MechDesigner and MotionDesigner Reference

USER MANUAL

MechDesigner & MotionDesigner 16.1

Reference Manual

by PSMotion Ltd

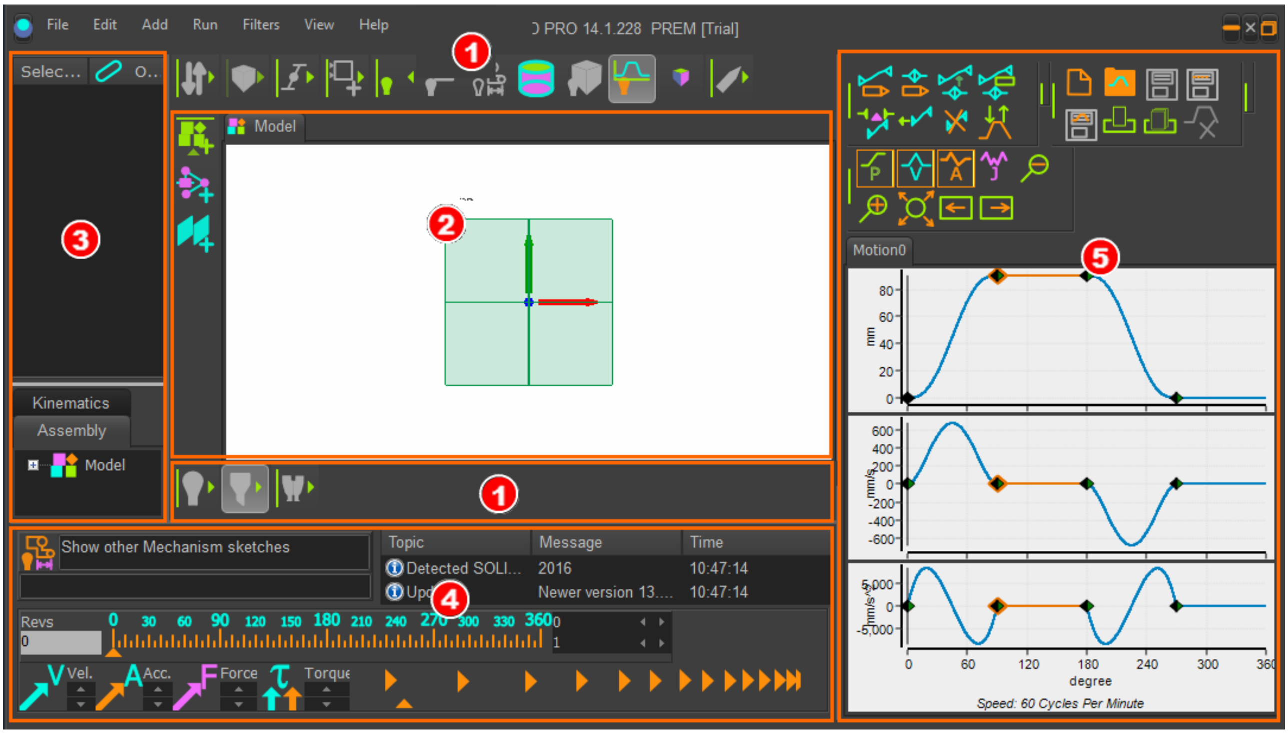
Together, MechDesigner and MotionDesigner are a superb solution for the design and analysis of Cams, Mechanisms, and Motions, for multi-axis machines.

MechDesigner Reference & User Interface

MechDesigner Reference Help documentation - MechDesigner is CAD-Software for Cam, Mechanism, and Motion Design and Analysis for Machines.

1 MechDesigner Reference & User Interface

User Interface - Visual Reference



1 Menus and Toolbars

- **Menus** are at the top of the application window.
- **Toolbars** are above, below, left, and right of the graphic-area. **Toolbars** are contextual to the editor-type.

2 Graphic-Area of Model-Editor, Mechanism-Editor, Part-Editor

To build models, repeat these steps in each editor-type :

STEP 1:	Add an ELEMENT	**See** Add... [45]
STEP 2:	Open the ELEMENT'S DIALOG	**See** Dialogs [276] , How to open a dialog [513]
STEP 3:	Edit the ELEMENT'S PARAMETERS	**See** How to edit a Parameter in a dialog [517]
STEP 4:	Close the dialog	

3 Project-Explorer - the **Project-Explorer** is to the left of the graphic-area.
See more: Project-Explorer [257]

4 Feedback-Area - the **Feedback-Area** is below the graphic-area.
See more: Feedback-Area [270]

5 MotionDesigner - **MotionDesigner** is docked (default) to the right of the graphic-area.
See more: MotionDesigner [273]

6 Memo - the MEMO opens when you start **MechDesigner** the first time. To turn the MEMO off, de-select the OPEN AUTOMATICALLY AT START check-box.
See more: Memo [275]

1.1 Known-Bugs

16.1.336

Hangs after Importing Library Model

When does it happen?

1. Open a Library file with **File toolbar > Import Library Model**

Result

You cannot select anything with your mouse.

The model seems to hang.

Temporary Solution:

1. Press the **ALT+C** keyboard combination to Cycle the model.
2. Press **ALT+C** again to stop the model cycling.

Version when corrected:

16.1.342 - not yet released

16.1.336

SOLIDWORKS 2018.

When does it happen?

Help about > Type Libraries are SOLIDWORKS 2018 - and all good to go.

1. Start SOLIDWORKS 2018, and open a Part document.
2. In MD: Edit a **CAD-LINE**
3. Import SOLIDWORKS document...crash

Result

Will not open a SW2018 document - my 16-1-336 tries to activate SOLIDWORKS 2020!

Temporary Solution:

None - can you use a different release of SOLIDWORKS?

Version when corrected:

16-1-342 - not yet released.

16.1.328

Lighting - Ambient increases to the Maximum

When does it happen?

1. Add a **PROFILE**
2. Enable **Show Solids in Mechanism-Editors**
3. Disable **Show Solids in Mechanism-Editors**

Result

The Ambient Lighting increases to a maximum

Solution:

1. Save the model if not already saved

2. Click the **MODEL** element at the top of the Assembly-Tree
3. Right-click the **MODEL** element

The **MODEL-OPTIONS DIALOG** is now open

4. Drag the **AMBIENT LIGHTING SLIDER** to almost the minimum.

Version when corrected:

Not yet corrected.

1.2 1: General Menus and Toolbars

1 Menu bar

The commands in the **File, Edit, Filters, View, Help** menus are common to most CAD applications.

The commands in the Add menu [45] are those that you need to build your **MechDesigner** model.

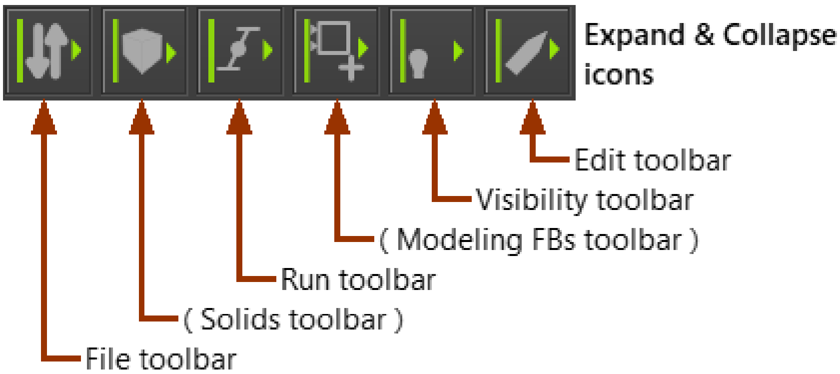
General toolbars

The **General toolbars** are above and below the graphic-area.

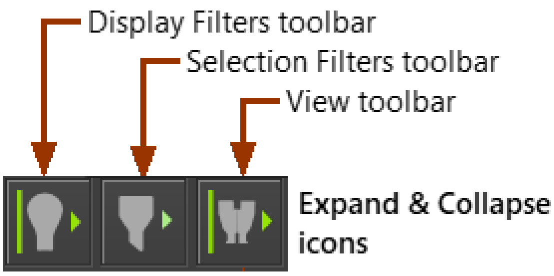
All **toolbars** are collapsed when you start **MechDesigner**.

Click the icon to **expand** and **collapse** each **toolbar**.

General toolbars ABOVE the graphic-area

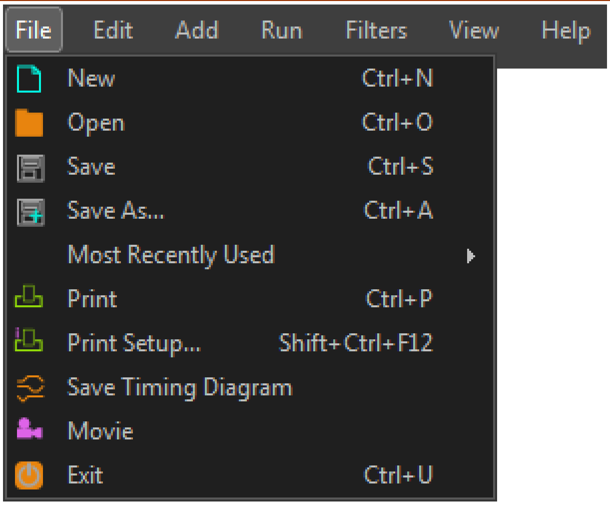


General toolbars BELOW the graphic-area



1.2.1 File

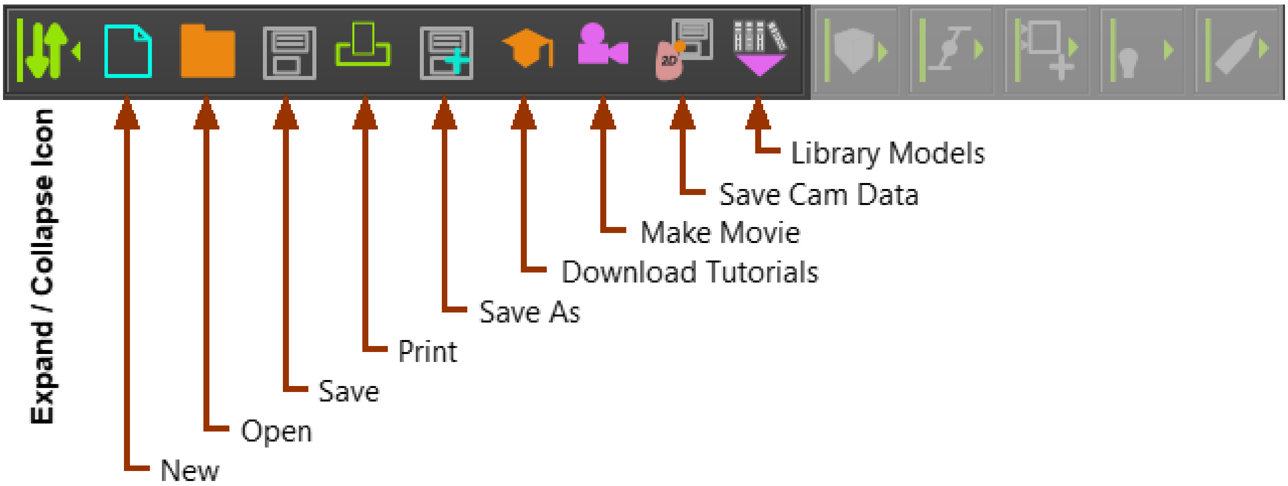
File menu



File menu

File toolbar

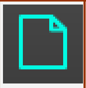
The **File toolbar** is **ABOVE** the graphic-area.



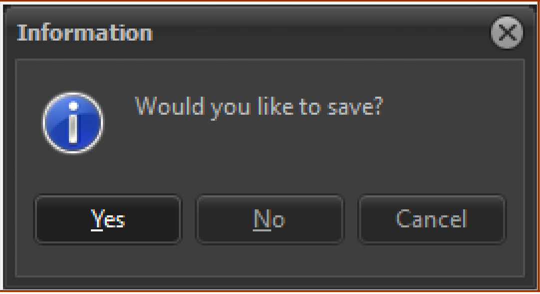
File toolbar

1.2.1.1 File | New

File > New

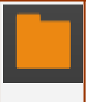
	File toolbar > New File menu > New
Keyboard Shortcut: CTRL+N	

If the active model is different from the model that has been saved, you must decide what to do before you start the new model.

	**Yes**: Close the dialog and save the active model **No**: Close the dialog and not save the active model **Cancel**: Close the dialog and do not save the active model

1.2.1.2 File | Open

File > Open

	File menu > Open File toolbar > Open

See also **File toolbar > Import Library Files** [32]

Open these File-types:

CXL

When you save a model, you save two files to the same path, with the same file-name.

CXL - the file-type that is for the **MechDesigner** model.

MTD - the file-type that is for the **MotionDesigner** motions that model needs.

OLD MODELS: **MechDesigner** may find a sketch-element that is over-constrained.

A pop-up asks you if we can delete for you the constraint so that we can solve the sketch. Click **YES** if you want to open the file! Check important sketches after you open the model, then save the model again.

See also: **Why so many CXL and MTD files?** [16]

LXL

Small Bug: After you open a Library LXL file-type, the model is unresponsive to your mouse-pointer. To solve this bug, do ALT+C on your keyboard to cycle the model, or do Run menu > Cycle, then do again to stop the model.

LXL - the file-type for a Library file.

There are three options when you open a **Library File**.

- **Preselect a Plane** - the Library File adds a new MECHANISM-EDITOR. The model in the Library file opens on the XY-Plane of the new MECHANISM-EDITOR.
- **Preselect a Part** - the Library File opens on the PART. The Library File is a child to the PART.
- **Preselect nothing** - the Library Fie is opened and merged with the active MECHANISM-EDITOR.

See more: **Open LXL File** [18]
See also: **Save as > LXL file-type** [24]

ZXL

ZXL is the **file-type** for a **MechDesigner ZIP file - see** Save as > ZXL [24]

The file-types that may be zipped-up in a ZXL file are the:

- **CXL** and **MTD file-types**
- **STL**, **SLDPRT**, **SLDASM** and **DXF file-types** that you may have linked to CAD-LINES.

After you open the ZXL file, the **STL, SLDPRT, SLDASM file-types** become linked to the ***path*** in which you extract the ZXL files.

Note:

You can choose to specify the path into which you want to extract ZXL file-types:

Application Settings | General tab | [284] FILE OPTIONS > Extract ZXL files to [284]:

- CREATE SUB-DIRECTORY AUTOMATICALLY - all files extract into a sub-directory to the path location of the ZXL file.
- SPECIFY ZXL OUTPUT DIRECTORY - you specify the path for the files.

I recommend you select **Specify ZXL Output Directory**. I usually specify the same path as that of the ZXL file itself. You can also select or make a new path.

See more: File menu > Save as > ZXL file-type. [24]

DXF

When you open a **DXF file-type**, you add a **DXF-ELEMENT** to the **ASSEMBLY-TREE** as a child to the **MODEL**. The **DXF ELEMENT** is the **container** for the **DXF-DRAWING**.

You must link the **DXF-ELEMENT** to a CAD-LINE to show the **DXF-DRAWING** in the graphic-area.

Thus, to show a **DXF-DRAWING**, do:

STEP 1. **File > Open DXF file-type**

STEP 2. Edit a CAD-LINE to open the CAD-LINE DIALOG

STEP 3. In the CAD-LINE DIALOG > **DXF tab**, select the DXF-ELEMENT

The **DXF-DRAWING** shows in the graphic-area.

See more: CAD-Line dialog > DXF tab [306]

MEC [Camlinks]

MEC is a Camlinks file-type.

See more: Open MEC files [21]

Why are there so many CXL and MTD files?

If you use Windows Explorer to find your model files, you also see many other files.

Why so many? They are the backup files.

The **CXL** is the file for the **MechDesigner** model. The **MTD** file is for the motions in **MotionDesigner,** It opens also when you open the **CXL** file.

When you open a model, we also look for the oldest backup files and copy them to another backup file.

- **CXL.*n*** and **MTD.*n*** are copied to **CXL.*n+1*** and **MTD.*n+1***

Thus if:

1. **CXL.2** and **MTD.2** are the oldest, they are copied to **CXL.3** and **MTD.3**
1. **CXL.1** and **MTD.1** are copied to replace **CXL.2** and **MTD.2**
2. **CXL** and **MTD** are copied to **CXL.1** and **MTD.1**
3. A copy of **CXL** and **MTD** are saved to **RAM**

AUTO-SAVE

When you Enable Auto-save [284], we **auto-save** the model for you after a number of commands:

1. **CXL** and **MTD** in **RAM** are saved to **~CXL** and **~MTD** on disk

2. New copies of **CXL** and **MTD** are saved to **RAM**

WARNING : **You overwrite the original model with each Auto-Save.**

See **Application Settings > General tab > File Options** 284 to:

- Enable or disable Auto-save
- Edit the number of Backup files
- Edit the number of commands between each Auto-Save

1.2.1.2.1 Open LXL file-type

File > Open > LXL file-type

Open a **Library file (LXL file-type)** that you have saved at another time - see Save as Library File [24].

SEE > NEW in MD16 **Import Library File** [32]

To open a Library File with File > Open

1. Click to select a PLANE, a PART; or select Nothing:

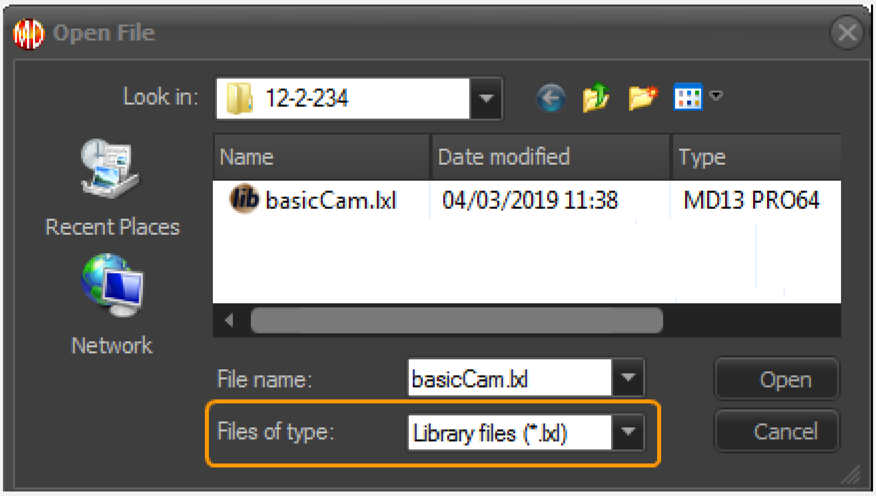
   If you:

   - Select a PLANE - the **Library file** opens onto a new **MECHANISM-EDITOR** as a child to the PLANE
   - Select a PART - the **Library fie** opens onto the PART as a child to the PART.
   - Select **nothing** - the **Library fie** opens and merges with the active **MECHANISM-EDITOR**.

2. Click **File > Open**

   The **Windows® File Open** interface starts.

3. In the **Files of type** drop-down, select **Library files (*.LXL)**



4. Select, a **Library File**
5. Click the **Open** button

   You can now save the original model before you open the Library File.

You should now see the **Library file** as a new **MECHANISM-EDITOR**, as a child to a PART, or merged with the active **MECHANISM-EDITOR**.

1.2.1.2.2 Open DXF File

File > Open > DXF file-type

When you do: **File menu** > **Open** > **DXF** file-type, you add a **DXF-ELEMENT** to the **ASSEMBLY-TREE.**

The **DXF-ELEMENT** is the container for the **DXF-DRAWING**.

To show the DXF-Drawing, you must link that **DXF-ELEMENT** and **DXF-DRAWING** to a **CAD-LINE.**

To show a DXF-drawing:

STEP 1 : File menu > Open > DXF file-type

1. Click **File menu (or toolbar) > Open**

Make sure the **File of type** filter is set to DXF.

2. Browse to a DXF file
3. Click the Open button

A **DXF-ELEMENT** shows in the **ASSEMBLY-TREE**.

STEP 2 : Show the DXF-Drawing

1. Edit a **CAD-LINE** to open its **CAD-LINE DIALOG**
2. Select the **DXF** tab

See CAD-Line dialog > DXF tab 306

Options

Edit the **DXF ELEMENT** in the **ASSEMBLY-TREE** to:

- Link the **DXF ELEMENT** with a different **DXF-DRAWING** file
- Define the units of the **DXF-DRAWING.** Usually, to the units of the original **DXF-DRAWING**.

See also: DXF-Element dialog 294

Top-Tips

When you save a **DXF-DRAWING** in your original CAD software:

- Save the **DXF-DRAWING** as **Release 1,** or as near to **Release 12** as possible.
- If you use 3D CAD, use **Hidden Lines Removed** before you save the **DXF-DRAWING**

1.2.1.2.3 Open MEC file-type

Why we recommend that you re-build Camlinks models in **MechDesigner**.

- You will remember your design decisions.
- We must add elements from the Camlinks model that you do not need in the MechDesigner model.
- We cannot support you with your Camlinks models.
- We want to help you use **MechDesigner.**
- We can guarantee **MechDesigner** will run with Windows® 10, 11, ...
- We **cannot** guarantee that the **MechDesigner** will represent the Camlinks model in every respect.
- We continue to improve **MechDesigner** in many new ways.

IMPORTANT: LOCKED-POINTS: After you open a .MEC file, Points in the model you import are **LOCKED**.

To edit the Position of a Locked-Point:

1. Double-click the Locked-Point to open the POINT PROPERTIES DIALOG[444]
2. Click **PART COORDINATES (READ-WRITE)** and
3. Edit the x and y values

To add and edit dimensions to edit the Position of the a Point:

1. Double-click the **LOCKED-POINT** to open the POINT PROPERTIES DIALOG[444]
2. Click **PART COORDINATES (READ-WRITE)**
3. Click **EDIT IN PART-EDITOR** radio-button
4. Click ✓ to close the dialog.
5. Edit the **PART** in the **PART-EDITOR** in the normal way.

History Note for Camlinks, and Camlinks Import Limitations.

MOTION: Camlinks4 vs Camlinks3

1. 'Camlinks4': Mechanisms and motions are a MEC file-type. The motions were defined with segment and motion parameters. Open a 'MEC' file in MechDesigner to recreate the motions with motion segments in MotionDesigner.
2. 'Camlinks3': Mechanisms and motions use also a MEC file-type. However, motions were defined with a list of numbers. Open a 'MEC' file in MechDesigner to recreate the motion, but with a **Z Raw Data** List Segment-Type in MotionDesigner.
3. To help you recreate the motion design (with segments), save the **Z Raw Data** List Segment-Type, and use the Overlay Motion tool to design a new motion MotionDesigner: Overlay Trace.

Throw Motion-Law

These are available in Camlinks but not imported into ***MotionDesigner***. However, it is possible to design the **Throw Motion-Law** with two(2) segments and the correct Blend-point Control Button conditions. We can give a Video on request.

Auto-Adjust

Auto-Adjust is not recreated. However, we have a better tool, which we call Magnetic-Joint[82].

View Outline On/Off as a Function of the Master Machine Angle

MechDesigner uses the Pattern Element - Release 7+

Camlinks Outlines

Camlinks Outlines are DXF-Drawings that you have imported onto Parts in Camlinks. The lines and arcs in the DXF-Drawing were converted to 'Camlinks Outlines'. They were 'dumb'. You could not edit them in Camlinks.

When you open a Camlinks file with 'Outlines' in MechDesigner, you have three options available. See below [21].

SSR

We do not import these dyads.

You can design the SSR and SSP dyas in **MechDesigner.**

RamR and RamP Dyads

We do not import these dyads. If you use these dyads, MechDesigner will crash (23-03-2013).

Forces, Forcebars, Torquebars

Use **MechDesigner** to add and show forces and torques in your kinematic-chains.

Cams

When a Cam-Follower Roller has 0mm diameter in Camlinks – to define the Pitch-Center of the Cam – **MechDesigner** will re-define it as 0.1mm, not as 0.0mm.

You must edit the 0.1mm dimension to make it equal to the real Cam-Follower Radius. Then, to show the Pitch-Curve, use the Pitch-Circle-Path check-box in the 2D-Cam dialog [327].

1.2.1.3 File | Save

File > Save

	File menu > Save File toolbar > Save
If the model does not have a file-name, the Save as dialog[24] opens.	
Keyboard Shortcut: CTRL+S	

Notes:

File > Save actually saves two files:

- **CXL** - for the **MechDesigner** model
- **MTD** - for the **MotionDesigner** motions

AUTO-SAVE

Auto-save starts **only** when the model has a file-name.

MOTIONDESIGNER.

You can also, in **MotionDesigner**, use **File toolbar > Save Active Motion, Save all Motions**, and **Save as** to save your motions.

See also:

File menu > Save as[24]

Edit menu > Application-Settings > General Tab > File Options: Auto-Save[284]

1.2.1.4 File | Save as

File > Save as

File menu > Save as

File toolbar > Save as

Save as:

CXL

CXL & MTD file-types

- **CXL file-type** is for the **MechDesigner** model
- **MTD file-type** is for **MotionDesigner** motions

LXL

See also: Open Library File [18], and Import Library File [32].

LXL file-type is a **Library File**.

1. Browse to a path, to save the file as a Library file.
2. Select **Save as Type: Library Files (.lxl)**

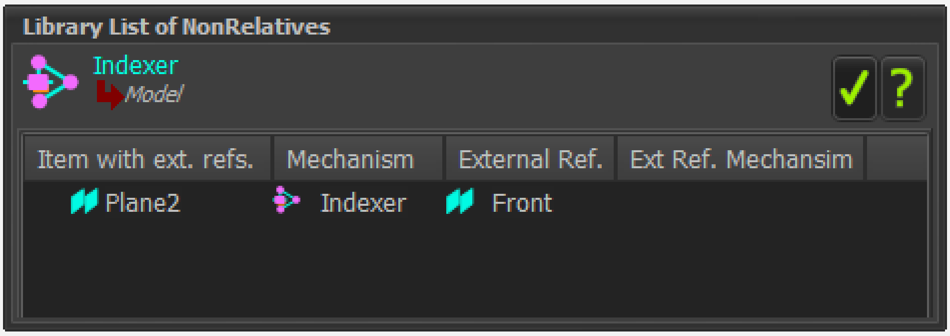
OR

1. Save the file to your User-Library

Notes:

If the active **MECHANISM-EDITOR** has a child **MECHANISM-EDITOR**, the **LXL file** CANNOT save.

The **Library List of Non-relatives** information pop-up shows if you cannot save the **LXL file**.



ZXL

This **file-type** is identical to a **ZIP file-type.** You can use **Win-Zip®** to unzip **ZXL file-types.**

You must save the model as a **CXL file-type** before you can save it with the **ZXL file-type.**

If these **file-types** are in the model, we save these files for you with the **ZXL file**:

- SLDPRT*
- SLDASM*
- STL
- DXF
- TXT (used with a Point-Cloud).

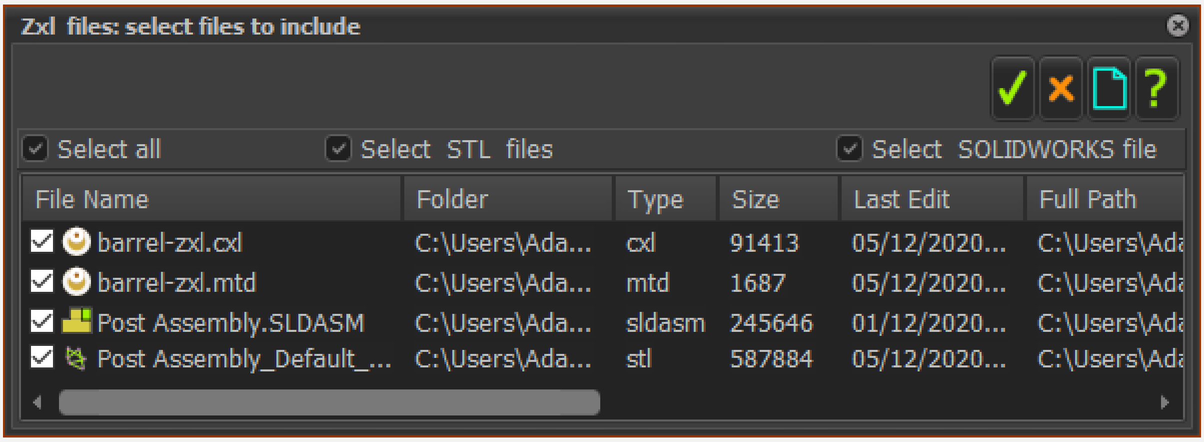
* IMPORTANT - save all SOLIDWORKS documents and exit SOLIDWORKS before you do File > Save as the ZXL file-type.

If you link a SLDASM assembly document with a CAD-LINE, you can also save the SLDPRT part documents that are in the assembly with the ZXL file.

When the ZXL file interface opens, you can:

- ☑ Select all
- ☑ Select STL files
- ☑ Select **SOLIDWORKS** file

Click ✓ to close the interface and save the ZXL file.



DXF

A **DXF** file of the model includes LINES, ARCS and PART-AXES.

You can save a DXF file of one PART or all of the PARTS in a MECHANISM-EDITOR:

1. Click to preselect a MECHANISM ELEMENT or PART ELEMENT

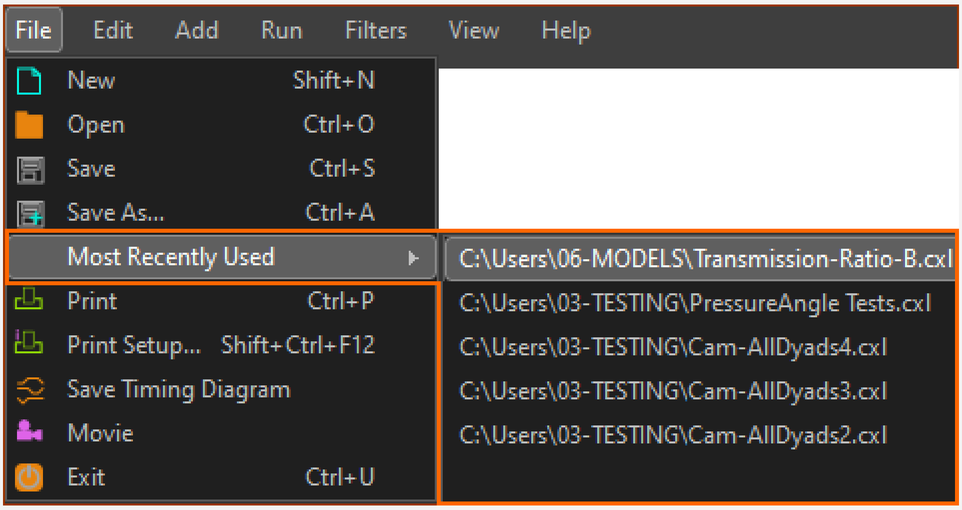
   The MECHANISM or PART element **must** show in the SELECTION-WINDOW.
2. Click **File menu > Save as**
3. Select **DXF** as the **File of type** in the **Save as.** dialog
4. Enter a file name.

1.2.1.5 File | Most Recently Used

File menu > Most Recently Used

The most recent file is at the top of the list.

If you have opened or saved **DXF** files, they are below the **CXL** files.



Most Recently Used files.

1.2.1.6 File | Print

File > Print

	File menu > Print File toolbar > Print

Use to:

- Print a screenshot of the graphic-area.

AND/OR

- Save a screenshot of the graphic-area as a JPG, BMP, PNG or GIF file-type.

The screenshot is a 1:1 scale of the graphic-area.

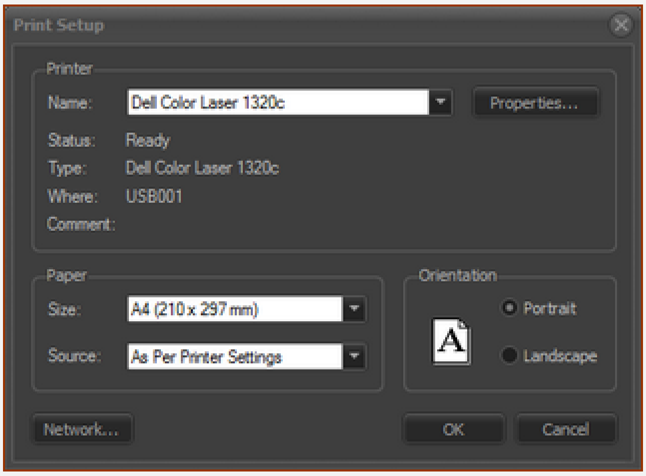
Re-size the graphic-area to increase or decrease the area of the screenshot.

1.2.1.7 File | Printer Set-up

File > Printer Set-up

See also: Print 27

This is a Windows® form that you can use to select the Printer, and other properties.



Print Setup dialog

1.2.1.8 File | Download Tutorial Videos

File toolbar (Help menu) > Download Tutorial Videos

See also : **Download Tutorials dialog** 507

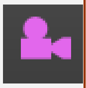
To open the Download Tutorial dialog:

	Click File toolbar > Download Tutorial Videos
OR Help menu > Download Tutorial Videos	

1.2.1.9 File | Make Movie

File > Make Movie

See also Make Movie dialog 495



File toolbar > Make Movie (video)

File menu > Make Movie

Do **Make Movie** to:

- record and save a movie as a GIF file-type

OR

- to record and save a sequence of images as JPEG, JPG, PNG and BMP file-types
  We append for you a number to each image. The numbers are 000, 001, 002, ...

OR

- save one image - a snapshot of the model.

Notes:

The GIF format produces a large file when the movie-frame-size and/or the number-of-frames is large.

You can edit the position and size of the movie-frame, and also number-of-frames in the **Make Movie dialog**.

TOP-TIP - **Before** you do **File menu > Make Movie** move the Master-Machine-Angle to zero(0).

1. **Run menu > Home**

OR

1. Use the **ALT+H** keyboard shortcut.

1.2.1.1 File | Save Timing Diagram
0

File menu > Save Timing Diagram

File toolbar - NO ICON

A Timing Diagram:

- Represents each motion, in a Motion name-tab, with a different graph.
- The graphs are above each other.
- Each graph has straight-lines between the positional values of each **BLEND-POINT**.

How to export and show a Timing Diagram in Excel

STEP 1: MechDesigner - Save the Timing-Diagram.

A: Do: File menu > Save Timing Diagram

We save for you a CSV file-type with the required data to display a timing diagram with the **Excel workbook**: **Timing Diagram.xlsm** - **see STEP 2, below**

The CSV file includes the:

- **MOTION NAME**
- **BLEND-POINT**: **X-axis** and **Y-axis** position values

The CSV does not include the:

- Motion-Law (Cam-Law)
- Motion derivatives

STEP 2: Microsoft Excel

A. Open the **Excel Macro-Enabled Worksheet: Timing-Diagram.XLSM**

Timing Diagram.xlsm is located in:

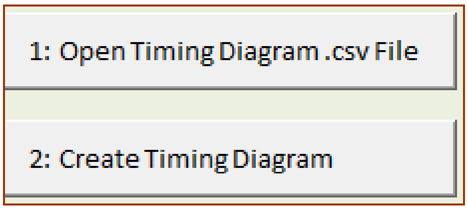
<LOCALAPPDATA> \ TIMING DIAGRAM.XLSM

OR

<COMMONAPPDATA> \ TIMING DIAGRAM.XLSM

B. Enable macros

There are two buttons in the Timing-Diagram | Motion worksheet.



C. Press Button: 

Find, and open, the CSV file you have saved to disc with MechDesigner : File menu > Save Timing Diagram - **see STEP 1, above**

NOTE - reformatting is required:

Look carefully at the image below - **Typical Timing-Diagram with 2 Motions**.

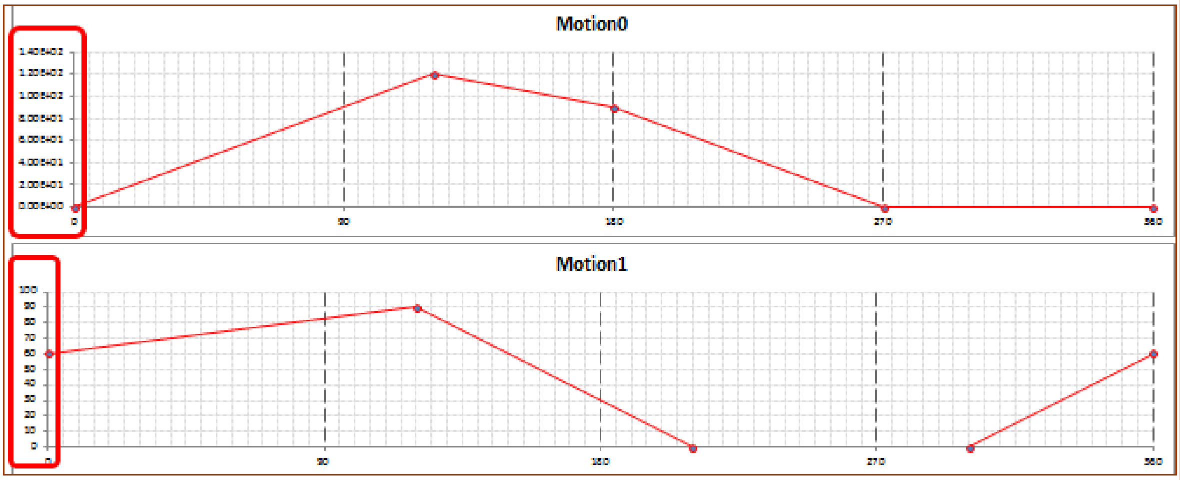
The data-format of the **Y-axis** of the top motion moves the plot of the actual Timing-Diagram slightly to the right.

To correct this:

1. Format the Y-axis data of 'Motion0' to make sure its data format is the same as 'Motion1, 2, ...'.

D. Press Button: 2: Create Timing Diagram

E. Look at the worksheet Timing Diagram to see the timing diagram for each motion



Typical Timing Diagram with 2 Motions

1.2.1.1 File | Import Library File
1

File toolbar > Import Library File

See also: File menu > Open > File of Type: LXL [15].

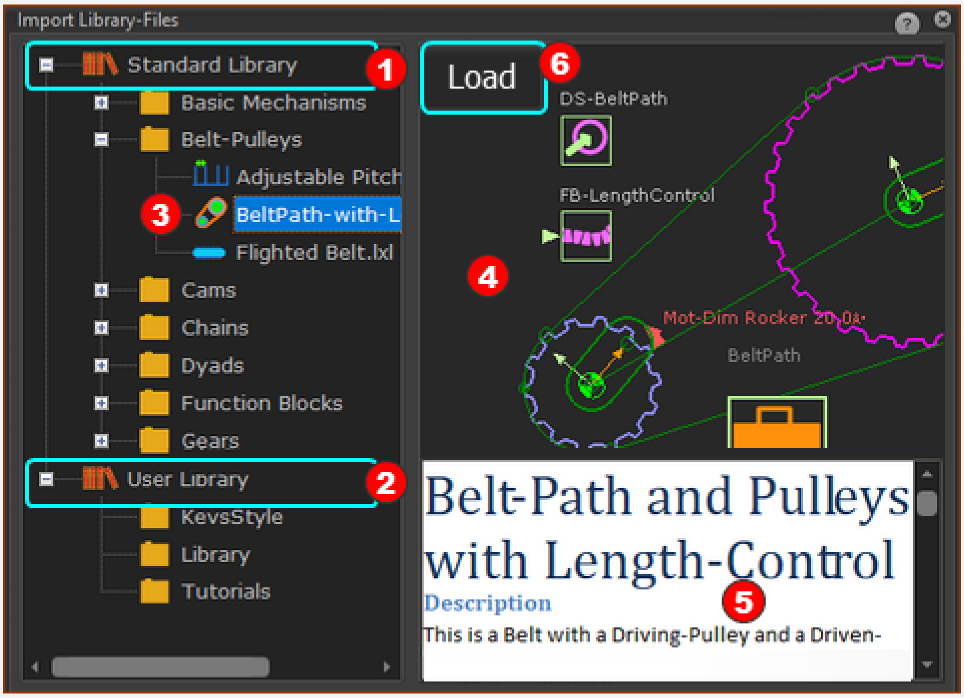
A **Library-File** is a model that you can import and merge with the active **MECHANISM-EDITOR**. It has the **LXL** file-extension.

Use **Import Library File** to save time. Use the **Import Library File** interface to review the **Library File** before you import it.

Warnings

1. You cannot undo Import Library File.
2. **Auto-save** starts only after you save the model with a file-name.

Import Library-File interface



Import Library File interface

There are two libraries:

❶ **STANDARD LIBRARY** - for those customers that have **Annual Support** can download these models from our website.

❷ **USER LIBRARY -** your Library Files.

You need these files in the User Library Path

❸ **LibraryFile.LXL** - see Save as > File of Type > Library File (.LXL)

Optionally, **LibraryFile.MTD** when the Library File needs a Motion.

To get the maximum use from your **USER LIBRARY** and the **Import Library-File** interface, you need these files in the same path as the **LibraryFile.LXL**

❹ **LibraryFile.PNG** - an image - ideal size is 480 x 360 that is a snapshot of the model in the Library File.

To save a **480×360** image, use **File menu > Movie** [29]: **File-type**: PNG ; **File-path**: User Library Path ; **Frame Size**: **430** × **360** ; Press Single Shot button.

❺ **LibraryFile.RTF** - a description of the model, and what to edit (ideally with a Design-Set [185]).

Use Word®, for example.

Optionally:

❸ **LibraryFile24.PNG** - an icon that is to the left of each Library-File

Use an image-editor to make an image that has exactly 24x24 pixels, with a transparent background (Note that the file-name is appended with 24).

Import a Library File:

STEP 1: Click to select a Mechanism-Editor OR Edit a Part

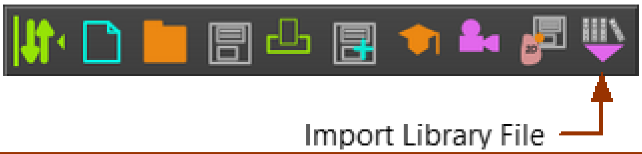
Model-Editor: You add the **Library File** to a new **MECHANISM-EDITOR** on the **Front** **PLANE**.

Mechanism-Editor: You will merge the **Library File** with the active **MECHANISM-EDITOR**.

Remember, you can add a **MECHANISM-EDITOR** to a **PLANE** that references a moving **LINE** in a **PART** - see Add Plane to Line [77]

Part: If the **Library File** is Geometry ONLY, then edit a **PART** to import the Geometry onto the **PART**.

STEP 2: File toolbar > Import Library-File



STEP 3: Use the Import Library File interface (see above) to find a Library file:

Standard Library ❶ - provided by PSMotion

User Library❷ - your Library Files

When you click a **Library file**, you see:

❹ an image that is a preview of the **Library-File**

❺ a note that describes the **Library File** and how and which dimensions and parameters to edit its, ideally with a Design-Set.

STEP 4: Click the LOAD button❺

We load the Library File immediately into the Mechanism-Editor or Part - see **STEP 1**

STEP 5: Click ⊗ to close the dialog

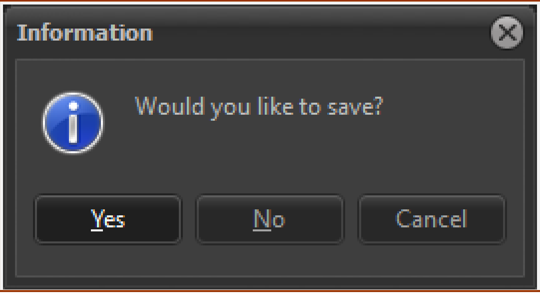
1.2.1.1 File | Exit 2

File > Exit

To close **MechDesigner**.

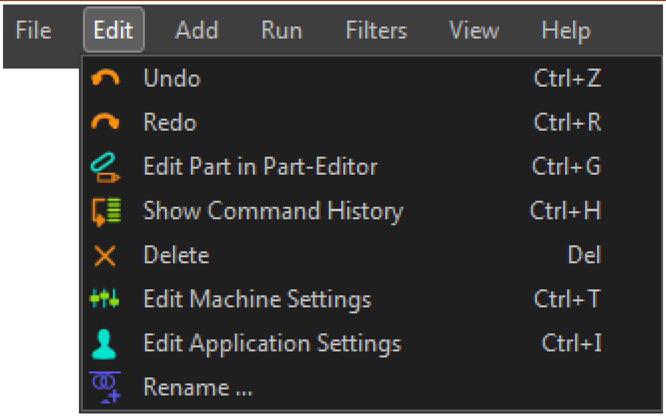
If the model has changes since you last saved it, the **Information** box shows:

1. Click Yes, No, or Cancel.



1.2.2 Edit

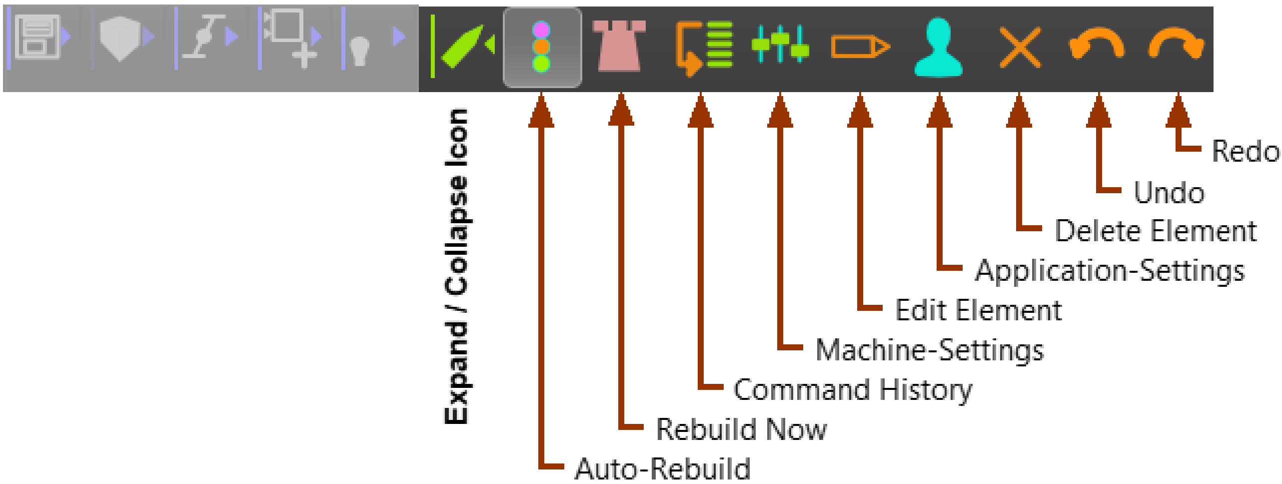
Edit menu



Edit menu

Edit toolbar

The **Edit toolbar** is **ABOVE** the graphic-area.



Edit Toolbar

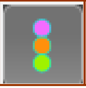
1.2.2.1 Edit | Auto Rebuild

Auto Rebuild

Note: Auto Rebuild is in the Edit toolbar only

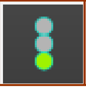
See also Rebuild Now [37]

Enable Auto-Rebuild.



The model **should** be up-to date - see Top-Tips, below.

Disable Auto-Rebuild



The model **may not** be up-to-date.

Top-Tips

To make sure the model is up-to-date - especially for complex models:

1. Click Rebuild Now [37]
2. Click to disable **Auto Rebuild**, and then click again to enable **Auto-Rebuild.**
3. Move the **Master-Machine-Angle** to 0 (Do ALT+H (Run menu > Home)).
4. Edit a PART, then exit the PART-EDITOR. This also rebuilds the model.
5. If there is a 2D-CAM in the model, open then close the 2D-CAM DIALOG.
6. Click Rebuild now [37].

1.2.2.2 Edit | Rebuild Now

Rebuild Now

Note: Auto Rebuild is not available in the Edit menu, only the Edit toolbar

Rebuild Now does a one-time, model rebuild.

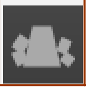
The button also indicates if the model is up-to-date or not up-to-date.

See also Auto Rebuild [37]

Model is up-to-date

The model is usually up-to-date.

Model is not up-to-date



The model is not up-to-date.

Click the **Rebuild Now** icon to rebuild the model (and the fort!).

1.2.2.3 Edit | Command History

Command-History

Use the **Command-History dialog** to undo or to redo more than one command at a time.

To open the **Command-History dialog:**

1. Click **Edit toolbar >Command History**

or

1. Click **Edit menu > Command History**

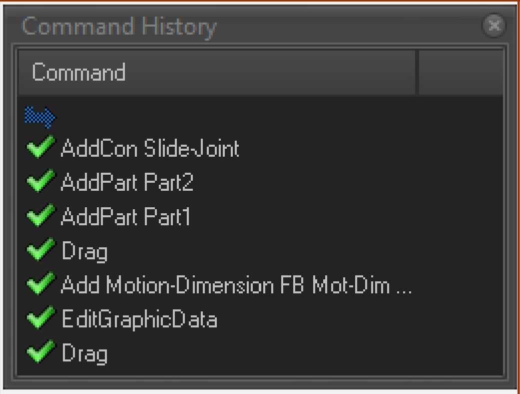
The **COMMAND-HISTORY DIALOG** is now open

See also:

Undo [43]

Redo [43]

Command-History dialog

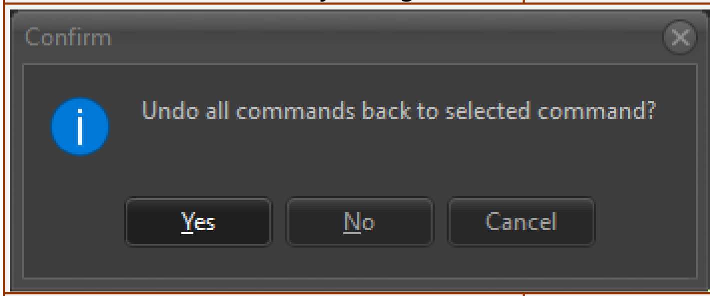


Command-History dialog

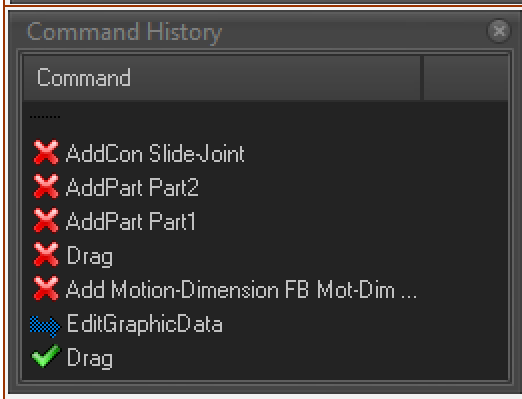
When you open the **Command-History dialog**, the most recent command is at the top of the list.

To **Undo the Command-History**:

1. Select a command lower in the **COMMAND-HISTORY**.



2. Click YES to confirm you want to undo all of the commands.
3. Wait until all of the commands are undone.



Command History dialog

To Redo the Command History:

1. Select a command higher than the undone commands
2. Wait until all commands are redone

Warning:

If you do a new command before you Redo commands in the Command History, the new command becomes the top command in the Command-History list.

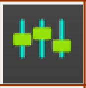
Then, you cannot redo the other commands.

1.2.2.4 Edit | Machine-Settings

Machine-Setting

Machine-Settings include: MACHINE-SPEED (RPM), NUMBER-OF-STEPS in a Machine-Cycle, and the ENGINEERING UNITS.

To open the **Machine-Settings dialog:**

	1. Click **Edit toolbar** > **Machine-Settings**
or	1. Click **Edit menu** > **Machine Settings**

The **MACHINE SETTING DIALOG** is now open.

See Machine Setting dialog [291]
See also : Application-Settings dialog [41]

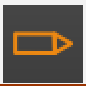
1.2.2.5 Edit | Edit Element

Edit Element

Note: You cannot edit all element-types - for example, you cannot edit a **START-POINT.**

To edit an **ELEMENT**:

1. Click the **ELEMENT** in
   - the graphic-area,
   - the **ASSEMBLY-TREE**, or
   - the **SELECTION-WINDOW** [257]



2. Click **Edit toolbar** > **Edit Element**

When it is possible to edit the element-type, the dialog to edit the **ELEMENT** is now open.

See also:

Delete Element [42]

How to open a dialog [513]

How to edit a parameter [517]

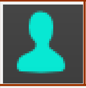
Dialogs [276]

1.2.2.6 Edit | Application-Settings

Application Settings

Application Settings include: colors, number-formats, element sizes, ... and many more settings.

To open the **Application-Setting dialog:**

	1. Click **Edit toolbar** > **Application-Settings**
or	1. Click **Edit menu** > **Edit Application-Settings**

The **APPLICATION-SETTINGS DIALOG** is now open.

See Application-Settings dialog 284
See also: Machine-Settings dialog 39

Notes:

Application-Settings are automatically saved when you exit **MechDesigner** to these files::

- **<LocalAppData>\IniFiles\MechDesigner.INI**
- **<LocalAppData>\IniFiles\ MechDesigner.XML**

Delete these two files to revert to factory-settings.

1.2.2.7 Edit | Delete Element

Delete Element

There are many ways to delete elements.

For example, select the element, then use your **Delete** key on your keyboard.

This topic is how to use the **Delete Element** tool.

To delete elements with **Edit menu** > **Delete element**:

1. Click an element in:
   - the graphic-area, or
   - the **ASSEMBLY-TREE**

One or more elements are now in the **SELECTION-WINDOW**.

To delete **ALL** of the elements that are in the **SELECTION-WINDOW**:

2. Click **Edit toolbar** > **Delete Element**

If there are dependent elements, the **Delete Dependent elements** opens.

Be careful! Only click ✓ in the **Delete Dependent element** form if you are sure.

To delete **ONE** of the elements that is in the **SELECTION-WINDOW**:

2. Click, in the **SELECTION-WINDOW** [257], the element that you want to delete.
3. Click **Edit toolbar** > **Delete Element**

If there are dependent elements, the **Delete Dependent elements** opens.

Be careful! Only click ✓ in the **Delete Dependent element** form if you are sure.

Notes:

- You **cannot** delete an element that is in a **DESIGN-SET** [185]. You must remove the element from the **DESIGN-SET** before you can delete it.
- You **cannot** delete a **2D-CAM** after you add to it a **POLYLINE**. You must delete the **POLYLINE**, then delete the **2D-CAM**.

1.2.2.8 Edit | Undo

Undo

To **undo** commands:

	1. Click **Edit toolbar** > **Undo**
or	1. Click **Edit menu** > **Undo**

The most recent command is now undone.

Do again to undo earlier commands.

Notes:

- Keyboard shortcut: **CTRL+Z**
- To redo [43] a command, do **not** do a new command.

See also:

Command History [38], Redo [43]

1.2.2.9 Edit | Redo

Redo

To **Redo** commands you have undone:

	1. Click **Edit toolbar** > **Redo last undone command.**
or	1. Click **Edit menu** > **Redo last undone command**

The command that you undone is now redone

See also:

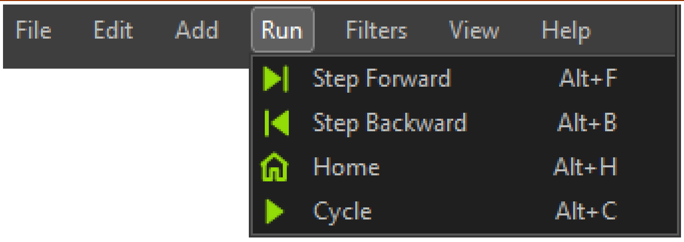
Command History [38], **Undo** [43]

Notes:

- Keyboard shortcut: **CTRL+R**
- To redo [43] a command you have undone, do ***not*** do a new command.

1.2.3 Run

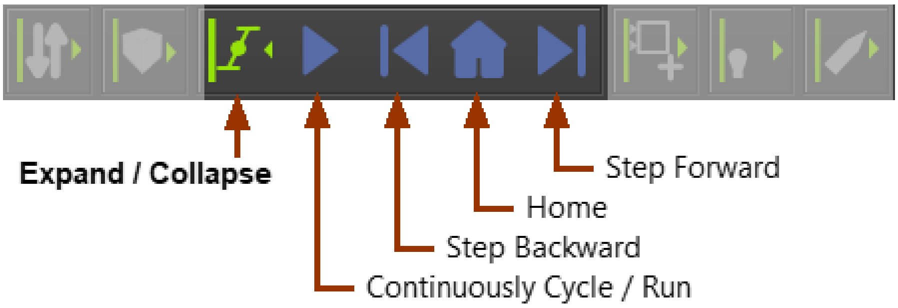
Run menu



Run menu

Run toolbar

The **Run toolbar** is **ABOVE** the graphic-area.



Run toolbar

Keyboard shortcuts

ICON	COMMAND	SHORTCUT (Not Case-Sensitive)
	CONTINUOUSLY CYCLE :	ALT+C
	To increase, at a constant rate, the **Master-Machine-Angle** (MMA).	
	STEP FORWARD 1 STEP* :	ALT+F
	To increase the **MMA** by **360°/** Number-of-Steps [291]	
	STEP-BACKWARD 1 STEP* :	ALT+B
	To decrease the **MMA** by **360°/** Number-of-Steps	
	CANCEL CYCLE & MOVE MMA TO ZERO* :	**Home key on your keyboard**
	To stop CYCLING, and move the **MMA** and **Revs** to **Zero** (0)	
* To edit the **MMA** to any machine-angle, approximately or exactly, **see Master-Machine-Angle -MMA**		

1.2.4 Add...

Add menu

The Add menu and sub-menus have the commands you need to build your models. The commands are contextual to each editor-type - MODEL-EDITOR, MECHANISM-EDITOR, PART-EDITOR.

To build a model, you do these steps:

STEP 1: Add an ELEMENT that is contextual to the editor-type:

See Model-Editor 65

See Mechanism-Editor 72

See Part-Editor 213

STEP 2: Open the ELEMENT'S DIALOG

Use the ELEMENT'S DIALOG to edit its PARAMETERS:

See Dialogs 276

See How to open a dialog 513

STEP 3: Edit the PARAMETERS to change the ELEMENT'S properties

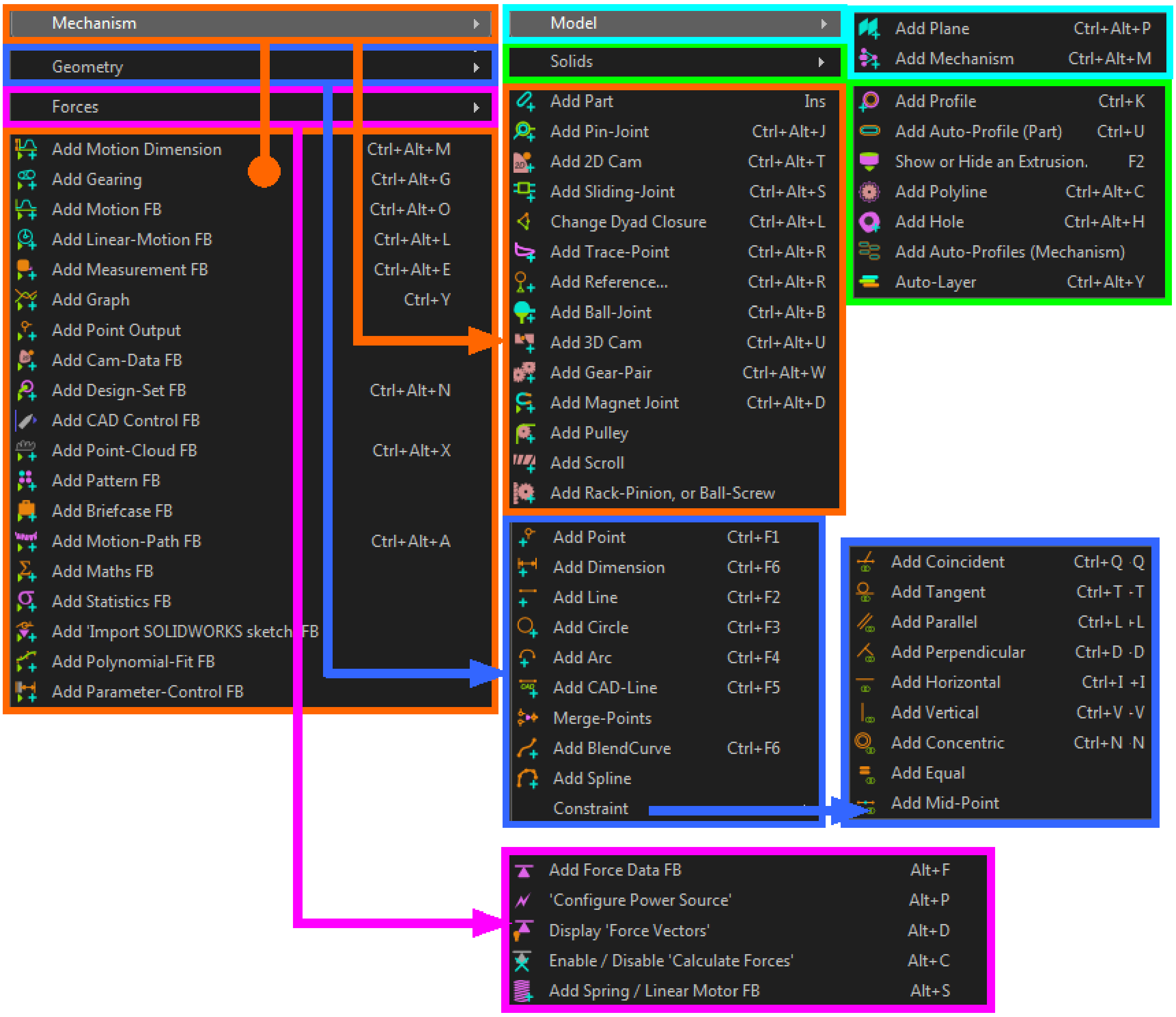
See How to edit a Parameter in a dialog 517

STEP 4: Close the dialog

STEP 5: Do STEPS 1 to 4 again, and again.

Add menu

This is the structure of the Add menu.



Structure of the Add menu

Model Editor menus and toolbar:

Model elements toolbar [73] - left of the graphic-area - add PLANES and MECHANISM-EDITORS to PLANES

Mechanism-Editor menus and toolbars:

Model elements toolbar [73] - left of the graphic-area - add PLANES and MECHANISM-EDITORS to PLANES

Kinematic elements toolbar [80] - left of the graphic-area - add basic elements: PARTS, JOINTS, to build kinematic-chains

Machine elements toolbar [108] - left of the graphic-area - add more complex, functional elements to the model

Kinematic Function-Blocks toolbar [142] - right of the graphic-area - plan and measure the motion of each kinematic-chain

Modeling Function-Blocks toolbar [176] - above the graphic-area - tools that help you do more complex modelling

Force elements toolbar [187] - right of the graphic-area - measure force, torque, and power that is required to drive each kinematic-chain

Solid element toolbar 199 - above the graphic-area - show kinematic elements as solid model elements

Part-Editor menus and toolbars:

Geometry toolbar 219 - left of the graphic-area - add sketch-elements and dimensions to PARTS

Constraints toolbar 244 - right of the graphic-area - add constraints to sketch-elements

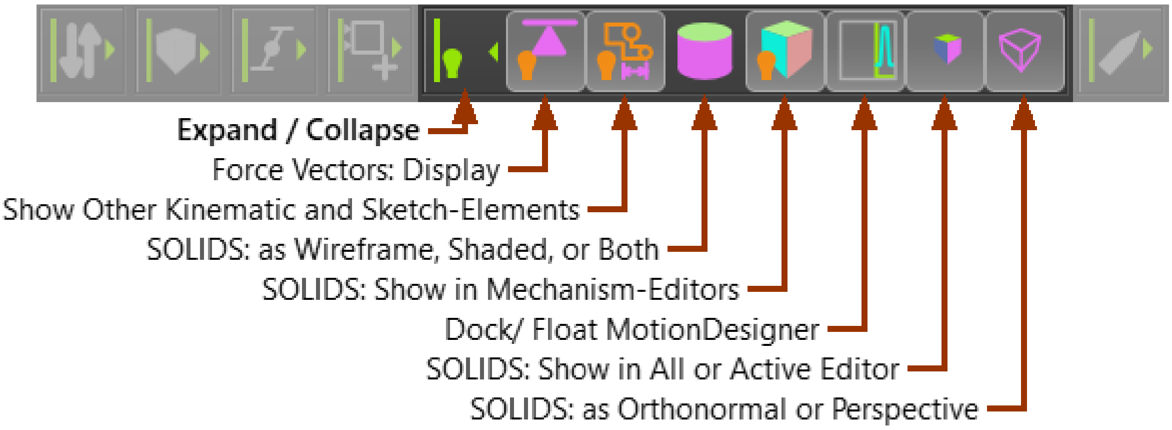
1.2.5 Visibility [toolbar only]

Visibility toolbar

Note: There is **not** a **Visibility menu**

The **Visibility toolbar** is **above** the graphic-area.

It uses toggles to enable or disable the visibility of different element-types.



Visibility toolbar

Visibility toolbar:

Visibility controls for SOLIDS

	- Enable "Force Vectors: Display" - See More [198]
	- Disable: "Force Vectors: Display"
	- Enable "Show other Kinematic and Sketch elements" - See More [49]
	- Disable "Show other Kinematic and Sketch elements"
	-Show "Solids as Shaded AND Wireframe edges" - See More [50]
	- Show "Solids as Wireframe only"
	- Show "Solids as Shaded only"
	- Enable "Show Solids in Mechanism-Editor" - See More [51]
	- Disable "Show Solids in Mechanism-Editor"
	- MotionDesigner is "Docked" : See More [52]
	- MotionDesigner is "Floating"

	- Show "Solids in ACTIVE Mechanism-Editor ONLY" - See More[52]
	- Show "Solids in ALL Mechanism-Editors"
	- Show "Solids as Orthonormal View" - See More[53]
	- Show "Solids as Projection View"

1.2.5.1 Show/Hide Force Vectors

Visibility toolbar > Force Vectors: Display

See Force toolbar > Force Vectors: Display[198]

1.2.5.2 Show Other Kinematic and Sketch elements

Visibility toolbar > Show Other Kinematic and Sketch-elements (Global Switch)

Use this command to switch the visibility of **Kinematic elements** and **Sketch elements** that are in the active and other MECHANISM-EDITORS.

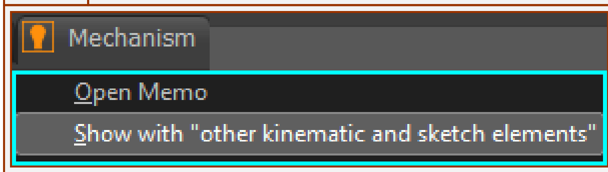
Use this control together with the Light-Bulb switch[49].

Global Switch

	Visibility toolbar > Show other Kinematic and Sketch elements : ON
	Visibility toolbar > Show other Kinematic and Sketch elements : OFF

Light-Bulb Switch

Each MECHANISM-EDITOR has a LIGHT-BULB.	
	Mechanism name-tab : light-bulb switch : ON
	Mechanism name-tab : light-bulb switch : OFF



To turn a Light-bulb switch to on :

1. Right-click on the name-tab of any MECHANISM-EDITOR
2. Click **Show with "other kinematic and sketch elements"**.

When the **GLOBAL SWITCH IS ON**, you can see:

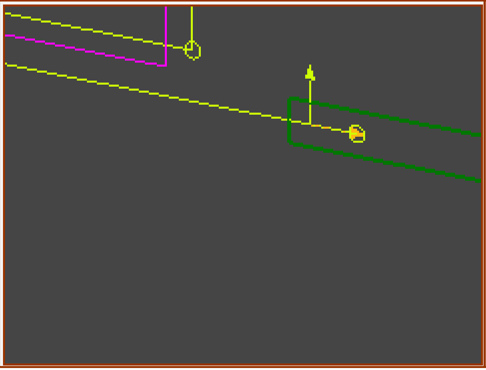
... those **Kinematic elements** and **Sketch elements** that are in the active MECHANISM-EDITOR

AND

... other MECHANISM-EDITORS that have their **LIGHT-BULB ON**

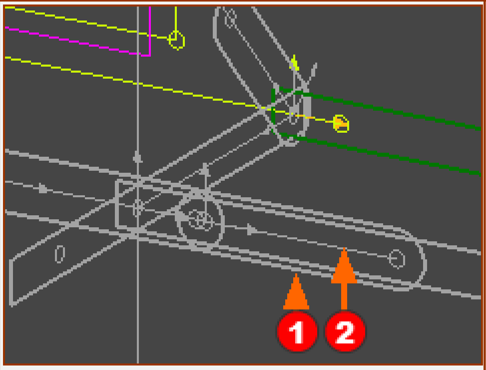
Example

Normal Visibility



Visibility Toolbar > Show other Kinematic and Sketch elements is OFF

Kinematic and Sketch-Elements of other Mechanism-Editors.



Visibility Toolbar > Show with other kinematic and sketch elements is ON

❶ A **PART-SYMBOL** in a different **MECHANISM-EDITOR** with its Light-Bulb ON

❷ A **LINE** or **CAD-LINE** in a different **MECHANISM-EDITOR** with its Light-Bulb ON

TOP-TIP :

To Identify Parts

- Single-Click a **PART-OUTLINE** (or **JOINT**, or **CAD-LINE**) to identify in the **SELECTION-WINDOW** and **ASSEMBLY-TREE** the name of the **PART** (or **JOINT**, or **CAD-LINE**) and the **MECHANISM-EDITOR** to which it is a child.
- Single-Click a **SKETCH-ELEMENT** to identify in the **SELECTION-WINDOW** the name of the **SKETCH-ELEMENT** and to which **PART** the **SKETCH-ELEMENT** is a child.

To Jump-to Mechanism-Editors

- Double-Click a **PART-OUTLINE** or **SKETCH-ELEMENT** to jump to the **MECHANISM-EDITOR** to which the **PART** or **SKETCH-ELEMENT** is a child.

See also:

Colors: To change the color of other kinematic and sketch elements:

Edit menu > Application-Settings > Graphics tab > GRAPHIC OPTIONS | Background Sketch

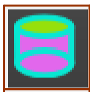
1.2.5.3 Solids Display Mode

Visibility toolbar > SOLIDS Display Mode

NOTE - there is not a Visibility menu

To change how you view **SOLIDS**.

The toggle switch has three states.

	Visibility toolbar > Solids as Shaded AND Wire-Frame
	Visibility toolbar > Solids as Wire-Frame ONLY (hidden lines visible)

 Visibility toolbar > Solids as Shaded ONLY

MUST also be enabled:

- Visibility toolbar> Show Solids in Mechanisms [51]

OPTIONS:

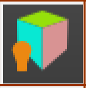
- Visibility toolbar > Show Solids in Active or All Mechanisms [52]
- Visibility toolbar > Show Solids as Orthonormal or as a Projection View [53]

1.2.5.4 Solids in Mechanisms

Visibility toolbar > Solids in Mechanisms (and Part-Editor)

NOTE - there is not a Visibility menu

Use this command to switch between Show and Hide **SOLIDS** in **MECHANISM-EDITORS**.

 Visibility toolbar > Enable Show Solids in Mechanisms

 Visibility toolbar > Disable Show Solids in Mechanisms

PLANES also show when you show **Solids in Mechanisms**, which you may find inconvenient.

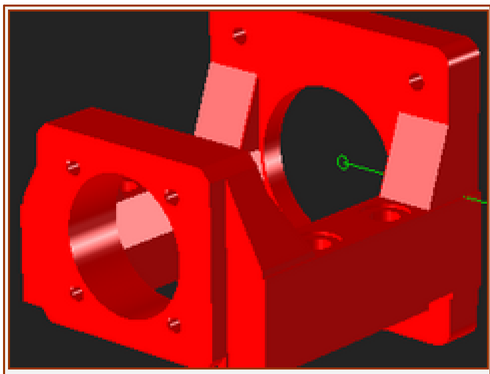
To hide PLANES, use:

- Filters menu > Display Filters sub-menu > Show or Hide Planes [54]

Use this command with:

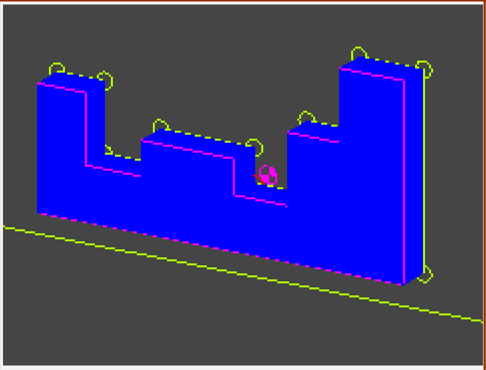
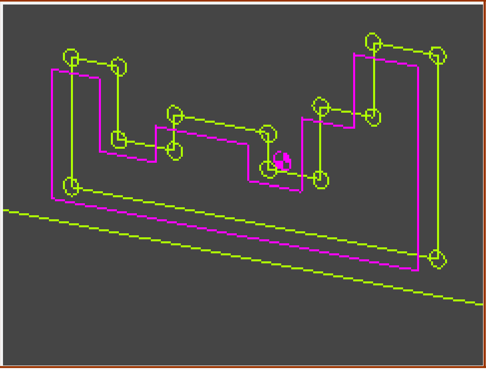
Visibility toolbar > Show Solids in Active or All Mechanism-Editors [52]
Visibility toolbar > Solids Display Mode [50]
Visibility toolbar > Show Solids as Orthonormal or Projection View [53]

Solids and Profiles in Mechanism-Editors



A CAD-Solid

This is a typical **CAD-SOLID** that you might import.

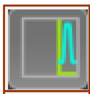
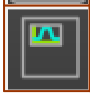
 An MD-Solid	This is a typical **MD-SOLID** you would sketch. Show Solids in Mechanisms is enabled. In **MECHANISM-EDITORS**, you can see Pink **PROFILE** and **EXTRUSION ELEMENTS**.
	This is: • Disable Show Solids in Mechanisms In **MECHANISM-EDITORS**, you can see only the Pink **PROFILE ELEMENTS**.

1.2.5.5 Dock or Float MotionDesigner

Visibility toolbar > Dock / Float MotionDesigner

NOTE - there is *not* a Visibility menu

This command has two(2) states:

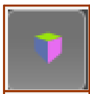
	Visibility toolbar > MotionDesigner is Docked
	Visibility toolbar > MotionDesigner is Floating

1.2.5.6 Solids In Active or All Mechanisms

Visibility toolbar > Show Solids in Active or All Mechanism-Editors

NOTE - there is not a Visibility menu

This command relates to **SOLID** and **PLANE** elements.

	Visibility toolbar > Show Solids in the active Mechanism-Editor ONLY
	Visibility toolbar > Show Solids in all Mechanism-Editors

MUST also be enabled:

- Visibility toolbar> Show Solids in Mechanisms 51

OPTIONS:

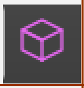
- Visibility toolbar > Solids Display Mode: 50 Shaded, Wire-Frame, or Shaded AND Wire-Frame 50
- Visibility toolbar > Model as Orthonormal or Perspective View 53

1.2.5.7 Orthonomal or Perspective Projection

Visibility toolbar > Orthonormal or Perspective Projection

NOTE - there is ***not*** a **Visibility** **menu**

Use this command to switch the view of **SOLIDS** between **Orthonormal Projection** to **Perspective Projection**.

	**Visibility toolbar > Show Solids with Orthonormal Projection**
	**Visibility toolbar > Perspective Projection**

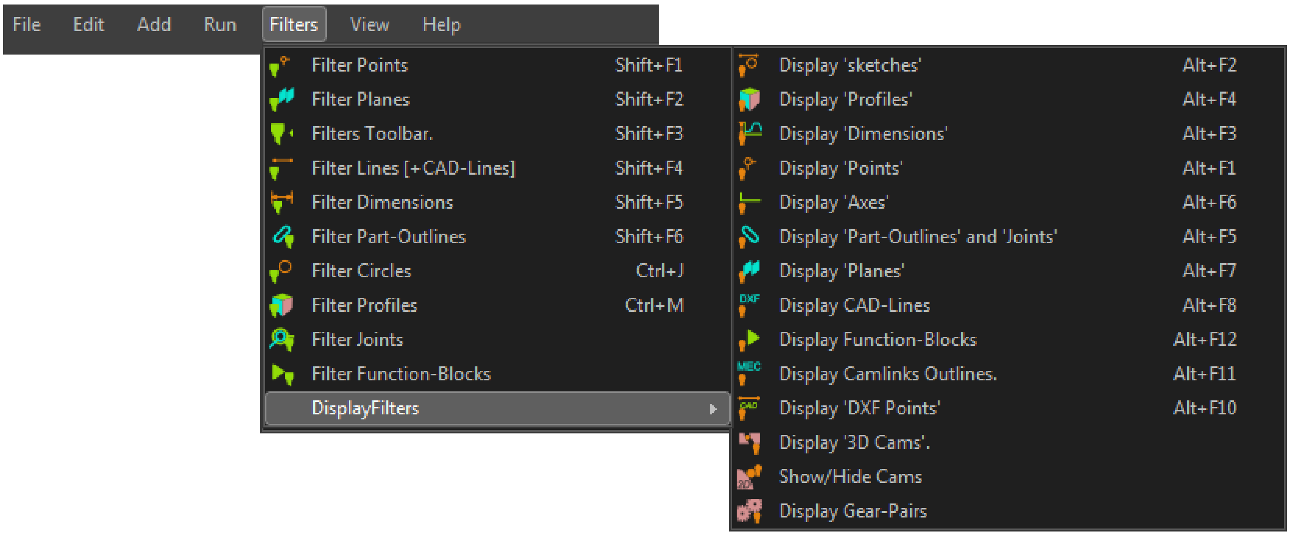
1.2.6 Filters

Filter and Display menus and toolbars

These toolbars are BELOW the graphic-area.

- Filter toolbar - to help you select different types of element.
- Display Elements - to show or hide different types of element.

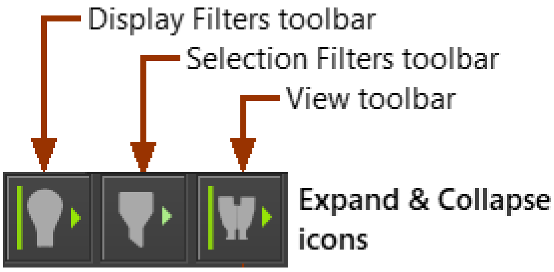
Filter and Display menus



Display Filters & Selection Filters menus

Filter, Display, (and View) toolbars

The Display Filters, Selection Filters (and View) toolbars are BELOW the graphic-area.



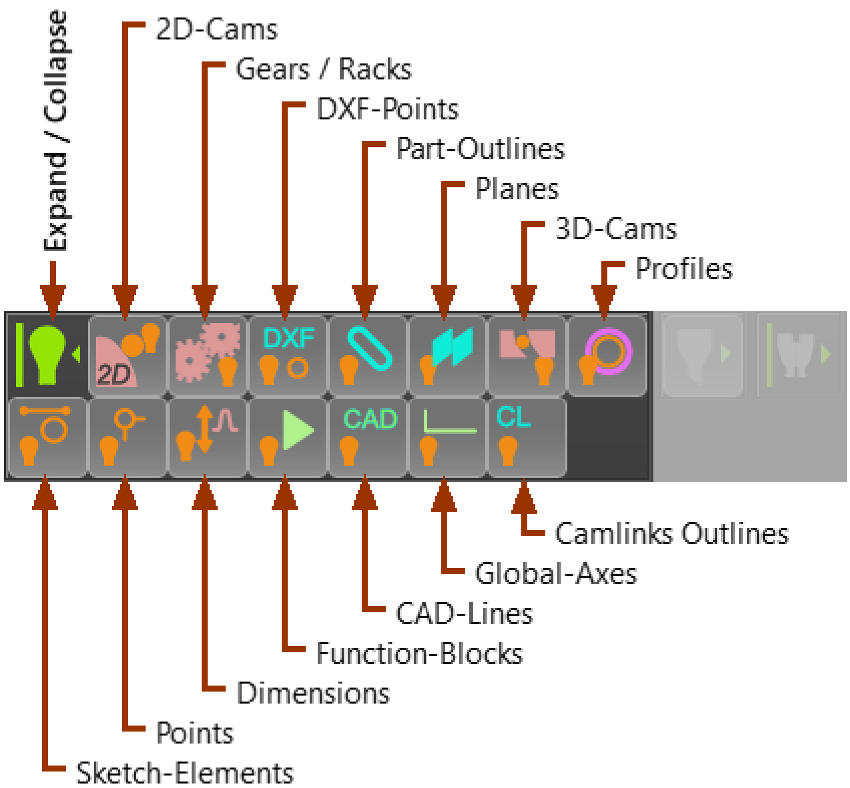
Display Filters & Selection Filters (and View) toolbars - collapsed

1.2.6.1 Display Filters

Display Filters toolbar

The Display Filters toolbar is BELOW the graphic-area.

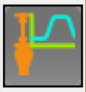
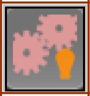
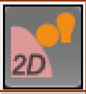
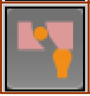
Default: All Display filters are SELECTED to display all element-types.



Display Filters toolbar
[Default: All Filters Selected to Display ALL element-types]

Display Filter icons

	Show or Hide PLANES from the graphic-area. **Note**: An active **MECHANISM-EDITOR** shows only those PLANES that are its 'children'.
	Show or Hide **MODEL AXES** from the graphic-area.
	Show or Hide PART-OUTLINES and Joints from the graphic-area.
	Show or Hide POINTS from the graphic-area.
	Show or Hide **SKETCH-ELEMENTS** AND **PART-AXES** from the graphic-area.
	Show or Hide **PROFILES** from the graphic-area.
	Click to hide CAD-Lines, all DXF Outlines and **CAD-SOLIDS** from the graphic-area.
	Show or Hide **DXF POINTS** in **DXF-DRAWINGS** from the graphic-area. Other **DXF ENTITIES** stay in the graphic-area. **DXF POINTS** show at the ends of DXF Entities in a **DXF-DRAWING** that you import. If there are many DXF Points, they can hide the **DXF-DRAWING**.
	Show or Hide **CAMLINKS OUTLINES** from the graphic-area. **CAMLINKS** was a 2D kinematics software package.
	Show or Hide FUNCTION-BLOCKS, INPUT-CONNECTORS AND OUTPUT-CONNECTORS from the graphic-area.

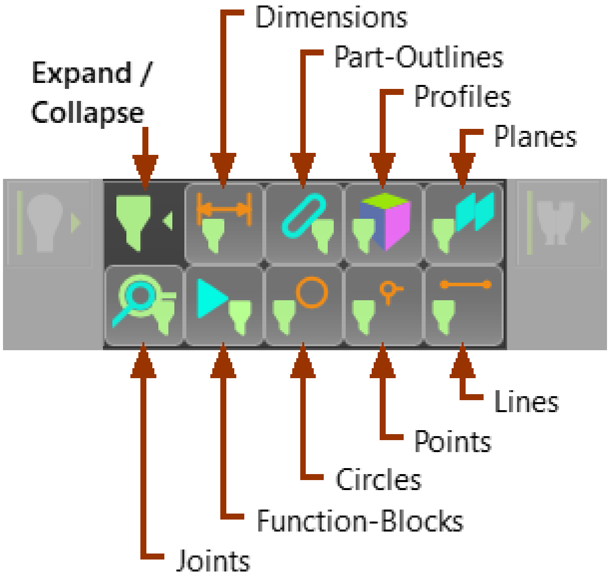
	Show or Hide: • MOTION-DIMENSIONS and MOTION-DIMENSION FBS • DIMENSIONS
	Show or Hide GEAR-PAIRS
	Show or Hide 2D CAM
	Show or Hide 3D-CAMS

1.2.6.2 Selection-Filters

Selection Filters toolbar

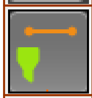
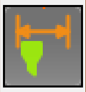
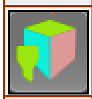
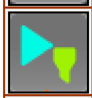
The **Selection Filters toolbar** is **BELOW** the graphic-area.

Special Case and default: Disable all Filters to select all element-types.



Selection Filters toolbar
[All filter elements selected]

Selection Filter icons

	Filter for PLANES
	Filter for **PART-OUTLINES** (to select a PART).
	Filter for POINTS
	Filter for LINES, CAD-LINES, **X-AXES**, and **Y-AXES** of a PART.
	Filter for CIRCLES and ARCS
	Filter for Dimensions: • Dimensions [222] in the **PART-EDITOR** • MEASUREMENT DIMENSIONS [166] and MEASUREMENT FBS in the **MECHANISM-EDITOR** • MOTION-DIMENSIONS [150] and MOTION-DIMENSION FBS in the **MECHANISM-EDITOR**
	Filter for PROFILES
	Filter for FUNCTION-BLOCKS
	Filter for JOINTS

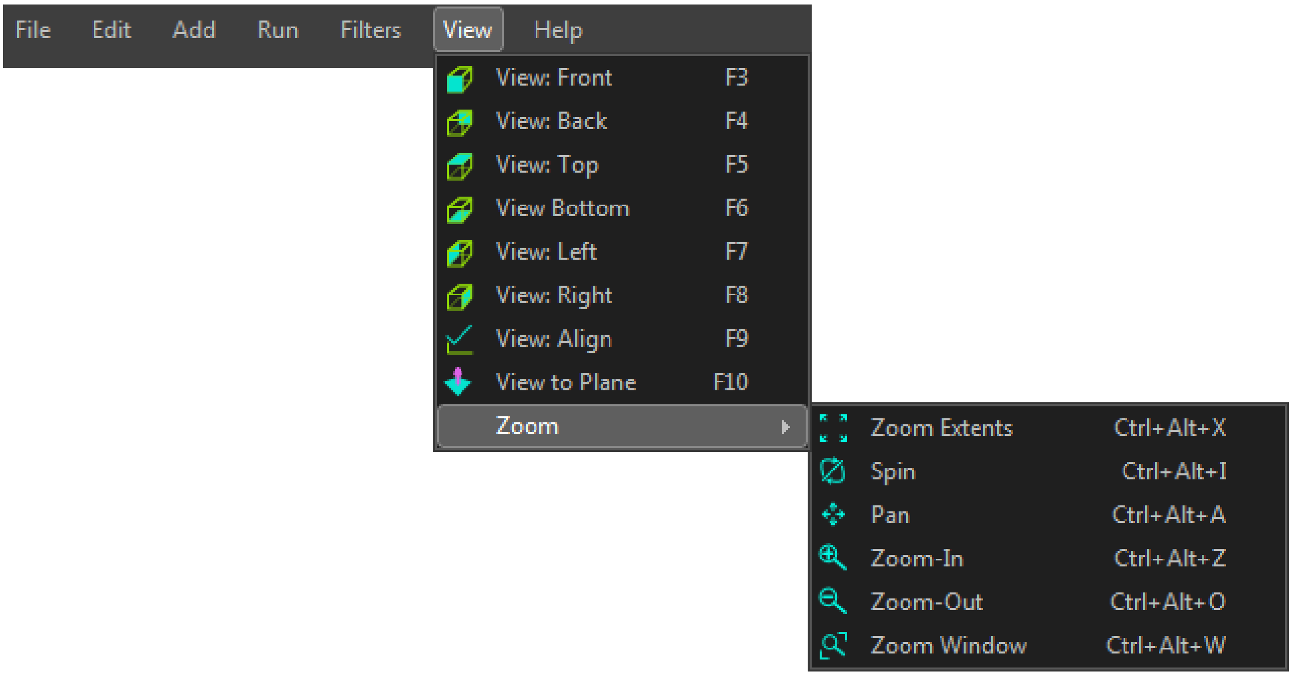
1.2.7 View

View toolbar and menu

The View toolbar is BELOW the GRAPHIC AREA.

It has the typical View options of CAD software.

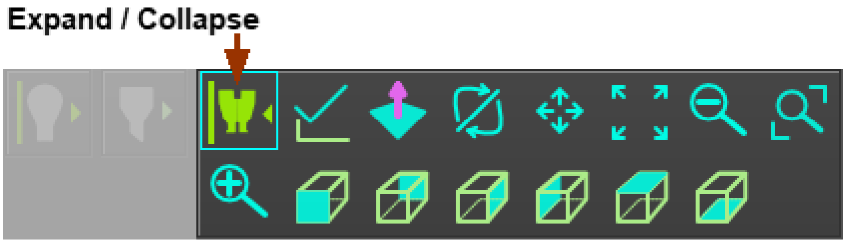
View menu



View menu

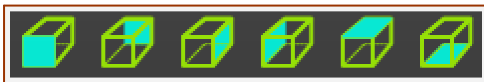
View toolbar

The View toolbar is BELOW the graphic-area.



View toolbar

View toolbar icons



View from the: **Front ; Back ; Left ; Right ; Top ; Bottom**

Front-View is the **XY-Plane** of the active MECHANISM-EDITOR

F3 key on your keyboard is the s short-cut key for the **Front-View**

TOP-TIP: **Clipped Models** - if part of the model is large, it may not display correctly. The model is **clipped.**

If the model is **clipped**:

1. Click Front-View (F3)
2. Click Zoom-Extents

View Align

- **PART-EDITOR** : Click **View Align** to view the **PART** with a horizontal **X-axis.**
- **MECHANISM-EDITOR** : Click **View Align** to view the **BASE-PART** with a horizontal **X-axis.**

View Normal to a Plane

To make the view normal(⊥) to a **PLANE**:

1. Click **View Normal to Plane**
2. Click a **PLANE**
3. Click ✓ in the **COMMAND-MANAGER**

SPIN

Spin button + **Mouse-Button-Down** + move your mouse-pointer to Spin the model.

Shortcut: - not usually recommended

SHIFT key + **Mouse-Button Down** + move your mouse-pointer to Spin the model.

Spin Shortcuts with keyboard arrow-keys - recommended

10º Spin : **LEFT / RIGHT & UP / DOWN** keyboard arrow-keys.

30º Spin : **CTRL** + **LEFT / RIGHT & UP / DOWN** keyboard arrow-keys.

90º Spin : **SHIFT** + **LEFT / RIGHT & UP / DOWN** keyboard arrows-keys.

PAN

Pan button + **Mouse-Button-Down** + **move your mouse-pointer** to **Pan** the model.

Shortcut:

CTRL key + **Mouse-Button Down** + **move your mouse-pointer** to Pan the model.

Notes:

With practice, you can also use your middle-mouse-wheel to Zoom Out then Zoom-In.

If the model is a large, wait for the model to move when you move your mouse-pointer.

Zoom Extents

To display the complete model.

If the complete model does not show, do **Front-View(F3),** then **Zoom-Extents**.

Zoom In

SHORTCUT: Mouse Wheel: Zoom in/out at your mouse position.

Zoom Out

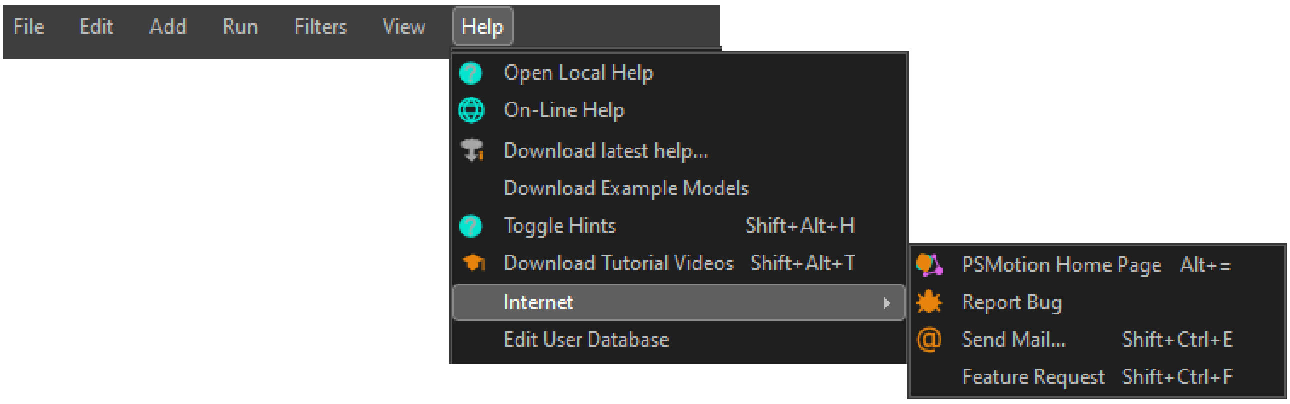
SHORTCUT: Mouse-Wheel: Zoom in/out at your mouse position.

Zoom Windows

With your mouse, drag a window for the new model view.

1.2.8 Help

Help menu



Help menu commands

Open Local Help

The **Local Help** is copied to **<CommonAppData>\ Help ** when you install **MechDesigner**.

To get contextual help:

- Click **Help menu > Open Local Help**, **AND/OR**
- Click the ? icon in a dialog [276], **AND/OR**
- Press the **F1** key when you move your mouse over a command icon.

On-line Help

You need an Internet connection.

We open for you the **MechDesigner's Help** Welcome Page.

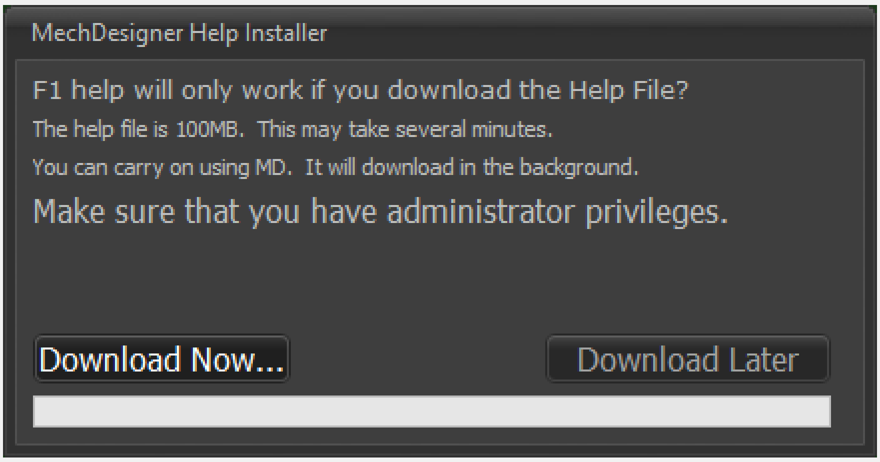
The On-Line help has many tutorials.

Download Latest Help...

You **MUST** have Administrator Rights

Download the Local Help to make sure you have the latest version.

1. Click **Help menu > Download Latest Help...**



2. Click **Download Now** button

Download Example Models

1. Click a model from the list of Examples

Wait for the model to download.

Some models are large. Please be patient to make sure the file downloads.

To show the **SOLIDS** that are in the model, click:

- Edit toolbar > Auto Update / Rebuild [37]
- Visibility toolbar > Show Model in Mechanisms [51] to show **SOLIDS**
- Filters menu > Display Filters > Show/Hide Planes [54] to hide **PLANES** that may hide other elements in the model.

Toggle Hints

Use to show tool-tips next to your mouse-pointer.

1. Click to **Toggle Off** (You may need to click this two(2) times)
2. Click again to **Toggle On**

See also **Application Settings** [289]

Download Tutorial Videos:

See **Tutorials dialog** [507]

Internet sub-menu [62] >

About dialog tabs:

MechDesigner - 16.1.54: (Release.Version.Build)

Graphics - your Graphics Card Supplier, OpenGL Version, Graphics Card Model, OpenGL Extensions

Type-Libraries: **Last used** version of **SOLIDWORKS** and installed version of **SOLIDWORKS** Type-Libraries

Protection Key - Your License Settings

See **About dialog** [63]

1.2.8.1 Internet menu

Help menu > Internet

You need an internet connection.

PSMotion Home Page

A link to: *www at psmotion dot com* in your web-browser.

Report Bug, Send Mail, Feature Request

Report Bug / Feature Request

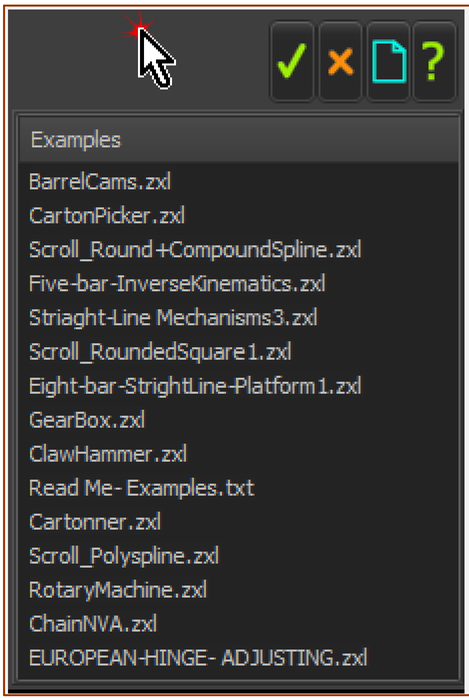
A Link to login to your account at www.psmotion.com/user

1. Login to your account.
2. Click on 'My Account' if necessary
3. Click 'Report a Bug' in the black menu, at the top of the screen (just below the main header).
4. At the top of the page, below the text 'Report a Bug'... Click 'Create new ...'.
5. Enter a title for the bug report
6. Enter a description of the bug
7. Scroll down to the bottom of the page
8. Use the two(2) Browse buttons to find and click on the CXL and MTD files that have the bug
9. Use the Upload buttons to upload them to the PSMotion server.
10. Click ***Save***

Send Mail

You can also send an email to '*support at psmotion dot com*'.

Download Example Models

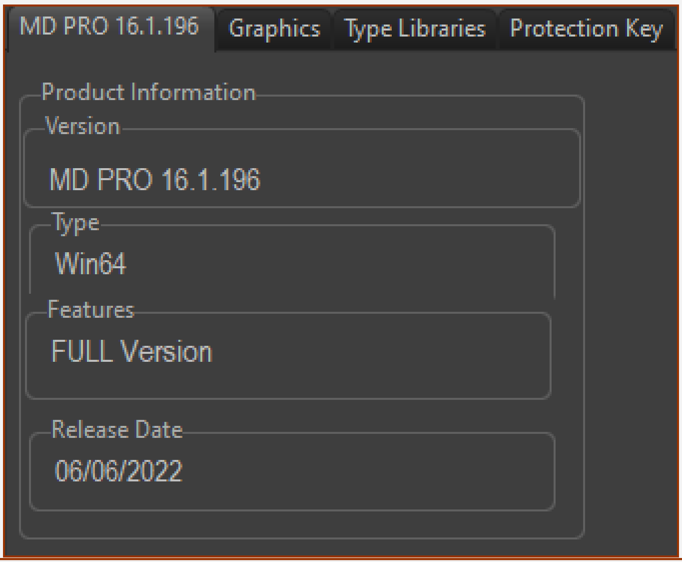


1.2.8.2 About

Help > About dialog

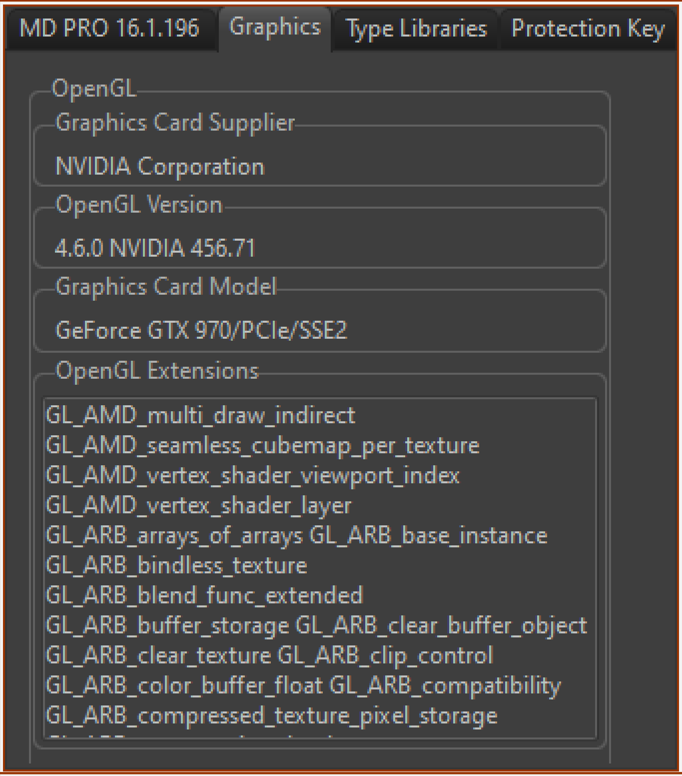
There are four tabs:

MD XXX V-R-B tab



Version: **MechDesigner XXX V.R.B**	
XXX = PRO - Professional ; STD = Standard ; TRL = Trial	
V. R . B : Version - Release - Build	
Type :	Win64
Features:	FULL Version
Release Date:	dd-mm-yyyy

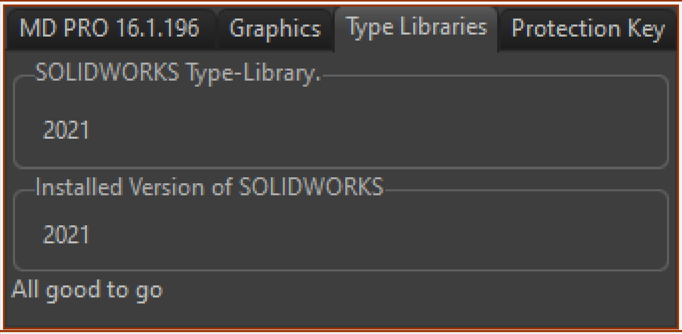
Graphics tab



Graphics Card Supplier:	For example: NVIDIA Corporation

OpenGL Version:	For example: 4.5.0 NVIDIA 361.91
Graphics Card Model	For example: GeForce GTX 970/PCIe/SSE2
OpenGL Extension	For example: GL...AMD_multi_draw_indirect, GL_AMD,_.... and others.

Type Library tab



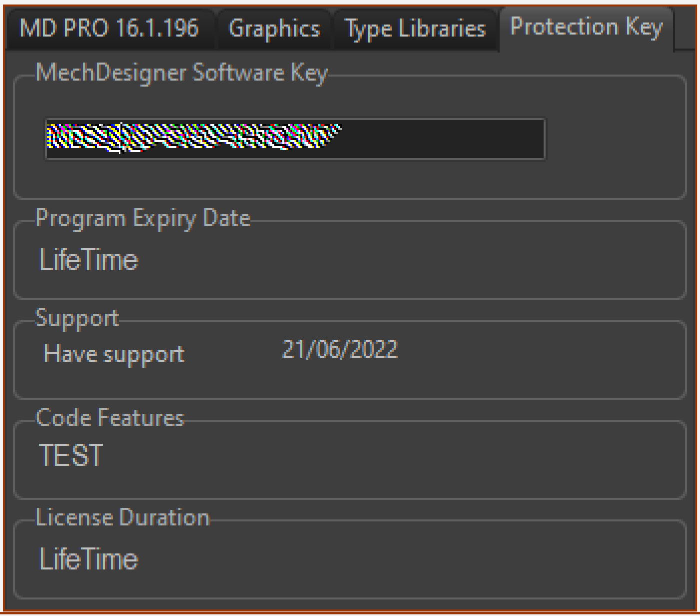
SOLIDWORKS® Type-Libraries / Installed Version of SOLIDWORKS
Each version of **SOLIDWORKS** uses a different Type Library file.
In the image above :

- **SOLIDWORKS Type Library:** 2021
- **Installed Version of SOLIDWORKS**: 2021

When the year is the same, you are **All good to go**.
If the versions are not the same, then **Run as Administrator** when you start **MechDesigner**, and check again.

Note: From MD15 - the correct Type Libraries are installed automatically with your installation, and it checks which version of **SOLIDWORKS** you are using each time you start **MechDesigner**.

Protection Key



MechDesigner Software Key	Mdxx_x_x-xxxx-xxxx-xx
Program Expiry Date	Lifetime / Expiry Date
Support	Have Support + Expiry date / Without Support
Code Features	PREM, STAN, LITE (TEST (code testing only))
License Duration	Lifetime / Annual

1.3 2.1 Model-Editor:

❷ Model-Editor

The **MODEL-EDITOR** is the view of the General-Assembly for your model. There is only one **MODEL-EDITOR**.

The **Reference Triad** shows the **Global XYZ-axes**.

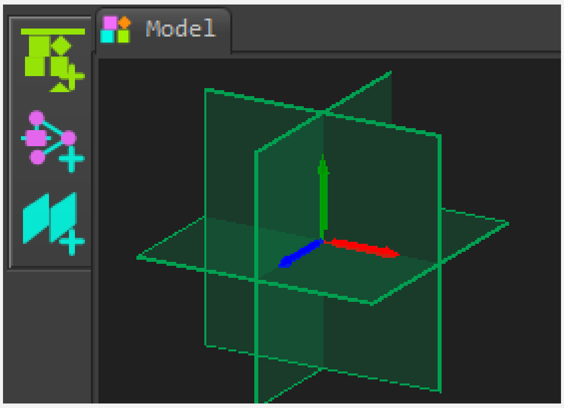
The direction of each axis uses the **right-hand rule**. The:

- thumb finger points to the positive X direction
- index finger points to the positive Y direction
- middle finger points to the positive Z direction, normal to the X and Y directions

There are three(3) fixed PLANES. They are the:

- **Front** PLANE, which is co-planar with the **Global** XY-axes
- **Top** PLANE, which is coplanar with the **Global** ZX-axes
- **Right** PLANE, which is coplanar with the **Global** YZ-axes

Model-Editor workspace



Model-Editor

Graphic-Area : shows the fixed PLANES (and other PLANES that you add to the model), the **Reference Triad,** and the SOLID elements that you add to the model.

Model toolbar : this toolbar is to the left of the graphic-area

Name-tab : the default name of the Model-Editor is **Model**

Menu and Toolbar:

The Model elements toolbar has two commands that you can use in the **MODEL-EDITOR**.

The two commands are also in the Add menu.

Model elements toolbar [66] - Add Plane, and Add Mechanism-Editor.

You can ad PLANES that are offset from one of the three fixed PLANES - **see** **Add Plane** [70]

You must add a minimum of one **MECHANISM-EDITOR** to a PLANE - **see** **Add Mechanism** [67]

You do most of your modelling in **MECHANISM-EDITORS**.

See also:

Model Options dialog [281]

1.3.1 Model elements

Model elements

The commands in **Model elements** arrange the layout of the machine.

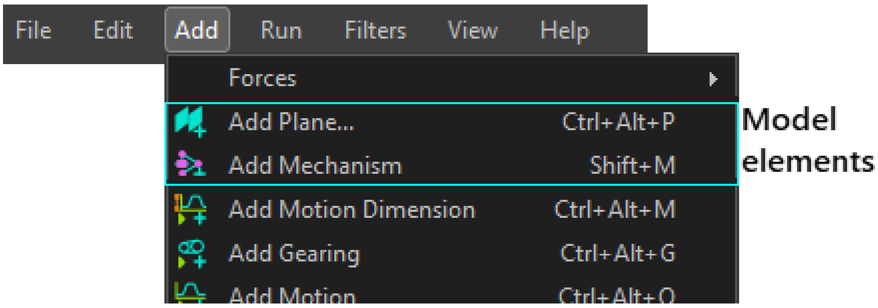
There are two commands.

- Add a new PLANE that is offset from any the three fixed PLANES - **see Add Plane** [70].
- Add a new **MECHANISM-EDITOR** to a PLANE - **see Add Mechanism** [67]*.

* A minimum of one **MECHANISM-EDITOR**. You do most of your modelling in **MECHANISM-EDITORS**.

Model elements menu

The **Add menu** has two commands you can use the in the **MODEL-EDITOR**.

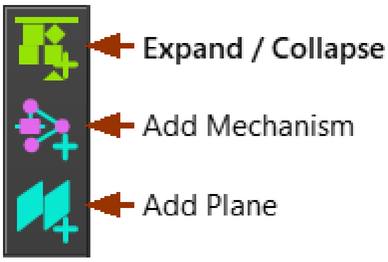


Model elements menu

Model elements toolbar

The **Model elements toolbar** is to left of the graphic-area.

It has two commands.



Model elements toolbar

1.3.1.1 Add Mechanism [Editor]

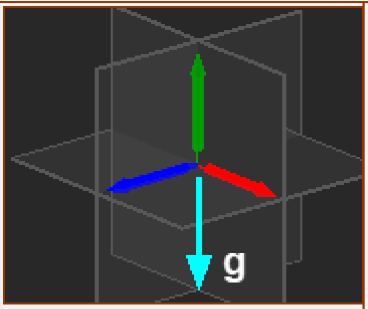
What is a Mechanism?

A **Mechanism**, in the context of **Add Mechanism**, is a **MECHANISM-EDITOR** with a **NAME-TAB**.

You add **MECHANISM-EDITORS** to different **PLANES** to model kinematic-chains (mechanisms, linkages), to design your machine.

You **must** use the **MODEL-EDITOR** to add a minimum of one(1) **MECHANISM-EDITOR.**

IMPORTANT: Gravitational Vector (g = 9.81665 m/s/s)



GRAVITY VECTOR is in the negative Y-axis direction (down the from the **Green Arrowhead** in the **MODEL-EDITOR**.

Before you add the **first** **MECHANISM-EDITOR**, to a **PLANE**, consider the type of machine.

For example, if your machine has a horizontal indexing or rotating table, add the first **MECHANISM-EDITOR** for the table to the **TOP PLANE**, or a **PLANE** that is parallel to the **TOP PLANE**.

Add Mechanism

Model elements toolbar

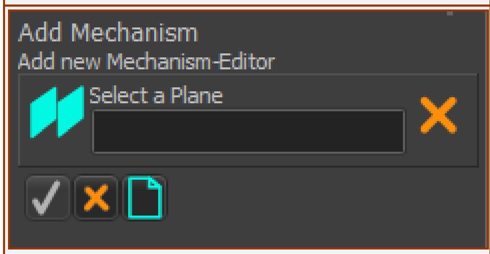
STEP 1: Start the Add Mechanism command:

2. Click **Add menu > Add Mechanism**

OR

2. Click Model elements toolbar [66] > **Add Mechanism**

The **COMMAND-MANAGER** has one selection-box.



Command-Manager - no elements

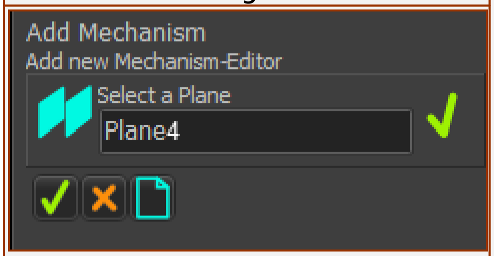
STEP 2: Select a Plane

1. Click a **PLANE** in the graphic-area

OR

2. Click a **PLANE** in the **ASSEMBLY-TREE**

The **PLANE** is now in the selection-box.



Command-Manager - with Plane element

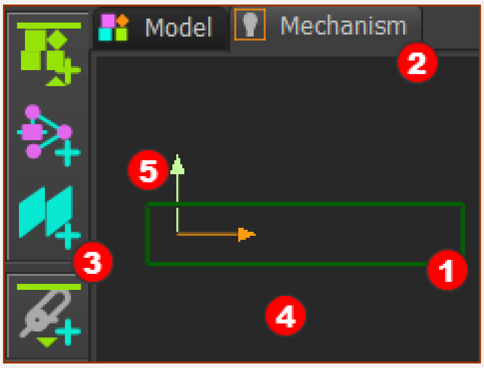
STEP 3: Complete the command

1. Click ✓ in the **COMMAND-MANAGER**

RESULT

You **jump** immediately to the new **MECHANISM-EDITOR**.

RESULT: the new Mechanism-Editor



Front-View of the Mechanism-Editor

Front-View of the **new** MECHANISM-EDITOR.

❶ : BASE-PART - the frame of the Mechanism-Editor. It has a rectangular **Green** PART-OUTLINE

(**See also: The Base-Part does not show correctly 68 ?)**

❷ : MECHANISM NAME-TAB

❸ : TOOLBARS (left and right of the graphic-area)

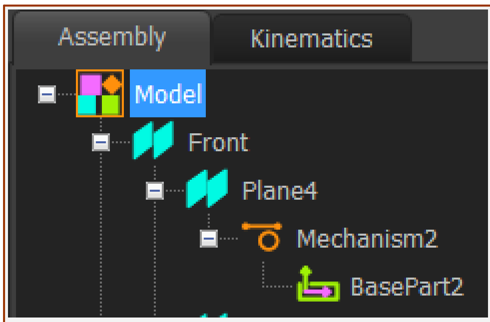
❹ : Graphic-Area

❺ : Local XY-AXES of the NEW MECHANISM-EDITOR and BASE-PART

It use the right hand rule. The:

- +X–axis is horizontal, to the right (>).
- +Y–axes is vertical, and upwards (^).
- +Z–axis is towards you (◎)

NEW Mechanism-Editor - and the Assembly-Tree



Assembly-Tree and the new Mechanism-Editor

The hierarchy of the new elements

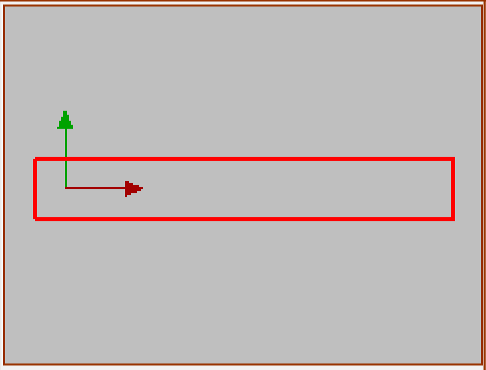
- MECHANISM2 - is a child to PLANE4
- BASEPART2 - is a child to MECHANISM2.

The XY-axes of PLANE4, MECHANISM2, and BASE-PART2 are coplanar and coincident.

Video Double-click for Video of Add Mechanism-Editor to a Plane

The Base-Part does not show correctly!

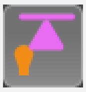
The Base-Part is not Green



Base-Part with Forces ON

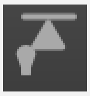
The Base-Part is a Random color.

If the BASE-PART is a not Green:

 Forces toolbar > Force Vectors: Display to display the Force Vectors.

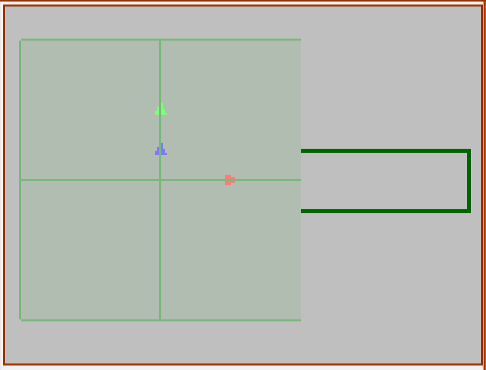
When this icon is enabled to display Force Vectors, each PART is given a random color.

To make the PART Green, you should:

 Click the Force Vectors: Display to not display the Force Vectors

When the icon is disabled the BASE-PART should be Green.

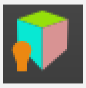
I cannot see the Base-Part properly



Base-Part partially hidden by a Plane.

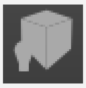
The Base-Part is hidden by a Plane.

This is because:

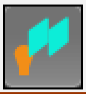
 Visibility toolbar > Show Solids in Mechanisms is enabled

When you enable Show SOLIDS in MECHANISM-EDITORS, we also show the PLANES.

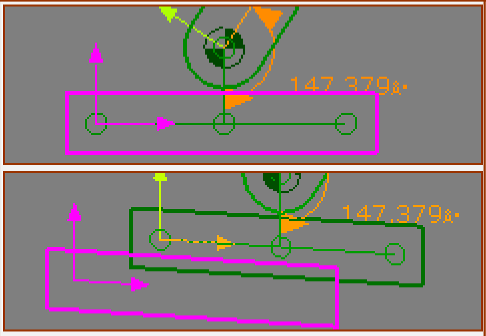
To hide the PLANES:

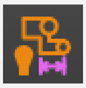
 Click Visibility toolbar > Show Solids in Mechanisms disabled.

OR

 Click Display toolbar > Display Planes to Hide Planes 54

The Base-Part is Pink



 Visibility toolbar > Show other Kinematic and Sketch elements

If you enable this, and another MECHANISM-EDITOR is in front of the active MECHANISM-EDITOR, you see the Pink* **Other Base-Part** (* default color)

Spin the model to see the BASE-PART of the active MECHANISM-EDITOR.

* See Application-Settings > Graphics tab > Display Colors > Background Sketch

1.3.1.2 Add Plane (Model-Editor)

What is a Plane?

A **PLANE** is a flat surface, defined by its Local XYZ-axes. The XY axes are coplanar with the **PLANE**. The Z-axis follows the usual Right-Hand-Rule.

You add **PLANES** to arrange the layout of your machine.

You can add **MECHANISM-EDITORS** to **PLANES**.

Add Plane (Model-Editor)

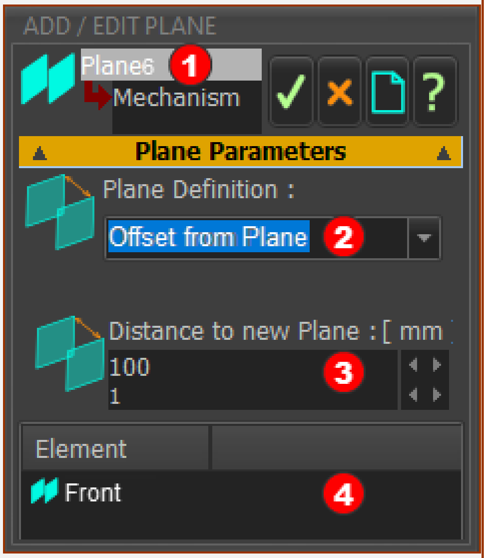
In the **MODEL-EDITOR**, a new **PLANE** is offset along the **Z-axis** of another **PLANE**.

STEP 1: Start the Add Plane command

Add menu > Add Plane

OR

Model elements toolbar [66] > Add Plane



FORMAT 1: Add Plane to Plane

The Add Plane dialog opens, immediately ❶

In the **MODEL-EDITOR**, its format is>

❶ **Name** - of new **PLANE**

PLANE PARAMETERS

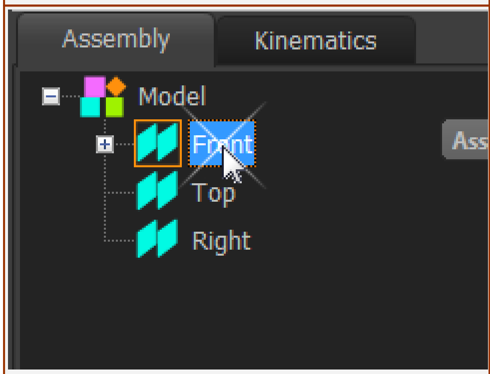
❷ **Plane Definition** for the new **PLANE** :

OFFSET FROM PLANE (always in the **MODEL-EDITOR**)

❸ **Parameter to define the new Plane :**

DISTANCE TO NEW PLANE

❹ **Element box -** the reference element for new **PLANE**



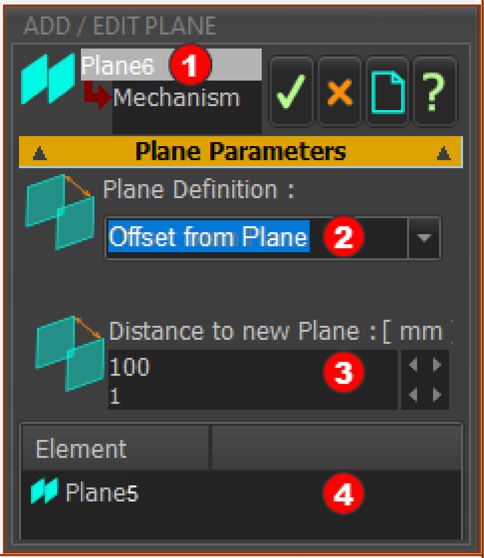
Click Plane in the Assembly-Tree

STEP 2: Click a Plane.

Click a **PLANE** in the graphic-area or the **ASSEMBLY-TREE**.

When you click a **PLANE**, it shows in the **Element box** ❹ of the **ADD PLANE DIALOG**

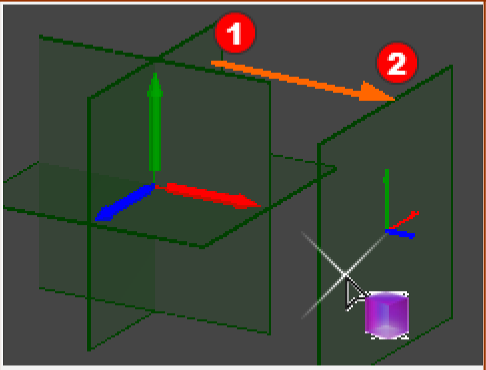
A preview of the new **PLANE** is also in the graphic-area immediately.



STEP 3: Edit the Parameter❸

Edit **DISTANCE TO NEW PLANE**❸

DISTANCE TO NEW PLANE from the **PLANE** in the **Element-box**❹ (the reference **PLANE**)



New Plane in the Graphic-Area

STEP 4: Complete the Command

Click ✅ in the **ADD-PLANE DIALOG**

RESULT

The new **PLANE**❷ is in the graphic-area and **ASSEMBLY-TREE**.

The new **PLANE** has its own **Local XYZ-axes**.

Note:

If you cannot see the new **PLANE** try:

View menu > Zoom-Out

View menu > Spin model 58

Filters menu (or Display toolbar) > Display Planes 54

Videos: Double-click to watch Video

1.4 2.2 Mechanism-Editor:

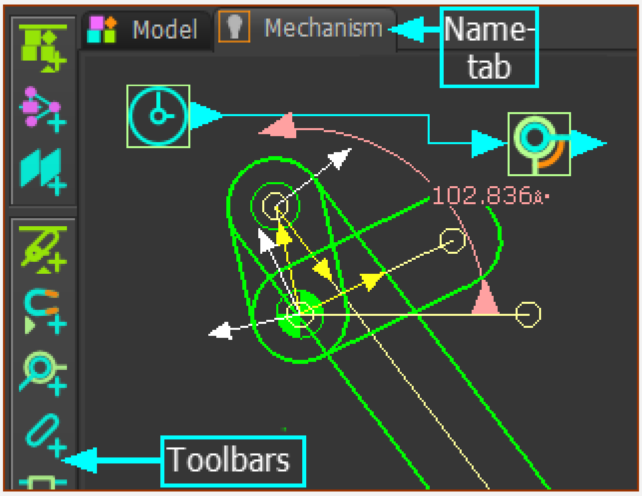
❷ Mechanism-Editor

You **MUST** add a minimum of one **MECHANISM-EDITOR** to a **PLANE** that is in the MODEL-EDITOR [65].

See Add Mechanism-Editor [67] - in MODEL-EDITOR.

See Add Mechanism-Editor [74] - in MECHANISM-EDITORS.

Mechanism-Editor workspace



Mechanism-Editor- Example

Graphic-Area : to add Kinematic and Machine elements, Kinematic and Modelling Function-Blocks, Solids and Force elements, ...

Toolbars : to the left, right, and above the graphic-area **- see Menus and Toolbars, below**

Name-tabs : each **MECHANISM-EDITOR** has a name tab.

Menus and Toolbars:

Toolbars provide the commands that you use in **MECHANISM-EDITORS**.

The commands are also in the **Add menu**.

Model elements toolbar [73] - left of the graphic-area - add **PLANES** and add **MECHANISM-EDITORS** to **PLANES**.

Kinematic elements toolbar [80] - left of the graphic-area - add the basic kinematic elements to build kinematic-chains.

Machine elements toolbar [108] - left of the graphic-area - add more complex functional parts to your model.

Kinematic Function-Blocks toolbar [142] - right of the graphic-area - plan and measure the motion of each kinematic-chain.

Modeling Function-Blocks toolbar [176] - above the graphic-area - tools that help you do more complex modelling.

Force elements toolbar [187] - right of the graphic-area - measure force, torque, and power that is required to drive each kinematic-chain.

Solid element toolbar [199] - above the graphic-area - show kinematic elements as solid model elements.

1.4.1 Model elements

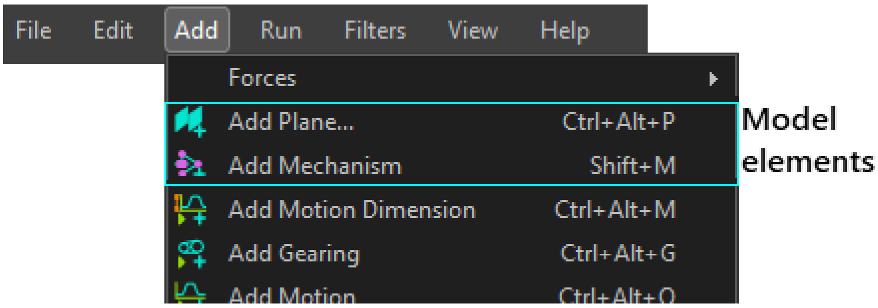
Model elements

Use **Model elements** to arrange the layout of the machine.

The **Add Model elements commands** are active in the **MODEL-EDITOR** ***and*** all **MECHANISM-EDITORS**.

Model elements menu

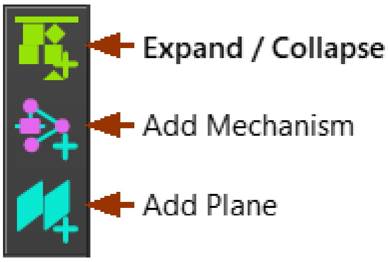
There are two commands.



Model elements : menu

Model elements toolbar

The **Model elements toolbar** is to left of the graphic-area



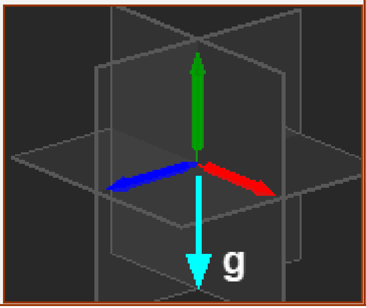
Model elements : toolbar

1.4.1.1 Add Mechanism [Editor]

Mechanism-Editor

A **Mechanism**, in the context of **Add Mechanism**, is a **MECHANISM-EDITOR** with a new **MECHANISM NAME-TAB**.

Use **MECHANISM-EDITORS** to add **PARTS**, **JOINTS**, **CAMS**, **GEARS**, **BELTS**, ... to design your machine.



IMPORTANT: Gravitational Vector (g = 9.81665 m/s/s)

The **GRAVITY VECTOR** is in the **negative Y-axis direction** (Green Arrowhead) in the **MODEL-EDITOR**.

Before you add the **first** **MECHANISM-EDITOR**, consider which **MECHANISM-PLANE** is the best to model your machine.

Add Mechanism-Editor

You add **MECHANISM-EDITORS** to **PLANES**. To see **PLANES**, enable Visibility toolbar >Show Solids in Mechanisms 51.

Model elements toolbar

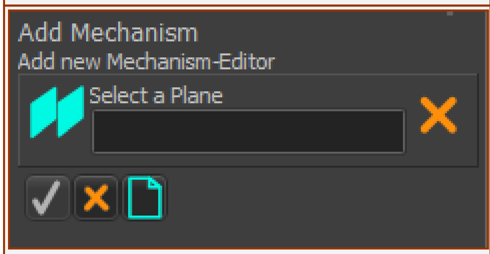
STEP 1: Start the Add Mechanism command:

2. Click **Add menu > Add Mechanism**

OR

2. Click Model elements toolbar 66 > **Add Mechanism**

The **COMMAND-MANAGER** has one selection-box.



Command-Manager

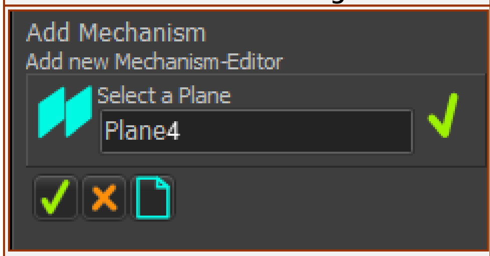
STEP 2: Select a Plane

1. Click a **PLANE** in the graphic-area

OR

2. Click a **PLANE** in the **ASSEMBLY-TREE**

The **PLANE** is now in the selection-box.



Command-Manager - Plane element

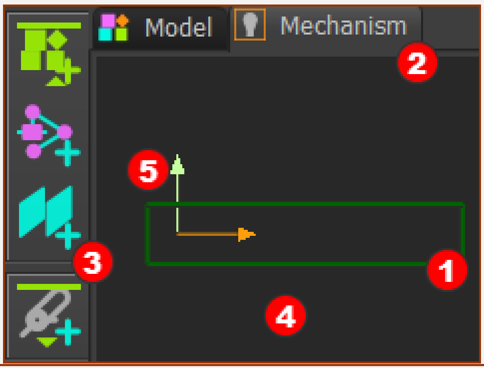
STEP 3: Complete the command

1. Click ✓ in the **COMMAND-MANAGER**

You **jump** immediately to the new **MECHANISM-EDITOR**.

The new **MECHANISM-EDITOR** is a child to the **PLANE** you select in **STEP 2**.

RESULT: the New Mechanism-Editor



Front-View of the new MECHANISM-EDITOR

1 : BASE-PART with a rectangular Green PART-OUTLINE

(**See also: The Base-Part does not show correctly** 75)

2 : Mechanism name-tab

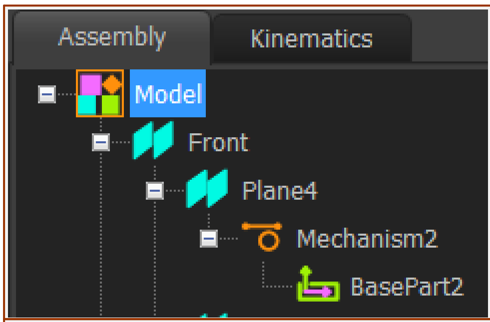
3 : Toolbars (left and right of the graphic-area)

4 : Graphic-Area

5 : Local XY-AXES of the NEW MECHANISM-EDITOR and BASE-PART

- The +X–axis is horizontal, to the right (>)
- The +Y–axes is vertical, and upwards (^)
- The +Z–axis is towards you (◎)

The New Mechanism-Editor - in the Assembly-Tree



The hierarchy of the new elements

- MECHANISM2 - is a child to PLANE4
- BASEPART2 - is a child to MECHANISM2.

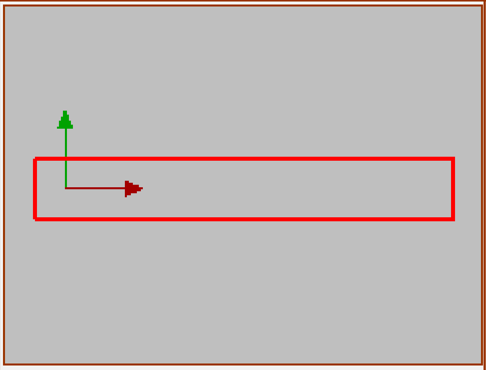
The XY-PLANE of PLANE4, MECHANISM2, and BASE-PART2 are coplanar and coincident.

Video:

Double-click to watch 'Add Mechanism to Plane'

The Base-Part does not show correctly!

The Base-Part is not Green



Base-Part with Forces ON

The Base-Part is a Random color.

If the BASE-PART is a not Green:

Forces toolbar > Force Vectors: Display to display the Force Vectors.

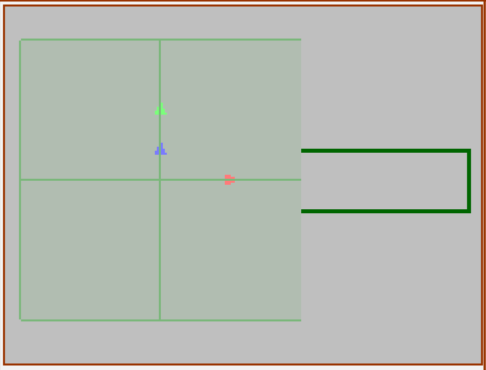
When this icon is enabled, each PART is given a random color.

To make the PART Green, you should:

Click the Force Vectors: Display to not display the Force Vectors

When the icon is disabled the BASE-PART should be Green.

I cannot see the Base-Part properly



Base-Part partially hidden by a Plane.

The Base-Part is hidden by a Plane.

This is because:

Visibility toolbar > Show Solids in Mechanisms is enabled

When you enable Show SOLIDS in MECHANISM-EDITORS, we also show the PLANES.

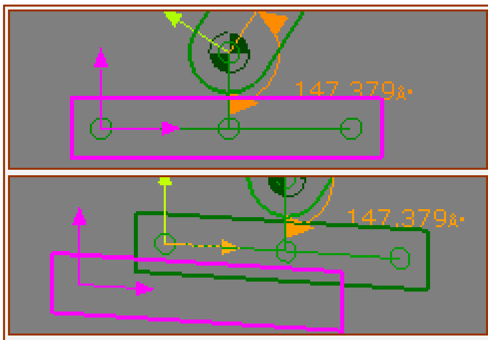
To hide the PLANES:

Click Visibility toolbar > Show Solids in Mechanisms disabled.

OR

Click Display toolbar > Display Planes [54] to hide PLANES

The Base-Part is Pink



Visibility toolbar > Show other Kinematic and Sketch elements

When you enable this, and another MECHANISM-EDITOR is in front of the active MECHANISM-EDITOR, you see the Pink* other BASE-PART (*default color)

Spin the model to see the BASE-PART of the active MECHANISM-EDITOR.

* See Application-Settings > Graphics tab > Display Colors > Background Sketch

1.4.1.2 Add Plane (Mechanism-Editor)

What is a Plane?

A **PLANE** is a flat surface, defined by its Local XYZ-axes. The XY axes are coplanar with each **PLANE**. The **Z-axis** follows the usual Right-Hand-Rule.

Use Visibility toolbar > Show Solids in Mechanisms (51) to see **PLANES** in **MECHANISM-EDITORS**.

See also: Add Plane - Model-Editor (70)

Add Plane (Mechanism-Editor)

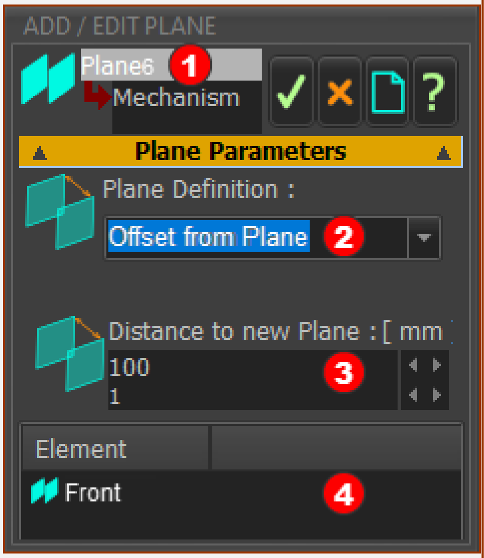
FORMAT 1: Add Plane to a Plane

STEP 1: Start the Add Plane command

Add menu > Add Plane

OR

Model elements toolbar (73) > Add Plane



FORMAT 1: Add Plane to Plane

The Add Plane dialog opens, immediately

In the **MECHANISM-EDITOR**, the default format is:

❶ **New Plane Name** - PLANE6

PLANE PARAMETERS

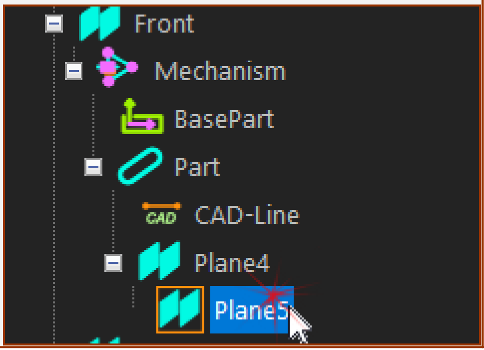
❷ **Plane Definition -** (default format)

OFFSET FROM PLANE (default definition)

❸ **Parameter to define the new Plane :**

DISTANCE TO NEW PLANE

❹ **Element box -** the name of the element(s) that are referenced by the new **PLANE**.

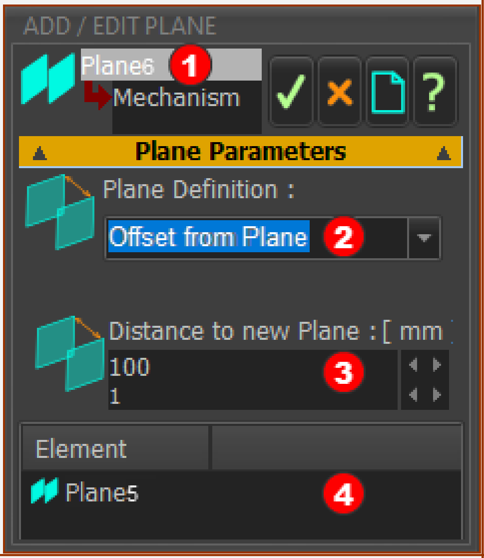


STEP 2: Click a Plane.

Click a **PLANE** in the graphic-area or the **ASSEMBLY-TREE**.

When you click a **PLANE**, it shows in the **Element box** ❹ of the **ADD PLANE DIALOG**.

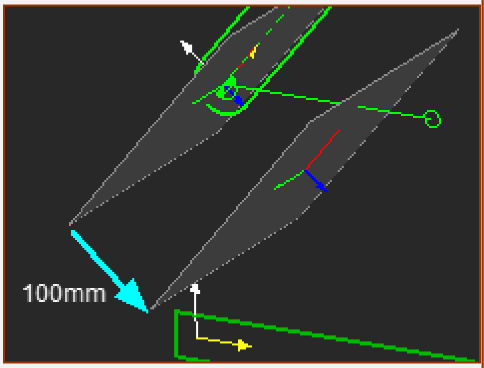
A preview of the new **PLANE** is also in the graphic-area immediately.



STEP 3: Edit the Parameter❸

Edit DISTANCE TO NEW PLANE❸

DISTANCE TO NEW PLANE - to PLANE 6❶ from PLANE5 ❹



STEP 4. Complete the Add Plane command

Click ✓ in the ADD-PLANE DIALOG

RESULT

The new PLANE❷ is in the graphic-area and ASSEMBLY-TREE.

It is on **Local Z-axis** of the PLANE you select to add the new PLANE.

The new PLANE has its own **Local XYZ-axes**.

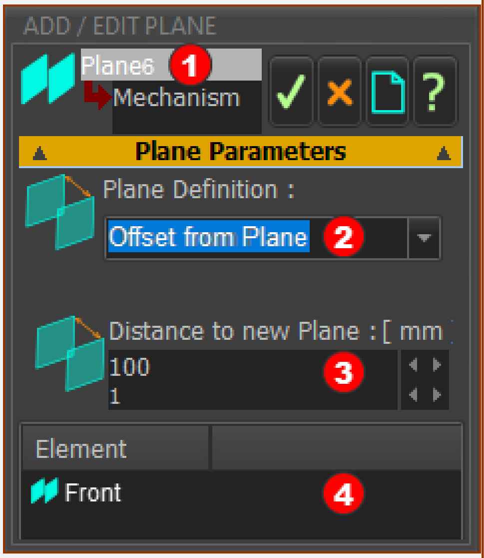
FORMAT 2: Add Plane to a Line (CAD-Line, Part's X-axis or Y-axis)

STEP 1: Start the Add Plane command

Add menu > Add Plane

OR

Model elements toolbar [66] > Add Plane



FORMAT 1: Add Plane to Plane

The Add Plane dialog opens, immediately

In the MECHANISM-EDITOR, the default format assumes you will select a Plane.

❶ **New Plane Name** - PLANE6

PLANE PARAMETERS

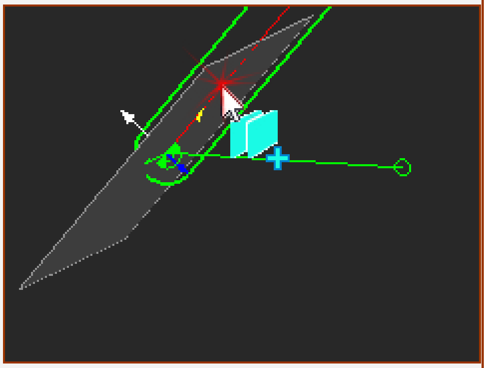
❷ **Plane Definition -** (default format)

OFFSET FROM PLANE (default definition)

❸ **Parameter to define the new Plane :**

DISTANCE TO NEW PLANE

❹ **Element box -** the name of the element(s) that are referenced by the new PLANE.



STEP 2: Click a Line*.

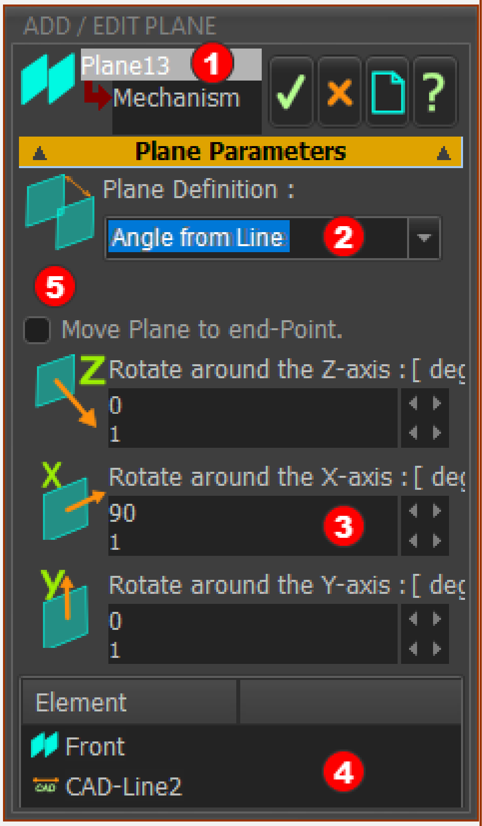
Click a LINE* in the graphic-area

* LINE, CAD-LINE, or a PART'S **X-axis** or **Y-axis**

When you click a LINE, it shows in the **Element box**❹ with the MECHANISM-PLANE (see below).

A preview of the new PLANE is also in the graphic-area immediately.

Note: The default ROTATE AROUND X-AXIS parameter is 90.



FORMAT 2: Add Plane to Line

STEP 3. Edit the parameters ❸❺

❸ ANGLE - rotate the new PLANE around its active X–AXIS

and

❸ ANGLE - rotate the new PLANE around its active Y–AXIS

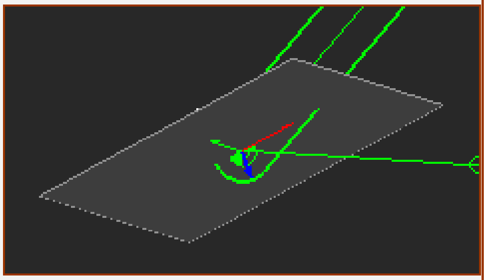
and

❸ ANGLE - rotate the new PLANE around its active Z–AXIS

Optional (for Bevel Gearboxes)

❺ MOVE PLANE TO END-POINT - move the new PLANE to the end of the LINE in the **Element box.**

If you enable MOVE PLANE TO END-POINT, (and the ROTATE AROUND X,Y,Z parameters are zero(0)), the Local X-axis is from the **END**-POINT to the **START**-POINT, the Local Y-axis is on the new PLANE, and the Local Z-axis follows the **Right-Hand-Rule.**



STEP 4. Complete the Add Plane command

Click ✔ in the ADD-PLANE DIALOG

RESULT

The new PLANE❷ is in the graphic-area and **ASSEMBLY-TREE.**

The new PLANE has its own **Local XYZ-axes.**

Video: Double-click to watch Video

1.4.2 Kinematic elements

Kinematic elements

Kinematic elements are coplanar with the **XY-Plane** of the active **MECHANISM-EDITOR***.

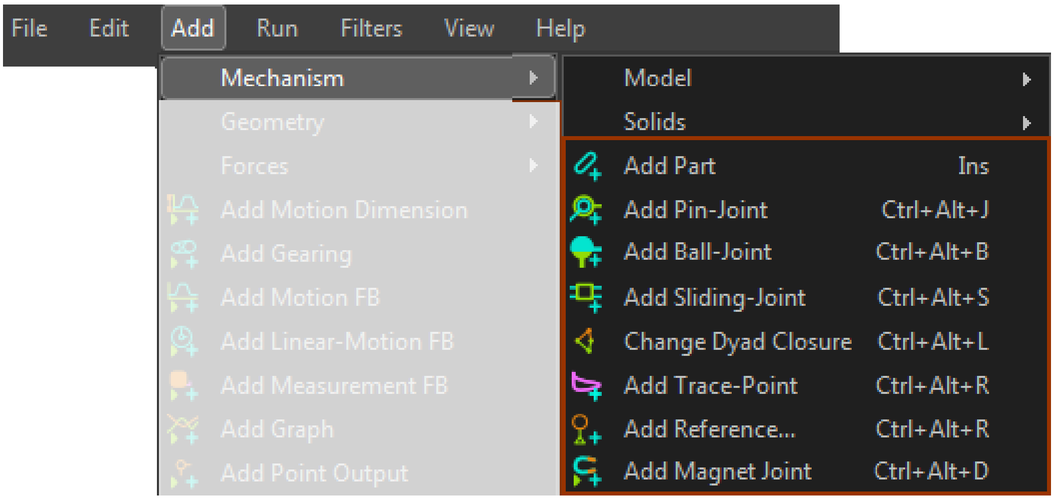
Kinematic elements represent the moving parts in your machine. You join **PARTS** to each other to form kinematic-chains (mechanisms, linkages).

Kinematic elements are NOT solid bodies; and they do NOT have shape or form**.

* An exception is a **PART** with a **BALL-JOINT** at its **start-Point** and **end-Point**.

** To add a solid body to a **PART**, add to it a **PROFILE** and/or import a **SOLIDWORKS** part/assembly document or an STL file from other CAD.

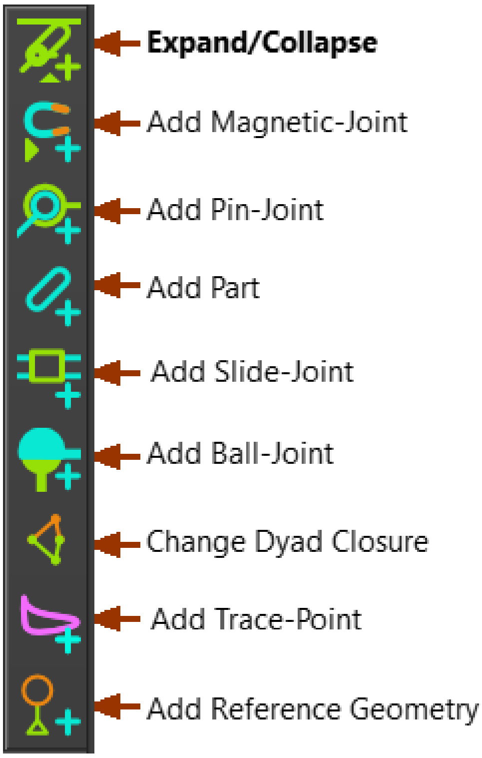
Kinematic elements menu



Kinematic-elements in Add menu

Kinematics elements toolbar

The **Kinematic elements toolbar** is to the left of the graphic-area.



Kinematic elements toolbar

1.4.2.1 Add Magnetic-Joint

Magnetic-Joint

See also: Magnetic-Joint dialog 457 **(YouTube Video : http://youtu.be/IME9Eb4mpQE)**

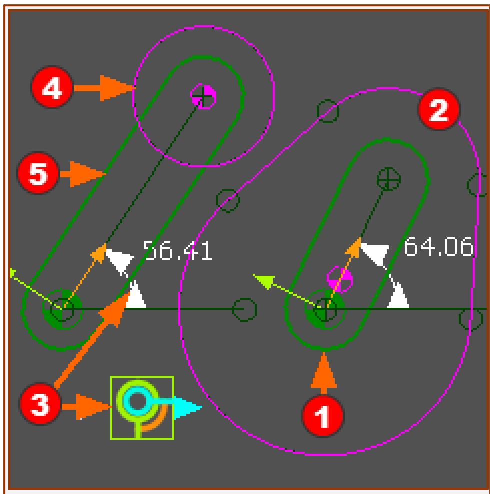
Use Magnetic-Joints ONLY for special modeling - for example, to reverse-engineer the motion of a Cam-Follower from the shape of an imported Cam.

MAGNETIC-JOINTS are solved with iterative, root-finding techniques. All other kinematic joints use closed-form equations for exact and fast evaluations. Therefore, a model with a MAGNETIC-JOINT is ~100s'× slower to solve than a model with only SLIDE-JOINTS, PIN-JOINTS, and BALL-JOINTS. Most models (99%?) do NOT need a MAGNETIC-JOINT.

Terminology

MAGNETIC-JOINT :	A MAGNETIC-JOINT pulls a **circular** PROFILE to be in continuous contact with an **irregular** PROFILE (or CURVE). The **circular** PROFILE and **irregular** PROFILE (or CURVE) are in different kinematic-chains. After you add a MAGNETIC-JOINT, the motions of the kinematic-chains are related by the contact between the **circular** and **irregular** PROFILES (or CURVE).
POINT-CLOUD :	X-Y or R-Θ coordinates. Use a POINT-CLOUD FB to import the **Point-Cloud** coordinates.
CURVE :	The smooth shape that we derive for you from the **Point-Cloud** coordinates.

Preparation for Add Magnetic-Joint (Typical)



The MAGNETIC-JOINT pulls a **Circular-Profile**❹ onto an **Irregular-Profile**❷.

❶ A PART (and kinematic-chain) to which the **Irregular-Profile**❷ is a child.

❷ An **Irregular-Profile**, which is a

- PROFILE that you add to a **sketch-loop**

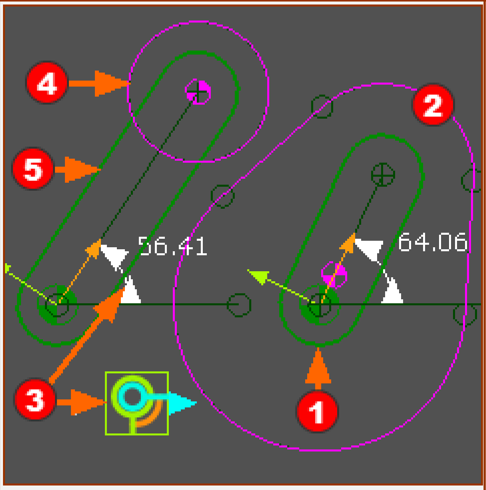
OR

- CURVE that you calculate from a Point-Cloud

❺ A PART (and kinematic-chain) to which the **Circular Profile**❹ is a child.

❸ The MOTION-DIMENSION FB that becomes driven by the action of the MAGNETIC-JOINT.

Add Magnetic-Joint



STEP 1: Pre-Position the Circular Profile

1. **HOME** the model

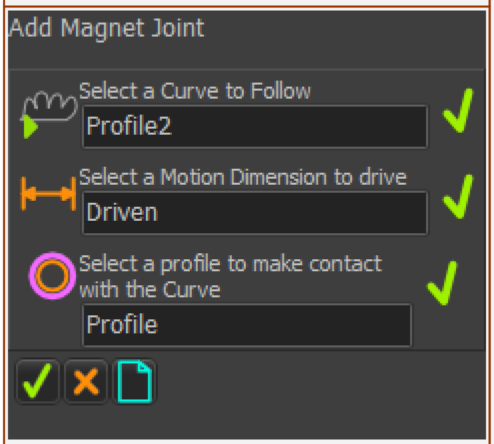
   Select the **Home** key, or use the **ALT+H** keyboard shortcut, or do **Run menu/toolbar** > **Home.**

2. Edit the MOTION-DIMENSION FB❸ | BASE-VALUE so that the:

   Distance from the **Circular-Profile**❹ to the **Irregular-Profile**❷ ...

   ... < **0.5 × Radius of Circular-Profile**❹

Note: See also [Magnetic-Joint dialog > ADVANCED](457) (459)



STEP 2: Start the Add Magnetic-Joint command

1. **Add menu** > **Mechanism sub-menu** > **Add Magnetic-Joint**

OR

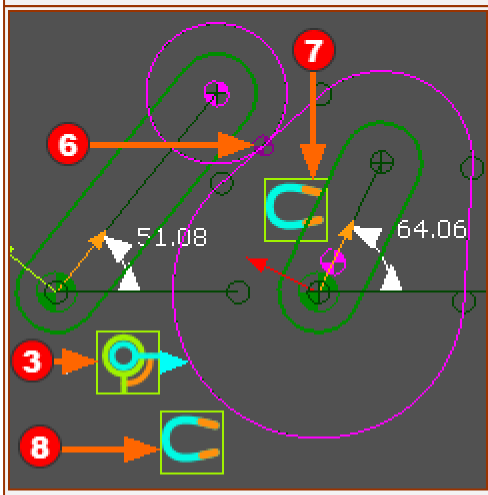
1. **Kinematic elements toolbar** > **Add Magnetic-Joint**

STEP 3: Select the elements:

1. Click the **Irregular Profile**❷
2. Click the MOTION-DIMENSION FB❸ **that you want to 'drive' with the Magnetic-Joint.**
3. Click **Circular Profile**❹

STEP 4: Complete the Command

1. COMMAND-MANAGER: **Click** ✓ **in the** COMMAND-MANAGER.



RESULT: (usually)

The MAGNETIC-JOINT ***pulls*** the **Circular-Profile**❹ to be in contact with the **Irregular-Profile**❷.

A **Point Symbol**❻ is at the contact.

The MAGNETIC-JOINT FB❼ is near to the **Point-Symbol**❻

There is a MAGNETIC-JOINT ICON❽ **below and right** of the MOTION-DIMENSION FB❸ that is driven by the MAGNETIC-JOINT.

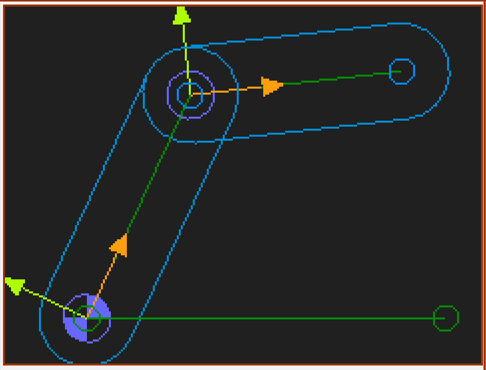
STEP 5: Open the [Magnetic-Joint dialog](457)

Video: Double-click to watch Video

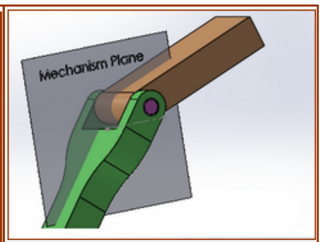
1.4.2.2 Add Pin-Joint

Pin-Joint

See also: **Edit-Pin-Joint**[483] **(Servo-motor and Gearbox Sizing)**


Pin-Joints in MechDesigner


Pin Joint between Connecting-Rod and Piston

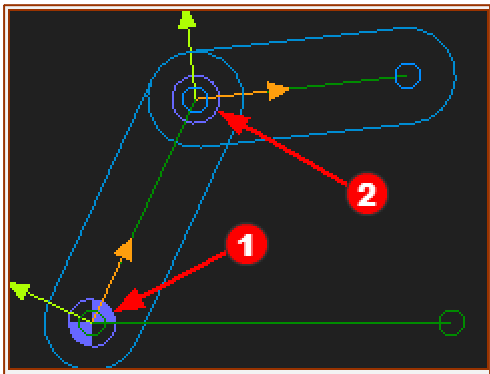


Pin-Joint in SOLIDWORKS

Terminology

PIN-JOINT :	A PIN-JOINT makes a POINT* in a PART to be coincident with a POINT* in a different PART. A PIN-JOINT allows only rotary motion between two PARTS. A PIN-JOINT removes two **degrees-of-freedom** from the model.
rotating-Part :	**derived-name - a rotating-Part** is a PART that you join with a PIN-JOINT to a different PART and can also rotate relative to the BASE-PART.
Revolute-Joint :	The kinematic and equivalent term for a PIN-JOINT.
KINEMATICS-TREE :	In **Dyads**, PIN-JOINTS are represented by the letter **R**, from **Revolute-Joint.**
* POINT, START-POINT, END-POINT, or CENTER-POINT.	

Pin-Joints in the Graphic-Area

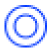


There are two symbols for a PIN-JOINT:

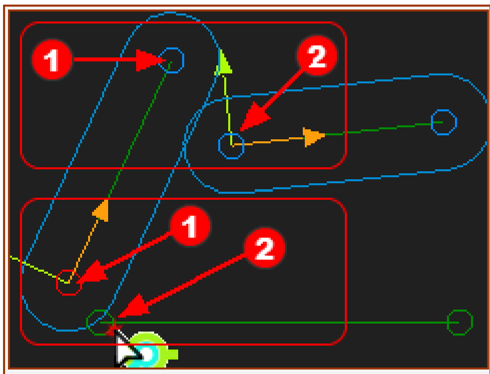
<<< PIN-JOINT❶ between a PART and the BASE-PART.

<<< PIN-JOINT❷ between two(2) added PARTS.

❶  - the symbol for a PIN-JOINT that cannot move relative to the BASE-PART.

❷  - the outer circle is the symbol for a PIN-JOINT that can move relative to the BASE-PART.

Simple Case - Add Pin-Joint



STEP 1: Start the Add Pin-Joint command

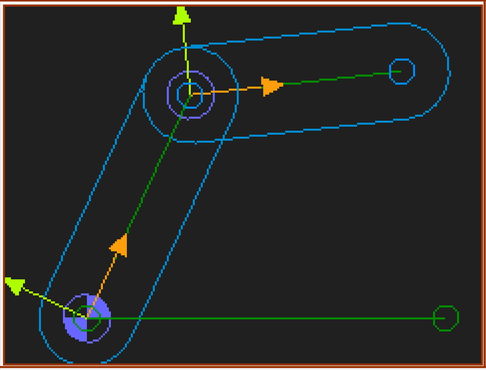
Add menu > Mechanism sub-menu > Add Pin-Joint

OR

 Kinematic elements toolbar > Add Pin-Joint

STEP 2: Select two(2) Points:

At least one POINT must be in a FREE PART.



1. Click a **POINT❶** in a **free PART**
2. Click a different **POINT❷** in a **PART**

The two(2) **POINTS** become coincident at the **PIN-JOINT**.

If the **SELECT-ELEMENTS DIALOG** opens, see Special-Case [86].

See also:

Add Slide-Joint [93]

Add Ball-Joint [97]

Add Motion-Dimension FB [150]

VIDEO : Add Pin-Joint - Simple-Case

DOUBLE-CLICK FOR VIDEO

Special-Case - Add two or more coincident Pin-Joints

Frequently, a design has **PIN-JOINTS** that are coincident, to join more than two **PARTS**.

Example:

You want to join **POINTS(#1,#2,#3)**, in **PARTS(#1,#2,#3)**. with **PIN-JOINTS(#1,#2)**.

Say, **PIN-JOINT#1** is between **POINTS #1 & #2** (in **PARTS #1 & #2**)

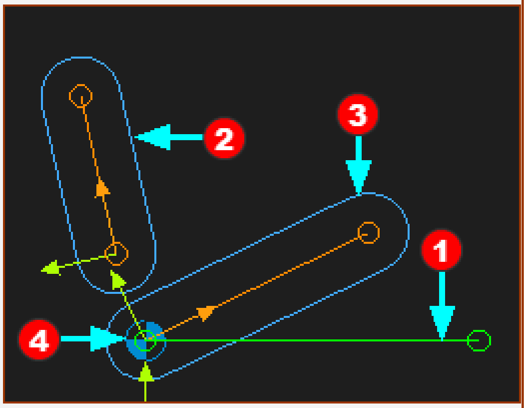
When you add **PIN-JOINT#2**, you have two(2) options:

- **Option A**: PIN-JOINT#2 is between POINTS #3 & #1 (in PARTS #3 & #1)

OR

- **Option B**: PIN-JOINT#2 is between POINTS #3 & #2 (in PARTS #3 & #2)

To select **Option A** or **Option B** is important if you also select the **PIN-JOINTS** to add MOTION-DIMENSION FBS [150].



Preparation example:

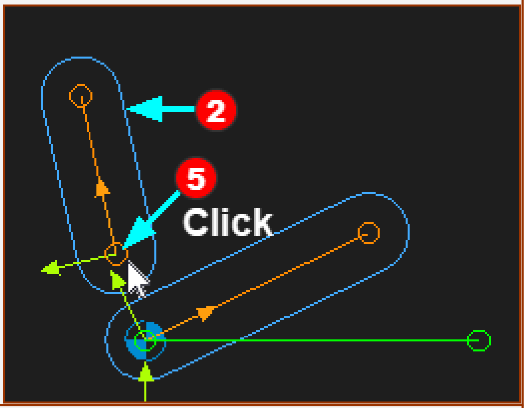
❶ **LINE** in the **BASE-PART** (any **PART** that is **kinematically-defined**)

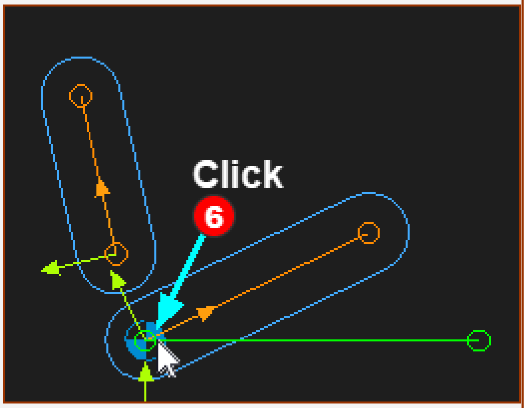
❷ Part that is completely-free

❸ Part that is free

PIN-JOINT❹ joins the **START-POINT** of **LINE❶** to the **START-POINT** of **PART❸**

You want to join **PART❷** that is **completely-free** with a **PIN-JOINT** that is coincident with **PIN-JOINT❹**.





STEP 1: Add Pin-Joint #2

1. Click **Kinematic elements toolbar > Add Pin-Joint**

1. Click POINT❺ at the end of the **Completely-Free-Part**❷
2. Click the PIN-JOINT❻ to select a POINT - see below

You have actually clicked a total of **three(3)** POINTS.

- One POINT at ❺
- Two POINTS at ❻

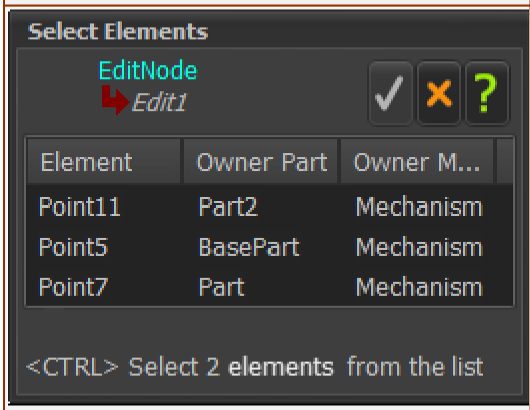
However, the PIN-JOINT is between two(2) POINTS.

Therefore, there is **ambiguity** ...

... Which two(2) POINTS, of the three(3) POINTS, do you want the PIN-JOINT to select?

Definition:

Ambiguity: it is not clear which element(s) to select.



Select-Elements dialog

When there is **ambiguity** as to which elements you want to select, the **Select Elements dialog** opens.

In this model, there are three(3) possible POINTS from which you want to select two(2) POINTS.

- POINT11 (the **START-POINT**❺ of the PART❷ that is completely-free)
- POINT5 (the POINT❻ at the left of the LINE in the BASE-PART)
- POINT7 (the **START-POINT**❻ of the PART that is free)

TOP-TIP: The element at the top of the list is the first element you click. Therefore, click the element you know you need before you click the other elements. Then, you know you need to select the top element.

STEP 2: Select the Points in the Select-Elements dialog:

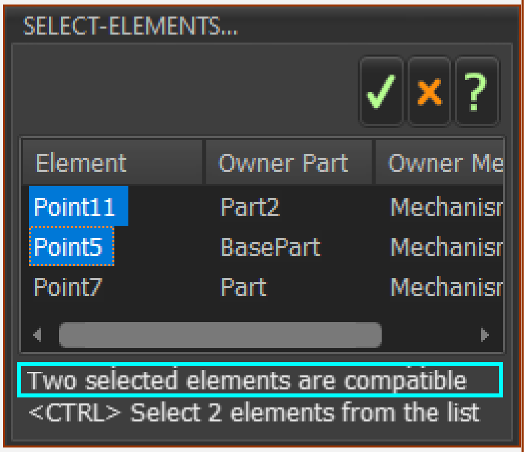
1. Ctrl key down
2. Click the two POINTS you want to use for the PIN-JOINT
3. Ctrl key up

The ✓ colorizes **only** when you select the elements that are compatible .

4. Click ✓ at the top, right of the dialog

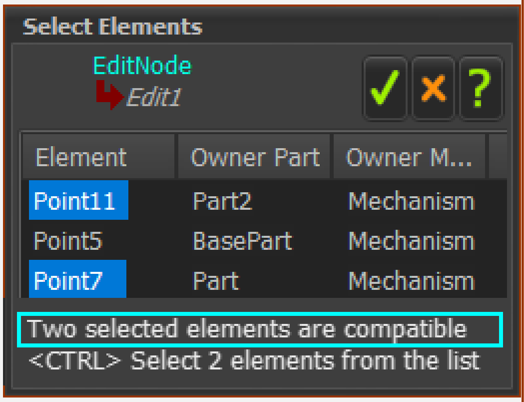
Which two POINTS do you select from the three POINTS?

There are three options to select two(2) ELEMENTS from three(3) elements.



OPTION A

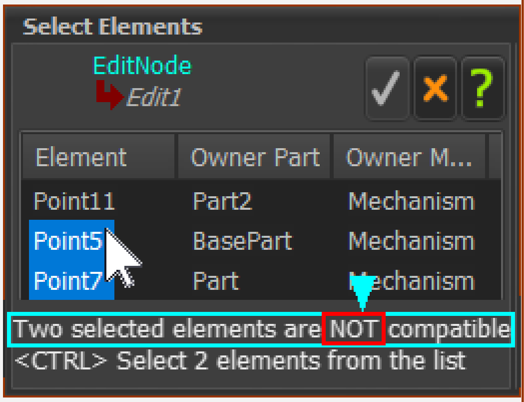
- **Option A**: Two elements are compatible or NOT compatible.
- Yes, you can select POINT11 and POINT5



OPTION B

OR

- **Option B**: Yes, you can select POINT11 and POINT7**.**



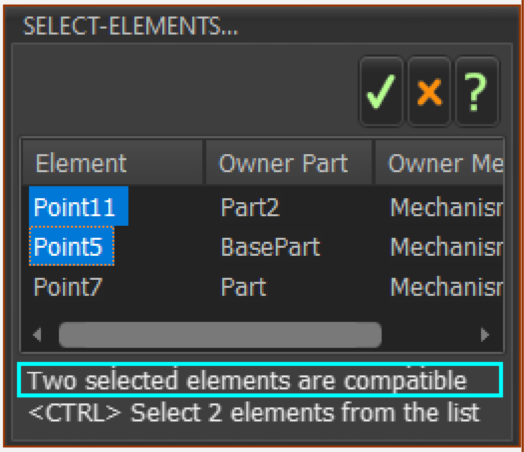
OPTION C

Option C: No, you cannot select POINT5 and POINT7.

Why? Because POINT5 and POINT7 are joined with a PIN-JOINT, already.

Note: In the image to the left, see the word **'not'** in the message that is below the element-box

'Two selected elements are not compatible'



Option A

STEP 2: To confirm again, select the TWO Points:

1. Ctrl key down
2. Click the two **POINTS** you want to use for the **PIN-JOINT**
3. Ctrl key up

The ✓ colorizes **only** when you select the elements that are compatible .

4. Click ✓ at the top right of the dialog

The **SELECT-ELEMENTS DIALOG** closes.

The **POINTS** (and **completely free-Part**) are coincident, at the new **PIN-JOINT**.

TOP-TIP:

If the **SELECT-ELEMENTS DIALOG** opens, the **ELEMENT** you click first is at the **top** of the list.

Therefore, if I know there will be ambiguity, I click the **ELEMENT** that I know I need to select before the elements that cause the ambiguity. Then, when the **SELECT-ELEMENTS DIALOG** open, the element I click first will at the top of the list.

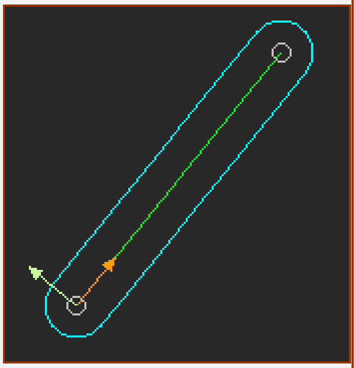
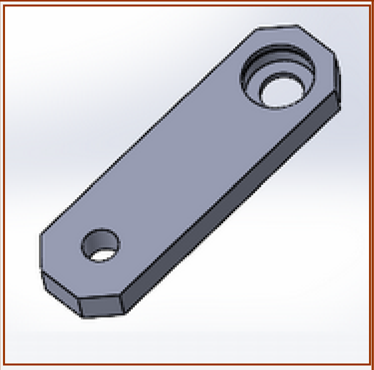
VIDEO - Add Pin-Joint - Special-Case:

Double-click to Video

1.4.2.3 Add Part

Part

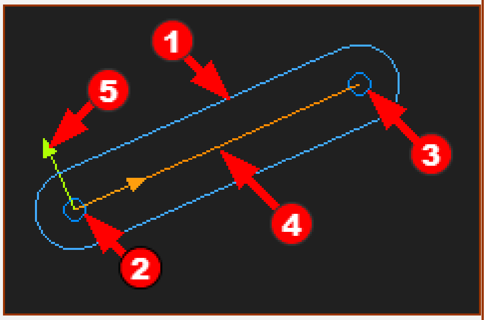
See also: Edit Part [214]

		
Part in MechDesigner	A Part in SOLIDWORKS	A physical part/link

Terminology

PART :	PARTS represent **any** physical and usually moving component in the machine (**see also** BASE-PART **below**) Use the Part-Editor [213] to edit its length and to add sketch-elements and constraints to the sketch-elements.
BASE-PART :	The PART that represents the fixed frame for each **MECHANISM-EDITOR**.
Part-Outlines :	The **Part-Outline** is the **symbol** you click to select or edit a PART. **Part-Outline** have two shapes: • The **Part-Outlines** of the PARTS that you add to the model have an **oval shape.** • The **Part-Outline** of the BASE-PART has a **rectangular shape.**
A PART has one of **TWO kinematic-states.**	
Not kinematically-defined :	The state of a PART when it has one or more degrees-of-freedom. Other terms: **Free-Part**, **Completely-Free Part**, a **Part that is not solved**. The **Part-Outline** is approximately Blue
kinematically-defined :	The state of a PART when its **Mobility** is Zero(0). Other term: is a **Part that is solved**. The **Part-Outline** is approximately Green
IMPORTANT: All of the PARTS in your model must be **kinematically-defined** before you can analyze kinematic (motion) and kinetostatic (force) data in your model.	

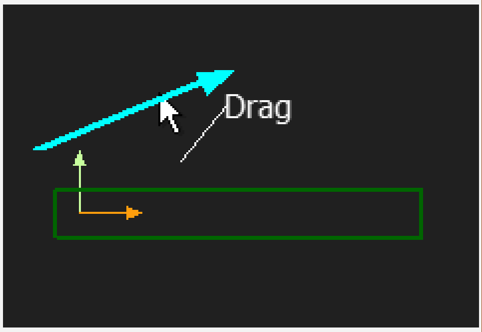
A Part in the graphic-area.

	❶ **PART-OUTLINE**: the symbol of the PART you select to edit the PART in the **PART-EDITOR**. You cannot edit the shape of the **PART-OUTLINE**. ❷ **START-POINT** and ❸ **END-POINT** of the PART ❹ CAD-LINE from the **START-POINT** to the **END-POINT**

	❺ **X & Y-AXES**: at the **START-POINT**

TOP-TIP: You can hide **PART-OUTLINES** (Display Filters - Part-Outlines 54) to see only the **CAD-LINE** and the **PART'S** axes, and edit double-click the **Y-AXIS** to edit the **PART**.

Add Part



STEP 1. Start the Add Part command

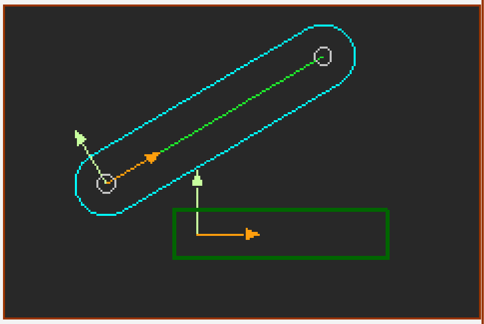
Add menu > Mechanism sub-menu > Add Part

OR

Kinematic elements toolbar > Add Part

OR

Click the **INSERT** key on your keyboard as a short-cut key



STEP 2. To add the Part

DRAG: in the graphic-area :

Mouse-button down, ... move your mouse, ... mouse-button up.

The **PART** is now in the graphic-area.

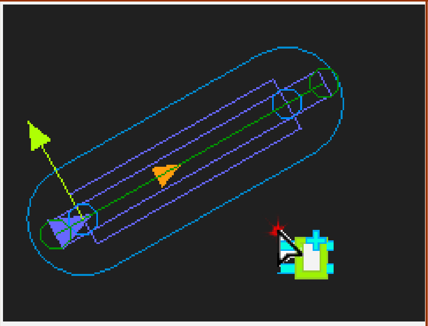
Add Part is **not** a persistent command.

Do **STEP 1** and **STEP 2** again to add a new **PART**.

Video: Double-click to watch Video Clip

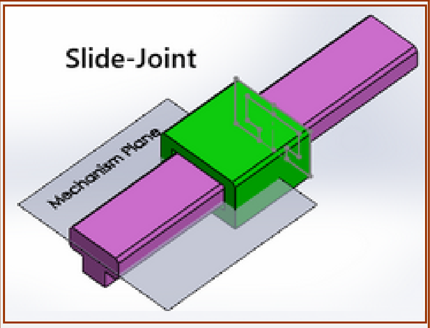
1.4.2.4 Add Slide-Joint

Slide-Joint



Slide-Joint in MechDesigner

Slide-Joint - THK

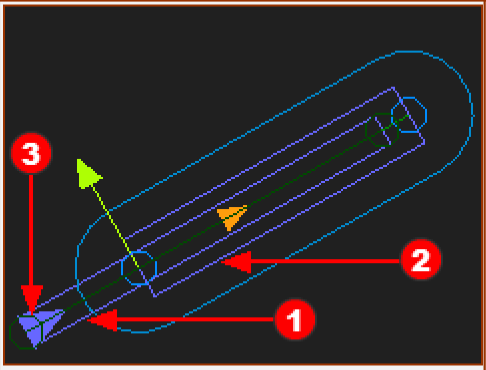


Slide-Joint in SOLIDWORKS

Terminology

SLIDE-JOINT :	A SLIDE-JOINT forces a LINE* in a PART to be collinear with a LINE* in a different PART. A SLIDE-JOINT removes two **degrees-of-freedom** from the model. A SLIDE-JOINT allows only rectilinear translation motion between two **parts** (rigid bodies).
Prismatic Joint :	The kinematic and equivalent term for a SLIDE-JOINT.
KINEMATICS-TREE :	Joints in a **Dyad** use the letter **P** to represent a SLIDE-JOINT. - for example: R-R-**P** dyad
* LINE or CAD-LINE.	
Derived Names:	
sliding-Part :	a PART that you join with a SLIDE-JOINT to a different PART.

A Slide-Joint in the Graphic-Area



Slide-Joint

The symbol for a SLIDE-JOINT:

1. A **Narrow Rectangle** that is equal to the length of the LINE in one of the PARTS.
2. A **Wide Rectangle** that is equal to the length of the LINE in the other PART.
3. An **Arrowhead** that is the **Positive-Direction*** of the SLIDE-JOINT.

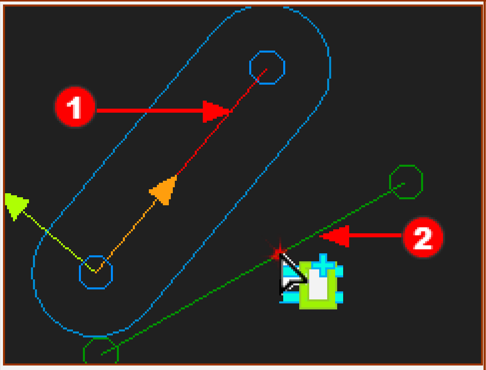
* The **Positive-Direction** is important if you want to add a motion to the SLIDE-JOINT.

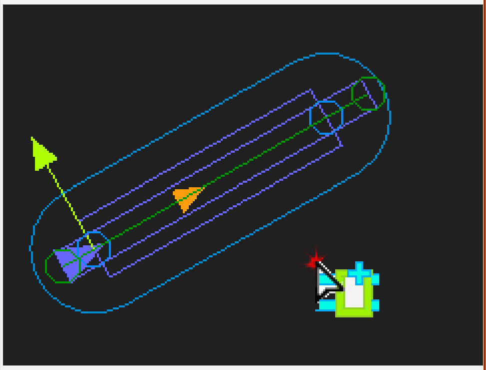
See Positive direction of Slide-Joints and Motion-Dimension 162.

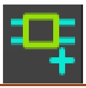
IMPORTANT: **To calculate reaction** FORCES **on a Slide-Joint:**

The **length and position** of the LINES should be **equivalent to the actual design**.

Add Slide-Joint:

<table>
<tr>
<td>


Select two Lines

A New Slide-Joint
</td>
<td>
STEP 1: Start the Add Slide-Joint command

Add menu > Mechanism sub-menu > Add Slide-Joint
OR
 Kinematic elements toolbar > Add Slide-Joint

STEP 2: Click the two(2) Lines

1. Click a LINE❶ in a **free-Part**
2. Click a LINE❷ in a different PART

The two(2) LINES become collinear to add the SLIDE-JOINT.

The **Positive Direction** of the SLIDE-JOINT is identified by the small arrowhead.

If the SELECT-ELEMENTS DIALOG opens, see Slide-Joint Special Case 94
</td>
</tr>
</table>

VIDEO: Add Slide-Joint - Simple-Case

Double-click to watch: Add Slide-Joints - Simple-Case

Special Case: two(2) or more collinear Slide-Joints

Frequently, a machine design has ***two(2)*** collinear SLIDE-JOINTS between ***three*** PARTS.

You must decide which ***two(2)*** LINES from which of two(2) PARTS you want to select for ***each*** SLIDE-JOINT.

For example:

There are a total three PARTS and you would like to join all three PARTS. You must add two(2) SLIDE-JOINTS:

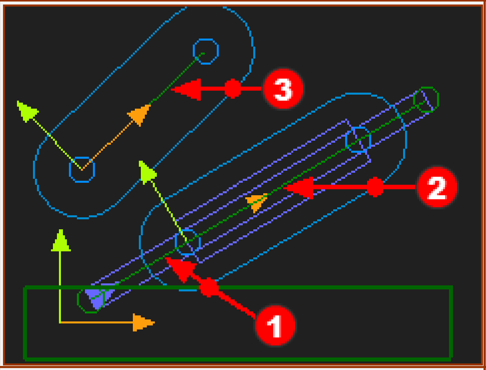
SLIDE-JOINT#1 is between LINES 1 & 2 (in PARTS 1 & 2),

SLIDE-JOINT#2 can be between:

OPTION A: SLIDE-JOINT#2 between LINES 3 & 1 (in PARTS 3 & 1)

OPTION B: SLIDE-JOINT#2 between LINES 3 & 2 (in PARTS 3 & 2)

When to do OPTION A or OPTION B becomes important when you need to also add MOTION-DIMENSION FBS 150 to specify the motions of the two(2) PARTS.



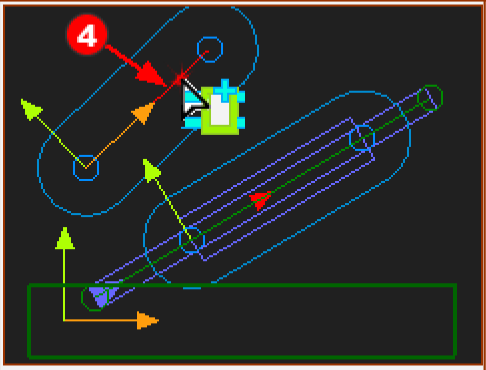
Preparation example:

❶ LINE in the BASE-PART

❷ CAD-LINE along the center of a SLIDING-PART

❸ CAD-LINE in a PART that is COMPLETELY-FREE❸

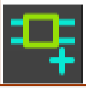
You want to join the PART that is **completely-free**❸ with to become collinear with the existing SLIDE-JOINT.



STEP 1: Add Slide-Joint

1. MECHANISM-EDITOR: Add menu > Mechanism menu > Add Slide-Joint

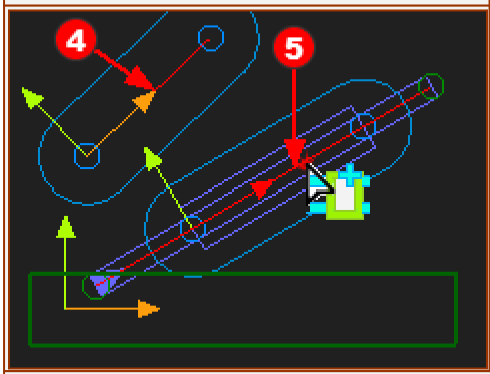
OR



1. MECHANISM-EDITOR: Kinematic elements toolbar > Add Slide-Joint

STEP 2: Select two(2) Lines:

1. MECHANISM-EDITOR: Click the CAD-LINE❹ in the COMPLETELY FREE PART.
2. MECHANISM-EDITOR: Click LINES❺

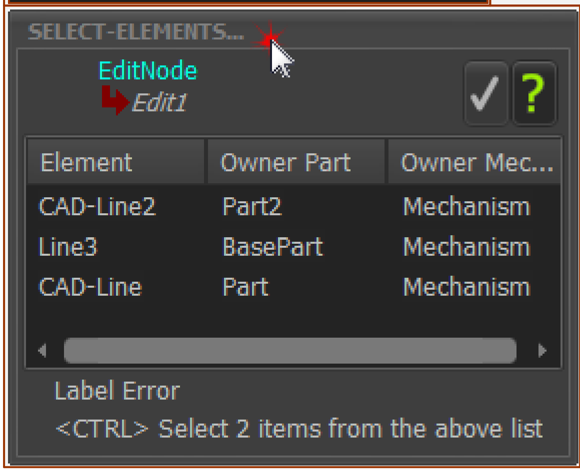


You have actually clicked a total of **three(3)** LINES.

However, the SLIDE-JOINT is between two(2) LINES.

Therefore, there is **AMBIGUITY** as to which two(2) LINES you want for the SLIDE-JOINT.

AMBIGUITY: when it is not clear which element to select.



Element	Owner Part	Owner Mec...
CAD-Line2	Part2	Mechanism
Line3	BasePart	Mechanism
CAD-Line	Part	Mechanism

When there is ambiguity as to which elements to select the SELECT ELEMENTS DIALOG opens.

<<< SELECT-ELEMENTS DIALOG

STEP 4: Select the TWO Lines in the Select-Elements dialog

1. **SELECT-ELEMENTS:** Click the LINE❹ at the top of the Element list (CAD-LINE2)
2. Ctrl key down
3. **SELECT-ELEMENTS:** Click one other LINE in the Element list.

You CAN select:

- CAD-LINE2 AND LINE3 are compatible (image above)

OR

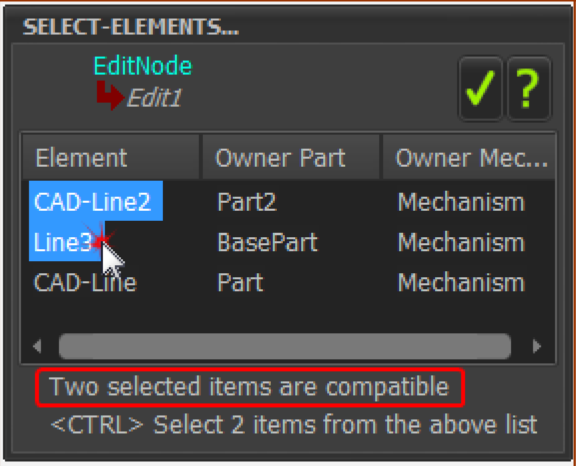
- CAD-LINE2 AND CAD-LINE are compatible (image left)

You CANNOT select:

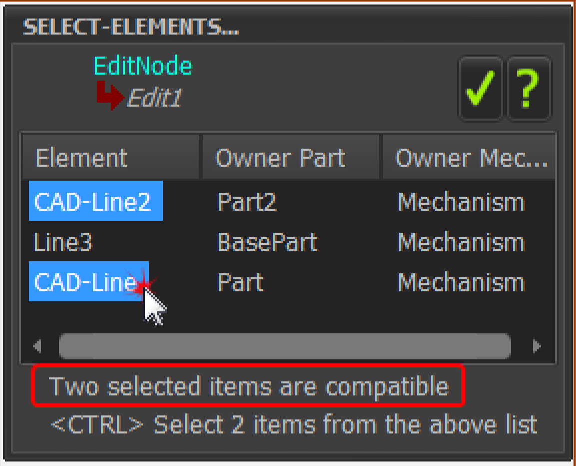
- LINE3 and CAD-LINE are NOT compatible (image below)

Why not LINE3 and CAD-LINE ?

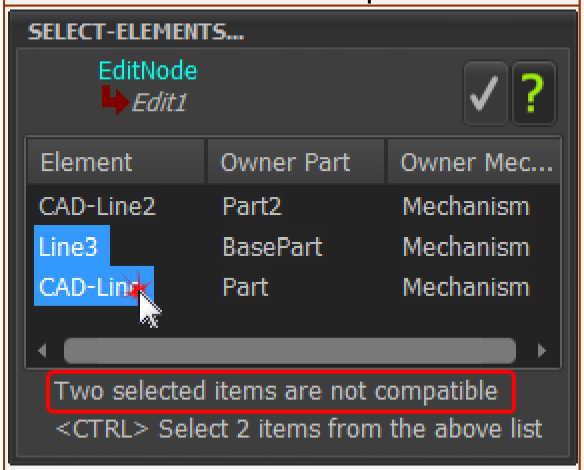
There is a SLIDE-JOINT between these two(2) CAD-LINES already.



Two Lines are Compatible



Two Lines are Compatible

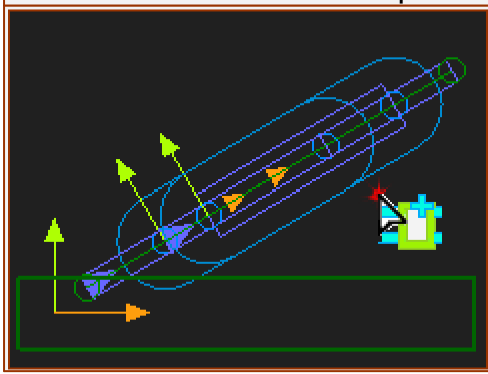


Two Lines are NOT Compatible

STEP 5: Close the Select-Elements dialog

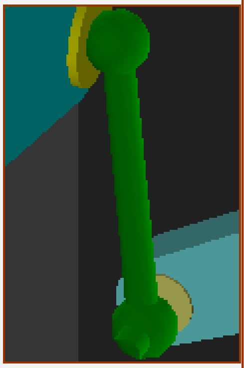
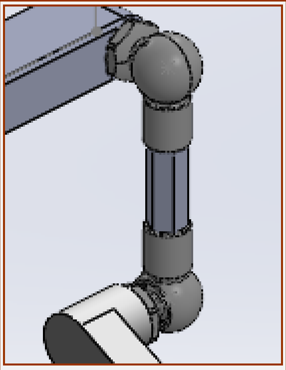
SELECT-ELEMENTS: Click ✓ to close the SELECT-ELEMENTS DIALOG.

There are now two(2) SLIDE-JOINT between three(3) LINES



1.4.2.5 Add Ball-Joint

Ball-Joint

		
2x Ball-Joints in MechDesigner	Rod-End Bearing	2 x Ball-Joint in SOLIDWORKS

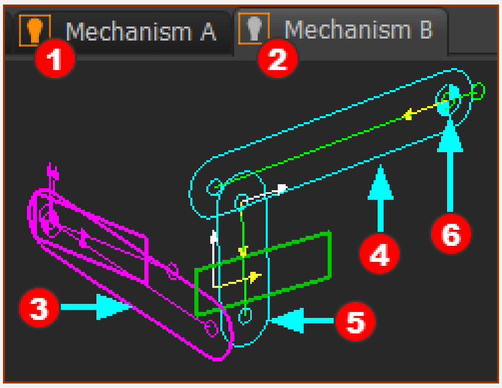
Terminology:

BALL-JOINT :	A **BALL-JOINT** makes a **POINT** in a **Connecting-Part** to be coincident with a **POINT** in a different **PART**. You must add 2 x **BALL-JOINTS** to add a **BALL-JOINT** to each end of a **PART** with a derived name of **Connecting-Part** : • 1 x **BALL-JOINT** between a **POINT** in the **Connecting-Part** and a **POINT** in a **PART**. The two **PARTS** are in the active **MECHANISM-EDITOR**. • 1 x **BALL-JOINT** between a **POINT** in the **Connecting-Part** and a Point in a **PART**. The two **PARTS** are in different **MECHANISM-EDITORS**.
Spherical-Joint :	The kinematic and equivalent term for a **BALL-JOINT.**
KINEMATICS-TREE :	Joints in a **Dyad** use the letter **S** for a **BALL-JOINT**. For example, the **R-S-S** and **P-S-S Dyads.**
Derived Names:	
Connecting-Part :	A **PART** with a **BALL-JOINT** at its **START-POINT** and at its **END-POINT**
Part that is Completely-Free :	A **PART** with 3 degrees-of-freedom (no joints)
Part that is Free :	A **PART** with 1 or 2 degrees-of-freedom

See also: Ball-Joint dialog [456], Connecting Part Length and Diameter [99].
See also Ball-Joint configurations

Preparation

A typical preparation:



A typical prerpation for Add Ball-Joint

Mechanism A

Usually, **Mechanism A** is perpendicular(⊥) to **Mechanism B**

- PART ❸ in a kinematic-chain that is **kinematically-defined** in the image is **Pink**

Mechanism B - the active MECHANISM-EDITOR

Three PARTS:

1 × PART. It is **kinematically-defined** - typically the BASE-PART.

2 × PARTS❹❺. These PARTS are not **kinematically-defined**.

- PART ❹ : A PART that is **free**. It is joined to the BASE-PART with a PIN-JOINT.
- PART ❺ : A PART that is **Completely Free** . It does not have joints before you do Add Ball-Joint.

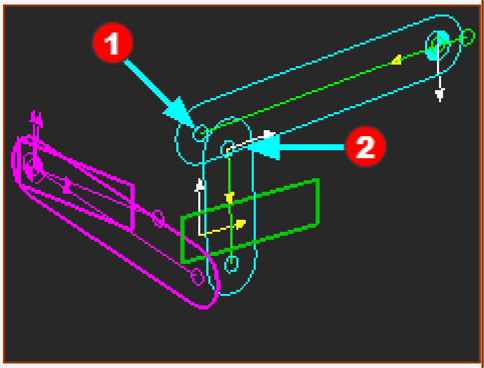
Notes:

- The MECHANISM-EDITORS cannot be parallel to each other. They are most often perpendicular(⊥) to each other.
- The PART that is **Completely-Free**❺ will joint to PARTS ❹ and to ❸ with BALL-JOINTS - these are two(2) **S** joints in the **S-S-R** or **S-S-P** dyads. **S** is for **Spherical-Joint.**
- As you add the PART that is **Completely-Free**, try to estimate its length.

  If *it* is too short or too long, the BALL-JOINTS cannot join it to PART❹ and PART❸ - you do not get the expected result.

Add Ball-Joint × two(2)

Do Add Ball-Joint two(2) times. One time to each end of the COMPLETELY FREE PART.



STEP 1: Start the Add Ball-Joint command

Add menu > Mechanism sub-menu > Add Ball-Joint

OR

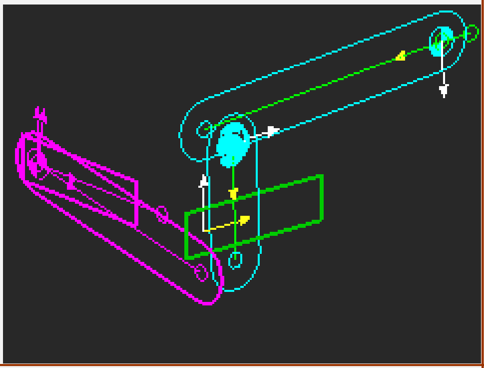
Kinematic elements toolbar > Add Ball-Joint

STEP 2: Add Ball-Joint #1

MECHANISM-EDITOR **B** is active.

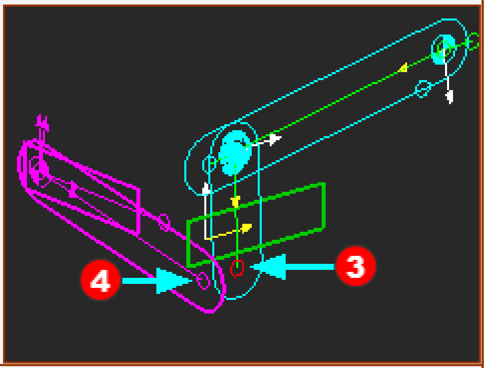
1. Click POINT❶ at the end of the PART that is **Completely-Free**
2. Click POINT❷ at the end of the PART that is **Free**.

If you cannot select the two(2) POINTS to add a BALL-JOINT, disable the Auto-Update tool [37], and try again.



When you do **Add Ball-Joint #1** you can see the **Ball-Joint symbol.**

However, POINTS❶❷ **do not** snap together before you do **Add Ball-Joint #2**



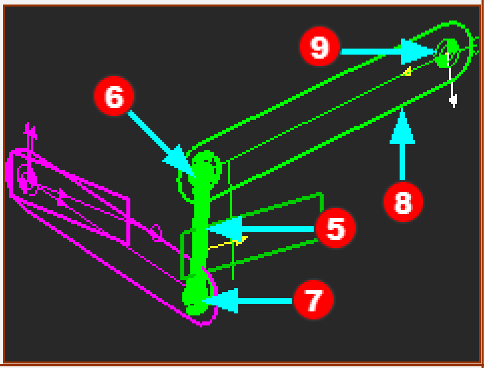
STEP 3: Add Ball-Joint #2

1. Click POINT❸ at the other end of the PART that is **Completely-Free**
2. Click POINT❹ - the **Pink Point** - at the end of the PART that is **kinematically-defined** in MECHANISM-EDITOR(**A**)

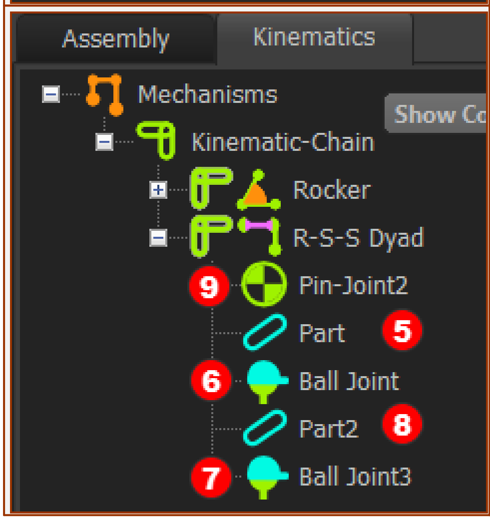
Result - expected

The BALL-JOINTS❻❼ snap to the POINTS in MECHANISMS **A** and **B**.

The PARTS that were **Completely-Free** and **Free** are now **kinematically-defined**❺❽



Note: If the arrangement of the PARTS that were **Completely-Free** and **Free** is not correct, then see Change-Dyad Closure [101]



KINEMATICS-TREE

In the KINEMATICS-TREE [262],

There is a new Dyad - the **R-S-S** dyad.

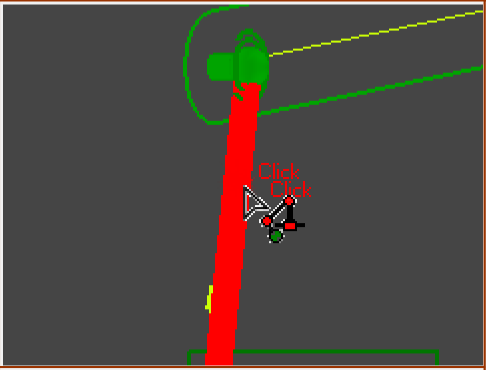
The three Joints in the Dyad are:

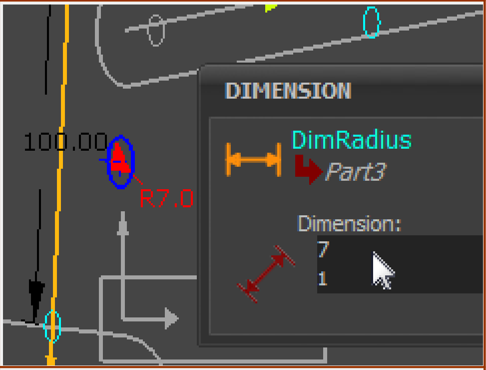
- PIN-JOINT❾ is the **R** joint - **R** is for **Revolute-Joint**
- BALL-JOINTS❻❼ are the **S** Joints - **S** is for **Spherical-Joint**

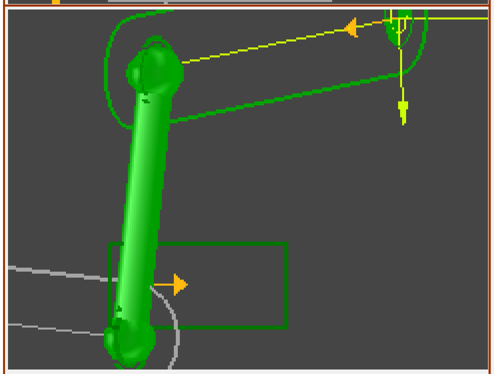
Connecting-Part: Edit Length, Rod Diameter, Ball-Diameter

Notes:

- You cannot add SOLIDS to **Connecting-Parts**.







To edit the LENGTH of the CONNECTING-PART

1. **MECHANISM-EDITOR**: Double-click the symbol for the **Connecting-Part**

It is now open in the **PART-EDITOR**.

2. **PART-EDITOR**: Edit the Length dimension of the **Connecting-Part**
3. Exit the **PART-EDITOR**

To edit the SYMBOLIC RADIUS of the CONNECTING-PART

1. **MECHANISM-EDITOR**: Double-click the symbol for the **Connecting-Part**

It is now open in the **PART-EDITOR**.

2. **PART-EDITOR**: Add a **LINE** or **CIRCLE**
3. **PART-EDITOR**: Add a dimension to the **LINE** or **CIRCLE**
4. Exit the **PART-EDITOR**

The dimension becomes the Radius of the **Connecting-Part**.

To edit the the symbolic diameter of the BALL-JOINT - see Ball-Joint dialog [456]

Notes:

In the **PART-EDITOR**, when the view is not the **Front View**, the **CAD-LINE**, **Axes**, sketch-elements and dimensions may show in unusual places!

If you cannot add or edit a dimension: do **View toolbar** > **Front Plane**.

Also, to see the new Symbolic Radius: enable **Visibility toolbar** > **Show Solids in Mechanisms** [51]

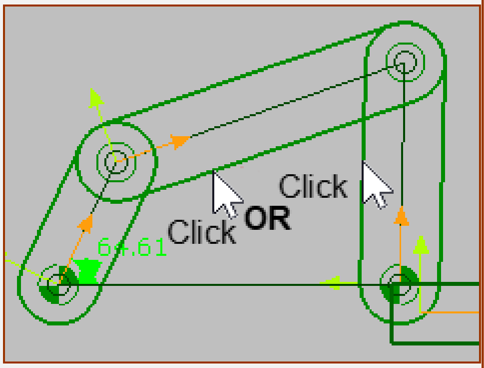
1.4.2.6 Change Dyad Closure

Change Dyad Closure / Change Configuration of Dyad's Assembly.

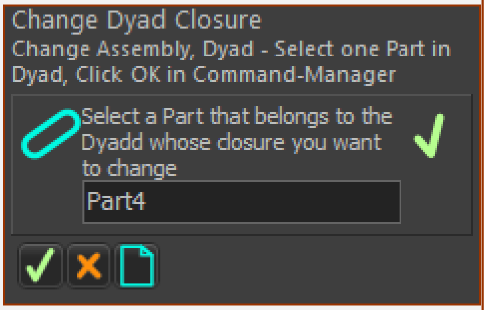
Terminology

Dyad : also known as Assur Group :	A **Dyad** is an assembly of two PARTS and three **joints**. There is one joint that joins the two PARTS together. The two other joints join the PARTS to two other PARTS that are **kinematically-defined**. The Dyad is **kinematically-defined**.
Assur Groups :	Another name for a **Dyad**
Dyad Closures :	The different ways you can assemble the two PARTS of a **Dyad**
Configuration of the Dyad's Assembly = another way of expressing '**Dyad Closure**'.	
Types of Dyad :	Planar Dyads: RRR, RRP, RPR, RPP, PRP Spatial Dyads: SSR, SSP
R :	Revolute Joint = Pin-Joint 85
P :	Prismatic Joint = Slide-Joint 93
S :	Spherical Joint = Ball-Joint 97
See more: Dyads	

Change Dyad Closure



RRR Dyad Closure 1

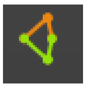


Command-Manager: Change Dyad Closure

STEP 1: Start the Change Dyad Closure command

1. MECHANISM-EDITOR: Add menu > Mechanism menu > Change Dyad Closure

OR



1. MECHANISM-EDITOR: Kinematic elements toolbar > Change-Dyad Closure

The COMMAND-MANAGER indicates you must select a PART in the **Dyad**.

STEP 2: Select one Part in the Dyad

2. Click the PART-OUTLINE of a PART in the **Dyad** whose assembly-configuration (Dyad-Closure) you want to change.

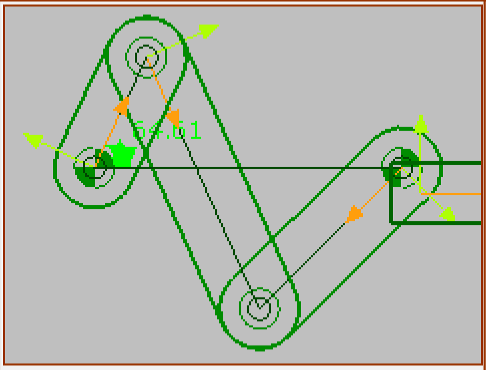
STEP 3: Complete the Command

3. Click ✓ in the COMMAND-MANAGER

The configuration of the assembly (Dyad Closure) should now be different.

<<< To the left, compare the bottom image with the top image.

STEP 4: Do again, if needed



RRR Dyad
Closure 2

4. If necessary, do **1 to 3** again and again to find the configuration of the assembly you want to apply to your design.

Top-Tip:

If there are many **Dyads** in the model and you are not sure which **Dyad** whose closure you need to change, expand and explore the **KINEMATICS-TREE** to identify each **Dyad**.

1. **KINEMATICS-TREE**: Click and Expand each **Kinematic-Chain** in the **KINEMATICS-TREE**
2. **KINEMATICS-TREE**: Click each **Dyad**

When you click to explore each dyad in the **KINEMATICS-TREE**, the five elements in the Dyad (**2** PARTS + **3** JOINTS) become red in the graphic-area.

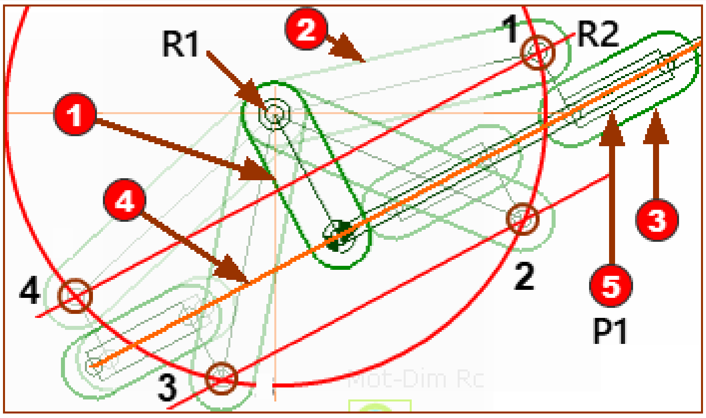
Assembly-Configurations of the RRP Dyad

The joints in the **R-R-P** Dyad are: **Revolute – Revolute – Prismatic.**

In **MechDesigner**, the equal joint names are: **Pin-Joint – Pin-Joint – Slide-Joint**.

There are 4 possible assembly configurations (Dyad Closures).

The image below shows an **Offset-Slider-Crank** mechanism.



4 x Assembly Configurations of an Offset-Crank-Slider

PART❶ is the **Crank** - to illustrate the 4 Dyad closures, it does NOT move.

Dyad: The two PARTS in the **R-R-P dyad** are PARTS ❷❸.

To imagine the four assembly-configurations, do not add the PIN-JOINT **(R2)** between PART❷ and PART❸.

PART❷ can only rotate around the PIN-JOINT, **R1**, at the end of the Crank.

Therefore, PIN-JOINT **R2** can ONLY be around the red circle **at the end of** PART❷.

PART❸ can only slide along the SLIDE-JOINT, **P1**, along LINE❹.

IMPORTANT: The POINT in PART❸ for the PIN-JOINT (**R2**) is offset from the SLIDE-JOINT and not on the SLIDE-JOINT.

Therefore, **PIN-JOINT** **R2** can, also, **ONLY** be along the **red lines** that are offset from the **SLIDE-JOINT**.

The **red circle** intersects the **red lines**. at 4 places.

Therefore, the **PIN-JOINT** **R2** can **ONLY** be at 4 places.

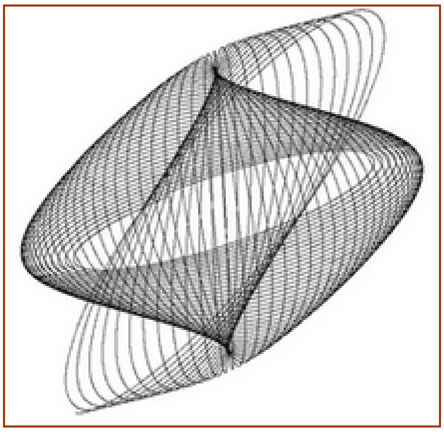
Therefore, the **Offset Slider-Crank** has 4 possible assembly configurations (Dyad Closures).

1.4.2.7 Add Trace-Point

Trace-Point

Terminology:

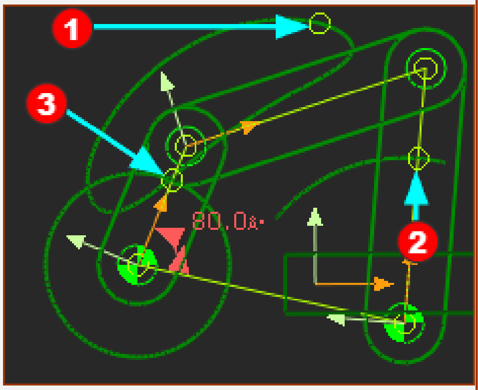
TRACE-POINT :	a locus, or a path, of a moving **POINT** relative to the **MECHANISM-PLANE** for one machine-cycle.



Add Trace-Point

Add Trace-Point does **NOT** add a **POINT** to your model.

You must select a **POINT** that is in your model before you do Add Trace-Point.



Three example Trace-Points

STEP 1: Start the Add Trace-Point command

1. MECHANISM-EDITOR: Add menu > Mechanism sub-menu > Add Trace-Point

OR

1. MECHANISM-EDITOR: Kinematic elements toolbar > Add Trace-Point

STEP 2: Click a Point

2. Click a POINT* in the graphic-area.

For example, in the image, POINTS ❶❷ and / or ❸

* START-POINT, END-POINT, CENTER-POINT, or POINT

See below: How to drag a Point with a Trace-Point. 104

If the SELECT-ELEMENT DIALOG 504 opens, you have clicked more than one POINT.

1.a. **CTRL + click** a POINT in the SELECT-ELEMENTS DIALOG

1.b. Click ✓ to close the SELECT-ELEMENTS DIALOG

STEP 3: Complete the command

3. Click ✓ **in the Command-Manager**

The TRACE-POINT is now in the graphic-area.

Video:

Double-click to watch Video

How to drag a Point with a Trace-Point

Normally, when the **MECHANISM-EDITOR** is active, you cannot move a sketch-element that is a child to a **PART**.

However, it is sometimes useful to search for a good **TRACE-POINT**, by moving the **POINT**.

You **can** drag **POINTS** in the **MECHANISM-EDITOR** **if ALL** of these conditions are met:

The **POINT** must be a **POINT** - that, is, it cannot be a **START-POINT**, **END-POINT** or **CENTER-POINT**.

IF...

In the **PART-EDITOR**:

a. You do: **Part-Editor > Add Point**

AND IF...

Also, in the **PART-EDITOR**

b. You do **not** add a dimension or a constraint to the **POINT**

AND IF...

In the **MECHANISM-EDITOR**:

c. You add a **TRACE-POINT** to the **POINT**

Then, you **can drag** the **POINT** and the **TRACE-POINT** in the **MECHANISM-EDITOR**.

Note:

To make it easier to move the **POINT**:

- Reduce the **NUMBER-OF-STEPS** to ~30 in the Machine Setting dialog 39

and / or

- Drag the **POINT** and then wait until **TRACE-POINT** moves.

To add a Trace-Point of a Point relative to a Part that is NOT the Base-Part

Add a **2D-CAM**.

The Point whose locus you want to see relative to a different Part is the center-Point of the Cam-Follower.

1. Edit the **PART** with the **POINT**
2. Add a **CIRCLE**, and then add a Coincident Constraint between the **CENTRE-POINT** coincident with the **POINT**.
3. Exit the **PART-EDITOR**
4. Add a **PROFILE** to the **CIRCLE**

Add 2D-Cam - you must select two elements:

5. **Cam-Part**: Select the **PART** in which you want to show the **TRACE-POINT**
6. **Follower Profile**: Select the **PROFILE** of the **CIRCLE** (see 1)

To show the **TRACE-POINT**

7. Edit the **2D-CAM** (see 2D-CAM DIALOG 327) and show only the **PITCH-CENTER** path.

The locus of the **PITCH-CENTRE** is the **TRACE-POINT** of the **POINT** relative to the **Cam-Part**

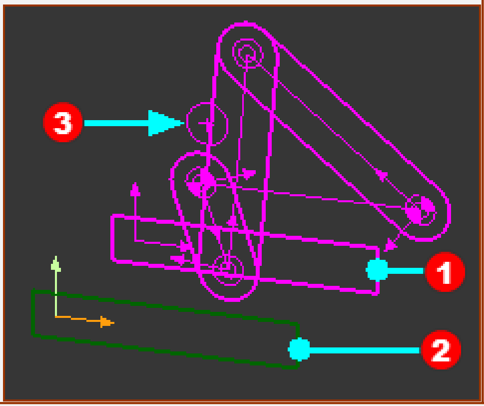
1.4.2.8 Add Reference Geometry

Reference Geometry

Terminology:

Reference Geometry :	General term for a sketch-element that is a copy of a sketch-element from a different MECHANISM-EDITOR to the active MECHANISM-EDITOR.
Source Sketch-Element :	A sketch-element, in a moving or stationary PART in a **different** MECHANISM-EDITOR.
Reference Sketch-Element :	A copy of the **Source sketch-element, and its motion,** to a PART in the **active** MECHANISM-EDITOR.

Preparation



Preparation example:

Two(2) MECHANISM-EDITORS

- SOURCE ❶
- REFERENCE ❷ - this is the active editor

A CIRCLE❸ in SOURCE SKETCH-ELEMENT

The CIRCLE is in the SOURCE MECHANISM-EDITOR.

To see the SOURCE MECHANISM-EDITOR when the REFERENCE MECHANISM-EDITOR is active.

- Visibility toolbar > Show other Kinematic and Sketch elements [49]

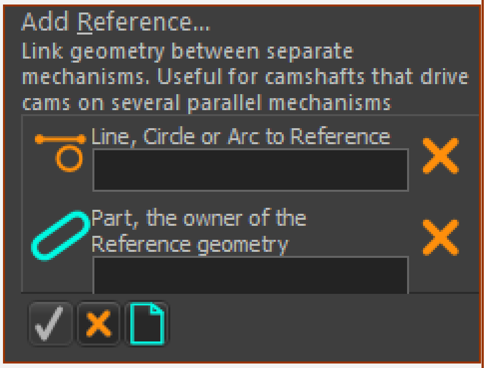
AND

- Right-click MECHANISM NAME-TAB of the SOURCE MECHANISM-EDITOR and select Show with 'other Kinematic and Sketch elements' [49].

AND

- View toolbar > Spin or use your arrow keys on your keyboard to spin the model to see the two editor.

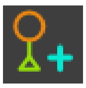
Add Reference Geometry



STEP 1: Start the Add Reference-Geometry command

1. Add menu > Mechanism menu > Add Reference Geometry

OR



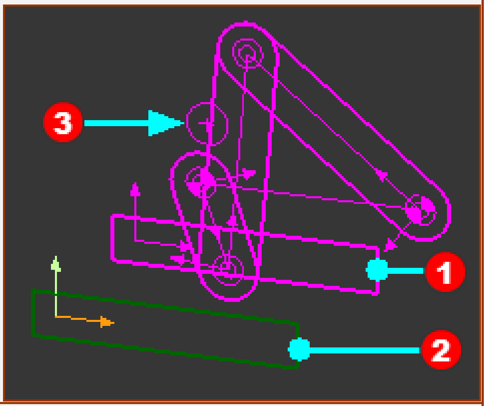
1. Kinematic elements toolbar > Add Reference Geometry

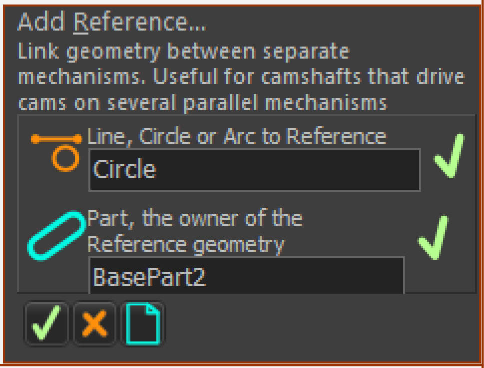
The COMMAND-MANAGER indicates you must select two(2) elements

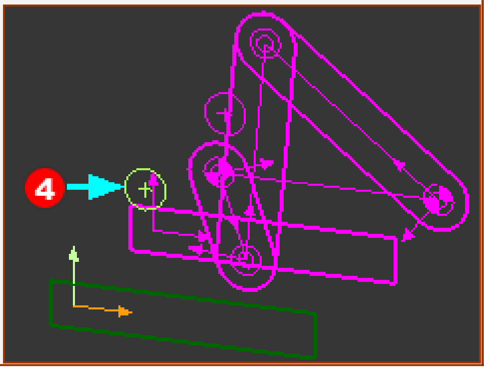
STEP 2: Select the two elements in the graphic-area

2. Click a sketch-element in the SOURCE REFERENCE-EDITOR.

Click the CIRCLE❸ in the SOURCE MECHANISM-EDITOR ❶.







3. Click a **PART** in the **REFERENCE MECHANISM-EDITOR**

In this case there is only one **PART** - the **BASE-PART** - in the **REFERENCE MECHANISM-EDITOR**.

STEP 3: Complete the Command

4. **COMMAND-MANAGER: Click ✓ in the COMMAND-MANAGER.**

RESULT :

The **REFERENCE GEOMETRY**❹ is now in the **REFERENCE MECHANISM-EDITOR**.

The motion of the **SOURCE GEOMETRY**❸ are also transferred to the **REFERENCE MECHANISM-EDITOR**.

Video:

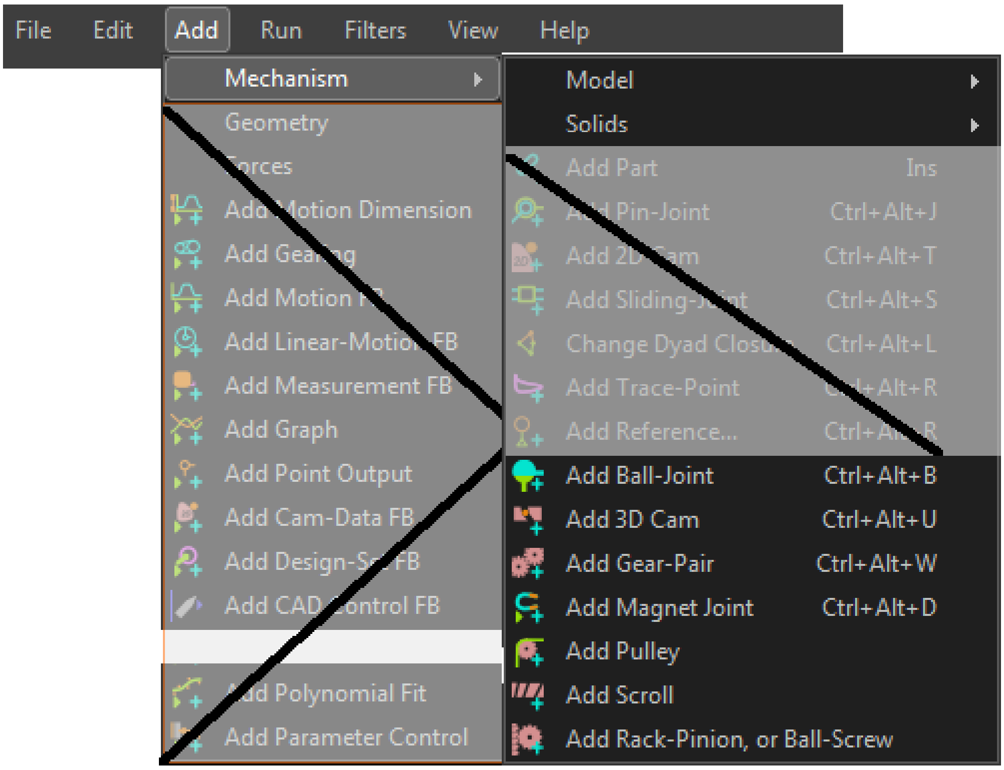
Double-click to watch Video Clip

1.4.3 Machine elements

Machine elements

Machine elements are hardware components that you install on industrial machines.

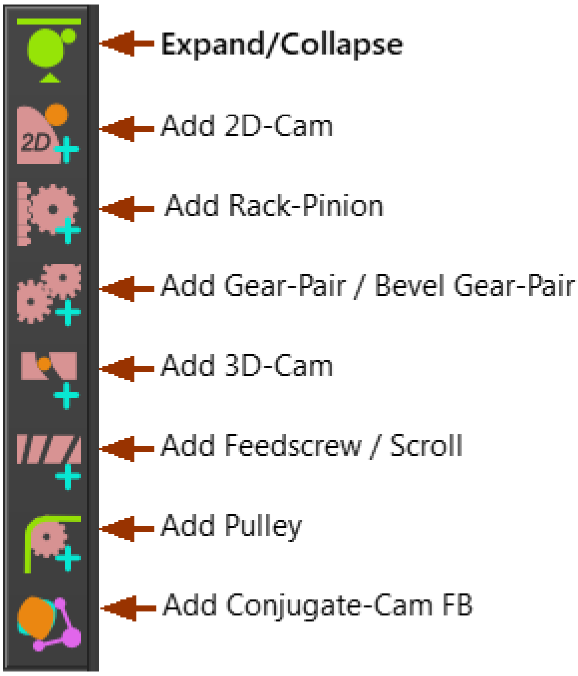
Machine elements menu



Machine element menu

Machine elements toolbar

The **Machine elements toolbar** is to the left of the graphic-area.



Machine elements toolbar

1.4.3.1 Add 2D Cam

2D-Cam

See also: 2D-Cam dialog [327]

We calculate for you the 2D-CAM from the relative motions of the **Cam-Part** and **Follower-Part**, together with the shape of the **Cam-Follower**.

Add 2D-Cam and Degrees of Freedom

Add 2D-Cam does NOT remove a degree-of-freedom.

Terminology:

CAM / 2D-CAM :	A **Cam** (2D-CAM element) is a curved profile or surface that imparts a displacement (a motion-law) to a Follower by either point or line contact with a Cam-Follower.
Cam-Part :	The PART to which you add the 2D-CAM
Cam-Follower :	The shape of the **sketch-loop** and PROFILE / EXTRUSION that is in continuous-contact with the 2D-CAM.
Cam-Follower Part :	The PART to which you add the the **Cam-Follower**.

2D-Cam: Work-flow

1. Add a 2D-CAM - **see** **Machine elements toolbar > Add 2D-Cam [109]**

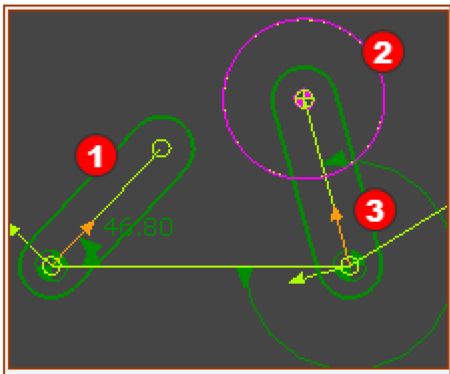
If the new 2D-CAM is one from a pair of **Conjugate-Cams**, or it is one flank of a **Groove-Cam**

1.a. Add a CONJUGATE-CAM FB - **see** **Machine elements toolbar > Add Conjugate Cam FB [139]**

1.b. Edit the CONJUGATE-CAM FB to select least two 2D-CAMS - **see** **Conjugate-Cam dialog [353]**.

2. Select a 2D-CAM or a CONJUGATE-CAM FB as the Power Source for the Follower - **see** **Configure-Power Source [480]**
3. Review the 2D-CAM : **Display, Properties, Roller-Life, Cam-Life, ...** - **see** **2D-Cam dialog [327]**
4. Add a CAM-DATA FB - **see** **Kinematic FBs > Add Cam-Data FB [171]**
5. Edit the CAM-DATA FB to link it to the 2D-CAM- - **see** **Cam-Data dialog : Cam Analysis [357]**
6. Connect wires from the CAM-DATA FB to a GRAPH FB to analyse 5 parameters - **see** **Cam-Data dialog : Cam Analysis [357]**
7. Edit the CAM-DATA FB again to calculate the Cam's Coordinates - **see** **Cam-Data dialog : Cam-Coordinates [361]**

Prepare to add a 2D-Cam (typical)



Preparation for Add 2D-Cam

The elements you need in the model before you do Add 2D-Cam:

Cam-Part ❶ - any PART that is **kinematically-defined**

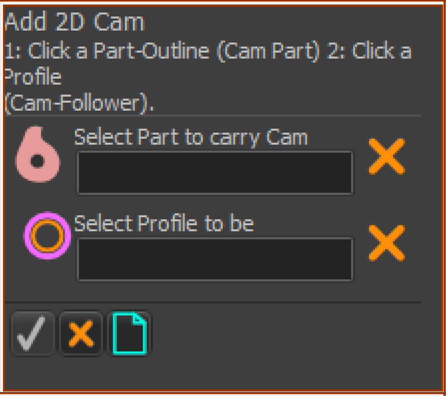
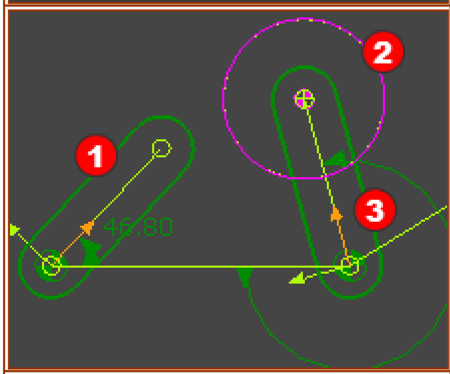
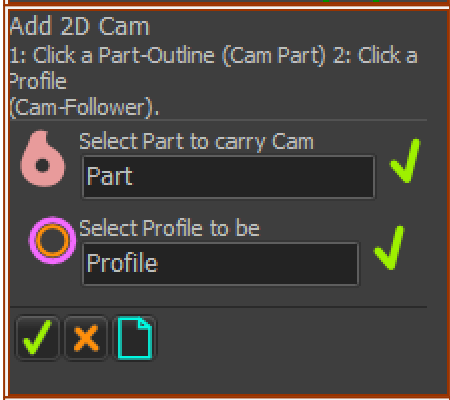
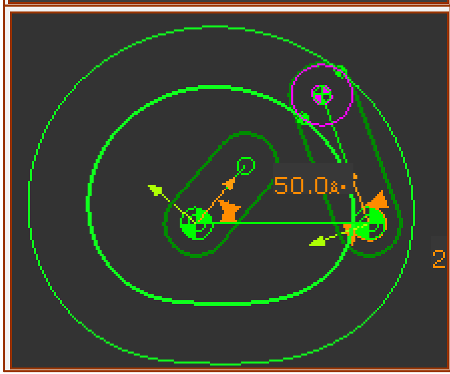
Follower-Part ❸ - any other PART that is also **kinematically-defined**

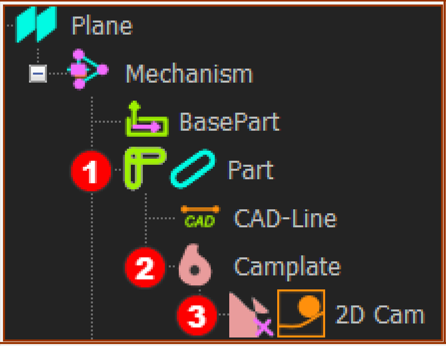
Cam-Follower ❷ - a PROFILE in the **Follower-Part**❸

The **Cam-Part** usually moves, but it may be stationary.

	The **Follower Part** usually moves, but it may be stationary. The **Cam-Part** and the **Follower Part** must move relative to each other. The **Cam-Part** and **Follower-Part** are usually in different **kinematic-chains**, but this is not necessary. **See also Shape of the Cam-Follower.**

Add 2D-Cam:

	STEP 1: Click Add 2D-Cam 1. MECHANISM-EDITOR: Add menu > Mechanism menu > Add 2D-Cam OR 1. MECHANISM-EDITOR: Machine-elements toolbar > Add 2D-Cam The COMMAND-MANAGER indicates you need to select two elements.
	STEP 2: Select the two elements Element 1 is the **Cam-Part** 1. Click the PART-OUTLINE ❶ in the graphic-area, or the Part element in the ASSEMBLY-TREE. Element 2 is the **Cam-Follower** in the **Follower-Part** 2. Click the PROFILE❷ in the graphic-area or the PROFILE element in the ASSEMBLY-TREE
	STEP 3: Complete the Command 3. COMMAND-MANAGER: **Click ✓ in the** COMMAND-MANAGER.
	Result : Graphic-Area The Inner and Outer 2D-CAMS show in the graphic-area.



Result : Assembly-Tree

There is a **Cam-Plate** ❷ as a child to the **PART** ❶ to which you add **2D-CAM** ❸.

The **2D-CAM** ❸ is a child to the **Cam-Plate** ❷.

The icon for the **2D-CAM** in the **ASSEMBLY-TREE** provides a warning that it may be "**Bad Cam**"!

We base the warning for you on the **Cam's Radius-of-Curvature** as a percentage of the **Follower's Radius.** You can set warning limit.

The default warning limit is specified in the **APPLICATION SETTINGS** - **see here**[284]

You can specify a warning limit for each **2D-CAM** - see **2D-Cam dialog**[329]

YOU must make sure the cam is suitable for your needs, even if it is a "**Good Cam**".

"**Good Cam**"

"**Bad Cam**"

Next Steps:

If Force-Analysis is important

Configure the Power-Source and select the 2D-Cam - see Configure Power-Source[480]

Add Mass-Properties to the important Parts in the model - see CAD-Line dialog[301]

Add Force Data FB to plot Torques, Forces over a machine-cycle - see Force-Data FB[191], Add Spring FB[194]

If the Cam-Part is actually a Cam-Shaft, then edit Motor - see Servo-motor and Gearbox[483]

Edit the 2D-Cam

Select and edit the **2D-CAM** to open the **2D-Cam dialog**[327].

You can display its Pitch-Curve, edit its color, the thickness of the cam-profile, ...

You can estimate the Roller-Life and Cam-Life.

Analyse the Cam over a Machine-Cycle the 2D-Cam to

Add and edit Cam-Data FB and link it to the **2D-CAM** you want to analyse - **see Add Cam-Data FB**[171] and **Cam-Data FB dialog : Cam Analysis**[357]

Analysis parameters include: **Maximum Shear-Stress, Contact-Force, Radius-of-Curvature, Pressure-Angle, Sliding-Velocity**

Calculate and export the Cam-Coordinates in different formats

Add and edit a **CAM-DATA FB** , edit link it to the **2D-CAM** you want the Cam-Coordinates - - **see Add Cam-Data FB**[171] and **Cam-Data FB dialog Cam-Coordinates**[361]

Calculate the Cam Coordinates as **XY-Points** or **Biarcs**.

Export the Cam Coordinates as TXT, DXF, STEP, or export the **Points** or **Biarcs** directly with SOLIDWORKS

Video (EN):

Double-click to watch Add a 2D-Cam

1.4.3.2 Add 3D Cam

3D-Cam

See also 3D-Cam dialog [315]

We recommend you do these tutorials if you have not used this command before: *Tutorial 6C: Barrel and Globoidal Cams*

Add 3D-Cam and Degrees of Freedom

Add 3D-Cam does NOT remove a Degrees-of-Freedom.

Terminology:

3D-CAM :	A 3D-CAM has complex Cam-Flanks that interact with Cam-Follower Rollers. The axis the rotating 3D-Cam is not parallel with the rotating-axes of the Cam-Follower Rollers.
Barrel, **Cylindrical**, and **Globoidal Cams** are 3D-CAMS. **Globoidal Cams** are also known as **Roller-Gear-Drives** and **Ferguson Cams**.	

Prepare to add a 3D-Cam (typical)

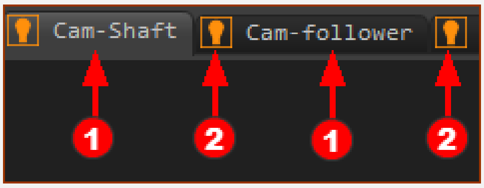
You must prepare the model with five(5) elements.

Also:

- 3 **Mechanism-Editors** for a **Barrel Cam**
- 4 **Mechanism-Editors** for a **Globoidal Cam**

ALL PARTS are **kinematically-defined** ***before*** you do Add 3D-Cam.

Top-Tip: Rename the **Mechanism-Editors** to the names of the elements you need to Add 3D-Cam. **See Rename dialog** [279]



Prepare the view:

1. Click **Visibility toolbar** > Show other Kinematic and Sketch elements [49]

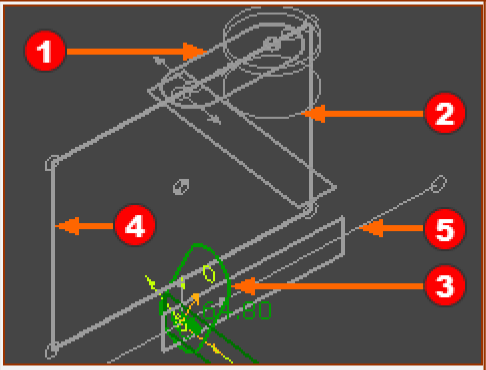
To show the **Other Kinematic and Sketch elements** in each MECHANISM-EDITOR:

1. Right-click the name-tab❶ of each MECHANISM-EDITOR that has an element you need to add a 3D-CAD
2. Click **Show with other Kinematic and Sketch elements.**

The Light-bulbs should now be orange❷

Do not show SOLIDS to Add 3D-Cam

1. De-select Visibility toolbar > Show Solids in Mechanism-Editor [51]



Prepare the five elements:

❶ PART : Follower-Part in Mechanism-B

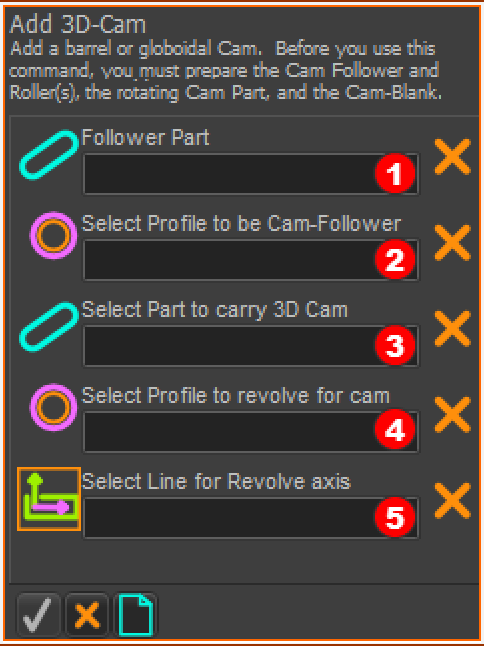
❷ PROFILE : Follower-Roller in Mechanism-B | the PROFILE must be circular.

❸ PART : Cam-Part in Mechanism-A - this must be the *active* MECHANISM-EDITOR

❹ PROFILE : Cam-Blank in Mechanism-C

❺ LINE : Line in Mechanism-C

Add 3D Cam



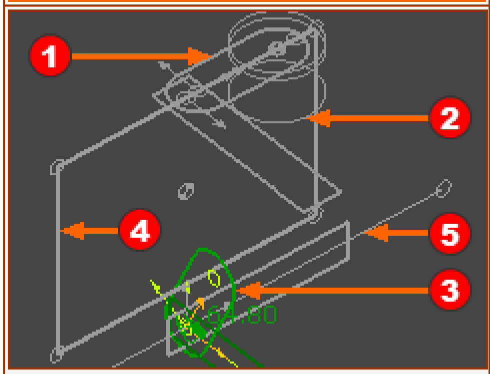
STEP 1: Click Add 3D-Cam

1. MECHANISM-EDITOR: Add menu > Mechanism menu > Add 3D-Cam

OR

1. MECHANISM-EDITOR: Machine-elements toolbar > Add 3D-Cam

There are five elements to select in the COMMAND-MANAGER



Elements to add a 3D-Cam:

STEP 2: Click the five elements, from top-to-bottom, in the graphic-area or the Assembly-Tree.

❶ PART: **Cam-Follower Part**: Click the *Grey* PART-OUTLINE

❷ PROFILE: Cam-**Follower Roller**: Click the *Grey* **Profile (contour)**

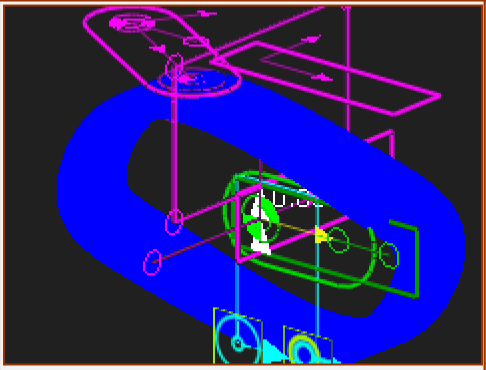
❸ PART: **Cam-Part**: Click the Green PART-OUTLINE in the active **Mechanism-Editor**

❹ PROFILE: **Cam-Blank Profile**: Click the Grey **Profile (contour)**

❺ LINES **Cam-Blank Axis of Rotation**: Click the Grey **Line**

STEP 3: Complete the command

1. COMMAND-MANAGER: **Click** ✅ **in the** COMMAND-MANAGER.

	**Result : Graphic-Area** You can see the **3D-Cam** in the graphic-area. The **3D-Cam** has **four surfaces**, which enclose the solid volume that you cut from a **Cam-Blank.** The four surfaces are: • **Cam Flank x 2** - one flank for each working side of the Follower-Roller • **Floor Surface -** a surface to connect the **bottom** edges of the **Cam-Flanks** • **Top Surface** - a surface to connect the **top** edges of the **Cam-Flanks**. This surface should be outside of the **Cam-Blank**. To edit the 3D-CAM parameters: **See 3D-Cam dialog** 315

Video:

Double-click to watch Add a 3D-Cam

1.4.3.3 Add Rack-Pinion - Ball-Screw

Rack-Pinion / Ball-Screw

See also Rack-Pinion dialog [470]

Add Rack-Pinion removes one(1) a degrees-of-freedom from a **rotating-Part** or a **sliding-Part**.

Terminology

RACK -PINION :	A RACK-PINION is the term for a sliding, linear-gear (RACK) that is in mesh with a rotating, circular-gear (PINION). The motion of the two PARTS are kinematically-related by the **module** and **number-of-teeth** on the PINION.
BALL-SCREW :	A BALL-SCREW is a device in which a **Screw** rotates to move a **Nut**, or vice-versa. The Linear-Motion of the **Nut** and the rotation of the **Screw** are related by the **lead** of the screw-thread.
sliding-Part :	a PART that is joined to another PART with a SLIDE-JOINT.
rotating-Part :	a PART that is joined to another PART with a PIN-JOINT.

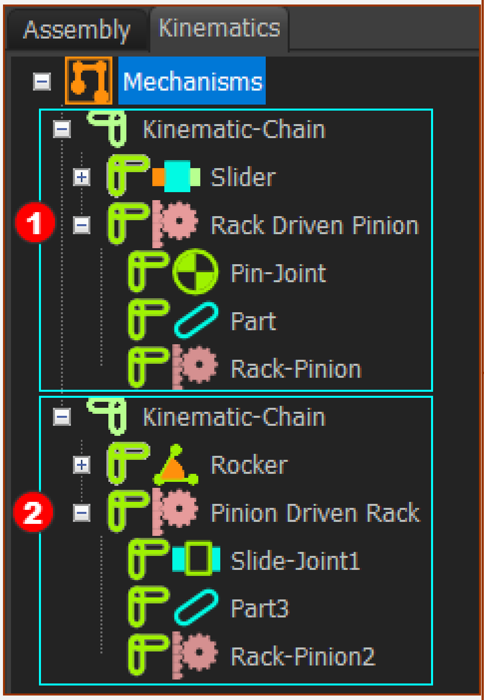
Kinematic State before you do Add Rack-Pinion.

There are three PARTS - 1,2,3.
PART(1) is joined to PART(2) with a PIN-JOINT.
PART(3) is joined to PART(2) with a SLIDE-JOINT.
PART(2) is **kinematically-defined**
PART(1) OR PART(3) is **kinematically-defined** before you do Add Rack-Pinion.

Kinematic State after you do Add Rack-Pinion.

Add Rack-Pinion removes one degrees-of-freedom.
PART(1) AND PART(3) are **kinematically-defined** after you do Add Rack-Pinion

Kinematics-Tree of Rack-Pinion



1 When the movement of the Pinion comes from the Rack

Before you do Add Rack-Pinion, there is a **sliding-Part** that is kinematically-defined and a **rotating-Part** that has one degree-of-freedom.

After you do Add Rack-Pinion, the **sliding-Part(Rack)** and the **rotating-Part (Pinion)** are **kinematically-defined.**

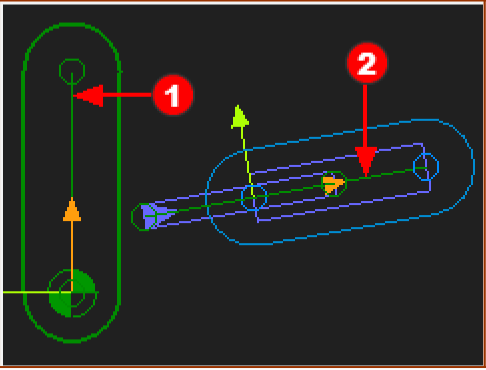
2 When the movement of the Rack comes from the Pinion

Before you do Add Rack-Pinion, the **rotating-Part** is **kinematically-defined** and the **sliding-Part** has one degree-of-freedom.

After you do Add Rack-Pinion, the **rotating-Part** (Pinion) and **sliding-Part (Rack)** are **kinematically-defined**, and in one kinematic-chain.

Preparation (typical)

When the movement of the Rack comes from the Pinion (Driving Pinion)

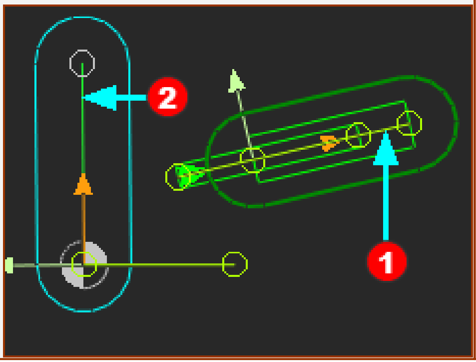


Elements before you do Add Rack-Pinion

LINE❶ - in a **rotating-Part** that is **kinematically-defined**

LINE❷ - in a **Sliding-Part** that is not **kinematically-defined**

When the movement of the Pinion comes from the Rack (Driving Rack)



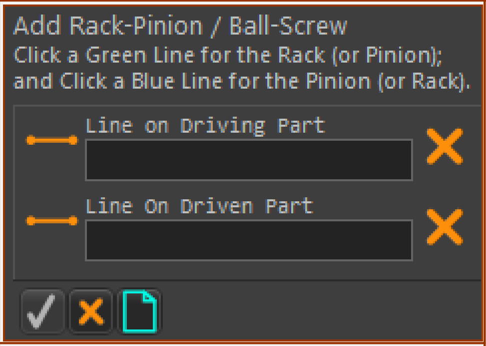
Elements before you do Add Rack-Pinion

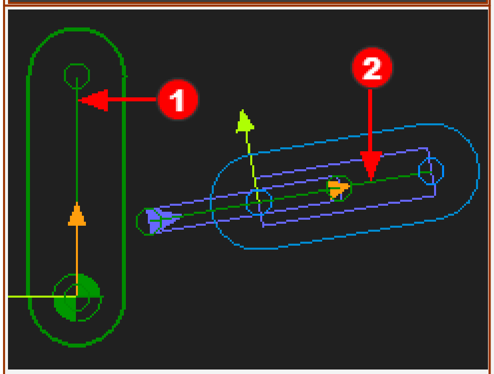
LINE❶ - in a **sliding-Part** that is **kinematically-defined**

LINE❷ - in a **rotating-Part** that is not **kinematically-defined**

Add Rack and Pinion

When the movement of the Rack comes from the Pinion (Driving Pinion)

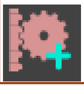




STEP 1: Click Add Rack-Pinion

1. Add menu > Mechanism menu > Add Rack-Pinion

OR



1. Machine-elements toolbar > Add Rack-Pinion

In the COMMAND-MANAGER, there are two elements to select.

STEP 2: Select the two elements in the graphic-area

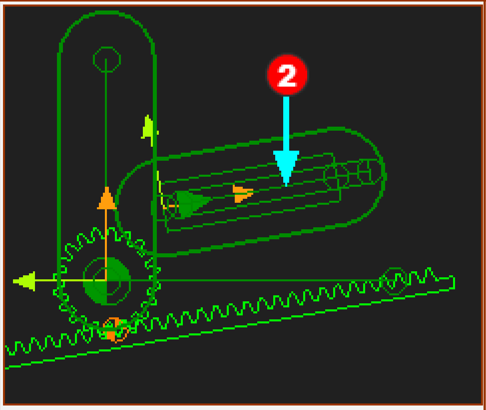
1. Click LINE❶ - it is in the PART that is **kinematically-defined.**
2. Click LINE❷ - it is in the PART that is NOT **kinematically-defined.**

STEP 3: Complete the Command

1. COMMAND-MANAGER: **Click** ✓ to complete Add Rack-Pinion.

RESULT :

The PINION is on the **Rotating-Part.**



The rotating-axis of the **PINION** is coincident with the **PIN-JOINT**.

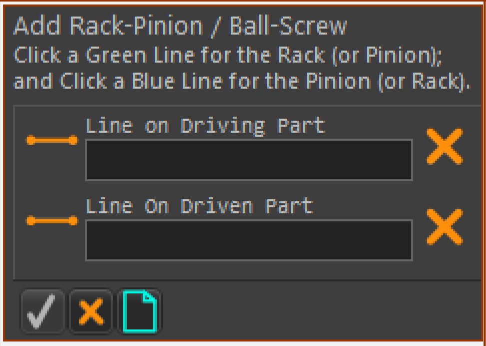
The **RACK** is on the **Sliding-Part**.

The **RACK** is parallel with the **SLIDE-JOINT**.

The **Pitch-Line** of the **RACK** is offset from the center of the **PINION** by the **Pitch-Circle Radius** of the **PINION**.

The **sliding-Part** is now a **PART** that is **kinematically-defined**.

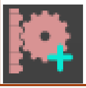
When the movement of the Pinion comes from the Rack (Driving-Rack)



STEP 1: Add Rack-Pinion

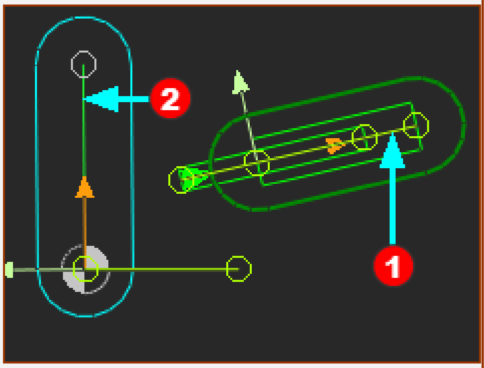
1. Add menu > Mechanism menu > Add Rack-Pinion

OR



1. Machine-elements toolbar > Add Rack-Pinion

In the **COMMAND-MANAGER**, there are two elements to select.



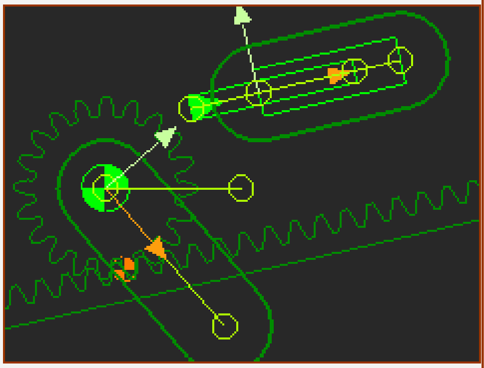
STEP 2: Select the two elements in the graphic-area

2. Click LINE❶ - it is in the **PART** that is **kinematically-defined**.
3. Click LINE❷ - it is in the **PART** that is **NOT kinematically-defined**.

STEP 3: Complete the Command

4. Click ✓ in the **COMMAND-MANAGER**.

The **RACK-PINION** is now in the **ASSEMBLY-TREE** and Graphic-Area.



RESULT :

The **PINION** is on the **Rotating-Part**.

The rotating-axis of the **PINION** is coincident with the **PIN-JOINT**.

The **RACK** is on the **Sliding-Part**.

The Pitch-Line of the **RACK** is parallel to the **Slide-Joint** and **sliding-Part**.

The **rotating-Part** is now a **PART** that is **kinematically-defined**.

Video of Add Rack-Pinion

Double-click to watch Video

1.4.3.4 Add Gear-Pair

Add Gear-Pair

See also: Gear-Pair dialog 462

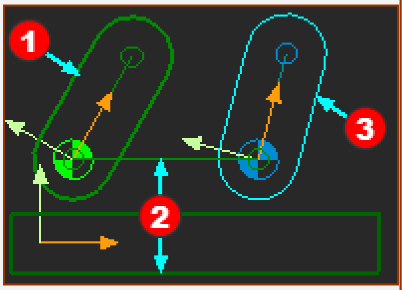
Add Rack-Pinion / Ball-Screw and Degrees of Freedom

Add Gear-Pair removes one(1) a degrees-of-freedom from a **rotating-Part**.

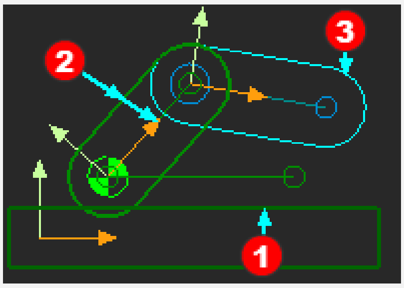
Terminology and definitions:

Gear-Wheel :	A **rotating-Part** with **gear-teeth** at a fixed diameter. The diameter is defined by the **module** of the gear-teeth and how many **gear-teeth**.
GEAR-PAIR :	Two **gear-wheels**. Their teeth are in **mesh**. The angular velocity of each **gear-wheel** is related by the number-of-teeth on each **gear-wheel**, and their **mesh**.
Driving Part :	The **rotating-Part** that is **kinematically-defined**, and whose motion is defined (even if it is stationary) before you do Add Gear-Pair
Driven Part :	The **rotating-Part** whose motion is defined after you do Add Gear-Pair by the **Driven-Part**, the NUMBER-OF-TEETH on each Gear in the Gear-Pair, and the MESH.
Mesh :	**Verb**: to engage the gear-teeth and gear-wheels with the correct phase and position. | **Noun**: the position of the gear-teeth and gear-wheels
External Mesh :	The two **gear-wheels** have **gear-teeth** that point outwards from their centers-of-rotation.
Internal Mesh :	One of the **gear-wheels** has gear-teeth that point inwards towards its center-of-rotation.
Simple Gear-Pair :	Two gear-wheels that rotate about fixed centers.
Epicyclic Gear-Pair :	One gear-wheel orbits around the center of the other gear-wheel.

Kinematic state before you do Add Gear-Pair



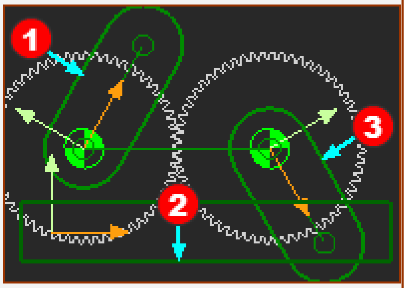
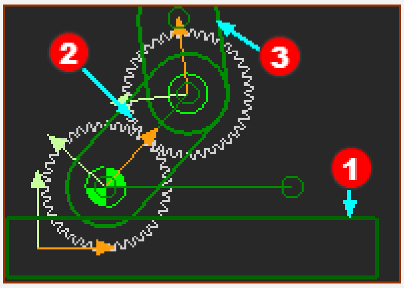
Kinematics before Add Gear-Pair Simple



Kinematics before Add Gear-Pair Epicyclic

There are three PARTS❶❷❸
PART❷ is **kinematically-defined**. There is a LINE in PART❷
PART❶ is **kinematically-defined** and it is joined to PART❷ with a PIN-JOINT.
PART❸ is not **kinematically-defined** before you do Add Gear-Pair, and it is also joined to PART❷ with a PIN-JOINT.
PART❸ is a **rotating-Part** with one degree-of-freedom.
The PIN-JOINTS must be at the START-POINT and END-POINT of the LINE in PART❷.
The LINE must have a dimension to define its length - which you add with the PART-EDITOR.

Kinematic state after you do Add Gear-Pair

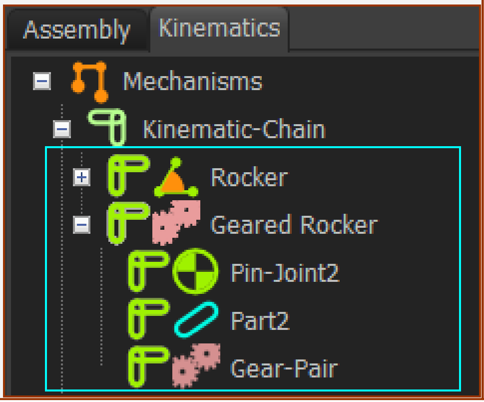
 Kinematics after Add Gear-Pair Simple	 Kinematics after Add Gear-Pair Epicyclic

Add Gear-Pair removes the one degree-of-freedom from PART❸.
The motion of PART❸ (the **Driven-Part**) is found from the motions of PART❶ and PART❷, the NUMBER-OF-TEETH, and the **mesh** on each gear-wheel in the GEAR-PAIR.

Note: The BASE-PART does not need to be one of the PARTS - all three PARTS❶❷❸ can move.

Kinematics-Tree of Gear-Pair

Add Gear-Pair removes one degrees-of-freedom.



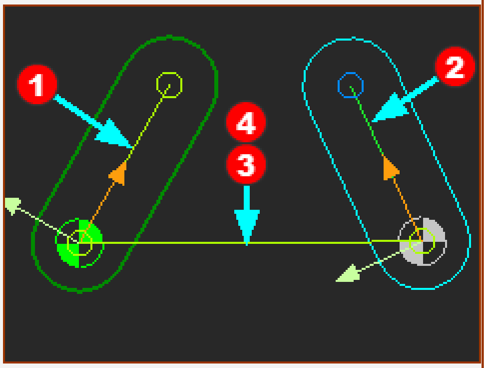
Gear-Pair

Before you do Add Gear-Pair, there is **rotating-Part** that is **kinematically-defined**, and a **rotating-Part** that has one degree-of-freedom.

After you do Add Gear-Pair, the **rotating-Part** that had one degree-of-freedom becomes a **Geared Rocker** and it is also **kinematically-defined**.

Preparation

Simple Gear-Pair



Three(3) PARTS, three(3) LINES, two(2) PIN-JOINTS ...

LINE❶ & LINE❸ are in Green PARTS that are **kinematically-defined**

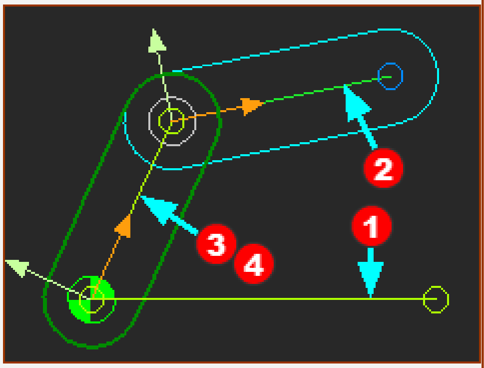
LINE❷ is in a PART that is not **kinematically-defined**

LINES❶❷ are joined with PIN-JOINTS to each end of LINE❸

LINE❸ has a DIMENSION❹ (in the PART-EDITOR)

LINE❷ is **kinematically-defined** after you do Add Gear-Pair.

Epicyclic Gear-Pairs



Three(3) **PARTS**, three(3) **LINES**, two(2) **PIN-JOINTS** ...

LINE❶ & **LINE❸** are in **PARTS** that are **kinematically-defined**

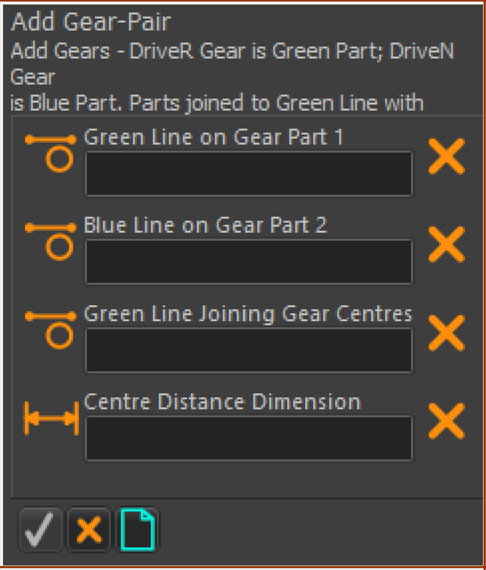
LINE❷ is in a **PART** that is **not kinematically-defined**.

LINES❶❷ are joined with **PIN-JOINTS** to each end of **LINE❸**

LINE❸ has a **DIMENSION❹** (in the **PART-EDITOR**)

LINE❷ (and the **PART**) is **kinematically-defined after** you do **Add Gear-Pair.**

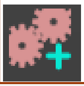
Add Simple Gear-Pair



STEP 1: Click Add Gear-Pair (Simple)

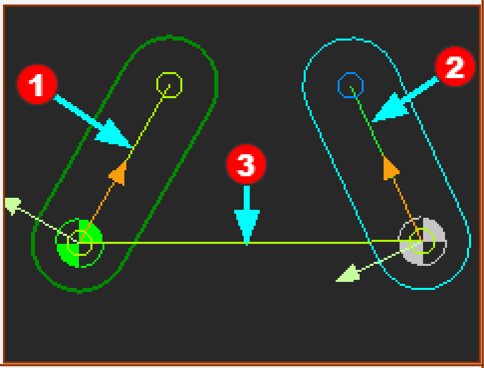
1. **MECHANISM-EDITOR**: **Add menu > Mechanism menu > Add Gear-Pair**

OR



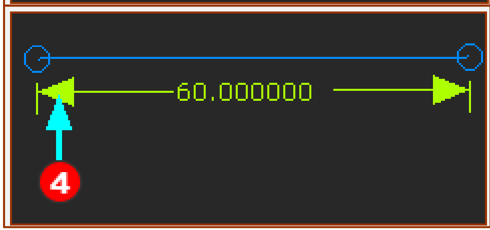
1. **MECHANISM-EDITOR**: **Machine-elements toolbar > Add Gear-Pair**

There are four elements to select in the **COMMAND-MANAGER**



STEP 2: Select the four elements

1. **MECHANISM-EDITOR** > Click **LINE❶**
2. **MECHANISM-EDITOR** > Click **LINE❷**
3. **MECHANISM-EDITORS** : Click **LINE❸**

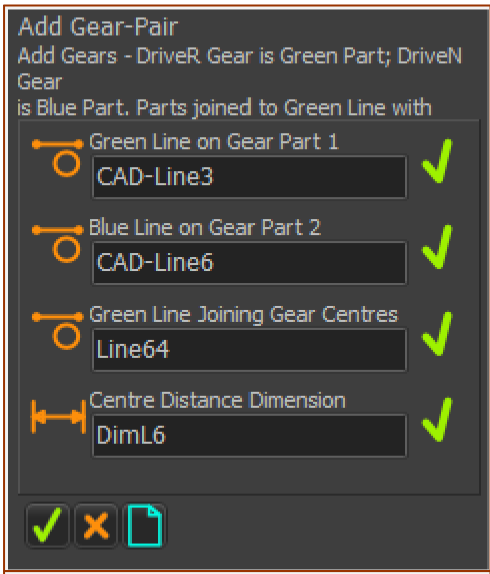


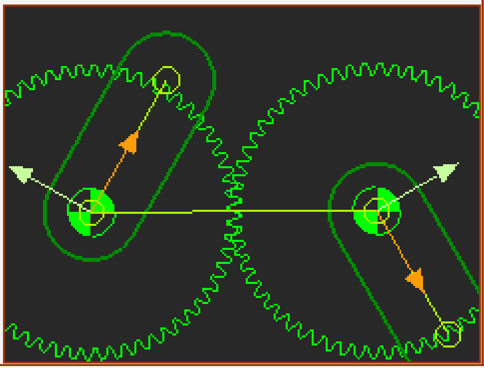
STEP 2: continued...

The **PART-EDITOR** opens automatically.

4. **PART-EDITOR**: Click the **DIMENSION❹**

The **PART-EDITOR** closes automatically.





STEP 3: Complete the Command

5. COMMAND-MANAGER: **Click ✓ in the COMMAND-MANAGER.**

RESULT:

The GEAR-PAIR is in the graphic-area.

The default GEAR-PAIR has an External Mesh and they have same number-of-teeth.

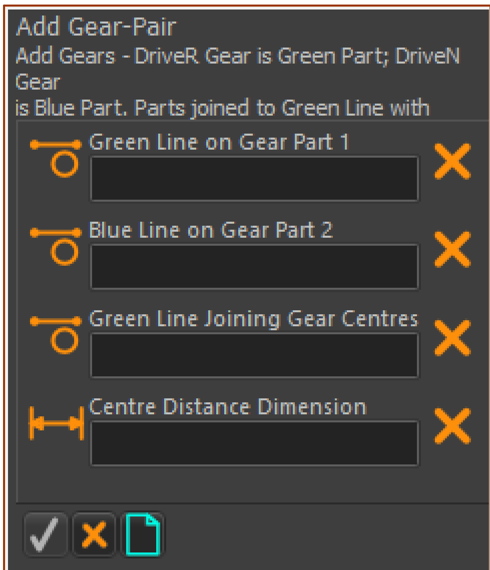
The length of the LINE between the Gear centers - called the **Line-of-centers** - becomes a **Driven-Dimension.**

The length of the Line-of-Centers = **(# Gear-Teeth 1 + # Gear-Teeth 2) × Module)**

The output PART is a Geared-Rocker in the KINEMATICS-TREE 262.

To edit the GEAR-PAIR parameters: **see Gear-Pair dialog** 462

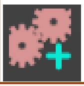
Add Epicyclic Gear-Pair



STEP 1: Click Add Gear-Pair (Epicyclic)

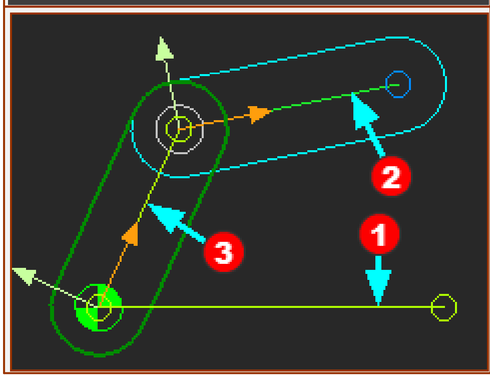
1. MECHANISM-EDITOR: Add menu > Mechanism menu > Add Gear-Pair

OR



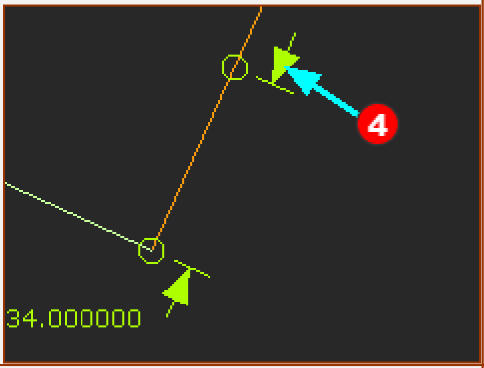
1. MECHANISM-EDITOR: Machine-elements toolbar > Add Gear-Pair

There are four elements to select in the COMMAND-MANAGER



STEP 2: Select the four elements

2. MECHANISM-EDITOR > Click LINE❶
3. MECHANISM-EDITOR > Click LINE❷
4. MECHANISM-EDITOR > Click LINE❸



STEP 2: continued...

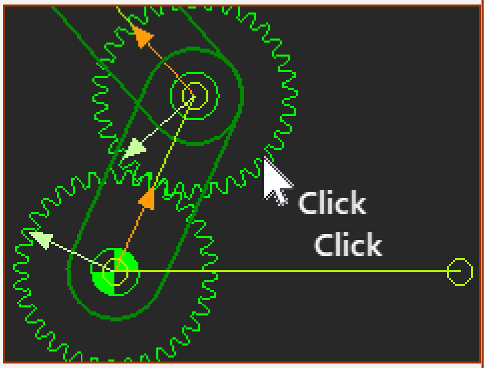
The **PART-EDITOR** opens automatically.

4. PART-EDITOR: Click the **DIMENSION** ❹

The **PART-EDITOR** closes automatically.

STEP 3: Complete the Command

5. COMMAND-MANAGER: Click ✓ in the COMMAND-MANAGER.



RESULT:

The **GEAR-PAIR** is in the graphic-area.

The default **GEAR-PAIR** has an External Mesh and the gears have same number-of-teeth.

The length of the **LINE** between the Gear centers - called the **Line-of-centers** - becomes a **Driven-Dimension.**

The length of the Line-of-Centers = **(# Gear-Teeth 1 + # Gear-Teeth 2) × Module)**

The output **PART** is a Geared-Rocker in the KINEMATICS-TREE 262.

To edit the **GEAR-PAIR** parameters, **see** **Gear-Pair dialog** 462

Videos of Add Gear-Pair

Simple Gear-Pairs: Double-click to watch 'Add Gear-Pair - Simple'

Epicyclic Gear-Pairs: Double-click to watch ' Add-Gear-Pair - Epicyclic'

1.4.3.5 Add Bevel Gear-Pair

Add Bevel Gear-Pair

See also: Gear-Pair dialog [462]

Add Rack-Pinion / Ball-Screw and Degrees of Freedom

Add Bevel Gear-Pair removes one(1) a Degrees-of-Freedom.

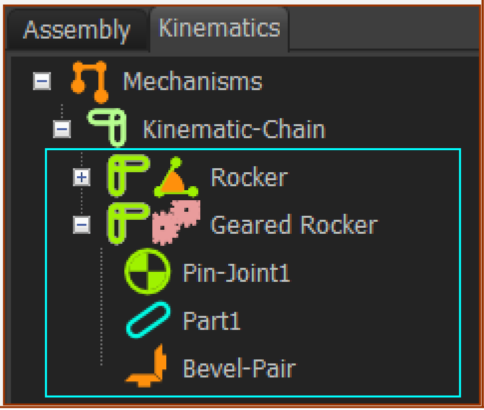
What is a Bevel Gear-Pair?

When you do Add Gear-Pair, you add two gears that are in mesh, with intersecting, rotating-axes.

The motion of a **Driven-Gear** is found from the motion of the **Driving-Gear**, the number-of-teeth, and the **mesh** (internal or external).

Kinematics-Tree of Bevel Gear-Pair

Add Bevel Gear-Pair removes one degrees-of-freedom.



Bevel Gear-Pair

Before you do Add Bevel Gear-Pair, there is a rotating-Part that is **kinematically-defined**, and a **rotating-Part** that has one degree-of-freedom.

After you do Add Bevel Gear-Pair, the **rotating-Parts** are **kinematically-defined**.

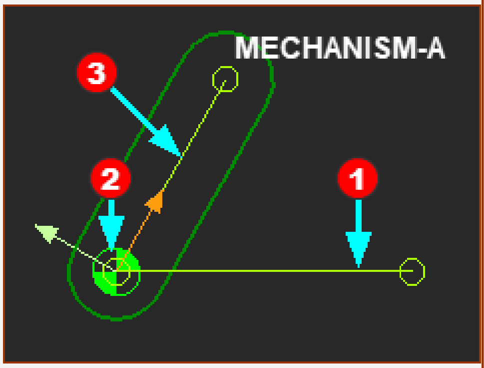
A **Bevel-Pair** is a child to the **Geared-Rocker.**

Prepare Bevel Gear-Pair

To prepare to **Add Bevel Gear-Pair** is more complex than Add Gear-Pair [119]. But you use the same command.

Compare the model before we can do Add Gear-Pair and Add Bevel Gear-Pair.

The Mechanism-Planes of the two(2) MECHANISM-EDITORS must intersect. They are frequently perpendicular to each other. For convenience, we name the two MECHANISM-EDITORS for you as **MECHANISM-A** and **MECHANISM-B.**



PREPARATION A1: MECHANISM A

LINE❶ & LINE❸ are in PARTS that are **kinematically-defined**

LINES❶❷ are joined with a PIN-JOINT

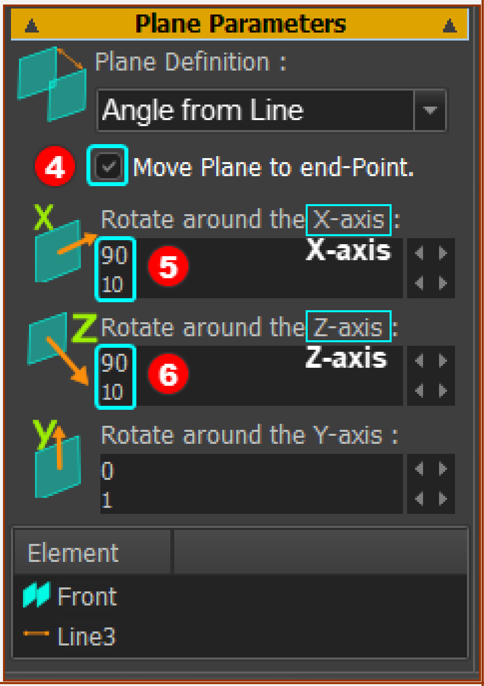
LINE❷ has a DIMENSION (in the PART-EDITOR)

See Tutorial 1: Add a Rotating-Part.

Preparation A2: Add a Plane to the Line in the Base-Part

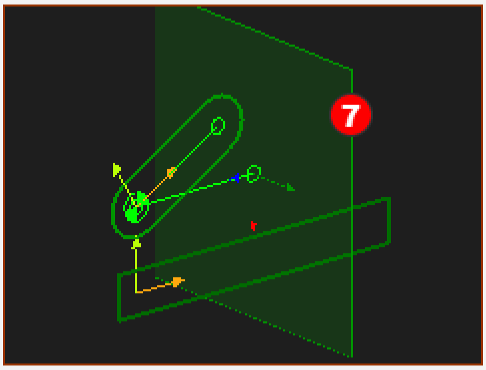
1. Click Model menu (or toolbar) > Add Plane
2. Click the LINE❶ in the BASE-PART

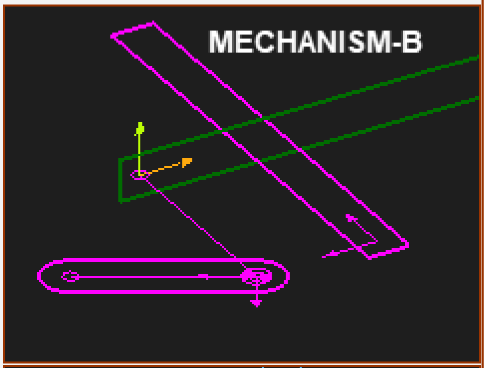
In the ADD PLANE DIALOG:

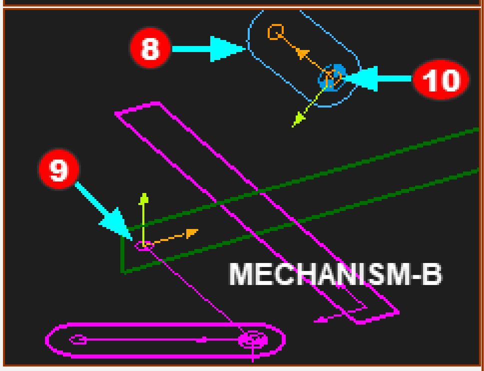


1. Click the **MOVE PLANE TO END-POINT❹** check-box
2. Enter **90º** in the **ROTATE AROUND THE Z-AXIS❺**
3. Enter **90º** (default value) is in **ROTATE AROUND THE X-AXIS❻**.

See **Tutorial 1: Add a Rotating-Part**.







Preparation A3: Add Mechanism-B to the new Plane

1. Click **Model toolbar > Add Mechanism-Editor**
2. Click the new **PLANE❼**

You jump to the new **MECHANISM-EDITOR - MECHANISM-B**

A) Enable **Visibility toolbar > Show other Kinematic and Sketch elements** [49]

B) Right-click the **Mechanism name-tabs Light-Bulb** > Enable **Show with other Kinematic and Sketch-Elements** [49]

C) Spin the model to show the other kinematic and sketch-elements

---

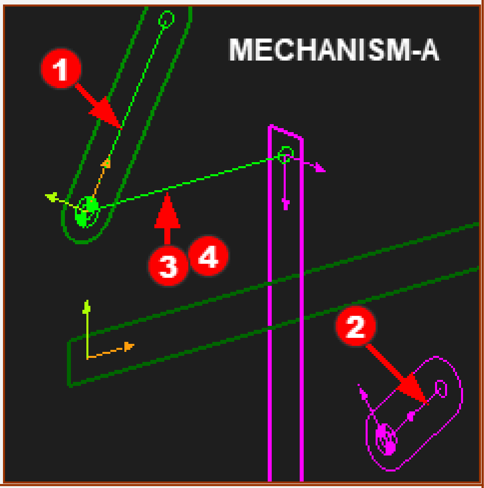
Preparation B1: Add a new Part and Pin-Joint

1. Click **Kinematic elements toolbar > Add Part** | Drag to add the **PART❽**
2. Click **Kinematic elements toolbar > Add Pin-Joint**
   a. Click the **START-POINT** of the new **PART❿**
   b. Click the **END-POINT❾** of the sketch **LINE** in the **MECHANISM-A**

The **PART❽** moves to a new place!!!

See also: Geared-Rocker, Dyads and Kinematics-Tree [262]

Add Bevel Gear-Pair

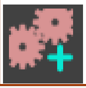


MECHANISM-A is the active MECHANISM-EDITOR - See Preparation above.

STEP 1: Click Add Gear-Pair

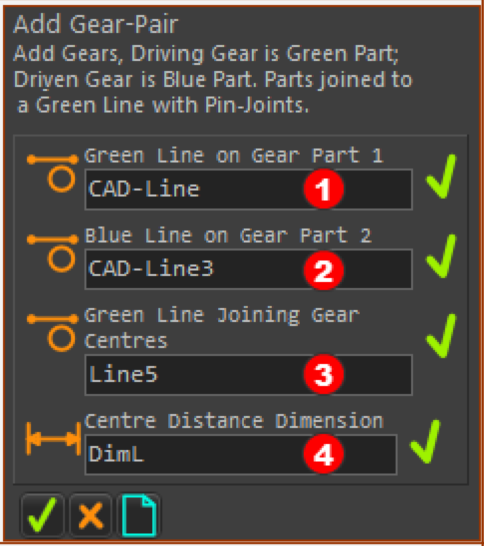
1. MECHANISM-EDITOR: Add menu > Mechanism menu > Add Gear-Pair

- or -



1. MECHANISM-EDITOR: Machine-elements toolbar > Add Gear-Pair

There are four elements to select in the COMMAND-MANAGER

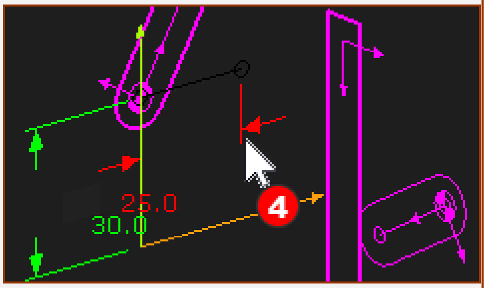


STEP 2: Select the four elements

❶ The LINE along the center of the rotating-Part

❷ The LINE along the center of the PART in the MECHANISM-B

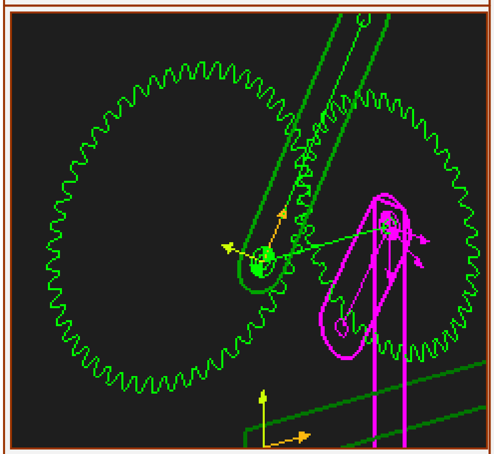
❸ The LINE in BASE-PART



The PART-EDITOR opens to show LINE❸ and its dimension❹.

STEP 3: Click the center Distance Dimension ❹

❹ Click the arrowhead, or the dimension line.



STEP 4: Complete the Command

COMMAND-MANAGER: **Click ✅ in the COMMAND-MANAGER.**

Result:

The BEVEL GEAR-PAIR is in the graphic-area.

The default BEVEL GEAR-PAIR has an External Mesh and the two gears have same number-of-teeth.

All the PARTS are **kinematically-defined**.

BEVEL GEAR PART 2 is now identified in the KINEMATICS-TREE as a Geared-Rocker

To edit the BEVEL-GEAR-PAIR parameters: **see [Gear-Pair dialog](462)**

1.4.3.6 Add Scroll

What is a scroll?

A **SCROLL** feeds products into a Rotating Machine. The products are in a buffer as they move into the start of the Scroll. At the end of the scroll, the products move into a star-wheel. The star-wheel feeds the products into the rotating machine.

The pitch of the products in the buffer, at the start of the scroll, is the same as the length of the product. The products are moved apart by the scroll until their pitch becomes the same as the pockets in the star-wheel, that is at the end of the scroll. The pitch of the products on the star-wheel is the same as the pitch of the products on the Rotary Machine.

Other names for a SCROLL are: FEEDSCREW, WORM.

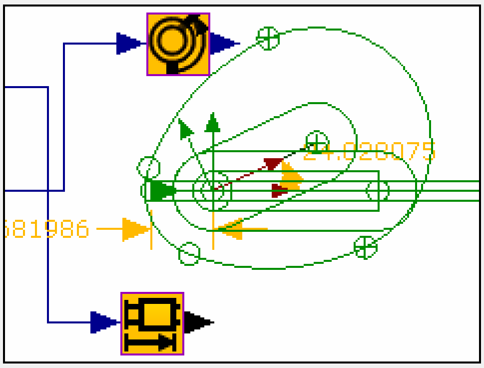
Add Scroll

	Machine Elements toolbar > Add Scroll
Menu :	Add menu > Mechanism sub-menu > Add Scroll
What to do :	*You must do the Preparation[127] before you can use 'Add Scroll'.* You need five elements: 1. A PART for the moving 'Pack'. A sketch-loop is the cross-section of the 'Pack'. You must also add a MOTION-PATH FB (see 5) and generate a number of MOTION-POINTS to the sketch-loop. 2. A PART that is the rotating SCROLL. 3. A sketch-loop and PROFILE that is the cross-section of the SCROLL. 4. A LINE to be the SCROLL'S axis-of-rotation. 5. A MOTION-PATH FB.
Result :	A SCROLL in the graphic-area.
Preparation :	There is a significant amount of preparation. You must prepare five elements in 2 MECHANISM-EDITORS. **See Preparation**[113] When you click Add Scroll, you see five selection-boxes in the COMMAND-MANAGER.

Preparation for 'Add Scroll'

STEP 1: Pack: Shape; Motion, and Motion-Points

> The Pack moves along a line that is parallel to the scroll's rotation-axis. It is also possible to rotate the Pack on its own axis as it moves from the entry to the exit of the scroll.



1. Add a Mechanism to a 'Plane' - the 'Front' Plane - in the Model-Editor.
2. Rename the Mechanism-Editor to 'Pack'.

In the new Mechanism-Editor:

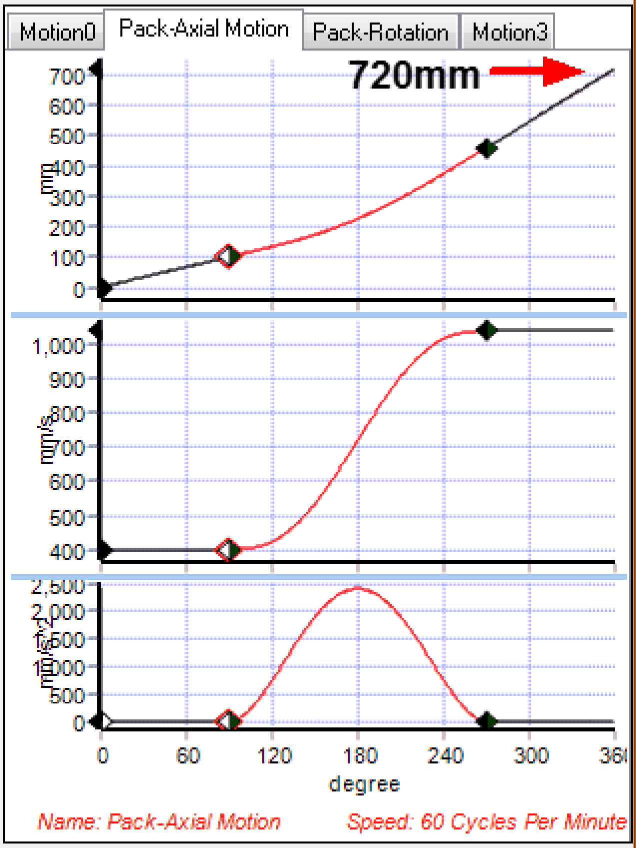
3. Add a Slider with a motion to define the axial motion of the Pack

If the Pack rotates as it moves along the scroll:

4. Add a Rocker to the Slider Part, and a motion to define the rotation of the Rocker / Pack
5. Then add the shape of the Pack to the Slider or Rocker.

The shape of the Pack must be a sketch-loop. If the Pack is not prismatic, then the shape should represent the its cross-section at the height of the scroll's rotational-axis.

The Motion for the Axial and Rotation of the Pack



Click to Expand and Collapse

Pack Axial Motion

The motion usually has a '*Constant Velocity Segment*' as its first and last segment.

Thus, there are two(2) Velocities you must consider.

Start Velocity, (SV)

Packs are usually adjacent '(bunched up', 'back-to-back')

SV = Product-Length ÷ Time for scroll to rotate one time

Final Velocity, (FV)

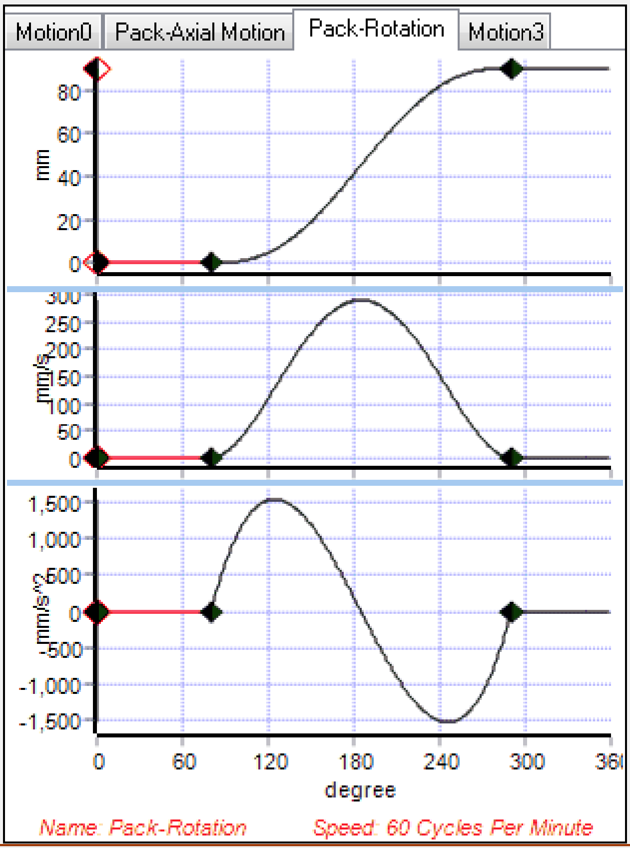
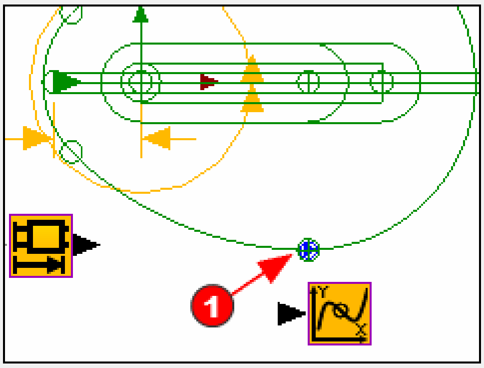
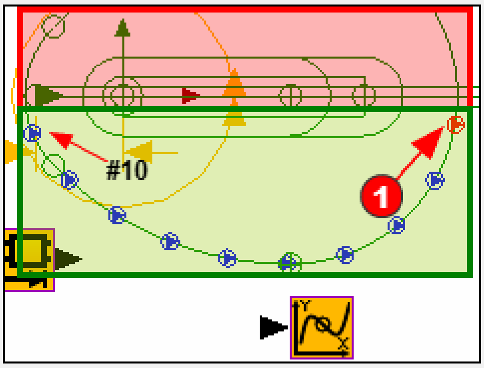
Packs should be at the same pitch as the star-wheel or circular pitch of the Rotary Machine

FV = Rotary Pitch ÷ Time for scroll to rotate one time

'Time', or 'Number of Machine Degree', for Scroll to rotate one time.

This is a function of the number-of-times the scroll rotates. Thus, of the length of the Scroll

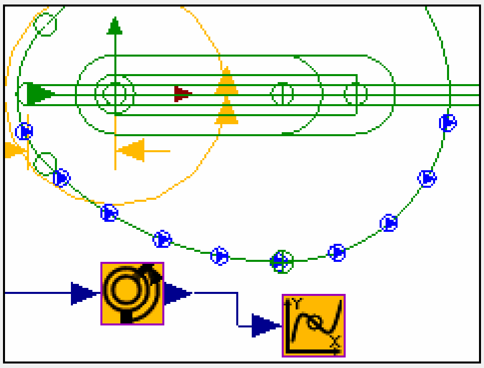
The first constant velocity is the speed of the bottle at its entry.

	The last constant velocity is the speed of the bottle at its exit. You need to edit the motion later.
	Pack Rotation The Pack rotates by 90° as it moves from the entry to the exit. The duration of the segment that rotates the pack is the same duration as the segment that accelerates the pack.
Motion-Path, Motion-Points.	
	In order to calculate the shape of the scroll, you must define the contact between the Pack and the Scroll. Rather than a Profile, you add Motion-Points to the shape of the Pack. Each Motion-Point define a 'Rim' along the scroll - as you see. **1.** *Click* Add Motion-Path **2.** *Click* one of the sketch-elements that give the shape to the Pack One Motion-Point 1 shows at the start-Point of the sketch-element that you click.
	1. *Double-click* the Motion-Path FB to open the Motion-Path dialog 2. Edit the '# Motion-Points' to 10 - we can edit this number later, if necessary. 3. *Click* the **'RE-GENERATE MOTION-POINTS'**. 4. Edit the 'Range' of the Motion-Points to 160° (use the Range option in degrees).

5. Edit the **'BASE-VALUE'** of the **MOTION-POINTS** until they are <u>only</u> in the half of the sketch-loop for the Pack that 'faces' the rotation-axis.

The Motion-Points must be in the **'Green Area'** The **MOTION-POINTS** cannot be in the **'Red Area'**.

A Motion for the Motion-Points

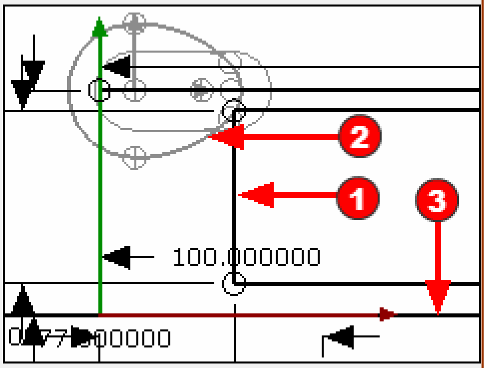


As the Pack rotates, we must also rotate the Motion-Points in the opposite direction so that they remain in the half of the Pack that faces the rotation-axis.

1. Connect a wire from the output-connector of the **MOTION-DIMENSION** that rotates the Pack to the input-connector of the Motion-Path.

You may need to add a **GEARING FB**, with a ratio of -1, to move the Points in the opposite direction.

STEP 2: Add the Scroll rotation-section and its rotation-axis.



(Note: I have rotated the Pack by 180º on the rotating Part. This helps later)

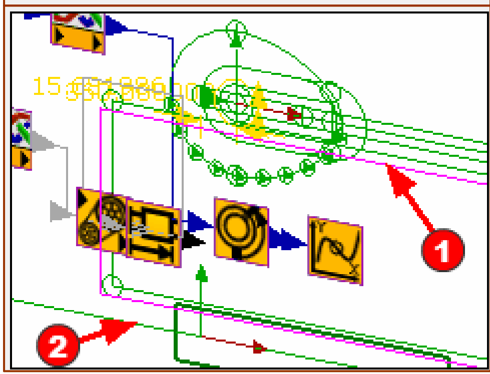
We must add a sketch-loop that represents the rotation-section of the scroll.

1. Edit the **BASE-PART**
2. Click **Visibility toolbar > Show/Hide other Kinematic and Sketch elements** - this shows the shape of the Pack ❷.
3. Add a rectangle❶, that represents the section-of-rotation of the Scroll-Blank.

The rotation-section needs a rotation-axis.

4. Add a **LINE**❸ that represents the rotation-axis of the scroll.

This should be along the X-axis of the **BASE-PART**.



We must also add a **PROFILE** to the rotation-section❶.

1. Close the **PART-EDITOR**
2. Add a **PROFILE**❶ to the rectangle

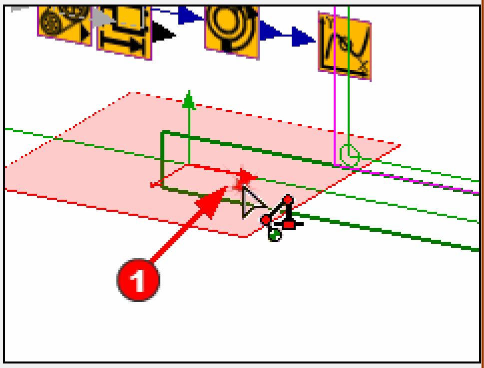
When we send the rotation-section and rotation-axis to SolidWorks, SolidWorks makes a 'revolved-section'.

STEP 3: Add a Plane, Mechanism, add the Scroll Part

We add the scroll in a different Mechanism-Editor. We must add a new Plane for the new Mechanism-Editor.

The Plane to which we add the new Mechanism-Editor must be orientated in a particular direction. If the Plane is not in the correct orientation, we cannot add the scroll.

Add the new Plane



1. Spin the model with your keyboard arrow keys
2. Click Add Plane
3. Click the arrowhead of the X-axis at the Origin of the Base-Part ❶

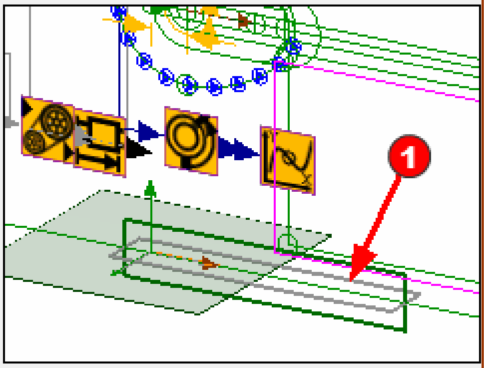
(you can also click the Y-axis)

The image shows the **default position** of the new Plane.

We must use the Add Plane dialog to re-orientate the Plane.

It is possible to use the Add Plane dialog to edit the orientation of the new Plane now.

However, I find it easier to 'Add a new Mechanism' to the Plane now, and then edit the Plane for the new Mechanism later.

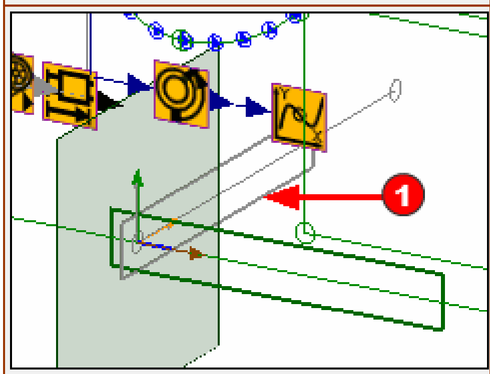


4. Close the Add Plane **dialog**
5. Click Add Mechanism
6. Click the new Plane
7. Click OK in the COMMAND-MANAGER

You 'jump' to the new Mechanism-Editor. Rename the new Mechanism-Editor to 'Scroll'

8. Click the 'Pack' Mechanism name-tab to return to the Pack Mechanism-Editor.
9. Click **Tools menu (or toolbar) > 'Show Sketches in Mechanism'**

You can now see the new Plane and the Base-Part ❶ (in **Grey**) to give the orientation of the new Mechanism-Plane. The XY-axes of the Base-Part are coplanar with Mechanism-Plane.



10. Edit the new Plane - double-click it to open the Plane dialog

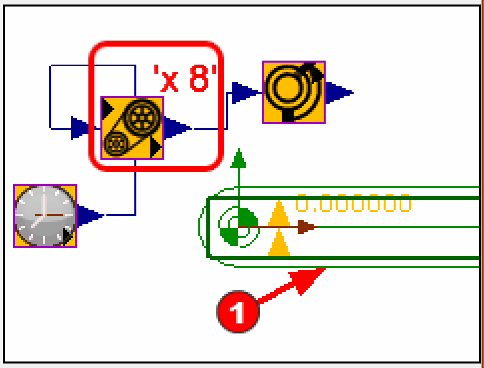
MechDesigner Rules - relative orientation of the new Plane with respect to the orientation of the Pack's Mechanism Plane.

- The Z-axis of the Scroll Mechanism's Base-Part must be co-axial with the X-axis of the Pack Mechanism's Base-Part
- The Y-axis of the Scroll Mechanism's Base-Part must be co-axial with the Y-axis of the Pack Mechanism's Base-Part

If you clicked the X-axis to add the new Plane, then the X, Y and Z-axis rotations are 90,90,90 respectively.

If you clicked the Y-axis to add the new Plane, then the X, Y and Z-axis rotations are 0,90,90 respectively.

The image shows the new orientation of the Base-Part and Mechanism-Plane of the 'Scroll' Mechanism.



11. Click the Scroll Mechanism name-tab

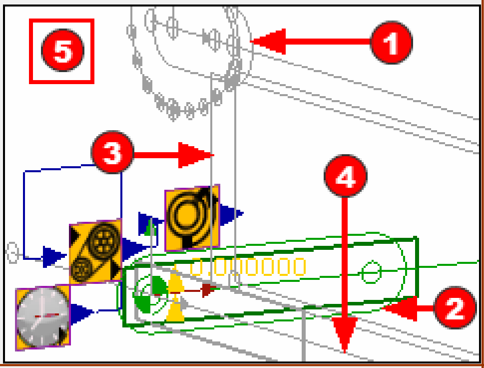
We now add the Part that represents the 'scroll'.

12. Add a rotating Part❶.

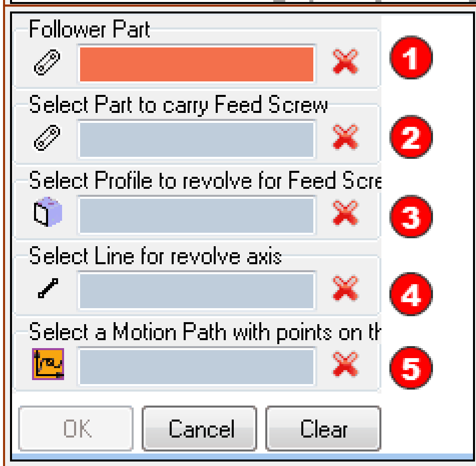
Make sure the Origin of the Part is at the Origin of the Base-Part.

Add a Gearing FB, and edit its Gearing Ratio parameter so that the Scroll Part rotates many times in a machine-cycle.

I suggest a Gearing Ratio of 'x8' as the first estimate.



13. Click **ALT+H** keyboard shortcut to Home the model, in which the MMA is at 0
14. Rotate the view so you can see the elements in the two(2) Mechanism-Editors.
15. Click Add Scroll.



Select the five elements in order.

❶ The Follower Part is the 'Pack' Part - in grey

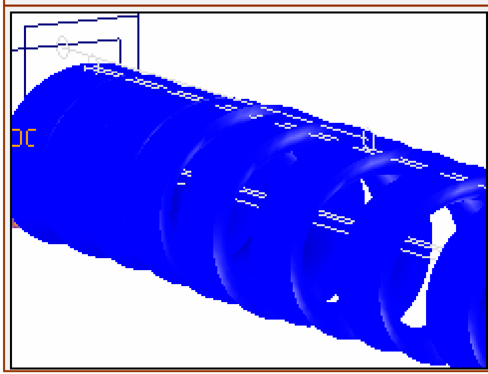
❷ The Part to carry the Feed-Screw is the 'Scroll', in the active Mechanism-Editor

❸ The Profile is the 'rotation-section' in grey - the Profile element

❹ The Line for the 'revolve axis' is the 'rotation-axis' in grey

❺ The MOTION-PATH FB in the graphic-area

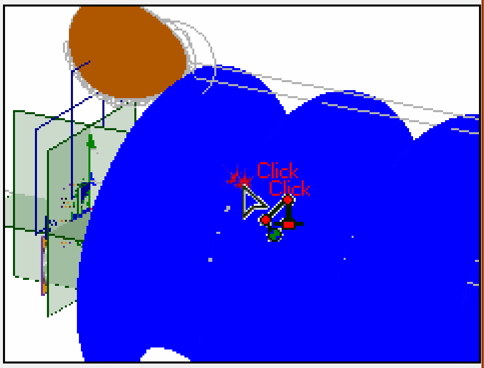
Click OK



You may need to click the Model-Editor to force a full model rebuild.

MechDesigner creates a surface for the part of the scroll that becomes in contact with the Pack, as the Pack advances.

STEP 4: Transfer the Scroll to SOLIDWORKS

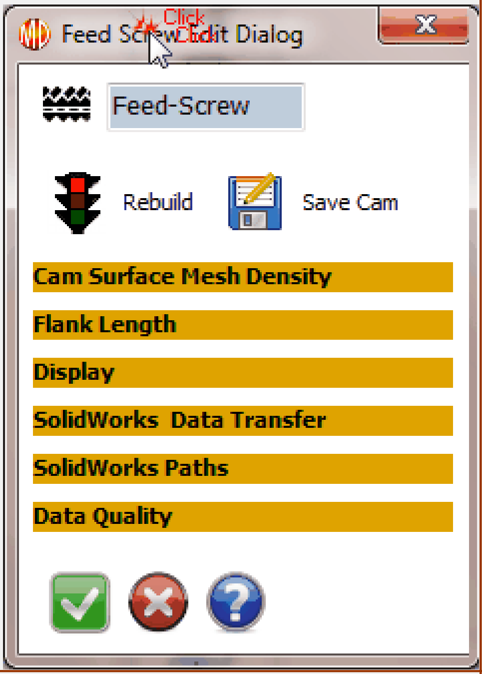


To transfer the Scroll to SOLIDWORKS, we need to edit the Scroll element.

1. *Double-click* the Scroll in the graphic-area

or

1. *Double-click* the Scroll element in the Assembly-Tree



The Scroll Element dialog (Also called the Feed Screw - dialog).

The important separators are:

Cam (that is: Scroll) <u>Surface Mesh Density</u>

- Edit the number of Points along each Rim.
- Typically 500 to 1500.

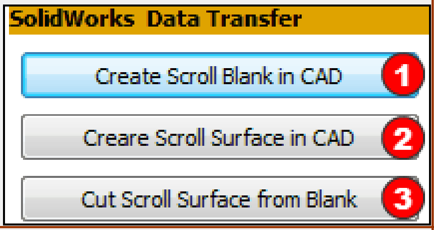
Flank-Length

The Flank Length is with respect to the MMA.

- Typically, set to 0 to 360

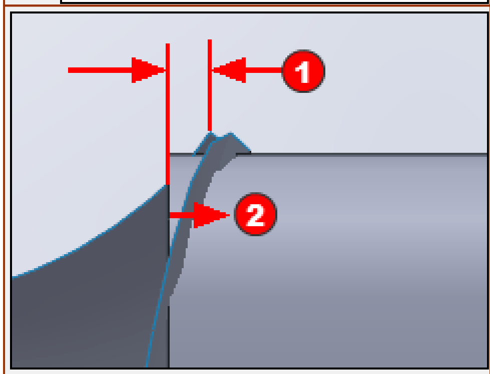
Display

- color
- Transparency - applies to view in the Model only
- Show as Solid (Surface) and Rims check-boxes.



SOLIDWORKS Data Transfer

1. Start SOLIDWORKS, add a new Part, and Save it.
2. You can click each button in turn



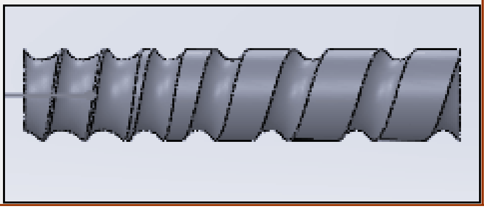
If the Scroll Surface is not cut from the Scroll Blank, there may be a couple of things you should look at.

1. The end of the Scroll Surface is outside the ends of the Scroll Blank.❶

In the image, you can see that the Scroll-Blank extends past the start of the Scroll Surface.

Edit the Scroll-Blank in SOLIDWORKS or ***MechDesigner***.

1. Move the left end of the sketch to the right by a small amount.❷

	When the Scroll Surface is cut from the Blank, then hide the Scroll Surface.

1.4.3.7 Add Pulley

Add Pulley

See also : Pulley dialog [474]

Terminology and definitions:

PULLEY :	A rotating-Part. It is a wheel with teeth around its circumference at a fixed radius. The teeth engage with a **Belt**. The angular motion of the PULLEY and the linear motion of a **Belt** are related by the radius of the PULLEY. The NUMBER-OF-TEETH and the TOOTH-PITCH specify the radius of the PULLEY
There are two types of PULLEY.	
Type 1: Driving Pulley :	The pre-defined angular motion of the PULLEY specifies the linear motion of the **Belt**.
Type 2: Driven Pulley :	The pre-defined linear motion of the **Belt** specifies the angular motion of the PULLEY.
Belt:	It is a **Timing-Belt** (we do not show you the Teeth on the **Belt**). The **Belt** does not slip relative to the Teeth on the PULLEYS. The motion of a MOTION-POINT on a **sketch-path** that represents the motion of the **Belt;** the shape of the **sketch-path** represents the path of the **Belt**. The **sketch-path** has an ARC for each PULLEY. A PULLEY must rotate about the CENTER-POINT of an ARC.

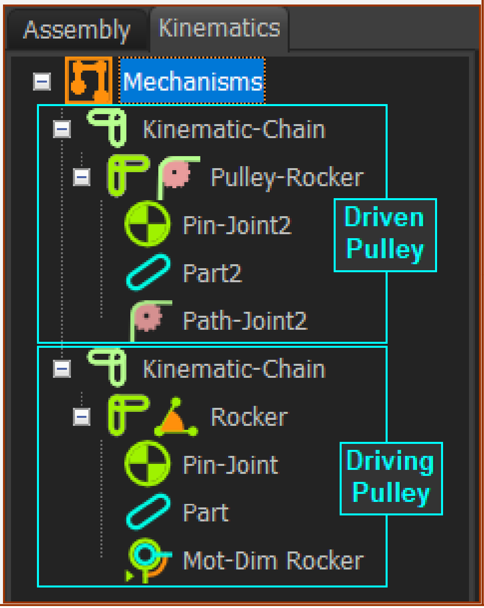
Kinematic State before you do Add Pulley.

Type 1: Driving-Pulley: A **rotating-Part** is **kinematically-defined**. **Type 2: Driven-Pulley:** A **rotating-Part** is **not kinematically-defined**.

Kinematic State after you do Add Pulley.

Type 1: The **rotating-Part** is **kinematically-defined.** It is a **Driving-Pulley**. The motion of the PULLEY, its NUMBER-OF-TEETH, and the TOOTH-PITCH now specify the motion of the MOTION-POINT along the **sketch-path** (the **Belt**). **Type 2: The rotating-Part is kinematically-defined.** It is a **Driven-Pulley**. The motion of the MOTION-POINT along the **sketch-path** (the **Belt**), the TOOTH-PITCH and the NUMBER-OF-TEETH on the PULLEY, now specify the motion of the PULLEY.

Kinematics-Tree of Pulley



Driven-Pulley

It is a PULLEY-ROCKER in the **Kinematics-Tree** after you do **Add Pulley**.

A PATH-JOINT is a child to the PULLEY-ROCKER.

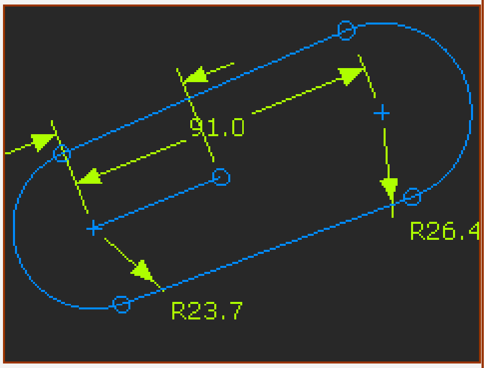
Driving-Pulley

It is a PART that is **kinematically-defined** in the **Kinematics-Tree** before you do **Add Pulley**.

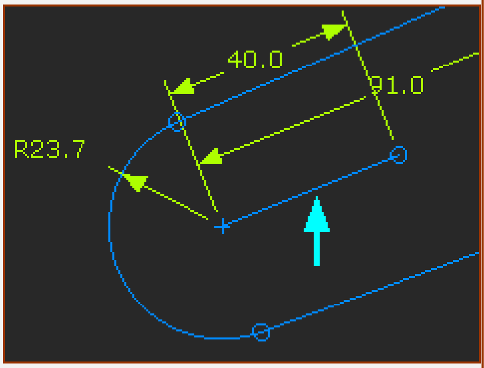
For example, a **Driving-Pulley** in the KINEMATICS-TREE is a **Rocker**.

PREPARATION (TYPICAL)

Prepare the sketch-path for the Belt.



Sketch-loop as the path of the Belt.



A Line for a Motion-Dimension FB

PREPARATION 1: Add a sketch-path to represent the path of the Belt.

The **sketch-path** should include only LINES and ARCS. Usually, add TANGENT constraints between each LINE and ARC. Do not add EQUAL constraints to the ARCS. The **sketch-path** is usually a closed **sketch-loop**. It is usually helpful to fully define the geometry of the sketch-path.

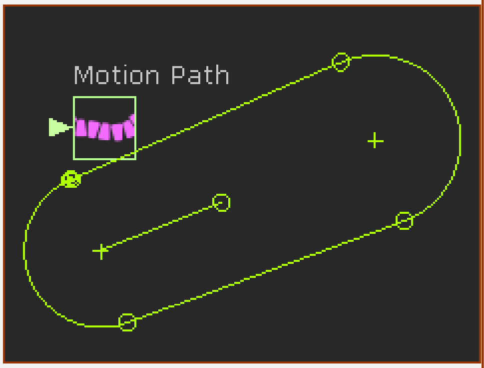
1. MECHANISM-EDITOR: Edit the BASE-PART (usually) or an ADDED-PART
2. PART-EDITOR: Add a **sketch-path** to represent the path of the **Belt**
3. PART-EDITOR: Add TANGENT constraints between each LINE and ARC.
4. PART-EDITOR: Add a dimension to the radius of each ARC. (Do not add EQUAL constraints between ARCS)

IF the **rotating-Part** is a **Rocker**:

5. PART-EDITOR: Add a LINE from the CENTER-POINT of the ARC that is the rotating-axis of the **Rocker** / PULLEY

You need the LINE to add a MOTION-DIMENSION FB to the **rotating-Part**.

Add the Motion-Path FB to the sketch-path



PREPARATION 2: MOTION-PATH FB

Add ONE MOTION-PATH FB to the **sketch-path**.

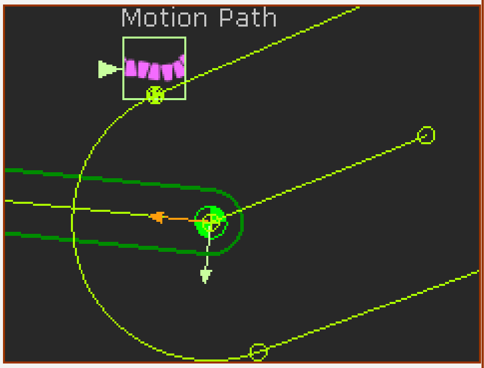
1. Click Kinematic FB toolbar > Add Motion-Path FB.
2. Click a sketch-element in the **sketch-path**

The MOTION-PATH FB is in the graphic-area.

A MOTION-POINT is at the START-POINT of the sketch-element you click.

Prepare the Rotating-Parts that become the Pulleys

A PULLEY is a **Driving-Pulley** or **Driven-Pulley**. It is a **rotating-part**.



rotating-Part for a Driving-Pulley

PREPARATION 3A: TYPE 1: DRIVING-PULLEY

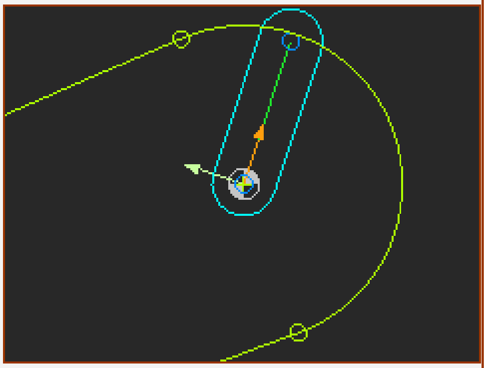
A **Driving-Pulley** is a **rotating-Part** that is **kinematically-defined.**

Add a maximum of ONE **Driving-Pulley** to a Belt.

3. Add a ROCKER with its PIN-JOINT at the CENTER-POINT of the ARC with a LINE (see PREPARATION 1: 5, above)

OR

3. Add PARTS to the model, in which there is a **rotating-Part** with its PIN-JOINT at the center of the ARC in the **sketch-path** of the **Belt** - it is **kinematically-defined.**



rotating-Part for a Driven-Pulley

PREPARATION 3B: TYPE 2: DRIVEN-PULLEY

The motion of the BELT and the **Pitch-Circle Diameter** specify the motion of **Driven-Pulleys.**

The **rotating-Part** is not **kinematically-defined** before you do Add Pulley.

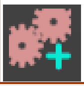
4. Add **rotating-Parts** to be **Driven-Pulleys**, with a PIN-JOINTS at the CENTRE-POINT of each ARC with a **Driven-Pulley**.

ADD PULLEY

STEP 1: Click Add Pulley

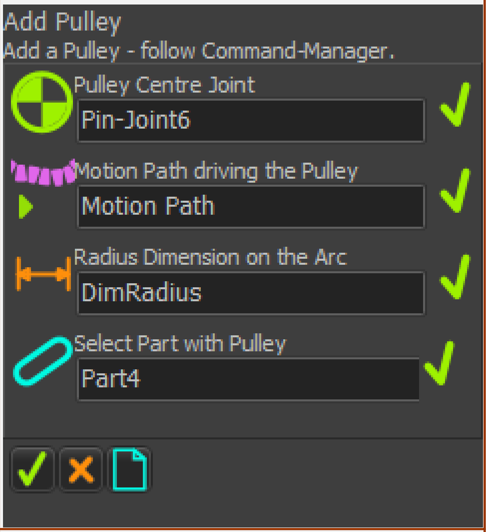
1. MECHANISM-EDITOR: Add menu > Mechanism menu > Add Pulley

- or -



1. MECHANISM-EDITOR: Machine-elements toolbar > Add Pulley

There are four elements to select in the COMMAND-MANAGER



STEP 2: Click the four elements

1. Click a PIN-JOINT at the center of a **rotating-Part***.
2. Click the MOTION-PATH FB.

The **PART-EDITOR** opens

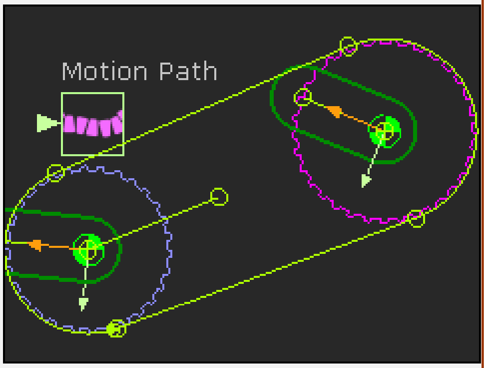
3. Click the dimension that defines the radius of the ARC.

The **PART-EDITOR** closes

4. Click the **rotating-Part*** that has a PIN-JOINT (**see 1**) at the CENTER-POINT of the ARC (**see 3**)

* If the **rotating-Part** is :

- is **kinematically-defined** - the motion of the PULLEY moves the BELT (**Driving-Pulley**).
- is not **kinematically-defined** - the motion of the BELT moves the PULLEY (**Driven-Pulley**).



1 x Driving-Pulley (Purple)
1 x Driven-Pulley (Magenta)

Do Add Pulley to add 1 x PULLEY.

STEP 3: Complete the Command

5. COMMAND-MANAGER: **Click** ✓ **in the COMMAND-MANAGER.**

RESULT:

The image shows two PULLEYS. You must do Add Pulley two times.

New PULLEY schematic shows in the graphic-area.

- Use the MOTION-PATH DIALOG 380 to edit the TOOTH-PITCH and LENGTH of the **Belt**.

The PULLEY has an integer number-of-teeth.

- Use the PULLEY DIALOG 474 to edit the NUMBER-OF-TEETH on each PULLEY.

Video: Add Pulley:

Double-click to watch Video

1.4.3.8 Add Conjugate Cam FB

Add Conjugate Cam FB

See Conjugate-Cam dialog 353

About Conjugate Cams

The design of a cam-system must make sure the Follower (usually a Roller) is in continuous contact with the cam-profile at all machine-speeds. There are two(2) design options:

A. **Force-Closed Cam** - an external force pushes or pulls the **Follower** against the **Cam-Profile**. For example, a Spring, Air-Cylinder, Gravity, Magnet, ...

B. **Body-Closed (Conjugate) Cam** - a minimum of two(2) **Followers** in the **Follower-Part** are in continuous contact with two(2) **Cam-Profiles** in the **Cam-Part**. The design prevents the **Followers** moving away from the **Cam-Profiles**.

Note: A Groove-Cam is also a Body-Closed Cam. It can be manufactured with one Follower. However to do the force analysis in which the contact-force moves from flank to flank, you need to add two 2D-CAMS - an Inner Cam and an Outer Cam with two concentric Followers, and also add a CONJUGATE-CAM FB.

Why use a Conjugate Cam FB?

Typically, to do the force and stress analysis of a 2D-CAM, you select one 2D-CAM as the Power Source in the Configure Power-Source dialog 193.

However, if two Cams (called Conjugate Cams) that drive the same follower system, you must add and configure a CONJUGATE CAM FB. Then select it as the Power-Source in the Configure Power-Source dialog, before you can do the force and stress analysis correctly. If you do not use and configure a CONJUGATE CAM FB, the Contact-Force is not correct.

2D-Cam: Work-flow

1. Add a 2D-CAM - **see Machine elements toolbar > Add 2D-Cam** 109

If the new 2D-CAM is one from a pair of **Conjugate-Cams**, or it is one flank of a **Groove-Cam**

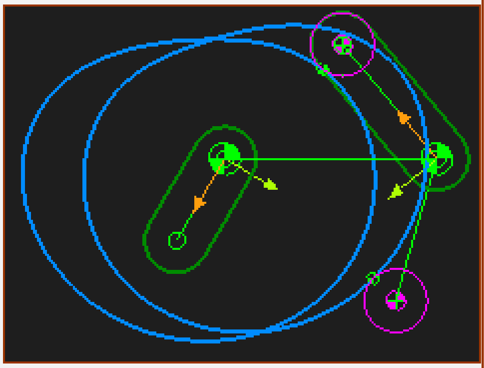
1.a. Add a CONJUGATE-CAM FB - **see Machine elements toolbar > Add Conjugate Cam FB** 139

1.b. Edit the CONJUGATE-CAM FB to select least two 2D-CAMS - **see Conjugate-Cam dialog** 353.

2. Select a 2D-CAM or a CONJUGATE-CAM FB as the Power Source for the Follower - **see Configure-Power Source** 480
3. Review the 2D-CAM : **Display, Properties, Roller-Life, Cam-Life, ...** - **see 2D-Cam dialog** 327

4. Add a CAM-DATA FB - **see Kinematic FBs > Add Cam-Data FB** 171
5. Edit the CAM-DATA FB to link it to the 2D-CAM- - **see Cam-Data dialog : Cam Analysis** 357
6. Connect wires from the CAM-DATA FB to a GRAPH FB to analyse 5 parameters - **see Cam-Data dialog : Cam Analysis** 357
7. Edit the CAM-DATA FB again to calculate the Cam's Coordinates - **see Cam-Data dialog : Cam-Coordinates** 361

Preparation



Preparation: The elements in the model before you do Add Conjugate-Cam FB:

- Two(2) 2D-CAMS on one PART. For example, a Cam-Shaft
- Two(2) or more Cam-Follower* on **ONE(1)** PART. For example, a **Rocker**

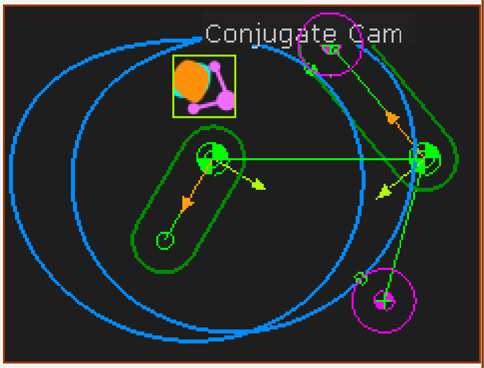
Each 2D-CAMS must display one cam-profile - the Inner or Outer.

See 2D-Cam dialog 333

* Rollers and/or Flat-Faced Followers.

See also Shape of the Cam-Follower.

Add Conjugate Cam FB



STEP 1: Add the Conjugate-Cam FB

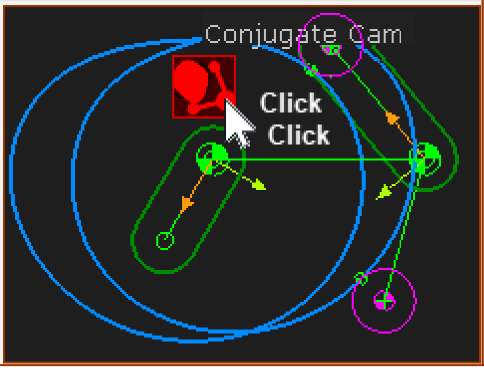
1. MECHANISM-EDITOR: Add menu > Mechanism menu > Add Conjugate-Cam FB

- or -

1. MECHANISM-EDITOR: Machine-elements toolbar > Add Conjugate-Cam FB

The CONJUGATE-CAM-FB is now in the graphic-area.

Edit Conjugate Cam FB



STEP 1: Edit the Conjugate-Cam FB

1. Double-click the CONJUGATE-CAM FB in the graphic-area

or

1. See How to open a dialog 513

The CONJUGATE-CAM DIALOG is now open.

STEP 2: **See Conjugate-Cam dialog 353**

1.4.4 Kinematic Function-Blocks

Kinematic Function-Blocks (FB)

Kinematic Function-Blocks are the 'brains' of kinematic-chains.

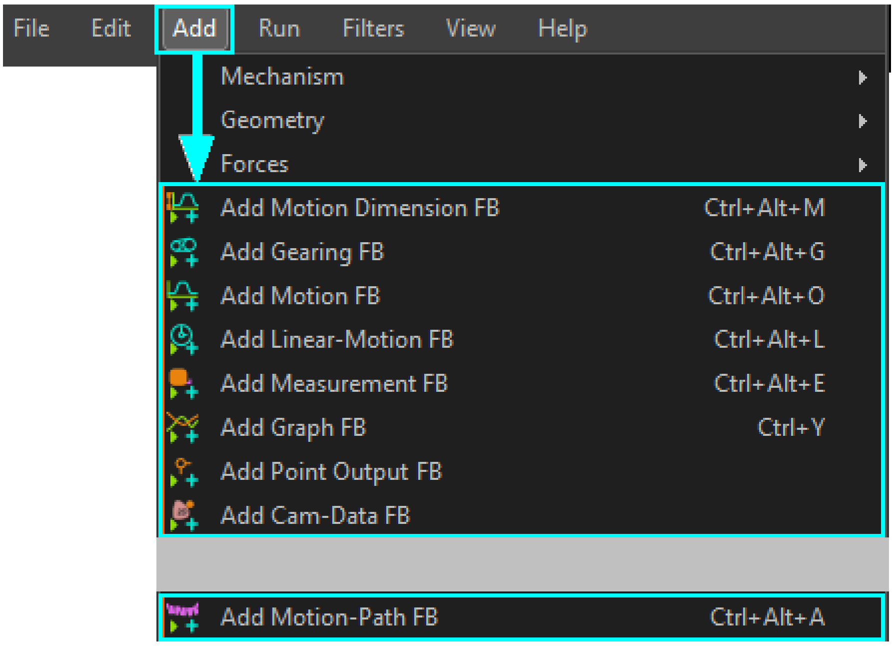
Connect FUNCTION-BLOCKS with wires. **Motion-data*** flows instantly through the wires and the FUNCTION-BLOCKS that you connect together.

Use FUNCTION-BLOCKS to plan and control the motion of kinematic-chains, to measure and plot linear or angular motion over a machine-cycle and also to calculate and export CAM-COORDINATES.

* **Motion-data** - Position, Velocity, Acceleration with Linear or Angular motion-units[292].

See also How to connect FBs[144] ; Forces menu and toolbar[187]

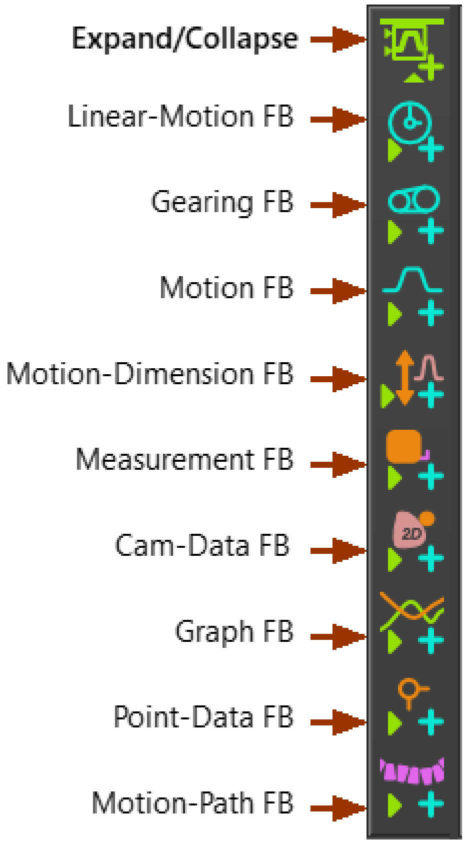
Kinematics FB menu (from Add menu)



Kinematic FB menu

Kinematics FB toolbar

The **Kinematic FB toolbar** is to the right of the graphic-area



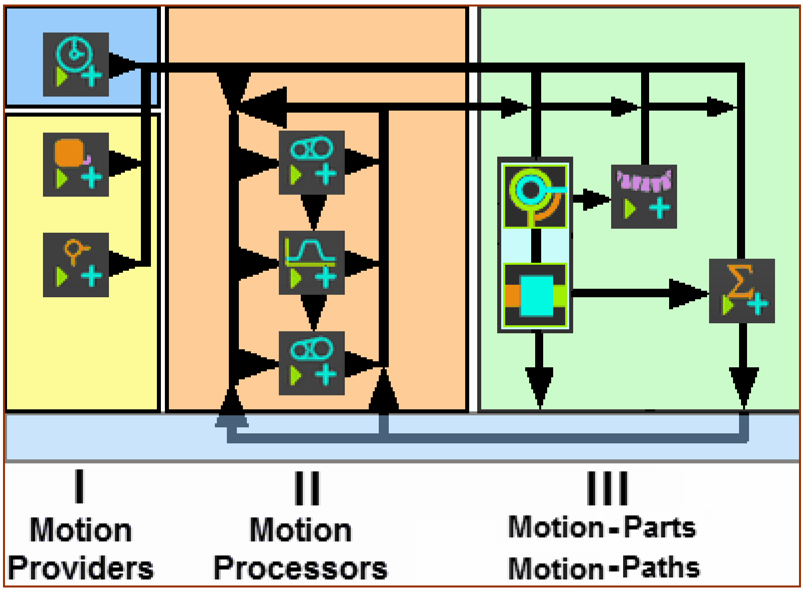
Kinematic FB toolbar

1.4.4.1 Function-Blocks | Connect, Parameter-Values, & Channels

Function-Blocks: Wires and Connectors

FUNCTION-BLOCKS have connectors: **input-connectors** and/or **output-connectors**. You connect **wires** from an **output-connector** of a FUNCTION-BLOCK to an **input-connector** of a different FUNCTION-BLOCK.

The motion-data that flows through the wires and KINEMATIC FUNCTION-BLOCKS has three Data-Channels, The Data-Channels are Position, Velocity and Acceleration, with Linear or Angular motion-units.



Schematic: Function-Block Wires and Connectors

The image above is a schematic of how you can add wires to connect FUNCTION-BLOCKS.

VIDEO : How to Connect Function-Blocks: Double-click to watch Video Clip

---

Data-Types of Motion Function-Blocks.

These FUNCTION-BLOCKS provide or process motion-data.

LINEAR-MOTION FB

Output DATA-TYPE = ROTARY

GEARING FB

Output DATA-TYPE = Input DATA-TYPE

MOTION FB

Output DATA-TYPE = ROTARY OR LINEAR

See **Motion FB dialog** 374

MOTION-DIMENSION FB

Output DATA-TYPE = Input DATA-TYPE

MOTION-PATH FB

No output-connector

Input DATA-TYPE = ROTARY OR LINEAR

MEASUREMENT FB

Output DATA-TYPE = ROTARY OR LINEAR

POINT-DATA FB

Output DATA-TYPE = ROTARY OR LINEAR

Function-Blocks - their Data-Types and Data-Channels

Output-Connector from	Data Type	Data-Channel 1	Data-Channel 2	Data-Channel 3
LINEAR MOTION FB	Rotary	Machine Angle	Machine Angular Velocity see Machine-Settings > Cycling Parameters [291]	Machine Angular Acceleration
GEARING FB	Linear	Position - mm	Linear Velocity - mm/s	Linear Acceleration - mm/s/s
	Rotary	Angle - deg	Angular Velocity deg/s	Angular Acceleration deg/s/s
MOTION FB	Linear	Position - mm	Linear Velocity - mm/s	Linear Acceleration - mm/s/s
	Rotary	Angle - deg	Angular Velocity - rad/s	Angular Acceleration - deg/s/s
MOTION-DIMENSION FB	Linear	Position (mm)	Linear Velocity - mm/s	Linear Acceleration - mm/s/s
	Rotary	Angle -deg	Angular Velocity - rad/s	Angular Acceleration - rad/s/s
MEASUREMENT FB	Linear	Position - mm	Linear Velocity - mm/s	Linear Acceleration - mm/s/s
	Rotary	Angle -deg	Angular Velocity - rad/s	Angular Acceleration - rad/s/s
POINT FB	Linear Linear Linear	X Position mm Y Position - mm Magnitude - mm	X Velocity - mm/s Y Velocity - mm/s Mag. Velocity- mm/s	X Acceleration - mm/s/s Y Acceleration - mm/s/s

Output-Connector from	Data Type	Data-Channel 1	Data-Channel 2	Data-Channel 3
				Mag. Acceleration - mm/s/s
CAM-DATA FB	Rotary	Pressure Angle - deg	Contact Pressure Angel 1 - deg	Contact Pressure Angle 2 - deg
	Linear	Inner Cam Radius-of-Curvature	Outer Cam Radius-of-Curvature	------
	Force	Contact Force Total - N	Contact Force, X component - N	Contact Force, Y component
	Stress/Pressure	Inner Cam Shear Stress- N/mm²	Outer Cam Shear Stress - N/mm²	------
	Linear	Sliding-Velocity (mm/s)	------	------
FORCE-DATA FB	Force	Joint Total Force / Torque	X Component - N	Y Component - N
	Force	Cam Contact Force - N	X Component - N	Y Component - N
	Force	Spring Force - N	X Component - N	Y Component - N
MATH FB	See Math FB dialog 397			

1.4.4.2 Add Linear-Motion FB

Linear-Motion FB

See also:

Linear Motion dialog [371]

How to Connect FBs [144]

Terminology and Definitions

MASTER MACHINE ANGLE :	**MMA** - when you cycle your model, the **MMA** increases at a constant rate from 0 to 360, again and again.
LINEAR MOTION FB :	The default output from a **LINEAR-MOTION FB** is equal to the **MMA**. You can edit the **LINEAR-MOTION FB** to add or subtract a value from the default value of the **MMA**. The **LINEAR-MOTION FB** is usually the first in a series of **FBS** that you connect together with wires.

Linear-Motion FB function

The function of the **LINEAR-MOTION FB** is:

Output = MMA ± 1

MMA = Master-Machine-Angle

Use the LINEAR-MOTION DIALOG [371] to edit 1 .

Add Linear-Motion FB

Add :	STEP 1: Add a **LINEAR-MOTION FB** to the graphic-area  1. Click Kinematics FB toolbar [142] > Linear-Motion FB OR 1. Click Add menu > Add Linear-Motion FB 2. Click the graphic-area The **LINEAR-MOTION FB** is now in the graphic-area (and **ASSEMBLY-TREE**).
Edit :	STEP 2: Open the **LINEAR-MOTION DIALOG**: 1. Double-click the **LINEAR-MOTION FB** in the graphic-area. OR 1. **See How to Open a dialog** [513] The **LINEAR-MOTION DIALOG** is now open.
Dialog :	STEP 3: **See Linear-Motion dialog** [371]

1.4.4.3 Add Gearing FB

Gearing FB

See also Gearing FB dialog [372]

Terminology

GEARING FB :	The GEARING FB modifies motion-data with a linear equation/function. The function has 3 parameters - see below

Gearing FB function

The function of the **GEARING FB** is:

Output = (1 * (Input + 2)) + 3

Use the GEARING FB DIALOG [372] to edit the three parameters 1 2 3

Add Gearing FB

Add :	**STEP 1: Add a Gearing FB to the graphic-area**  1. Click Kinematics FB toolbar [142] > **Gearing FB** OR 1. Click **Add menu** > **Add Gearing FB** 2. Click the graphic-area The **GEARING FB** is now in the graphic-area (and **ASSEMBLY-TREE**).
Edit :	**STEP 2: Open the Gearing FB dialog** 1. Double-click a **GEARING FB** in the graphic-area or **ASSEMBLY-TREE** OR 1. See How to Open a dialog [513]
Dialog :	**STEP 3: See Gearing FB dialog** [372]

Notes:

- The default parameters do not change the motion-data.
- Connect a **GEARING FB** to the input of a **MOTION FB** to change the frequency of a motion.
- Connect a **MOTION FB** to the input of a **GEARING FB** to change the amplitude and/or direction of a motion.

1.4.4.4 Add Motion FB

Motion FB

See also :

Motion FB dialog [374]

Terminology and Definitions

MOTION FB :	The MOTION FB provides a link to a Motion in **MotionDesigner**. The input to and the output from the Motion FB correspond directly to the X-axis and Y-axis values, respectively, of the Motion you select.

Motion FB function

Edit the MOTION FB to select with which **Motion** in **MotionDesigner** you want to make a link.

Input = X-axis value of selected Motion

Output = Y-axis value corresponding to X-axis value of selected Motion

Add Motion FB

Add :	STEP 1: **Add a Motion FB to the graphic-area.**  1. Click Kinematics FB toolbar [142] > **Motion FB** or 1. Click **Add menu** > **Add Motion FB** 2. Click the graphic-area The MOTION FB is now in the graphic-area and ASSEMBLY-TREE.
Edit :	STEP 2: **Open the Motion FB dialog** 1. Double-click the MOTION FB in the graphic-area or ASSEMBLY-TREE OR 1. **See How to Open a dialog** [513] The MOTION FB DIALOG is now in open.
Dialog :	STEP 3: See Motion FB dialog [374]

1.4.4.5 Add Motion-Dimension FB

Motion-Dimension FB

See also Motion-Dimension dialog [377]

Terminology and Definitions

Motion-Part :	The **derived name** for a PART whose **Linear or Angular motion**, relative to the motion of a different PART, is specified at the output of a MOTION-DIMENSION FB.
ROCKER :	A **Motion-Part** that rotates.
SLIDER :	A **Motion-Part** that slides.
MOTION-DIMENSION FB :	A MOTION-DIMENSION FB removes one degree-of-freedom from a PART - the PART becomes a **Motion-Part** and **kinematically-defined**.

Motion-Dimension FB function

The output from a MOTION-DIMENSION FB specifies the **Linear or Angular motion** of a PART, (with a derived-name of **Motion-Part**).

- **Angular motion** of a **Motion-Part = Input motion-value + Base-Value (offset angle) - this is a Rocker**

OR

- **Linear motion** of a **Motion-Part = Input motion-value + Base-Value (offset position) - this is a Slider**

Use the MOTION-DIMENSION DIALOG [377] to edit ❶

Add Motion-Dimension FB - ROCKER

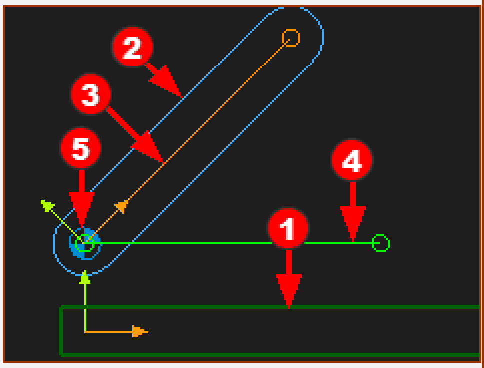
Prepare the model with these elements.

Note the kinematic state of each element before you add a MOTION-DIMENSION FB.

ROCKER

Element #	Element Type	Rules
#1	PIN-JOINT	• Is between two(2) POINTS • Each POINT is the START-POINT or END-POINT of a LINE (or CAD-LINES) • Each POINT is a child to a different PART • One PART IS **kinematically-defined**, the other PART IS NOT **kinematically-defined**
#2	LINE	• This LINE radiates from the PIN-JOINT (see ELEMENT #1) • AND this LINE is in the PART that IS **kinematically-defined**
#3	LINE	• This LINE also radiates from the PIN-JOINT (see ELEMENT #1) • AND this LINE is in the PART that IS NOT **kinematically-defined**

Preparation: Motion-Dimension - Rocker (typical)

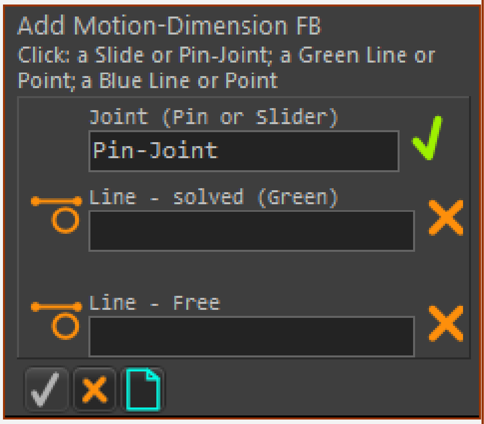


Preparation example:

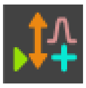
- Two(2) PARTS. In this case the BASE-PART❶ and a ROTATING-PART❷
- LINES❸❹ in each PART
- PIN-JOINT❺ joins the START-POINT and/or END-POINTS of each LINE❸❹
- PART❶ is kinematically-defined
- PART❷ is not kinematically-defined

See Add Line 228, Add Part 91, Add Pin-Joint 85

Add Motion-Dimension FB (Rocker)



STEP 1: Select the Add Motion-Dimension FB command

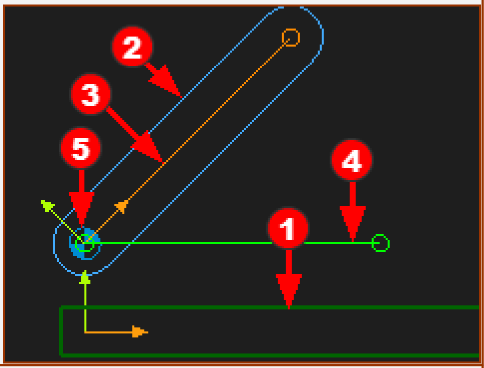


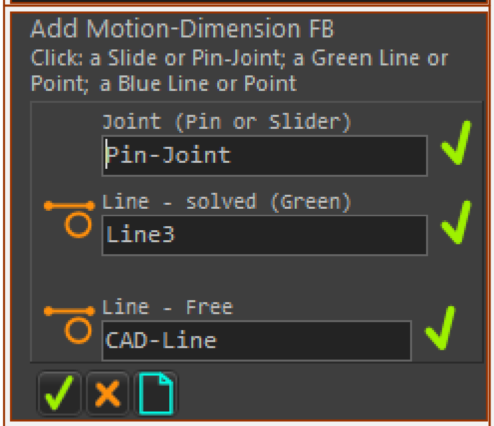
1. Click Kinematics FB toolbar 142 > Motion-Dimension FB

or

1. Click Add menu > Add Motion-Dimension FB

The COMMAND-MANAGER indicates that there are three elements to select:





STEP 2: Select the three elements in the graphic-area

ELEMENT #1: PIN-JOINT

1. Click a PIN-JOINT❺

The PIN-JOINT must be a joint between two(2) PARTS, and:

- One PART must be **kinematically-defined**
- The other PART must be a **free** and joined with a PIN-JOINT to the PART that is **kinematically-defined**

ELEMENTS #2 and ELEMENT #3: LINES ❸❹

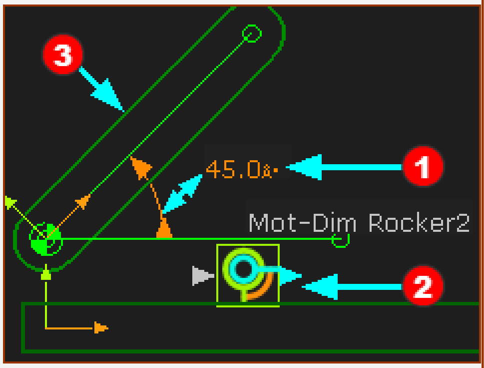
Each LINE must be a child to a POINT and also a child to the PIN-JOINT. (ELEMENT #1)

2. Click LINE❹ in the PART that is **kinematically-defined**❺
3. Click LINE❸ in the **free** PART❷

STEP 3: Complete the Command

1. **Click** ✓ in the COMMAND-MANAGER to complete Add Motion-Dimension FB (Rocker).

RESULT:



The **Mobility** of the kinematic-chain is now Zero(0).

PART❸ is now a PART that is **kinematically-defined**.

A MOTION-DIMENSION❶ and MOTION-DIMENSION FB ❷ specifies the angle between the LINES

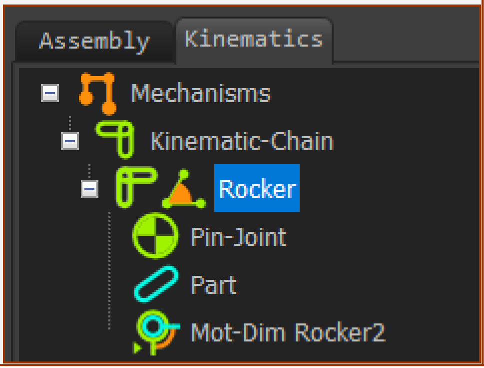
The MOTION-DIMENSION FB icon❷ in the graphic-area for a **ROCKER**.

Note: In the graphic-area:

- To move the **Motion-Dimension❶** : **CTRL+ DRAG the FB icon❷**
- To move the **MOTION-DIMENSION FB❷** : **DRAG the FB icon❷**

To edit: STEP 3: Open the dialog - **See [Motion-Dimension dialog](377)**

Kinematics-Tree - Motion-Dimension (Rocker)



When you add a MOTION-DIMENSION FB to a PIN-JOINT, you add a **Rocker** to the KINEMATICS-TREE.

The kinematic elements are:

Part + Pin-Joint + Motion-Dimension

=

ROCKER

Add Motion-Dimension FB and SLIDER

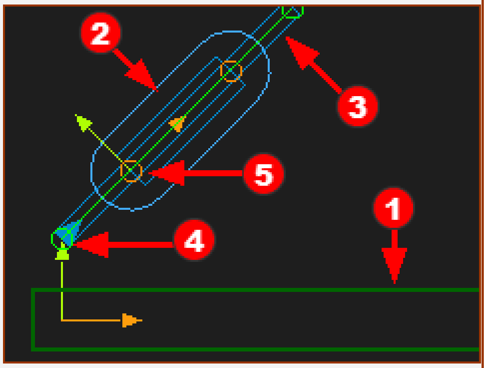
Prepare the model with these elements.

Note the kinematic state of each element **before** you add the MOTION-DIMENSION FB.

SLIDER

Element #	Element type	Rules
#1	SLIDE-JOINT	• Is between two(2) LINES* • Each LINE* is a child to a different PART • One PART IS **kinematically-defined**, the other PART IS NOT **kinematically-defined** * LINE and/or CAD-LINE
#2	POINT	• the START-POINT or the END-POINT of one of the LINES in the SLIDE-JOINT (ELEMENT #1), • AND the POINT (and LINE) is in the PART that is **kinematically-defined**
#3	POINT	• the START-POINT or the END-POINT of one of the LINES in the SLIDE-JOINT (ELEMENT #1) • AND the POINT (and LINE) is in the PART that is **free**

Preparation: Add Motion-Dimension - Slider (typical)



Preparation example:

1. Two(2) PARTS❶❷

One PART is **kinematically-defined** …the BASE-PART❶

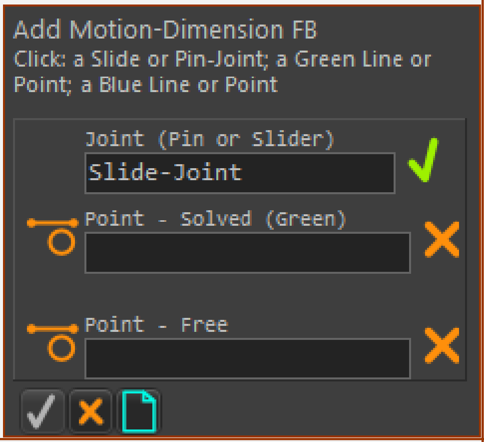
The other PART that is **free** ❷

2. The PARTS are Joined… with a SLIDE-JOINT❸
3. START POINT❹ (or END-POINT) of a LINE in the PART that is **kinematically-defined**
4. START-POINT❺ (or END-POINT) of a LINE in the PART that is **free**

The LINES are those that you select for the SLIDE-JOINT❸.

See Add Line [228], Add Part [91], Add Pin-Joint [85]

Add Motion-Dimension FB (Slider)

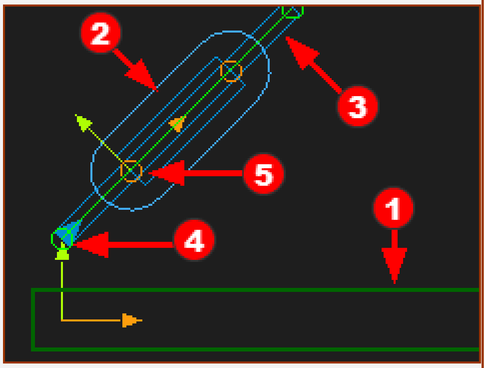


STEP 1: Add the MOTION-DIMENSION FB

1. Click Kinematics FB toolbar [142] > Motion-Dimension FB

or

1. Click Add menu > Add Motion-Dimension FB



STEP 2: Select three elements in the graphic-area

ELEMENT #1: PIN-JOINT

1. Click a SLIDE-JOINT❸

The SLIDE-JOINT must be a joint between two(2) PARTS, and:

- One PART is **kinematically-defined** ❶
- The other PART is **free** ❷

ELEMENTS #2 and ELEMENT #3: POINTS ❹❺

Each POINT must be a child to a LINE that is also a child to the SLIDE-JOINT. (ELEMENT #1)

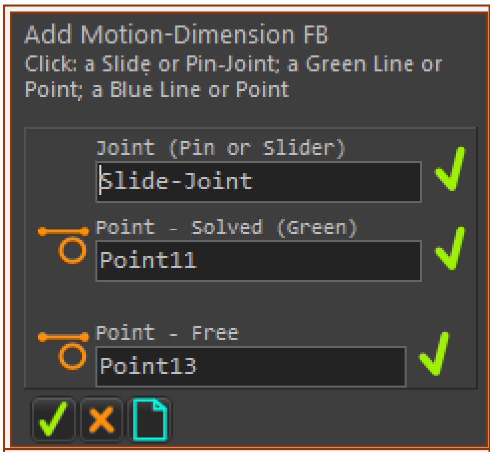
2. Click the POINT❹ in the PART that is **kinematically-defined**❶
3. Click the POINT❺ in the PART that is **free**❷

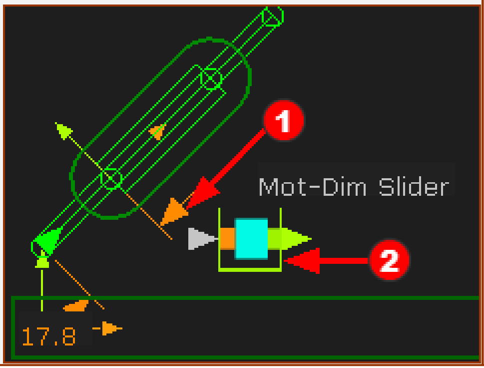
STEP 3: Complete the Command

1. **Click ✓ in the** COMMAND-MANAGER to complete Add Motion-Dimension FB (Slider).

RESULT :

The free PART is now **kinematically-defined**.





A MOTION-DIMENSION❶ and MOTION-DIMENSION FB ❷ specifies the angle between the LINES

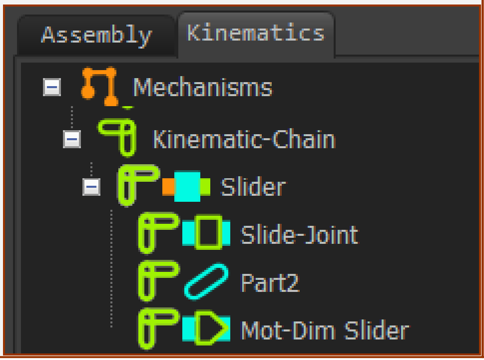
The MOTION-DIMENSION FB icon❷ in the graphic-area for a SLIDER.

Note: To drag in the graphic-area the:

- Motion-Dimension : CTRL + DRAG the FB icon
- MOTION-Dimension FB : DRAG the FB icon

STEP 4: Open the dialog - See Motion-Dimension dialog 377

Kinematics-Tree - Motion-Dimension (Slider)



When you add a MOTION-DIMENSION FB to a SLIDE-JOINT, you add a **Slider** to the KINEMATICS-TREE.

The kinematic elements are:

Part + Slide-Joint + Motion-Dimension =

SLIDER

Videos: Add Part, Add Joint, Add Motion-Dimension FB:

Add Motion-Dimension FB to a Pin-Joint Double-click to watch 'Add and Edit Rocker Motion-Dimension FB

Add Motion-Dimension FB to a Slide-Joint Double-click to watch 'Add and Edit Slider Motion-Dimension'

Special Methods

Frequently, in a machine, PIN-JOINTS can be coincident (co-axial) and SLIDE-JOINTS can be collinear.

For example, there are two or more **rotating-Parts** that rotate about one center, or two or more **sliding-Parts** that slide along one axis.

Special Method: Coincident Pin-Joints

If you want to specify the motion of each **rotating-part** with a different MOTION-DIMENSION FB, you must carefully plan as to which POINTS you select when you add the PIN-JOINTS.

Then, when you add the MOTION-DIMENSION FBS, the MOTION-DIMENSIONS can reference the correct PARTS.

Special Method: Coincident Slide-Joints

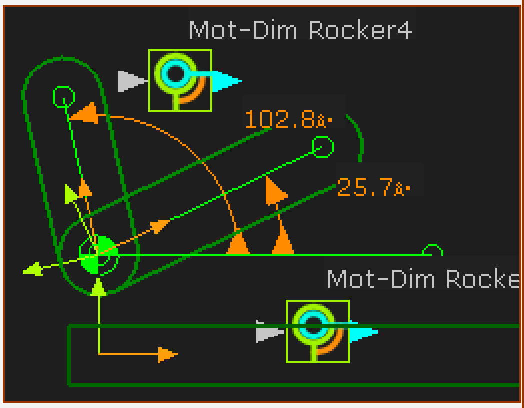
If you want to specify the motion of each **sliding-part** with a different MOTION-DIMENSION FB, you must carefully plan as to which LINES you select **when you add the** SLIDE-JOINTS.

Then, when you add the MOTION-DIMENSION FBS, the MOTION-DIMENSIONS can reference the correct PARTS.

The LINES you select when you add the SLIDE-JOINT determine between which two PARTS you can specify the MOTION-DIMENSION FB.

Special Method 1

The methods and principles described below apply to Rockers and Sliders. However, the descriptions below are given for Rockers only.



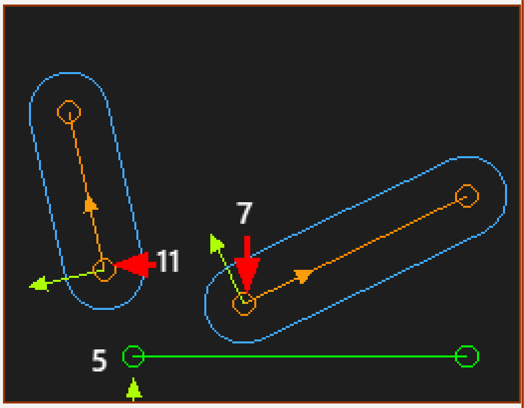
Result of Special Method 1:

The model shows

- **Two** MOTION-DIMENSION FBS
- They reference the same LINE in the BASE-PART.

You can see the dimension of the two MOTION-DIMENSIONS FBS start at the green horizontal LINE.

STEP 1: Add Pin-Joints two times



There are two added PARTS

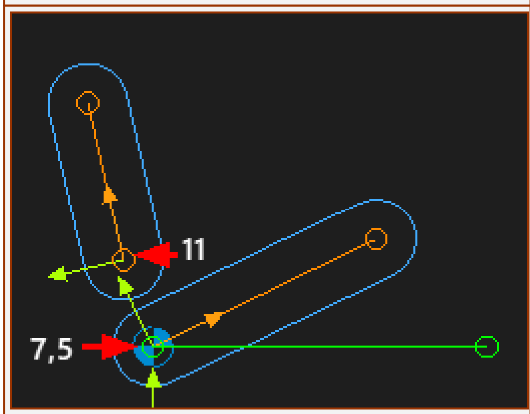
There is an added PART. As always, it has a CAD-LINE, with a START-POINT and an END-POINT.

There is also LINE. This LINE is in the BASE-PART, which you cannot see.

There are **three(3)** LINES in **three(3)** PARTS in total.

The START-POINTS of the LINES are **5**, **7** and **11.**

Note: You can select the START-POINTS and/or the END-POINTS. However, when possible, use the START-POINTS.



ADD PIN-JOINT #1

In the image to the left, PIN-JOINT **#1** is between POINT **7** & **5**

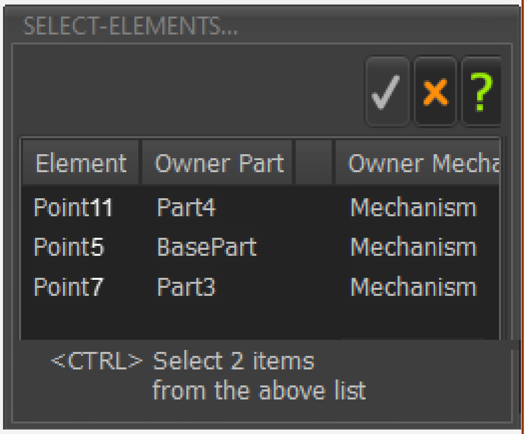
Now, add a PIN-JOINT. **#2**

ADD PIN-JOINT #2

Add PIN-JOINT **#2** between POINTS **11** & **5.**

Q: Why POINTS **11** & **5** and not **11** & **7** ?

A: You want the **MOTION-DIMENSIONS** to reference the horizontal **LINE** in the **BASE-PART.**



Add Pin-Joint :

1. Click **Point 11** and then ***try*** to click **Point 5.**

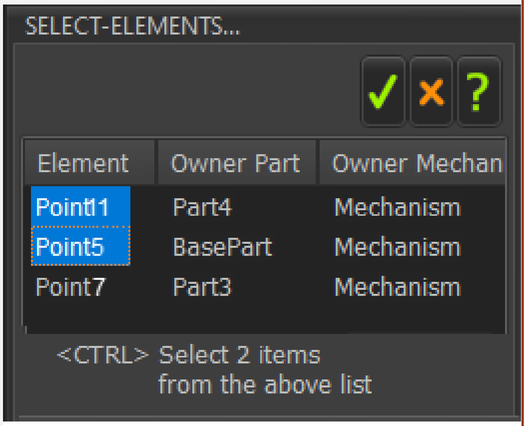
The **SELECT-ELEMENTS DIALOG** opens! Why? Because there is ambiguity.

POINTS 5 & **7** are at the same position. You have actually clicked three **POINTS** (**11, 7 & 5**).

You must select the two **POINTS** for the **PIN-JOINT**.

About the **SELECT ELEMENTS DIALOG**:

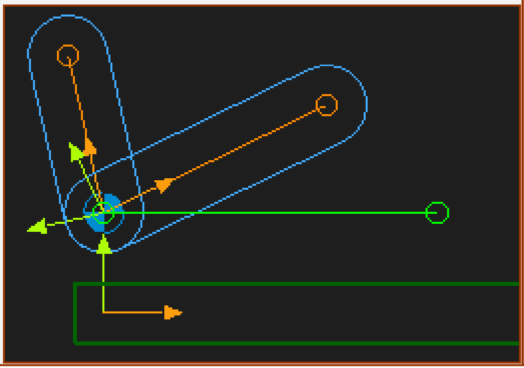
- Is a list of **ALL** of the elements that you have clicked
- The element that you selected first is always at the top of the list.
- None of the elements are selected.



2. **CTRL + CLICK ALL** of the elements you need to complete the command.

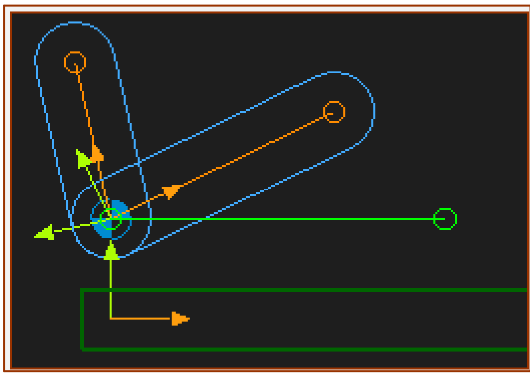
In this image, select **POINT 11** and **POINT 5**

1. **CTRL + CLICK Point11 AND Point5**
2. **Click ✓**



Now, the two **PARTS** are joined with a **PIN-JOINT** to one **POINT** in the **BASE-PART**.

STEP 2: Add Motion-Dimension FB two times

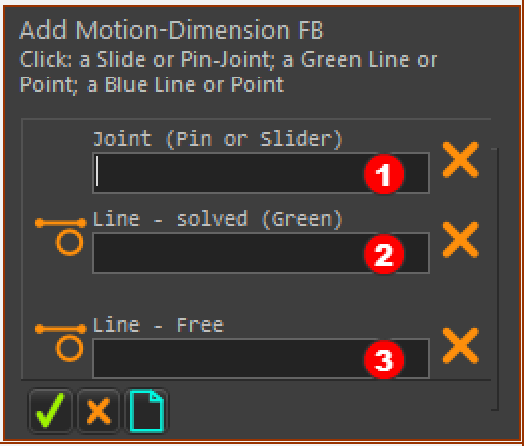


The two **PARTS** are now joined to the **BASE-PART** with two **PIN-JOINTS**

Now, use the **MOTION-DIMENSION FB** two times

ADD MOTION-DIMENSION 1

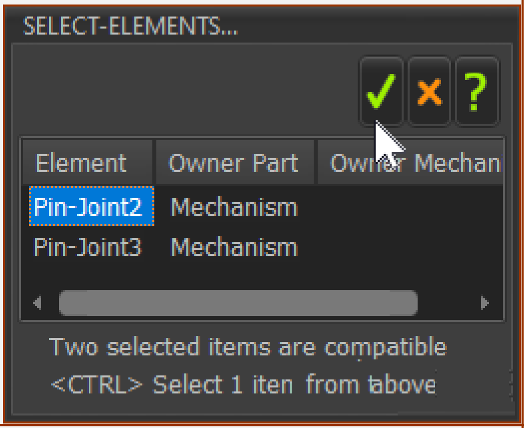
1. Click **Function-Block toolbar > Add Motion-Dimension FB**.



The **COMMAND-MANAGER** show that you must select three elements ❶, ❷ and ❸.

The first-element must be a **PIN-JOINT** or **SLIDE-JOINT**.

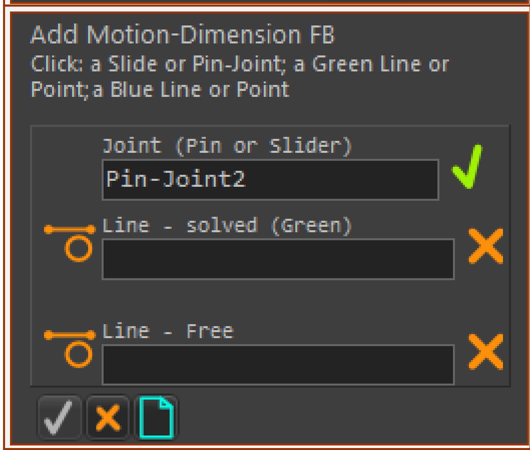
There are two **PIN-JOINTS**!

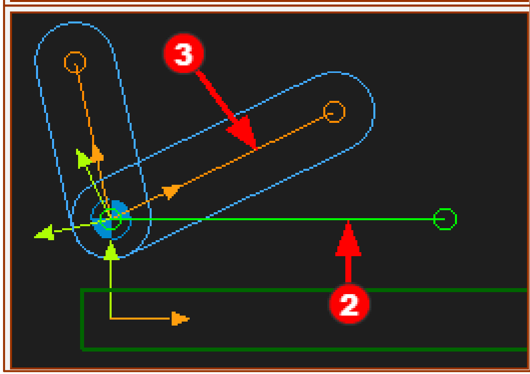


When you click the **PIN-JOINTS** in the graphic-area the **SELECT-ELEMENTS DIALOG** opens as there are two **PIN-JOINTS**.

In the **SELECT-ELEMENTS DIALOG**:

Click the **PIN-JOINT**.

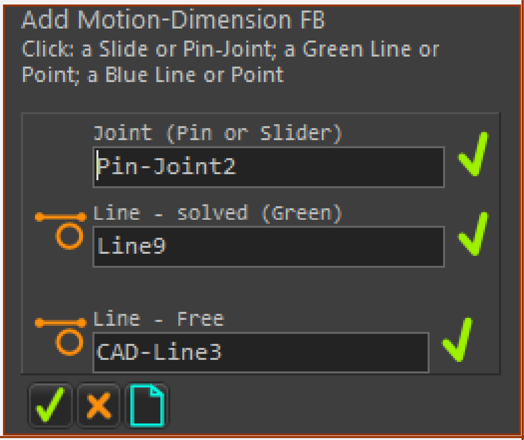




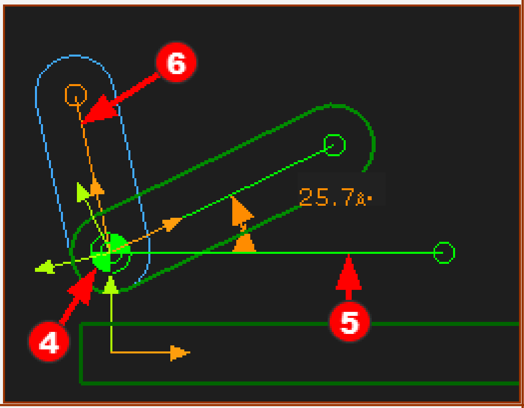
Now, select two **LINES**.

Click the horizontal **LINE** ❷ as the second element.

Click the **CAD-LINE** ❸ along one the **PARTS** with which the **PIN-JOINT** is made.



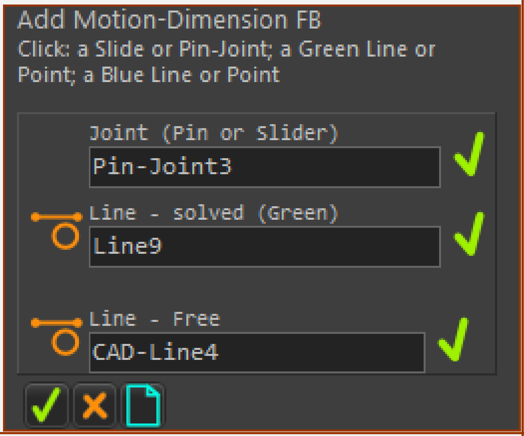
This image shows the three elements in the Command-Manager boxes.



The first Rocker **MOTION-DIMENSION FB**.

ADD MOTION-DIMENSION #2

You must now select another three elements,❹❺❻



Function-Block Toolbar > Motion-Dimension FB again.

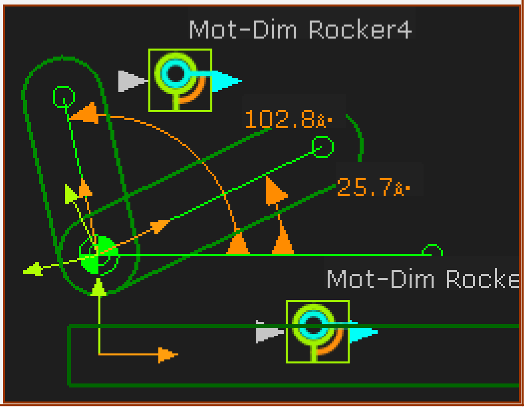
These are the elements.

❹ **JOINT: PIN-JOINT2**.

Now, there is no ambiguity. There is only one **PIN-JOINT** available.

❺ **LINE** – Solved: select the horizontal **LINE**.

❻ **LINE** – Free: select the **CAD-LINE** along the center of the Blue **PART**.



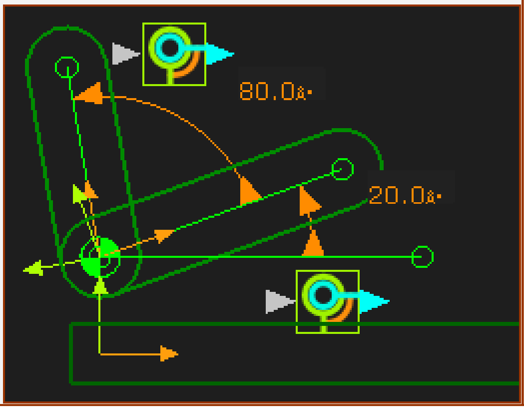
Here, you can see the result.

Both **MOTION-DIMENSIONS FBS** use the Horizontal **LINE** in the **BASE-PART** as a reference for the motion-dimension.

Note: When you add two **MOTION-DIMENSIONS**, the **FBS** may be on top of each other.

Drag each **FB** so you can see them.

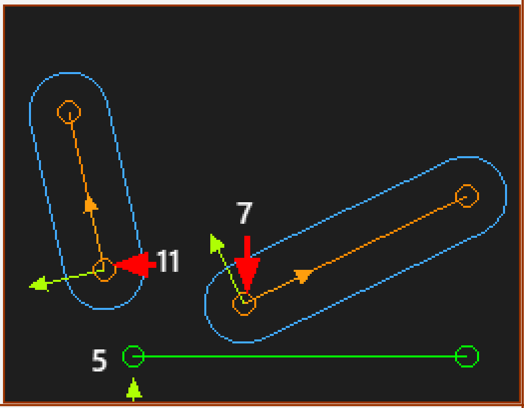
Special Method 2:



Result of Special Method 2

- **MOTION-DIMENSION 1**:
  From the **LINE** in the **BASE-PART**
  To a **LINE** in **'PART 1'**
- **MOTION-DIMENSION 2**:
  From the **LINE** in **'PART 1'**
  To the **LINE** in **'PART 2'**

STEP 1: Add Pin-Joint two times

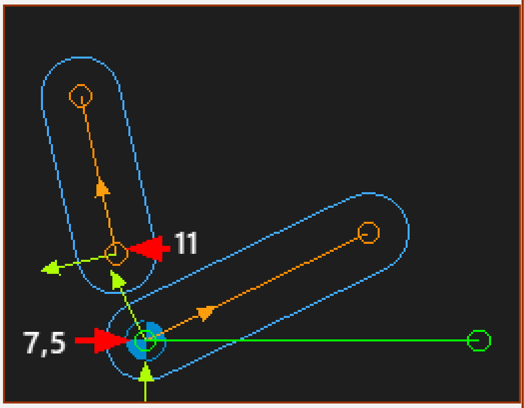


Again, there are two **PARTS** and a **LINE** in the **BASE-PART**.

The **POINTS** at the ends of the **LINES** are 5, 7 and 11.

You want to:

- Add a **PIN-JOINT** between '**POINT7**' and '**POINT5**', and
- Add a **PIN-JOINT** between '**POINT11**' and '**POINT7**'.

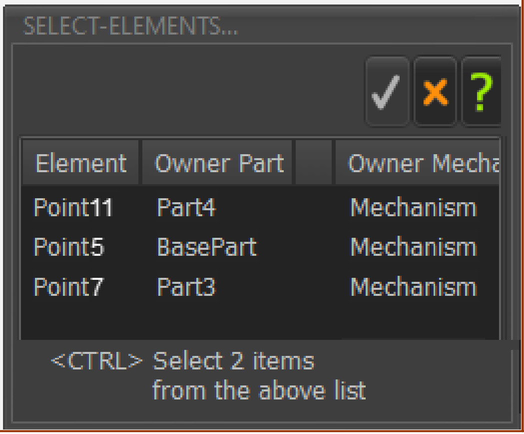


In the image to the left, **POINT7** is joined to **POINT5**

CONDITION 2

Add the second **PIN-JOINT** between **POINT11** to **POINT7**.

- **Q:** Why **POINT7** and **not POINT5** ?
- **A:** Because **POINT7** is a child to **CAD-LINE❶**, you want this **MOTION-DIMENSION** to reference **CAD-LINE❶**.
- We do not want the **MOTION-DIMENSION** to reference the **LINE** in the **BASE-PART**. That was **Method 1**, above.

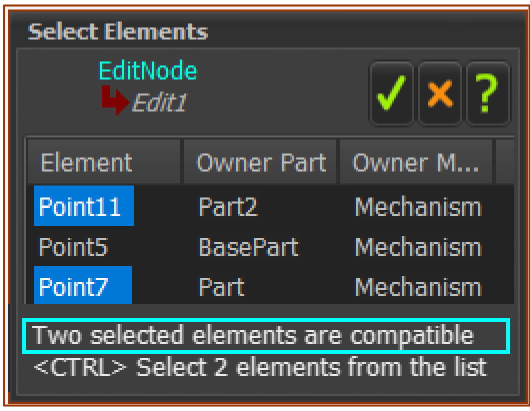


Select **POINT11**.

Then try to select **POINT7**

There is ambiguity between **POINT5** and **POINT7**. Hence, the **SELECT ELEMENTS DIALOG** opens.

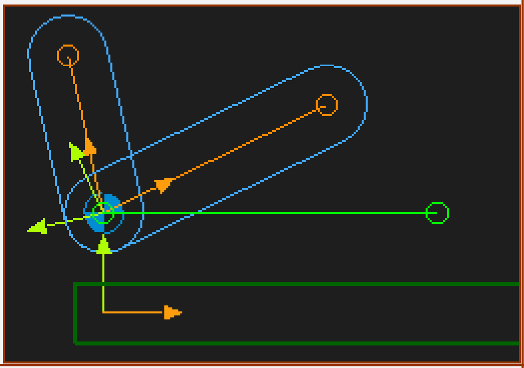
- ALL the elements you need for the command are made available to you
- The first element you select is at the top of the list.



In the image to the left, we have selected for you POINT11 and POINT7.

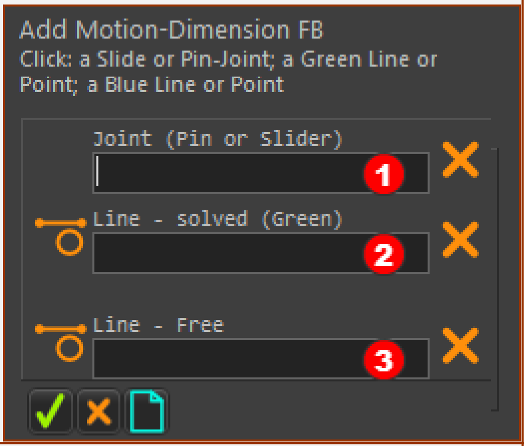
1. CTRL + CLICK Point11 AND Point7
2. Click ✓

STEP 2: Add Motion-Dimension FB two times

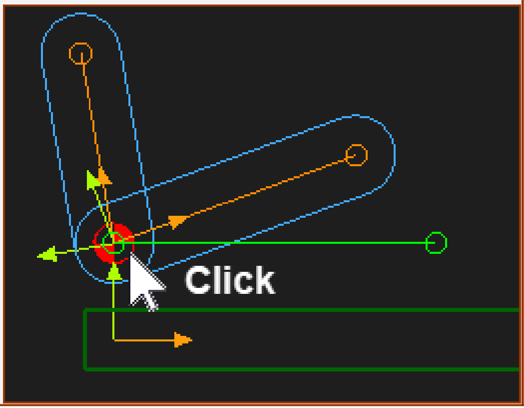


The two PARTS become joined with a PIN-JOINT.

Do Add MOTION-DIMENSION FBS **two times.**



You must select three elements to do each ADD MOTION-DIMENSION FB

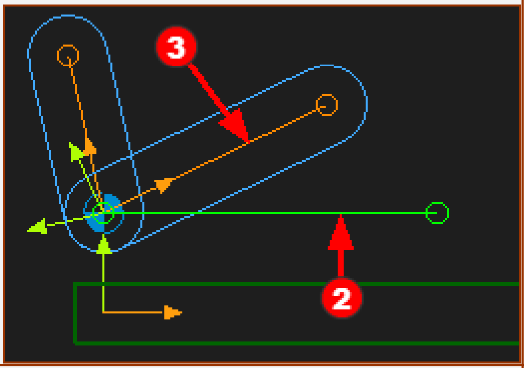


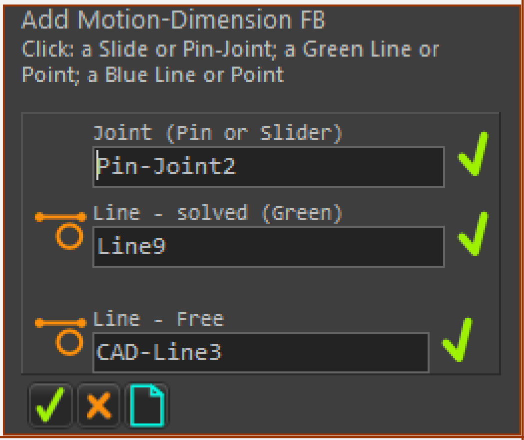
ELEMENT 1: A Joint.

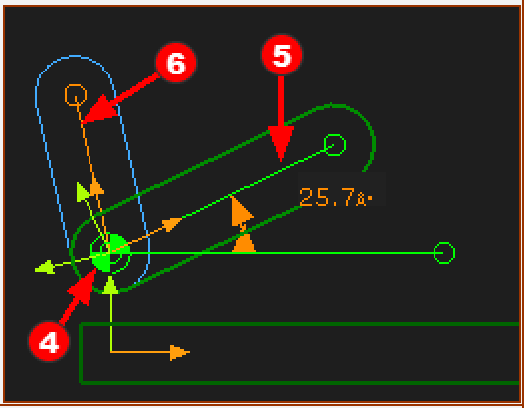
There is NO ambiguity when you select the PIN-JOINT.

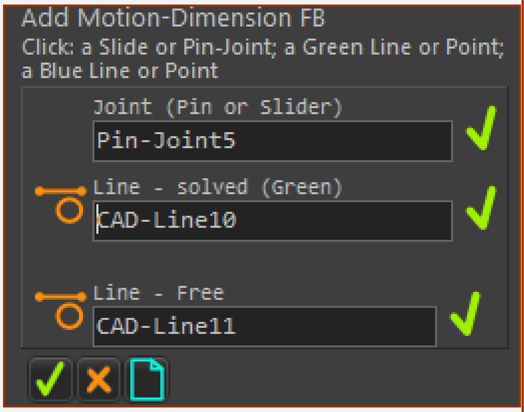
There is only ONE PIN-JOINT between a POINT in the PART that is **kinematically-defined** and the other POINT in the PART that is **free**.

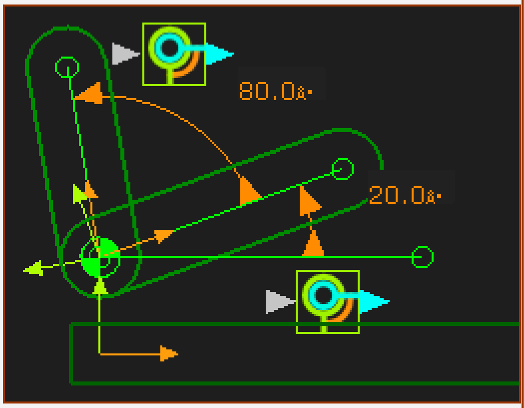
The SELECT ELEMENTS DIALOG does NOT open.

<table>
<tr>
<td>
</td>
<td>ELEMENTS 2 & 3: Two LINES – a Solved LINE❷ then a Free LINE❸

Select the Horizontal LINE❷ and the CAD-LINE❸</td>
</tr>
<tr>
<td>
</td>
<td>The three elements in the COMMAND-MANAGER

Click ✅ in the COMMAND-MANAGER</td>
</tr>
<tr>
<td>
</td>
<td>You can see the First MOTION-DIMENSION FB.
<hr>Add the second Motion-Dimension.

You must select three elements,❹❺❻</td>
</tr>
<tr>
<td>
</td>
<td>These are the element-types
1. PIN-JOINT5, there is no ambiguity because the existing MOTION-DIMENSION FB uses PIN-JOINT2.
2. CAD-LINE10: you must select this CAD-LINE because it is a child to a PART that IS kinematically-defined and its START-POINT is referenced by PIN-JOINT6
3. CAD-LINE11: you must select this CAD-LINE because it is a child to a PART that IS NOT kinematically-defined and its START-POINT is referenced by PIN-JOINT6</td>
</tr>
</table>

	This image shows the end result. MOTION-DIMENSION FB **#1:** uses the Horizontal LINE that is a child to the BASE-PART as a **reference Line**. MOTION-DIMENSION FB **#2:** uses the CAD-LINE along the center of the ***added*** PART as the **reference Line**. The CAD-LINE is **kinematically-defined** after you add MOTION-DIMENSION FB #1 - see Method 1 Note: When you add two MOTION-DIMENSIONS, the FBs may be on top of each other. **DRAG** each FB so you can see each of them.

Positive' Direction of Slide-Joints and Slider

The nursery rhyme : **She swallowed a cat, that swallowed a mouse, that swallowed a spider, that swallowed a fly**... comes to mind!

A Slider

To add a MOTION-DIMENSION FB (for a Slider) you must select a SLIDE-JOINT, and then one POINT from each of the two LINES that define the SLIDE-JOINT.

- The properties of the MOTION-DIMENSION FB are determined by the properties of the SLIDE-JOINT.
- The properties of the SLIDE-JOINT are determined by the properties of the LINES you select to add the SLIDE-JOINT.
- The properties of the LINES are determined by the properties their START-POINTS and END-POINTS.

Therefore, the **positive-direction** of the MOTION-DIMENSION for a SLIDER is defined by the START-POINT and END-POINT of the two LINES in the SLIDE-JOINT.

If the positive-direction of the MOTION-DIMENSION is not as you want, you need to add the LINES and SLIDE-JOINT again.

Top-Tip - plan. Drag LINES and PARTS from the START-POINT to the END-POINT in the **same positive direction** of the MOTION-DIMENSION.

Always, the **Positive Direction** of a MOTION-DIMENSION for a SLIDER :

'...is the **Positive Direction** of the LINE, or CAD-LINE, that is a child to the PART that is **kinematically-defined** before you add the MOTION-DIMENSION FB to the SLIDE-JOINT.

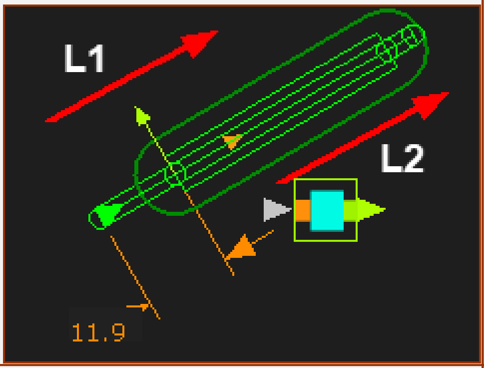
What is the Positive Direction of a Line, CAD-Line, or Part?

The direction in which you drag - from its START-POINT to its END-POINT - when you add the LINE, CAD-LINE, or PART.

EXAMPLES: positive direction of SLIDERS

- LINE 1 is a child to a PART that IS **kinematically-defined** before you add the MOTION-DIMENSION FB
- LINE 2 is a child to a PART that IS **kinematically-defined** after you add the MOTION-DIMENSION FB

I have used the ***red arrows*** to show the direction in which I dragged my mouse-point to add the LINES and PARTS.

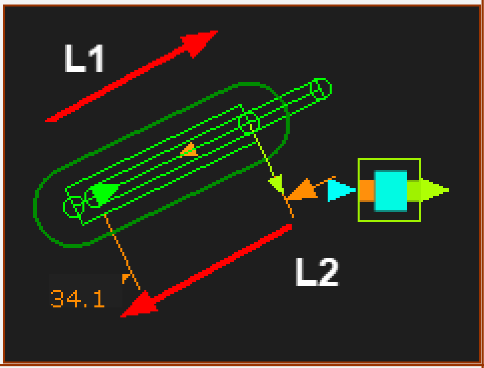


- LINE 1 – **L1** in the BASE-PART- Drag up to the right
- CAD-LINE/PART 2 – **L2** - Drag up to the right

The MOTION-DIMENSION is 11.9mm.

If you Increase the MOTION-DIMENSION value, you move the SLIDER up to the right.

The small **arrowhead** shows the positive direction of the SLIDER.

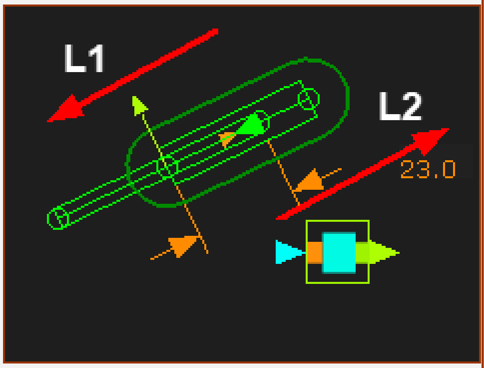


- LINE 1 – **L1** in the BASE-PART - Drag up to the right
- CAD-LINE/PART 2 – **L2** - Drag down from the left

The MOTION-DIMENSION is 34.1mm.

If you increase the MOTION-DIMENSION value, you move the SLIDER up to the right.

The small **arrowhead** shows the positive direction of the SLIDER.

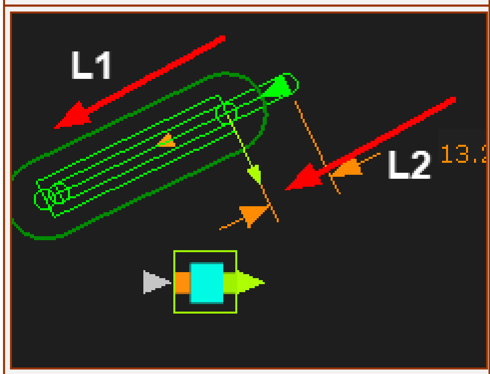


- LINE 1 – **L1** in the BASE-PART - Drag down from the right
- CAD-LINE/PART 2 – **L2** - Drag up from the left

The MOTION-DIMENSION is 23mm.

If you increase the MOTION-DIMENSION value, you move the SLIDER down to the left.

The small **arrowhead** shows the positive direction of the SLIDER.



- LINE 1 – **L1** in the BASE-PART - Drag down Right to Left
- CAD-LINE/PART 2 – L2 - Drag from Right to Left

The MOTION-DIMENSION is 13.2mm.

If you increase the MOTION-DIMENSION value, you move the SLIDER down to the left.

The small **arrowhead** shows the positive direction of the SLIDER.

1.4.4.6 Add Motion-Path FB

Motion-Path FB

See also Motion-Path dialog [380]

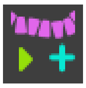
Terminology and Definitions

MOTION-PATH FB :	A **FB** that adds a **MOTION-POINT** to a **sketch-path** - see **sketch-loop**
MOTION-POINT :	The input to the **FB** specifies the motion of the **MOTION-POINT** along the **sketch-path**.

Example applications of a Motion-Path FB:

To define the motion of a robot along a sketch-path To define the motion of a chain or belt along a sketch-path To model an extending-cylinder to raise a Scissor-Lift

Add Motion-Path FB

Add :	**STEP 1: Start the Add Motion-Path FB command**  1. Click Kinematics FB toolbar [142] > **Motion-Path FB** **or** 1. Click **Add menu** > **Add Motion-Path FB** The **COMMAND-MANAGER** has one selection-box. 2. Click a sketch-element* from a sketch-path** 3. Click ✅ in the **COMMAND-MANAGER** The **MOTION-PATH FB** is now in the graphic-area (and **ASSEMBLY-TREE**). AND A **Motion-Point** is at the **START-POINT** of the sketch-element* you click on the sketch-path**.
Open :	**STEP 2: Open the Motion-Path dialog** 1. Double-click a **MOTION-PATH FB** in the graphic-area **or** 1. See How to Open a dialog [513] The **MOTION-PATH DIALOG** is now open.
Dialog :	**Motion-Path dialog** [380]

Compare a Motion-Path FB with a Motion-Dimension FB

FUNCTION-BLOCK	MOTION-PATH FB	MOTION-DIMENSION FB
Motion Element	MOTION-POINT	MOTION-PART
Constraint	SKETCH-PATH	JOINT
Input	MOTION-DATA AT INPUT-CONNECTOR	MOTION-DATA AT ITS INPUT-CONNECTOR
SUMMARY	*Motion-Point is constrained by a sketch-element and motion-data controls its*	*A Motion-Part is constrained by a joint and motion-data controls its position, velocity, and acceleration.*

FUNCTION-BLOCK	MOTION-PATH FB	MOTION-DIMENSION FB
	position, velocity, and acceleration	

1.4.4.7 Add Measurement FB

Measurement FB

Use a **MEASUREMENT FB** (in a **MECHANISM-EDITOR**) to dimension and measure the motion between sketch-elements that are (usually) in different **PARTS**.

Measurements / Dimensions

You can measure the:

- Length of a **LINE, CAD-LINE** , Radius of a **CIRCLE**
- Distance between: two **POINTS****, a **LINE*** and a **POINT****, or two parallel **LINES***
- Angle between two **LINES*** or three **POINTS****.

* **LINE, CAD-LINE, X-AXIS, Y-AXIS.**

** **POINT, START-POINT, END-POINT,** and **CENTER-POINT.**

Output from Measurement FB

The motion-data at the output-connector(s)§ from the **MEASUREMENT FB** includes:

- the dimension
- the dimension's first and second motion-derivative.

§ Two(2) output-connectors when you measure between two **POINTS** - see more details below

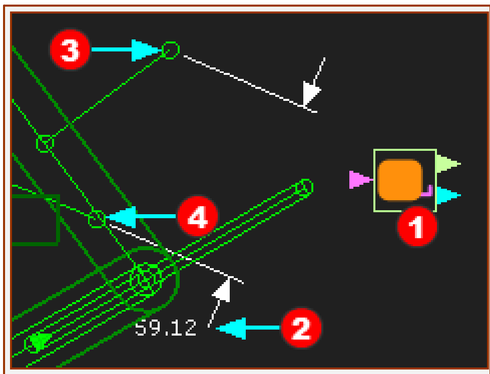
Connect the Measurement FB to a:

- GRAPH FB [172] or STATS FB [179] - to analyse the dimension and motion-data
- **MOTION-DIMENSION FB** or - to use the motion-data as the motion for a **Motion-Part**.
- **MOTION-PATH FB** - to use the motion-data as the motion for a **Motion-Point**.
- **MOTION FB** - to use the motion-data as the **X-axis** of a Motion.
- **MATH FB** - to use the motion-data in an equation, or to change the units of the motion-data.

Messages:

Message in the Feedback-Area [270] may be: '1 Mechanism dependencies detected'. This means, that the motion of a kinematic-chain is a function of (dependent on) the motion (measurement) of a different kinematic-chain.

Add Measurement FB



Measurement FB between two Points

1. Click Kinematics FB toolbar [142] > **Measurement FB**

or

1. Click **Add menu** > **Measurement FB**

2. Click a sketch-element or the sketch-elements between which you want to measure.

In this example, the **MEASUREMENT FB**❶ and **DIMENSION** ❷ is between two **POINTS**❸❹

3. Click the graphic-area.

	The MEASUREMENT FB❶ and DIMENSION❷ are now in the graphic-area.

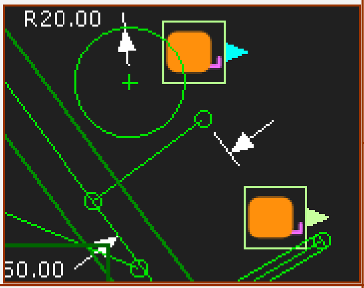
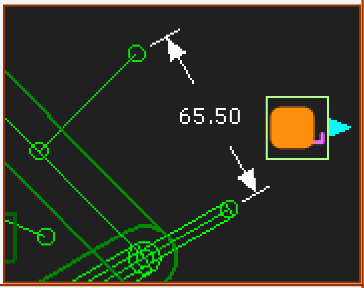
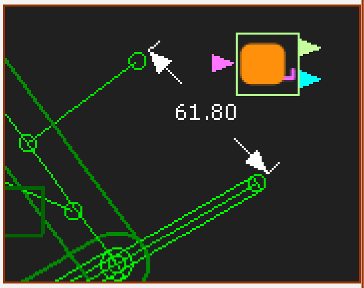
NOTE:

In the graphic-area, to move the:

- FB❶ – Drag the FB❶
- Dimension❷ – CTRL + Drag the FB❶ or the DIMENSION❷.

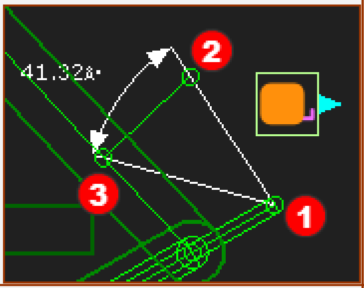
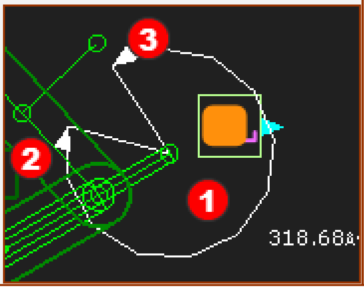
Measurement FB

Linear Distance

* LINE, CAD-LINE, X-axis, Y-axis ** POINT, START-POINT, END-POINT, CENTER-POINT		
Image	Sketch-Elements	Measurement Dimension
	LINE or CAD-LINE.	Length: Click the LINE*
	CIRCLE or ARC.	Radius: Click the CIRCLE or ARC.
	POINT - LINE	Perpendicular Distance between Line and Point. 1. Click the POINT** 2. Click the LINE*
	POINT - POINT	Linear Distance AND Angle between two Points. • Click a POINT** • Click a POINT** A MEASUREMENT FB between two POINTS** has two output-connectors: • **Top output-connector**: distance between two POINTS, and first and second motion-derivatives • **Bottom output-connector**: angle between two POINTS, and first and second motion-derivatives

Angle between Three Points

Measurement: Angle between Three Points - Internal Angle and External Angle. ** POINT, START-POINT, END-POINT, CENTER-POINT		
Image	Sketch-Elements	Measurement Dimension

	Three **POINTS** 1. Click 'Apex' POINT** ❶ 2. Click POINT** ❷ 3. Click POINT** ❸	**Internal Angle** The angle is from ❷ to ❸ in a **Counter-Clockwise** direction.
	Three **POINTS** 1. Click 'Apex' POINT**❶ 2. Click POINT** ❷ 3. Click POINT** ❸	**External Angle** The angle is from ❷ to ❸ in a **Counter-Clockwise** direction.

Angle between Two Lines

After you select the two **LINES**, and before you click in the graphic-area to place the **MEASUREMENT DIMENSION** and **FB**, you can display one of eight angles. Move your mouse-pointer around the APEX of the two **LINES**: CASE 1: Mouse -Pointer moves COUNTER-CLOCKWISE : ***4 angles < 180°*** : the ***Acute, Obtuse, Supplementary*** and ***Vertical*** angles CASE 2: Mouse-Point moves CLOCKWISE : ***4 angles > 180°*** : the ***reflex angles*** - or the angles that are external to those angles of CASE 1.
Video: Eight Measurement Angles between two Lines: Double-click to watch Video

1.4.4.8 Add Point-Data FB

Point-Data FB

See also **Point-Data dialog** [392]

Use a POINT FB to measure the motion of a POINT* has with respect to the MECHANISM PLANE.

It has three output-connectors. From top to bottom, the motion-data at the output-connectors are the:

- Motion of the POINT* parallel to the X-axis
- Motion of the POINT* parallel to the Y-axis
- Magnitude of the motion equal to:
  - $\sqrt{(X^2 + Y^2)}$

Note: Motion include: **Position**, **Velocity**, and **Acceleration** of the POINT*.

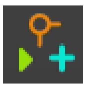
---

* POINT, START-POINT, END-POINT, CENTRE-POINT, MOTION-POINT.

The POINT must be a child to a PART that is **kinematically-defined**.

Add Point-Data FB

STEP 1: Add a Point-Data FB to the graphic-area



1. Click **Kinematics FB toolbar** [142] > Point-Data FB

OR

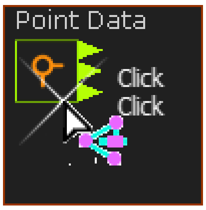
1. Click Add menu > Add Point-Data FB

Then:

2. Click the graphic-area

The POINT-DATA FB is now in the graphic-area (and ASSEMBLY-TREE).

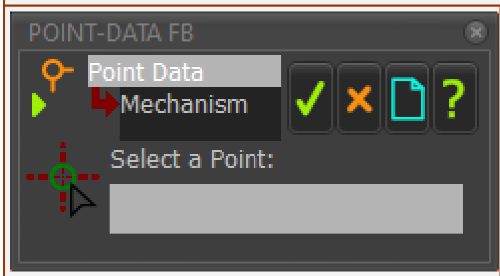
STEP 2: Open the Point-Data dialog:



1. Double-click the POINT-DATA FB in the graphic-area

OR

See How to Open a dialog [513]



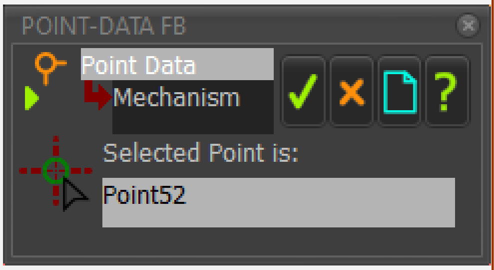
Point-DATA FB - with no Point

STEP 3: Select a Point

When the box is SELECT A POINT:

1. Click a POINT* in the graphic-area.

* POINT, START-POINT, END-POINT, CENTRE-POINT, MOTION-POINT.



Point-Data FB- with a Point

The box is now: SELECTED POINT IS:

STEP 4: Close the Point-Data dialog

1. Click ✓ in the POINT-DATA DIALOG

The motion-data of the POINT is now at the output-connectors of the POINT-DATA FB.

How to use the Point-Data FB:

You can connect a wire from the output-connector of a POINT-DATA FB:

- To plot the Position, Velocity and Acceleration of the Point with a GRAPH FB [172] **- see Note 1 below**
- To the input-connector of a MOTION FB. The motion-data from the POINT-DATA FB become the independent variable for a Motion in ***MotionDesigner***, More usually, the independent variable is the output of a LINEAR-MOTION FB [147] - **see Note 2 below**
- To the input-connector of a MOTION-DIMENSION FB - **see Note 2 below**

Note 1:

To plot the Position, Velocity **and** Acceleration of the POINT in the same graph.

1. Drag a wire from the **same** output-connector of the POINT-DATA FB three times to different input-connectors on a Graph FB [172],
2. Use the Y–axis display options in the **GRAPH INTERFACE** to display the three different motion-derivatives.

Note 2:

There is a message in the Feedback-Area: '1 Mechanism dependencies detected'.

You should Click Rebuild Now [37] when the number of dependencies becomes more than 1 if the data is critical, before you use the data for a Cam, for example.

1.4.4.9 Add Cam-Data FB

Cam-Data FB

See also: Cam-Data FB > Cam Analysis [357] ; Cam-Data FB > Cam Coordinates [361]

Add and then edit a **CAM-DATA FB** to link it to a 2D-CAM. The 2D-CAM must already be in the model (**see Add 2D-Cam)** [109].

You use the **CAM-DATA FB** to calculate and save coordinates of the **2D-CAM** cam-profile - see Cam-Coordinates [361], Cam-Coordinates File Formats

You can also analyze five different design parameters of the **2D-CAM** - see Cam Analysis [357]

Terminology:

CAM / 2D-CAM :	A **CAM** is a curved profile or surface that imparts a displacement (a motion-law) to a **FOLLOWER-PART** by either point or line contact with a **CAM-FOLLOWER**.
CAM-DATA FB :	A FB that you can link to a 2D-CAM.

2D-Cam Work-Flow

1. Add a 2D-CAM - **see Add 2D-Cam** [109]

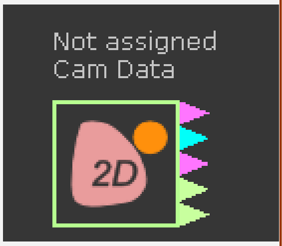
IF the new 2D-CAM is a **Conjugate-Cam** or a **Groove-Cam** (or other **BODY-CLOSED CAM**):

1.a. Add a CONJUGATE-CAM FB - **see Add Conjugate Cam FB** [139]

1.b. Edit the CONJUGATE-CAM FB to add at least two 2D-CAMS - **see Conjugate-Cam dialog** [353].

2. Select the **2D-CAM** or the **CONJUGATE-CAM** as the Power Source for the Follower - **see Configure-Power Source** [480]
3. Review the **2D-CAM** : **Display, Properties, Life, ... - see below**
4. YOU ARE HERE: Add a CAM-DATA FB - **see below**
5. Edit the CAM-DATA FB to link it to the 2D-CAM- - **see Cam-Analysis** [357]
6. Connect wires from the CAM-DATA FB to a GRAPH FB to analyse 5 parameters **see below**
7. Edit the CAM-DATA FB again to Calculate the Cam-Coordinates - **see Cam-Coordinates dialog** [361]

Add Cam-Data FB



Cam-Data FB in Graphic-Area

STEP 1: Add a CAM-DATA FB to the **GRAPHIC AREA**

1. Click Kinematics FB toolbar [142] > Cam-Data FB

or

1. Click Add menu > Cam-Data FB

2. Click the graphic-area

The CAM-DATA FB is now in the graphic-area - see left - (and **ASSEMBLY-TREE**).

Now, you must link the CAM-DATA FB to a 2D-CAM in the model.

See Cam-Data dialog [357]

1.4.4.1 Add Graph FB
0

Graph FB

Terminology:

GRAPH FB :	You connect wires to the input-connector of a GRAPH FB to plot any data that is available at the output-connectors of other FUNCTION-BLOCKS over a complete machine-cycle.
Data-Channel :	Each wire provides three parallel streams of data. For example, the data is linear or angular position, plus the first and second motion-derivatives. We name each stream of data on a wire a **Data-Channel**.

Add Graph FB

STEP 1: Add a Graph FB to the graphic-area

1. Click Kinematics FB toolbar[142] > Graph FB

OR

1. Click Add menu > Add Graph FB

Then :

2. Click the graphic-area

The GRAPH FB is now in the graphic-area.

STEP 2: Connect wires to the input-connectors of the GRAPH GB

The GRAPH FB has five(5) INPUT-CONNECTORS.

From the top input-connector the data is:

- Input 1 – 4 : Y-axis inputs - **See below: Y-axis Data-Channel Options**[174]
- Input 5 (bottom) : X-axis input - **See below: X-axis Data-Channel Options**[173]

Note: Data-Values at the input-connector for the X-axis (#5 input-connector)

OPTION 1: no data (not connected). The X-axis is equal to the MASTER-MACHINE-ANGLE

OPTION 2: data increases steadily. For example, the output-connector of a LINEAR-MOTION FB, or GEARING FB

OPTION 3: data increases and/or decreases within a range of values. For example, the output-connector of a MOTION FB, MEASUREMENT FB, or POINT-DATA FB

Note: if there is a **fly-back** across the graph, click Rebuild Now[37] - this command may remove the **fly-back**.

If **Rebuild Now** fails, increase the number-of-steps to ~360 - **see Machine-Settings > Number-of-Steps**[291]

STEP 3: Open the GRAPH FB

1. Double-click the GRAPH FB in the graphic-area

OR

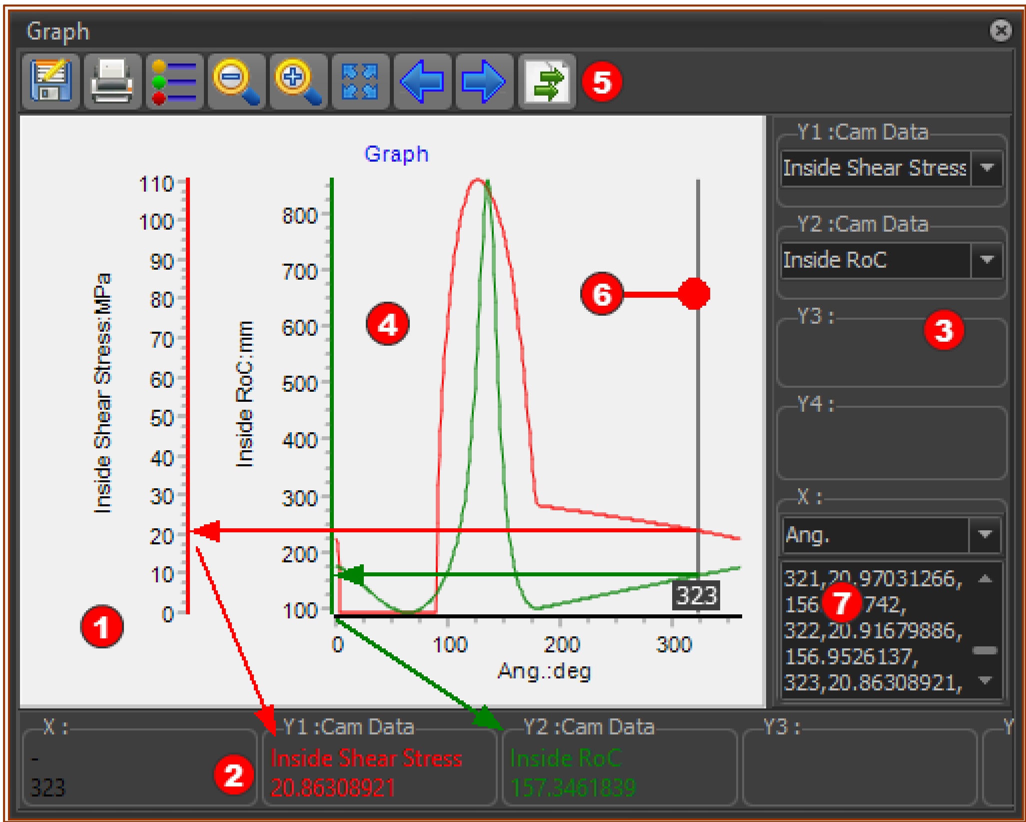
See How to open a dialog[513]

The GRAPH DIALOG is now open.

STEP 4: **See: Graph FB Interface**[173].

To review the kinematic, kinetostatic and cam data that you can plot with a **GRAPH FB**.
See also: [Application Settings | Number Format | DATA FORMAT](286)

Graph FB Interface



Graph FB:

1 Y–axis : Scales

- Maximum 4 scales, that correspond to motion-value at input-connectors Y1 to Y4
- The color of the Y–axis and the plot-line are the same.

2 Y–axis : Digital Readout (D.R.O.)

- The X-axis and the Y1 – Y4-axis values at the position of the vertical cursor 6.
- The Number-Format for the X–axis, Y–axis, and data in Clipboard 7 are configured in Application Settings | Number Format tab. (286)

3 X-axis and Y-axis Data-Channel Selector

- Each wire that is connected to a Y-axis orthe X-axis input-connector has up to 3 **Data-Channels**
- Select the **Data-Channel** in the drop-down box - ***see*** **Data-Channel Selectors** (174)

4 The Graph Area:

- The color-coded graphs for each Y–axis input
- **Vertical-Cursor 6 - see more below**

5 Toolbar:

- Save, Print, Graph Settings (389), Zoom Extents, Pan Left, Pan Right, Copy Graph Data to Clipboard – see 7 below

---

6 Vertical-Cursor

When you click in the area of the Graph❹, the ***Vertical Cursor***❻ snaps to your mouse pointer

When you drag your pointer in the graph :

- the Vertical-Cursor follows your pointer
- the Digital Readouts❸ continuously updates to indicate the Y-axis values
- the **MASTER MACHINE ANGLE** updates to the position of the Vertical Cursor
- the kinematic-chains that are **kinematically-defined** move to a position that is defined by the **MASTER MACHINE ANGLE**.

❼ **Copy Graph Data to Clipboard**

- When you click the right-most icon in the toolbar❺, the graph data is copied to the Clipboard❼.
- Paste this data to Excel, or
- To ***MotionDesigner*** in the **DATA TRANSFER TABLE**. You can plot it as an **OVERLAY** or use it as a **MOTION** (use Z Raw-Data).

Notes:

- Configure the Number-Format for the X–axis, Y–axis, and 'Data in Clipboard' in Application Settings | Number Format tab. [286]

❷ **Data-Channel Selectors:**

Each wire you connect to the **GRAPH FB** has three **Data-Channels**. The data-channels that are available are a function of the **FB** from which you connector the wire to the **GRAPH FB INPUT-CONNECTOR**.

Use the **Data-Channel Selectors** to select which of the three data-channels to plot.

The data-channels from a:

- **Function-Block** [142] (**LINEAR-MOTION, GEARING, MOTION, MOTION-DIMENSION**):
  - Linear position, velocity, acceleration.
  - Angular position, velocity, acceleration.
- **Measurement FB** [166] or **Point-Data FB** [169]:
  - Linear position, velocity, acceleration of the Measurement dimension or Point's position.
  - Angular position, velocity, acceleration of the Measurement dimension.
- **Cam-Data FB** [171]
  - **PRESSURE-ANGLE.**
    You can plot three Pressure Angles – See Pressure Angle
  - **CAM RADIUS-OF-CURVATURE.**
    You can plot Radii-of-Curvature for Inner Cam and the Outer Cam.
  - **CONTACT CAM-FORCE**
  - **MAXIMUM CONTACT-SHEAR-STRESS.**
    You must know the material-properties to obtain this data - See 2D-Cam dialog [327] .
  - **ENTRAINMENT VELOCITY**
    The Entrainment-Velocity for the Inner Cam and the Outer Cam.
- **Force-Data FB** [191]

If the **FORCE-DATA FB** references a:

- **Pin-Joint that is a Power-Source,** you can plot Application-Torque for a Servo-motor and Gearbox.
- **Slide-Joint that is a Power-Source**, you can plot the Application-Load for a Linear Servo-motor.
- **PIN-JOINT** or **SLIDE-JOINT** that is NOT a Power-Source:

  $$F_{total} = \sqrt{F_x^2 + F_y^2}$$

  - F_x - Force in the X-axis direction :
  - F_y - Force in the Y-axis direction:
- **2D-CAM**: you can plot the Contact Force between the Cam and the Cam-Follower.
- **SPRING FB**: you can plot the Total Force, the X Force and the Y Force acting on the **anchor-Point** of the Spring.

TOP-TIP:

To show all three Data-Channels from the same output-connector of a Function-Block,

1. Drag three wires from the same output-connector of the Function-Block to three(3) different **GRAPH FB** input-connectors.
2. Select the different **Data-Channels** with the **Y-axis Data-Channel Selector** drop-down (right-side of graphs)

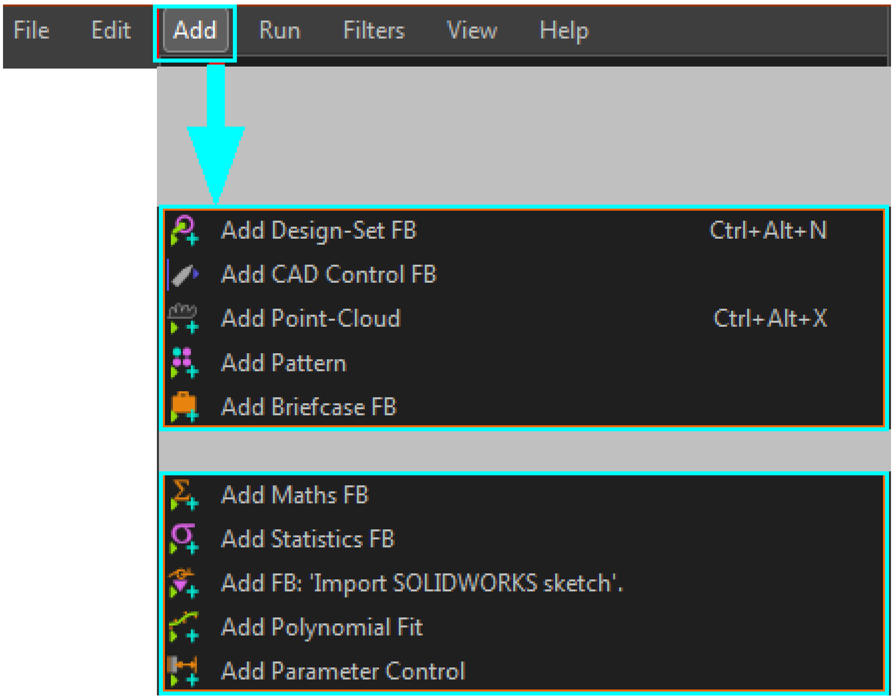
1.4.5 Modelling Function-Blocks

Modeling Function-Blocks

Use **Modeling Function-Blocks** for more complex modeling.

See also How to Connect FBs 144

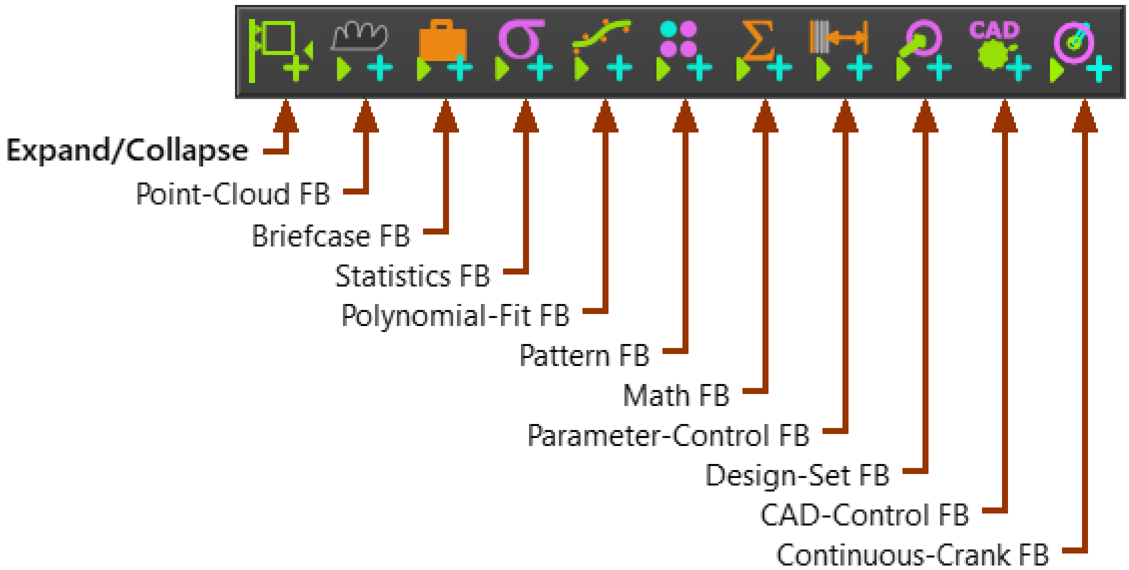
Modeling FB menu (from Add menu)



Modelling FB menu

Modeling FB toolbar

The **Modeling Function-Block** toolbar is above the GRAPHIC-AREA.



Modeling FB toolbar

1.4.5.1 Add Point-Cloud FB

Why use a Point-Cloud FB?

Generally, a **Point-Cloud** is an array of coordinates that specify the shape of a 2D or 3D object. You can use a **POINT-CLOUD FB** for many reasons - for example, to reverse-engineer the shape of a Cam. From the shape, you can also reverse-engineer the motion of a Cam-Follower.

Use the **POINT-CLOUD FB** to:

- Import data that represents the shape of a 2D object
- Fit a **CURVE** to the data and shape of the 2D object

See also : Point-Cloud FB dialog [425]

Add Point-Cloud FB

What to do :	**STEP 1 Add a Point-Cloud FB to a Part** 1. Click **Add menu > Add Point-Cloud FB**  **OR** 1. Click **Modeling FB toolbar** [176] **> Add Point-Cloud FB** The **COMMAND-MANAGER** has one selection box. You must select a **PART**. 2. Click a **PART** in the graphic-area or **ASSEMBLY-TREE** 3. Click ✓ in the **COMMAND-MANAGER** The **POINT-CLOUD FB** is now in the graphic-area.
	STEP 2 Open the Point-Cloud dialog: 1. Double-click a **POINT-CLOUD FB** in the graphic-area or **ASSEMBLY-TREE** **OR** **See How to Open a dialog** [513]. The **POINT-CLOUD DIALOG** is now open.
	STEP 3 See Point-Cloud dialog [425].
Video :	See YouTube: http://youtu.be/IME9Eb4mpQE

1.4.5.2 Add Briefcase FB

Why use a Briefcase FB?

When a model becomes complex, and there are many kinematic-chains in each **MECHANISM-EDITOR**, it becomes more difficult to identify which Function-Blocks control which Kinematic-Chain.

- You can use **BRIEFCASE FB**s to hide **FUNCTION-BLOCKS** from the graphic-area
- You can use a different **BRIEFCASE FB** to group each set of **FUNCTION-BLOCKS**.
- You can name each **BRIEFCASE FB** to help remind you which FBs are in which **BRIEFCASE FB**
- You can select and edit each **FUNCTION-BLOCK** directly from the **BRIEFCASE FB** interface
- You can add a **BRIEFCASE FB** to a different **BRIEFCASE FB**.

See also : Briefcase FB dialog [422]

Add Briefcase FB

What to do :

STEP 1: Add a Briefcase FB to the graphic-area:

1. Add menu > Add Briefcase FB

OR

1. Modeling FB toolbar [176] > Add Briefcase FB

Then:

2. Click the graphic-area

The **BRIEFCASE FB** is now in the graphic-area.

STEP 2: Open the Briefcase FB dialog

1. Double-click the **BRIEFCASE FB** in the graphic-area or **ASSEMBLY-TREE**

OR

See How to Open a dialog [513]

STEP 3: See Briefcase FB dialog [422]

1.4.5.3 Add Statistics FB

Why use a Statistics FB?

Statistical data may include values as RMS , Maximum, Minimum, etc. values of a variable (wire data-channel).

Use a STATISTICS FB to list the **Statistical data** for the data that is available at the output of a different **Function-Block.**

See also : **Statistics dialog** [404]

Add Statistics FB

What to do : STEP 1: Add a Statistics FB to the graphic-area

1. Click Add menu > Add Statistics FB

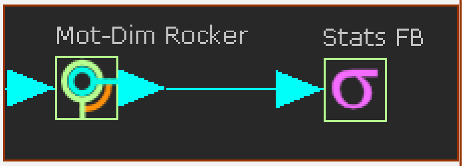
OR

1. Click Modeling FB toolbar [176] > Add Statistics FB

Then:

2. Click the graphic-area

The STATISTIC FB is now in the graphic-area



STEP 2: Connect a wire to its input-connector

1. Drag a wire from a different FB to the input-connector of the STATISTICS FB

A wire is now connected to the input-connector of the STATISTICS FB.

STEP 3. Open the Statistics dialog

1. Double-click a STATISTICS FB in the graphic-area

OR

1. **See** **How to Open a dialog** [513]

STEP 4: See Statistics dialog [404]

Notes:

Wires that connect FUNCTION-BLOCKS have **3 Data-Channels** (in most cases).

The three(3) **Data-Channels** are usually:

- Position, Velocity, Acceleration (Angular or Linear).
- Total Force, Force X, Force Y (relative to the MECHANISM-PLANE

See Data-Channels [144] for more information.

Compare a GRAPH FB with a STATS FB:

- GRAPH FB : plot data at its input-connectors over one machine-cycle.
- STATS FB : summarize a machine-cycle of data at its input-connector with different statistical parameters.

1.4.5.4 Add Polynomial-Fit FB

Why use a Polynomial-Fit FB?

With Inverse-Kinematics, your motion-design (that you design with **MotionDesigner**) is defined for a Tool/End-Effector. Then, typically, you add more parts, which you join to the Tool-Part, to create a more complex kinematic-chain. Finally, you use a servo-motor to drive a part that in the kinematic-chain, that has a different motion to the Tool-Part.

To plot the motion for the servo-motor you can use a Measurement FB [166], add a Graph FB [172], and then export a list of positional data-points for the servo-controller.

While this is often satisfactory, it is often preferable to give the servo-controller a series of polynomials rather than a list of data-points.

The output from the POLYNOMIAL FIT FB is the best-fit motion to the data at its input-connector, but defined as a series of 5th order Polynomials.

You can export the Polynomials directly to **MotionDesigner**. In **MotionDesigner**, you can further manipulate the Polynomials, as required.

Add Polynomial-Fit FB

What to do : **STEP 1: Add a Polynomial-Fit FB to the graphic-area**

1. Click **Add menu > Add Polynomial-Fit FB**

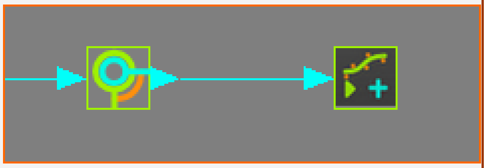
OR

1. Click Modeling FB toolbar [176] **> Add Polynomial-Fit FB**

Then:

2. Click the graphic-area

The POLYNOMIAL-FIT FB is now in the graphic-area



STEP 2: Connect a wire to its input-connector.

1. Drag a wire from a different FB to the input-connector of the POLYNOMIAL-FIT FB

A wire is now connected to the input-connector of the POLYNOMIAL-FIT FB.

STEP 3. Open the Polynomial-Fit dialog:

1. Double-click a POLYNOMIAL-FIT FB in the graphic-area

OR

1. **See How to Open a dialog** [513]

The POLYNOMIAL-FIT DIALOG is now open.

STEP 4: See Polynomial Fit dialog [406]

Notes:

Wires that connect FUNCTION-BLOCKS have **3 Data-Channels** (in most cases).

The three(3) **Data-Channels** are usually:

- Position, Velocity, and Acceleration, or
- Total Force, Force X, and Force Y

See Data-Channels [144] for more information.

1.4.5.5 Add Pattern FB

Why use a Pattern tool?

A **PATTERN FB** makes an array of **SOLIDS** that are in the model.

A **PATTERN** is similar to, but more capable than, a circular or linear array as found in your CAD.

The **PATTERN FB** must anticipate the motion of each **SOLID** in the array at each instant of a machine-cycle relative to the other **PARTS** in the model.

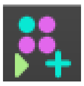
When you use the **PATTERN FB** to make an array of a **SOLID**, but hide a **Copy** at different phases in the machine-cycle, you also remove its **MASS-PROPERTIES.**

See also: Pattern dialog [414]

Add Pattern FB

What to do :

STEP 1 Add a Pattern FB to the graphic-data



1. Click **Add menu** > **Add Pattern FB**

OR

1. Click Modeling FB toolbar [176] > **Add Pattern FB**

Then:

2. Click the graphic-area

The **PATTERN FB** is in the graphic-area.

STEP 2 Open the Pattern dialog

1. Double-click a **PATTERN FB** in the graphic-area (or **ASSEMBLY-TREE**)

OR

See How to Open a dialog [513].

The **PATTERN DIALOG** is now open.

STEP 3 See Pattern-FB dialog [425].

Application Note: Pattern FB and a DRIVING Pulley:

A **Driving Pulley** is one in which the motion is applied to the **PULLEY** and not to the **BELT**.

If the timing is such that the Pulley makes one rotation in a machine-cycle, the Belt does not complete its motion along the sketch-path in one machine-cycle.

If you want to use a **PATTERN FB** to model an array (Pattern) of a **SOLID** element to be along the complete length of a **BELT**, you must make changes to the timing of the **PULLEY**.

1. Add a **GEARING FB** to the input-connector of the **MOTION-DIMENSION FB** that is driving the **PULLEY**
2. Open the **GEARING FB DIALOG**
3. Edit the **GEARING RATIO** = **BELT-LENGTH / PULLEY-CIRCUMFERENCE**

The **PATTERN** of the **SOLID** elements should now be at equal spaces along the complete **BELT-LENGTH**.

1.4.5.6 Add Math FB

Why use a Math FB?

A **MATH FB** allows you to develop new functions that may not be available with other FBs.

Examples:

- Mechanism-Synthesis, when combined with Measurement FBs[166] and Parameter-Control FBs[184].
- Add two or more Motions to create a new Motion - for example, add a Modified-Sinusoid motion-law to a Constant-Velocity motion-law to give a Starting-Velocity and Ending-Velocity that are not zero.
- Define Parametric equations for Piggyback Sliders.
- ...

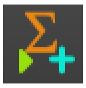
See also : Math FB dialog[396]

Add Math FB

What to do :

STEP 1 Add a Math FB to the graphic-data

1. Click **Add menu > Add Math FB**



OR

1. Click **Modeling FB toolbar**[176] **> Add Math FB**

Then:

2. Click the graphic-area

The **MATH FB** is in the graphic-area.

STEP 2. Open the Math dialog

1. Double-click the **MATH FB** in the graphic-area

OR

See How to Open a dialog[513].

The **MATH DIALOG** is now open.

STEP 3. See Math FB dialog[396]

1.4.5.7 Add Parameter-Control FB

Why use a Parameter-Control FB?

Use a PARAMETER-CONTROL FB to:

- control a DIMENSION of a sketch-element
- control an EXTRUSION DEPTH of an MD-SOLID*
- control an EXTRUSION-OFFSET of an MD-SOLID*

Apply a PARAMETER-CONTROL FB for Mechanism-Synthesis, animations with more realistic effects.

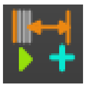
* When a PARAMETER-CONTROL FB controls the EXTRUSION-DEPTH or EXTRUSION-OFFSET of an EXTRUSION that has many facets, the model is slow to cycle.

See also : Parameter-Control dialog 411

Add Parameter-Control FB

What to do :

STEP 1 Add a Parameter-Control FB to the graphic-data



1. Click **Add menu > Add Parameter-Control FB**

OR

1. Click Modeling FB toolbar 176 > **Add Parameter-Control FB**

Then:

2. Click the graphic-area

The PARAMETER-CONTROL FB is in the graphic-area.

STEP 2: Open the Parameter-Control dialog

1. Double-click the PARAMETER-CONTROL FB in the graphic-area or ASSEMBLY-TREE

OR

1. **See How to Open a dialog** 513.

The PARAMETER-CONTROL DIALOG is now open.

STEP 3: See Parameter-Control dialog 411

1.4.5.8 Add Design-Set FB

Why use a Design-Set?

Use DESIGN-SETS to edit a number of dimensions and parameters in one place - the DESIGN-SET.

Add to a DESIGN-SET those dimensions and parameters that you believe to be important to the outcome of a design-objective. When dimensions are in a DESIGN-SET, you cannot edit them in the model. You can add more than one DESIGN-SET to the model.

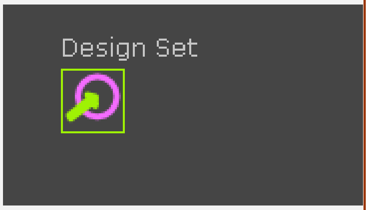
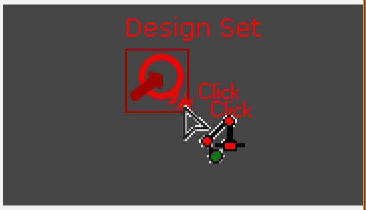
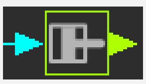
Different DESIGN SETS can be used for different purposes. A Design-Set to help make changes to a machine for a packaging size change.

A Design-Set should also remind you, or instruct other design engineers, which dimensions and parameters to edit.

You can even add to a DESIGN-SET those dimensions that you do not want to edit. Give it the Label - **Do Not Edit!** In this way, you cannot accidentally edit the dimensions that you want to protect.

See Design-Set dialog 431

Add Design FB

	**STEP 1 Add a Design-Set FB to the graphic-data** 1. Click **Add menu > Add Design-Set FB** **OR**  1. Click Modeling FB toolbar 176 > **Add Design-Set FB** Then: 2. Click the graphic-area The DESIGN-SET FB is in the graphic-area.
	**STEP 2: Open the Design-Set dialog** 1. Double-click the DESIGN-SET FB in the graphic-area or **ASSEMBLY-TREE.** **OR** **See How to open a dialog** 513 The DESIGN-SET DIALOG is now open. **STEP 3: See Design-Set dialog** 431
Notes :	• When you add a dimension to a DESIGN-SET, you can edit the dimension ***only*** in the DESIGN-SET. • Dimension and its extension lines you add to the DESIGN-SET become gray in the PART-EDITOR.
	• A MOTION-DIMENSION FB becomes ***gray*** if you add the MOTION-DIMENSION to the DESIGN-SET.
See also:	***Online Video : Tutorial 7: Design-Sets***

1.4.5.9 Add CAD-Control FB

Why add a CAD Control FB?

Use a CAD CONTROL FB to synchronize the motion of one or more parts that are in a **SOLIDWORKS**® Assembly with the motion of same PARTS in a ***MechDesigner*** model/assembly.

Question: Why do this when **MechDesigner** is perfect to synchronize the motions of all of the PARTS in a model?

Answer: Because SOLIDWORKS® has useful tools. For example, **Collision Verification, Interference Detection, ...**

The **Clearance Detection** tool in SOLIDWORKS® evaluates the exact gap between parts. You can use this tool to check that a 3D-CAM you export to SOLIDWORKS has the same Clearance as the Radial Clearance that you enter in the 3D-CAM DIALOG.

After you configure the CAD CONTROL DIALOG, motion-data that controls PARTS in **MechDesigner** model/assembly are piped to the SOLIDWORKS Distance or Angle mates that define the position of a Part in the assembly in SOLIDWORKS®.

Add CAD Control FB

STEP 1 Add a CAD-Control FB to the graphic-data

1. Click **Add menu > Add CAD-Control FB**

OR

1. Click Modeling FB toolbar [176] > **Add CAD-Control FB**

Then:

1. Click the graphic-area

The CAD-CONTROL FB is in the graphic-area.

STEP 2: Open the CAD-Control dialog:

1. Double-click a CAD-CONTROL FB in the graphic-area

OR

See How to Open a dialog [513].

The CAD-CONTROL DIALOG is now open.

STEP 3: See CAD Control dialog [394]

Note:

Angles in SOLIDWORKS® cannot be negative, but they can be in **MechDesigner**. Thus, you may need to use GEARING FBS to add a constant to the output values so that they have the same minimum and maximum values as those in SOLIDWORKS®.

1.4.5.1 Add Continuous Crank FB 0

Why add a Continuous-Crank FB?

In a mechanism that is driven by an **oscillating cam-follower**, the motion is usually designed for a Tool-Part and the **oscillating motion** of the Cam-Follower is found with Inverse-Kinematics.

Similarly, a mechanism may be driven by a **crank that rotates completely** (360º). Again, the motion is designed for the Tool, and the continuously rotating motion for the crank is found with Inverse-Kinematics.

This is a typical application of the CONTINUOUS-CRANK FB - it finds the correct length and rotational motion of the Crank.

See also: Continuous-Crank FB dialog[437]

Add Continuous Crank FB

STEP 1: Add a Continuous-Crank FB to the graphic-area

1. Click Modeling FB toolbar[176] > Continuous-Crank FB

Then:

2. Click the graphic-area

The CONTINUOUS-CRANK FB is now in the graphic-area and ASSEMBLY-TREE.

STEP 2: Open the Continuous-Crank dialog:

1. Double-click a CONTINUOUS-CRANK FB in the graphic-area

OR

See How to Open a dialog[513].

The CONTINUOUS-CRANK DIALOG is now open.

STEP 3: **See Continuous-Crank FB dialog**[437]

1.4.6 Force elements

To calculate **Force vectors** that act on PARTS, there must be a minimum of one kinematic-chain that is **kinematically-defined** and a minimum of one PART with **Mass and Inertia** properties.

PARTS have three sources of **Mass Properties (Mass & Inertia)**:

They **Mass Properties** are:

- defined by the shape and EXTRUSION-DEPTH of a PROFILE/ EXTRUSION,
- defined by you as USER MASS-PROPERTIES
- imported with a SOLIDWORKS part document onto a CAD-LINE

You can use each of these to add together the **Mass Properties** from each source

It is also important to **Configure the Power Source** to make sure the Power that flows through the kinematic-chain is correct.

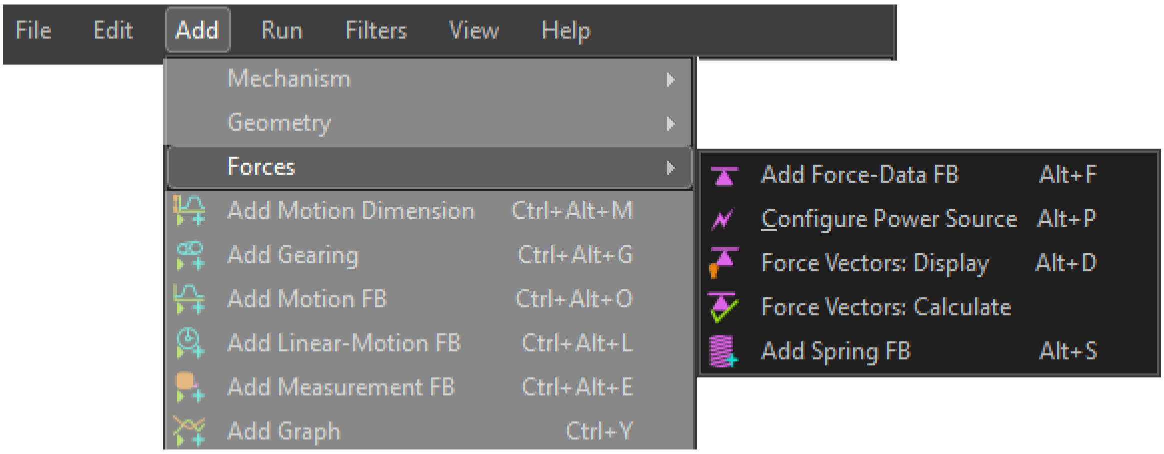
Force elements

Use the tools and options in the **Force elements menu** or **toolbar** to calculate and show the Force vectors that act on PARTS at **Joints** and the machine elements in your model.

On-line Tutorial: **Tutorial 13: Forces Introduction**

Force sub-menu

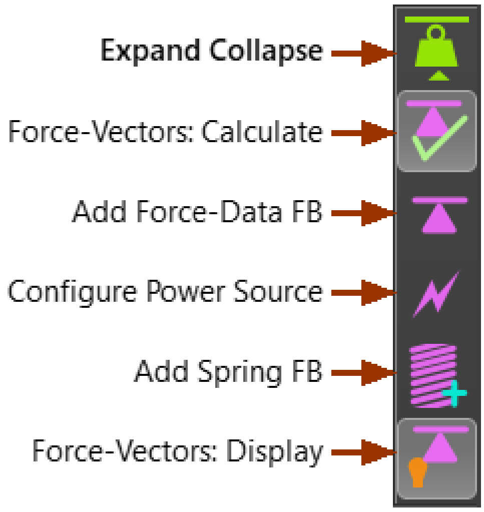
The **Force menu** is a sub-menu to the **Add menu**.



Force menu

Force toolbar

The **Force toolbar** is to the right of the graphic-area.



Forces toolbar

Kinetostatic Forces - Definitions:

FORCE
Action of its surroundings on a body tending to change its state of rest or motion.
LINE OF ACTION OF A FORCE
The line along which the vector that represents a given force lies.
MAGNITUDE OF A FORCE
Number of units of force obtained by comparing a given force with a standard, taken as unit force.

FORCE
ACTIVE [APPLIED] FORCE
Force capable of producing motion.
REACTION
Force arising in a constraint and acting upon a constrained body due to the action of an active force upon that body.
CENTRIPETAL FORCE
Force causing the centripetal acceleration of a particle.
INERTIA FORCE
Product of the mass of a particle and the negative of its acceleration. Following D'Alembert, the inertia force can be regarded as being in equilibrium with the resultant of all the forces acting on the particle.
CENTRIFUGAL FORCE
Inertia force of a particle moving uniformly along a circular path.
CORIOLIS FORCE
Inertia force equal to the product of the mass of a particle and the negative of its Coriolis component of acceleration.
GRAVITATIONAL FORCE
Force equal to the product of the mass of a particle and the Gravitational Acceleration on Earth - taken as 9.806m/s/s.

Acceleration Definitions:

We calculate for you those Forces that result from these Accelerations :

CORIOLIS ACCELERATION
Component of the absolute acceleration of a point due to its velocity relative to a rotating frame of reference. It equals twice the vector product of the angular velocity of the moving frame of reference and the relative velocity of the given moving point.
CENTRIPETAL ACCELERATION
Acceleration of a point towards the center of curvature of its path as it moves along a fixed curve.
TANGENTIAL ACCELERATION
Component of acceleration of a point collinear with its velocity.
NORMAL ACCELERATION
Component of acceleration of a point normal to its velocity.
ANGULAR ACCELERATION
Rate of change of angular velocity with respect to time.

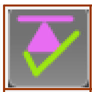
1.4.6.1 Force-Vectors: Calculate

Forces-Vectors: Calculate

Enable **Force-Vectors: Calculate** to silently calculate **Kinetostatic Forces Vectors** at each joint in each kinematic-chain that is **kinematically-defined**.

Force-Vectors: Calculate also calculates the **Application Load** or **Application Load Torque** for each kinematic-chain.

You must select the correct element as the **Power-Source** - see Configure Power Source [480].

	Forces toolbar > Enable Force-Vectors: Calculate
	Forces toolbar > Disable Force-Vectors: Calculate
	Add menu > Forces sub-menu > Enable / Disable Forces-Vectors: Calculate

Note:

To calculate the **Forces Vectors**:

- A minimum of one **PART** has mass and/or inertia

AND / OR

- A **SPRING FB** applies an external force to the kinematic-chain

See also:

Force-Vectors: Display [198]

Kinetostatic Servo-motor and Gearbox selection [483]

1.4.6.2 Add Force-Data FB

Why add a Force-Data FB?

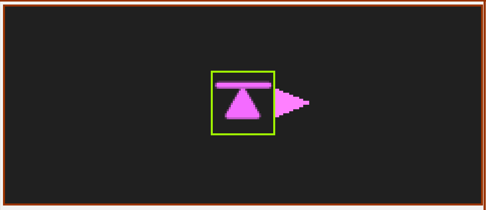
See also: Force-Data FB dialog [442]

Use a **FORCE-DATA FB** to measure the **force*** that **acts ON** a **POINT** at a **JOINT**, **SPRING**, **2D-CAM**, **PULLEY**, or **RACK-PINION**.

It is important to CONFIGURE POWER SOURCE [480] correctly to calculate Force*.

* **Force** is the generic term we use for Force and Torque data.

Add Force-Data FB

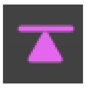


Force-Data FB in graphic-area

STEP 1: Add the Force-Data FB to the graphic-area.

1. Click **Add menu** > **Forces sub-menu** > **Add Force-Data FB**

OR

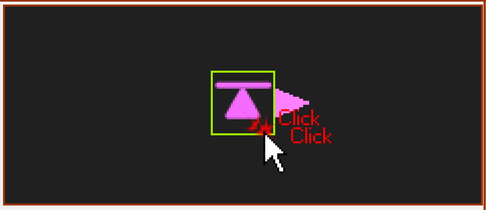


1. Click Forces toolbar [187] > **Add Force-Data FB**

Then:

2. Click the graphic-area

The **FORCE-DATA FB** is now in the graphic-area.



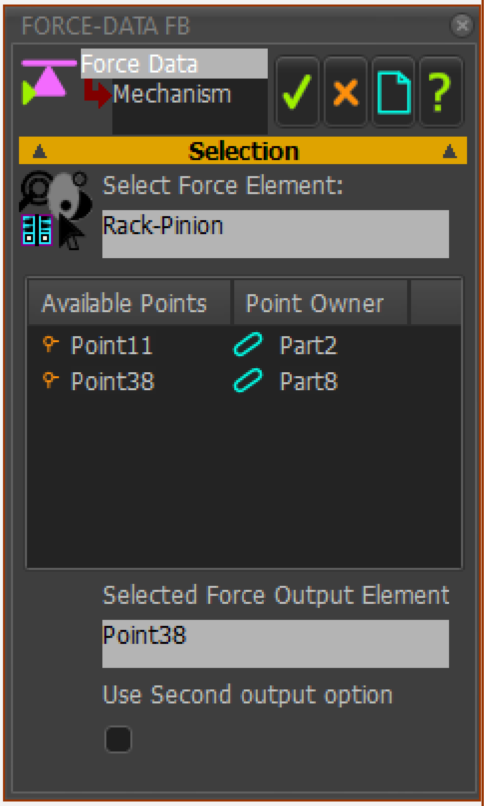
Doube-click to edit Force-Data FB

STEP 2. Open the Force-Data FB dialog

1. Double-click the **FORCE-DATA FB** to open the **FORCE-DATA DIALOG**.

OR

See 'How to open a dialog' [513]



Force-Data FB dialog

STEP 3. Link the Force-Data FB to a Force-Element and a Point

A. Select a Force-Element

1. Click a Force-element in the graphic-area or the **ASSEMBLY-TREE**

Force-elements include: **JOINT**, **2D-CAM, SPRING, PULLEY, RACK-PINION,**

A list of **AVAILABLE POINTS** shows in the box below.

B. Select a Point in the box of Available Points

1. Click a **POINT** in the box below **AVAILABLE POINTS**

If you select a:

- **PIN-JOINT** - there are two(2) **AVAILABLE POINTS**
  One **POINT** on each **PART** of the **PIN-JOINT**
- **SLIDE-JOINT** - there are four(4) **AVAILABLE POINTS**
  Two **POINTS** on each **LINE** of the **SLIDE-JOINT**
- 2D-CAM [109] - zero(0) **AVAILABLE POINTS**
  CONTACT POINT is selected

- SPRING FB 194 - two(2) AVAILABLE POINTS
  The SPRING'S Anchor POINTS
- RACK-PINION 116 - two(2) AVAILABLE POINTS
  The CONTACT-POINTS between the RACK AND PINION

Note: Even if a POINT shows in the SELECTED FORCE OUTPUT ELEMENT box, click the POINT in the AVAILABLE POINTS box.

C. Complete the command

3. Click ✅ to close the FORCE-DATA FB

1.4.6.3 Configure Power Source

Why Configure Power Source?

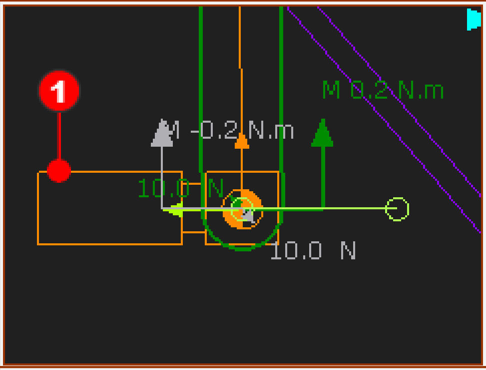
There is one **Power-Source** for each degree-of-freedom in a kinematic-chain.

In the default case, we locate for you the **Power-Source** for a degree-of-freedom at the Joint whose position and motion are defined by a MOTION-DIMENSION FB. The Power-Source may need to be moved to a different joint or element.

You use the **Configure Power-Source** interface to specify which element is the **Power-Source** for each degree-of-freedom in a kinematic-chain.

These elements can be a **Power-Source**: JOINTS (as Motors), 2D-CAMS, SPRINGS, and PULLEYS.

To identify the Power-Source



In the default case, there is a **Power-Source** at a joint whose motion is identified by a MOTION-DIMENSION FB.

A **Motor-Symbol**❶ represents a **Power-Source** at a Joint.

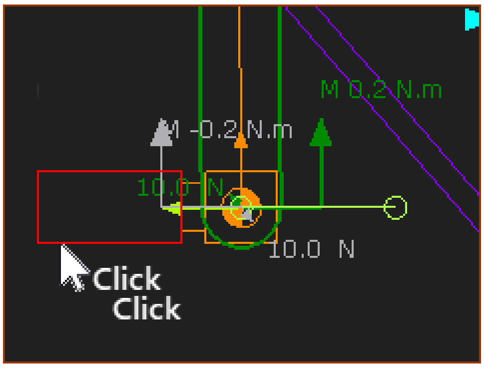
To see the **Motor-Symbol,** enable:

- Force toolbar > Force Vectors: Display 198
- Forces toolbar > Force Vectors: Calculate 190

To open the Configure Power-Source dialog

There are four(4) methods to open the CONFIGURE POWER-SOURCE DIALOG.

METHOD 1



Double-Click the Motor-Symbol

If you can see a **Motor-Symbol**

1. Double-click the **Motor Symbol**.

The CONFIGURE POWER SOURCE DIALOG is now open.

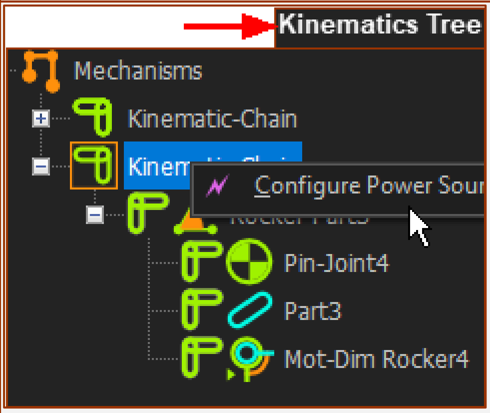
METHOD 2

1. Click the **KINEMATICS-TREE** in the **Element-Explorer**[261]
2. Click the **Kinematic-Chain** for which you want to identify the Power-Source.

The Kinematic-Chain should be Blue.

3. Click **Force toolbar**[187] > **Configure Power Source** icon.

The CONFIGURE POWER SOURCE DIALOG is now open.



Configure Power Source from Kinematics-Tree

METHOD 3 (<<< see image above)

In the **Kinematics-Tree:**

1. Click the **KINEMATICS-TREE** in the **Element-Explorer**[261]
2. Click the **Kinematic-Chain**

The Kinematic-Chain label in the **Kinematics-Tree** should be Blue.

3. Right-click the Kinematic-Chain in the **Kinematics-Tree**
4. Click **Configure Power Source** in the shortcut menu

The CONFIGURE POWER SOURCE DIALOG is now open.

METHOD 4

1. Click the **KINEMATICS-TREE** in the **Element-Explorer**[261]
2. Click the **Kinematic-Chain**
3. Press the **ALT+P** keyboard shortcut

The CONFIGURE POWER SOURCE DIALOG is now open.

See Configure Power Source dialog[480].

1.4.6.4 Add Spring FB

Why add a Spring FB?

See also: Spring FB dialog[439]

Use a SPRING FB to add a force between two POINTS*. The POINTS* are usually in different PARTS.

* POINT, START-POINT, END-POINT, CENTER-POINT.

The SPRING FB can superimpose these forces:

- SPRING FORCE - a force proportional to the dimensional difference between the Spring's ANCHOR-POINT and its FREE-LENGTH.

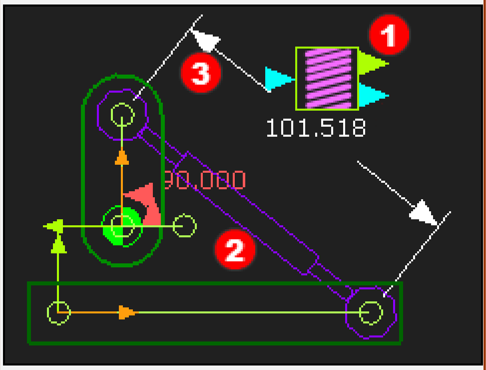
- CONSTANT FORCE - a constant force that pulls or pushes the Spring's ANCHOR-POINTS, throughout the machine-cycle.
- COULOMB CONSTANT-FORCE - a constant force that acts in the opposite direction to the relative velocity between the SPRING'S ANCHOR-POINT.
- DRAG-FORCE - a force that is proportional to the relative velocity between the Spring's ANCHOR-POINTS.
- DRAG FORCE-SQUARED - a force that is proportional to the square of the relative velocity between the Spring's ANCHOR-POINTS.
- FORCE-FUNCTION - a force-function that you add to the input-connector

The **Forces values** are superimposed (added) to together to give the **Total-Force** at the output-connector.

Terminology:

Anchor-Points :	The two(2) POINTS that define the axis through which the SPRING FB applies its Force. You select the Points when you add a SPRING FB.
Force element :	A SPRING FB is a **Force-Element**.
Linear Motive Force :	The SPRING FB can become a Power-Source. It indicates the Application Force that you must supply to drive the Kinematic-Chain. **A Linear Motor would provide the Linear-Force.**

Add Spring FB



Spring FB in the graphic-area

STEP 1: Start the Add Spring FB command

1. Click **Add menu** > **Forces sub-menu** > **Add Spring FB**

OR

1. Click Forces toolbar 187 > **Add Spring FB**

STEP 2: Select the Anchor-Points

1. Click two different POINTS - the ANCHOR-POINTS - in two different PARTS.

   The distance between the two POINTS shows next to your mouse-pointer.
2. Click the graphic-area to place the dimension and the SPRING FB.

The SPRING FB is now in the graphic-area.

STEP 3: Open the Spring FB dialog 439

SPRING FB in the graphic-area:

SYMBOL ❶ FB - see SPRING FB CONNECTORS

SYMBOL ❷ SPRING between the ANCHOR-POINTS

❸ DIMENSION between the ANCHOR-POINTS

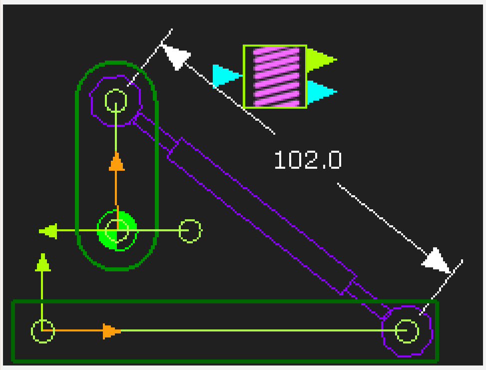
NOTE: Positive (Negative) force values Pull (Push) the Anchor-Points together (apart).

Spring FB States:

The SPRING FB can be in three different **states**.

STATE 1: The Spring FB is enabled.

The default state. The **SPRING FB** exerts a Force that is defined by the parameters in the **Spring FB dialog** 439.



Spring FB - Enabled

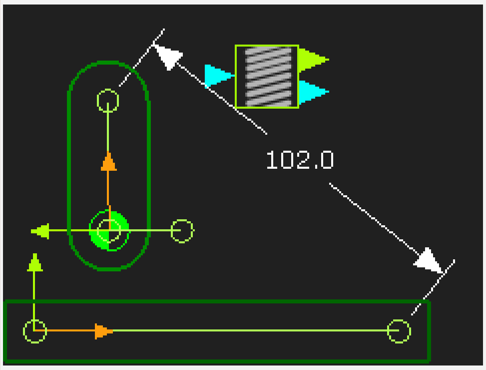
STATE 2: The Spring FB is not enabled.

To enable/disable the **SPRING FB**

1. Edit the **SPRING FB** to open the **SPRING FB DIALOG** 439

   In the **SPRING FB DIALOG**

2. Select / deselect the **ENABLE CHECK-BOX**.
3. Click ✅ to close the dialog.



Spring FB > Not enabled

STATE 3: The Spring FB is a Linear Motor

To change the **SPRING** to a **Linear-Motor**

1. Open the **Configure Power Source dialog** of the kinematic-chain for which the **SPRING FB** will be **Linear-Motor.**

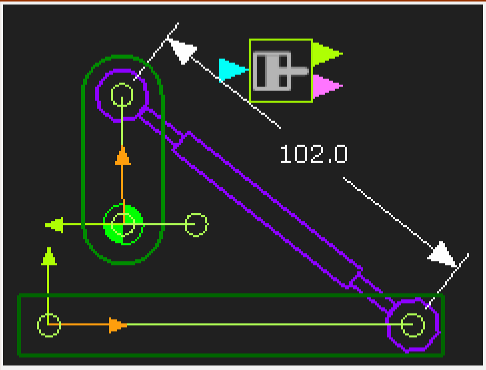
In the **Configure Power Source dialog**:

2. Click the **MOTION-DIMENSION FB** that applies the motion to the kinematic-chain
3. Click the **SPRING FB** as the **POWER SOURCE**,

The **SPRING FB** is now a **Linear Motor**.

4. Close the **CONFIGURE POWER SOURCE DIALOG**.

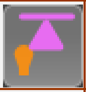
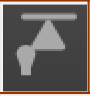
See - Configure Power Source 193



Spring FB > Linear Motor

1.4.6.5 Force-Vectors: Display

Forces toolbar > Force Vectors: Display

	Forces toolbar > Enable Force-Vectors: Display
	Forces toolbar > Disable Force-Vectors: Display
Menu :	Add menu > Forces sub-menu > Enable/Disable Force Vectors: Display

Force Vectors show ONLY when...

- Force toolbar > Force Vectors: Calculate [190] is enabled

AND

- Force toolbar > Force Vectors: Display is enabled

AND

- A minimum of one PART that is **kinematically-defined** has mass(kg) - see CAD-LINE DIALOG | MASS PROPERTIES [304].

AND/OR

- A SPRING FB [194] applies a Force between two POINTS and two PARTS that are kinematic-defined.

AND

- Display Filters toolbar > Display Part-Outlines is active

Force-Vectors in the graphic-area

Each **Force-Vector** is the **Force** that ACTS-ON a PART at a Joint, or other POINT, in a kinematic-chain.

Vector Location: as rays from POINTS at PIN-JOINTS, SLIDE-JOINTS, SPRINGS, CAM CONTACTS, GEAR CONTACTS, BELTS.

Vector Direction: they are in the direction of the **Force-Vector**

Vector Length: their length is scaled to the magnitude of the Force - **see Vector Scale buttons** [271]

Vector Magnitude: the Force that acts on the POINT is at the arrowhead of each vector

Vector Color: - **see below Colors: Part-Outlines and Force-Vectors** [199].

Rotary Driving Moment / Driving Torque

The **Torque-Vector** is the **Torque** to drive the kinematic-chain.

Location: at a PIN-JOINT that you select to drive the kinematic-chain - **see Configure Power Source** [193]

Length: proportional to the magnitude of the Torque.

IMPORTANT:

The **Torque Vector** does **NOT** include the torque to accelerate the inertia of a **Servo-motor** or **Gearbox**

To select and include a **Servo-motor** and **Gearbox** - **see Kinematic Servo-motor and Gearbox Sizing** [483].

Linear Driving-Force

The **Driving-Force** is the instantaneous **Force** to drive the kinematic-chain.

Location: at a SLIDE-JOINT that you select to drive the kinematic-chain **- see also: Configure Power Source** [193]**)**

Length: proportional to the magnitude of the Driving-Force.

IMPORTANT:

The **Driving-Force** does **NOT** include a **Linear Servo-motor** to drive the kinematic-chain.

There is **NOT** a database of **Linear Servo-motors.**

Colors: Part-Outlines and Force-Vectors

When you enable **Force-Vectors: Display**, the color of each **PART-OUTLINE** is changed.

The colors become the same color as the **Force-Vector** that ACTS-ON the PART, at a POINT in the PART.

TOP-TIP

To help identify which **Force-Vectors** that act on a PART:

1. Hover above a **PART-OUTLINE**

The **PART-OUTLINE** AND the **Force-Vector** that ACTS-ON the PART change to the selected color, usually RED.

Use **Configure Power Source** [193] to edit the color of each **Force-Vector**, and/or to hide **Force-Vectors** that act on different PARTS.

See also: Kinetostatic Torque and Speed dialog [483]

1.4.7 Solid elements

MD-SOLIDS is the term for **SOLIDS** that you add with **Add Profile/Auto-Profile**, and **CAD-SOLIDS** is the term for **SOLIDS** that you import onto a CAD-LINE.

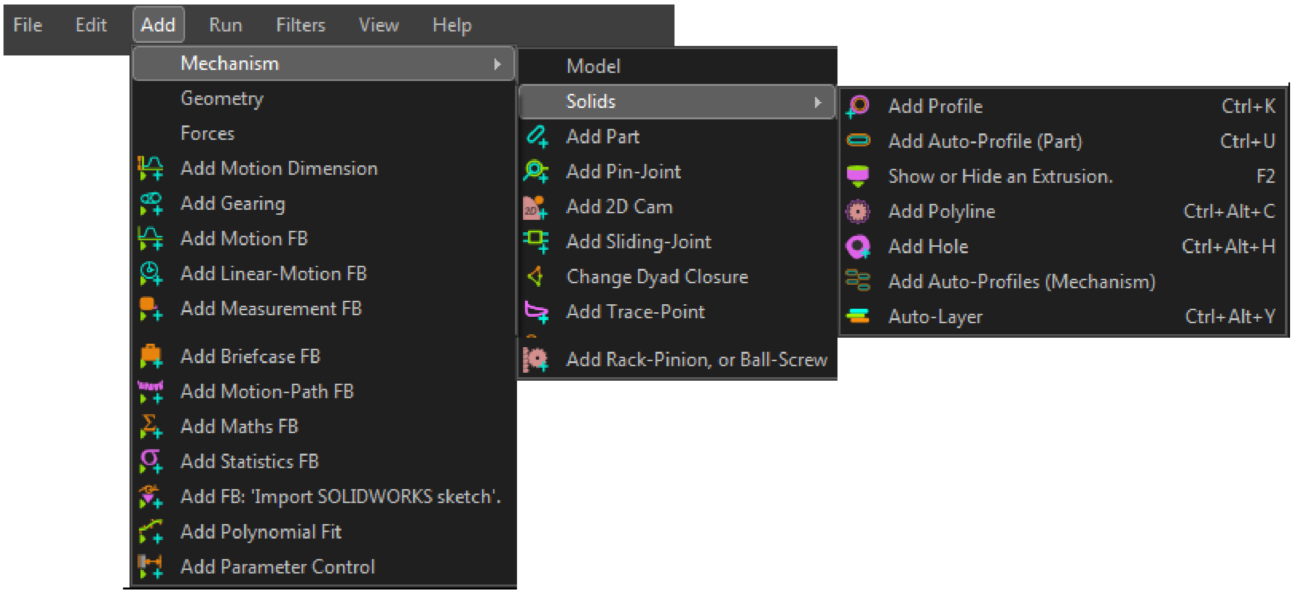
Solid elements

The commands in the **Solids menu** and **Solids toolbar** relate to **MD-SOLIDS**

To see **MD-SOLIDS** and **CAD-SOLIDS**, you must enable **Visibility toolbar > Show Solids in Mechanisms** [51]

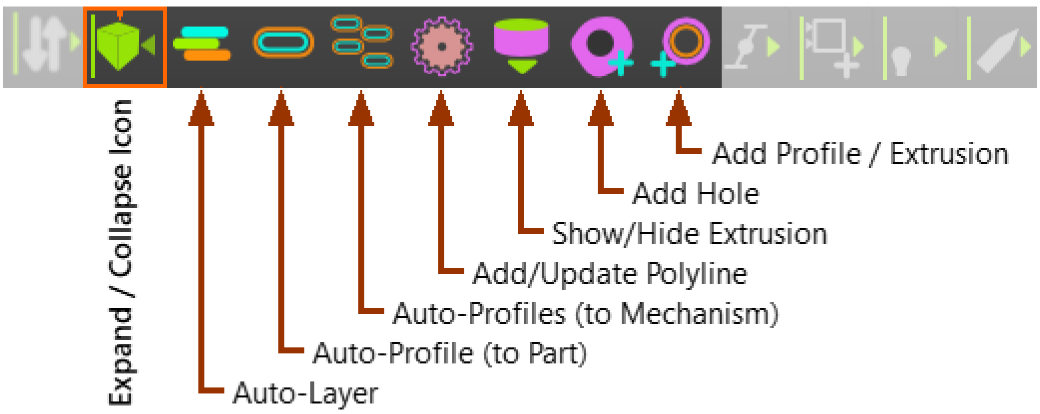
See also : Add CAD-Line [230]**, CAD-Line dialog** [301]

Solids menu



Solids toolbar

The **Solids toolbar** is **above** the graphic-area



Solids toolbar

1.4.7.1 Auto-Layer

What is 'Auto-Layer'?

Auto-Layer automatically adds an **EXTRUSION-OFFSET** to all **EXTRUSIONS**. It is intended that the extrusions do not collide when the mechanism moves.

Note:

Before you do **Auto-Layer**, you would usually:

- Do Solids toolbar > Auto-Profile (Mechanism) 204

To see **MD-SOLIDS** and also **CAD-SOLIDS**,

- Enable **Visibility toolbar** > Show Solids in Mechanisms 51 **to**

Auto-Layer

What to do : **STEP 1: Do the one command.**

1. **Add menu > Mechanism sub-menu > Solids sub-menu > Auto-Layer**

OR

1. **Solids toolbar > Auto-Layer**

Auto-Layer uses the **EXTRUSION-OFFSET** and **PART-OFFSET** parameters of each **EXTRUSION** to offset each **EXTRUSION** element by a different offset distance from the **MECHANISM-PLANE**.

You may wish to also edit the **EXTRUSION-OFFSET** and **PART-OFFSET** parameters of each **EXTRUSION** with the **EXTRUSION DIALOG**.

Video:

Double-click to watch Solids menu > Auto-Layer

1.4.7.2 Auto-Profile (Part)

What is Auto-Profile (Part)?

Do **Add Auto-Profile (Part)** to add a **SKETCH-LOOP** and an **AUTO-PROFILE / EXTRUSION** to one **PART**.

Do **Add Profile (Part)** again to the same **PART**, to remove the **SKETCH-LOOP** and **AUTO-PROFILE / EXTRUSION** from the **PART**.

An **AUTO-PROFILE** is an **MD-SOLID.**

See also

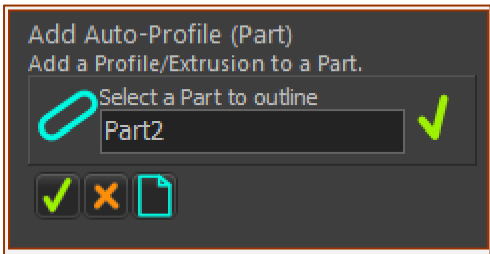
Auto-Profile (Mechanism) [204].

Notes:

Enable **Visibility toolbar** > Show Solids in Mechanisms [51] to see the **MD-SOLIDS** in a **MECHANISM-EDITORS**.

To edit an Extrusion: **SHIFT+CLICK** the **AUTO-PROFILE** to see the **AUTO-PROFILE** AND **EXTRUSION** elements in the **SELECTION-WINDOW**.

Auto-Profile (Part)

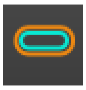


Command-Manager
Add Auto-Profile

To do **Add Auto-Profile:**

1. Click **Add menu** > **Mechanism menu** > Solids menu [199] > **Add Auto-Profile (Part)**

OR



1. Click **Solids toolbar** > **Add Auto-Profile (Part)**

The **COMMAND-MANAGER** has one selection-box.

In the graphic-area or **ASSEMBLY-TREE**:

2. Click a **PART-OUTLINE**
3. Click  in the **COMMAND-MANAGER**.

IF the **PARTS** does not have an **AUTO-PROFILE:**

Add a **SKETCH-LOOP** and **AUTO-PROFILE** / **EXTRUSION** to the **SKETCH-LOOP** ...

... to **ONE PART**

ELSE

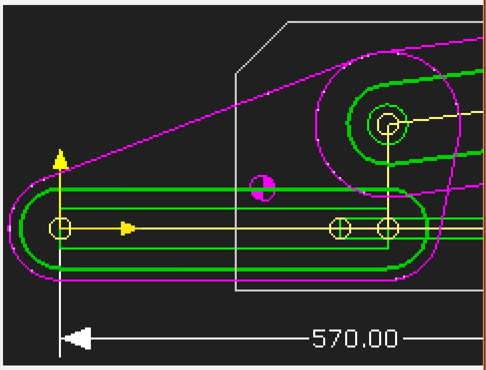
IF the **PART** has an **AUTO-PROFILE:**

Delete the AUTO-PROFILE and **SKETCH-LOOP** ...

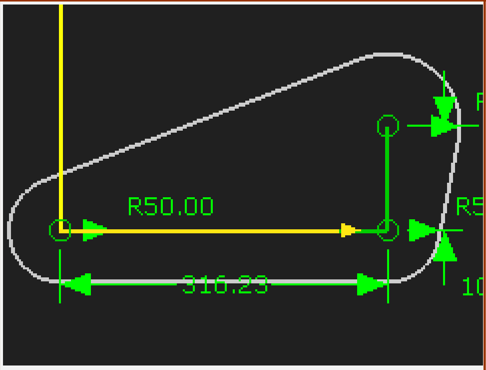
... from **ONE PART**

The shape of the **SKETCH-LOOP** is a function of the **JOINTS** in the **PART**.

<< The image shows the **AUTO-PROFILE** (**Pink**) around a **SLIDE-JOINT** and a **PIN-JOINT**.



Auto-Profile in Mechanism-Editor



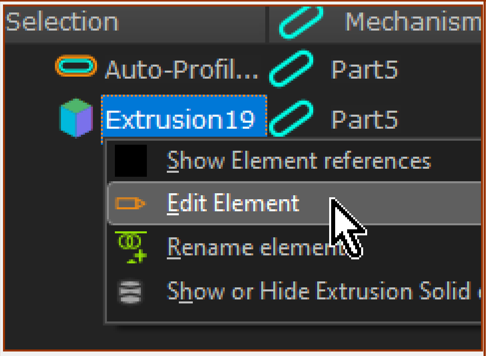
Auto-Profile in Part-Editor

Auto-Profile sketch-elements in the sketch-loop:

- you cannot select, edit, or delete them or their **constraints**
- you cannot add a joint to the LINES or POINTS.
- you can edit the **Radius** (dimension) to edit the size of the SKETCH-LOOP around PART'S POINTS

<< The image shows the SKETCH-LOOP around the POINTS in the PART that reference a SLIDE-JOINT and also a PIN-JOINT.

To edit an Extrusion - Special-Case



Edit an Extrusion:

1. SHIFT+CLICK the AUTO-PROFILE in the graphic-area

The AUTO-PROFILE and EXTRUSION show in the SELECTION-WINDOW

2. Right-click the EXTRUSION in the SELECTION-WINDOW

A shortcut menu shows next to your pointer.

3. Click **Edit element** from the shortcut menu

The EXTRUSION DIALOG opens.

See Extrusion dialog 298 for more information

1.4.7.3 Auto-Profiles (Mechanism)

Auto-Profiles (Mechanism)

Auto-Profiles (Mechanism) is a toggle command.

Enable Add Auto-Profiles (Mechanism):

- to ADD **sketch-loops*** and AUTO-PROFILES / EXTRUSIONS to ALL PARTS.

Disable Add Auto-Profiles (Mechanism):

- to DELETE the AUTO-PROFILES/EXTRUSIONS and **sketch-loops*** from ALL PARTS.

The command applies to PARTS in the active MECHANISM-EDITOR.

---

* The shape of each **sketch-loop*** has:

- at a PIN-JOINT - an ARC at the POINTS selected to add the PIN-JOINT
- at a SLIDE-JOINT - an ARC at each end of the LINE selected to add the SLIDE-JOINT
- ARCS at the START-POINT and END-POINT of the CAD-LINE when the PART is not **kinematically-defined**.

and:

- LINES between the ARCS

You cannot use the LINES or the ARCS for kinematic-elements. E.g. PIN-JOINTS, SLIDE-JOINTS, ...

Tips::

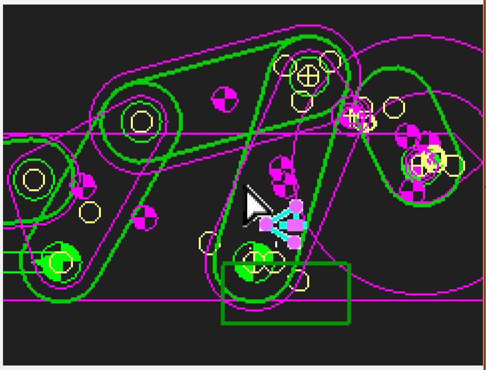
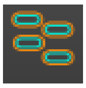
To see EXTRUSIONS (MD-SOLIDS) in MECHANISM-EDITORS.

- Enable Visibility toolbar > Show Solids in Mechanisms [51]

To edit an EXTRUSION you cannot see:

- SHIFT+CLICK the AUTO-PROFILE in the graphic-area, edit the EXTRUSION from the SELECTION-WINDOW.

Add Auto-Profiles (Mechanism)

What to do :  Parts with Auto-Profiles.	To Add Auto-Profiles or To Delete Auto-Profiles. 1. Click Add menu > Mechanism menu > Solids menu [199] > Add Auto-Profile (Mechanism) **OR**  1. Click Solids Toolbar [199] > Add Auto-Profiles (Mechanism) Then: 2. Click the graphic-area
	RESULT: This command toggles: The Rule to add or delete the AUTO-PROFILES is: **IF ≤ 50%** of the PARTS have AUTO-PROFILES: Add SKETCH-LOOPS and AUTO-PROFILES / EXTRUSIONS ... ... to ALL PARTS in ALL kinematic-chains in the active MECHANISM-EDITOR.

	The colors of the **AUTO-PROFILES** are different from each other. **ELSE** **IF** > **50%** of the **PARTS** have **AUTO-PROFILES:** **Delete** **AUTO-PROFILES** and **SKETCH-LOOPS** ... ... from **ALL** **PARTS** in **ALL** kinematic-chains in the active **MECHANISM-EDITOR**.

Auto-Profile sketch-elements:

- you **cannot** edit or delete the **SKETCH-ELEMENTS** or their **CONSTRAINTS**
- you **cannot** add Joints to their **LINES** or **POINTS**
- you **can** edit the **Radius** (dimension) at each Joint.

Note: we do not add for you **TANGENT** constraints between the **ARCS** and **LINES** of each **SKETCH-LOOP**.

Video : Double-click for a Video

1.4.7.4 Add/Update Polyline

Why add a Polyline?

A **POLYLINE** is a sketch-element.

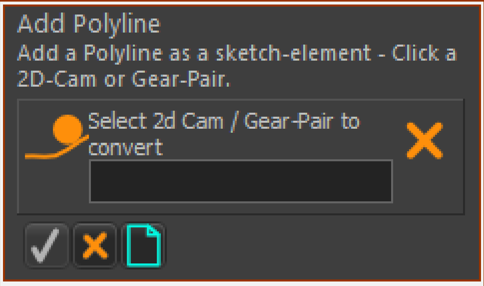
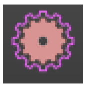
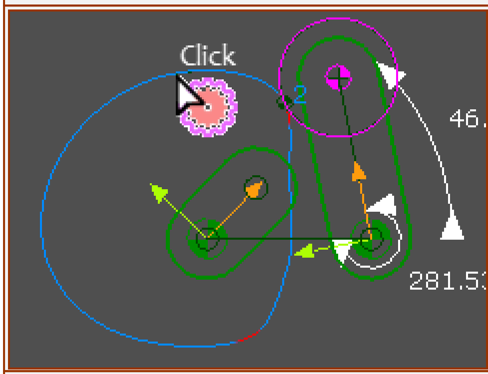
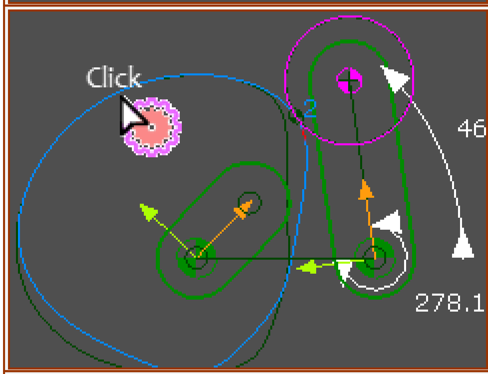
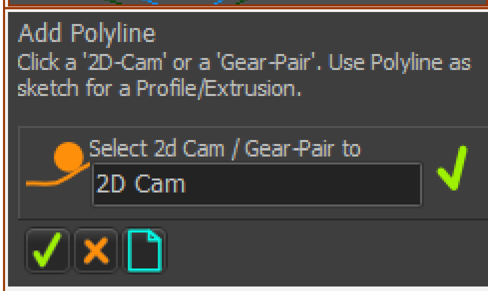
Use **Add Polyline** to:

- Add a **POLYLINE** to a **GEAR-PAIR** or **2D-CAM**.
- Update the shape of a **POLYLINE if** you edit a **GEAR-PAIR** or **2D-CAM**

You can add one **POLYLINE** to each **2D-CAM** and each **GEAR-PAIR**.

You can select one **POLYLINE** to add many **PROFILES**.

Add Polyline & Update Polyline

	STEP 1: Put the MMA to Zero degrees 1. **Press ALT+H or the Home key on your keyboard**
	**STEP 2: Start the Add Polyline command:** 2. Click **Add menu** > **Mechanism menu** > [Solids menu](199) > **Add Polyline**  **OR** 2. Click [Solids Toolbar](199) > **Add Polyline** The **COMMAND-MANAGER** has one selection-box:
	**STEP 2: To ADD a Polyline** 1. Click a **2D-CAM** or **GEAR-PAIR** in the graphic-area
	**STEP 2: To UPDATE a Polyline** 1. Click a **2D-CAM** or **GEAR-PAIR** with has a **POLYLINE** that is not the correct copy of the **2D-CAM or GEAR-PAIR**. <<< image shows the **2D-CAM** (in Blue) and the out-of-date **POLYLINE** in Dark-Green **before** Update Polyline.
	**STEP 3: End the command** 1. Click ✓ in the **COMMAND-MANAGER** **If necessary:** 2. **Click [Rebuild Now](37) to make sure the Polyline is correct.**

	The **POLYLINE** is now a **SKETCH-ELEMENT** in the **PART** that is equal to the shape or size of the **2D-CAM** or **GEAR-PAIR**.

1.4.7.5 Show/Hide Extrusion

What does this tool do?

Use this toggle tool to switch the visibility of an **EXTRUSION.**

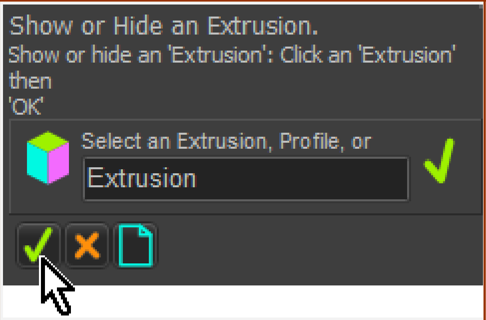
Before you use this tool, to see if the Extrusion is showing or hidden.

- Use Visibility toolbar [48] > Show Solids in Mechanisms [51]

Show-Hide Extrusion

Solids toolbar [199] > Show-Hide Extrusion

Add menu > Mechanism sub-menu > Solids sub-menu > Show-Hide Extrusion



You must pre-select the **EXTRUSION** to use this command.

If you **cannot** see the **EXTRUSION**, (and only the **PROFILE**):

1. **SHIFT-CLICK** a **PROFILE** in the graphic-area
2. Click Solids toolbar [199] > **Show-Hide Extrusion**

The **COMMAND-MANAGER** has one selection-box - the **EXTRUSION** is selected.

Click ✓ in the **COMMAND-MANAGER**

Now, you can see the **EXTRUSION**

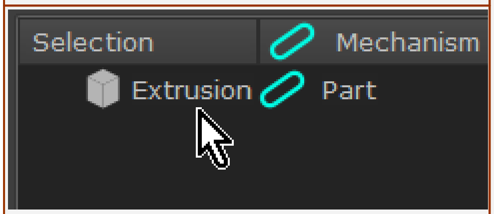
If you **can** see the **EXTRUSION**

1. **CLICK** the **EXTRUSION** in the graphic-area
2. Click Solids toolbar [199] > **Show-Hide Extrusion**

The **COMMAND-MANAGER** has one selection-box - the **EXTRUSION** is selected.

Click ✓ in the **COMMAND-MANAGER**

Now, you cannot see the **EXTRUSION**



When you hide the **EXTRUSION** the color of the **EXTRUSION** element-icon in the **ASSEMBLY-TREE** and **SELECTION-WINDOW** is **gray.**

Note: Other methods to Switch the Visibility of an Extrusion:

- EXTRUSION DIALOG [300]

Also:

a. Shift+Click a **PROFILE** in the Graphic-Area.

The **EXTRUSION** shows in the **SELECTION-WINDOW**

b. **RIGHT-CLICK THE EXTRUSION**

In the shortcut menu

c. Select **Show or Hide Solid Elements**

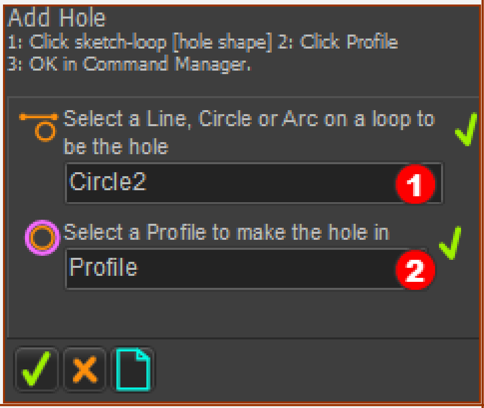
1.4.7.6 Add Hole

What is a Hole?

A **HOLE** cuts the shape of a **SKETCH-LOOP** through a **PROFILE / EXTRUSION** (MD-Solid) that is in the model.

Note: To see the **HOLE** through the **PROFILE / EXTRUSION**, enable **Visibility toolbar** > Show Solids in Mechanisms 51

Add Hole



STEP 1: Start the command Add Hole:

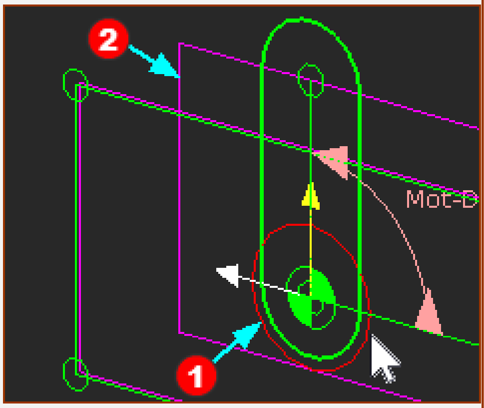
1. Click **Add menu** > **Mechanism menu** > **Solids menu** > **Add Hole**

OR

1. Click **Solids toolbar** > **Add Hole**

The **COMMAND-MANAGER** starts.

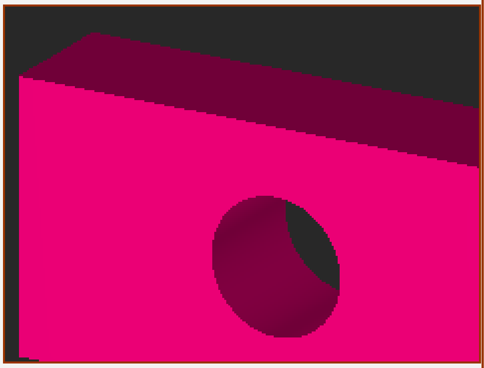
There is one selection-box.



STEP 2: Select the two elements for the Command-Manager

1. Click a **SKETCH-LOOP** ❶, **all** of which is inside the bounds of a **PROFILE** ❷
2. Click the **PROFILE** ❷ through which you want to cut the **HOLE**

The **SKETCH-LOOP** and the **PROFILE** are now in the selection-boxes in the **COMMAND-MANAGER**



STEP 3: Complete the Command

1. Click ✓ in the **COMMAND-MANAGER**

RESULT:

Assembly-Tree: the **HOLE** element is a child to the **PROFILE** element.

Graphic-Area: the **HOLE** cuts through the **PROFILE / EXTRUSION**.

<<< image, in the **MODEL-EDITOR**, of the **HOLE** that is cut through the **MD-SOLID**.

See also: **Sketch-Loop, Profile, Command-Manager**

Video:

Double-click to watch 'Add Hole'

1.4.7.7 Add Profile / Extrusion

What is a Profile / Extrusion?

A **PROFILE/EXTRUSION** is one type of **SOLID**.

Use **Add Profile** to add, to a **SKETCH-LOOP**, a **PROFILE** and an **EXTRUSION** as a child to the **PROFILE**.

You can add many **PROFILES** to the same **SKETCH-LOOP**.

Notes:

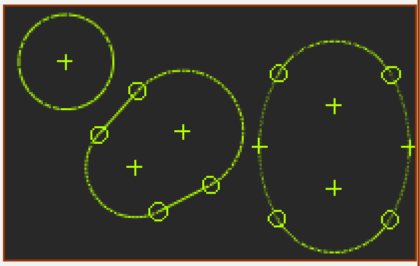
The **Cam-Follower** you select to do **Add 2D-Cam / 3D-Cam** must be a **PROFILE**. **See also Shape of the Cam-Follower.**

See: How to EDIT an EXTRUSION ; How to DELETE a PROFILE (212)

Terminology:

Sketch-Loop :	sketch-elements with **START-POINTS** and **END-POINTS** that you merge to form a closed loop. The sketch-elements cannot cross over each other.
PROFILE :	two offset contours that are a copy of the **sketch-loop** you select when you do **Add Profile**
EXTRUSION :	the **SOLID** between the two offset contours.

Preparation for Add Profile / Extrusion



3x Sketch-Loops in a Part

The elements in the model before you can do **Add Profile:**

A **SKETCH-LOOP** is in a **PART.**

There are three different sketch-loops in the image to the left.

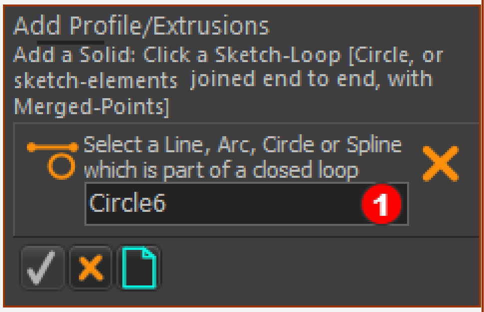
Rules of Sketch-Loops:

The **START-POINT** and **END-POINT** of a sketch-element are **MERGED** with those of a different sketch-element.

The sketch-elements are in a continuous loop.

The sketch-elements **CANNOT** cross each other.

Add Profile / Extrusion



STEP 1: Start the Add Profile command

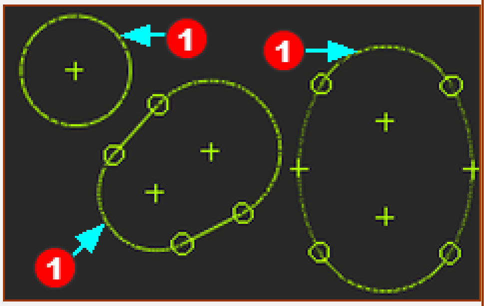
1. Click **Add menu > Mechanism menu > Solids sub-menu > Add Profile**

OR

1. Click **Solids toolbar > Add Profile**

The **COMMAND-MANAGER** starts.

There is one selection-box.



3x Sketch-Loops in a Part

STEP 2: Select a sketch-element

1. Click a **SKETCH-ELEMENT** 1 that is in a **sketch-loop**, or a **POLYLINE**

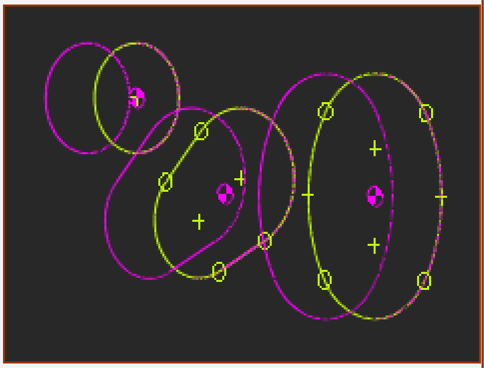
The name of the sketch-element is in the **COMMAND-MANAGER**.

STEP 3: Complete the command:

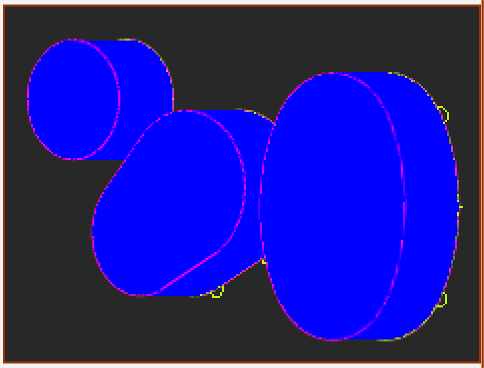
3. Click ✓ in the **COMMAND-MANAGER.**

The **PROFILE** and the **EXTRUSION** are now in the model.

Do 1 - 3 again to add another **PROFILE**.



3 x Profile Contours in graphic-area



3 x Extrusions in graphic-area

RESULT:

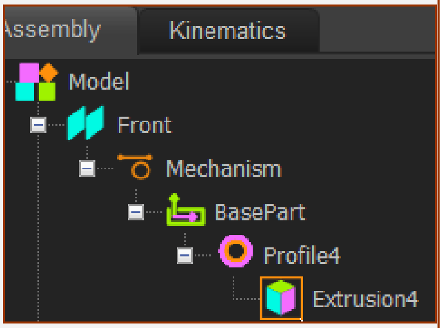
Graphic-Area: To see the **EXTRUSION**, enable **Visibility toolbar** 48 > **Show Solids in Mechanisms** 51

The images show 3 × **PROFILES**

Each **PROFILE** has two contours, which are offset from each other.

Each **EXTRUSION** is an **MD-SOLID** that shows between the **PROFILE** contours.

The default color of the **EXTRUSION** is **Blue.**



Assembly-Tree:

The **PROFILE** is a child to the **PART** from which you selected the **sketch-loop**. An **EXTRUSION** is a child to the **PROFILE**.

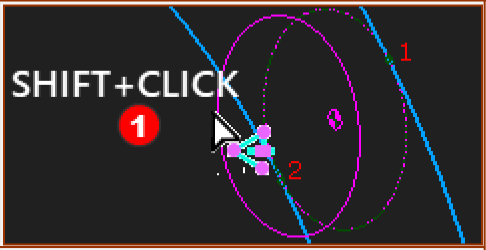
See also :

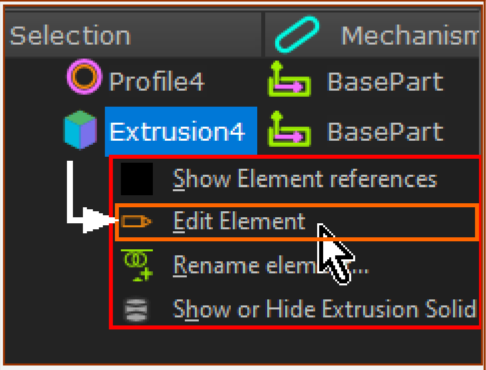
Tutorial 4: Add Profile **(external link)**

Add Auto-Profile 202, **Add Auto-Profiles** 204, **Polyline Sketch-Loop**

Video Add Profile:Double-click to watch the Video

How to edit an Extrusion





SELECTION-WINDOW: Right-Click

To **EDIT** an **EXTRUSION ELEMENT** - with the **SELECTION-WINDOW**.

In the Graphic-Area

1. **SHIFT-CLICK** the **PROFILE**

The **PROFILE** and **EXTRUSION** show in the **SELECTION-WINDOW**.

In the **SELECTION-WINDOW**:

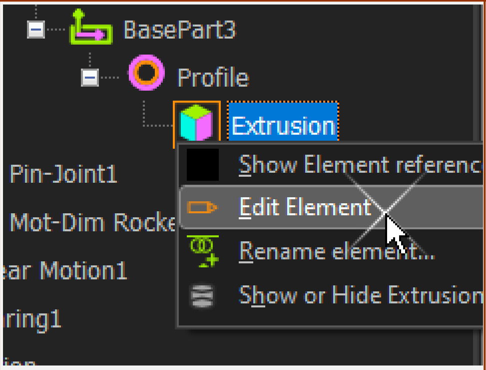
2. Right-click the **EXTRUSION**

In the shortcut menu

3. Click **Edit element**

The **EXTRUSION DIALOG** opens.

See Extrusion dialog 298



ASSEMBLY-TREE - Right-click Extrusion

OR

EDIT EXTRUSION ELEMENT - with the **ASSEMBLY-TREE**

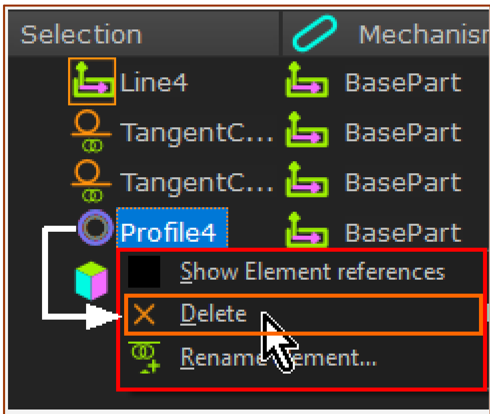
In the **ASSEMBLY-TREE**

1. Click the **PROFILE** to expand and see the **EXTRUSION** element - a child to the **PROFILE** element
2. Click the **EXTRUSION** to make sure it has a square **ORANGE** box around the **EXTRUSION** icon
3. Right-click the **EXTRUSION**

In the shortcut menu

4. Click **Edit element**.

How to delete a Profile



ASSEMBLY-TREE - Delete Profile

To **DELETE** a **PROFILE ELEMENT**

In the Graphic-Area

1. **CLICK** the **PROFILE**

The **PROFILE** shows in the **SELECTION-WINDOW**.

In the **SELECTION-WINDOW**:

2. Right-click the **PROFILE**

In the shortcut menu

3. Click **Delete**

The **PROFILE and** the **EXTRUSION** elements are deleted from the model.

1.5 2.3 Part-Editor:

❷ Part-Editor

See: **Why edit a Part?** [547]

After you add a **PART**, use the **PART-EDITOR** to:

- edit the length of a PART
- add sketch-elements

Then:

- add dimensions and constraints to, or between, sketch-elements
  You can only dimensions and add constraints AFTER you add the sketch-elements.

You can edit one **PART** at a time. To edit a different **PART**:

1. Close the Part-Editor [217],

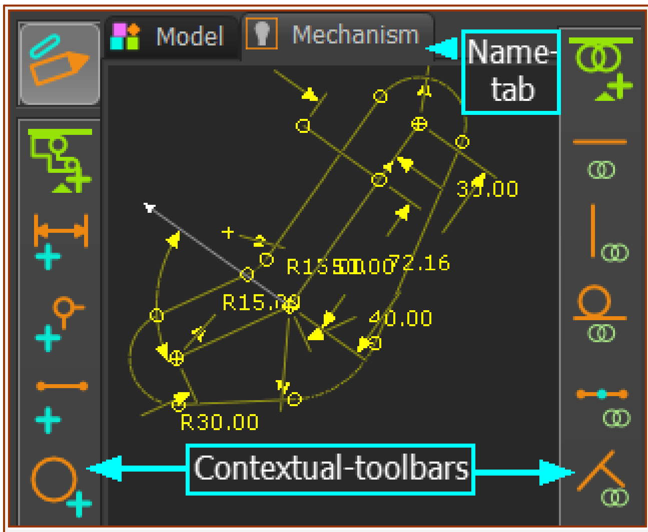
In the MECHANISM-EDITOR, select a different PART

2. Start the Part-Editor [214] again.

Video: Five methods to start the Part-Editor, and one method to close the Part-Editor

Double-click to watch

Part-Editor workspace



Part-Editor: Example

Graphic-Area : shows sketch-elements, dimensions, Part-Axes.

Toolbars : are to the left and right of the graphic-area.

Name-tabs : are above the graphic-area. The name of the tab is the name of the MECHANISM-EDITOR to which the PART is a child.

Menus and Toolbars:

Geometry menu and toolbar [219] to add sketch-elements.

Constraints menu and toolbar [244] to add constraints to, or between, sketch-elements.

1.5.1 How to start (open) the Part-Editor

How to start (open) the Part Editor

See also: How to close the Part-Editor217
See also: I cannot select the Part-Outline. What to do?533

Notes:

The **PART-EDITOR** can edit ***one*** PART at a time.

To edit a different PART, you must close the **PART-EDITOR** to return to the **MECHANISM-EDITOR**, and then edit the different PART.

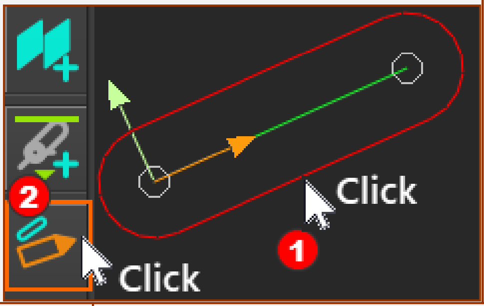
Video: **How to start the Part-Editor:**

Double-click to watch

How to start the Part-Editor - more details

To start (open) the **PART-EDITOR**, do one these methods.

METHOD 1: Edit-Part icon



STEP 1: Click a **PART-OUTLINE** ❶ in the graphic-area

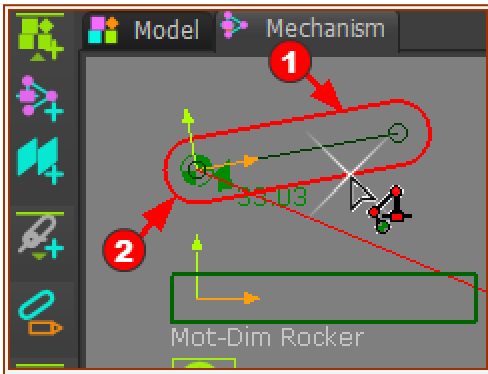
STEP 2: Click **Edit toolbar** > **the Edit Part in Part-Editor** ❷

The **PART-EDITOR** is now open.

Note: The **Edit Part** icon is to the left of the graphic-area.

See also: How to close the Part-Editor: Method 1217

METHOD 2: Selection-Window



STEP 1: Click a Part-Outline

STEP 1: **Select the Part**

Graphic-area:

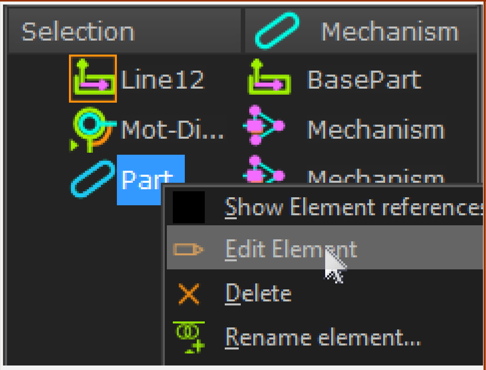
1. Click the **PART-OUTLINE** ❶ (or ❷ with some graphic-cards)

OR

Assembly-Tree:

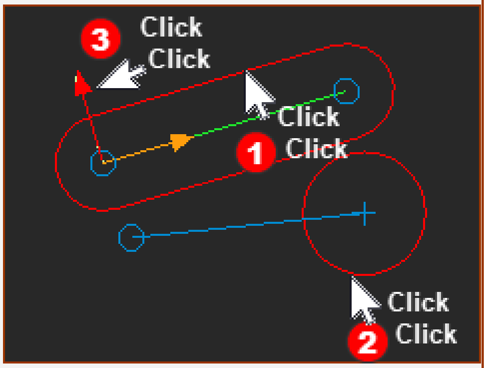
1. Click the PART element

STEP 2: **In the Selection-Window**

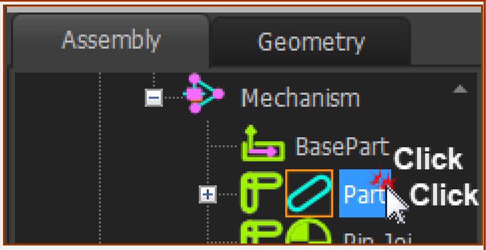


STEP 2: Right-click the Part in the Selection-Window, Click Edit element

METHOD 3: Double-Click



Double-click a Part-Outline in the graphic-area



Double-click a Part in the Assembly-Tree

STEP 1: Double-click ...

... the Part-Outline in the Graphic-area:

1. Double-click the PART-OUTLINE ❶

OR

1. Double-click a LINE*, ARC*, or CIRCLE* SKETCH-ELEMENT ❷ that you have added to the PART

OR

1. Double-click the Y-AXIS ❸ of the PART

The PART is now open in the PART-EDITOR.

OR

... the Part element in the Assembly-Tree:

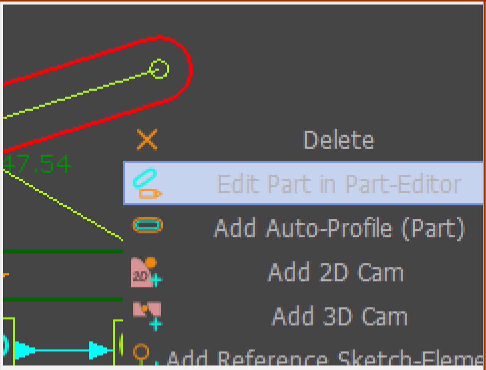
1. Double-click the PART element

The PART is now open in the PART-EDITOR.

*** If you double-click a:**

CAD-LINE	... the CAD-LINE DIALOG (301) opens
BLEND-CURVE	... the BLEND-CURVE DIALOG (449) opens
POINT	... the POINT PROPERTIES DIALOG (444) opens
More than one element	... nothing happens.

METHOD 4: Right-Click



Hover+ Right-click a Part-Outline in the graphic-area

In the graphic-area:

1. Hover so the **PART-OUTLINE** becomes ***red***

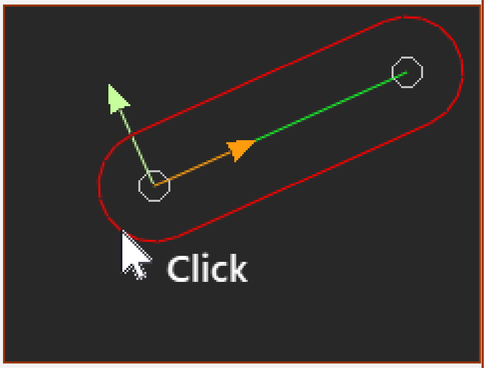
Note: With some graphic-cards, you must move your mouse-pointer to the arc of the Part-Outline near to the Origin❷.

2. Right-Click the **PART-OUTLINE**
3. Click **Edit Part Geometry** from the shortcut menu.

The **PART** is now open in the **PART-EDITOR**.

Occasionally, there is a problem with a Graphic-Card

With some graphics cards, you cannot select the **PART-OUTLINE** to edit a **PART**. Try these methods:



Select the Part-Outline near start-Point.

Method 1: Graphic-area

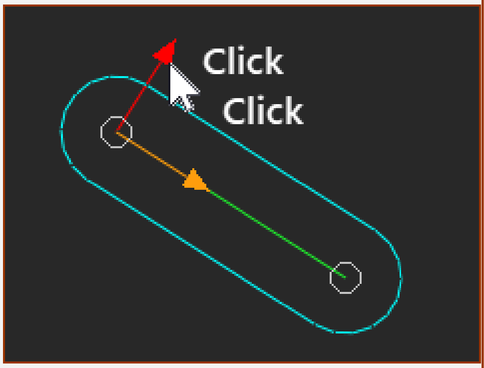
1. Move your mouse-pointer above the ***arc*** of the **PART-OUTLINE**
2. Double-click your mouse

Method 2: Selection-Window

1. Move your pointer above the ***arc*** of the **PART-OUTLINE** - see image
2. Click your mouse one time.

Then, in the **SELECTION-WINDOW**:

3. Right-Click the **PART**
4. Click **Edit element**

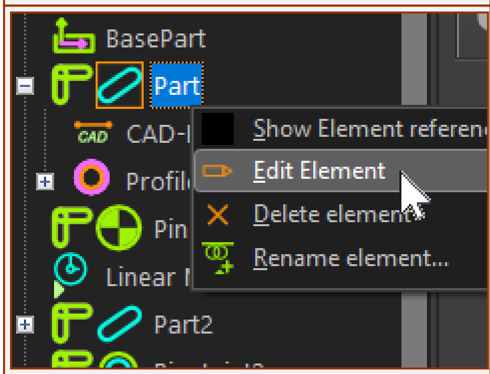


OR Double-click the Y-axis

Method 3:

OR

1. Double-click the **Y-axis** to edit a **PART**,



Select a Part in the Assembly-Tree

Method 4:

OR

In the **ASSEMBLY-TREE**

1. Right-Click the **PART**
2. Select **Edit element** in the shortcut menu.

1.5.2 How to exit (close) the Part-Editor

How to exit (close) the Part-Editor

See also : How to Start the Part-Editor [214]

When you close the **PART-EDITOR**, you return to the **MECHANISM-EDITOR.**

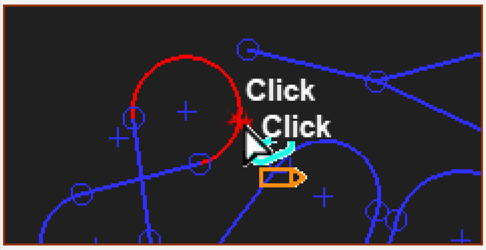
To close the **PART-EDITOR**, do one of these methods.

METHOD 1: De-select Edit-Part in Part-Editor icon

1. Click the **Geometry toolbar** > **Edit Part** icon, at the **left** of the graphic-area.

METHOD 2: Double-click a sketch-element

1. Double-click a **LINE**, **CIRCLE** or **ARC** sketch-element.



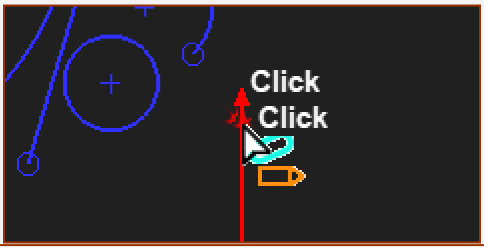
Note: If you double-click a...

CAD-Line	...the CAD-Line dialog [301] opens
Blend-Curve	...the Blend-Curve dialog [449] opens
Point	...the Point Properties dialog [444] opens
...more than one element, nothing happens.	

METHOD 3: Double-click the Y-axis

1. Double-click the **Y-axis**.

Note: We recommend the **Y-axis** because the **X-axis** is collinear with the **CAD-LINE.**

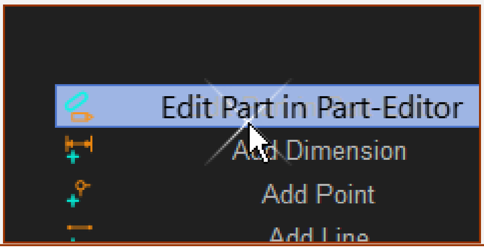


METHOD 4: Right-click

1. Right-click the graphic-area

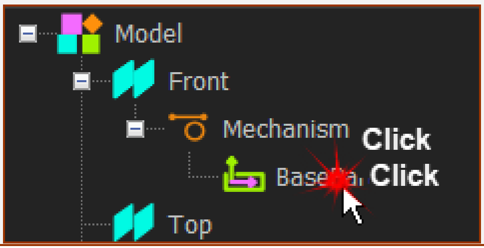
The shortcut menu shows. The **Edit-Part in Part-Editor** icon is active.

2. Click the **Edit Part in Part-Editor** command to de-select it



METHOD 5: Double-click element in Assembly-Tree

1. Double-click the **PART** in the **ASSEMBLY-TREE**.



1.5.3 Geometry

Part-Editor - Geometry menu and toolbar

In the **Part-Editor**, use the **Geometry menu and toolbar**

- to edit the length of an Added-Part
- to add sketch-elements to a **PART**
- to add dimensions to the sketch-element that you add to a **PART**
- to import a SOLIDWORKS sketch onto a **PART**
- to use the **MERGE-POINTS** tool to combine two **POINTS** to one **POINT**

NOTE:

You **cannot** add constraints **AS** you add **SKETCH-ELEMENTS** (**Merge-Points** is the exception).

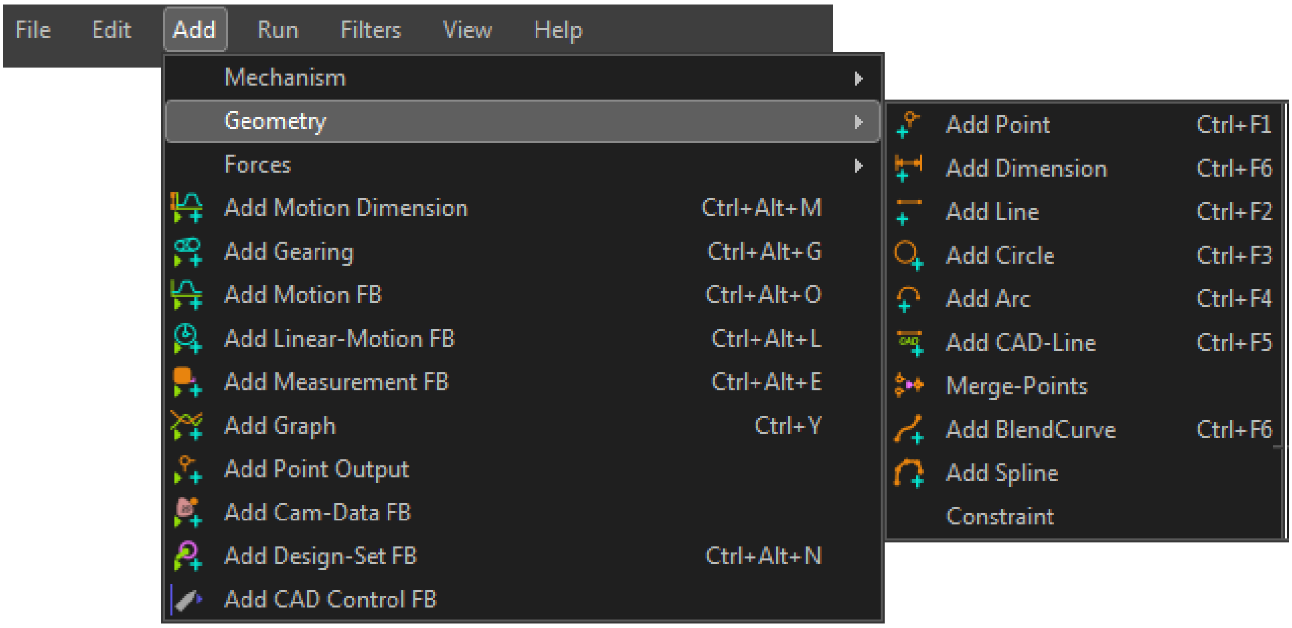
You **must** add constraints **AFTER** you add the **SKETCH-ELEMENTS**.

TOP-TIP:

Add **SKETCH-ELEMENTS** in ways that are clearly not constrained. E.G Add a **LINE** that is obviously not Horizontal. Then add the Horizontal Constraint.

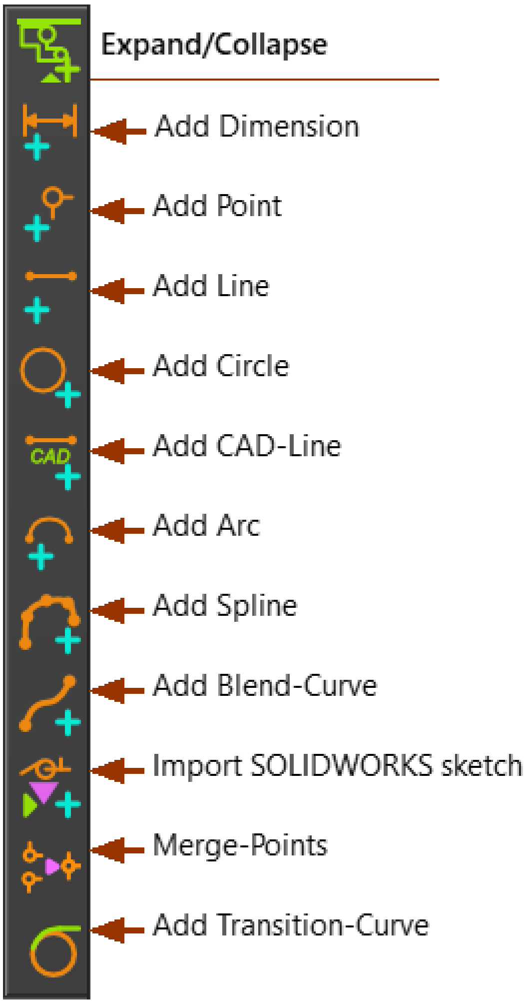
See also: Constraints menu and toolbar [244]

Part-Editor: Geometry menu



Part-Editor: Geometry toolbar

Part-Editor : Geometry toolbar is to the **left** of the graphic-area.



Geometry toolbar
[Click text to see more...]

SKETCH-ELEMENTS and the BASE-PART

Before you can add a joint between the BASE-PART and a PART, you must edit the BASE-PART to add to it a minimum of one(1) sketch-element. You can, of course, add any number of sketch-elements to the BASE-PART.

The first sketch-element in the BASE-PART?

We recommend that you add a LINE as the first sketch-element to the BASE-PART.

Why?

A LINE gives you flexibility when you need to join other PARTS to the BASE-PART.

You can select the LINE'S START-POINT and/or END-POINT to add PIN-JOINTS, and the LINE to add SLIDE-JOINTS, between the BASE-PART and a different PART.

You can also reference the LINE and/or the START-POINT or END-POINT to add a MOTION-DIMENSION FB, and so define the motion of a **Rocker** and/or **Slider** relative to the BASE-PART.

ABOUT GEOMETRY

Point names

- POINT - the SKETCH-ELEMENT you add to the PART in the PART-EDITOR with **Add Point**.
- START-POINT - the POINT where you **mouse-button down** at the start of the Drag to add a LINE, CAD-LINE, ARC, or BLEND-CURVE.

- **END-POINT** - the **POINT** where you **mouse-button up** at the end of the Drag to add a **LINE**, **CAD-LINE**, **ARC**, or **BLEND-CURVE**.
- **CENTER-POINT** - the **POINT** where you **mouse-button down** at the start of the Drag to add a **CIRCLE**.
- **CENTER-POINT** - the **POINT** at the center of an **ARC** after you **mouse-button up** at the end of the Drag to add an **ARC**.

Edit a Dimension

To edit a dimension:

1. If necessary, **de-select** **Add Dimension** and all commands.
2. Double-click the **arrowhead** of a dimension or extension line (do **not** double-click the dimension number).

Sketch-Elements in the Part-Editor:

- **Blue:** sketch-elements are not Fully defined
- **Black:** sketch-elements are Fully defined

Sketch-Elements in the Mechanism-Editor:

- **Green:** **PART** is **kinematically-defined** / solved - **PART-OUTLINE** is also **Green**.
- **Blue:** **PART** is not **kinematically-defined** / solved - **PART-OUTLINE** is also **Blue**.

Note: The default colors may have changed. See:

Application-Settings > Graphics > **DISPLAY COLORS** > Geometry Defined / Under-Defined

Application-Settings > Graphics > **DISPLAY COLORS** > Part Solved / Part Unsolved

---

Trouble-shoot - Geometry

- **CTRL+Z** should return the geometry to the previous state.,
- If a sketch-element is **Black** but it should be **Blue:** delete the sketch-element, and add the sketch-element again

Trouble-shoot - Constraints

To delete a Constraint that you suspect is a problem:

1. **SHIFT+CLICK** a **sketch-element** or **POINT** with a **Constraint**

The **sketch-element**, and/or **POINT**, and all **Constraints** that you have added show in the **SELECTION-WINDOW**

2. **Delete** the **Constraint** from the **SELECTION-WINDOW**.

Trouble-shoot - Dimensions

- **Delete** an angle dimension if it does not increase to be greater than 180º. Move the sketch-element, and add the dimension again.
- **Delete** a dimension if it does not pass through 0, (angular or linear dimension), and add the dimension again.

Fix Constraint

To **FIX** a **POINT**, **START-POINT**, **END-POINT**, or **CENTER-POINT**: use the **LOCK & SPECIFY** check-box in the Point Properties dialog 444.

However, you **cannot** **FIX** a **POINT** with the **LOCK AND SPECIFY** check-box if the **POINT** is constrained with a Constraint or a Dimension.

1.5.3.1 Add Dimension

Add Dimension

Use Add Dimension to add Linear and Angular Dimensions to sketch-elements.

These include the:

- **Distance** between **POINTS***
- **Length** of a **LINE**** ; **Radius** of a **CIRCLE** or **ARC**.
- **Perpendicular (shortest) Distance** from a **POINT*** to a **LINE****, **X-AXIS**, or **Y-AXIS**.
- **Angle** between two **LINES**, or the **Angle** from the **X-AXIS** or **Y-AXIS** to a **LINE****
- **Angle** between three **POINTS***

---

* **POINT**, **START-POINT**, **END-POINT**, and **CENTER-POINT**.

** **LINE** or **CAD-LINE**

Notes:

Usually, do not make a dimension zero. It is better to add a constraint. **See Sketch-Constraints** 244

See How to add dimensions 223**, How to edit a dimension** 222**.**
See also: Number-Format: Precision and Digits 286**, Dimension Font Size** 286

Add Dimension

To Add a Dimension:

1. Click **Geometry toolbar** > **Add Dimension**

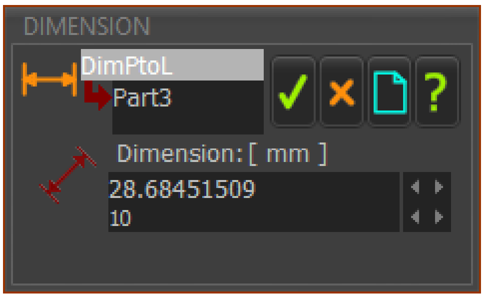
OR

1. Click **Add menu** > **Geometry sub-menu>** > **Add Dimension**

2. Click the sketch-element, or sketch-elements, to add a linear or angular dimension - **see below How to Add a Dimension** 223

   The **DIMENSION** shows near to your mouse-pointer.

3. Click in the graphic-area to place the dimension-line and extension lines next to your mouse-pointer.



The **DIMENSION DIALOG** opens.

4. Edit the **DIMENSION** in the **DIALOG**

> **Negative Dimensions?**
> **Do not delete the negative sign from a negative dimension.**
> **See: Why a dimension is negative** 551**.**

5. Click ✓ to close the **DIMENSION DIALOG**

The **DIMENSION** is now in the graphic-area.

How to EDIT a dimension:

1. De-select all other commands
2. Double-click the arrowhead of a **dimension-line**

Why the arrowhead?

- It is easier to click the **arrowhead** than click the **dimension-line** or **extension-line**.

AND

- Nothing happens if you click the **dimension-number**

More details: How to ADD a dimension

* POINT, START-POINT, END-POINT, and CENTER-POINT.

** LINE or CAD-LINE

Dimensions to locate a Point

There are two methods to locate a POINT.

METHOD 1: Add Dimensions

Examples	Dimension	How?	Selection-Window & Geometry-Tree
	• Minimum distance between two POINTS*	• Click each POINT*	DimPtoP*n*
	• Perpendicular distance between a POINT* and a LINE**, X–AXIS, or Y–AXIS	• Click the POINT* then the LINE**	DimPtoL*n*

METHOD 2: Use the Point Properties dialog

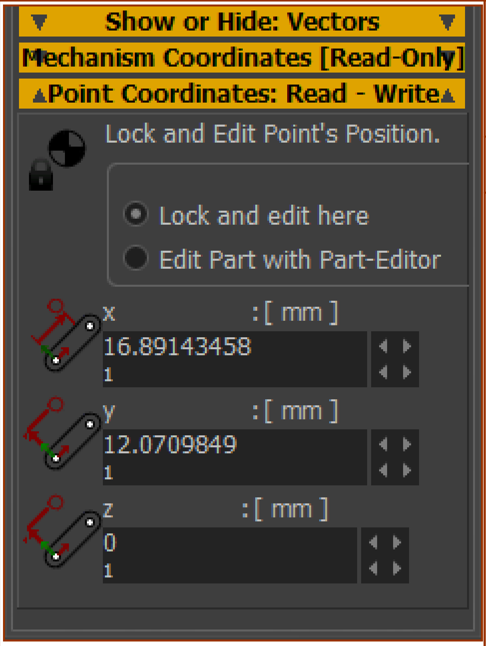
This method is possible:

- ONLY with a POINT sketch-element

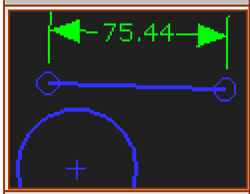
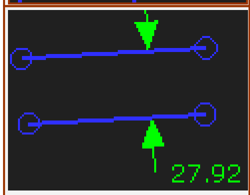
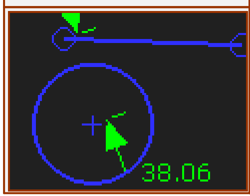
  It is not possible with a START-POINT, CENTRE-POINT, CENTER-POINT

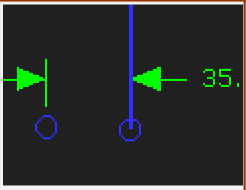
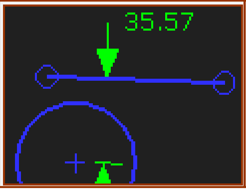
AND

- NOT possible if the POINT has a dimension or constraint applied to it.

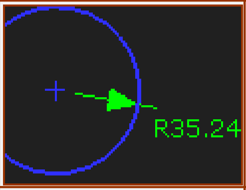
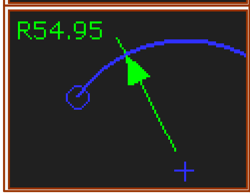
Example	Dimension	Selection-Window / Geometry-Tree
 Point Properties dialog	• LOCK AND SPECIFY the exact location of a POINT* in a PART. How? 1. Double-Click a POINT* in the PART-EDITOR OR 1. Double-Click a POINT* in the MECHANISM-EDITOR The **Point Properties dialog** 444 opens - see left Expand POINT COORDINATES 2. Select the LOCK AND SPECIFY check-box. 3. Enter 'x' and 'y' values for the POINT*, with **Part Coordinates** * in this case, ONLY a POINT sketch-element.	Pointn

Distance dimensions

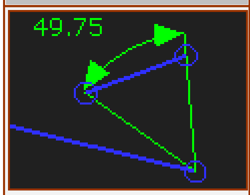
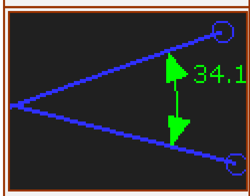
Example	Dimension:	How?	Selection-Window / Geometry-Tree
	• Length of a LINE**	1. Click the LINE**	Dimn
	• Distance between two parallel LINES** • Distance from Y-axis (or X-axis) to a Vertical (or Horizontal) LINE**	1. Click a POINT* at one end of the LINE** 2. Click the other LINE** (or axis).	DimLtoLn
	• Shortest distance between two POINTS*	1. Click each POINT*	DimPtoPn

Example	Dimension:	How?	Selection-Window / Geometry-Tree
	• Horizontal Distance between two POINTS*	1. Add a Vertical LINE** from one of the POINTS* 2. Click the POINT** and then the Vertical LINE**	
	• Vertical Distance between POINTS*	1. Add a Horizontal LINE** from one of the POINTS* 2. Click the POINT* then the Horizontal LINE**	
	• Shortest distance between a POINT* and a LINE** (or axis)	1. Click the POINT* then the LINE** (or axis).	DimPtoLn

Radius

Example	Dimension:	How?	In Selection-Window as:
	• Radius of a CIRCLE	• Click the Circumference of a CIRCLE	DimRadiusn
	• Radius of an ARC	• Click the Radius of an ARC	DimRadiusn

Angle

Example	Dimension:	How?	In Selection-Window as:
	• Angle between three POINTS*	• Click the POINT* that is the apex of the angle, followed by the other two POINTS*. The angle is the clockwise angle from the second POINT* to the third POINT* you click.	DimAngPPPn
	• Angle between two LINES** • Angle between an axis and a LINE**	• Click each LINE** The angle that we display for you has rules.	DimAngLtoLn

Example	Dimension:	How?	In Selection-Window as:
	The LINES** do not need to cross.	See RULES, below	

RULES: There are eight possible angle dimensions:

Before you click your mouse again to fix the angle dimension between the LINES**, you can select which angle to display.

- Move your mouse **counter-clockwise** around the apex of the two LINES**, to show the **acute**, **obtuse**, **supplementary**, or the **vertical** angle <180°.
- Move your mouse **clockwise** around the apex of the two LINES**, to show the **reflex** angles to the other angles.

Video: Double-click to watch Add an Angle

1.5.3.2 Add Point

Add Point

1. Click **Geometry toolbar** > **Add Point**

OR

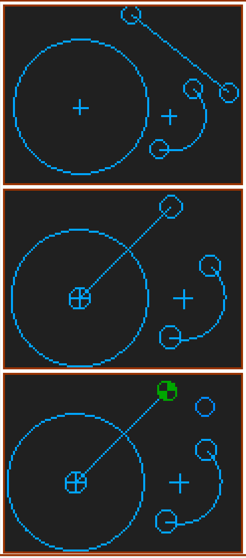
1. Click **Add menu** > **Geometry sub-menu>** > **Add Point**

Then:

2. Click the graphic-area

The **POINT** is now in the graphic-area.

DIFFERENT POINT SYMBOLS



: **POINT** sketch-element

: START-POINT or END-POINT - at the end of an **ARC** or a **LINE**

+ : CENTER-POINT - at the center of an **ARC** or a **CIRCLE**

+ : CENTER-POINT - **Merged** with a START-POINT or END-POINT of an **ARC** or **LINE**

⊕ : CENTER-POINT - **Coincident** with a START-POINT or END-POINT

: LOCKED-POINT - see POINT PROPERTIES DIALOG [444]

: MOTION-POINT - added with **Add Motion-Path FB** [164].

: MOTION-POINT - coincident with a **POINT**, START-POINT or END-POINT.

To add a dimension or locate a Point:

METHOD 1: PART-EDITOR: Add dimensions and constraints - see Add Dimension [223]

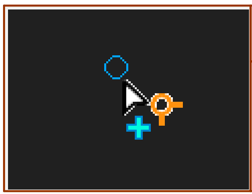
OR

METHOD 2: MECHANISM-EDITOR: Use Point Properties dialog [446] > LOCK AND EDIT HERE

OR

METHOD 3: MECHANISM-EDITOR: Drag a **POINT** if it has no constraints or dimensions and it has a **TRACE-POINT** - **see Add Trace-Point** [104]

Example

	How:	Selection-Window
	• Click the graphic-area to approximately locate the **POINT**.	Point*n*

1.5.3.3 Add Line

Add Line

1. Click **Geometry toolbar** > **Add Line**

OR

1. Click **Add menu** > **Geometry sub-menu>** > **Add Line**

Then:

2. Drag in the graphic-area from the **START-POINT** to the **END-POINT** of the **LINE**.

Drag: mouse-button-down (**START-POINT**), move your mouse-pointer, mouse-button-up (**END-POINT**).

The **LINE** is now in the graphic-area.

Do **Add Line** with a **free START-POINT** and/or **END-POINT**.

> **Do NOT** HOVER above a different **POINT*** before you **mouse-button-down** or **mouse-button-up** to add the **LINE**.
>
> **See also:** How to Delete a Constraint [245]

Do **Add Line** with a **merged START-POINT** and/or **END-POINT**.

> **Do** HOVER above a different **POINT*** before you **mouse-button-down** or **mouse-button-up** to add the **LINE**.
>
> **See:** Hover+Drag Video [241]

* **POINT**, **START-POINT**, **END-POINT**, **CENTER-POINT**.

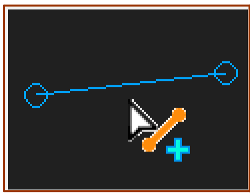
Coordinate System of a Line / CAD-Line

The **Origin** (0,0,0) of the **Coordinate-System** is the **START-POINT** of the **LINE / CAD-LINE**

The **+X–AXIS** is from the **START-POINT** to the **END-POINT** of the **LINE / CAD-LINE**

The **+Y-AXIS** is at **+90º** from the **+X-AXIS**, and on the **MECHANISM-PLANE**.

Example

	How:	*Selection-Window*
	• *Drag* from the **START-POINT** to the **END POINT** of the **LINE**	• **Linen** + • **PointN** at each end of the **LINE**

1.5.3.4 Add Circle

Add Circle

1. Click **Geometry toolbar** > **Add Circle**

OR

1. Click **Add menu** > **Geometry sub-menu>** > **Add Circle**

Then:

2. Drag in the graphic-area from the **CENTER-POINT** to approximate its radius

Drag: mouse-button-down (**CENTER-POINT**), move your mouse-pointer, mouse-button-up

The **CIRCLE** is now in the graphic-area.

Do **Add Circle** with a free **CENTER-POINT**

> **Do NOT** HOVER above a **POINT*** before you ***mouse-button-down***
>
> **See also:** **How to Delete a Constraint** 245

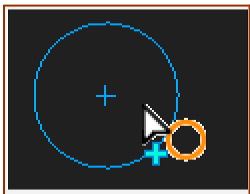
Do **Add Circle** with a merged **CENTER-POINT**

> **Do** HOVER above a different **POINT*** before you ***mouse-button-down***
>
> **See:** **Hover+Drag Video** 240

* **POINT**, **START-POINT**, **END-POINT**, **CENTER-POINT**.

Notes :	A **CIRCLE** is a **sketch-loop**. A **CIRCLE** is the most basic shape of a **Cam-Follower**.
Top tip :	Add the **dimension** to define the **Radius** of the **CIRCLE** before you add other **constraints**.

Example

	How to Add...	Selection-Window
	Mouse-button-down at its **CENTER-POINT**... and Drag to estimate the radius of the **CIRCLE**, and ... Mouse-button-up	Circle*n* + Point*n* at the **CENTER-POINT**

1.5.3.5 Add CAD-Line

Add CAD-Line

See CAD-Line dialog [301]

Add a **CAD-LINE** to:

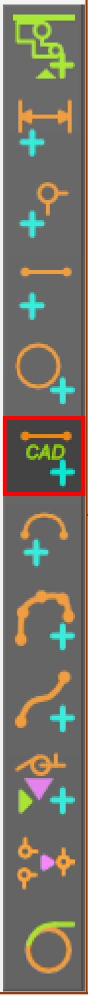
- Display [306] * a DXF-Drawing [306]
- Import and display a SOLIDWORKS Part [302]
- Import and display an STL File [302]
- Specify Mass & Inertia Properties [304]

There is one **CAD-LINE** from the **START-POINT** to the **END-POINT** of each **PART** you add to the model.

Add more **CAD-LINES** to a **PART** (and **BASE-PART**) to import more CAD files onto the same **PART**.

* **Before** you can display a **DXF-DRAWING** with a **CAD-LINE**, you must use File menu > Open > DXF file-type [16] to add a minimum of one **DXF-ELEMENT** to the **ASSEMBLY-TREE**.

Add CAD-Line



1. Click **Geometry toolbar** > **Add CAD-Line**

OR

1. Click **Add menu** > **Geometry sub-menu>** > **Add CAD-Line**

Then:

2. Drag in the graphic-area from the **START-POINT** to its **END-POINT**.

Drag: mouse-button-down (**START-POINT**), move your mouse-pointer, mouse-button-up (**END-POINT**).

The **CAD-LINE** is now in the graphic-area.

See CAD-Line dialog [301]

Do **Add CAD-Line** with a free **START-POINT** and/or **END-POINT**.

> **Do NOT** HOVER above a different **POINT*** before you **mouse-button-down** or **mouse-button-up** to add the **LINE**.
>
> **See also: How to Delete a Constraint** [245]

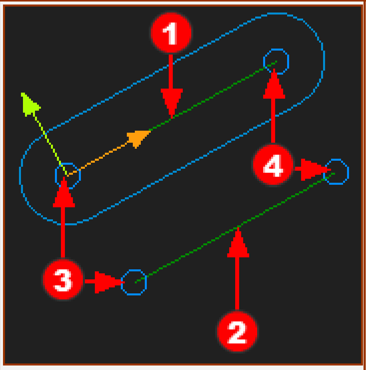
Do **Add CAD-Line** with a merged **START-POINT** and/or **END-POINT**.

> **Do** HOVER above a different **POINT*** before you **mouse-button-down** or **mouse-button-up** to add the **LINE**.
>
> **See: Hover+Drag Video** [241]

* **POINT**, **START-POINT**, **END-POINT**, **CENTER-POINT**.

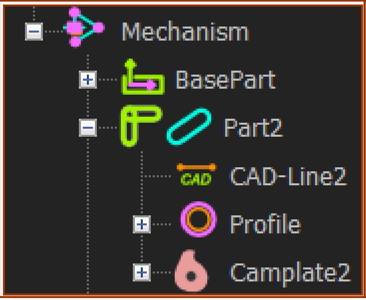
Note :	You can open the **CAD-LINE DIALOG** from the: • **PART-EDITOR**, **MECHANISM-EDITOR** AND, if you link a **SOLID** to the **CAD-LINE**, **MODEL-EDITOR**

The CAD-Line in the graphic-area and Assembly-Tree

<table>
<tr>
<td>


CAD-Lines in an Added-Part
</td>
<td>
In the Graphic-Area:
MECHANISM-EDITOR : Add-Part - a CAD-Line❶ along the center of the Part.
PART-EDITOR : Add CAD-Line to add one or more CAD-LINE❷ to a PART
<hr>
CAD-LINES have a COORDINATE SYSTEM, which is important to orientate CAD files that you import

+X–AXIS: from its START-POINT❸ to its END-POINT❹
+Y–AXIS: at +90° from the +X–AXIS
+Z-AXIS: perpendicular to the MECHANISM PLANE (towards you in the default view (FRONT-VIEW))

</td>
</tr>
<tr>
<td>


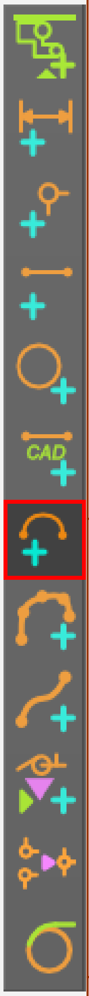
</td>
<td>
In the Assembly-Tree:

The CAD-LINE is the only SKETCH-ELEMENTS that show in the ASSEMBLY-TREE.
The CAD-LINE is a child to a PART.

</td>
</tr>
</table>

1.5.3.6 Add Arc

Add Arc



1. Click **Geometry toolbar** > **Add Arc**

OR

1. Click **Add menu** > **Geometry sub-menu>** > **Add Arc**

Then:

2. Drag in the graphic-area from the **START-POINT** of the **ARC** to the **END-POINT** of the **ARC**
3. Drag **again** between the **START-POINT** and **END-POINT** to approximate the radius of the **ARC**

Drag: mouse-button-down (**CENTER-POINT**), move your mouse-pointer, mouse-button-up

The **ARC** is now in the graphic-area.

Add Arc with a free **START-POINT** and/or **END-POINT**.

> **Do NOT** HOVER above a different **POINT*** before you **mouse-button-down** or **mouse-button-up** to add the **ARC**.
>
> **See also: How to Delete a Constraint** 245

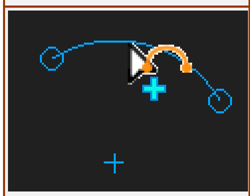
Add Arc with a merged **START-POINT** and/or **END-POINT**.

> **Do** HOVER above a different **POINT*** before you **mouse-button-down** or **mouse-button-up** to add the **ARC**.
>
> **See: Hover+Drag Video** 241

* **POINT**, **START-POINT**, **END-POINT**, **CENTER-POINT**.

Top tip:	**Add a Dimension** 222 to define the **RADIUS** of the **ARC** **before** you drag it or add other **CONSTRAINTS** 255

Example

Example	How:	Selection-Window
	**You must Drag Two times.** Refer to the image to the left: 1. **FIRST DRAG**: from the **START-POINT** to the **END-POINT** of the **ARC**, then release you mouse. 2. **SECOND DRAG**: Position your pointer between the **START-POINT** and **END-POINT** of the **ARC**, and Drag towards the apex of the **ARC** as you intend it to be drawn, then release your mouse. The **ARC** has a **CENTER-POINT**, a **START-POINT** and **END-POINT**.	Circle*n* + Point*n* At each end of the **ARC**.

Video

Double-click to watch 'How to Add an Arc'

1.5.3.7 Add Spline

Add Spline

1. Click **Geometry toolbar** > **Add Spline**

OR

1. Click **Add menu** > **Geometry sub-menu>** > **Add Spline**

Then

2. Click in the graphic-area to add the **first Nodes** for the SPLINE
3. Continue to click at different places* to add more **nodes**

To end **Add Spline**:

4. Click the **first node** again to add a **CLOSED SPLINE**
   OR
   Click the **last node** again to add an **OPEN SPLINE.**

* Click a POINT (POINT, START-POINT, END-POINT, CENTER-POINT) of a different sketch-element to merge it with a **node**.

The SPLINE curve is now in the graphic-area.

Edit a Node's Position

PART-EDITOR: Add dimensions 222 or Constraints

OR

PART-EDITOR: Drag the node

OR

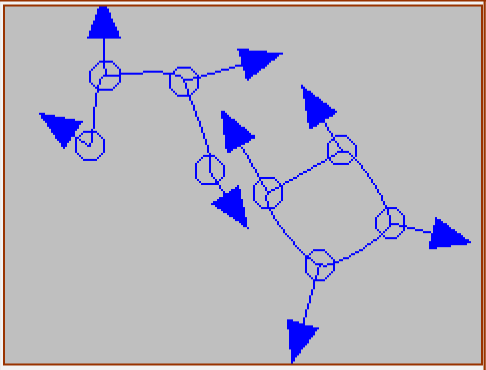
MECHANISM-EDITOR or PART-EDITOR : **Use the** Point Properties dialog 444

Edit the Spline's Angle at a Node

1. MECHANISM-EDITOR or PART-EDITOR : Click the SPLINE to show the SPLINE-HANDLES
2. MECHANISM-EDITOR or PART-EDITOR : Drag the arrowhead of a SPLINE-HANDLE around its NODE
3. MECHANISM-EDITOR or PART-EDITOR : Click the SPLINE again to hide the SPLINE-HANDLES

Edit the Spline's Curvature at a Node

1. MECHANISM-EDITOR or PART-EDITOR : Click the SPLINE to show the SPLINE-HANDLES
2. MECHANISM-EDITOR or PART-EDITOR : Drag the arrowhead of a SPLINE-HANDLE further from or nearer to a NODE
3. MECHANISM-EDITOR or PART-EDITOR : Click the SPLINE again to hide the SPLINE-HANDLES

 Spline-Handles of an Open-Spline and a Closed-Spline	**Notes:** • The **direction** of the SPLINE at a **NODE** is normal to its **SPLINE-HANDLES** • The **Radius-of Curvature** of the SPLINE at a **NODE** is proportional to the length of its **SPLINE-HANDLE** • The **Radius-of-Curvature** of the SPLINE at a **NODE** is symmetrical to the two sides of its **SPLINE-HANDLE**. • You can add **Constraints** to **Nodes**, but not the SPLINE'S curve.

1.5.3.8 Add Blend-Curve

Add Blend-Curve

See also: Blend-Curve dialog [449]

1. Click Geometry toolbar > Add Blend-Curve

OR

1. Click Add menu > Geometry sub-menu> > Add Blend-Curve

Then:

2. Drag in the graphic-area from the **START-POINT** to its **END-POINT**.

Drag: mouse-button-down (**START-POINT**), move your mouse-pointer, mouse-button-up (**END-POINT**).

The BLEND-CURVE is now in the graphic-area.

Note:

Add with Free-Points

Do NOT **HOVER** above a **POINT*** before you **mouse-button-down** or **mouse-button-up**.

See also: How to Delete a Constraint [245]

Add with Merged-Points

Do **HOVER** above a **POINT*** before you **mouse-button-down** or **mouse-button-up**.

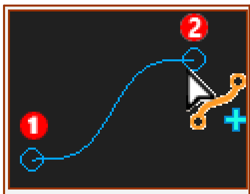
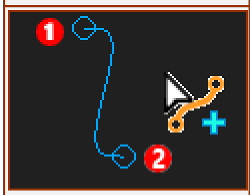
See: Hover+Drag Video [241]

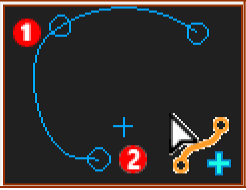
Note:

The BLEND-CURVE automatically adjusts its ANGLE, CURVATURE, and CURVATURE-RATE to equal those of the sketch-element with which you merge the BLEND-CURVE.

* POINT, START-POINT, END-POINT, CENTER-POINT.

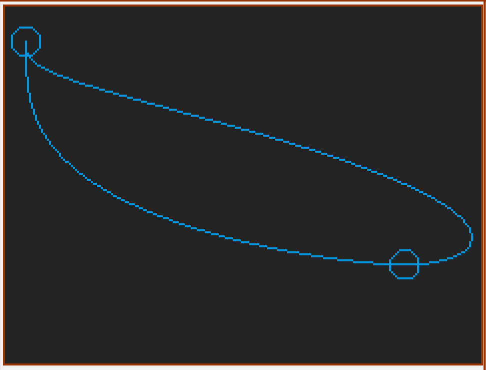
Example

	How:	*Selection-Window* *Assembly-Tree*
	• Drag from the **START-POINT** 1 to the **END POINT** 2 of BLEND-CURVE	Blend-Curve*n* Point*n* at each end of the CAD-Line The Blend-Curve is a child to the Part.
	The <u>default</u> ANGLE, CURVATURE and CURVATURE RATE at the **START-POINT** and **END-POINT** of the BLEND-CURVE are zero. The images to the left and above are default BLEND-CURVES. • The BLEND-CURVE at the top has been added with a drag from bottom-left to top-right. • The BLEND-CURVE at the bottom has been added with a drag from top-right to bottom-left.	



- **Hover+Drag** from the END-POINT of a different sketch-element to merge the START-POINT of a new BLEND-CURVE.
- The BLEND-CURVE matches the ANGLE, CURVATURE and CURVATURE RATE with the other sketch-element.

What is a Blend-Curve?



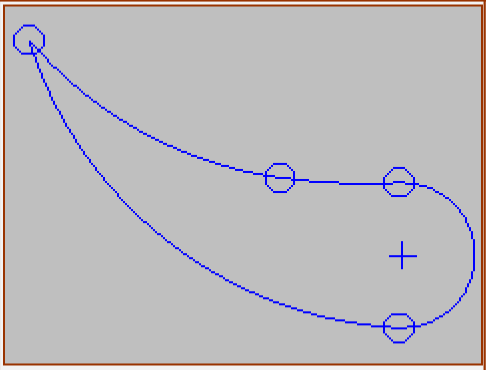
Continuous Curvature AND Curvature Rate

A BLEND-CURVE can give a complex and useful 2D path

Blend-Curve Parameters at its START-POINT and END-POINT:

- ANGLE
- CURVATURE
- CURVATURE-RATE
- VELOCITY SCALING

Why use a Blend-Curve?



Tangent are NOT Smooth

This path is NOT smooth - Tangent ARCS (with different Radii) have discontinuous CURVATURE.

Use Blend-Curve to give a smooth open or closed sketch-path. See also Smoothness 237.

If the **Blend-Curve** is to guide a MOTION-POINT which also guides a PART, it is important that:

- The motion of the MOTION-POINT along a **sketch-path** does not have discontinuities in position, velocity, acceleration, and jerk.

AND

- The **shape** of the sketch-path does not have discontinuities in angle, curvature, and rate-of-change of curvature.

Smoothness definitions

Smoothness of Sketch-elements.

You are often concerned with the smoothness of the sketch. The smoothness at the transition between sketch-element is important when a MOTION-POINT moves along the sketch-elements.

	Angle	Curvature	Curvature Rate
Line joins a Line	Can 'kink', be the same angle (why two Lines?)	NA	NA
Arc joins a Line	Keep the same Angle	Keep the same Curvature (or Radius)	NA
Arc joins an Arc	Keep the same Angle	Keep the same Curvature (or	NA

		Radius)	
Blend-Curve joins a Line	Keep the same Angle	Keep the same Curvature=0 (infinite Radius)	Keep at 0
Blend-Curve joins an Arc	Keep the same Angle	Keep the same Curvature; Radius	Keep at 0
Blend-Curve joins a Blend-Curve	Keep the same Angle	Keep the same Curvature; Radius	Keep the same Curvature Rate

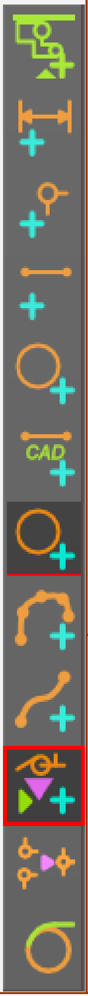
1.5.3.9 Add Import SOLIDWORKS Sketch FB

Import a SOLIDWORKS sketch

To save you time, you can import SOLIDWORKS sketches directly onto a **PART**.

See also : **Import SW Sketch dialog** 454

Add Import SOLIDWORKS Sketch FB



1. Click **Geometry toolbar** > **Add Import SOLIDWORKS sketch FB**

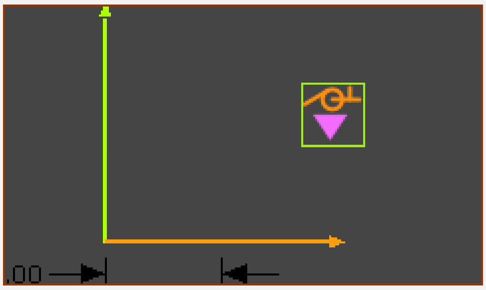
OR

1. Click **Add menu** > **Add Import SW sketch FB**

Then:

2. Click the graphic-area

The **IMPORT SOLIDWORKS SKETCH FB** is now in the graphic-area.



See : **Import SW Sketch dialog** 454

1.5.3.1 Merge Points 0

About Merge-Points

The **Merge-Points** command combines two **POINTS*** into one **POINT**.

* **START-POINTS**, **END-POINTS**, **CENTRE-POINTS**, and/or **POINTS.**

When do you need to Merge-Points?

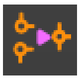
- To sketch a **sketch-loop** for a **PROFILE / EXTRUSION**
- To sketch an **sketch-path** for a **MOTION-POINT** (with **Add Motion-Path FB**)
- To merge the **END-POINTS** of Lines-of-Centers when you add two or more **GEAR-PAIRS** to design a Gear-Train

---

See also: Coincident-Constraint[255].
See also: Hover+Drag technique[241] - it is more efficient than **Merge-Points**.

Merge-Points

Merge-Points :



1. Click **Geometry toolbar** > **Merge-Points**

OR

1. Click **Add menu** > **Geometry sub-menu>** > **Merge-Points**

Then, **IF** the **POINTS*** are not coincident:

2. Click a **POINT*** at the end of a sketch-element
3. Click a **POINT*** that at the end of a different sketch-element
4. Click the graphic-area

OR, **IF** the **POINTS*** are coincident:

2. Click the two coincident **POINTS***
3. Click the graphic-area

The two **POINTS** are now one **POINT**.

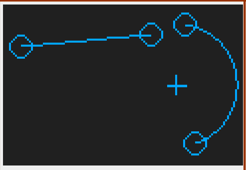
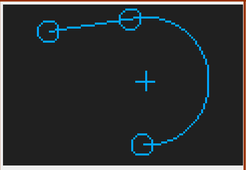
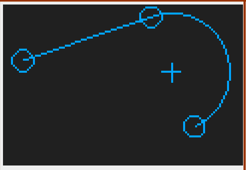
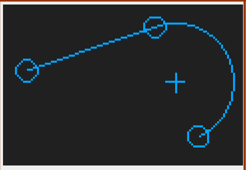
---

* **POINTS** : **START-POINT** and/or **END-POINT** of a **LINE, ARC** or **BLEND-CURVE** sketch-element.

Notes:

- You CANNOT merge a **POINT*** with the **START-POINT** or **END-POINT** of the **CAD-LINE** along the X-axis of an **ADDED-PART**.
- You CAN, however, merge a **START-POINT** or **END-POINT** of a sketch-element if you drag to or from the **START-POINT** or **END-POINT** of the **CAD-LINE.**

Merge-Points

Before	*After*	*Select:*	*In Selection-Window / Geometry-Tree*
 Free Points	 Merged Points	• Two different **POINTS** at the end of sketch-elements. Click a **POINT** and then a different **POINT**. One **POINT** is deleted (Compare with CoincidentPtoP) [255], and the sketch-elements move together.	PointN
 Coincident Points	 now Merged-Points	• Two Coincident **POINTS** A **POINT** and the Coincident constraint is deleted	PointN

Video of Merge-Points with Hover Technique.

Double-click to watch 'How to add a Sketch-Loop - Hover Technique'

1.5.3.1 Add Transition Curve 1

The Transition-Curve is not available with the Trial License.

It is ONLY available with the Premium License.

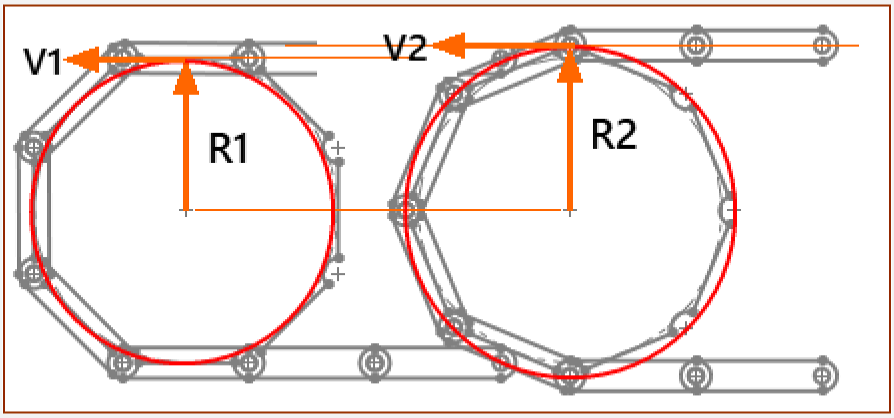
About Transition-Curve

The **PSMotion Transition-Curve©** is used to eliminate:

- Polygonal-Action, also referred to as Chordal-Action, **AND**
- Linear Velocity-Discontinuity,

each time a chain-link moves onto a chain-sprocket (chain-tooth).

About Polygonal-Action (Chordal Action) and Chains



Standard Sprocket and Polygonal Action

As a chain-link moves onto a chain-tooth the chain moves upwards and downwards. Its radius changes from R1 to R2 and back again to R1 (see image above)

- the Linear-Velocity of the chain varies between :
  - minimum Linear-Velocity of $V_1 = \omega R_1$
  - maximum Linear Velocity of $V_2 = \omega R_2$
- the Linear-Velocity variation is NOT sinusoidal - it is similar to sinusoidal bumps ($0 \rightarrow \pi$), without the valley!
- the Linear Acceleration of the chain continually varies
- the Linear-Acceleration is discontinuous at the instant the chain starts to move upwards from R1 (or equivalently, when it ends its move downwards).
- each Pin (that joins the chain-links together) impacts the root of the chain-tooth

Polygonal-Action creates two(2) problems

- Velocity Variation in the Chain

  In the general case, if a Driving Sprocket is rotating at constant Angular Velocity, the chain does not move at constant Linear Velocity.

  The ratio of Maximum and Minimum Velocity increases as you reduce the number-of-teeth on the drive sprocket.

- Tension Variation in the Chain.

  As a chain link engages with a tooth, the chain accelerates, and thus there is a tension variation.

  Also, the pitch of the chain link is not constant.

Usually, it is recommended that you use more than 17 teeth on a standard sprocket. Bicycle sprockets have fewer, but the derailleur compensates for the velocity and tension-variations.

Add Transition-Curve

Toolbar :	Part-Editor : Geometry Toolbar > Transition-Curve
Menu :	Part-Editor : Add menu > Geometry sub-menu> > Transition-Curve

To add a Transition-Curve

1. Click Geometry-toolbar > Transition-Curve
2. Drag to add a Line

   The Transition-Curve is added.

You can see that the Transition Curve is easy to add. However, it must be used with Pulleys and Chains to be useful.

A Tutorial will show you how to apply the Transition Curve to a Long-Link Chain.

1.5.4 Constraints

Part-Editor - Constraints Menu and Toolbar

In the **PART-EDITOR**, use the **Constraints menu and toolbar** to add **Constraints.**

You can add **Constraints**

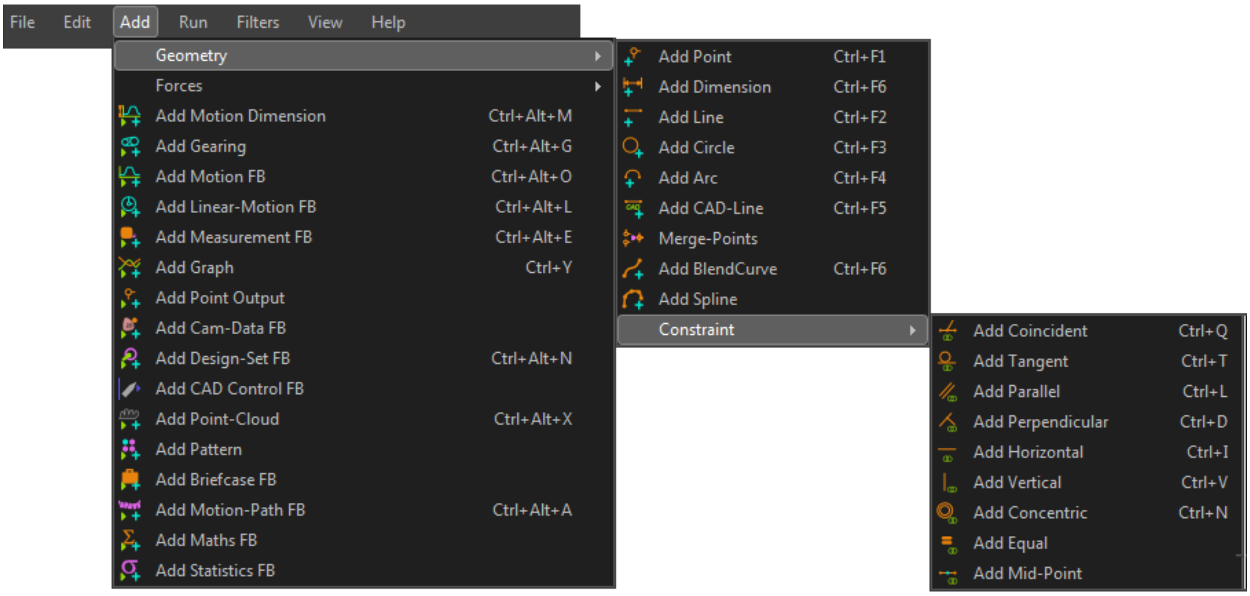
- to a sketch-element
- between two sketch-elements
- between a sketch-element and the X-axis or Y-axis

See below: Why cannot I add another Constraint? [246]

See below: How to Delete a Constraint [245].

See also: Geometry menu and toolbar [219]

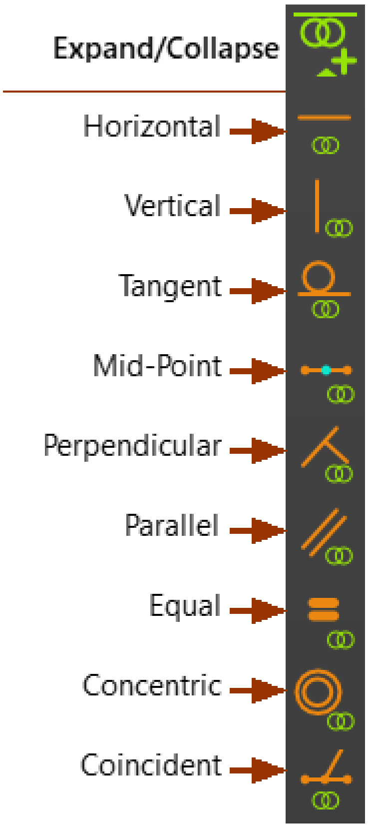
Part-Editor: Constraints menu



Constraints menu

Part-Editor: Constraints Toolbar

PART-EDITOR > **Constraints toolbar** is to the **right** of the graphic-area.



Constraints toolbar

How to Delete a Constraint

In the graphic-area, you cannot see **Constraints** that you have added to or between SKETCH-ELEMENTS.

To DELETE a **Constraint**:

1. SHIFT+CLICK a SKETCH-ELEMENT or a POINT*

   All of the **Constraints** that apply to the SKETCH-ELEMENT or POINT* are now in the SELECTION-WINDOW

   In the SELECTION-WINDOW:

2. Right-click the **Constraint** you want to delete
3. Click **Delete element** in the shortcut menu

* POINT, START-POINT, END-POINT, or CENTER-POINT

About Merge-Points and Coincident Constraint between Points

Add MERGE-POINTS is between two POINTS*. The command will **delete** one of the POINTS*.

You CANNOT delete POINTS* that you merge with Merge-Points.

Usually, do Merge-Points to make a **sketch-loop** or **sketch-path**.

Add Coincident Constraint is between two POINTS*. The command does NOT delete one of the POINTS*.

You CAN delete the COINCIDENT-CONSTRAINT between the two POINTS* - see How to delete a constraint 245.

Usually, add Coincident Constraint for construction geometry.

* POINTS, START-POINTS, END-POINTS, CENTER-POINTS.

OVER-Constrain a sketch-element?

You cannot (usually) add a constraint that OVER-constrains a sketch-element.

For example, a **POINT*** has two degrees-of-freedom.

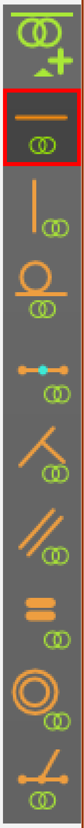
1. If you add a **Coincident Constraint** between a **POINT** and a **LINE** (**CoincidentPtoL**) you remove 1 degrees-of-freedom.
2. Then, you attempt to add a **Coincident Constraint** between two **POINTS** (**CoincidentPtoP**) you attempts to remove 2 degrees-of-freedom.

However, it is not possible to remove three(3) degrees-of-freedom from a **POINT**.

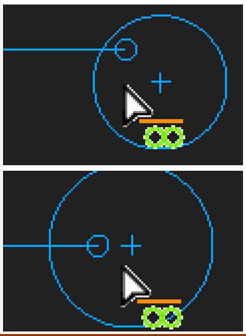
To correct the problem, delete the **CoincidentPtoL - see** Delete Constraint 245 - then Add a constraint between the two **POINTS**.

1.5.4.1 Add Horizontal Constraint

Add Horizontal



Two Points (HorizontalPtoP)



1. Click **Constraints toolbar > Add Horizontal**

OR

1. Click **Add menu > Constraints sub-menu> > Add Horizontal**

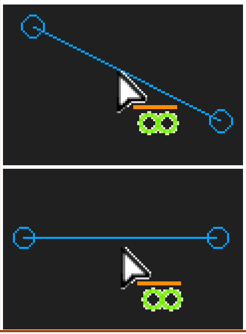
Then:

2. Click a **POINT***
3. Click a different **POINT***

   The two **POINTS*** are **Horizontal** - parallel with the **X-AXIS**

* **POINT**, **START-POINT**, **END-POINT**, or **CENTRE-POINT**

A Line:



1. Click **Constraints toolbar > Add Horizontal**

OR

1. Click **Add menu > Constraints sub-menu> > Add Horizontal**

Then:

2. Click a **LINE****

   The **LINE**** becomes **Horizontal** - parallel with the **X-AXIS**

** **LINE** or **CAD-LINE**

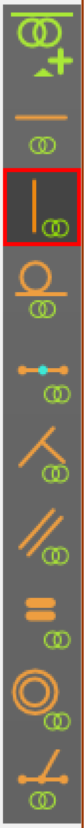
Delete Horizontal

1. **SHIFT+CLICK** a sketch-element with a **Horizontal** constraint.

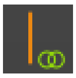
   The sketch-element and **Horizontal** constraint show in the **SELECTION-WINDOW**.
2. Right-click the **HorizontalL** or **HorizontalPtoP** constraint in the **SELECTION-WINDOW**
3. Click **Delete** in the shortcut menu

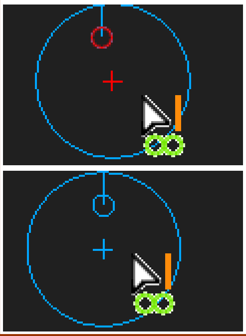
1.5.4.2 Add Vertical Constraint

Add Vertical



Two Points:





1. Click **Constraints toolbar** > **Add Vertical**

OR

1. Click **Add menu** > **Constraints sub-menu>** > **Add Vertical**

Then:

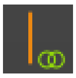
2. Click a **POINT***
3. Click a different **POINT***

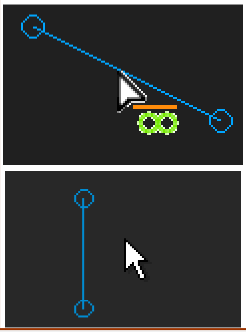
   The two **POINTS*** become **VERTICAL** - parallel with the **Y-AXIS**

---

* **POINT**, **START-POINT**, **END-POINT**, or **CENTRE-POINT**

A Line:





1. Click **Constraints toolbar** > **Add Vertical**

OR

1. Click **Add menu** > **Constraints sub-menu>** > **Add Vertical**

Then:

2. Click a **LINE****

   The **LINE**** becomes **VERTICAL** - parallel with the **Y-AXIS**

** **LINE** or **CAD-LINE**.

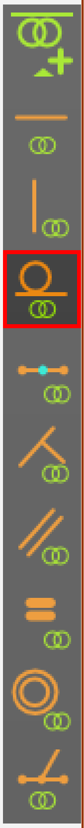
Delete Vertical

1. **SHIFT+CLICK** a sketch-element with a **Vertical** constraint

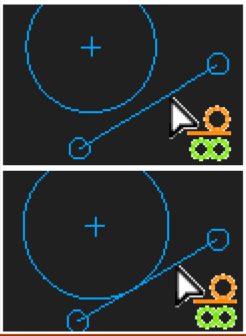
   The sketch-element and **VerticalL or VerticalPtoP** constraint show in the **SELECTION-WINDOW**.
2. Right-click the **VerticalL** or **VerticalPtoP** constraint in the **SELECTION-WINDOW**
3. Click **Delete** in the shortcut menu

1.5.4.3 Add Tangent Constraint

Add Tangent



A Line and Circle or Arc :



1. Click **Constraints toolbar** > **Add Tangent**

OR

1. Click **Add menu** > **Constraints sub-menu>** > **Add Tangent**

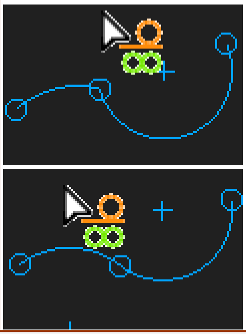
Then:

2. Click a **CIRCLE** or **ARC**
3. Click a **LINE***

   The **CIRCLE/ARC** is **TANGENT** with the **LINE***

* **LINE**, **CAD-LINE**, **X-AXIS**, or **Y-AXIS**

Circle/ Arc and Circle / Arc :



1. Click **Constraints toolbar** > **Add Tangent**

OR

1. Click **Add menu** > **Constraints sub-menu>** > **Add Tangent**

Then:

2. Click a **CIRCLE** or **ARC**
3. Click a different **CIRCLE** or **ARC**

   The **CIRCLE** and/or **ARCS** are **TANGENT**

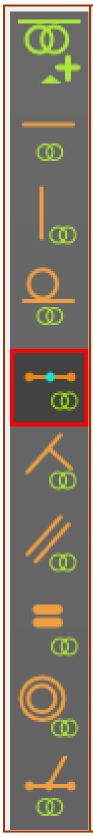
Delete Tangent

1. **SHIFT+CLICK** a sketch-element with a **Tangent** constraint

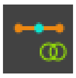
   The sketch-element and **TangentCtoL** or **TangentCtoC** constraint show in the **SELECTION-WINDOW**.
2. Right-click the **TangentCtoL** or a **TangentCtoC** constraint in the **SELECTION-WINDOW**
3. Click **Delete** in the shortcut menu

1.5.4.4 Add MidPoint Constraint

Add Mid-Point

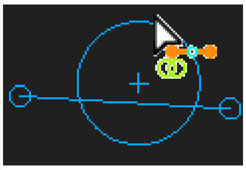


A Point and Line :



1. Click **Constraints toolbar** > **Add Mid-Point**

OR

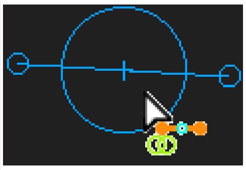


1. Click **Add menu** > **Constraints sub-menu>** > **Add Mid-Point**

Then:

2. Click a **POINT***
3. Click a **LINE****

The **POINT*** is at the **MID-POINT** of the **LINE**



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* **POINT**, **START-POINT**, **END-POINT**, or **CENTRE-POINT**

** **LINE** or **CAD-LINE**

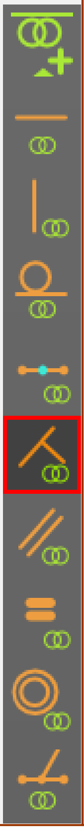
Delete Mid-Point

1. **SHIFT+CLICK** a sketch-element with a **Mid-Point** constraint

   The sketch-element and **Mid-Point** constraint show in the **SELECTION-WINDOW**.
2. Right-click the **Mid-Point** constraint in the **SELECTION-WINDOW**
3. Click **Delete** in the shortcut menu

1.5.4.5 Add Perpendicular Constraint

Add Perpendicular

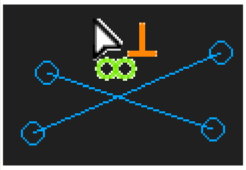


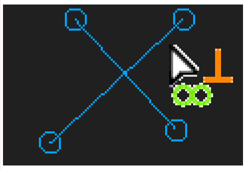
A Line and Line :

1. Click **Constraints toolbar > Add Perpendicular**

OR

1. Click **Add menu > Constraints sub-menu> > Add Perpendicular**





Then:

2. Click a **LINE***
3. Click a different **LINE***

   The **LINES** are **Perpendicular**

* **LINE**, **CAD-LINE**, **X-AXIS**, or **Y-AXIS**

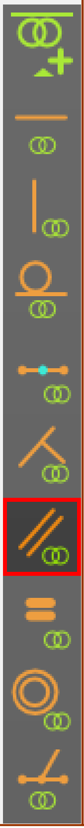
Delete Perpendicular

1. **SHIFT+CLICK** a sketch-element with a **Perpendicular** constraint

   The sketch-element and **Perpendicular** constraint show in the **SELECTION-WINDOW**.
2. Right-click the **Perpendicular** constraint in the **SELECTION-WINDOW**
3. Click **Delete** in the shortcut menu

1.5.4.6 Add Parallel Constraint

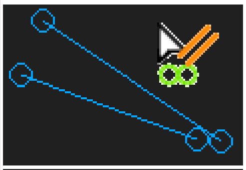
Add Parallel



A Line and Line :

1. Click **Constraints toolbar** > **Add Parallel**

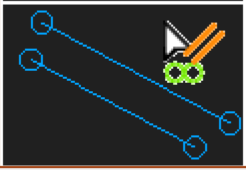
OR



1. Click **Add menu** > **Constraints sub-menu>** > **Add Parallel**

Then:

2. Click a **LINE*** sketch-element



3. Click a different **LINE*** sketch-element

The two **LINES** are **Parallel**

* **LINE**, **CAD-LINE**, **X-AXIS**, or **Y-AXIS**

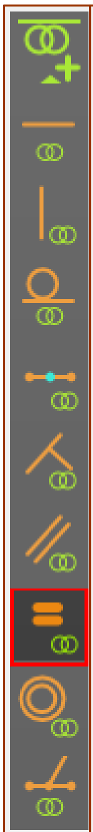
Delete Parallel

1. **SHIFT+CLICK** a sketch-element with a **Parallel** constraint

   The sketch-element and **Parallel** constraint show in the **SELECTION-WINDOW**.
2. Right-click the **Parallel** constraint in the **SELECTION-WINDOW**
3. Click **Delete** in the shortcut menu

1.5.4.7 Add Equal Constraint

Add Equal



A Line and Line :

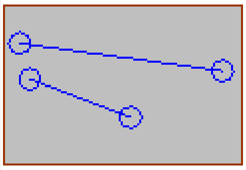
1. Click **Constraints toolbar** > **Add Equal**

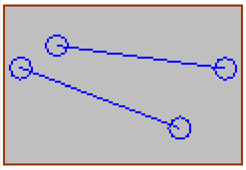
OR

1. Click **Add menu** > **Constraints sub-menu>** > **Add Equal**

Then:

2. Click a **LINE***
3. Click a different **LINE***





The **Length** of the two **LINES** are **Equal**

* **LINE** or **CAD-LINE**

A Circle (or Arc) and Circle (or Arc) :

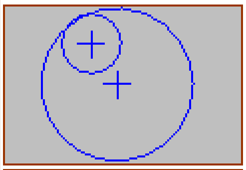
1. Click **Constraints toolbar** > **Add Equal**

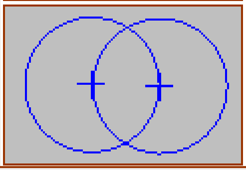
OR

1. Click **Add menu** > **Constraints sub-menu>** > **Add Equal**

Then:

2. Click a **CIRCLE*** sketch-element
3. Click a different **CIRCLE*** sketch-element





The **Radius** of the two **CIRCLES*** are **Equal**

* **CIRCLES** or **ARCS**

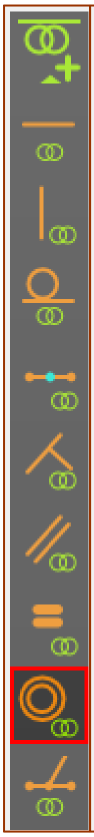
Delete Equal

1. **SHIFT+CLICK** a sketch-element with an **Equal** constraint

   The sketch-element and **Equal** constraint show in the **SELECTION-WINDOW**.
2. Right-click the **Equal** constraint in the **SELECTION-WINDOW**
3. Click **Delete** in the shortcut menu

1.5.4.8 Add Concentric Constraint

Add Concentric



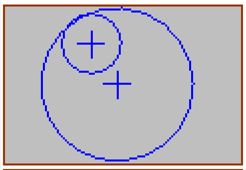
A Circle (or Arc) and Circle (or Arc) :

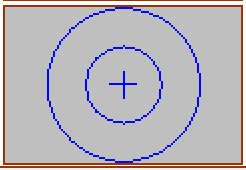
1. Click **Constraints toolbar** > **Add Concentric**

OR

1. Click **Add menu** > **Constraints sub-menu>** > **Add Concentric**

Then:





2. Click a **CIRCLE** (or **ARC**)
3. Click a different **CIRCLE** (or **ARC**)

   The **CIRCLE** and/or **ARC** are **Concentric**

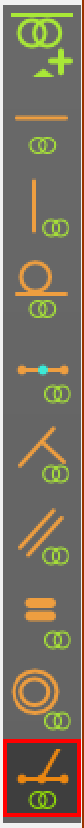
Delete Concentric

1. **SHIFT+CLICK** a sketch-element with an **Concentric** constraint

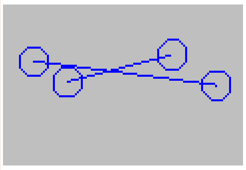
   The sketch-element and **Concentric** constraint show in the **SELECTION-WINDOW**.
2. Right-click the **Concentric** constraint in the **SELECTION-WINDOW**
3. Click **Delete** in the shortcut menu

1.5.4.9 Add Coincident Constraint

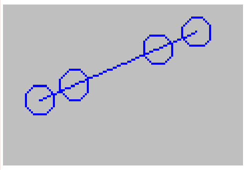
Add Coincident



A Line and Line (CoincidentLtoL) :



Before Coincident LtoL



After: CoincidentLtoL

1. Click **Constraints toolbar** > **Add Coincident**

OR

1. Click **Add menu** > **Constraints sub-menu>** > **Add Coincident**

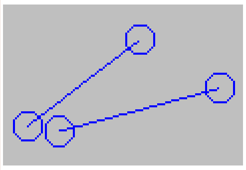
Then:

2. Click a **LINE***
3. Click a different **LINE***

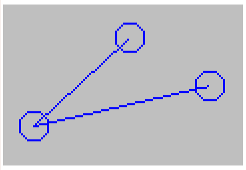
The **LINES*** are now **Collinear.**

* **LINE, CAD-LINE, X-AXIS**, or **Y-AXIS**

A Point and Point (CoincidentPtoP) :



Before CoincidentPtoP



CoincidentPtoP

1. Click **Constraints toolbar** > **Add Coincident**

OR

1. Click **Add menu** > **Constraints sub-menu>** > **Add Coincident**

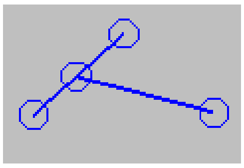
Then:

2. Click a **POINT***
3. Click a different **POINT***

The **POINTS*** are now **Coincident.**

* **POINT, START-POINT, END-POINT**, or **CENTRE-POINT**

Point and Line (CoincidentPtoL) ; Point and Circle (CoincidentPtoC)



After CoincientPtoL

1. Click **Constraints toolbar** > **Add Coincident**

OR

1. Click **Add menu** > **Constraints sub-menu>** > **Add Coincident**

Then:

2. Click a **POINT***

Then:

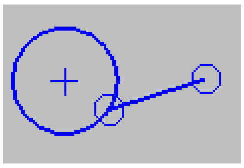
3. Click a **LINE****

The **POINT*** is now **Coincident** with the ray of the **LINE****

OR

3. Click a **CIRCLE** or **ARC**

The **POINT*** is now **Coincident** with the circumference of the **CIRCLE** or **ARC**

		 After CoincidentPtoC	* **POINT**, **START-POINT**, **END-POINT**, or **CENTRE-POINT** ** **LINE, CAD-LINE,** **X-AXIS**, or **Y-AXIS**

Delete Coincident

1. **SHIFT+CLICK** a sketch-element with a **Coincident** Constraint.

   The sketch-element and **CoincidentPtoP**, **CoincidentPtoL**, and/or **CoincidentPtoC** constraint show in the **SELECTION-WINDOW**.

In the **SELECTION-WINDOW**:

2. Right-click the **CoincidentLtoL**, **CoincidentPtoP**, **CoincidentPtoL**, or **CoincidentPtoC** constraint
3. Click **Delete** in the shortcut menu

Note: Coincident Constraints, Points and Lines, and Degrees-of-Freedom

COINCIDENT CONSTRAINT, POINTS, and **DEGREES-OF-FREEDOM**

A **POINT*** has two degrees-of-freedom.

1. If you add a **Coincident Constraint** between a **POINT*** and a **LINE**** (**CoincidentPtoL**) you remove 1 degrees-of-freedom.
2. Then, add a **Coincident Constraint** between two **POINTS*** (**CoincidentPtoP**) - which attempts to remove 2 more degrees-of-freedom.

However, it is not possible to remove three(3) degrees-of-freedom from a **POINT***.

Therefore, the **POINTS*** do **not** move together when you add the **CoincidentPtoP**

To correct the problem, you must delete the **CoincidentPtoL** - see Delete Constraint 245.

---

* **POINT**, **START-POINT**, **END-POINT**, or **CENTRE-POINT**

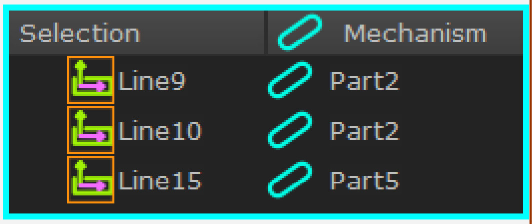
** **LINE, CAD-LINE,** **X-AXIS**, or **Y-AXIS**

1.6 3: Project Explorer

❸ Project-Explorer

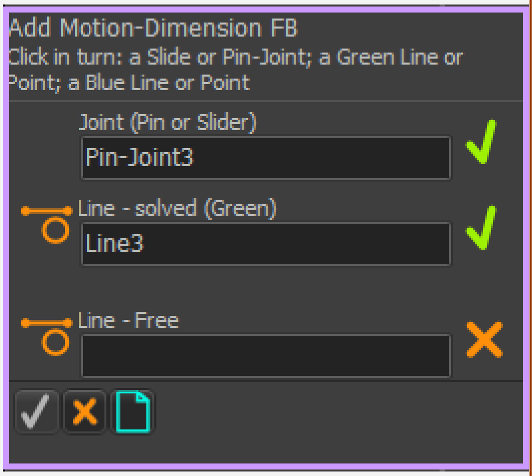
The **PROJECT-EXPLORER** is to the left of the graphic-area.

It has three(3) areas.



SELECTION-WINDOW [257]

The **SELECTION-WINDOW** shows the list of elements that you **CLICK**, or **SHIFT+CLICK**, in the graphic-area or **ELEMENT-EXPLORER** (see below).

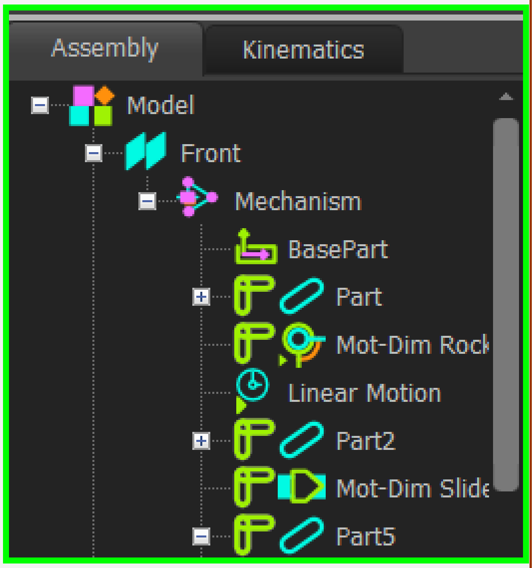


COMMAND-MANAGER [259]

Commands add new elements to the model. To do many of the commands, you must select other elements that are in the model.

The **COMMAND-MANAGER** is a small interface that has a selection-box for each element you must select to add the new element.

Select elements for each selection-box from the graphic-area or **ELEMENT-EXPLORER** (see below).



ELEMENT-EXPLORER [261]

The **ELEMENT-EXPLORER** shows the elements that are in the **MODEL-EDITOR** and **MECHANISM-EDITORS**.

The **ELEMENT-EXPLORER** has three(3) trees, but shows only two(2) trees at one time.

In the **MODEL-EDITOR** and **MECHANISM-EDITORS**, the trees are the:

- ASSEMBLY-TREE [261] - see image
- KINEMATICS-TREE [262] (If necessary, click Rebuild Now [37] to compile the **KINEMATICS-TREE**)

In the **PART-EDITOR**, the trees are the:

- ASSEMBLY-TREE [261]
- GEOMETRY-TREE [267] (If necessary, click Rebuild Now [37] to compile the **GEOMETRY-TREE**)

1.6.1 Selection-Window

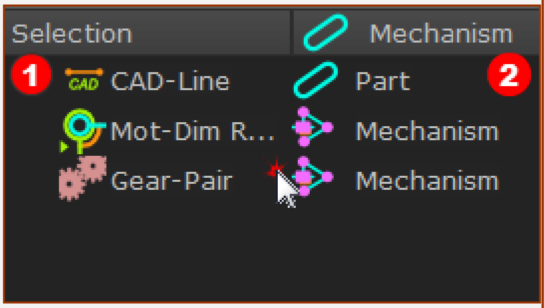
Selection-Window

The **SELECTION-WINDOW** is at the top of the **PROJECT-EXPLORER**.

The Selection-Window shows the list of the elements that you **Click** or **Shift+Click** in the graphic-area or the Element-Explorer [261].

Click ...

... Elements



Elements in the Selection-Window

Selection list ❶

The list of elements that you **Click** in the graphic-area or ELEMENT-EXPLORER 261.

Owner list ❷

Mechanism: when the element is a child to a **MECHANISM-EDITOR.**

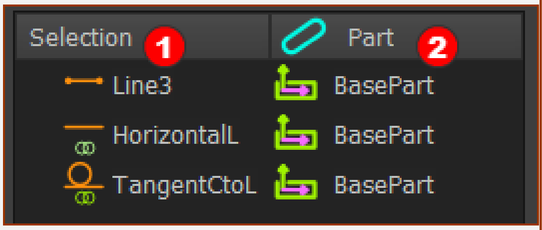
Part: when the element is a child to a PART.

Shift-Click ...

... Sketch-Elements to list their Constraints

You cannot see a CONSTRAINT in the graphic-area after you add it to a **sketch-element**.

SHIFT+CLICK a **sketch-element** to list in the **SELECTION-WINDOW** those CONSTRAINTS that you have added to the **sketch-element**.



Constraints in the Selection-Window with SHIFT+CLICK

Selection list ❶

The **sketch-element** that you SHIFT+CLICK

+ The CONSTRAINTS that you have added to the sketch-element - for example **HorizontalL.**

+ The CONSTRAINTS that you have added between the **sketch-element** and a different **sketch-element** - for example, **Perpendicular.**

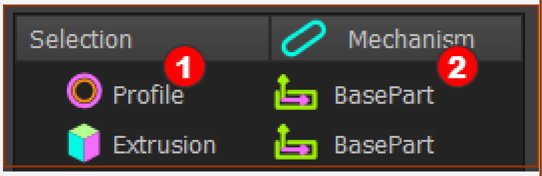
Owner list ❷

Part - the PART to which you have added the **sketch-element**.

... Profiles to list the Extrusion

When you click a PROFILE, the EXTRUSION does not show in the **SELECTION-WINDOW** if it is not in the graphic-area.

SHIFT+CLICK the PROFILE to add to the list the EXTRUSION that is a child to the PROFILE.



You can EDIT the Extusion or DELETE the Profile

Selection list ❶

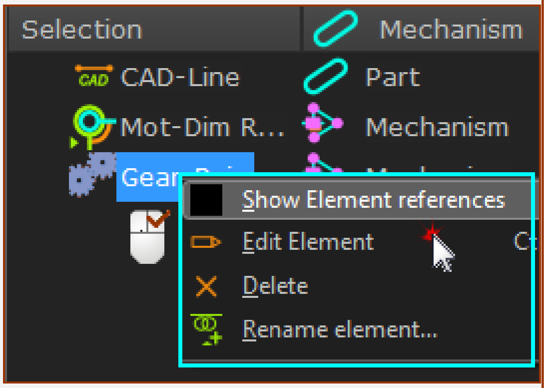
The PROFILE that you SHIFT+CLICK

The EXTRUSION that is a child to the PROFILE.

Owner list ❷

Part: the PART to which you have added the PROFILE.

Shortcut menu in Selection-Window



Contextual menu in Selection-Window

To show the **shortcut menu**

1. Right-click an element*

Use the shortcut menu to:

- **Show Element References - see below** 259
- **Edit element**
- **Delete element**
- **Rename element**

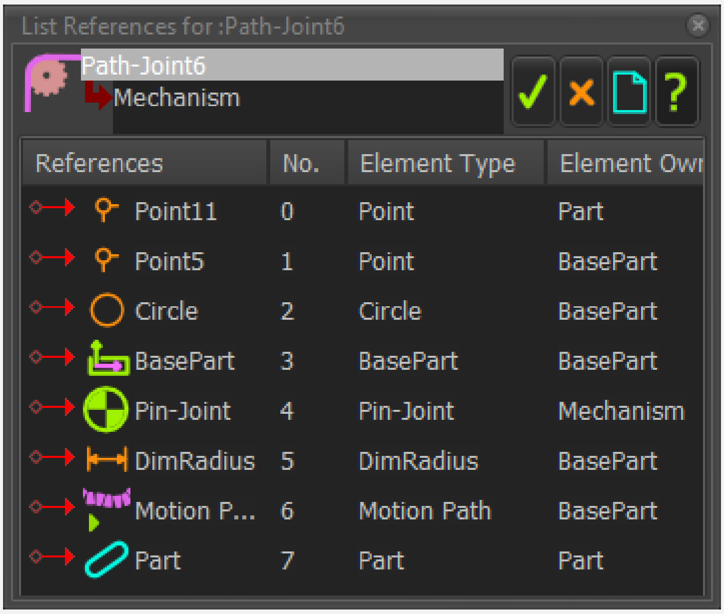
* **Show or Hide Extrusion** if you right-click an EXTRUSION

Show Element References

Many elements need **other elements** to exist in the model. The **other elements** are **Reference Elements**.

For example, the **Reference Elements** of a:

- LINE are its **START-POINT** and **END-POINT**
- PIN-JOINT are two POINTS, which may be the **START-POINT** or **END-POINT** of a LINE or the **CENTER-POINT** of a CIRCLE.



Reference Elements for a Pulley

The columns in the dialog identify the element's:

References - the name of the Reference Elements. For example, PROFILE
Element Type - the type of element. For example, POINT
Element Owner - to which the **Reference Elements** is a child. For example, PART

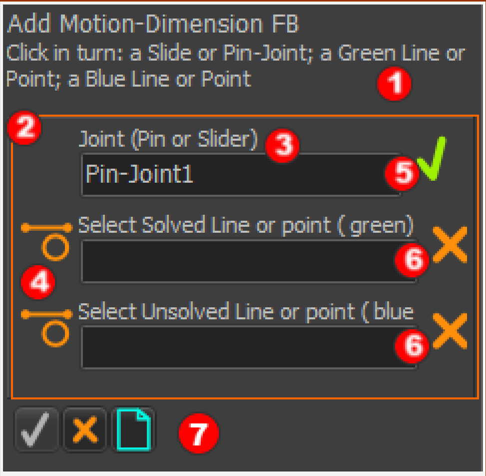
1.6.2 Command-Manager

Command-Manager

The **COMMAND-MANAGER** shows in the **PROJECT-EXPLORER** when you must select other elements to do a command.

Select the elements from the graphic-area, ASSEMBLY-TREE 261, and/or the SELECTION-WINDOW 257.

Command-Manager example :



Command-Manager -
Before you select elements

Add Motion-Dimension FB

❶ **Name of the Command and Short Instruction**

❷ **Selection-Boxes** - each **Selection-Box** has:

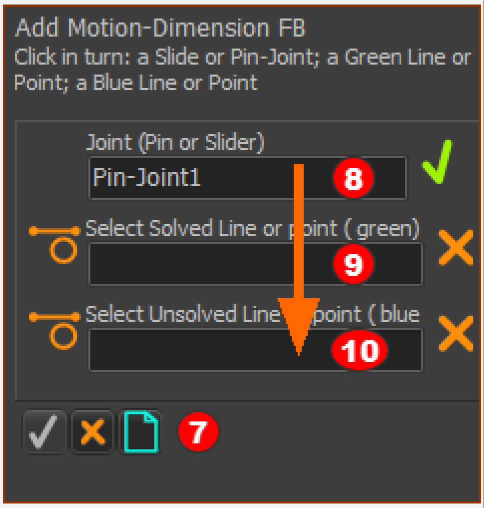
❸ **Above** : the name of the element-type you must select

❹ **Left** : an icon to help you identify the element-type

❺ **Right** : the icon when an element is in the selection-box

OR

❻ **Right** : the icon when an element is not in the selection-box



Command-Manager -
Select elements from top to bottom

continued:

To select the elements for each selection-box:

1. Click an element that is contextual to the top selection-box❽

The next selection-box❾ automatically becomes active.

2. Continue to click elements that are contextual to each selection-box

To complete the command:

When the bottom selection-box❿ has an element, the ❼ becomes colorized.

To complete the command:

1. Click ❼

OR

1. Right-click in the graphic-area (usually recommended).

To exit or start the command again:

1. Click ❼ to exit the command, but do not add the new element.

OR

1. Click ❼ to clear the elements from all selection-boxes. You must select each element again.

1.6.3 Element-Explorer

Element-Explorer

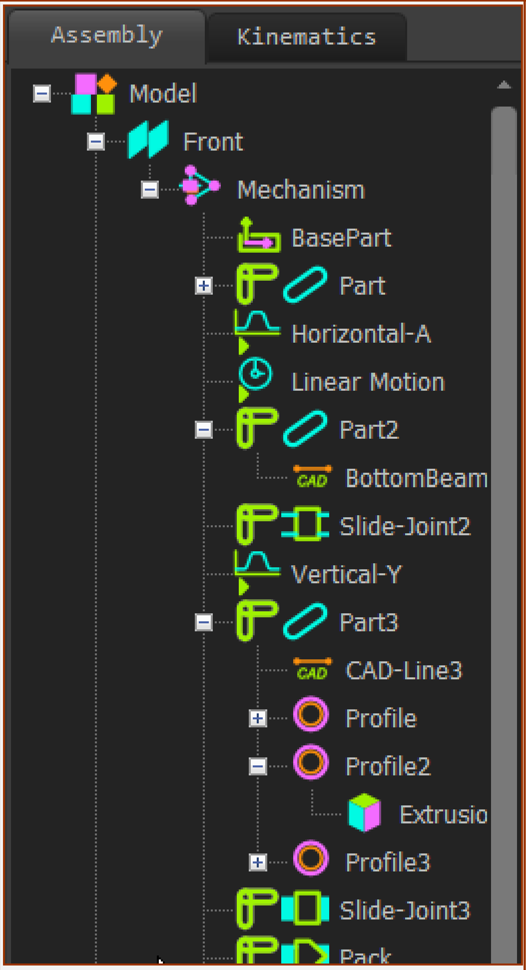
The **ELEMENT-EXPLORER** is at the bottom of the PROJECT-EXPLORER [257].

It shows most of the elements that are in the model.

It does not show sketch-elements or sketch-constraints.

There are three(3) **Trees** to explore.

Element-Explorer example:



Example Elements in the Assembly-Tree

ASSEMBLY-TREE [261]

It is a chronological and hierarchical list of the elements in the **MODEL** and **MECHANISM-EDITOR.**

KINEMATICS-TREE [262]

It shows when a **MECHANISM-EDITOR** is active.

It compiles a list of the kinematic-elements into kinematic-chains for the active **MECHANISM-EDITOR**.

GEOMETRY-TREE [267]

It shows when the **PART-EDITOR** is active.

It is a list of the constraints and dimensions that you have added to or between sketch-elements.

It does not list the sketch-elements.

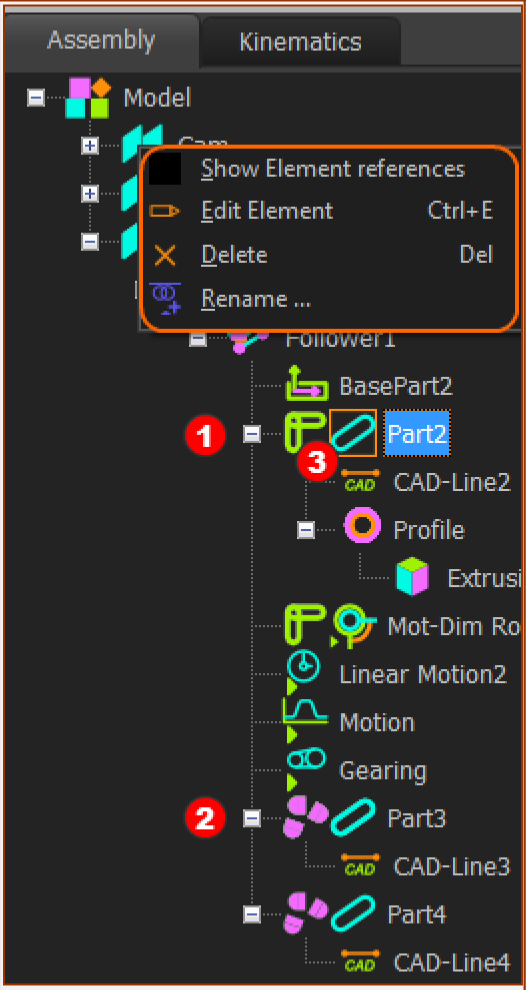
If you click an element in the **ELEMENT-EXPLORER** it becomes RED (default selection color) in the graphic-area.

1.6.3.1 Assembly-Tree

Element-Explorer - Assembly-Tree

The elements you add to the model are in the **ASSEMBLY-TREE.**

Each element has an **Element Icon** and an **Element Name.**



Assembly-Tree in Explorer-Explorer

The **ASSEMBLY-TREE** has a hierarchy. For example:

The **Add Mechanism-Editor** command adds a **MECHANISM** element as a child to a **PLANE** element.

Similarly, **PARTS, FUNCTION-BLOCKS, JOINT, GEARS, BELTS**, ... are children to the **MECHANISM** element.

Finally, **CAD-LINES, PROFILES, 3D-CAMS,** and **2D-CAMS** are children to **PARTS**.

PARTS

To the left of each **PART**, an icon identifies if the kinematic-chain to which it is associated, is solved, broken, or unsolved

❶ **Solved Part and solved kinematic-chain:** Mobility is zero AND all **PARTS** remain joined for all positions of the machine-cycle.

❷ **Solved but broken kinematic-chain**. Mobility is zero BUT **PARTS** cannot remain joined at all positions of the machine-cycle.

Unsolved kinematic-chain. Mobility is greater than zero(0).

Selected Elements

When you click an element:

- a **square**❸ shows around the icon
- the element shows in the **SELECTION-WINDOW** [257]

shortcut menu:

Right-click an element in the **ASSEMBLY-TREE**, to show a shortcut menu.

You can use the shortcut menu to:

- Show Element References [259]
- Edit Element
- Delete (element)
- Rename ... (element)

1.6.3.2 Kinematics-Tree

Element-Explorer - Kinematics-Tree

Note: If you cannot see or explore the **KINEMATICS-TREE**, click **Rebuild now** [37].
See also: Kinematic-Symbols [264], and Configure-Power Source/ Change Dyad Closure [266]

The **KINEMATICS-TREE** is the kinematic-structure of each kinematic-chain in the model.

We compile the **KINEMATIC-TREE** for you as you add Kinematic elements and Kinematic FBs to your model.

You can explore to see which elements are in each kinematic-chain. If you click on any kinematic-chain or element in the kinematic-structure, it will show as selected elements in the graphic-area.

You can use the **KINEMATIC-TREE** to Configure the Power Source [266] of each kinematic-chain.

This icon is at the top of the **KINEMATIC-TREE** to the left of the name **Mechanisms.**

Note: If you cannot see **Mechanisms**, or explore the **KINEMATICS-TREE**, click **Rebuild now** [37].

The structure of the Kinematics-Tree is as follows

⌧ MECHANISMS:

  ⌧ KINEMATIC-CHAINS

    ⌧ KINEMATIC SUB-ASSEMBLIES

      ⌧ KINEMATIC ELEMENTS

  ⌧UNSOLVED MECHS

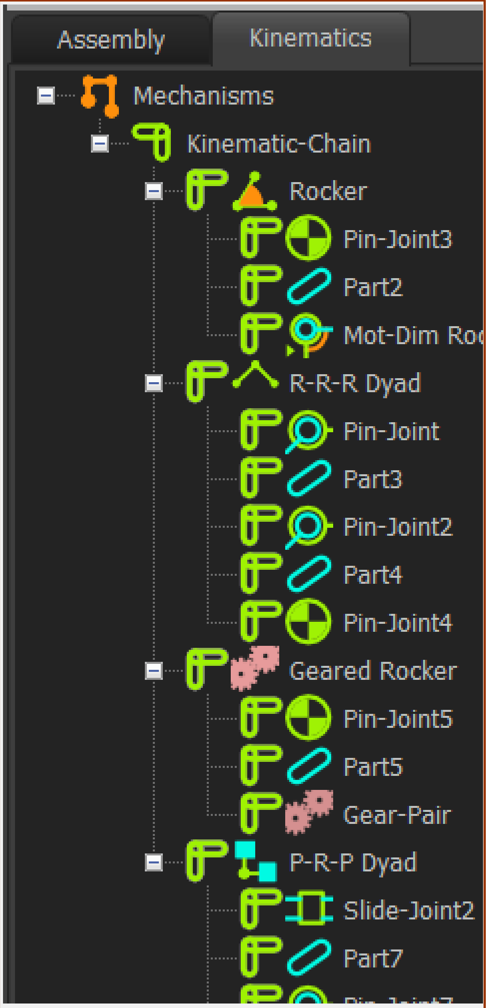
KINEMATIC-CHAIN

These kinematic-chains have a **MOBILITY** of zero(0). The kinematic-chain is **kinematically-defined**.

You can explore the kinematic-chain to find **KINEMATIC SUB-ASSEMBLIES** (see below) and **KINEMATIC ELEMENTS**.

UNSOLVED MECHS

These kinematic-chains have a mobility that is greater than zero(0). The kinematic-chains are not **kinematically-defined**.



Example: Kinematics-Tree in Element-Explorer

KINEMATIC-SUB-ASSEMBLIES (Solved Mechanisms ONLY):

These are compiled as:

1. Motion-Parts
   a. Rockers
   b. Sliders
2. **Motion-Paths**
3. Dyads - all dyads have **two** PARTS and **three** JOINTS

We represent Dyads for you with a letter for each of its joints:

R = **Revolute-Joint** = PIN-JOINT

P = **Prismatic-Joint** = SLIDE-JOINT

S = **Spherical-Joint** = BALL-JOINT

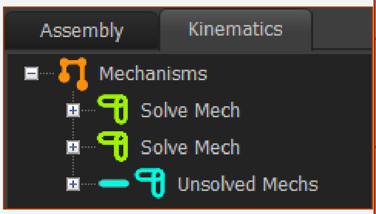
   a. RRR
   b. RRP
   c. RPR
   d. PRP
   e. RPP
   f. Ram-R
   g. Ram-P
   h. SSR
   i. SSP
4. Geared-Rockers
5. Rack-Pinion / Pinion-Rack
6. Geared-Pulley

1.6.3.2.1 Kinematic-Symbols

Kinematics-Tree Structure & Symbols

Symbol at top of **KINEMATICS-TREE**

TOP-LEVEL



Symbol	State	Mobility	
	Solved Mech	M = 0	The kinematic-chain: - is solved - is **kinematically-defined**
	Unsolved Mechs	M > 0	The kinematic-chain : - is not fully defined - cannot solve its kinematic-data

SUB-ASSEMBLY LEVEL

Kinematic Sub-Assemblies

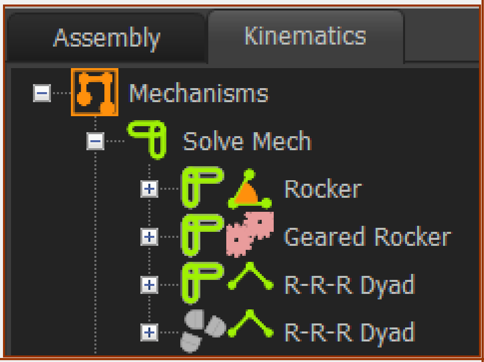
- Motion-Parts (Rockers, Sliders)
- Geared-Rockers
- Dyads
- Pulley-Rockers
- Rack-Pinions

Sub-assembly Symbols:

 : the kinematic-chain solves for the complete machine-cycle.

 : the kinematic-chain is solved now, but not for the complete machine-cycle.

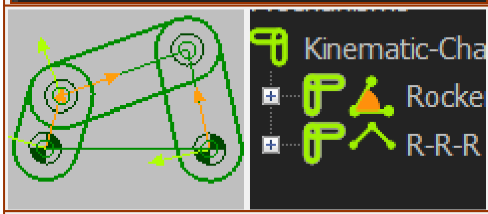
 : the kinematic-chain is not solved now, at this machine-angle.



Example.

Kinematic-Chain with four sub-assemblies:

- Rocker,
- Geared-Rocker
- R-R-R Dyad
- R-R-R Dyad



EXAMPLE 1 : A Dyad that never Breaks ...

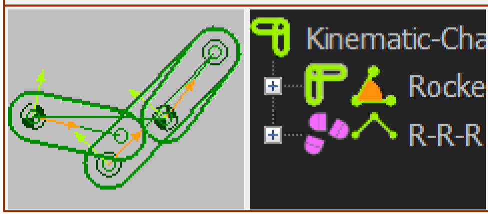
The Symbol is  :

The Dyad is solved for a machine-cycle (0-360 of the Master Machine Angle).

EXAMPLE 2 : A Dyad that Breaks ...

The Symbol is  :

The Dyad is solved now, but it does **not solve** for **ALL positions** of the machine-cycle (0-360 of the Master Machine Angle).



EXAMPLE 3 : -The Dyad is broken

The Symbol is  :

The Dyad is **not solved at this position** of the machine-cycle (0-360 of the Master Machine Angle).

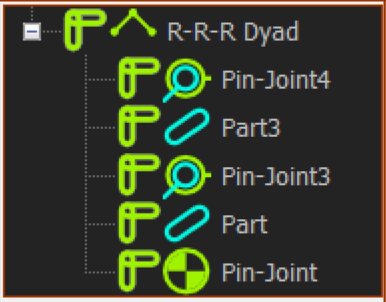
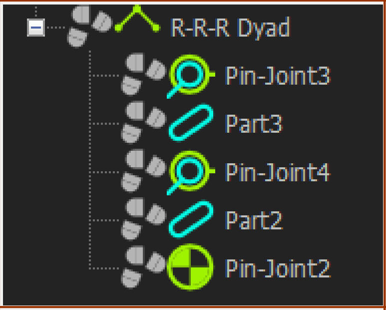
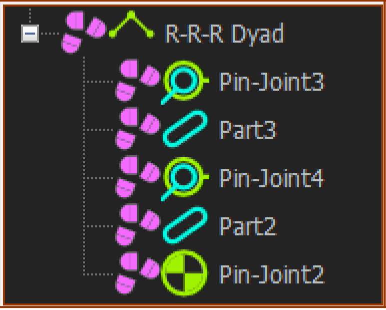
ELEMENT LEVEL

You can explore the fundamental kinematic elements of each kinematic-chain.

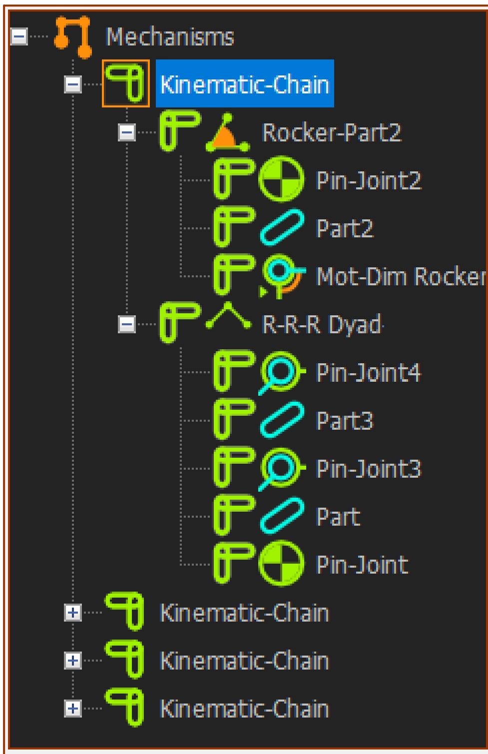
At this level you can see:

- Parts
- Joints
- Motion-Dimensions
- Gear-Pairs
- Pulley-Joints

The symbol, , also shows next to the elements in each sub-assembly.

	**Example 1 : 5-Elements in a Dyad** ***A Dyad that never Breaks...*** The symbol is to show the Dyad is solved for a machine-cycle. The symbol is next to each element in the dyad to show they are solved for a machine-cycle.
	**Example 2 : 5-Elements in a Dyad** ***A Dyad that never Breaks...*** The symbol is to show the dyad does not solve for a period in the machine-cycle. The symbol also shows next to the elements in the dyad that do not solve for a period in the machine-cycle.
	**Example 2 : 5-Elements in a Dyad** ***A Dyad that never Breaks...*** The symbol is to show the joints in the Dyad do not solve at this instant in the machine-cycle. The symbol also shows next to the elements in the dyad that do not solve at this instant in the machine-cycle.

A typical Kinematics-Tree

	This Kinematics-Tree has four(4) Kinematic Defined Chains, identified by the icon. A **Rocker** is expanded to show three elements: • PIN-JOINT, PART, MOT-DIM ROCKER The **R-R-R Dyads** is expanded to show it is assembled from: • Three(3) PIN-JOINTS , two(2) PARTS **Dyad and Joint Symbols:** This symbol is to the left of a Dyad and Joint when it solves for the machine-cycle. This symbol is to the left of a Dyad and Joint if a Joint does not solve for period of the machine-cycle. This symbol is to the left of a Dyad and Joint if a Joint does not solve at this instant of the machine-cycle.

1.6.3.2.2 Configure Power Source / Change-Dyad-Closure

Kinematics-Tree - shortcut menu

Configure Power Source

See also : Forces toolbar > Configure Power Source [193]

To open the **Configure Power-Source dialog:**

1. Click a **kinematic-chain** so that a Red or Orange square encloses the icon
2. Right-click to show 'CONFIGURE POWER SOURCE'
3. Left-click the 'CONFIGURE POWER SOURCE' text to open the dialog**.**

Change Dyad Closure

See also : Kinematic elements toolbar > Change Dyad Closure [101] - that tool is more convenient and easier to use.

This command rearranges the assembly of the two PARTS in **RRR, RRP, RPP dyads**.

To change the closure of a dyad from the **KINEMATICS-TREE**:

1. Click the **Dyad** so that a Red or Orange square encloses its icon
2. Right-click the icon to show **Change Dyad Closure** next to you mouse-pointer
3. Click the **Change Dyad Closure** text to change the closure of the Dyad

The **SELECT ELEMENTS DIALOG** shows because you select a PART.

4. Click a PART in the **SELECT-ELEMENTS DIALOG**
5. Close ✅ to close the **SELECT-ELEMENTS DIALOG**
6. Click ✅ in the **COMMAND-MANGER** to change the closure of the **Dyad**

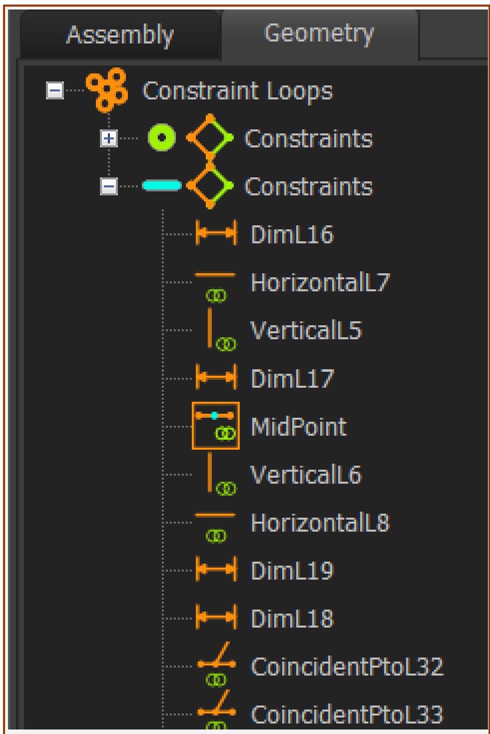
1.6.3.3 Geometry-Tree

Element-Explorer - Geometry-Tree

Note: If you cannot explore the **GEOMETRY-TREE**, click Edit menu > Rebuild now [37].

The **GEOMETRY-TREE** replaces the **KINEMATICS-TREE** when you start the **PART-EDITOR**.

Example Geometry-Tree



Example: Geometry in Element-Explorer

What is a Constraint-Loop?

A **Constraint-Loop** (labeled Constraints) is a list of the constraints that are between a set of sketch-elements.

If you add a Constraint between sketch-elements that are in different Constraint-Loops, they combine to give one Constraint-Loop.

For example:

- Add Horizontal Constraint to Line1 = Constraint Loop #1
- Add Horizontal Constraint to Line2 = Constraint Loop #2

There are 2 × Constraint Loops.

- Add a Horizontal Constraint between a Point in Line 1 and a Point in Line 2, =

Now there is 1 × Constraint Loop

Constraints and the Selection-Window

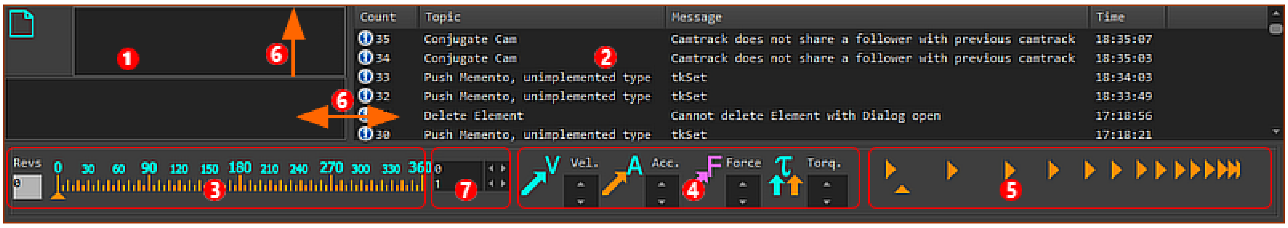
	If you: • **Click** a **LINE** : the **LINE** shows in the **SELECTION-WINDOW** • **SHIFT+Click** a **LINE** : the **LINE** **AND** any **CONSTRAINTS** that have been added to the **LINE** show in the **SELECTION-WINDOW**.

1.7 4: Feedback Area

❹ Feedback-Area

The **Feedback-Area** is below the graphic-area.

It has five areas.



❶ Extended Hints

When you move your mouse over a command icon in a toolbar, the tool-tips shows in the **EXTENDED HINTS** box.

Note: To see all of the Extended Hint, you may need to drag the separator bar❻ upwards, and/or to the right❻, a short distance.

You do not need to enable HINTS AND TOOL-TIPS.

See Application Settings 289

❷ Message Area

The most recent message is at the top of the list.

There are four columns in the **MESSAGE AREA**.

COUNT :	The message number since you last cleared the **Message-Area** of messages.
TOPIC :	Importance Level: – Information; – Warning; – Error - save your work now!
MESSAGE :	Relates to the command. This is the message you should read!
TIME :	When the message was displayed.

To clear messages from the **MESSAGE AREA**:

1. Right-Click, with your mouse-pointer in the **MESSAGE AREA**
2. Click **CLEAR MESSAGES** in the shortcut menu

Message Sounds:

If you enable sounds in Application Options | Accessibility tab 289, you hear a **honk** with each new message.

❸ Master-Machine-Angle (MMA) ; Revs ; Machine-Angle-Indicator

The **MASTER-MACHINE-ANGLE** is the virtual axis, and the virtual input to all kinematic-chains in the model. You can imagine it as a drive shaft that rotates one time per machine cycle.

MASTER-MACHINE-ANGLE (MMA) :	Range 0 – 360, equal to one **MACHINE-CYCLE**
REVS :	Number of **MACHINE-CYCLES**
MACHINE-ANGLE-INDICATOR ❼ :	**Read-write** - the exact machine-angle

To edit the **MASTER-MACHINE-ANGLE:**

EXACTLY : use the **MACHINE-ANGLE INDICATOR**❼ to:

MOVE TO A MACHINE ANGLE :	Enter a Machine-Angle between **0** and **360.**
INCH :	Use the **SPIN-BOX** to increase or decrease the **MACHINE ANGLE** by the **SPIN-INCREMENT**. Edit the **SPIN-INCREMENT** (by factors of 10) to change the **INCH-STEP**.
JOG :	**Press & Hold the Spin-Box** to continuously increase or decrease the **MACHINE-ANGLE**.

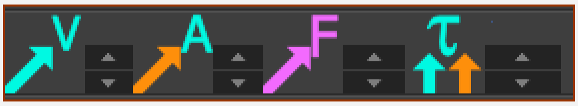
APPROXIMATELY : Drag in the **MMA**❸

Mouse-button-down and move you mouse to the left or right, with your mouse-pointer INSIDE the **MASTER-MACHINE-ANGLE** area.

The orange triangle (▲) below the **MMA** moves with your mouse-pointer.

See also : Run menu 44

❹ Kinematic and Kinetostatic Vector Scale Buttons



Use the **VELOCITY**, **ACCELERATION**, **FORCE**, and **TORQUE** scale buttons to increase or decrease the length of vectors in the graphic-area.

The buttons do **NOT** change the values of the vector.

Velocity (V) and Acceleration (A) Vector buttons

Click the up /down arrow-buttons next to:

- **V** to increase or decrease the length of **Velocity** vectors
- **A** to increase or decrease the length of **Acceleration** vectors

See Point Properties dialog 444 to display the **Velocity Vector** and **Acceleration Vector** of a moving **POINT**

Force (F) and Torque (τ) Vector buttons

Click the up /down arrow-buttons next to:

- **F** to increase or decrease the length of **Force** vectors
- τ to increase or decrease the length of **Torque** vectors

See Force Vectors: Calculate 190 and **Force Vectors: Display** 49 to display **Force** and **Torque Vectors** at joints.

See Configure Power Source 480 to make sure the Power flows through the model as intended.

❺ Animation-Speed Slider

Use the **ANIMATION SPEED SLIDER** to speed-up or slow-down the **Animation Speed** of the model.

> Mouse-button-down and move you mouse to the left or right, with your mouse-pointer **INSIDE** the **Animation-Speed Slider** to change the animation speed.

The **Animation-Speed** does not change the kinematic or force analysis, or motion synchronization between kinematic-chains.

To edit the simulation speed or machine-speed: **see** **Edit menu > Machine-Settings** [39] **>** CYCLING PARAMETERS.

See also: **Run >Cycle** [44]

1.8 5: MotionDesigner

❺ MotionDesigner

Use **MotionDesigner** to add, name, and design motions. Each motion graph has a unique name. Identify its name in its **Motion name-tab**.

Use a **MOTION FB** to link a motion that you have designed in **MotionDesigner** to a **MOTION-PART** or **MOTION-POINT** in **MechDesigner.**

To Link a Motion to a Motion-Part

STEP 1: Add a Motion FB to the graphic-area

1. Click Kinematics FB toolbar [142] > Motion FB
2. Click in the graphic-area

The **MOTION FB** is now in the graphic-area.

STEP 2: Open the Motion FB dialog

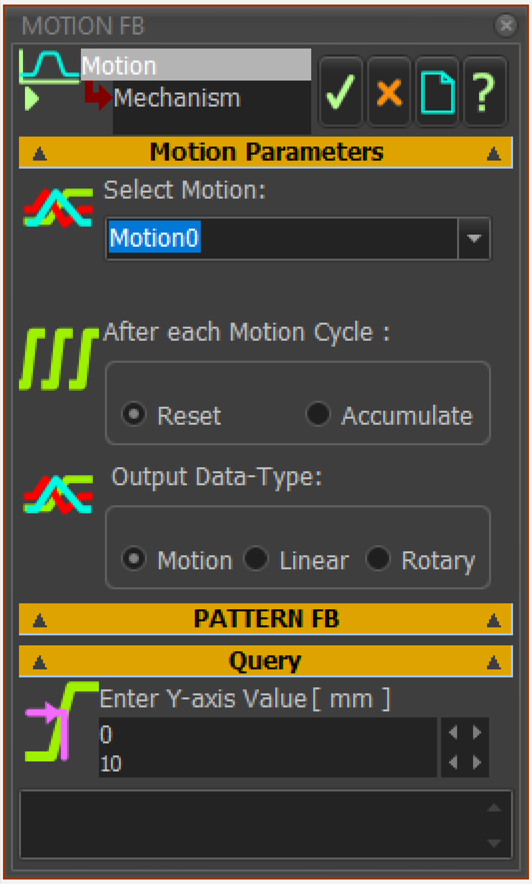
1. Double-click the **MOTION FB** in the graphic-area or **ASSEMBLY-TREE**

OR

1. **See How to Open a dialog** [513]

The **MOTION FB DIALOG** is now open.

STEP 3: Edit the Parameters



MOTION PARAMETERS

SELECT MOTION

In the **SELECT MOTION** list:

1. Select a Motion from a list of the **Motion name-tabs**

Now, the **Motion name** has a link with the **MOTION FB.**

AFTER EACH MOTION CYCLE

- **RESET OUTPUT** : **Default,** used most frequently.
- **ACCUMULATE OUTPUT** : Select when the motion is a progressive motion - for example, the motion is indexing.

See more details : Motion FB dialog [375]

OUTPUT DATA-TYPE

- **MOTION** : Default option, use in most cases
- **LINEAR OR ROTARY** : Select when you connect the **MOTION FB** to a **MOTION-PATH FB** or **MATH FB.**

PATTERN FB

Not normally used.

QUERY

Optionally: enter a **Y-axis** value to find the corresponding **X-axis** value or values in the SELECTED MOTION.

STEP 4: Click ✅ to close the Motion FB dialog.

STEP 5: Connect wires to the input-connector and from the output-connector.

1. Connect a wire to the input-connector of the **MOTION FB**

Usually, (but not always), connect a wire from the output of a **LINEAR-MOTION FB** or a **GEARING FB** to the input-connector of the **MOTION FB**.

When you cycle the model, the motion-data at its input-connector control the **X-axis** value for the **Motion** you select in **STEP 3**.

2. Connect a wire from the output-connector of the **MOTION FB**

Usually (not always), connect a wire from the **MOTION FB** to the input of a **MOTION-DIMENSION FB** or a **MOTION-PATH FB**

When you cycle the model, the motion-data at the output-connector is the **Y-axis** value that corresponds to the **X-axis** value at its input, for the **Motion** you select in **STEP 3.**

1.9 6: Memo

❻ Memo dialog

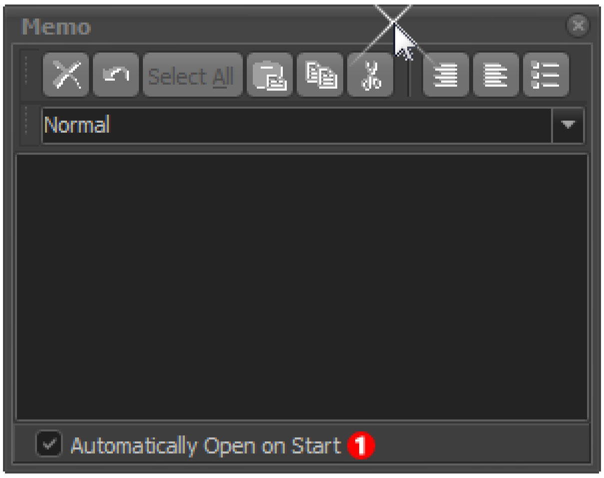
The **Memo** is a **Windows NotePad®** style interface.

The **MODEL-EDITOR** and all **MECHANISM-EDITORS** can have their own **Memo**. We save the Memo for you with the model file.

In the default case, the **Memo** opens each time you start **MechDesigner**.

To disable the **Memo** when you start **MechDesigner**:

1. Clear the **Automatically Open on Start** check-box, at the bottom of the interface ❶**.**



Memo dialog

To open a **Memo** at any other time.

1. Right-click the **MODEL NAME-TAB** or any **MECHANISM NAME-TAB**
2. Click **Open Memo** in the shortcut menu.

1.10 Dialogs

Dialogs

To build a model:

STEP 1:	Add an ELEMENT	- see **Model Editor** [65] ; **Mechanism Editor** [72] ; **Part-Editor** [213]
STEP 2:	Open the ELEMENT'S DIALOG	- see **How to open a dialog** [513]
STEP 3:	Edit the ELEMENT'S PARAMETERS	- see **How to edit a Parameter in a dialog** [517]
STEP 4:	Close the DIALOG	Close the dialog before you open a different dialog.
Do Steps 1 to 4, again, and again, ...		

Dialogs

GENERAL DIALOGS	
RENAME [279]	Replace Old Name with New Name
MODEL / MECHANISM [281]	Lighting, Synchronize **CAD-Solids**
APPLICATION SETTINGS [284]	General, Number-Format, Auto-Profiles, Accessibility, Colors, ...
MACHINE SETTINGS [291]	Machine Speed, # Steps in Machine-Cycle, Units
ELEMENT PROPERTIES [498]	Read-only - an element's Motion and/or Force properties
MODEL-EDITOR	
ADD OR EDIT PLANE [296]	Edit Angle of PLANE or Offset of PLANE
PROFILE / EXTRUSION [298]	Edit a an EXTRUSION - the MD-SOLID
CAD: DXF, SOLIDWORKS	
DXF-ELEMENT [294]	Select a different DXF File, specify the DXF Units of the new DXF element
CAD-LINE [301]	
> DXF TAB [306] >> DXF LAYER EDITOR [309]	Import DXF File, Convert Entities to MD Sketch-Elements.
> SOLIDWORKS [302]	> CAD-SOLID - Import SOLIDWORKS file-type
> MASS PROPERTIES [304]	> User Mass-Properties, SOLIDWORKS Mass-Properties
> STL IMPORT [310]	> Import an STL file
> DISPLAY OPTIONS [313]	> Edit the Color, Transparency, ...
CAMS: 2D & 3D	
2D-CAM [327]	2D-Cam:
> PARAMETERS [329]	> Edit the Start-Angle and Range (defaults are 0 and 360), Enable Lifetime, Edit Safety-Factor.
> DISPLAY [332]	> Display Colour Profile as: Cam-Profile, Pressure-Angle, Contact-Force, or Shear-Stress
> ROLLER LIFE [336]	> Edit the Roller bearing, enter factors for ISO 281 calculations
> CAM LIFE [347]	> Edit the Cam steel, Quality, Heat-Treatment, and Enter its Hardness

CONJUGATE CAM FB 353	Required for Force Analysis, Cam-Life, Roller-Life with Conjugate-Cams and Groove-Cams.
CAM-DATA FB 357	Plot Contact-Force, Hertzian Shear-Stress, Radius-of-Curvature, Pressure-Angle, or Sliding-Velocity
CAM-COORDINATES 361	Calculate and Export 2D-Cam Coordinates, Save as STEP, TXT, DXF
3D-CAM 315	Calculate and Export 3D-Cams - Barrel, Cylindrical, Globoidal.
FUNCTION-BLOCKS	
LINEAR-MOTION FB 371	Output is 0 - 360, again and again - a machine 'clock'.
GEARING FB 372	Modify motion-values with a Linear function.
MOTION FB 374	Link a Motion in MotionDesigner
MOTION-DIMENSION FB 377	Edit the Base-Value Angle or Position of a Motion-Part.
MOTION-PATH FB 380	Edit the starting position of a Motion-Point, Control the length of a sketch-loop
POINT-DATA FB 169	Measure Point's Position, Velocity and Acceleration relative to the Mechanism-Plane.
MEASUREMENT FB 166	Measure Linear or Angular Position, Velocity and Acceleration.
GRAPH FB 386	Plot up to 4 graphs as a function of one X-axis
> GRAPH SETTINGS 389	> Graph Display
SPRING FB 439	Define a force for a Spring
FORCE-DATA FB 442	Measure Force or Torque
CAD CONTROL FB 394	Control SOLIDWORKS mates from MechDesigner
MAGNETIC-JOINT 457	Glue Profiles/Curves together
MATH FB 396	User defined functions
STATISTICS FB 404	Statistics of a motion-values or force-values
POLYNOMIAL FIT FB 406	Fit Polynomials to data at its input-connector
PARAMETER-CONTROL FB 411	Control the length of a 'static-dimension', Extrusion-Depth, Extrusion Offset with motion-values at its input-connector.
CONTINUOUS CRANK FB 437	Make a Crank rotate fully with a modulated speed, as a function of the motion of a different part in the kinematic-chain.
MODELING TOOLS	
DESIGN-SET 431	Add different dimensions and parameters to one dialog
BRIEFCASE FB 422	Put other FBs in a Briefcase - to make you model easier to review
MAKE MOVIE 495	Produce a GIF file or series of PNG, JPG , of BMP images
POINT-CLOUD 425	Import XY data, convert to a curve
PATTERN 414	Copy **CAD-SOLIDS** and/or **MD-SOLIDS** multiple times in space and time
OTHER DIALOGS	
POINT PROPERTIES 444	Edit Position of a **POINT**, Display Velocity and Acceleration Vectors
BLEND-CURVE 449	Edit the Angle, Curvature, and Curvature-Rate at the **START-POINT** and **END-POINT** of a **BLEND-CURVE**
BALL-JOINT 456	3D Joint for Spatial Mechanisms, edit its offset from the Mechanism-Plane.
GEAR-PAIR 462	Edit the module, number-of-teeth, mesh, ...

RACK-PINION 470 / BALL-SCREW	Edit the module, number-of-teeth, ...
PULLEY 474	Number of Teeth on the Pulley
KINETOSTATIC: SERVO-MOTOR AND GEARBOX SIZING 483	Size a Gearbox and Servo-motor for the Application.
CONFIGURE POWER SOURCE 480	To move a motor or other power source to the correct joint or cam
DIMENSION 448	Dimension Value
MISCELLANEOUS	
TUTORIAL VIDEOS 507	Play Tutorial Video
EXAMPLE MODELS	Help menu
SELECT ELEMENTS 504	Choose one or more elements for a list of elements
DEPENDENT ELEMENTS 505	Show those elements that you also delete
VIEW REFERENCES 506	Elements that are referenced by another elements

1.10.1 Dialog: Rename Element

Rename

You can rename all elements.

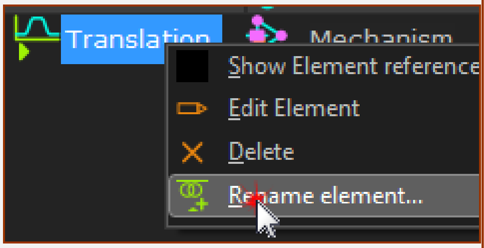
After you rename the element, you can identify the element-type with the icon that is to the left of the element-name.

How to Rename an Element:

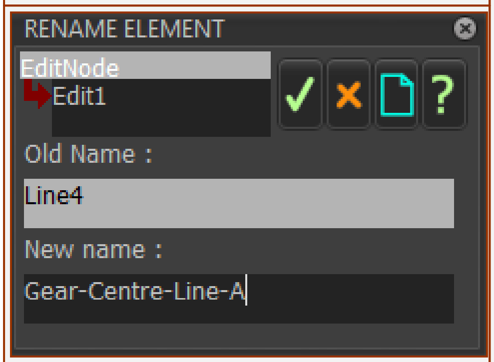
You can rename an element with three(3) methods:

METHOD 1: Rename Element dialog-box

Use this method when you can select the element in the graphic-area, and/or **ELEMENT-EXPLORER**.



Selection-Window: shortcut menu



Rename dialog

STEP 1: Select the element:

1. Click an element, in the graphic-area or the **ELEMENT-EXPLORER**.

The element is now in the **SELECTION-WINDOW**.

In the **SELECTION-WINDOW**:

2. Right-click the element that you want to rename
3. Click **Rename element** in the shortcut menu

The **RENAME ELEMENT DIALOG** is now open.

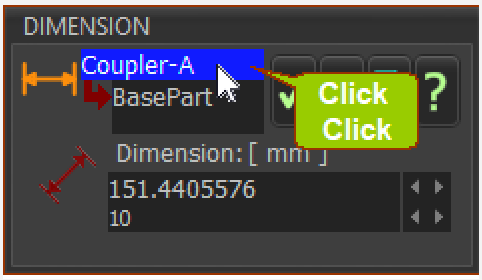
STEP 2: Enter a New Name to replace the Old Name:

1. Enter a **New Name** (Top-Tip: add a **Letter** to the end of the new-name. For example, **NewName-A** and NOT **NewName-1**)

In this case, the new name for the **LINE** is: **Gear-Centre-Line-A**

2. Click ✓ to close the **RENAME ELEMENT DIALOG**.

METHOD 2: Name the Element with its dialog-box



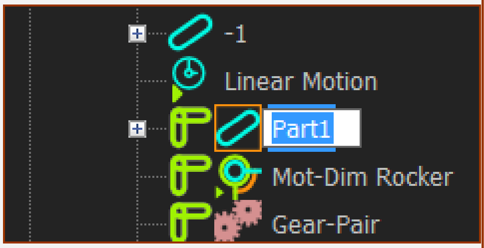
Rename element:
An open dialog

Use this method in an open dialog:

1. Double-click the element-name at the top of the dialog < see image
2. Enter a new element-name E.g. **Coupler-A**
3. Press the Enter key (↵) on your keyboard.

Unfortunately, not all element-types have a dialog.

METHOD 3: Rename elements in the Element-Explorer

<table>
<tr>
<td>

Rename element:
From the Assembly-Tree</td>
<td>Do the Windows® method:
1. Click, pause, and click the element-name in the ELEMENT-EXPLORER or the SELECTION-WINDOW
2. Enter a new element-name
3. Press the Enter key (↵) on your keyboard
Unfortunately, not all element-types show in the ELEMENT-EXPLORER or the SELECTION-WINDOW.</td>
</tr>
</table>

1.10.2Dialog: Model / Mechanism Options

Model / Mechanism Options

See [**known bugs - 16.1.268**] [10]

How to open the Model Options dialog

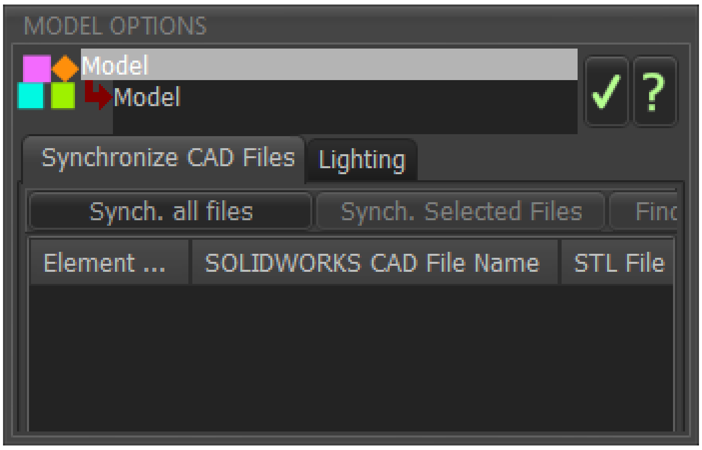
1. Double-click the **Model** or **Mechanism** element in the **ASSEMBLY-TREE**

OR

1. Right-click the **Model** or **Mechanism** element at the top of the **ASSEMBLY-TREE**
2. Click **Edit element** in the shortcut menu

The **MODEL OPTIONS DIALOG** is now open.

Model Options dialog.



Model Options dialog

There are two tabs.

Synchronize CAD Files

The list of **CAD-SOLID** files that you have imported onto a **CAD-LINE** show in the box (see image below).

There are three buttons you can use to import **CAD files** again if they are not the latest **CAD files.**

Background

You use the CAD-Line dialog | Solids tab [302] to import a CAD file (a SOLIDWORKS document or an STL file) as a CAD-**SOLID**.

CAD files have the ***time-stamp*** when you used your CAD software to save them to disc. When we import a CAD file, we link that ***time-stamp*** with the **CAD-LINE**.

When you open this **MODEL OPTION DIALOG**, we compare for you the:

- **Active Time-stamp** of the CAD file at the time it was linked to the **CAD-LINE**
- **Current Time-stamp** of the CAD file with the time-stamp of the CAD file-name that is on disc.

CAD-File box:

Element Name

CAD-LINES that have a link to a CAD file. Each CAD-LINE has a status icon:

- the CAD file on disc is newer than SOLIDWORKS document on the CAD-LINE.

- the CAD file is not on disc in the place expected

- the 'time-stamp' is the same as that of the CAD file on your hard-disk.

SOLIDWORKS CAD File Path

The CAD file is either:

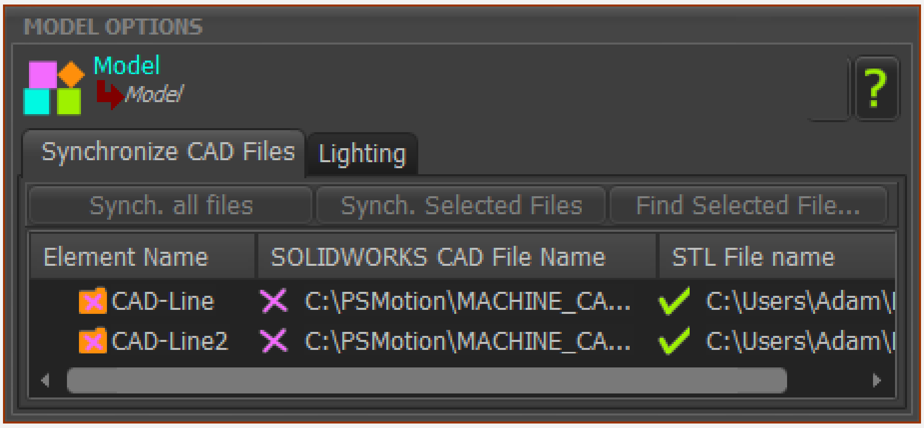
Not found on your hard-disk

Found on your hard-disk

STL File name

The source file is the STL file that is linked to the CAD-LINE.

We check for you that it is in the path, as expected.



Model Option dialog

The control buttons allow you to re-import or find SOLIDWORKS files if they are out-of-date.

Control Buttons:

Synch. all files

1. Click **Synch. all files** button to check you have the most up-to-date SOLIDWORKS files.

The files import again, if needed.

Synchronize Selected Files

1. CTRL+Click each CAD-LINE, in the Element Name list.
2. Click **Synch. Selected Files**

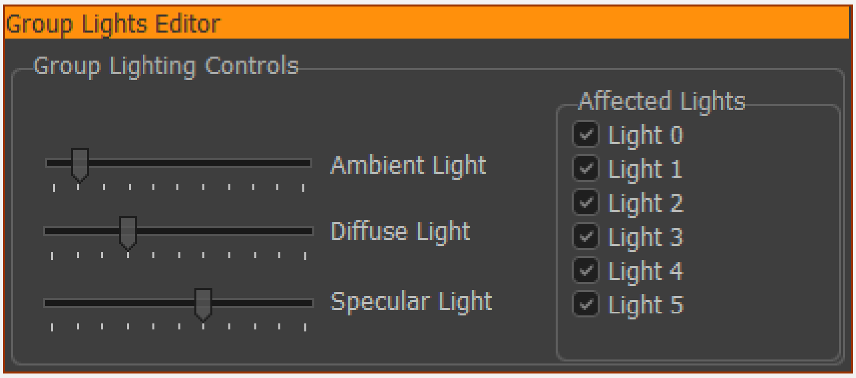
MD re-imports the files again

Find Selected File

1. Click a CAD-LINE in the list
2. Click **Find Selected File...** button
3. Browse to find a file that **MechDesigner** cannot find.

Lighting

GROUP LIGHTS EDITOR



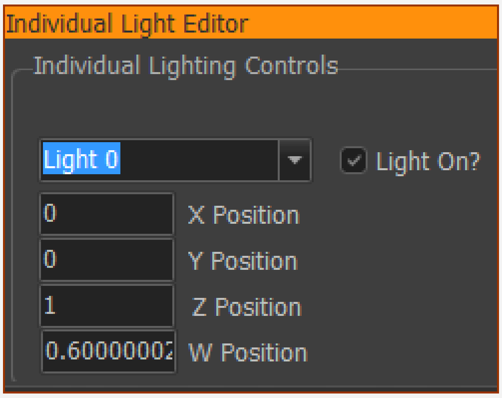
Model Options > Lighting tab

The slider controls those ***Affected Lights*** that have a check in the check-box

1. Click each ***Affected Light*** check-box to make lights active.
2. Use the sliders to change their:
   - Ambient, Diffuse, and Specular Light Levels
2. Move your mouse-pointer over the graphic-area to update the lighting

Note: It is often better to decrease ***Ambient Light Levels***, and increase ***Diffuse*** and ***Specular Light Levels*** to view ***Solids*** and SOLIDWORKS Parts. I often set one light at 1-1-1 - over my right-hand shoulder!

INDIVIDUAL LIGHT EDITOR



To turn on or off each light and to control its source point:

1. Use the drop-down box to select a light (Light 1 to 6)
2. Click the check-box to turn the light that is in the drop-down list on or off.
3. Enter numbers into the X, Y, Z and W Position boxes to change the direction of the light in the drop-down box.
   - ±X, ±Y ±Z directions for each light
   - W - not sure how this works!
4. Move your mouse-pointer over the graphic-area to update the lighting

An External link to Light : Good Luck!

1.10.3Dialog: Application-Settings

Application-Settings

Use the **APPLICATION-SETTINGS DIALOG** to edit the working environment.

We save for you the **APPLICATION-SETTINGS** when you exit **MechDesigner** to two files:

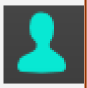
- <LocalAppData>\Inifiles\MechDesigner.INI
- <LocalAppData>\Inifiles\MechDesigner.XML

Delete the two files to reset **MechDesigner** to its factory-settings.

<LocalAppData> is typically C:\users\username\AppData\Local\PSMotion\

See also: Application-Settings, Themes, Styles, and the Save and Load buttons [510]

How to open the Application-Settings dialog

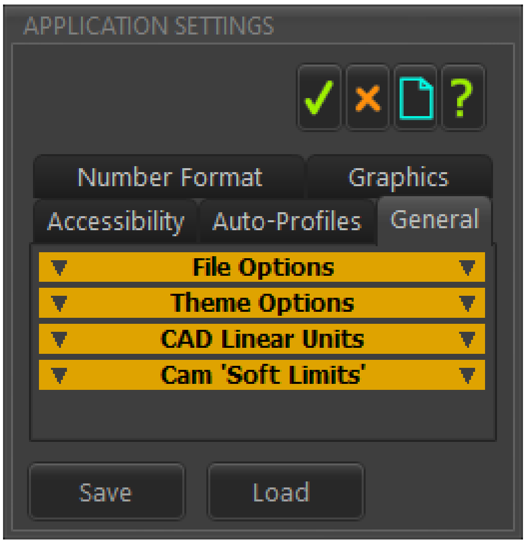


1. Click Edit toolbar > Application-Settings

OR 1. Click Edit menu > Application-Settings [41]

The **APPLICATION SETTINGS DIALOG** is now open.

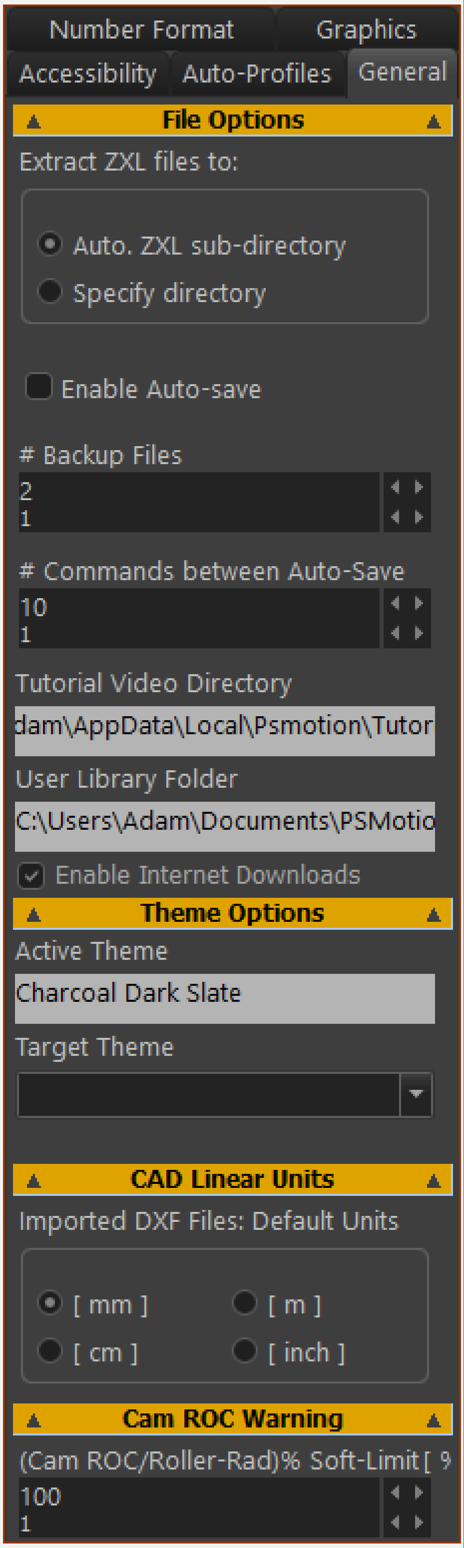
Application-Settings dialog:



Application Settings dialog

Application-Settings tabs

General



FILE OPTIONS **See also: ZXL file-type.**

EXTRACT ZXL FILES TO:

⦿ **AUTO. TO ZXL SUB-DIRECTORY**: Automatically, add a sub-directory to that of the ZXL file in which to extract the files that are in the ZXL file.

⦿ **SPECIFY DIRECTORY**: Specify a path in which to extract files that are in the ZXL file.

☑ **ENABLE AUTO-SAVE** check-box:

Enable to automatically save the model to overwrite the model file on disc

Note: Before **Auto-Save** becomes active, the model **must** have a file-name.

 # OF BACKUP FILES

Automatically make a backup of the models files. Backup file-type: CXL.1, CXL.2, ... CXL.#; and MTD.1, MTD.2, ...MTD.#

 # OF COMMANDS BETWEEN AUTO-SAVE

ENABLE AUTO-SAVE is to save after **# OF COMMANDS BETWEEN AUTO-SAVE**. For you, we:

1. Rename model file-name.CXL and .MTD, to file-name.~CXL and file-name.~MTD
2. Save the model to file-name.CXL and file-name.MTD

TUTORIALS VIDEO DIRECTORY - click the path to specify a path for **TUTORIALS VIDEOS** - **see** **Tutorials** [28]

USER LIBRARY FOLDER - click the path to specify a **USER LIBRARY FOLDER** - see **Import Library Files** [32]

ALLOW INTERNET DOWNLOADS : Enable to allow downloads.

THEME OPTIONS

ACTIVE THEME: After you select a **TARGET THEME**, the box shows the **Theme**.

TARGET THEME: Select from:

- **CHARCOAL DARK STYLE (DEFAULT)** , **WINDOWS**, **AQUA LIGHT SLATE**, **WINDOWS10 DARK**, **TABLET DARK**, **SLATE CLASSICO**, **WINDOWS 10 SLATE GRAY**, and **WINDOWS 10**.

Note: Mostly, in this help documentation, the **Theme** is **CHARCOAL DARK SLATE.**

CAD LINEAR UNITS

IMPORTED DXF FILES: UNITS

The scale of a **DXF-DRAWING** are 1:1 when the DXF units you select are equal to those units of the imported **DXF-DRAWING**.

See also: Edit DXF-Element 294

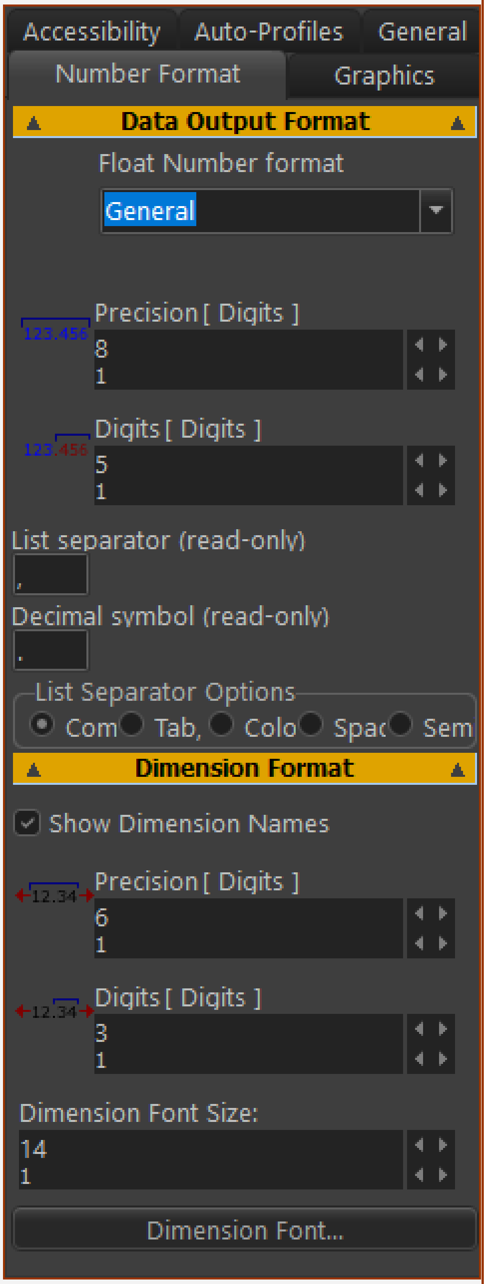
CAM ROC WARNING

If | Radius of the Cam Profile | < % × Radius of Cam-Follower Roller

- The Cam profile is more likely to be **Undercut**
- The Cam profile becomes **red** in the graphic-area.

It is a **WARNING ONLY** - a soft-limit.

Number Format



Note: Precision of calculations do not change if you edit these parameters.

DATA OUTPUT FORMAT

Data-format for data you save from a GRAPH FB or CAM-COORDINATES FB.

- FLOAT NUMBER FORMAT: **See Internet Link here about number formats**
- PRECISION and DIGITS - see below

Regional Characters as Separators

LIST SEPARATOR (READ-ONLY)- the active regional character that separates values.

For example, “,” in a CSV file.

DECIMAL SEPARATOR (READ-ONLY) - the active character that separates integer and fractional parts of a number.

For example, “.” in 12.3456

List Separator Options:

Select a separator to use for files that you **EXPORT**

DIMENSION FORMAT

☑ SHOW DIMENSION NAMES check-box

Top-tip: Rename the DIMENSION if the DIMENSION is in a DESIGN-SET 431 - **see Rename element** 279

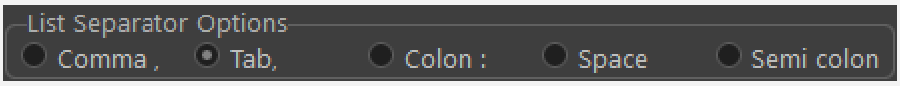
Data-format of dimensions, in the graphic-area

- PRECISION and DIGITS - see below
- DIMENSION FONT SIZE - experiment (typically 14)
- DIMENSION FONT - the Windows font picker opens. Experiment.

Notes:

	PRECISION: The number-of-digits in the - before and after the decimal mark. **DIGITS**: The number-of-digits after the decimal separator. Usually, set : **PRECISION ≥ DIGITS +3,** . However, it depends on your model requirements.

LIST SEPARATOR OPTIONS

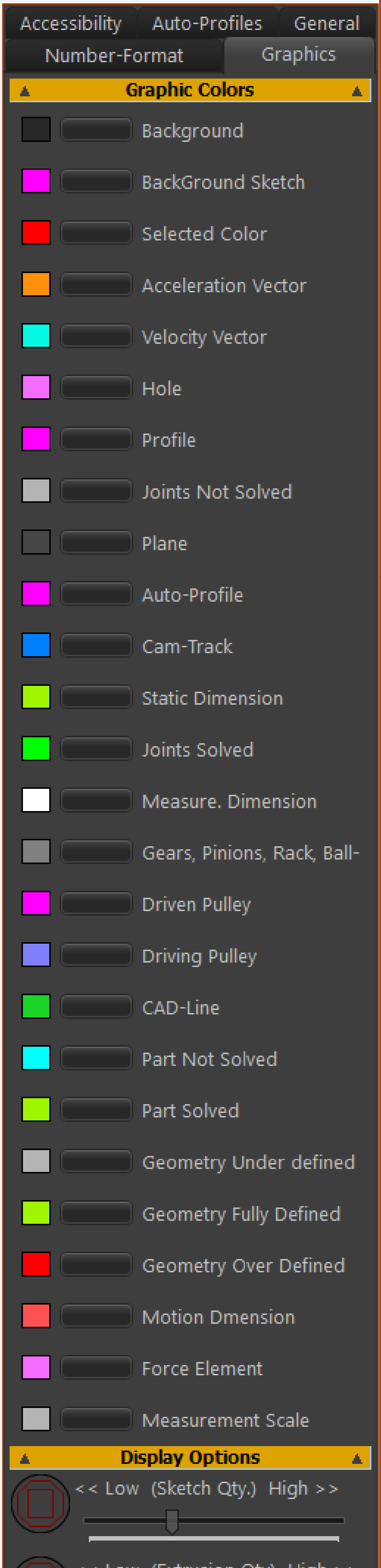


Comma (,) Tab ( ), Colon (:), Space ( ), Semi-colon (;)

PRECISION AND DIGITS

Example ~ 1234.56789		DIGITS			
		2	4	6	8
PRECISION	2	1.2E03	1.2E0003	1.2E3	1.2E3
	4	1,235.00	1,234.0000	1,234.000000	1,235.00000000
	6	1,234.57	1,234.5700	1,234.570000	1,234.57000000
	8	1,234.57	1,234.5679	1,234.567900	1,234.56790000

Graphics tab



DISPLAY COLORS

Edit the color of the symbols in the graphic-area.

Standard colors:

- **Part Not Solved** (equivalent term is : **not kinematically-defined**) is blue
- **Part Solved** (equivalent term is: **kinematically-defined**) is a green

XY-axes in the **MECHANISM-EDITOR** and **PART-EDITOR** are:

- Orange: X–axis
- Green: Y–axis

XYZ-axes (**RGB**) at the Origin in **MODEL-EDITOR**:

- Red: X–axis,
- Green: Y–axis,
- Blue: Z-Axis

DISPLAY OPTIONS

There are two sliders

<< LOW (SKETCH QUALITY) HIGH >>

Slide the handle to the left to decrease, and to the right to increase, the number of facets around the CIRCLE and ARC sketch-elements.

<< LOW (EXTRUSION QUALITY) HIGH >>

Slide the handle to the left to decrease, and to the right to increase, the number of facets around EXTRUSIONS

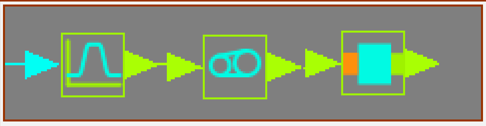
☑ **SHOW FUNCTION-BLOCK NAMES** (see images left)

Enable to show the names of each FUNCTION-BLOCKS above the FB in the graphic-area.

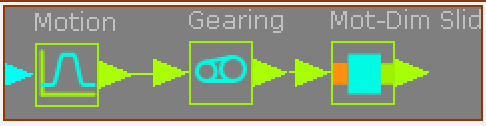
This is more helpful if you rename FUNCTION-BLOCK - **see** **Rename element** [279]

☑ **ENABLE ANTI-ALIASING***

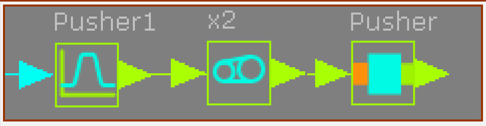
Enable to remove the jagged edges from the symbols of elements in the graphic-area.



Do not display Function-Block names

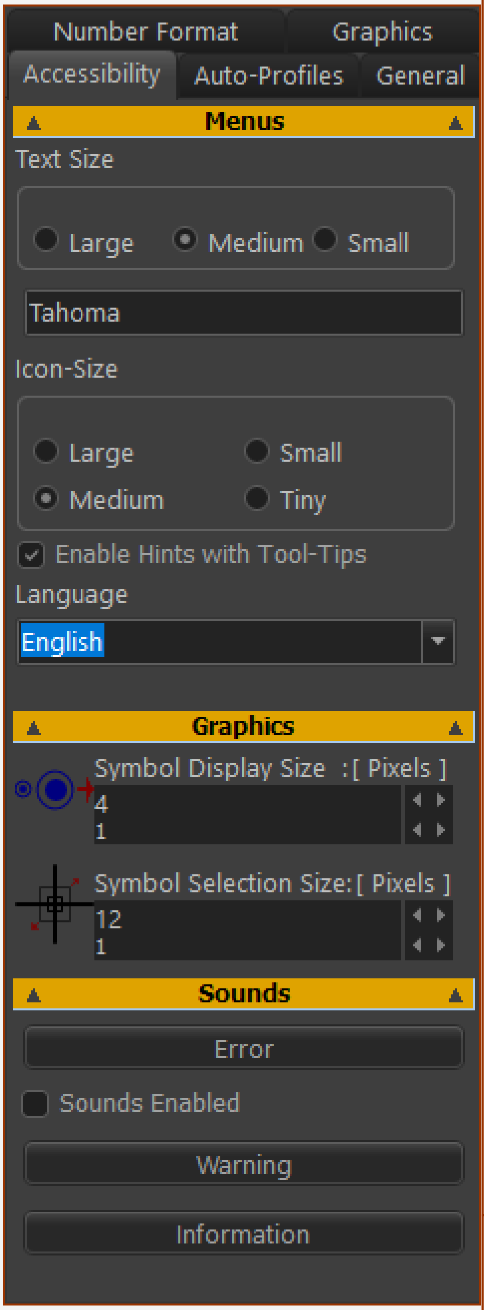


Do display Function-Block names



Display renamed Function-Blocks

Accessibility tab



MENUS

TEXT SIZE

Text size of menus and in dialogs.

Large - too big ?

Medium - typical and recommended

Small - OK?

They are a relative to the **ICON SIZE** (see below)

FONT - choose a font for the dialogs - for example, Tahoma

ICON SIZE

Large - size is 48×48 pixels (too big?)

Medium - size is 32×32 pixels (OK, recommended)

Small - size is 24×24 pixels (OK)

Tiny - size is 16×16 pixels- (too small?)

Note : When you change the **ICON SIZE** the command icons in the toolbars do not arrange correctly, then reset the **ICON-SIZE** to **Medium** and, if necessary, re-start **MechDesigner.**

LANGUAGE (An English Joke about language)

Languages: English, Italian, German, French, Spanish, Portuguese, Dutch, Polish, Russian, Japanese, Chinese.

ENABLE HINTS WITH TOOL TIPS

We also show for you the hints in the Feedback Area [270].

See also Help menu [60]

GRAPHICS

SYMBOL DISPLAY SIZE (PIXELS):

The size of symbols that represent the elements in the graphic-area.

SYMBOL SELECTION SIZE (PIXELS):

The size of the pointer **pin-head**. Increase to make it easier to select an element, but you may also select more than one element.

SOUNDS

☑ SOUNDS ENABLED

Enable the check-box to hear a different sound with each message in the Feedback-Area [270]. There are many messages!

Click the buttons to hear the sound for the three types of message.

Auto-Profile tab

DEFAULT SIZES

DEFAULT PART-OFFSET (default = 0)

The default offset of all PROFILE/EXTRUSIONS you add to a PART.

DEFAULT BALL RADIUS

The default size of the BALL-JOINT symbol.

See also: **Add Ball-Joint** [97] ; **Ball-Joint dialog** [456]

DEFAULT BEARING DIA.

DEFAULT GROUND BEARING DIA.

The default dimension given to ARCS at moving and fixed PIN-JOINTS when you do Add Auto-Profile(s).

See also: **Add Auto-Profiles** [204]

DEFAULT EXTRUSION DEPTH

The default EXTRUSION-DEPTH parameter in the EXTRUSION DIALOG.

See also: **Add Auto-Profile** [202] ; **Add Auto-Profiles** [204] ; **Extrusion dialog** [298]

1.10.4Dialog: Machine-Settings

Machine-Settings

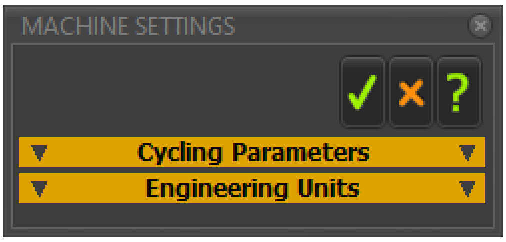
Use the **MACHINE-SETTINGS DIALOG** to edit the **Machine Speed, Number-of-Steps** in a machine-cycle, and to edit the **Engineering Units**.

How to open the Machine-Settings dialog

	1.	Click Edit toolbar [41] > Machine-Settings
OR	1.	Click Edit menu > Machine-Settings [39]

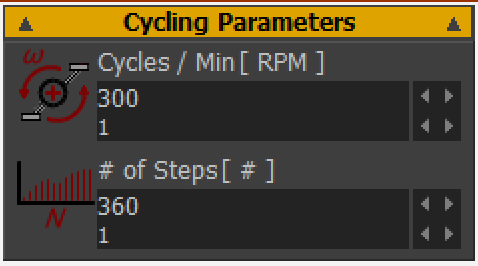
The **MACHINE-SETTINGS DIALOG** is now open.

Machine Settings dialog



Machine Settings dialog

CYCLING PARAMETERS



Cycling Parameters

CYCLES / MIN: (RPM) (**See also** Note 1 [292])

RPM - Revolutions per Minute.

The **Master Simulation-Speed**. It sets the rate-of-change of the MASTER-MACHINE-ANGLE.

MotionDesigner reads this parameter.

NUMBER-OF-STEPS: (**See also** Note 2 [292])

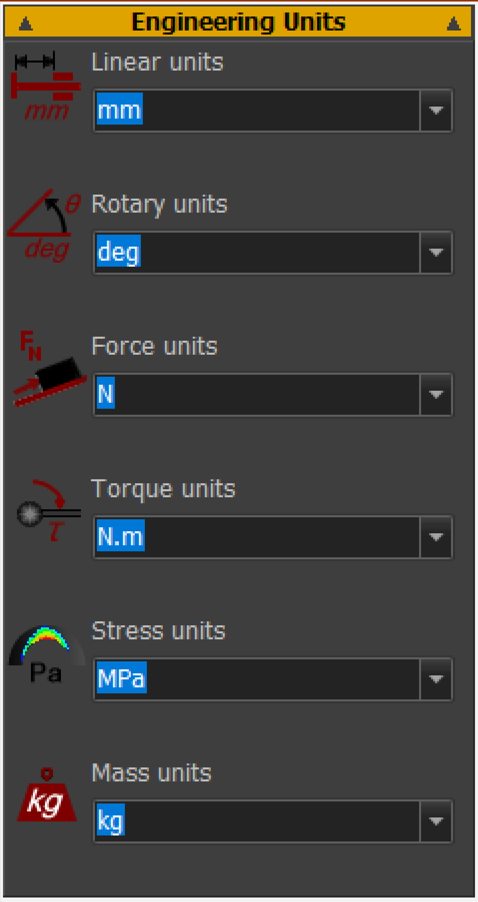
Number of calculation points in a machine-cycle.

Notes:

- NUMBER-OF-STEPS does not change accuracy of a Cam **- see** Cam-Coordinates dialog [361]
- NUMBER-OF-STEPS is the number of data-points along each graph - **see** Graph FB [386]
- NUMBER-OF-STEPS is the number of facets along the curve of a Trace-Point - **see** Trace-Point
- NUMBER-OF-STEPS is the number of points/dots on the **Motor Torque and Speed Curves**
- Usually, 120 is good. More than 360 is not usually needed.

See also: Run menu > Step Forward / Step Backward [44]

ENGINEERING UNITS



Engineering Units

ENGINEERING UNITS : (See also Note 3 [292])

LINEAR:	mm, cm, meters, inch.
ROTARY:	deg, radians, cycles
FORCE:	N, lbf (pound-force)
TORQUE:	N.m, N.mm, ft.lbf
STRESS:	Pa(N / m2), MPa(N / mm2), PSI(lbf / in2).
MASS:	kg, gms (grams), lb, oz

More about the Machine Settings Parameters.

Note 1:

CYCLES / MIN does not change the **animation-speed**.

Change the **animation-speed** with the Animation Speed Slider [271].

Note 2:

Do NOT increase **Machine-Settings >Number-of-Steps** [291] to increase accuracy of a 2D-CAM! Use the **Cam-Coordinates dialog** [361] to increase the accuracy of the 2D-CAM for manufacture.

120 Steps is ideal. 360 Steps should be the maximum, for nearly all models.

To move the model with small steps (increments), we recommend you use the Spin-Box [518] in the MASTER MACHINE ANGLE.

Note 3:

If you change the dimension units, you do **not** change the size of the model.

For example, **100 mm** becomes **3.94 inches** when you change dimensional units from **mm** to **inches.**

For example, **100 degrees** becomes **1.75 radians** when you change dimensional units from **degrees** to **radians.**

In all cases, we do **not** display for you the units in the graphic-area.

1.10.5 Dialog: DXF-Element

DXF-Element

Do **File Open > DXF file-type** 16 to add a DXF-ELEMENT to the ASSEMBLY-TREE.

The DXF ELEMENT is the **container** for a DXF-DRAWING.

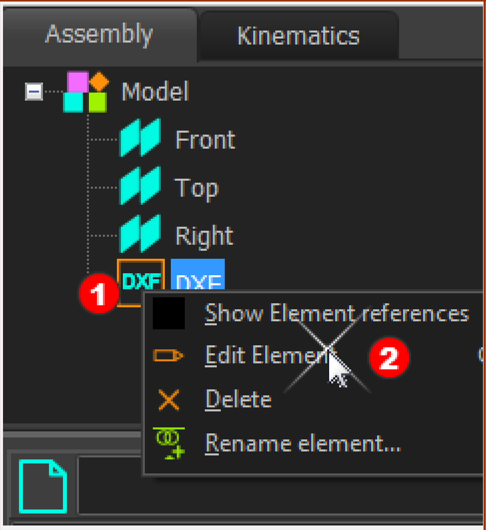
Use the DXF-ELEMENT DIALOG to:

- Link the DXF-ELEMENT to a different DXF-DRAWING
- Apply different LINEAR UNITS to the DXF-DRAWING

Use the CAD-LINE > DXF TAB 306 to:

- Display the DXF-DRAWING.

How to open the DXF-Element dialog



Editing the DXF-Element from the Assembly-Tree

To edit the DXF-Element

1. Find a DXF-ELEMENT❶ in the ASSEMBLY-TREE
2. Right-click the DXF-ELEMENT

In the shortcut menu:

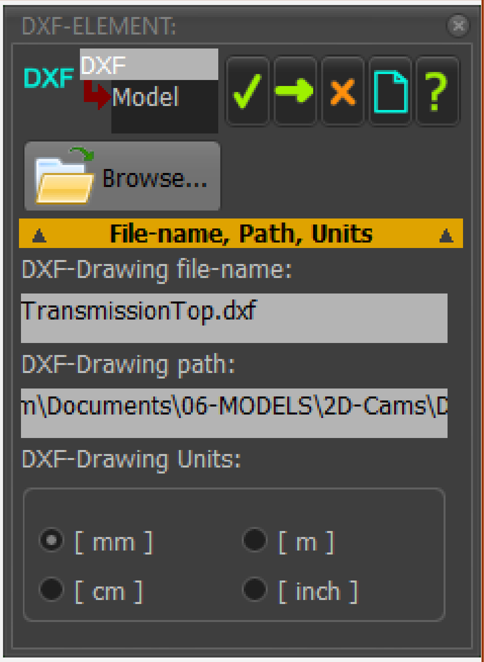
3. Click **Edit Element**❷

OR

See How to open a dialog 513

The **DXF-Element dialog** is now open.

DXF-Element dialog



FILE NAME, PATH, AND UNITS

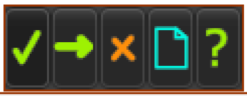
DXF-DRAWING FILE-NAME ; PATH

1. Click the BROWSE button to find and select a DXF-DRAWING.

DXF-DRAWING UNITS

To show the DXF-DRAWING to the correct scale, select the DXF UNITS of the **original** DXF-DRAWING.

To apply a new **DXF-Drawing** and **Units** to a **DXF-Element**



	1. Click **APPLY** to link the **DXF-DRAWING** with the **DXF-ELEMENT**. 2. Click **CLOSE** to close the dialog.

1.10.6 Dialog: Plane

Add / Edit Plane

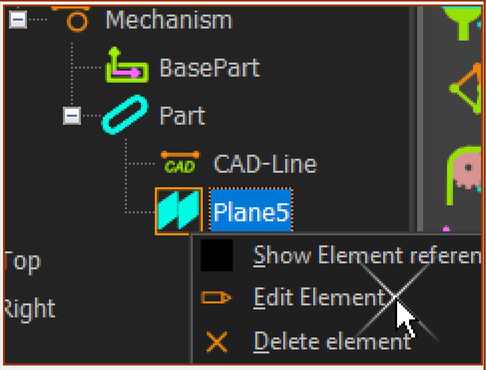
See also: Model Editor > Add Plane [70] **and Mechanism-Editor > Add Plane** [77]

Why add PLANES: they define the layout of a Machine. You add **MECHANISM-EDITORS** to PLANES.

When you do Add Plane, the PLANE DIALOG opens immediately to specify its parameter(s).

Edit a PLANE to edit its parameters with the PLANE DIALOG.

How to open the Plane dialog-box



Edit Plane

The PLANE DIALOG opens automatically when you do Add Plane. To open the PLANE DIALOG at a different time:

1. Right-click a PLANE in the **ASSEMBLY-TREE.**
2. Click **Edit-element** in the shortcut menu

OR

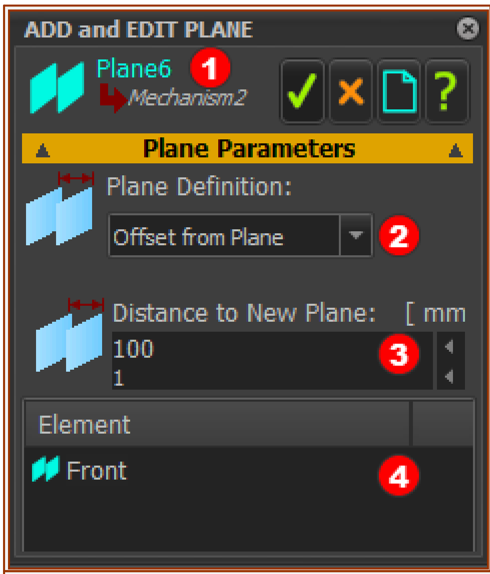
1. **See How to Open a dialog** [513]

The PLANE DIALOG is now open.

Plane dialog

The PLANE DIALOG has two configurations - which we call the **Plane Definitions**.

PLANE DEFINITION 1: Offset from Plane



❶ **New Plane:** PLANE6

PLANE PARAMETERS

❷ **Plane Definition**: OFFSET FROM PLANE

❹ **Element box**- the element you select as the reference for the **Plane Definition**

Parameter to edit:

❸ DISTANCE TO NEW PLANE

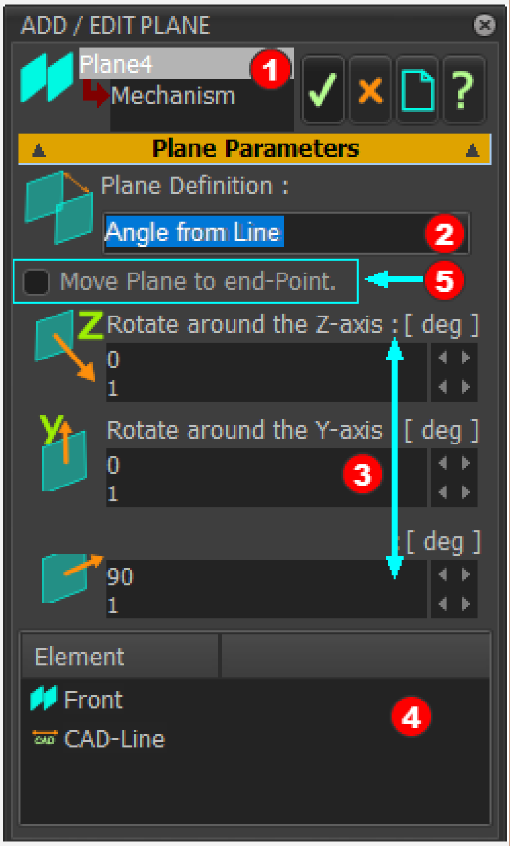
The offset dimension from the PLANE in the **ELEMENT** box❹ to the new PLANE❶

Notes:

The new PLANE is parallel to the selected PLANE

The Origin of the new PLANE is on the **Z-axis** of the selected PLANE

PLANE DEFINITION 2: Angle from Line



❶ **New Plane:** PLANE4

PLANE PARAMETERS

❷ **Plane Definition:** ANGLE FROM LINE

❹ **Element:** The element(s) you select as the reference for the **Plane Definition**

Parameters to edit:

❺ MOVE PLANE TO END-POINT check-box.

Enable to move the origin of the PLANE from the **START-POINT** to the **END-POINT** of the LINE that is in the **ELEMENT** box❹

NOTE: You must MOVE PLANE TO END-POINT to add a BEVEL GEAR-PAIR[124].

❸ ROTATIONAL AXES

- ROTATE AROUND X-AXIS : Rotate the new PLANE around the active X-AXIS direction of the new PLANE
- ROTATE AROUND Y-AXIS : Rotate the new PLANE around the active Y-AXIS direction of the new PLANE
- ROTATE AROUND Z-AXIS : Rotate the new PLANE around the active Z-AXIS direction of the new PLANE

Notes:

Default **Origin** of the new PLANE is coincident the **START-POINT** of the selected LINE

Default **+ve X-axis** of the new PLANE is in the direction from the **START-POINT** to the **END-POINT** of the selected LINE.

Related topics

Model toolbar > Add Plane[70] (in the **MODEL-EDITOR** tab)

Model toolbar > Add Plane[77] (in any **MECHANISM-EDITOR** tab)

How to open a dialog

How to edit a parameter[513]

1.10.7 Dialog: Profile/Extrusion

Profile/Extrusion

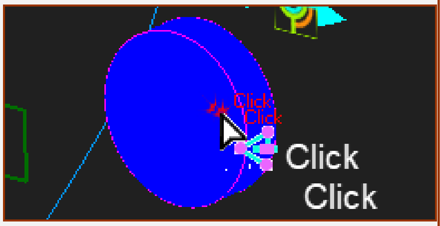
See Add Profile/Extrusion[210]

You add **PROFILE / EXTRUSIONS** to **sketch-loops**.

Use the **EXTRUSION DIALOG** to edit the properties of an **EXTRUSION**.

How to open the Extrusion dialog

If you CAN see the Extrusion in the Graphic-Area



Double-click Extrusion

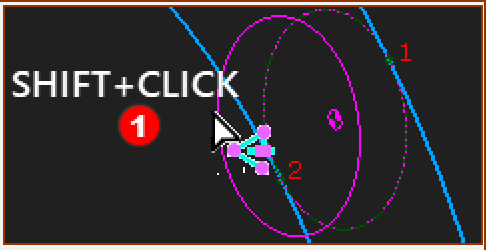
To open the **EXTRUSION DIALOG**

1. Double-click the **EXTRUSION** in the **GRAPHIC-AREA**

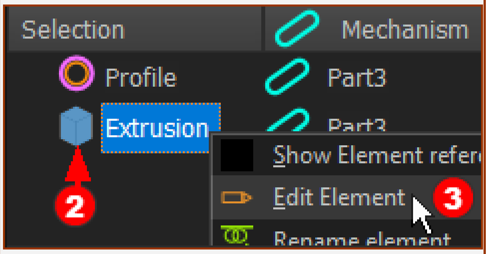
or

1. See How to open a dialog[513]

If you CANNOT see the Extrusion in the Graphic-Area



SHIFT-CLICK Profile



Profile/Extrusion in Selection-Window

STEP 1: In the Graphic-Area

❶ **SHIFT+CLICK** the (Pink) **PROFILE**

STEP 2: In the Selection-Window

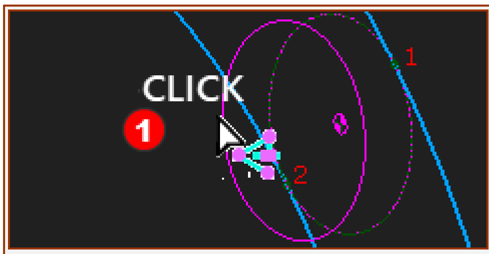
The **PROFILE** and **EXTRUSION** show in the **SELECTION-WINDOW**

❷ **SELECTION-WINDOW**: Right-click the **EXTRUSION** element

In the shortcut menu:

❸ Click **Edit Element.**

Alternatively, if you CANNOT see the Extrusion in the Graphic-Area

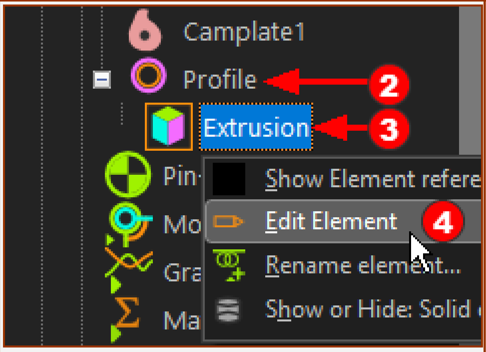


CLICK Profile

STEP 1: In the Graphic-Area

❶ **CLICK** a (Pink) **PROFILE**

The **ASSEMBLY-TREE** expands to identify the **PROFILE**❷ and **EXTRUSION**



Profile/Extrusion in Assembly-Tree

STEP 2: In the Assembly-Tree

1. ASSEMBLY-TREE: Click the EXTRUSION❸

The Element name is now blue, the ICON has an orange box❸

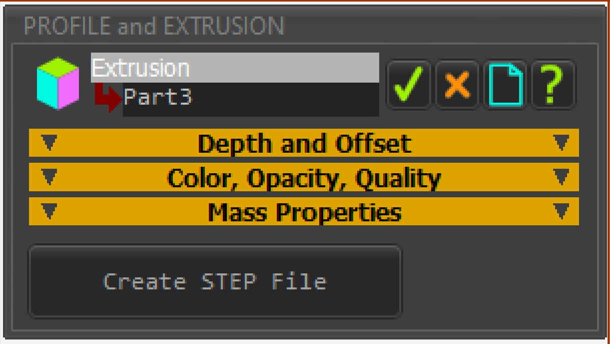
2. ASSEMBLY-TREE: Right-Click the EXTRUSION❸

A shortcut menu shows next to your pointer:

3. SHORTCUT MENU: Click Edit Element❹

The EXTRUSION DIALOG is now open.

The Extrusion dialog



Extrusion dialog

There are three separators in the EXTRUSION DIALOG.

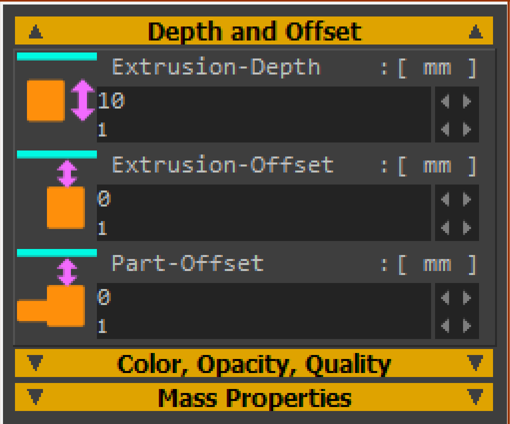
DEPTH AND OFFSET

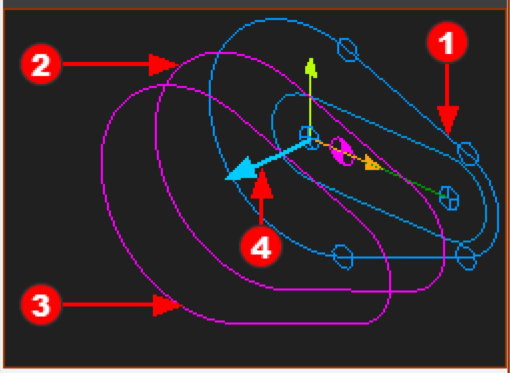
COLOR, OPACITY AND QUALITY

MASS PROPERTIES

Create STEP File - to save the EXTRUSION as a STEP file-type.

DEPTH AND OFFSET





EXTRUSION-DEPTH (default = see Application-Settings> Auto-Profile tab[290])

The distance to the SECONDARY-CONTOUR❸ from the PRIMARY-CONTOUR❷, along the +Z-axis❹ of the PART.

EXTRUSION-DEPTH must be positive. **Minimum value = 0.001mm**

EXTRUSION-OFFSET (default = 0)

The distance to the PRIMARY CONTOUR from the dimension that is specified by PART-OFFSET.

EXTRUSION-OFFSET can be negative or positive.

PART-OFFSET (default = 0)

Applies to **ALL** EXTRUSIONS in a PART.

The distance to the PRIMARY CONTOUR from the **Mechanism-Plane** to which we add the distance specified by the EXTRUSION-OFFSET parameter.

PART-OFFSET can be negative or positive.

Note: Distance to PRIMARY CONTOUR❷ from MECHANISM-PLANE = PART OFFSET + EXTRUSION OFFSET

The Primary and Secondary Contours of the PROFILE:

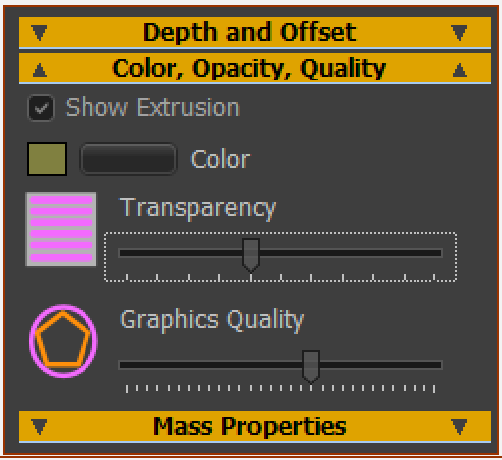
❶ Sketch-Loop

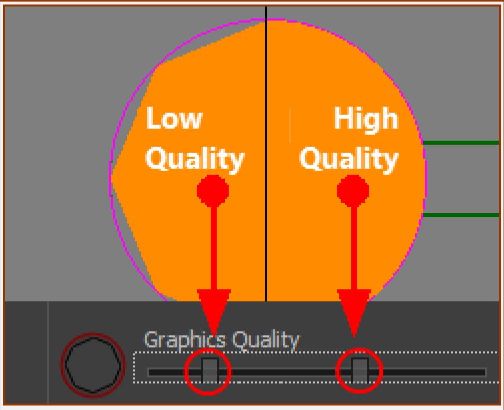
❷ Primary-Contour

❸ Secondary-Contour

❹ Z-axis is perpendicular(⊥) to the Mechanism-Plane

COLOR, OPACITY, QUALITY





Low Quality High Quality

SHOW EXTRUSION check-box

To show or hide the EXTRUSION

Note: To hide PROFILE (contours), see **Display toolbars > Show/Hide Profiles**

COLOR

Button to select the color of the EXTRUSION with the Windows® Color-Picker

TRANSPARENCY (SLIDER)

Slider to change the TRANSPARENCY of the EXTRUSION in the graphic-area

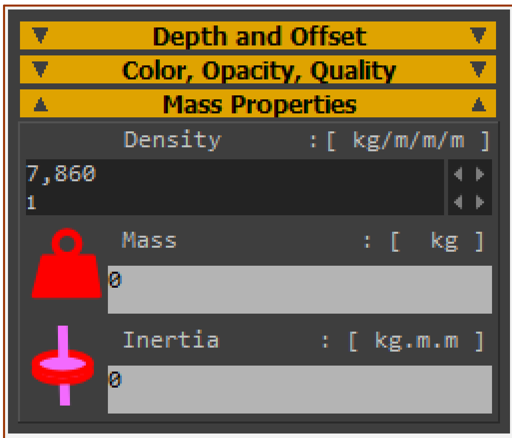
GRAPHICS QUALITY (SLIDER)

Slider to change the GRAPHICS QUALITY of the EXTRUSION

Note: To update the GRAPHICS QUALITY, you may need to open the PART in the PART-EDITOR and then exit the PART-EDITOR immediately.

See also: Application Settings | Graphics [287]

MASS PROPERTIES



DENSITY Units $[kg/m^3]$

The MASS and INERTIA properties (below) change as you edit the DENSITY.

MASS (read-only) Units $[kg]$

MASS = DENSITY × area of **sketch-loop** × EXTRUSION DEPTH

INERTIA (read-only) Units $[kg.m^2]$

INERTIA is calculated with respect to the center-of-mass.

The position of the center-of-mass is a function of the shape of the **sketch-loop** you select to add the PROFILE. The center-of-mass is on the **MECHANISM-PLANE**. The mass is uniformly distributed.

See also CAD-Line dialog > Mass Properties tab [304]

1.10.8Dialog: CAD-Line >

CAD-Line

See also: **Add CAD-Line** 230, and About **CAD-Line** details.

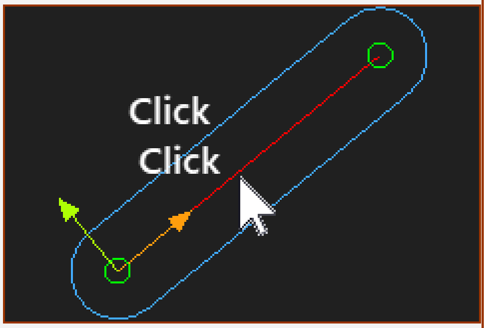
There is one CAD-LINE from the **START-POINT** to **END-POINT** of all PARTS you add to the model.

You can use the **Part-Editor** to add CAD-LINES to a PART.

Use the CAD-LINE DIALOG to:

- Import and display **one**(1) **CAD-SOLID**
- Display **one**(1) DXF-DRAWING - to import the DXF-DRAWING, do **File menu > Open DXF-Drawing** 19
- Import and/or edit user Mass-Properties
- Change how to display the **CAD-SOLID**

How to open the CAD-Line dialog



Double-Click CAD-Line element in an Added-Part

To open the CAD-LINE DIALOG

1. Double-click a CAD-LINE in the graphic-area, or the ASSEMBLY-TREE

OR

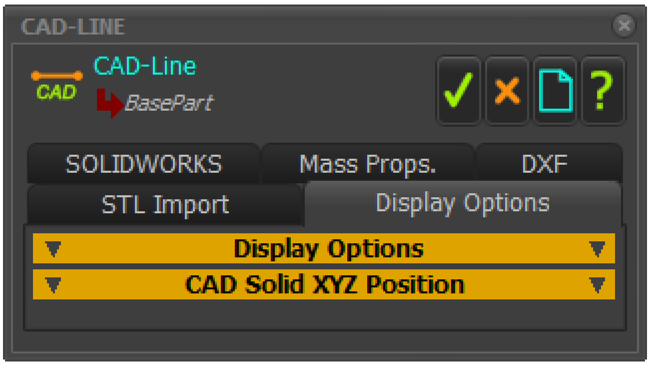
1. **See** **How to Open a dialog** 513

Note:

You can open the CAD-LINE DIALOG from MECHANISM-EDITORS AND the PART-EDITOR.

The CAD-LINE DIALOG is now open - see below

CAD-Line dialog



CAD-Line dialog

CAD-Line dialog tabs.

- **SOLIDWORKS tab** 302
- **Mass Properties tab** 304
- **DXF tab** 306
- **STL Import tab** 310

- Display Options tab [309]

1.10.9-> SOLIDWORKS tab

CAD-Line > SOLIDWORKS tab

See also: Mass-Properties tab [304], DXF tab [306], STL Import tab [310], Display Options tab [313].

Enable Visibility toolbar > Show Solids in Mechanisms [51] to see **CAD-SOLIDS** in **MECHANISM-EDITORS**.

Use the **CAD-Line dialog** > **SOLIDWORKS** tab to:

- Import a Part/Assembly document directly from SOLIDWORKS - see below [302]
- Remove from a **CAD-LINE** a Part/Assembly document that you have already imported from SOLIDWORKS
- Open in SOLIDWORKS a Part/Assembly document that you have already imported from SOLIDWORKS
- Specify the surface accuracy in MechDesigner of the Part/Assembly document that you will import from SOLIDWORKS
- Rename the CAD-Line to the SOLIDWORKS Part/Assembly document name.

In **SOLIDWORKS**:

STEP 1. Make sure the document you want to import from SOLIDWORKS is open and it is the active document.

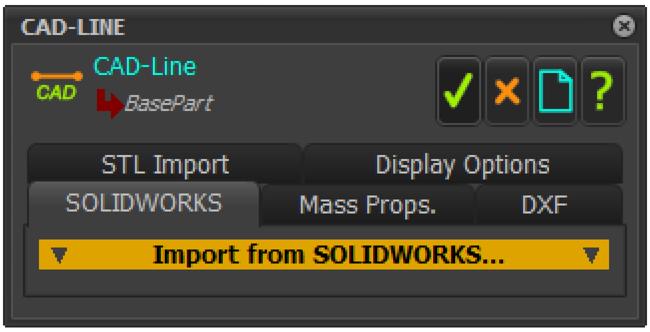
STEP 2. Make sure the SOLIDWORKS document has a file-name.

STEP 3. Usually, add a Coordinate-System to the SOLIDWORKS document to align it to the **CAD-LINE.**

Warning: **The model does not import into MechDesigner if Configuration name uses a Chinese Tradition Font. Rename the Configuration with a 'western font'.**

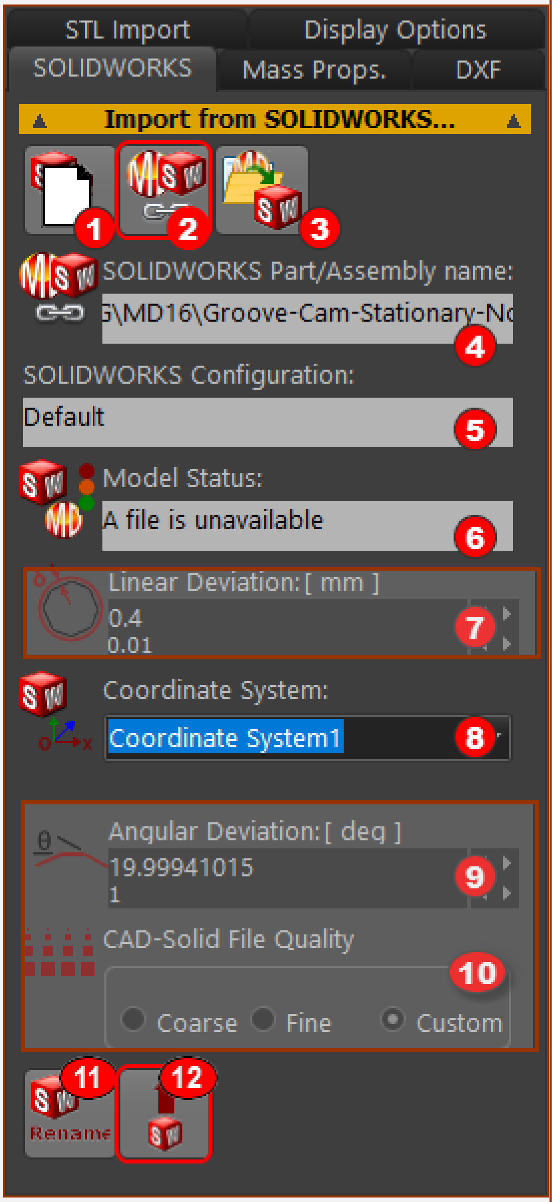
CAD-Line dialog

SolidWorks tab



CAD-Line dialog> SOLIDWORKS tab

IMPORT FROM SOLIDWORKS



CAD-Line dialog > SOLIDWORKS tab

STEP 1: Click the READ❷ button

WAIT until the:

- SOLIDWORKS® PART/ASSEMBLY (read-only) shows in box❹
- SOLIDWORKS CONFIGURATION (read-only) shows in box❺
- MODEL STATUS (read-only) shows in box❻

Model Status:
Unavailable - before you import SolidWorks® document the first time.
Out of Date - the file in box❹ is older than the current date/time.
Up to Date - the date/time of file in box❹ is the equal to the date/time when imported.

STEP 2: If there is more than one Coordinate-System, select a SOLIDWORKS COORDINATE-SYSTEM❽.

STEP 3: Click the IMPORT ⓬ Button

WAIT - a large model can take time to import.

OPTIONAL BUTTONS(❶❸)

Button❶ : REMOVE : the SOLIDWORKS® document in box ❹ from the CAD-LINE

Button❸ : OPEN : the document in box❹ (that you have previously imported) in SOLIDWORKS®

Button⓫ : **RENAME CAD-LINE** to the file-name of the SOLIDWORKS document in box❹.

CAD-SOLID FILE QUALITY ❿

⦿ **Fine**: more vertices

⦿ **Coarse**: fewer vertices

⦿ **Custom** : edit with:

- LINEAR DEVIATION❼: Maximum =0.5mm; Minimum =0.014mm
- ANGULAR DEVIATION❾: Maximum =30º; Minimum =0.4º

WARNING : Number-of-Vertices[313] may increase rapidly as you reduce the LINEAR/ANGULAR DEVIATION.

Often the appearance of the **CAD-SOLID** does not improve. Experiment! **See: Edge Angle Limit**[312].

1.10.10 Mass Properties tab

CAD-Line > Mass Properties tab

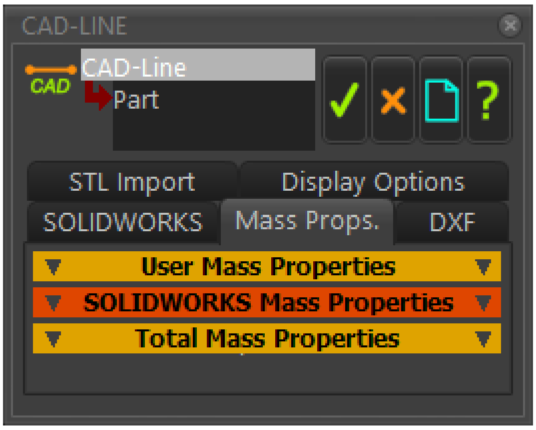
See also : CAD-Line dialog 301

Use the **Mass-Properties** tab

- Enter user-defined mass properties - see below 304
- Import from SOLIDWORKS the mass-properties of a SOLIDWORKS document - see below 305
- Review the total mass-properties on the **PART** - see below 305

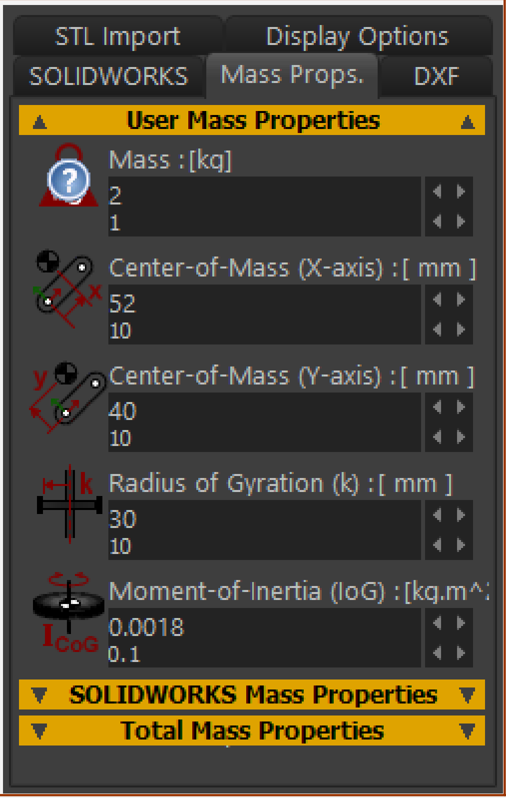
CAD-Line dialog

Mass Properties tab



CAD-Line dialog > Mass- Properties tab

USER MASS PROPERTIES



MASS (Units: kg)

CENTER-OF-MASS: X–AXIS - (mm)

CENTER-OF-MASS: Y–AXIS - (mm)

Note:

The X-axis is the **CAD-LINE**, the Y-axis is at +90º.

The 0,0 is the **START-POINT** of the **CAD-LINE**.

MOMENT OF INERTIA (I_g) - ($kg.m^2$ **NOT** $kg.mm^2$)

The Inertia about the Center-of-Mass.

You can enter of:

RADIUS-OF-GYRATION (mm),

OR

MOMENT-OF-INERTIA ($kg.m^2$)

Moment-of-Inertia: Radius-of-Gyration, and Point-Mass.

CASE 1:

If you enter a MASS but do NOT enter a RADIUS-OF-GYRATION or MOMENT-OF-INERTIA, the MASS is reduced to a **Point Mass**.

The position of the MASS is at (CENTER-OF-MASS: X–AXIS, CENTER-OF-MASS: Y–AXIS).

CASE 2:

If you enter MOMENT-OF-INERTIA, then the equivalent RADIUS-OF-GYRATION, k_g is :

$$k_g = \sqrt{\frac{I_g}{m}}$$

CASE 3:

If you enter RADIUS-OF-GYRATION, then the equivalent MOMENT-OF-INERTIA, I_g is:

$$I_g = M.k_g^2$$

PARALLEL AXIS THEOREM

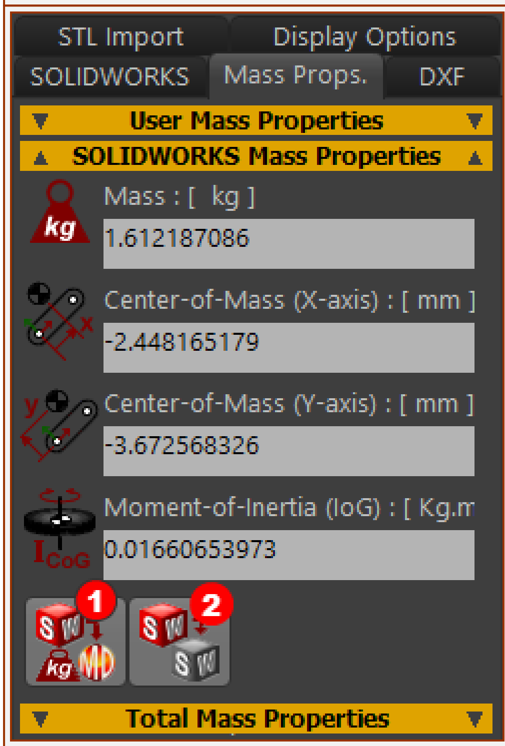
We use for you the **Parallel Axis Theorem** to calculate the **Moment-of-Inertia** about the PART'S instantaneous center-of-rotation.

$$I_o = I_g + M.h^2$$

h = distance to the Center-of-Mass from the center-of-rotation.

SOLIDWORKS MASS PROPERTIES

When you import a **SOLIDWORKS** document onto a CAD-LINE, you can also import the **SOLIDWORKS Mass-Properties**



Button ❶ IMPORT / UPDATE SOLIDWORKS MASS PROPERTIES

Note: Before you click button ❶, make sure the active **SOLIDWORKS** document is on the CAD-LINE.

MASS : (kg)

MASS of the active SOLIDWORKS document.

CENTER-OF-MASS - X-AXIS: (mm)

The distance to the CENTER-OF-MASS along the X-axis from the **START-POINT** of the CAD-LINE.

CENTER-OF-MASS - Y-AXIS (mm)

The distance to the CENTER-OF-MASS along the Y-axis from the **START-POINT** of the CAD-LINE.

MOMENT-OF-INERTIA $(kg.m^2)$

The INERTIA with respect to the CENTER-OF-MASS

Button ❷ REMOVE SOLIDWORKS MASS PROPERTIES

TOTAL MASS PROPERTIES

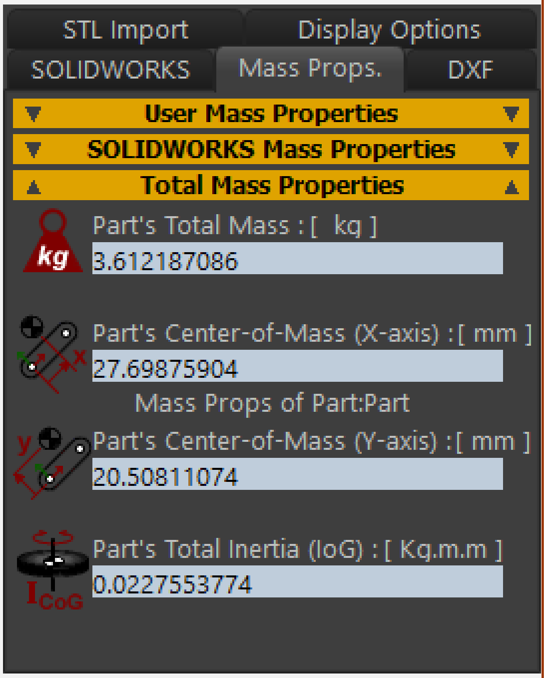
Total Mass Properties:

The **Total Mass Properties** is for ALL CAD-LINES in one PART.

EXPLICITLY: the **Total Mass Properties** include, for each PART, ALL of those:

- **Mass** and **Mass-Properties** as calculated for ALL PROFILE/EXTRUSIONS in the PART

- **User Mass-Properties** as defined for ALL CAD-LINES in the PART
- SOLIDWORKS **Mass-Properties** as defined for ALL CAD-LINES in the PART



NOTES:

Parallel-Axis Theorem

Inertia about Part's Rotational-Axis = Total Inertia about center-of-Mass + Mass × (Distance to Part's Rotational-Axis)^2

Relationship between Moment-of-Inertia and Radius of Gyration

Total Inertia (about center-of-Mass) = Total Mass × (Radius-of-Gyration)^2

Total Mass of Part =

Mass of all Extrusions [300] + Total mass of other CAD-Lines in the same Part

Total Inertia about the Part's center-of-Gravity =

Inertia of all Extrusions + Total Inertia of other CAD-Lines in the same Part

1.10.11 > DXF tab >

CAD-Line > DXF tab

See also: DXF-Element [294]

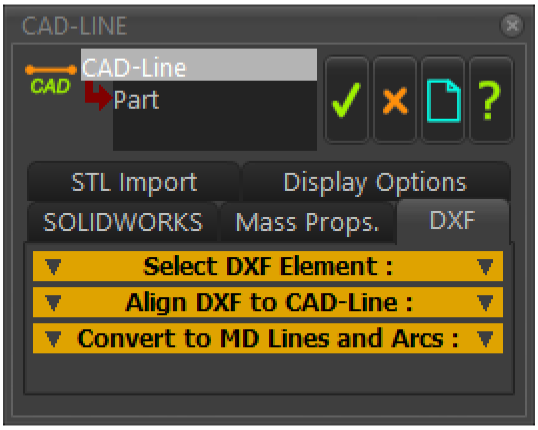
Use the **DXF tab** to:

- Display* a **DXF-DRAWING**
- Align the **DXF-DRAWING** with the CAD-LINE
- Show or Hide different **DXF LAYERS** and/or change the Layer colors
- Convert **DXF ENTITIES** in the **DXF DRAWING** to MD sketch-elements
- Remove the **DXF-DRAWING** from the CAD-LINE

* Before you can display a **DXF-DRAWING** with a CAD-LINE, you must use File menu > Open > DXF file-type [16] to open and add a minimum of one DXF-ELEMENT to the **ASSEMBLY-TREE**.

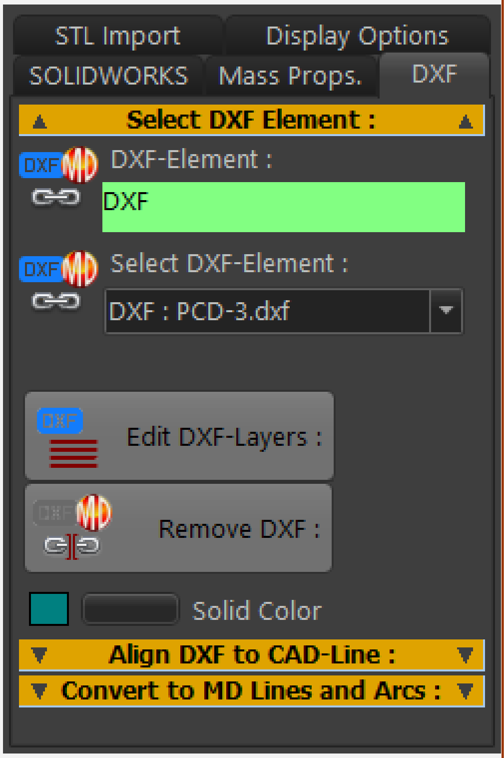
CAD-Line dialog

DXF tab



CAD-Line dialog > DXF tab

SELECT DXF-ELEMENT



After you do **File menu > Open DXF file-type**, select the **DXF-ELEMENT** in this dialog to show it in the graphic-area.

SELECT DXF-ELEMENT: drop-down list-box

Each **DXF-ELEMENT** has two parts - see example[307] below

1. Click the drop-down arrow to select a **DXF-ELEMENT**

The **DXF-DRAWING** shows in the graphic-area immediately.

See here[306] if you cannot see the **DXF-DRAWING** in the graphic-area.

Optional buttons:

EDIT DXF LAYERS :

To open the DXF LAYERS DIALOG[309] to show or hide different **LAYERS** in the **DXF-DRAWING**

REMOVE DXF :

To remove the **DXF-DRAWING**.

DXF-ELEMENT

For example: DXF : PCD-3.DXF

DXF is the name of the **DXF-ELEMENT** in the **ASSEMBLY-TREE**. It is the container for the **DXF-DRAWING**.

PCD-3.DXF is the name of the **DXF-DRAWING** that you opened File menu > Open > DXF file-type[16].

If you cannot see the **DXF-DRAWING**:

Change the background color of the graphic-area

See Application Settings | color tab[287]. Mid Gray is often a good color.

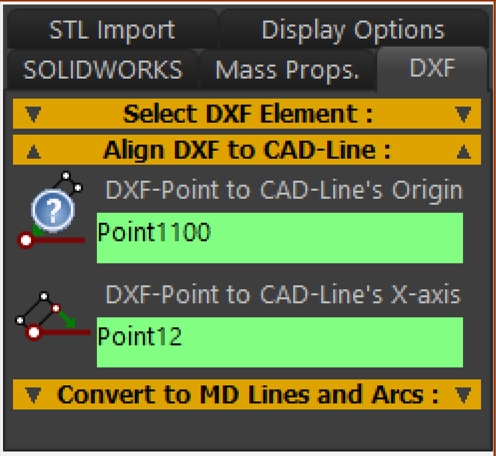
The **DXF-DRAWING** may be outside of the current graphic view.

See **View menu > Zoom-Out** [58]. (or use the wheel button on you mouse)

The **LINEAR UNITS** of the **DXF-DRAWING** may be incorrect, which may make the drawing small or big. It is normal to use the same **LINEAR UNITS** as the original **DXF-DRAWING**.

See **Edit DXF-Element dialog** [294].

ALIGN DXF TO CAD-LINE



Align the DXF-Drawing with the CAD-Line, select two(2) DXF-Points.

DXF POINT TO CAD-LINE'S ORIGIN

1. Click in the box to make the box **ORANGE** - see **Note**
2. Click the **DXF-POINT** in the **DXF-DRAWING** that you want to move to the **START-POINT** of the **CAD-LINE**

   The **DXF-POINT** shows in the box and the box becomes Green

DXF POINT TO CAD-LINE

3. Click in the box to make the box **ORANGE**
4. Click a **DXF-POINT** in the **DXF-DRAWING** to move it to the X-axis of **CAD-LINE**.

   The **DXF-DRAWING** moves the two(2) **DXF-POINTS** to the **CAD-LINE**

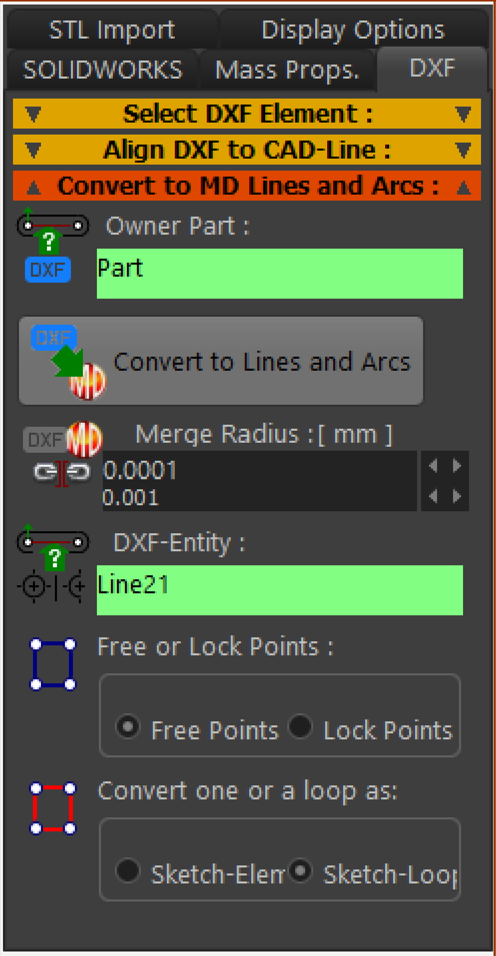
Note:

If you can see the **DXF-DRAWING** but cannot see DXF-Points, then:

1. Click the **Display Filter toolbar > DXF Points** (the toolbar is below the graphic-area)

Display toolbar > DXF-Points

CONVERT DXF ENTITIES TO MD LINES AND ARCS



DXF Line, **DXF Arc**, and **DXF Point** entities are **dumb**.

To convert the DXF entities to sketch-elements that are native to MechDesigner:

OWNER PART

This box specifies the **PART** onto which you convert the DXF entity

Usually, you add the sketch-elements to the same **PART** as the **CAD-LINE**.

1. Click the **OWNER PART** box to make it **ORANGE**
2. Click a **PART** in the graphic-area or the **ASSEMBLY-TREE**.

   The **PART** should now be in the **OWNER PART** box

DXF ENTITY

This box is the **DXF-ENTITY** you convert to a sketch-element.

3. Click the **DXF ENTITY** box to make it **ORANGE**

   Before you click a DXF-Entity, you may want to:

   - Click the **SELECT-LOOP** radio-button so that, if possible, you convert a number of DXF-Entities to a sketch-loop
   - Click the **LOCK POINTS** radio-button to lock/fix the **POINTS** in the sketch-elements to the **PART**, after you convert the DXF entities to sketch-elements
   - Optionally, edit the **MERGE RADIUS** value to help make a sketch-loop with those DXF entities inside the **MERGE-RADIUS'** value - I have never found a reason to edit **MERGE RADIUS**.

4. Click a **DXF ENTITY** (a **DXF LINE** or **DXF ARC**) in the graphic-area
5. Click the **CONVERT TO MD ENTITIES** button

Repeat **1** to 5 again to convert other DXF Entities to MD sketch-elements.

Note: After you **CONVERT DXF TO MD LINES AND ARC**, you can click the **REMOVE DXF** button, in **SELECT DXF ELEMENTS**.

1.10.12> -> DXF Layers

CAD-Line > DXF tab > DXF Layers

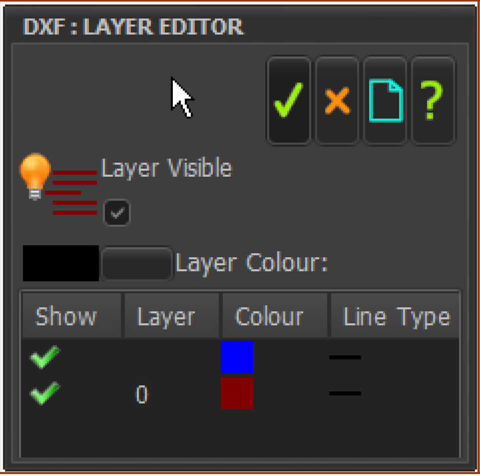
DXF-Drawings frequently use **Layers** for different drawing information.

Use the **DXF LAYERS DIALOG** to show or hide each Layer in a **DXF-Drawing**.

To open the **DXF LAYERS DIALOG**:

1. Click the **EDIT LAYERS** button in the CAD-Line dialog > DXF tab [306]

DXF Layers dialog



To Show or Hide a Layer:

1. Click a ✓ (or ✗) in the **Show** column
2. Click the **Layer Visible** (or **Layer Hidden**) check-box

To change the color of a Layer:

This option does not work!

See **Solid Color** in the DXF tab [306]

1.10.13 > STL Import tab

CAD-Line > STL Import tab

See also : CAD-Line dialog [301]

Use the **STL Import** tab if you do not have SOLIDWORKS.

Use **STL Import** to:

- Import 1 x **STL** file - it is a **CAD-SOLID**
- Edit the **STL FILE UNITS** to the Linear-Units of the original STL file
- Edit the **EDGE ANGLE LIMIT** to remove STL-file facet lines.

Before and when you save an STL file in your CAD:

Rotate the model in your CAD to the correct orientation relative to the **XY-Plane** of the **MECHANISM-EDITOR** before you import it.

Save the model (in your CAD software) as a **Binary STL file-type.**

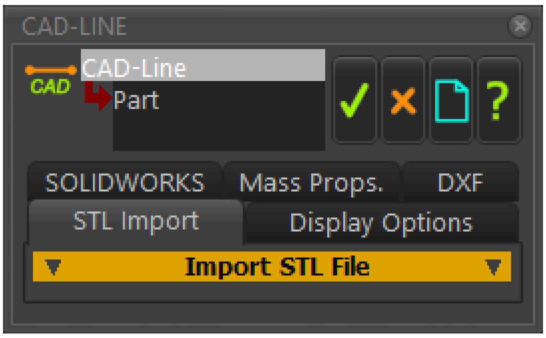
Remember the **Linear units.** If you cannot edit the STL units, they will be SI units - meters (m).

After you import an STL file:

See Display Options tab [314] to move the STL file in the X, Y, Z-axis directions.

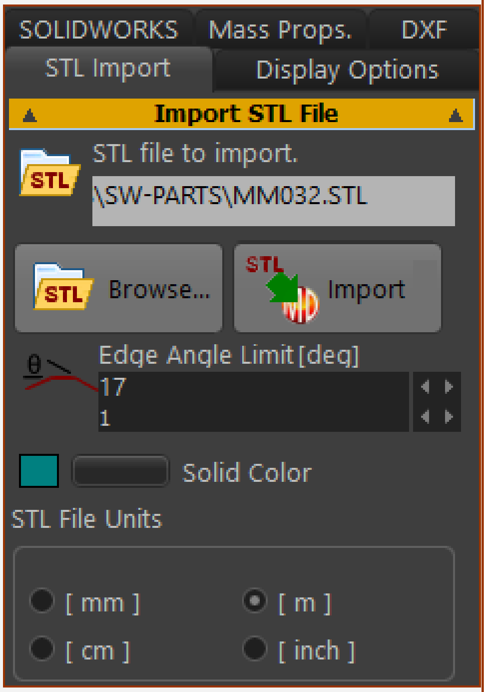
CAD-Line dialog

STL Import tab



CAD-Line dialog > STL Import tab

STL FILE IMPORT



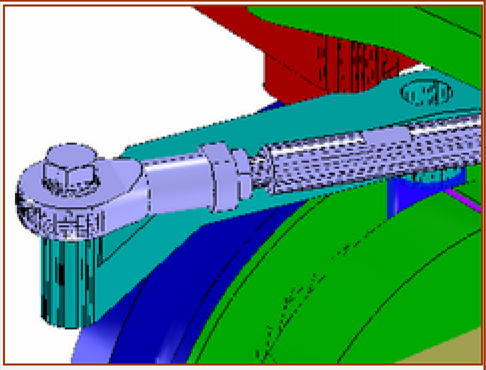


Image with too many Facet-Edges

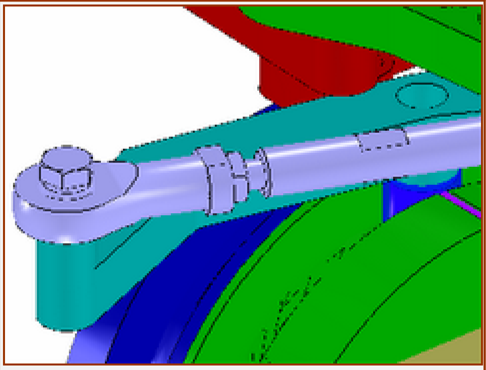


Image with Facet-Edges OK

To import an STL file:

STL FILE IMPORT

1. Click the **BROWSE** button
2. Explore and select an STL file
3. Click the **IMPORT** button.

Note: To see the **CAD-SOLID**, enable Visibility toolbar > Show Solids in Mechanisms [51]

EDGE ANGLE LIMIT -

1. Enter a Value in the **EDGE-ANGLE LIMIT** box

This parameter can change the ***apparent*** quality of the STL file in the graphic-area.

EDGE ANGLE LIMIT analyzes the acute angle between two adjacent facets and compares it to the **ANGULAR-DEVIATION** parameter (SOLIDWORKS tab [302])

- If the angle is **less than** the **EDGE ANGLE LIMIT**, we **do not** show a line between the two facets.
- If the angle is **more than** the **EDGE ANGLE LIMIT**, we **do** show a line made by the two facets.

SOLID COLOR button - to edit the color of the STL file:

1. Click the **SOLID COLOR** button
2. Select a color in the Windows Color Picker®
3. Click OK in the Windows Color Picker®

STL FILE UNITS

1. Select the units that are equal to the units of the **original STL** file.

Try meters, [m], if you are not sure.

STL FILE-SIZE

The **STL** file **tessellates** the surfaces of the CAD model as a mesh of triangles. Each triangle appears as a small face - which we call a **facet**. The number of facets is a function of:

- the complexity of the original CAD model, and
- how accurately you want the STL file to tessellate the original CAD model.

To build complex models, and to import many STL files, you want:

SMALL file-size for each STL file

- AND each STL file to appear as the original CAD model

- AND to see only those facets that define real edges in the CAD model.

STRATEGIES TO REDUCE THE STL FILE-SIZE.

STRATEGY 1: Reduce the complexity of the original CAD Model = SMALL file-size

Add a new configuration for the CAD model. In the new configuration:

- Do a geometry check of the CAD model file to find if it has any small gaps or whiskers.
- Suspend small fillets, especially if they do not have a function.
- Replace fillets with chamfers, especially along straight edges.
- Suspend all fasteners
- Suspend fastener holes and hole features.
- Suspend all cosmetic details.
- Simplify bearing models, and other complex models you may have downloaded from the internet.

STRATEGY 2: Reduce the accuracy of the STL file

There parameters are in the SOLIDWORKS tab.

LINEAR DEVIATION :

The maximum LINEAR DEVIATION (distance) between the surface of the original CAD model and the tessellated surface of the STL file.

ANGULAR DEVIATION :

The maximum ANGULAR DEVIATION between adjacent facets of the tessellated surface of the STL file.

If you do not have SOLIDWORKS, you must find these parameters in your 3D-CAD before you save to the STL file. It is possible they are options in the **Save as > STL file-type** dialog.

STRATEGY 3: Use the EDGE ANGLE LIMIT parameter - see STL FILE IMPORT above

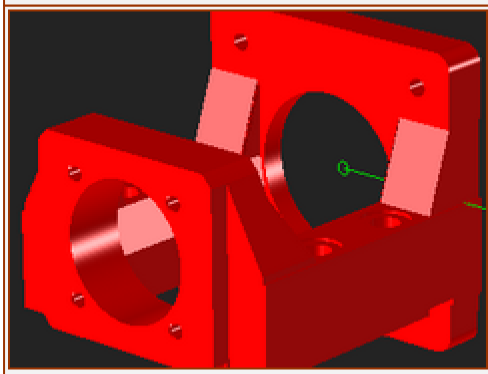
STL files may show their facets. You can improve how the facets display with my **Magic Number**!

EDGE ANGLE LIMIT = ANGULAR DEVIATION × 3.14159÷2

EXAMPLES

Below, the two images are of the same SOLIDWORKS part.

They appear to be identical. However, there is a large difference in the number of facets and vertices.



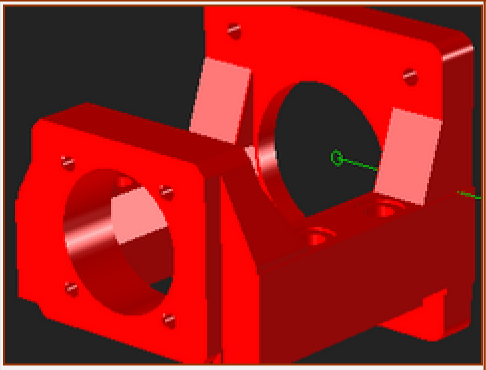
STL tessellation: Large number of Vertices

BAD - many facets

- Linear Deviation = 0.014mm and
- Angular Deviation = 0.4º

There are

- 119132 vertices
- ~40000 triangles
- The STL file size is **3.5 Mega Bytes**



STL tessellation: Few number of Vertices

GOOD - few facets

- Linear Deviation = 0.5mm and
- Angular Deviation = 5°

There are:

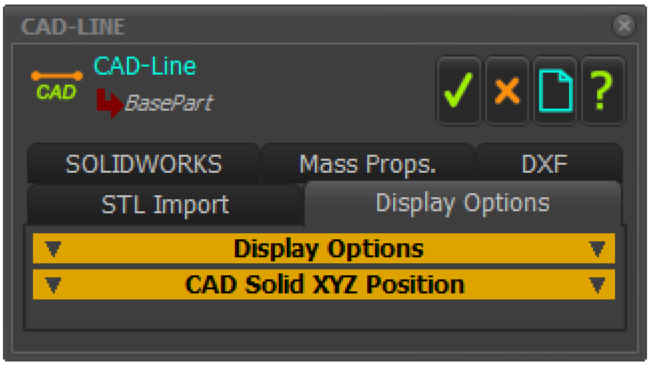
- 7533 vertices
- 2500 triangles
- The STL file size is **0.365 Mega Bytes**

1.10.14 Display Options tab

CAD-Line > Display Options tab

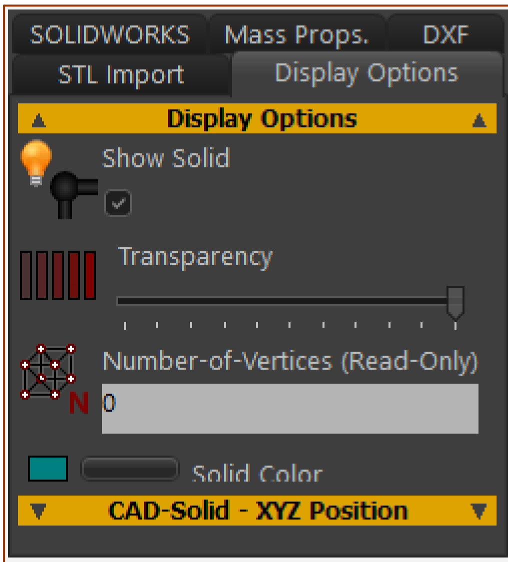
Use **Display-Option** tab to:

- Edit the color and transparency of an imported **CAD SOLID**
- To Toggle the Show or Hide an imported **CAD SOLID**
- To move an imported **CAD SOLID** in the X, Y, and/or Z-axis directions.



CAD-Line dialog > Display Options tab

DISPLAY OPTIONS



SHOW SOLID

Click the **SHOW SOLID** check-box to show or hide the **CAD-SOLID** that is on the **CAD-LINE**.

Note: Enable Visibility toolbar > Show Solids in Mechanisms [51] to show **CAD-SOLIDS** in **MECHANISM-EDITORS**

TRANSPARENCY:

Drag the **TRANSPARENCY** slider-bar to change the transparency of the **CAD-SOLID** on the **CAD-LINE**. Experiment!

SOLID COLOR:

Click the **SOLID COLOR** button to open and edit the color of the **CAD-SOLID** with the Windows® color picker,

NUMBER OF VERTICES: (**read-only**)

Complex files have more vertices. **MechDesigner** may become slow to respond to new commands when there are **many CAD-SOLIDS**, each with many NUMBER-OF-VERTICES.

How many is too many? Try to make NUMBER-OF-VERTICES < 10,000.

Experiment with the FINE, COARSE and CUSTOM radio-buttons in the SOLIDWORKS tab 302.

See also: Top-Tips

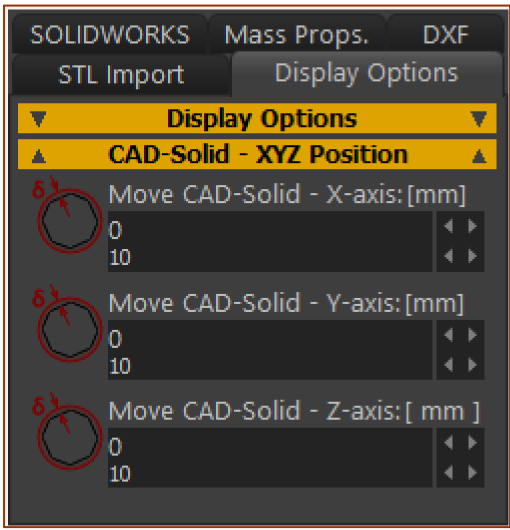
Top Tips:

Make a new SOLIDWORKS Configuration if there are many features that add vertices for cosmetic reasons.

In the new configuration, reduce the number of vertices:

- Remove Fillets, cosmetic fillets.
- Use Chamfers to show model edges.
- Do not import models with balls in ball bearings!
- Make the model as simple as possible.

CAD SOLID XYZ POSITION



If the imported **CAD-SOLID** is not in the correct place, you can move it in the X, Y, and Z-axis directions.

Note:

You can **not** rotate an imported **CAD-SOLID**

You must rotate the CAD model in your CAD software **before** you import it.

1.10.15 Dialog: 3D-Cam

3D-Cam

See Add 3D Cam[113]

Note: After you open a model that includes a **3D-CAM**, explore the **ASSEMBLY-TREE** to open the **3D-CAM DIALOG**, and click the Rebuild button to show the **3D-CAM** in the model.

How to open the 3D-Cam dialog-box ...

To open the **3D-CAM DIALOG**:

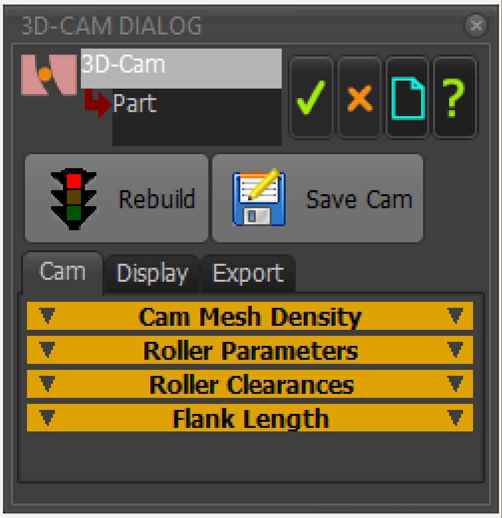
1. Click a **3D-CAM** in the graphic-area or **ASSEMBLY-TREE**.
2. Right-click the **3D-CAM** in the **SELECTION-WINDOW**
3. Click **Edit element** in the shortcut menu

OR

1. **See How to Open a dialog**[513].

The **3D-CAM DIALOG** is now open.

3D-Cam dialog



3D-Cam dialog

There are two buttons at top of the **3D-CAM DIALOG**:

Rebuild Save Cam

See below[323] for details.

There are three tabs:

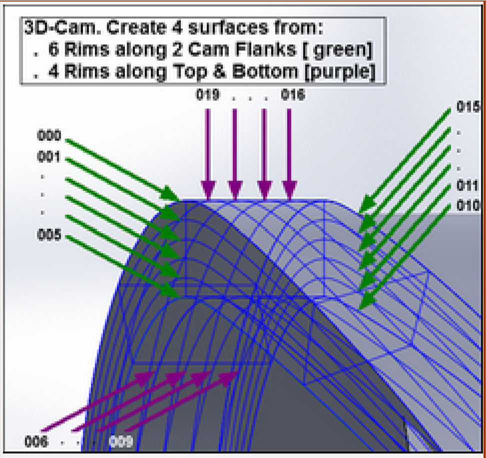
Cam[315]

Display[321]

Export[315]

CAM tab

CAM SURFACE MESH DENSITY



Rims along the Cam-Flanks (Purple), the Top and Bottom (Orange), and Points along each Rim.

PREAMBLE:

Four(4) Cam Faces

We specify for you **3D-CAMS** with four(4) cam faces. The four (4) cam faces are the:

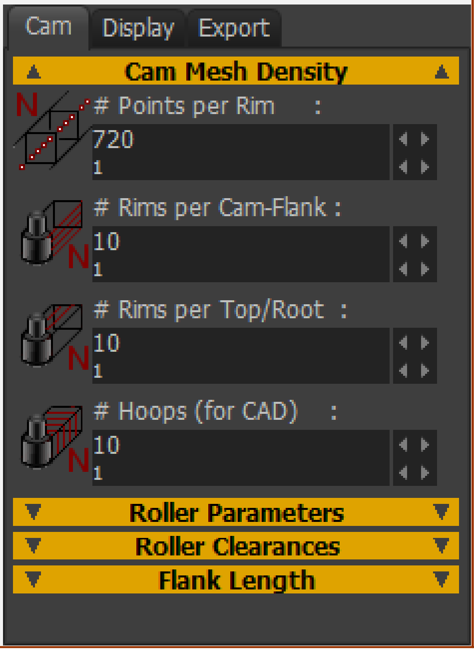
- **Two working faces** (also called **Cam-Flanks):** in contact with the **Cam-Roller**
- **Root-Face**: the floor of the cam. It should be deeper into the cam than **Cam-Roller**
- **Top Face**: you do not need this face in the actual cam. We add it to close the **3D-CAM** in the model.

Rims and Points

Rims: along the total length of each Cam face.

Points: We specify each **Rim** for you with **Points**.

You specify the number of **Rims** and the number of **Points** you want to save with the CAD model.



 # POINTS PER RIMS:
The number of **Points** along each **Rim**

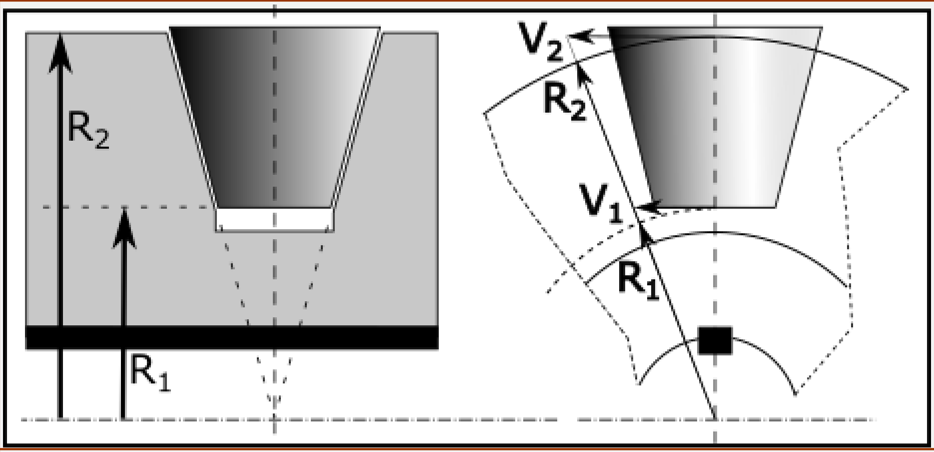
 # RIMS PER FLANKS:
The number of **Rims** across each flank (working faces) of the Cam

 # RIMS PER TOP & BOTTOM:
The number of **Rims** across the Root and Top faces (non-working faces) of the **3D-CAM**

There are also **Hoops** around the the **Rims**.

 # HOOPS (FOR CAD)
The number of **hoops** that wrap around the **3D-CAM** and connect the same **Point** along each **Rim.** We use the **hoops** to help SOLIDWORKS.

ROLLER PARAMETERS



PREAMBLE:

These parameters are for a **Tapered Cam-Follower Roller**.

$V = r.\omega$ [Linear Velocity (V) = Radius(r) × Angular Velocity (ω)]

Cam Flank Velocity: Different points on the cam-flank surface are at different radii from the cam's rotational axis.

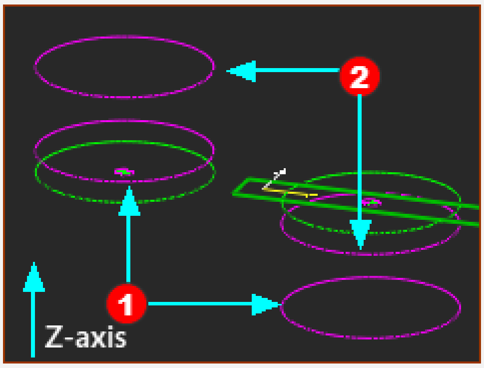
Therefore, a point that is at a large radius from the Cam's rotational-axis has a greater velocity than a point that is at a small radius.

Cylindrical Follower Velocity: the surface of a cylindrical follower bearing are at equal radii from its own rotational-axis. Therefore, the tangential velocities of all points on the surface of a rotating cylindrical roller are identical.

The velocity of the cam's surface and the follower's surface can only be the equal at one contact radius from the center of the Cam. The follower must be skidding at all other points!

Tapered Follower Velocity: the surface velocity of a tapered-roller increases as its radius increases. If its apex is coincident with the rotational-axis of the cam, it can have a surface speed that is equal to the surface speed of the cam. Thus, it can roll on the cam surface as the cam rotates. This is be possible only during the dwell periods.

Nevertheless, cylindrical or barrel rollers are most often used as they are readily available, cheap, and they can withstand high loads.



Two Extrusions - different Extrusion-Offset

The Roller Diameter of a Tapered Roller?

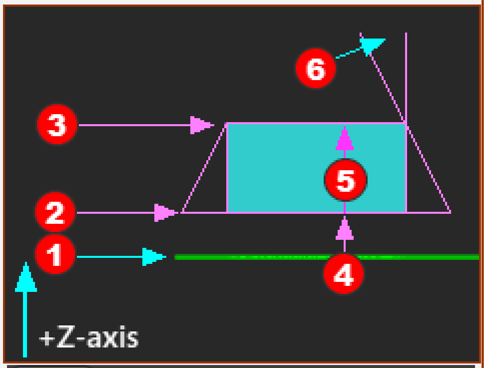
First, review the EXTRUSION-OFFSET and EXTRUSION-DEPTH parameters that define the PROFILE EXTRUSION of the Cam-Follower Roller.

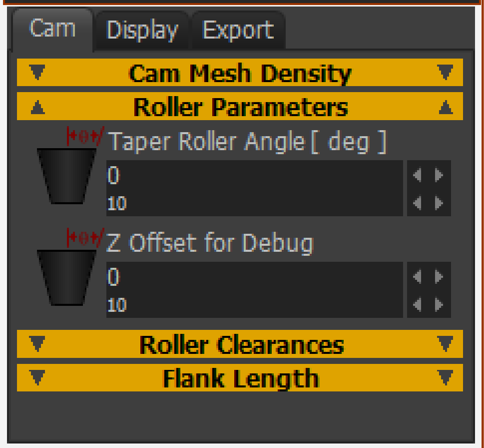
Use the EXTRUSION DIALOG 298 to make the EXTRUSION-DEPTH parameter equal to the Width or Length of the Cam-Follower Roller (Frequently, the **'B'** parameter in bearing catalogs).

Extrusion dialog: (Note: PART-OFFSET parameter = 0 mm).

EXTRUSION-OFFSET (mm): Position (Z-axis) to the Primary-Contour❶ from the Mechanism-Plane

EXTRUSION-DEPTH (mm): = Dimension between the Primary❶ and Secondary❷ - Contours. ONLY a positive value





Note: In image to the left, +Z-axis direction is down

TAPER-ANGLE of the Cam-Follower Roller.

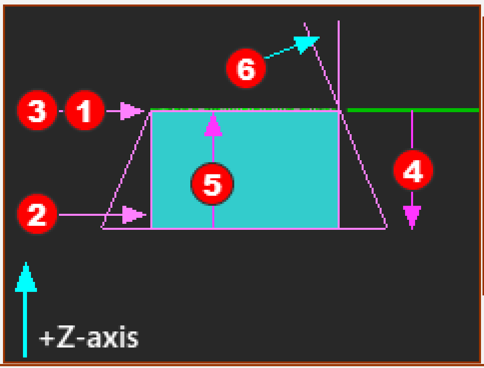
Taper-Angle (degrees) inclines the Flanks of the 3D-CAM. The taper angle pivots about the Secondary-Contour❸.

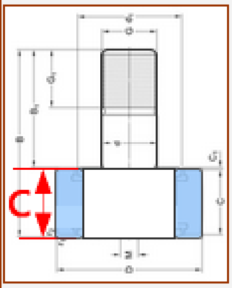
This is not as convenient as the Primary-Contour❷, for which we apologize.

With reference to the image to the left, you can see that a:

- **Negative Taper Angle**❻ (shown in image): the diameter of the Roller **decreases** in the +Z-axis direction.
- **Positive Taper Angle** : the diameter of the Roller **increases** in the +Z-axis direction.

Z OFFSET FOR DEBUG - ignore.





C = Actual Roller Width

Recommendation , if you want to use the Taper-Roller parameter:

Assume the MECHANISM-PLANE of the Cam-Follower is half (mid-)way through the width of the Cam-Follower. It if is not, then change the dimension below to suit your model.

STEP 1

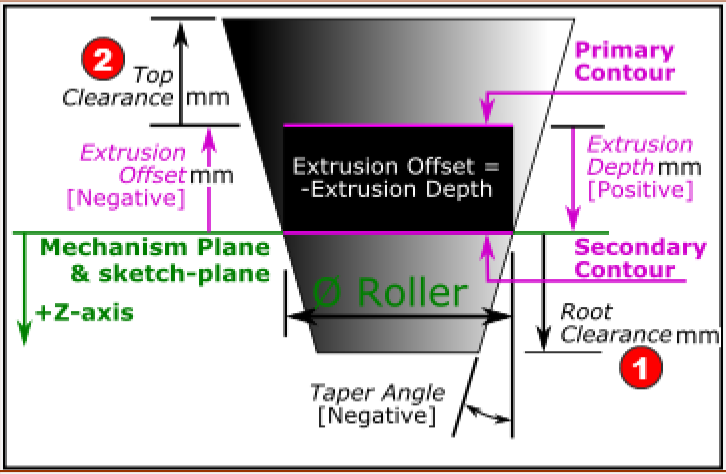
Edit the Cam-Follower EXTRUSION with the EXTRUSION DIALOG

The EXTRUSION-OFFSET❹ should be the **negative of the** EXTRUSION-DEPTH❺.

EXTRUSION-DEPTH(mm) = **ACTUAL ROLLER WIDTH**(mm) (Equal to dimension C in SKF catalog

EXTRUSION-OFFSET (mm) = **–** EXTRUSION-DEPTH(mm) (Equal to negative of dimension – C in SKF catalog.

The Secondary Profile❸ is now **on** the MECHANISM-PLANE❶. The TAPER-ANGLE❻ parameter rotates and pivots on the MECHANISM-PLANE.

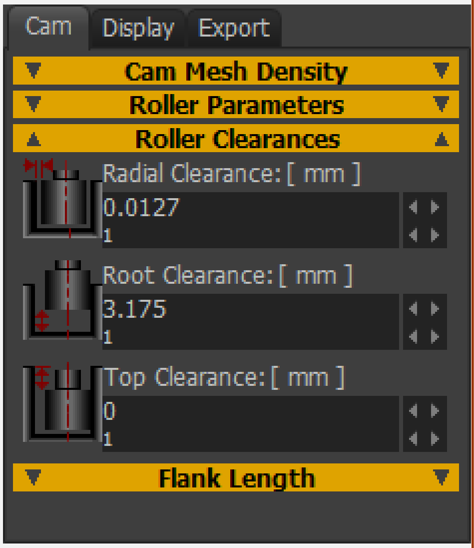


STEP 2:

In the Clearances separator (see below 319):

ROOT-CLEARANCE and TOP-CLEARANCE to extend the flanks as need for the actual roller and 3D-CAM

ROLLER CLEARANCES



PREAMBLE:

There must be a small clearance for a Cam-Follower bearing to roll between the cam's flanks.

Also, you must make sure the Cam-Follower bearing does not touch the Root/Floor of the cam.

Thus, Clearances are practical considerations.

To calculate the basic 3D-CAM (without Roller Clearances), we use for you the:

- **Radial** dimension of the CIRCLE you add to the model to represent the Follower.
- EXTRUSION-DEPTH of the CIRCLE as the depth of the Cam-Flanks
- PART-OFFSET + EXTRUSION-OFFSET of the EXTRUSION from the **MECHANISM-PLANE** to locate the **Cam-Flanks**.

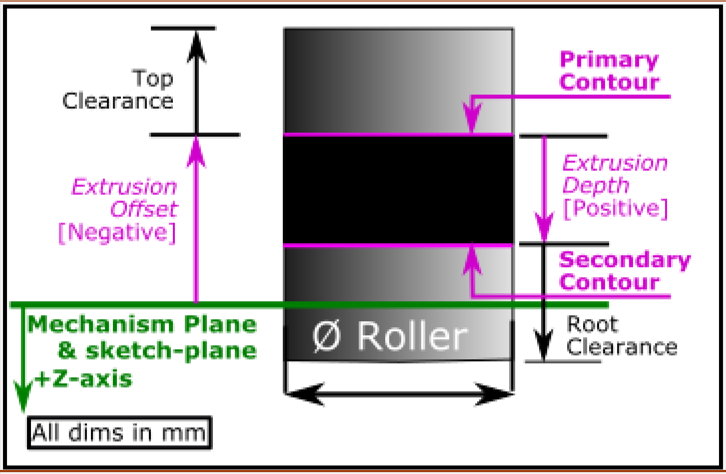
You should use the actual dimensions of the Cam-Follower Bearing, and use ROLLER CLEARANCES to give the clearances you need.

RADIAL-CLEARANCE (mm)

IMPORTANT: Total Clearance = 2 × RADIAL CLEARANCE.

A positive RADIAL CLEARANCE moves the cam-flanks outwards

A negative Radial Clearance moves the cam-flanks inwards.



ROOT-CLEARANCE:

A positive value that moves the Root-Face towards the center of the Cam -see **Note**.

Add ROOT-CLEARANCE to make sure the **Follower-Roller** does not touch the actual **Root** (the ***floor***) of the cam-groove/track.

TOP-CLEARANCE:

A positive value that moves the Top-Face away from the center of the Cam - see **Note**.

Add TOP-CLEARANCE to make sure the top-face of the model is outside the Cam-Blank

Note:

The **Top-Face** may be the **Root-Face** of the Cam - vice-versa!

To find which is which, Experiment! Enter a value for the ROOT-CLEARANCE and click the REBUILD button. Observer in the model which face moves.

FLANK LENGTH

PREAMBLE:

Flank Length parameters apply to Indexing Cams.

When you add a 3D-CAM, we calculate for you the cam over a complete machine-cycle, 0 – 360. However, cam-follower rollers of indexing cams do not stay in contact with the cam for a complete machine-cycle 0 – 360. The cam-follower rollers progressively engage, then disengage with the cam, in a similar way to gear teeth.

You use the FLANK-LENGTH separator to calculate the cam coordinates for the period when the cam-follower is engaged with the cam.

You may need to edit the Flank Length to calculate the cam for angles that are not from 0 and 360.

FLANK-LENGTH:

START-ANGLE:

- Edit the START-ANGLE

END-ANGLE:

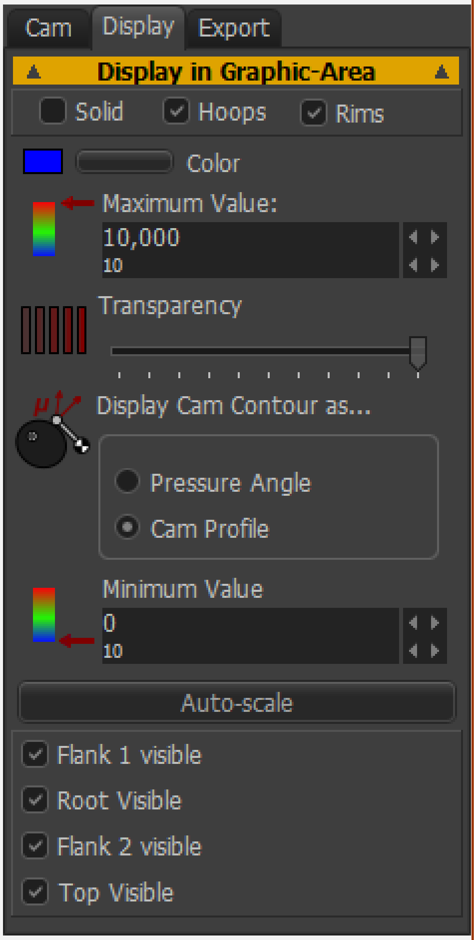
- Edit the END-ANGLE

As you edit the START-ANGLE and END ANGLE, the Cam becomes shorter in length.

The number of points that we calculate for you along each **rim** in the cam does not change. Thus, the points along each **rim** become nearer to each other.

DISPLAY tab

DISPLAY IN GRAPHIC-AREA



PREAMBLE:

The 'Display' edits how the 3D-CAM shows in the graphic-area.

SOLIDS* | HOOPS | RIMS *check-boxes*.

☑ SOLIDS*: show or hide, 3D-Cam as a Solid

☑ HOOPS: show or hide a hoop at each Point along each of the Rims - see Point [315]

☑ RIMS: show or hide the RIMS along the 3D-CAM

* You MUST enable the SOLIDS check-box to see the 3D-CAM in the other **MECHANISM-EDITORS** and the **MODEL-EDITOR**

COLOR

- Use the Windows® color-picker to select a color of the 3D-CAM

TRANSPARENCY

- Use the slider to change the TRANSPARENCY of the 3D-CAM.

DISPLAY CAM-CONTOUR AS...

- PRESSURE-ANGLE
- CAM-PROFILE

When you show the 3D-CAM as PRESSURE ANGLE, the faces have a color-code.

- **Red** identifies a high PRESSURE-ANGLE
- **Blue** identifies a low PRESSURE-ANGLE

MINIMUM / MAXIMUM

By default, the color-code scales to the maximum and minimum values of the PRESSURE-ANGLE

If you want to change the scale, use the Maximum Value and Minimum Value boxes.

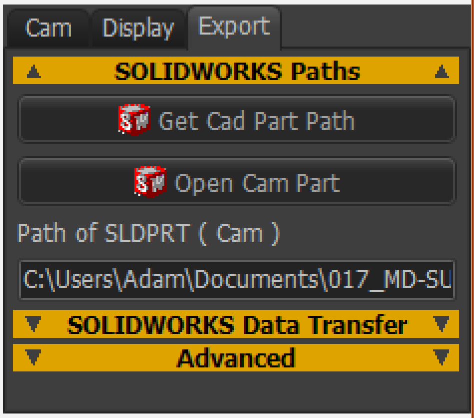
AUTO-SCALE BUTTON

Click the auto-scale button to reset to the maximum and minimum values again.

EXPORT tab

NOTE: From **MD13.2**, it is easier to save the 3D-CAM AS a **STEP** file with the Save button [323].

SOLIDWORKS PATHS



PREAMBLE:

It is often useful to link a SOLIDWORKS document file-name with the **3D-CAM.**

GET CAD PART PATH

Link the active SOLIDWORKS document file-name with the 3D-Cam dialog.

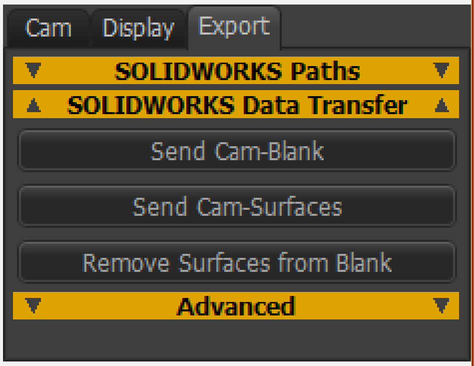
The file-name is put into the **Path of SLDPRT (Cam)** box

OPEN CAM PART

Open, in SOLIDWORKS, the file-name of the SOLIDWORKS document that is in the *Cam Part Path* box

SOLIDWORKS DATA TRANSFER

Note: From **MD13.2**, it is **much** easier to save the **3D-CAM** to a **STEP** file with the Save button [323].



PREAMBLE:

We recommend you use save as a STEP file - See Rebuild and Save buttons [323]

Alternatively, you can use these buttons to transfer the 3D-Cam data to SOLIDWORKS.

1. Open SOLIDWORKS
2. Add a new SolidWorks Part and make it the active document.
3. Press the three buttons in the **SOLIDWORKS DATA TRANSFER** separator

SEND CAM BLANK

- We export for you the sketch of the **Cam-Blank** and its **Axis-of-Rotation** to SOLIDWORKS.
- We instruct SOLIDWORKS for you to add a revolved feature with the **Cam-Blank** and **Axis-of-Rotation**.

Note: Usually, the sketch of the **Cam-Blank** is symmetrical about the middle of the sketch. If the sketch is not symmetrical (top to bottom), then it is possible that SOLIDWORKS puts the sketch up-side-down with respect to the **3D-CAM**. To flip the Cam-Blank: (in SOLIDWORKS), mirror the sketch about the Y–axis and then the X–axis.

SEND CAM SURFACES Note you can also save the Cam as a STEP file, to import directly into any CAD.

- We export for you each Rim and Hoop
- We instruct SOLIDWORKS to:
  - Create a surface from the Rims for each face and flank of the Cam
  - Add end-caps to the surfaces, if it is an Indexing-Cam [320]
  - Knit the surfaces together

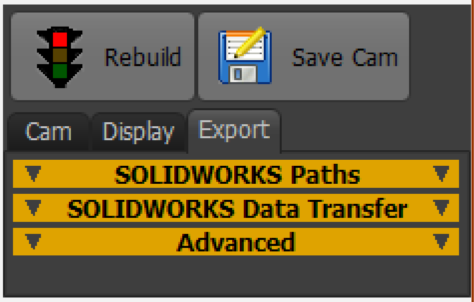
REMOVE SURFACES FROM BLANK

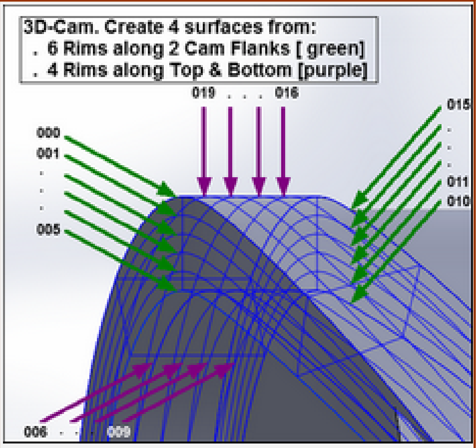
- We instructs SOLIDWORKS to:
  - Cut the knitted surfaces into the Cam Blank
  - Hide the knitted surfaces

Note: You may need to edit the Surface Cut feature to reverse the direction of the cut to leave the cam is a cam-track.

REBUILD and SAVE buttons

Rebuild and Save Cam buttons





Rims along the Cam-Flanks (Purple), the Top and Bottom (Orange), and Points along each Rim.

Rebuild button:

- to recalculate the **3D-CAM** with updated parameters
- before you transfer the **3D-CAM** to SOLIDWORKS
- before you save the **3D-CAM**

Save Cam button:

- to Save as STEP, TXT or SLDCRV file-types

STEP (recommended).

You can open the STEP file in most **3D CAD** programs.

See **EXAMPLE 1 : Export the 3D-Cam as a STEP file**, below.

TXT, SLDCRV, DAT

Each Rim is saved as a separate file.

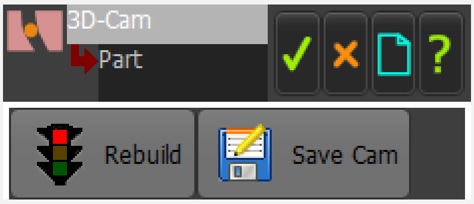
The XYZ coordinates of each Rim are saved to a different file.

For example:

In the image to the left, there 20 Rims and there is one file to each Rim.

EXAMPLES - Export 3D-CAM

Example 1: Export the 3D-Cam as a STEP file (RECOMMENDED)



IN MECHDESIGNER:

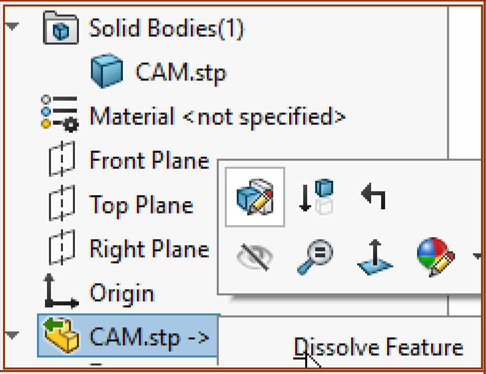
In the **3D-CAM DIALOG:**

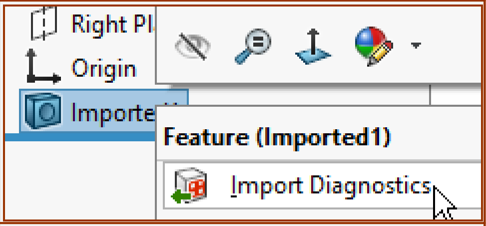
1. Click **Rebuild button**
2. Click **Save button**
3. Save the file-type as ***.STP**, and save the cam - for example, **CAM.STP**

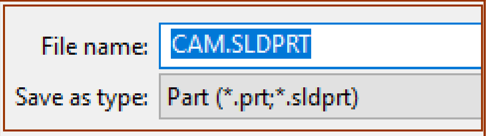
Note: STP = STEP

SOLIDWORKS

1. Open **Cam.STP** file in SOLIDWORKS®
2. If you are prompted, do **Import Diagnostics**.







3. Close **Import Diagnostics** command

Depending on your SOLIDWORKS® settings, you may need to:

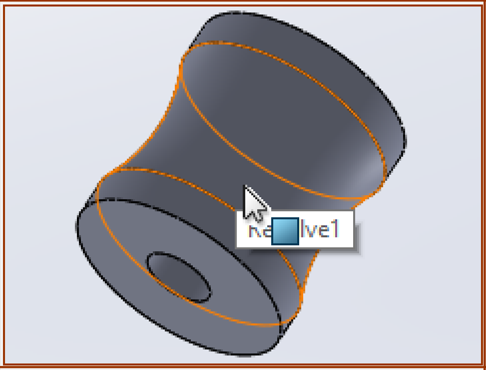
a. Right-Click **Cam.STP** in the **Feature Manager**
b. In the shortcut menu: Click **Dissolve Feature** and **Yes, break the link**
c. Right-Click - again - the **CAM.STP** in the **Feature-Manager**
d. In the shortcut menu: Click **Import Diagnostics**

In the **Import Diagnostics dialog**

e. Click the **Attempt to Heal All** button, and then close the **Import Diagnostics** command.

When the Cam is ***healed:***

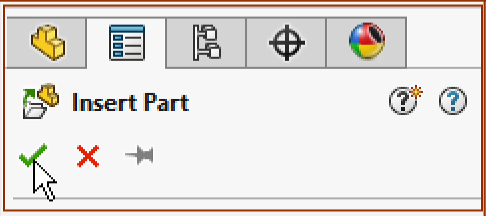
4. Save **Cam.STP** as a SOLIDWORKS® file-type. E.g. **Cam.SLDPRT**.



IN SOLIDWORKS

Open the Cam-Blank:

1. Open the prepared SOLIDWORKS® part **Cam-Blank.SLDPRT**.



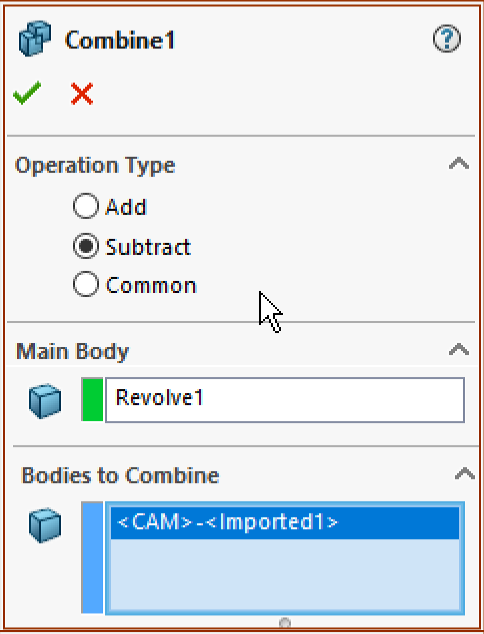
Insert the Cam.SLDPRT file (see PART 1)

1. **Insert menu** > **Part...** ; select "**CAM.SLDPRT**"
2. Click the **TICK** at the top of the SOLIDWORKS feature manager.

Cam.SLDPRT becomes a feature and a **Solid Body (Cam<imported)**

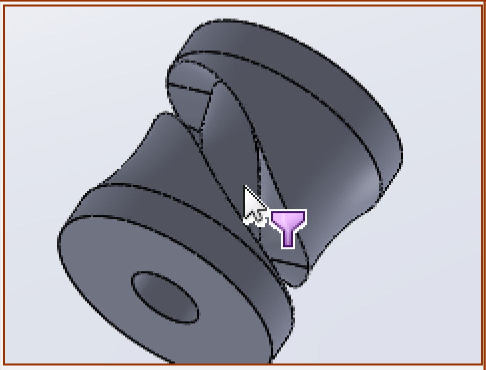
Ideally the orientation of the **CAM.SLDPRT** is correct relative to the **Cam-Blank.SLDPRT**

If not, I find it easier to move the sketch of the **Cam-Blank** to the correct Plane - experiment!



Finally:

1. **Insert menu** > **Feature** > **Combine:**
   a. Select **Subtract** as the **Operation Type** and,
   b. In the **Bodies to Combine** box, select
      i. **Cam-Blank** as the **Main-Body** and
      ii. **Cam<imported>** as the **Body to Subtract**.



You should now see the **Cam-Track** in the **Cam-Blank**.

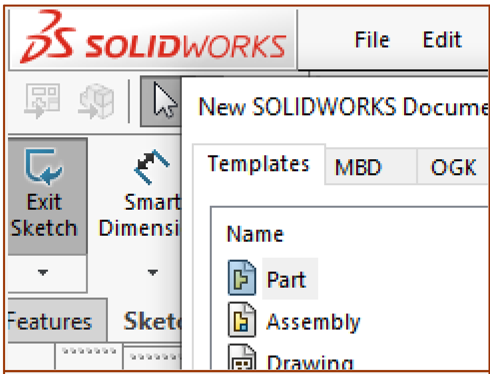
Note: It is possible that the Solid Body "**Cam<imported>**" is not in the correct place relative to the **Cam-Blank**. Most usually, **Cam<imported>** and **Cam-Blank.SLDPRT** are defined on the incorrect Planes.

In this case, redefine the sketch-planes of "**Cam.Blank**" (easiest method) or move "**Cam<imported>**" (more difficult to do)

To move **Cam<imported>**, do:

1. **Insert menu** > **Feature** > **Move/Copy...** and select **Cam.SLDPRT** as the body to move.

Example 2: Transfer the data directly to SOLIDWORKS



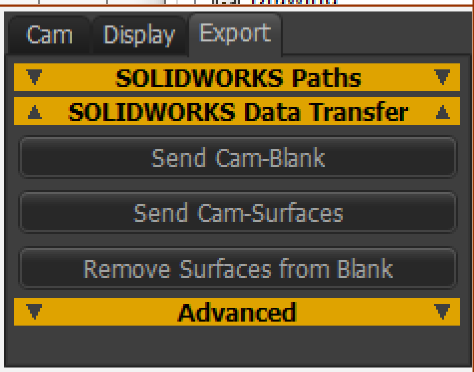
Not usually recommended - we recommend that you save the Cam as a STEP file - see EXAMPLE 1 - above.

In SOLIDWORKS®, make sure the active document is a Part document.

STEP 1: **Open** SOLIDWORKS®

STEP 2: **Add a new Part**

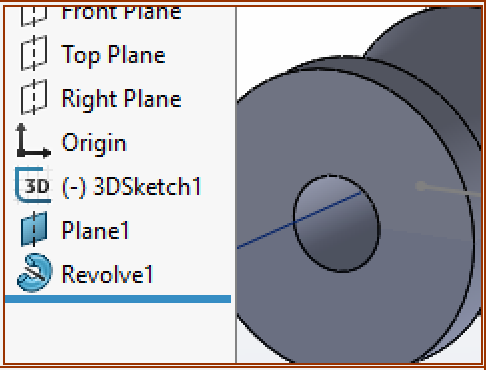
STEP 3: **Save the new Part**



STEP 4: Click the SEND CAM BLANK button

Before you can add the 3D-CAM in **MechDesigner**, you must add a sketch-loop (to represent the rotational-section of the Cam-Blank) and a LINE (to represent the rotation-axis of the Cam-Blank).

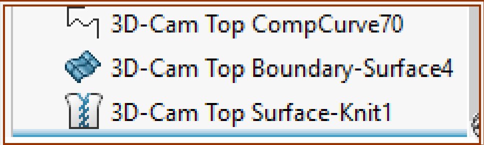
We send for you these sketch-elements to SOLIDWORKS to create the Cam-Blank feature, when you click SEND CAM-BLANK.

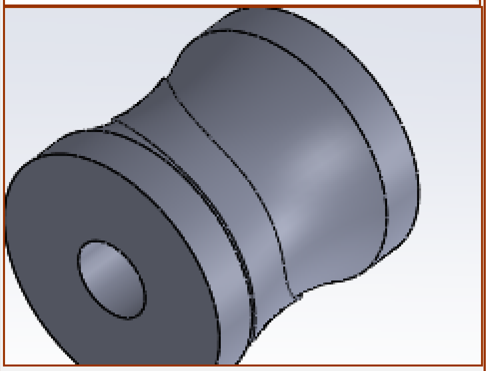


See Getting Started Tutorials 6C for more information.

In the image to the left, you can see in SOLIDWORKS:

- **3DSketch1** which is the rotational-axis
- **Revolve1** which is the feature that revolves the sketch-loop into the Solid.





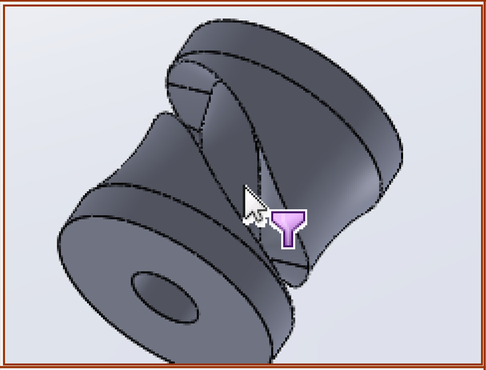
STEP 5: Click the **SEND CAM SURFACES** button

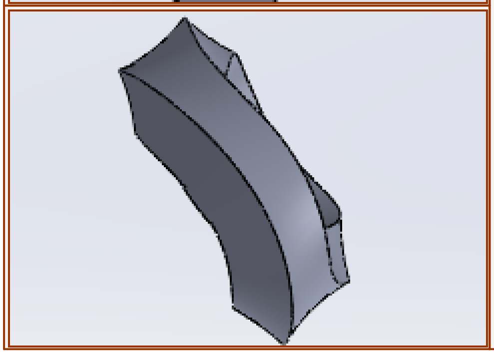
To make the transfer quicker, show a drop-down menu in SOLIDWORKS®. For example, show the help menu drop-down.

We do not know why this makes the transfer faster, but it works.

We control SOLIDWORKS to:

1. Add the Rims as **Curve** entities
2. Add four Surfaces features (or six if the Flank-Lengths are not from 0 – 360)
   a. two Cam Flanks; a Floor, and a Top
   b. plus two 'End-Caps' **if** the Flank-Lengths are not from 0 – 360.
3. Add the Knit-feature to knit the Surfaces





STEP 6: Click **REMOVE SURFACES FROM CAM** button

The **3D-Cam Surfaces** Cut into the Cam Blank.

WAIT! - you should now see the image to the left

If you see ONLY the CAM TRACK and not the Finished Cam **(see image below)**

STEP 7: **Save the Part In** SOLIDWORKS®

In SOLIDWORKS®:

1. **Insert** > **Cut** > **With Surface**
2. Select the **Knitted 3D-Cam Surface** - the last feature in the **Feature Manager**.
3. Edit the **Surface Cut** feature in SOLIDWORKS® and click the Change Direction button.
4. Hide the **Knitted Surface** feature of the **3D-Cam**.

1.10.10 Dialog: 2D-Cam >

2D-Cam

2D-Cam: Work-flow

1. Add a 2D-CAM - **see Machine elements toolbar > Add 2D-Cam** [109]

If the new 2D-CAM is one from a pair of **Conjugate-Cams**, or it is one flank of a **Groove-Cam**

1.a. Add a CONJUGATE-CAM FB - **see Machine elements toolbar > Add Conjugate Cam FB** [139]

1.b. Edit the CONJUGATE-CAM FB to select least two 2D-CAMS - **see Conjugate-Cam dialog** [353].

2. Select a 2D-CAM or a CONJUGATE-CAM FB as the Power Source for the Follower - **see Configure-Power Source** [480]
3. Review the 2D-CAM : **Display, Properties, Roller-Life, Cam-Life, ...** - **see 2D-Cam dialog** [327]

4. Add a CAM-DATA FB - **see Kinematic FBs > Add Cam-Data FB** [171]
5. Edit the CAM-DATA FB to link it to the 2D-CAM- - **see Cam-Data dialog : Cam Analysis** [357]
6. Connect wires from the CAM-DATA FB to a GRAPH FB to analyse 5 parameters - **see Cam-Data dialog : Cam Analysis** [357]
7. Edit the CAM-DATA FB again to calculate the Cam's Coordinates - **see Cam-Data dialog : Cam-Coordinates** [361]

How to open the 2D-Cam dialog

To open the 2D-CAM DIALOG:

1. Double-click a 2D-CAM in the graphic-area or **ASSEMBLY-TREE**.

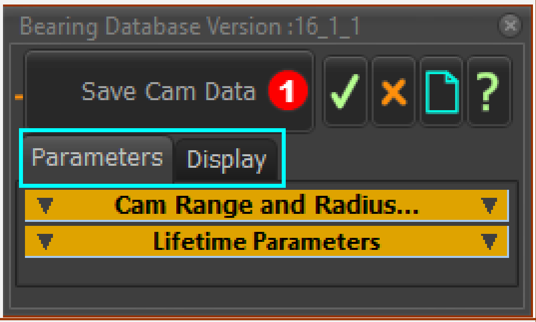
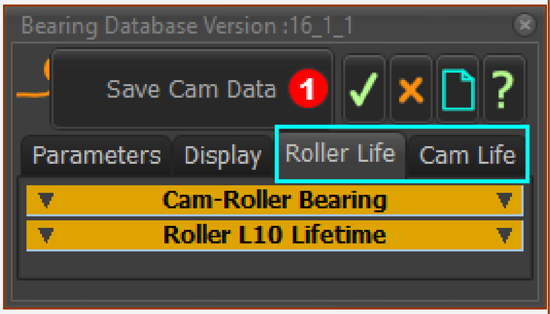
OR

1. **See How to Open a dialog** [513].

The 2D-CAM DIALOG is now open.

2D-Cam dialog

The 2D-CAM DIALOG has two formats:

SHORT	EXTENDED
	
The two tabs are: **Parameters** [329] **Display** [332]	The two NEW tabs are: **Roller Life** [336] **Cam Life** [347]

❶ Click Save Cam Data to save the data that you calculate with this dialog.

Save Cam Data button

IMPORTANT: Do not use the X and Y Points that you can save with this dialog to manufacture the cam. Add and use a **CAM-DATA FB** to calculate the Cam's Profile for manufacture as XY-Point, Biarcs, or save it as a STEP file.

Save Cam Data

Click **Save Cam Data** at the top of the dialog to save the data produced by this dialog to a CSV file-type. The CSV file also saves the data at each step in the machine cycle - see Machine-Settings dialog > Number of Steps[291]. The CSV saves the Roller and the Cam details when the Lifetime is enabled.

Time	X-Coordinates	Y-Coordinates	Contact Force	Contact Shear Stress	Radius of Curvature	Pressure Angle	Sliding Velocity	Cam Angular Velocity	Interferes?
s	mm	mm	N	MPa	mm	degree	mm/s	rad/s	Y/N

See also the output connectors of the Cam-Data FB[357] for further Cam-Analysis.

2D-Cam: Work-flow

1. Add a 2D-CAM - **see Machine elements toolbar > Add 2D-Cam**[109]

If the new 2D-CAM is one from a pair of **Conjugate-Cams**, or it is one flank of a **Groove-Cam**

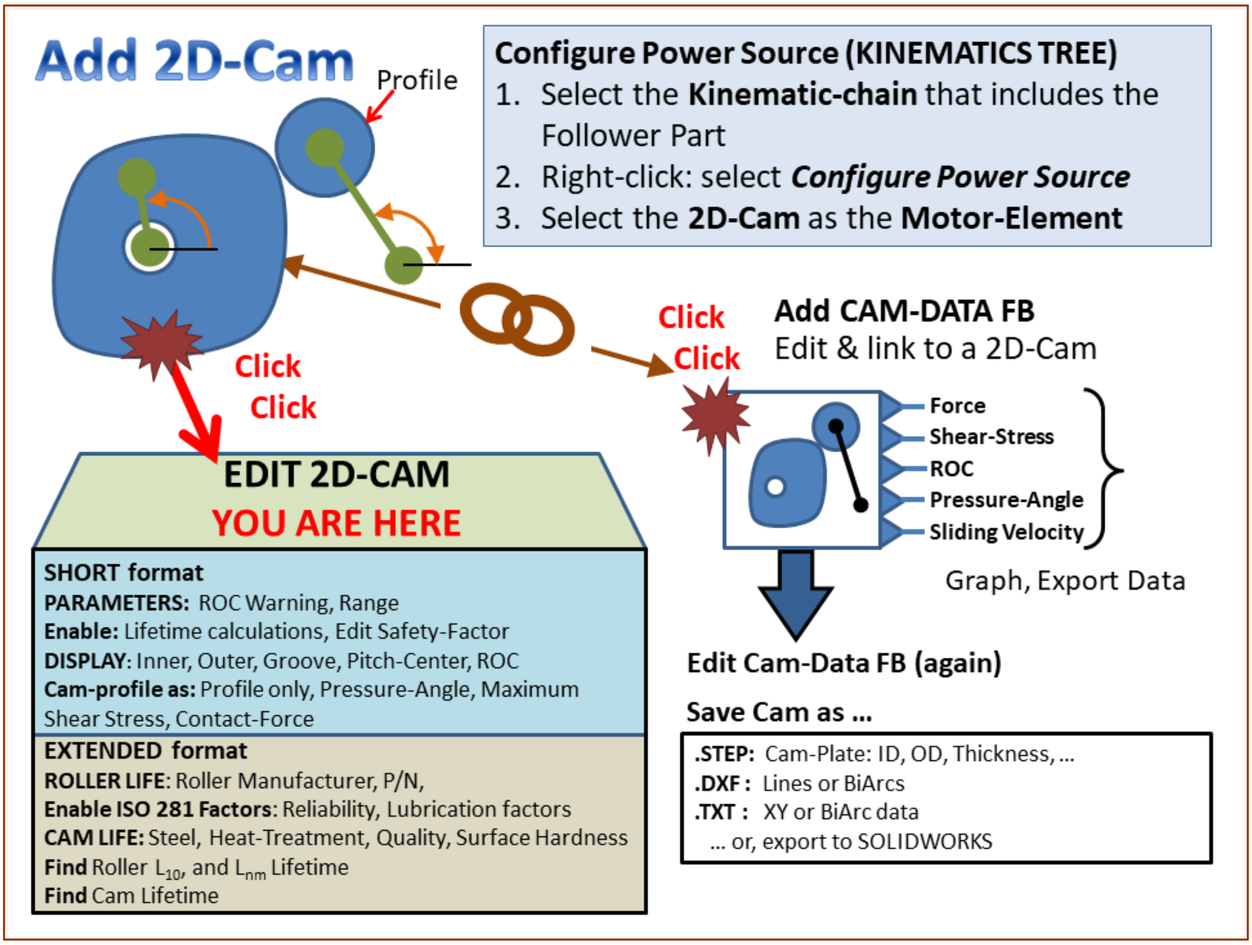
1.a. Add a CONJUGATE-CAM FB - **see Machine elements toolbar > Add Conjugate Cam FB**[139]

1.b. Edit the CONJUGATE-CAM FB to select least two 2D-CAMS - **see Conjugate-Cam dialog**[353].

2. Select a 2D-CAM or a CONJUGATE-CAM FB as the Power Source for the Follower - **see Configure-Power Source**[480]
3. Review the 2D-CAM : **Display, Properties, Roller-Life, Cam-Life, ...** - **see 2D-Cam dialog**[327]

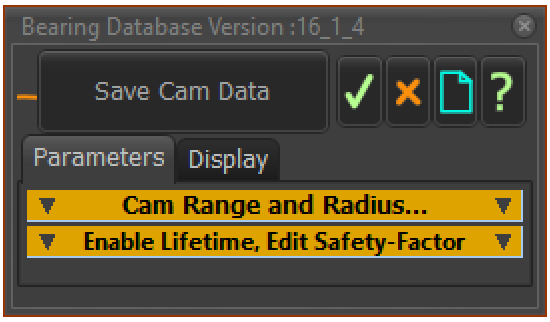
4. Add a CAM-DATA FB - **see Kinematic FBs > Add Cam-Data FB**[171]
5. Edit the CAM-DATA FB to link it to the 2D-CAM- - **see Cam-Data dialog : Cam Analysis**[357]
6. Connect wires from the CAM-DATA FB to a GRAPH FB to analyse 5 parameters - **see Cam-Data dialog : Cam Analysis**[357]
7. Edit the CAM-DATA FB again to calculate the Cam's Coordinates - **see Cam-Data dialog : Cam-Coordinates**[361]

2D-Cam - work-flow as a diagram



1.10.17 Parameters tab

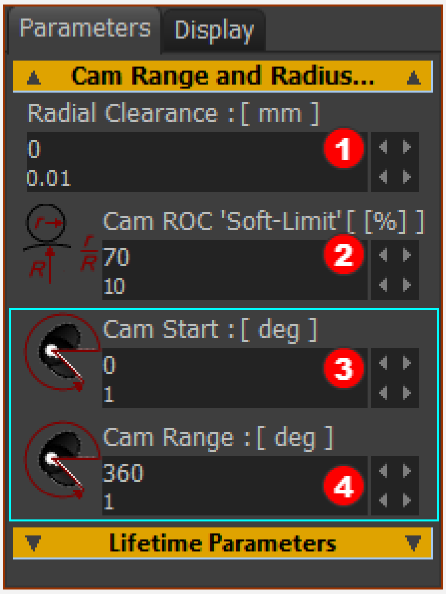
2D-Cam dialog: Parameters tab



2D-Cam dialog > Parameters tab

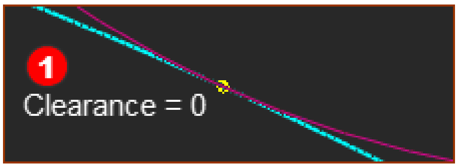
Parameters tab

CAM RANGE AND RADIUS...

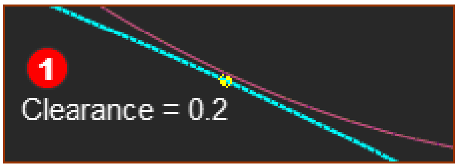


Cam Range and Radius tab

❶ RADIAL CLEARANCE (MM) Default = 0



Cam without Radial Clearance



Cam with 0.2mm Radial Clearance

When RADIAL CLEARANCE ≠ 0 we calculate for you a Cam-Profile that is offset from the true Cam-Profile.

Examples RADIAL CLEARANCE ≠ 0

1: Groove-Cam: RADIAL CLEARANCE = +0.05 : a small clearance for a Roller bearing to fit between the Cam-Flanks of a Groove cam-type.

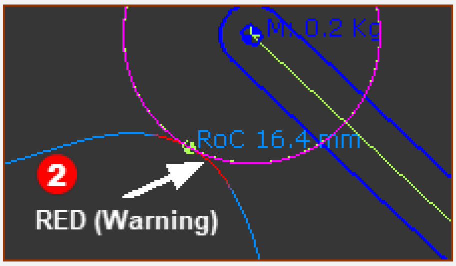
2: Conjugate-Cam: RADIAL CLEARANCE = +0.05 : a small clearance to compensate for assembly tolerances with a Conjugate cam-type.

3: Rough-Cut: RADIAL CLEARANCE = –0.5 : to export data for a rough-cut of the cam (opposite to clearance)

Notes:

a) You can also edit this parameter in the Cam-Coordinates dialog [361].

b) Stud-type Cam-Followers usually have a large negative tolerance bias - for example. $0 - 50\mu m$ - which may be enough, or too much, clearance for a Groove cam-type.



Cam-Profile RED = Warning

❷ CAM ROC 'SOFT-LIMIT' (%)

Default: see **Application Settings** 284 **> General tab > Cam Radius Soft-Limits = Default value.**

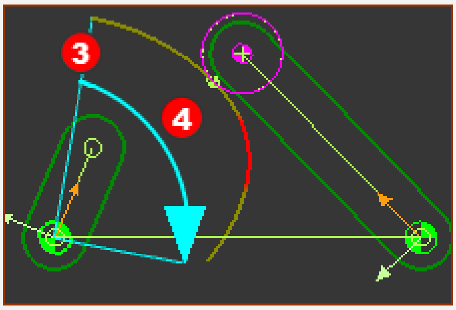
This parameter compares the Cam's radius of curvature (ROC) with the Roller's radius.

Warning if ... $\left|ROC_{cam}/R_{foll}\right| \times 100\% < (Soft - Limit)\%$

... then the **Cam-Profile** is RED as a warning.

FOR EXAMPLE: CAM ROC 'SOFT-LIMIT' = 100%

The **Cam-Profile** is RED where the **Cam's Radius-of-Curvature** is less than the **Roller's Radius.**



Cam: Range < 360

❸CAM START - Default = 0 ; ❹CAM RANGE - Default = 360

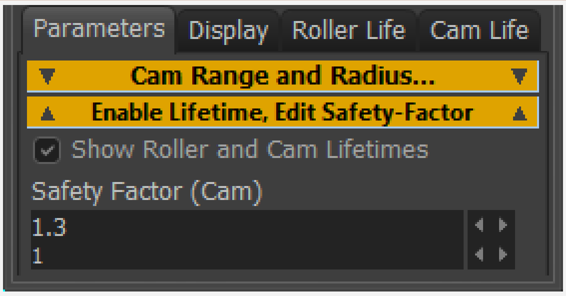
- Use CAM-START to start calculating the Cam-Profile at a different MASTER MACHINE ANGLE.
- Use CAM-RANGE to display the cam for less than 360° of the machine-cycle

Application examples:

E.G.1: A Cam-Part rotates 2 times / machine-cycle, then enter CAM START: 0 and **CAM RANGE : 360 / 2** (REVS)

E.G.2: To hide one side of a **Slot-Cam.**

ENABLE LIFETIME, EDIT SAFETY-FACTOR



Enable Show Roller and Cam Lifetimes

☑SHOW ROLLER AND CAM LIFETIMES

If you enable SHOW ROLLER AND CAM LIFETIME, the 2D-CAM DIALOG has two new tabs:

- ROLLER LIFE - **see** **Roller Life** [336]
- CAM LIFE - **see** **Cam Life** [347]

With these tabs:

- We can calculate for you the life of the **OUTER OR** the **INNER** cam-profile. We remove the **GROOVE** options in the **Display tab**
- We display **Force Vectors**. To hide **Force Vectors**, see Forces toolbar > Force Vectors - Display [198]

See Display tab > CAM DISPLAY OPTIONS [333]

SAFETY-FACTOR (CAM)

We apply for you the **SAFETY-FACTOR** to the **Cam's Contact-Stress** and the **Roller's Equivalent Contact-Load**

Cam Life tab

Cam's Dynamic Maximum Contact-Stress = Max Contact-Stress × SAFETY-FACTOR (CAM)

Roller Life tab

SAFETY-FACTOR (Roller) = SAFETY FACTOR (CAM)^{2}

∴ Roller's Dynamic Equivalent Load = Equivalent Load × SAFETY FACTOR (CAM) × SAFETY FACTOR (CAM)

E.g: If **SAFETY-FACTOR (CAM)** = 1.1 ; **SAFETY-FACTOR (ROLLER)** = 1.21

The **SAFETY FACTOR** should consider at least the following.

What drives the Cam-Shaft - is it a constant speed servo-motor or a single-cylinder diesel engine.

What the Cam-Follower drives - from a simple paper-tucker to a forging, stamping press?

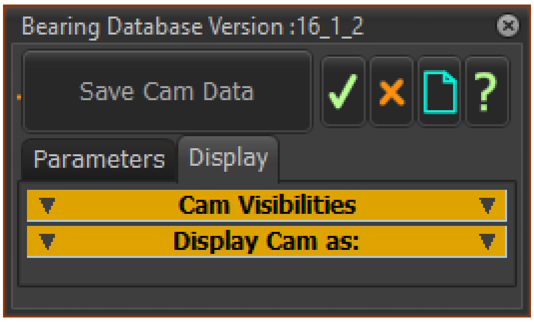
What is the stiffness, rigidity, inertia, natural-frequency, of the load - from long, thin shafts to short, stub spindles

What is the severity of the motion - from jerk continuity to acceleration discontinuity

Is there any backlash in the driving and/or the driven system. - from poor quality gears to preloaded bearings, coupling gears.

1.10.18 Display tab

2D-Cam dialog: Display tab



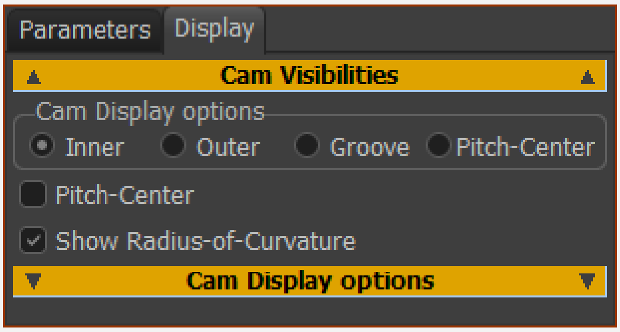
2D-Cam dialog > Display tab

Display tab

CAM VISIBILITIES

There are two formats: **SHORT** and **EXTENDED**.

SHORT : CAM VISIBILITY - when you DISABLE SHOW ROLLER AND CAM LIFETIMES **(see Parameters tab)**



Cam Visibilities when Show Roller and Cam Lifetimes is NOT enabled

Select :

⊙ INNER - Inside / Internal cam-profile only

⊙ OUTER - Outside / External cam-profile only

⊙ GROOVE - the INNER and OUTER cam profiles

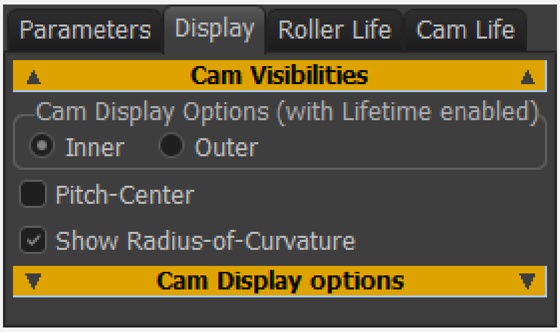
⊙ PITCH-CENTRE - the path of the center-point of a Roller Cam-Follower bearing relative to the **Cam-Part**

You can enable:

❑ PITCH-CENTRE - enable to show the path of the centre-point of a Roller Cam-Follower relative to the **Cam-Part**.

❑ SHOW RADIUS-OF-CURVATURE - enable to show RADIUS-OF-CURVATURE at the contact-point of the **Cam-Profile** and the **Cam-Follower**.

EXTENDED : CAM VISIBILITY - when you ENABLE SHOW ROLLER AND CAM LIFETIMES **(see Parameters tab)**



Cam Visibilities when Show Roller and Cam Lifetimes is enabled

Select:

⊙ INNER - Inside / Internal cam-profile only

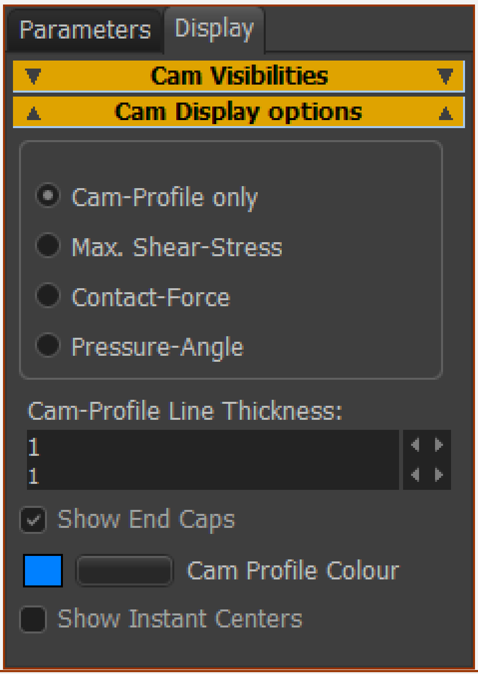
⊙ OUTER - Outside / External cam-profile only

You can enable:

❑ **PITCH-CENTRE** - enable to show the path of the centre-point of a Roller Cam-Follower relative to the **Cam-Part**.

❑ **SHOW RADIUS-OF-CURVATURE** - enable to show the RADIUS-OF-CURVATURE at the contact-point of the **Cam-Profile** and the **Cam-Follower Roller**

CAM DISPLAY OPTIONS



Select one from:

- ⦿ **CAM-PROFILE** (default)
- ⦿ **MAXIMUM SHEAR-STRESS***
- ⦿ **CONTACT FORCE***
- ⦿ **PRESSURE-ANGLE**

* Configure the Power Source 480 to calculate correctly.

CAM-PROFILE LINE THICKNESS: (default =1)

Edit the thickness of the **Cam-Profile.**

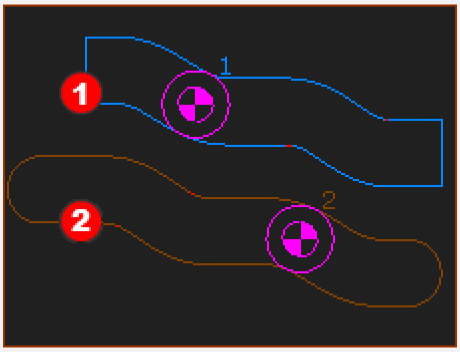
CAM-PROFILE COLOR button

Edit the color of the **Cam-Profile**.

It is useful when there many **Cam-Profiles** in your model if each Cam-Profile has a different color.

TOP-TIP - rename the 2D-CAM element to the color you select for the **Cam-Profile**.

SHOW INSTANT CENTERS : For the kinemagicians!



Without and With End-Caps

Enable **SHOW END-CAPS** check-box

❶ Without End-Caps

❷ With End-Caps

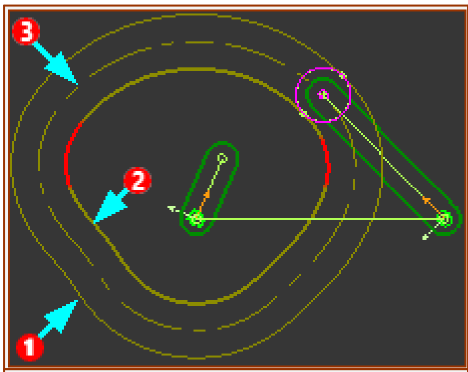
This check-box shows **only** when **MechDesigner** detects an **open-cam**.

An **open-cam** is one in which the start and end of the **Cam-Profile** are not in the same place.

With Slot-Cam types, it is often possible to close the **Cam -Profile** with End-Caps that have radii equal to that of the **Cam-Follower Roller**.

Note: Very often, Slot-Cams do not transfer to SolidWorks. This is because the can be duplicate XY-Points.

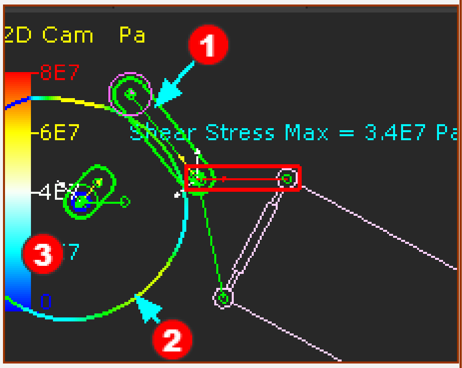
Example Display Options:



⦿ **CAM PROFILE**

Outer❶ and **Inner**❷ cams are continuous lines.

Pitch-Centre Path❸ is a dashed line.



⦿ **MAXIMUM SHEAR-STRESS**

❶ **MAXIMUM SHEAR STRESS** value next to the contact point

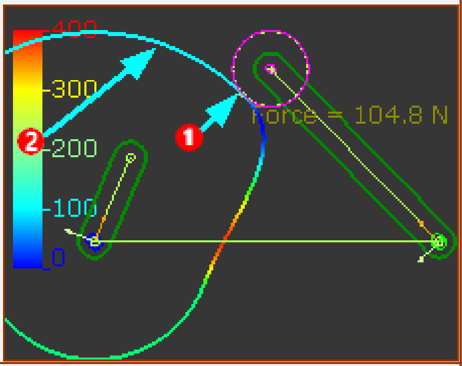
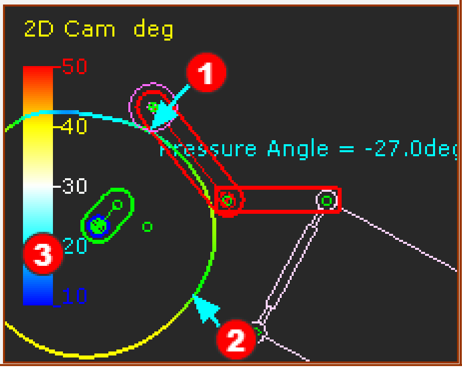
❷ Color of the cam agrees with the color-bar❸

❸ Color-bar indicates the scale of the **MAXIMUM SHEAR-STRESS**

Notes:

MAXIMUM SHEAR-STRESS is below the surface, at 0.78 × Contact Width for Line-Contact.

MAXIMUM SHEAR-STRESS is = 0.3 × Contact Stress for Line-Contact.

	⦿ CONTACT FORCE ❶ CONTACT FORCE is at the contact point. ❷ Color of the Cam and Color-Bar agree with the CONTACT FORCE
	⦿ PRESSURE ANGLE ❶ PRESSURE ANGLE is adjacent to the contact point ❷ Color of the cam agrees with the color-bar❸ ❸ Color-bar as scale of the PRESSURE ANGLE See also **Pressure Angle**

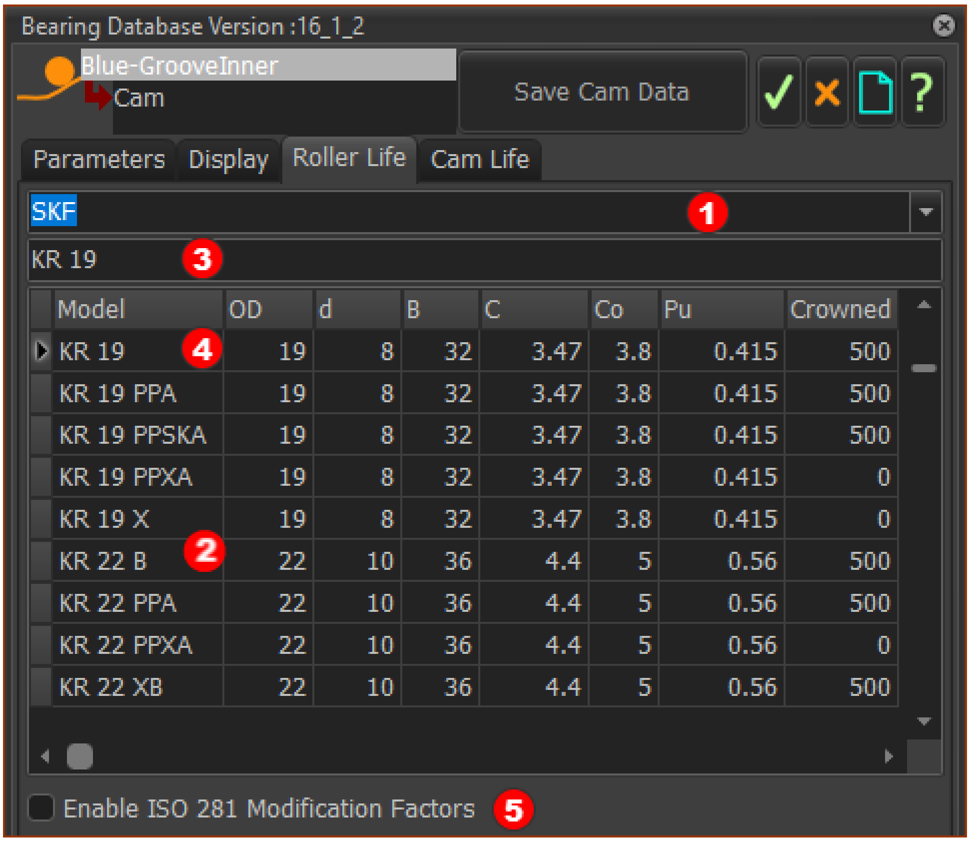
1.10.19 ► Roller Life tab

2D-Cam dialog: Roller Life tab

Note: other catalogue and brand names for **Roller bearings** are **Track-Rollers, Track-Followers,** and **Cam-Followers.**

To **Calculate Roller Life**, you must:

Parameters tab (Parameters tab > 331 ENABLE LIFETIMES, EDIT SAFETY-FACTOR 329)
STEP 1: Enable **SHOW ROLLER AND CAM LIFE**
STEP 2: **Enter a SAFETY FACTOR (CAM)**
Note: The **Roller's Safety-Factor** is equal to the **Square of the Cam's Safety-Factor.**
Top-Tip - Enter SAFETY-FACTOR = 1 to help you the basic Roller Life results, then enterparameters.
Roller Life tab
STEP 3: **Select a Roller bearing manufacturer and Roller bearing**
If you **ENABLE ISO 281 MODIFICATION FACTORS**:
STEP 4: **Select or enter a RELIABILITY FACTOR**
STEP 5: **Enter the OIL OPERATING TEMPERATURE, and OIL VISCOSITY at two other TEMPERATURES (usually 40°C and 100°C)**
STEP 6: **Enter the LUBRICATION TYPE and select the CONTAMINATION LEVEL.**
REVIEW THE ROLLER'S LIFETIME



2D-Cam dialog > Roller Life tab
Select a Manufacturer and Bearing Part-Number of Parameter

Roller Life tab

Select a Roller bearing Manufacturer and Part Number.

STEP 1: Select a Roller bearing Manufacturer ❶

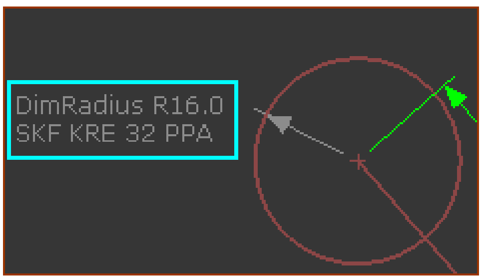
Available Manufacturers : **SKF, INA/FAG, KOYO, NSK, THK**

When you select a manufacturer, their bearing Part/Numbers, and other parameters, show in the table below❷.

STEP 2: Select a Roller Bearing ❷

Note: Click a **Header** to sort by that Header. For example, click **OD** to sort by **Outside-Diameter**

1. Click a **Part-Number** for the Roller



Cam-Follower Radius
& Roller Part- Number

IMPORTANT NOTES

A. The Radius of the sketch-element that represents the Cam-Follower Roller bearing changes to the **OD/2 of the bearing you select** (± **Radial Clearance** [329]).

B. If you edit the **Cam-Follower** (PART) in the **PART-EDITOR**, you see that the **Radius of the Roller** and the **Roller's Part-Number** are read-only - see image.

C. To edit the Radius of the Roller:

- Select a different **Part-Number** for the **Roller** bearing

or

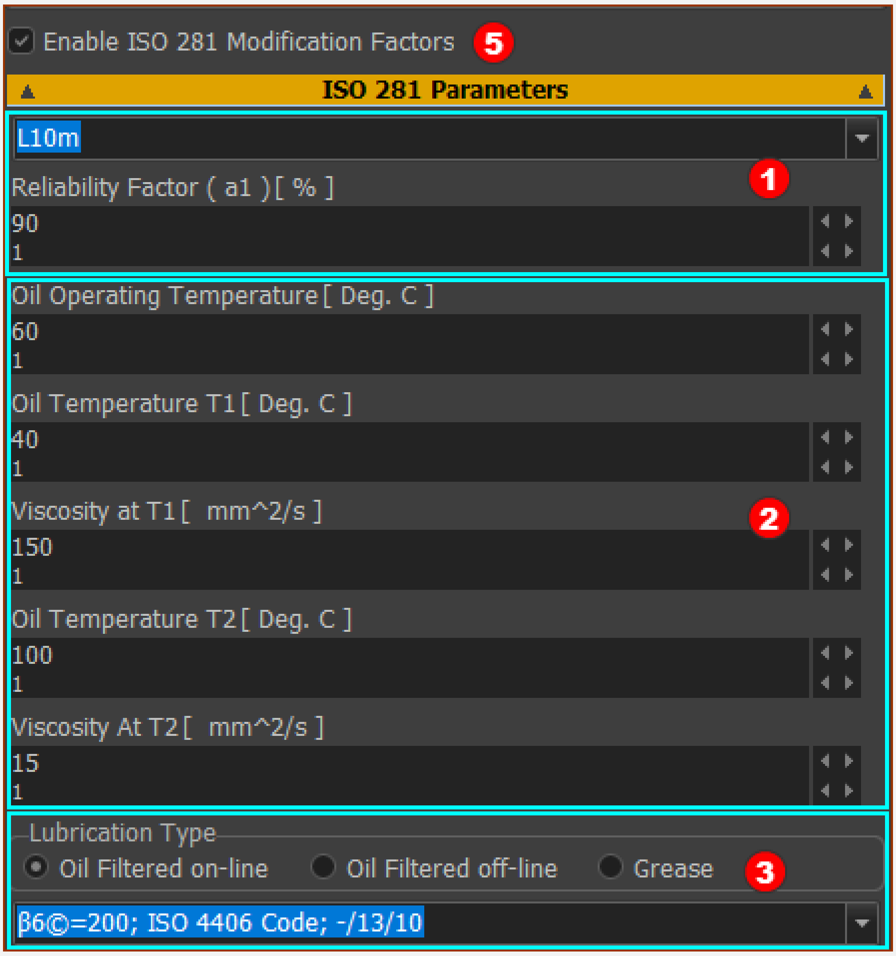
- Disable Show Roller and Cam Lifetimes check-box 331, and then edit its Radius dimension with the **PART-EDITOR**.

You can see immediately the **Basic Rating Life**, L_{10} in the **L10 FOR ROLLER: PART NUMBER** separator.

ISO 281 PARAMETERS

To see these factors, enable the ISO 281 Modification Factors 5

We calculate for you the a_1 and a_{iso} modification factors when as you enter the parameters below.



ISO 281 Modification Factors

1 Reliability Factor (a1)

There are two ways to specify the **Reliability Factor.**

- **Reliability drop-down**: Use the drop-down to select, from a standard list of reliability percentages.

  The drop-down list is the standard range of **Reliability Factors** (90 to 99.95%), as given in ISO 281.

or

- **Reliability Factor:** as a percentage.

  In this case, we find for you the nearest **Reliability Factor** to the **Reliability-Factors** in the drop-down list.

❷ Operating Viscosity

The viscosity of the Oil (or Base Oil in a Grease) is function of its Viscosity Grade, Viscosity Index, and the Operating Temperature.

To calculate these, you must enter parameters.

- Operating Temperature, ℃
- Temperature,
- Temperature T_1, and Viscosity ν_1 at Temperature T_1
- Temperature T_2, and Viscosity ν_2 at temperature T_2

Rules:

- $T_1 < T_2$ and $\nu_1 > \nu_2$

Note:

The ISO VG of an oil equates to $T_1 = 40℃$ and $\nu_1 = ISO\ VG$

Lubricating Oil and Grease datasheets usually provide the Viscosities at $40℃$ and $100℃$.

❸ Lubrication Type and Contamination Factors

The contamination factors can be determined for the following lubrication methods:

- **OIL FILTERED ON-LINE**: Circulating oil lubrication with the oil filtered on-line before being supplied to the bearings
- **OIL FILTERED OFF-LINE**: Oil bath lubrication, or circulating oil lubrication with off-line filters (or without filtration)
- **GREASE**.

Oil-Mist Lubrication is not considered.

OIL FILTERED ON-LINE

Select, in the drop-down list box, the **Filtration Ratio**, $\beta_{x(c)}$.

The operating, or actual, Filtration-Ratio should be as high, and if not higher, than the Filtration-Ratio you select.

- x - the contamination particle size, in µm, with ISO 11171 calibration.
- $\beta_{x(c)}$ - filtration ratio at contamination particle size x

The designation (c) signifies that the particle counters — of particles of size $x\ \mu m$ — shall be APC (automatic optical single-particle counter) calibrated in accordance with ISO 11171.

Also, the oil system shall have cleanliness within the range indicated by the cleanliness code according to ISO 4406 - for example, **17/14**

Filtration Ratio and Filtration Code
$\beta_{6(c)} = 200$ **; ISO 4406 Code; -/13/10**
$\beta_{12(c)} = 200$ **; ISO 4406 Code; -/15/12**

Filtration Ratio and Filtration Code
$\beta_{25(c)} = 75$; **ISO 4406 Code; -/17/14**
$\beta_{40(c)} = 75$; **ISO 4406 Code; -/19/16**

OIL FILTERED OFF-LINE

Select, in the drop-down list box, the **Cleanliness Codes, according to ISO 4406**, that best represents the anticipated operating condition.

Filtration Code
ISO 4406 Code; -/13/10
ISO 4406 Code; -/15/12
ISO 4406 Code; -/17/14
ISO 4406 Code; -/19/16
ISO 4406 Code; -/21/18

GREASE

Select, in the drop-down list box, the **Level of Contamination** that best represents the operating conditions.

Level of contamination	Operating Conditions
High cleanliness	Clean assembly with careful flushing; good sealing in relation to operating conditions; re-greasing carried out continuously or at short intervals Bearings, greased for life with effective sealing capacity in relation to operating conditions - for example, sealed bearings
Normal cleanliness	Clean assembly with flushing; good sealing in relation to operating conditions; re-greasing according to manufacturer's specification Bearings, greased for life with proper sealing capacity in relation to the operating conditions - for example, shielded bearings
Slight to typical contamination	Clean assembly; moderate sealing capacity in relation to operating conditions; re-greasing according to manufacturer's specifications
Severe contamination	Assembly in workshop; bearing and application not adequately washed after mounting; poor sealing capacity in relation to operating conditions; re-greasing intervals longer than recommended by manufacturer
Very severe contamination	Assembly in contaminated environment; inadequate sealing; long re-greasing intervals

ROLLER LIFE : BEARING P/N

These parameters are calculated automatically when you select a Roller, and/or enable ISO 281 factors.

Lifetime for Roller : SKF KR 19		
Result	Valid Data	Units
Equiv Load	258.29763	[N]
Roller rev Count	4.0248855	#
L10m N	5765.3793	[Rev x 1million]
L10m Hrs	91822.636	[Hour]
L10m Yrs	10.482036	[Year]

Life for the seleced Roller(Cam-Follower Bearing)

Basic Rating Life, L_{10}

This states that if the bearing load, P, is equal to the **Basic Dynamic Load Rating**, C, then there is a 90% reliability that the bearing survives 1 million rotations, if manufactured with commonly used high quality material, of good manufacturing quality, and operating under conventional operating conditions.

Basic Rating Life

$$L_{10} = \left(\frac{C}{P}\right)^{p}$$

Operation in Hours

If the cam speed is constant, it is often preferable to calculate the life in the number of operating hours.

Basic Rating Life (Hours)

$$L_{10}\,hrs = \frac{10^6}{60.n}.L_{10}$$

Operation in Years

Also, it is often preferable to calculate the life in operating years. This is based on 8760 hours (24hours, 365 days) a year.

Basic Rating Life (Years)

$$L_{10}\,years = \frac{L_{10h}}{8760}$$

L_{10}	Basic Load Rating (at 90% reliability) (millions of revolutions)
$L_{10}\,hrs$	Basic Load Rating (at 90% reliability) (operating hours)
$L_{10\,yrs}$	Basic Load Rating (at 90% reliability) (operating years)
C	Basic dynamic load rating (kN) of the Roller bearing
P	Equivalent dynamic bearing load (kN) (See image above : **Equiv Load**) The load continuously changes as the Cam rotates. Thus, we calculate for you an equivalent load.
n	Roller rotating speed (RPM) The speed continuously changes as the Cam rotates. Thus, we calculate for you an equivalent speed.

p	Exponent of the life equation = 3 for ball bearings =10/3 for roller bearings

Modified Rating Life, L_{nm}

Two factors must be calculated:

- a_1 - a factor for Reliability - see below 343
- a_{iso} - a factor calculated with a System Approach - see below 344

Modified Rating Life - as per ISO 281

$$L_{nm} = a_1.a_{iso}.L_{10}$$

see above for L_{10}

see below for an explanation of the a_1 and a_{iso} factors.

Operation in Hours

If the cam speed is constant, it is often preferable to calculate the life in the number of operating hours.

Modified Rating Life (Hours)

$$L_{nm}hrs = \frac{10^6}{60.n}L_{nm}$$

Operation in Years

Also, it is often preferable to calculate the life in operating years. This is based on 8760 hours (24hours, 365 days) a year.

Modified Rating Life (Years)

$$L_{nm}\ years = \frac{L_{nmh}}{8760}$$

L_{10}	Basic Life Rating (at 90% reliability) (millions of revolutions)
L_{nm}	Modified Life Rating (at $100 - n$% reliability) (millions of revolutions)
$L_{nm}\ hr$	Modified Life Rating (at $100 - n$% reliability) (operating hours)
$L_{nm}\ yr$	Basic Load Rating (at 90% reliability (operating years))
a_1	Life modification factor for Reliability - **see below** 343
a_{iso}	Life modification factor for Systems Approach - **see below** 344
C	Basic dynamic load rating (kN) of the Roller bearing
P	Equivalent Dynamic bearing load (kN) The load usually continuously changes as the Cam rotates. Thus, we calculate for you an equivalent load.
n	Roller rotating speed The speed usually continuously changes as the Cam rotates. Thus, we calculate for you an equivalent load.

p	Exponent of the life equation = 3 for ball bearings =10/3 for roller bearings

Operating Life

The operating life is defined as the life actually achieved by the bearing. It may differ significantly from the calculated life.

The operating life cannot be calculated!

Due to the wide variety of possible installation and operating conditions, it is not possible to precisely predetermine the operating life. The most reliable way of arriving at a close estimate is by comparison with similar applications.

Possible factors influencing the operating life

Wear or fatigue as a result of:

- deviating operating data
- misalignments between the shaft and housing
- insufficient or excessive operating clearance
- contamination
- insufficient lubrication
- excessive operating temperature
- oscillating bearing movement with small swivel angles (false brinelling)
- high vibration and false brinelling
- high shock loads (static overloading)
- prior damage during mounting

More information regarding the Modification Factors

Life Modification Factors for Reliability, a_1

The **Reliability Factor** is constant for all application conditions.

The drop-down list has the standard Reliability Factor percentages (90 to 99.95%), as given in ISO 281.

Modified Life Rating (at $100 - n\%$ reliability) (millions of revolutions)

Reliability	Failure Probability	L_{nm}	a_1 Factor
%	%	millions rotations	-
90	10	L_{10m}	1
95	5	L_{5m}	0,64
96	4	L_{4m}	0,55
97	3	L_{3m}	0,47
98	2	L_{2m}	0,37
99	1	L_{1m}	0,25
99,2	0.8	$L_{0.8m}$	0,22
99,4	0.6	$L_{0.6m}$	0,19
99,6	0.4	$L_{0.4m}$	0,16

Reliability	Failure Probability	L_{nm}	a_1 Factor
%	%	millions rotations	-
99,8	0.2	$L_{0.2m}$	0,12
99,9	0.1	$L_{0.1m}$	0,093
99,92	0.08	$L_{0.08m}$	0,087
99,94	0.06	$L_{0.06m}$	0,080
99,95	0.05	$L_{0.05m}$	0,077

Life Modification Factor for System Approach, a_{iso}

The **Life Modification factor**, a_{iso}, is a complex interaction between Oil or Grease Viscosity Grade, Filtration, Contamination, Oil Operating Temperature, the Fatigue load capacity of the Roller, the rotational-speed of the Roller, and the diameter of the Roller.

The equations given in ISO 281 to calculate these factors are empirical, complex, and interrelated. All of the factors, except C_u, are a function of the bearing speed and bearing load. Because, in a cam mechanism, the speed and load on the roller continually change, we calculate for you the factors at each step and integrate them to find their equivalent values.

$$a_{iso} = f\left(\frac{e_c.C_u}{P}, \kappa\right)$$

a_{iso}	Life modification factor, using a systems approach
C_u	Fatigue Limit of Bearing (N)
P	Dynamic Load (N)
κ	Viscosity Ratio (-)
e_C	Contamination Factor (-)

Fatigue Limit of Bearing. C_u (N)

If the Fatigue Limit of the Roller bearing is not in the Catalogue, we calculate its value for you.

In analogy to the static load rating in ISO 76, C_u is defined as the load at which the fatigue stress limit, σ_u, is just reached in the most heavily loaded raceway contact. For bearing steels, this is typically 1500MPa. The ratio σ_u/σ can be approximated by the ratio C_u/P

Contamination Factor, e_C

To calculate the Contamination Factor, e_C , you must select the Lubrication-Type and its Contamination Level. The Contamination Level is also a function of the Viscosity Ratio - see below.

Lubrication Type

You can choose from three Lubrication-Types:

- Circulating oil with the oil filtered in-line before being supplied to the bearings.
- Oil bath lubrication or circulating oil lubrication with off-line filters.
- Grease lubrication.

Oil Filtration

There are three types of filtration

- In-Line Filtration
- Off-Line Filtration

Grease Contamination Level

For grease, the level of contamination is a function of the

- assembly conditions for the machine/mechanism, the
- operating conditions, and the
- protection from dirt ingress into the Grease.

Grease Contamination

Each Lubrication-Type has a list of Contamination Levels from which you can choose.

Level of contamination	Operating conditions
High cleanliness	Very clean assembly with careful flushing; very good sealing in relation to operating conditions; re-greasing carried out continuously or at short intervals Sealed bearings, greased for life with effective sealing capacity in relation to operating conditions - for example sealed bearings.
Normal cleanliness	Clean assembly with flushing; good sealing in relation to operating conditions; re-greasing according to manufacturer's specification Sealed bearings, greased for life with proper sealing capacity in relation to operating conditions - for example, shielded bearings
Slight to typical contamination	Clean assembly; moderate sealing capacity in relation to operating conditions; re-greasing according to manufacturer's specifications
Severe contamination	Assembly in workshop; bearing and application not adequately washed after mounting; poor sealing capacity in relation to operating conditions; re-greasing intervals longer than recommended by manufacturer
Very severe contamination	Assembly in contaminated environment; inadequate sealing; long re-greasing intervals

Contamination Levels ISO 4406

The table below provides scale values as a function of particle concentration (particles/ml) - it is from ISO 4406

A three number code defines the amount of contamination at 4, 6, and 14 µm. Each time a scale number increases the quantity of particles is doubled.

Example: **ISO code** = 21 / 19 / 17

This Contamination Class describes a fluid containing:

- between 10 000 and 20 000 particles ≥ 4 µm(c) per 1 ml
- between 2 500 and 5 000 particles ≥ 6 µm(c) per 1 ml
- between 640 and 1 300 particles ≥ 14 µm(c) per 1 ml

If the leading number is missing, then that size of particle is not counted.

Scale Number	More than	Up to
28	1300000	2500000
27	640000	1300000
26	320000	640000
25	160000	320000
24	80000	160000
23	40000	80000
22	20000	40000
21	10000	20000
20	5000	10000
19	2500	5000
18	1300	2500
17	640	1300
16	320	640
15	160	320
14	80	160
13	40	80
12	20	40
11	10	20
10	4	10
9	2.5	4
8	1.3	2.5
7	0.64	1.3
6	0.32	0.64
5	0.16	0.32
4	0.08	0.16
3	0.04	0.08
2	0.02	0.04
1	0.01	0.02
0	0	0.01

Viscosity-Ratio, κ

The viscosity ratio, κ, is an indication of the quality of lubricant film formation.

The effectiveness of the lubricant is primarily determined by the degree of surface separation between the rolling contact surfaces. If an adequate lubricant separation film is to be formed, the lubricant must have a given minimum viscosity when the application has reached its operating temperature. The condition of the lubricant separation is described by the viscosity ratio, κ, as the ratio of the actual kinematic viscosity, ν, to the reference kinematic viscosity,ν_1. The kinematic viscosity, ν, is considered when the lubricant is at operating temperature.

$$\kappa = {}^{\nu}/_{\nu_1}$$

κ	Viscosity-Ratio

ν	Kinematic viscosity of the lubricant at operating temperature (mm^2/s)
ν_1	Reference Kinematic Viscosity of the lubricant (mm2/s)
λ	Film Thickness Ratio - the ratio of the actual film thickness to the composite roughness of the rolling elements and raceway surfaces.

ν_1 - **Reference Kinematic Viscosity.**

ν_1 is the viscosity that should separate the rolling elements from the raceways, to give a **Film Thickness Ratio,** $\lambda = 1 \rightarrow 1.5$.

It assumes that the oil is a mineral oil, with a Viscosity-Index of approximately 100. Synthetic Hydrocarbon SHC type synthetic oils can be used.

ν - **Viscosity at Operating Temperature**

We calculate for you the viscosity at the Operating Temperature from the viscosity, ν , at two other temperatures.

The Viscosities at 40℃ (ν_{40}) and 100℃ (ν_{100}) are usually specified on the data sheet of the lubricant.

If you have manufactured the internal and/or external raceways, then it is better to calculate the **Viscosity Ratio** from:

$$\kappa = \lambda^{1.3}$$

$$\lambda = \frac{h_{min}}{\left(R_{q1}^2 + R_{q2}^2\right)^{0.5}}$$

λ	Film Thickness Ratio - the ratio of the actual film thickness to the composite roughness of the rolling elements and raceway surfaces.
h_{min}	Minimum film thickness
R_q	RMS Roughness of the Rolling-Elements, or Rolling-Raceways.

Notes:

A **Viscosity-Ratio** less than 0.1 is outside of the limits of **ISO 281**. It is near to metal-to-metal contact.

A **Viscosity-Ratio** greater than 4 is the maximum that can be used by **ISO 281**. However, **ISO 281** simply states that the **Viscosity-Ratio** is equal to 4 if it is actually greater than 4. A viscosity-ratio greater than 4 is getting too high for the bearings. The needles or balls may slide and refuse to roll in the 'thick-oil', or the oil may churn and increase the oil and bearing temperatures.

A viscosity of approximately 2-3 is approximately ideal.

1.10.20 Cam Life tab

2D-Cam dialog: Cam Life tab

To **Calculate Cam Life**, you must :

Parameters tab

STEP 1: Enable Show Roller and Cam Life

STEP 2: Enter a safety factor (CAM)

See Parameters tab > [331] ENABLE LIFETIMES, EDIT SAFETY-FACTOR [329]

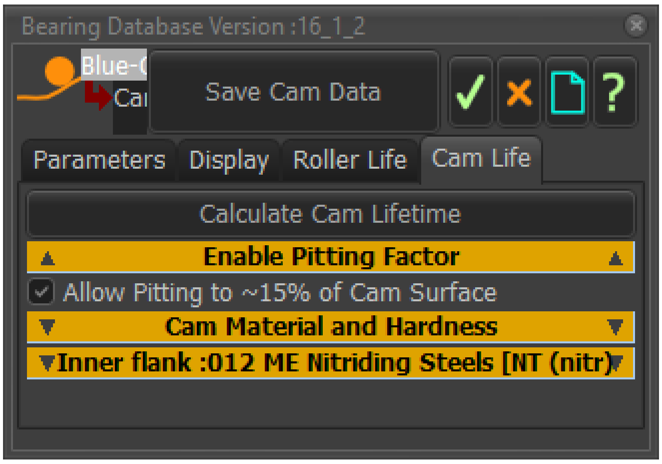
Cam Life tab

STEP 3: Enable / Disable “Allow Pitting to ~15% of the Cam Surface”

STEP 4: Select a Steel Category, Heat-Treatment, and Steel Quality for the Cam

STEP 5: Enter the Steel's Hardness as HV, HB, or HRC, within the Minimum and Maximum Hardness Limits

Review the Lifetime Results



2D-Cam dialog > Cam Life tab

Cam Life: tab

Enable Pitting Factor

Allow Pitting to ~15% of Cam Surface

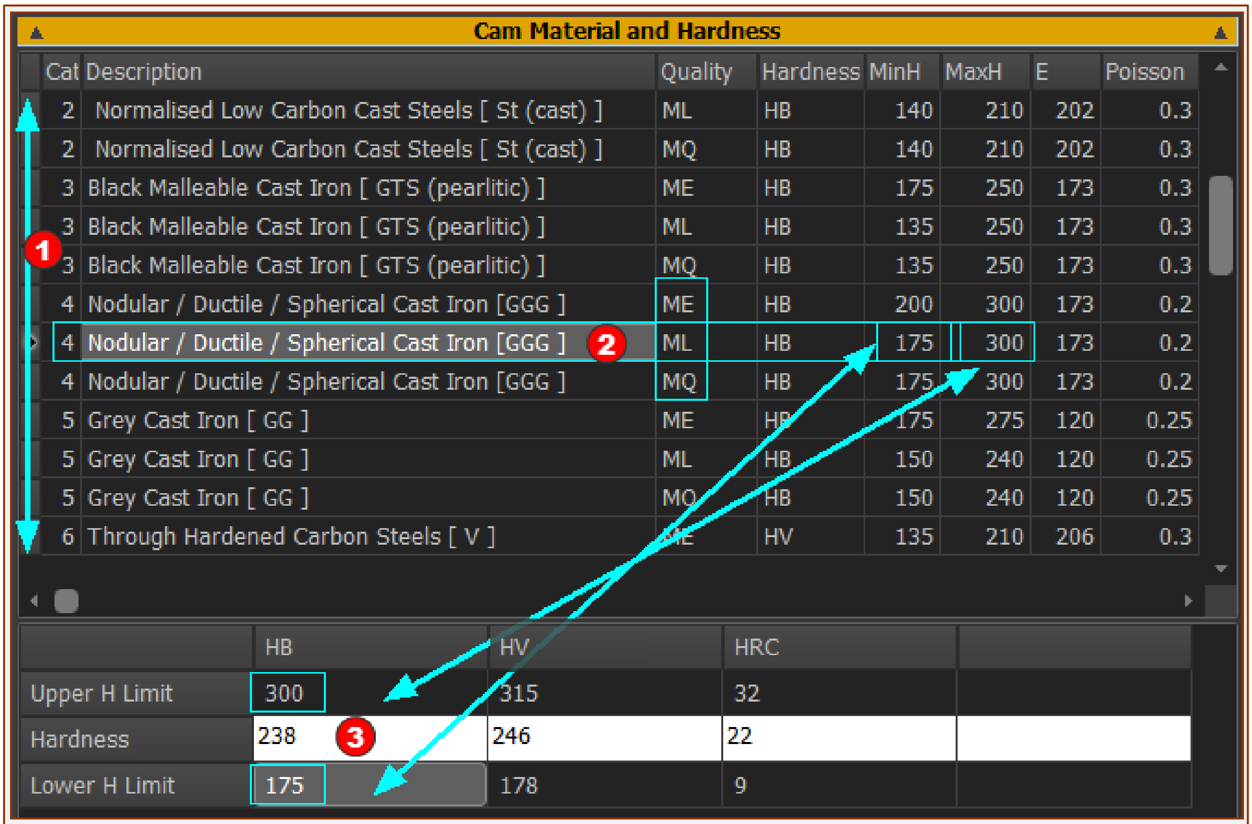
Steels and Cast-Irons of Categories 1, 2, 3, 4, 6, 8, 9, 10, and 11 (See Cam Life Results below)

These categories allow a small amount of surface pitting to occur, at which time the pitting stabilizes. This is called non-progressive pitting.

If you allow up to ~15% pitting, the cam-life is extended.

In **mission-critical** applications, for example **manned flight** - you would **not** allow pitting, of course.

CAM MATERIAL AND HARDNESS



Cat	Description	Quality	Hardness	MinH	MaxH	E	Poisson
2	Normalised Low Carbon Cast Steels [St (cast)]	ML	HB	140	210	202	0.3
2	Normalised Low Carbon Cast Steels [St (cast)]	MQ	HB	140	210	202	0.3
3	Black Malleable Cast Iron [GTS (pearlitic)]	ME	HB	175	250	173	0.3
3	Black Malleable Cast Iron [GTS (pearlitic)]	ML	HB	135	250	173	0.3
3	Black Malleable Cast Iron [GTS (pearlitic)]	MQ	HB	135	250	173	0.3
4	Nodular / Ductile / Spherical Cast Iron [GGG]	ME	HB	200	300	173	0.2
4	Nodular / Ductile / Spherical Cast Iron [GGG]	ML	HB	175	300	173	0.2
4	Nodular / Ductile / Spherical Cast Iron [GGG]	MQ	HB	175	300	173	0.2
5	Grey Cast Iron [GG]	ME	HB	175	275	120	0.25
5	Grey Cast Iron [GG]	ML	HB	150	240	120	0.25
5	Grey Cast Iron [GG]	MQ	HB	150	240	120	0.25
6	Through Hardened Carbon Steels [V]	ME	HV	135	210	206	0.3

	HB	HV	HRC
Upper H Limit	300	315	32
Hardness	238	246	22
Lower H Limit	175	178	9

Enter the Hardness as HB, HV, or HRC

Select a Steel Category, Heat-Treatment, and Quality

❶ There are 14 Steel/Cast-Iron and Heat-Treatment categories, each with 3 Quality levels (ML,MQ,ME) - identical to the ISO 6336-5 Standard for Gears.

STEELS CATEGORIES AND HEAT-TREATMENT

1. Normalized, Low-Carbon Wrought-Steels[a] - St
2. Normalized, Low-Carbon, Cast-Steels[a] - St(cast)
3. Black Malleable Cast-Iron - GTS (perl)
4. **Nodular Spheroidal Cast-Iron - GGG (❷ - see image above)**
5. Grey Cast-Iron - GG
6. Through-Hardened Wrought Carbon-Steel[b] - V
7. Through-Hardened Wrought Alloy-Steels[b] - V
8. Through-Hardened Cast Carbon-Steel - V(cast)
9. Through-Hardened Cast Alloy-Steel - V(cast)
10. Case-Hardened Wrought-Steels[c] - Eh
11. Flame or Induction-Hardened Wrought or Cast-Steels (IF)
12. Nitrided Nitriding Wrought-Steels[d] - NT(nitr)
13. Through-Hardened Nitrided-Steel[b] - NV(nitr)
14. Through-Hardened Nitro-Carburized Wrought-Steel[e] - (NV (nitr-car)

a - ISO 4948-2 (ISO10020) Part 2 - Classification of unalloyed and alloy steels according to main quality classes and main property or application

b - ISO 683-1 - Heat-treated Steels, alloy steels and free-cutting steel Part 1 - Quenched & Tempered unalloyed steels

c - ISO 683-11 - Heat-treated Steels, alloy steels and free-cutting steel Part 11 - Wrought Case Hardening steels

d - ISO 683-10 - Heat-treated Steels, alloy steels and free-cutting steel Part 1 - Quenched & tempered unalloyed steels

e - **ISO 683-1**, **ISO683-10**, or **ISO 683-11**

STEEL QUALITY❷

The **Steel Quality Standards** are:

ML - Modest demands on the material quality and the heat treatment process

MQ - Material Quality and Heat-Treatment standards met by experienced manufacturer.

ME - High degree of reliability of Material Quality and Heat-Treatment process.

The requirements to be met for different steel qualities include: Chemical Analysis, Melting Practice, Surface Crack Detection after machining, Hardness must be measured. Sometimes with a test piece of steel that is manufactured, and heat-treated together with cam - See ISO 3663-5.

Enter the Steel's Hardness within the High and Low Hardness Limits

❸ Enter a Hardness value in the HB, HV, or HRC hardness scale - see image above

The default Hardness is the average of the Upper and Lower Hardness Limits

The **Hardness** you enter must be within the **Upper Hardness Limit** and **Lower Hardness Limit** (These limits are specified in ISO 6336-5).

Lower H Limit ≤ Actual Hardness (HB, HV, HRC) ≤ Upper H Limit

In the example above, you can enter a Hardness in the range of **175-300HB**, **178-315HV**, or **9-32HRC**.

When you enter a value with one of the **Hardness Scales**, the other two **Hardness Scales** update automatically.

Cam-Life Results

INNER / OUTER FLANK + STEEL CATEGORY # , QUALITY, AND CATEGORY NAME

Cam Life - Inner flank : 010 ME Case Hardened Steels [Eh]

Quantity	Valid Data	Units
Contact Stress (Max.)	862.11288	[MPa]
Dynamic Contact Stress(Max.)	862.11288	[MPa]
Allowable Contact Stress(Max.)	1650	[MPa]
Dynamic / Allowable (Max. Contact Stress)	0.52249265	Ratio
Depth at Max. Stress	0.10278249	[mm]
Shear Stress(Max.)	268.36295	[MPa]
Dynamic Shear Stress(Max.)	268.36295	[MPa]
CamLife N	10000	[10^6 Rotations]
CamLife Hrs	641025.64	[Hrs]
CamLife Yrs	73.176443	[Yrs]

The Orange Separator indicates the Flank, Steel Category, Steel Quality, Heat-Treatment Type and/or Steel-Type with Steel Acronym are indicated in the separator.

E.G: INNER FLANK: 004 ML NODULAR / DUCTILE / SPHERICAL CAST IRON (GGG)

Flank : Inner

Steel Category : 004

Steel Quality : ML

Steel Name : Nodular / Ductile / Spherical Cast Iron

Steel Acronym : GGG

THE RESULTS ARE:

Maximum Contact Stress (MPa - Mega-Pascals)

The maximum contact-stress that we calculate for you from the Geometry, Materials, and Contact-Force, for both cylindrical and barrel shaped Cam-Followers.

Maximum Dynamic Contact-Stress - (MPa - Mega-Pascals)

Maximum Contact-Stress × Safety Factor - see Parameters tab > Enable Lifetime, Edit Safety-Factor 329

Maximum Allowable Contact Stress

Function of selected **Steel-Category, Heat-Treatment, Steel Quality,** and its **Surface Hardness**

Dynamic /Allowable

Ratio of Maximum Dynamic Contact-Stress to the Maximum Allowable Contact Stress

This ratio gives the value on the Y-axis on the plot of Life vs Y (Dynamic/Allowable) - see image below.

From that, we derive for you the life of the Cam.

Depth of Max Shear Stress

The maximum shear-stress is below the surface. Its depth is a function of the size and shape of the contact zone.

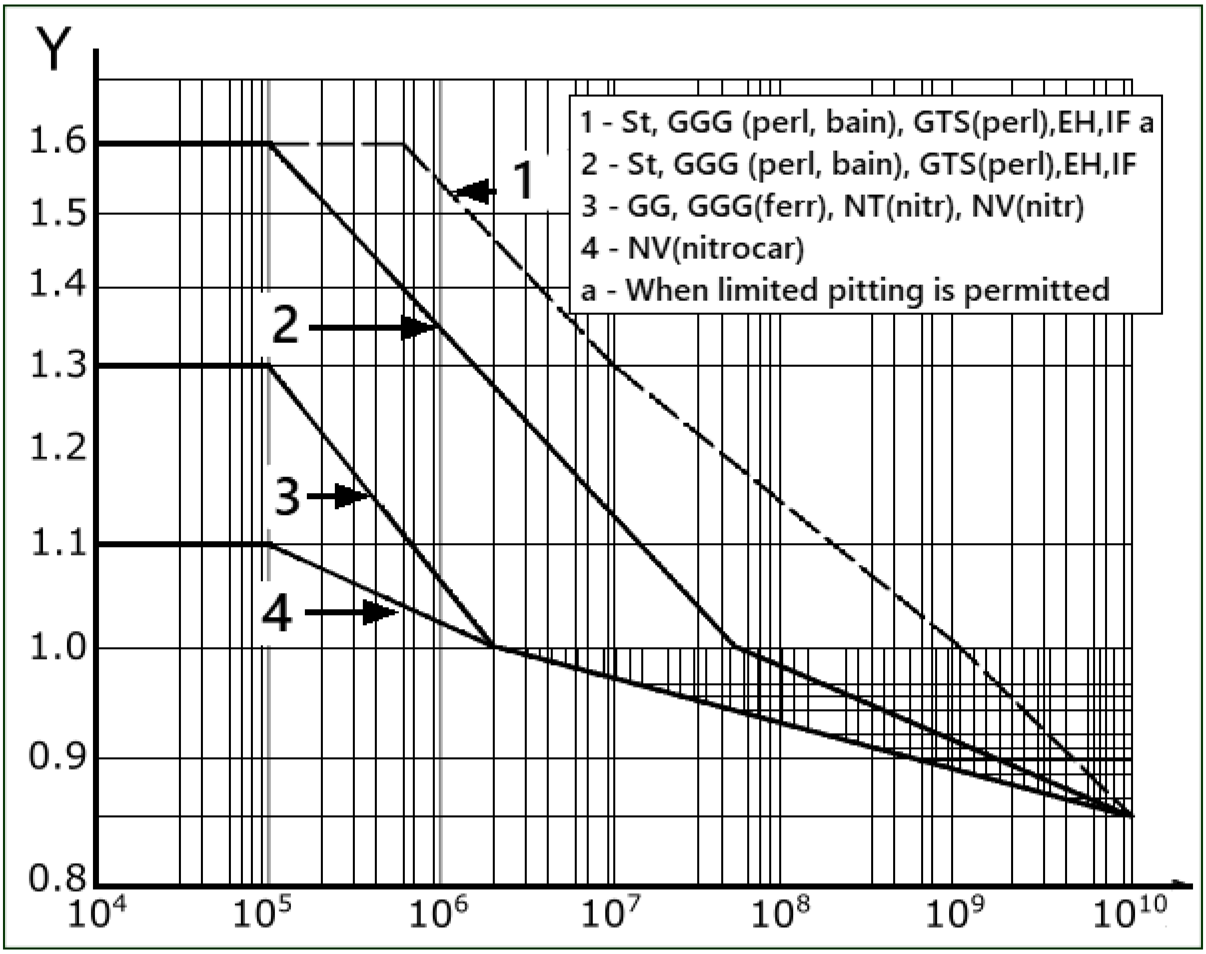
The minimum depth-of-hardness for any surface hardening should be approximately 2 x this value.

Maximum Shear Stress

Approximately 0.78 x Maximum Contact-Stress (depending on the contact geometry.

Dynamic Shear-Stress

Maximum Shear-Stress x Safety-Factor - see Parameters tab > Enable Lifetime, Edit Safety-Factor 329



S-N Curves for different Steel Categories (based on ISO 6336)

Cam Lifetime:

Y (in graph below) = Maximum Dynamic Contact Stress / Allowable Contact Stress

When Y=1, Cam Lifetime = 1.

1 Life of Steels-Categories 1: 10^9 cycles,

1 Life of Steel-Categories 2:1.2^10^7cycles;

1 Life of Steel Categories 3 and 4: 5x10^6 cycles

When Y > 1, the Cam Lifetime is reduced.

When Y < 0.85, Cam-Lifetime is infinite (Greater than 10^10 cycles)

When Life < 10^5 cycles, Contact-Stress plays a more important role, also, ratcheting and shakedown stress.

Cam-Life N (Millions of Cam Rotations)

Millions of Cycles, 99% reliability.

Maximum - 10,000,000,000 (10^{10} rotations / 10 billion)

Cam-Life Hrs

Number of Hours, based on Millions of Cam-Life Rotations and Machine Speed (RPM).

Cam-Life Yrs

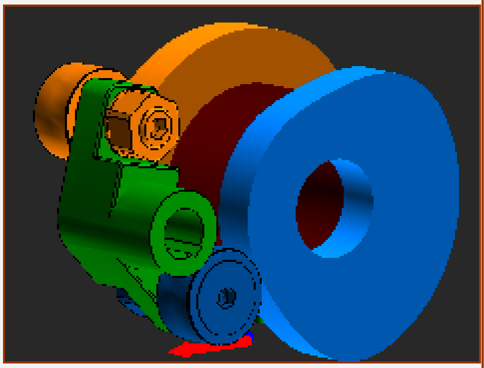
Number of Years, based on 8760 hours per year.

1.10.21 Dialog: Conjugate-Cam-FB

Conjugate-Cam FB

See: Add Conjugate Cam FB [139].

What are Conjugate Cams?



A Conjugate Cam pair

A **Conjugate Cam** is Body-Closed Cam with typically two Cams that rotate on one axis/shaft, which drive two **Cam-Follower Rollers** that oscillate or index on another shaft. E.g. Commercial **Parallel Cams** that oscillate or index a Cam-Follower.

A **Conjugate Cam** may have one **Cam-Follower Roller** held in a Groove or Track. This is a Conjugate-Cam this is called a **Groove Cam** or **Track-Cam**.

To do the **Force Analysis** of **Conjugate-Cams**, you need two **Cam-Follower Rollers** and to define the **Inner-Cam** and the **Outer Cam** separately, and then use the CAM-CONJUGATE FB.

In the image to the left, the blue and orange Cams rotate together on one shaft. The blue Cam-Profile is in continuous contact with the blue Cam-Follower Roller, and the Orange Cam-Profile is in continuous contact with the Orange Cam-Follower Roller. The Cam-Follower rotates one Part.

Why use a Conjugate Cam FB?

Typically, to do the force and stress analysis of a 2D-CAM, you select one 2D-CAM as the Power Source in the Configure Power-Source dialog [193].

However, if there are two Cams (**Conjugate Cams**) that drive the same **Cam-Follower**, you must add and configure a CONJUGATE CAM FB. Then, select the CONJUGATE-CAM FB as the **Power-Source** in the CONFIGURE POWER-SOURCE DIALOG - only then should you do the force and stress analysis.

You cannot use the Inner and Outer Flanks of one 2D-CAM when it is one Cam in a pair of **Conjugate-Cams**.

2D-Cam: Work-flow

1. Add a 2D-CAM - **see Machine elements toolbar > Add 2D-Cam** [109]

If the new 2D-CAM is one from a pair of **Conjugate-Cams**, or it is one flank of a **Groove-Cam**

1.a. Add a CONJUGATE-CAM FB - **see Machine elements toolbar > Add Conjugate Cam FB** [139]

1.b. Edit the CONJUGATE-CAM FB to select least two 2D-CAMS - **see Conjugate-Cam dialog** [353].

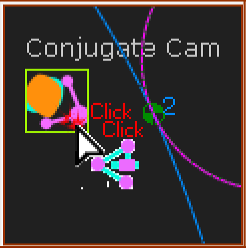
2. Select a 2D-CAM or a CONJUGATE-CAM FB as the Power Source for the Follower - **see Configure-Power Source** [480]

3. Review the 2D-CAM : **Display, Properties, Roller-Life, Cam-Life, ...** - **see 2D-Cam dialog** [327]

4. Add a CAM-DATA FB - **see Kinematic FBs > Add Cam-Data FB** [171]

5. Edit the CAM-DATA FB to link it to the 2D-CAM- - see Cam-Data dialog : Cam Analysis [357]
6. Connect wires from the CAM-DATA FB to a GRAPH FB to analyse 5 parameters - see Cam-Data dialog : Cam Analysis [357]
7. Edit the CAM-DATA FB again to calculate the Cam's Coordinates - see Cam-Data dialog : Cam-Coordinates [361]

How to open a Conjugate Cam dialog



To open the CONJUGATE-CAM DIALOG:

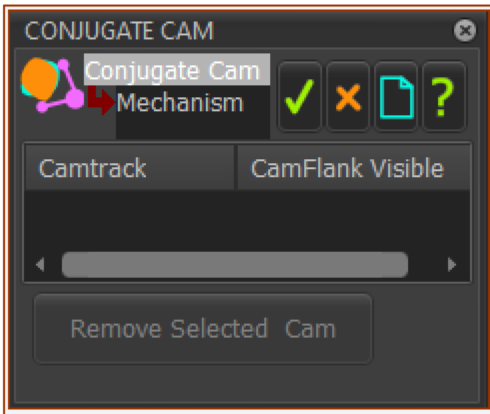
1. Double-click the CONJUGATE-CAM FB

OR

1. **See** How to Open a dialog [513]

The CONJUGATE-CAM DIALOG is now open.

Conjugate-Cam dialog



Conjugate-Cam dialog

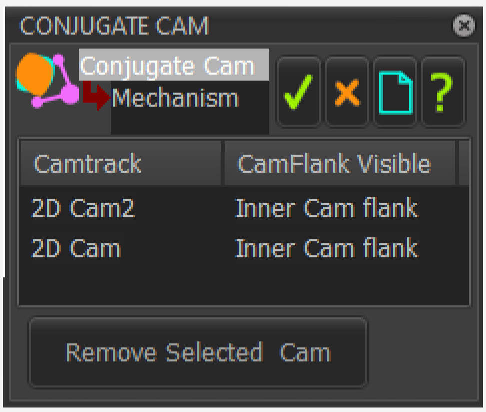
Note: if you want to analyse the forces of a Groove Cam - you MUST add 2 x 2D-Cams to the model. The Cam-Follower Rollers are above each other.

You cannot use the Inner and Outer Flanks of 1 x 2D-Cam.

Add 2 x 2D-Cams

In the graphic-area or ASSEMBLY-TREE:

1. Click a 2D-CAM
2. Click a different 2D-CAM



The two **2D-CAMS** must be capable of being the Power-Source to the same **Cam-Follower Part**

Each **2D-CAM** drives the **Inner-Cam-Flank** OR the **Outer Cam-Flank**.

3. **Click ✓ to close the dialog.**

Note 1 : If the **2D-CAMS** you select do **not** drive the same **FOLLOWER-PART**, you get this message in the Feedback Area. 270

Message

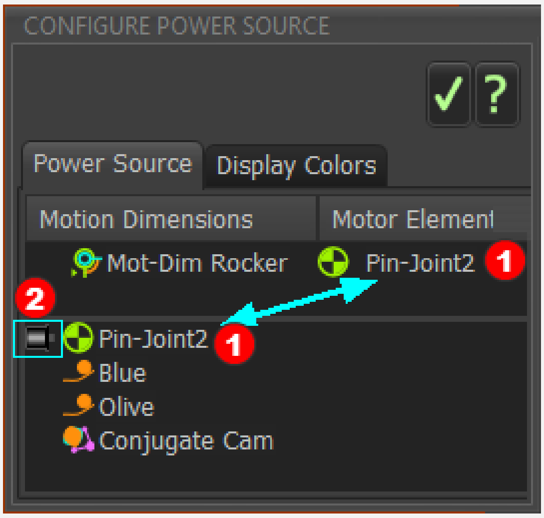
Count	Topic	Message
29	Conjugate Cam	Camtrack does not share follower with previous camtrack

To do **Force Analysis** of the **Conjugate Cams**, you must select the **CONJUGATE-CAM FB** in the **CONFIGURE POWER SOURCE DIALOG**.

To remove a 2D-Cam

1. Select the **2D-CAM** in the list
2. Click Remove Selected Cam

Select Conjugate Cam in the Configure Power Source dialog-box



DEFAULT: Configure Power Source dialog

Configure the Power Source

1. Click the **Kinematics-Tree** tab in the **ELEMENT-EXPLORER**
2. Click the **Kinematics-Chain** that includes the **Follower Part**
3. Right-click the **Kinematic-Chain**
4. Click **CONFIGURE POWER-SOURCE** to open the **CONFIGURE POWER-SOURCE DIALOG** - **see image above**

The **PIN-JOINT** ❶ is the **DEFAULT MOTOR ELEMENT** (**Power-Source**) - **see image above**

Notice also the **Motor Symbol**❷ to the left of the **PIN-JOINT** - **see image above**

In the **CONFIGURE POWER-SOURCE DIALOG** - **see image below**

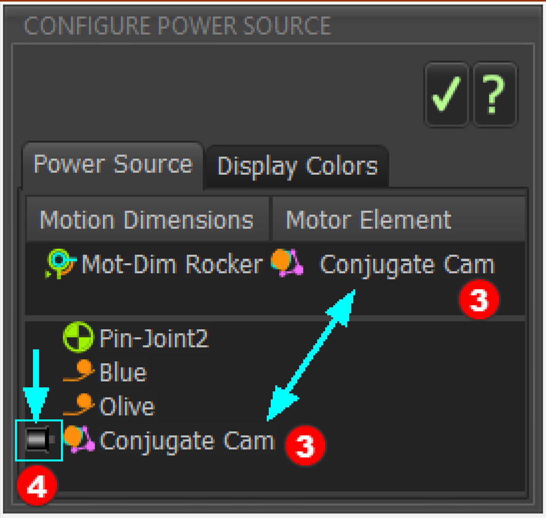
5. Click the **CONJUGATE CAM**❸ in the list of elements - **see image below**

The CONJUGATE-CAM❸ replaces the PIN-JOINT as the MOTOR ELEMENT❸ (Power-Source) - **see image below**

Notice the **Motor Symbol**❹ is now to the left of the CONJUGATE-CAM - **see image below**

6. **Click** ✅ to close the dialog.

Now you can do **Contact-Force Analysis** for the **Conjugate-Cam**.



Conjugate-Cam as the Power-Source

1.10.2 Dialog: Function-Block: Cam-Analysis

Cam-Data FB > Cam-Analysis

2D-Cam: Work-flow

1. Add a 2D-CAM - **see Machine elements toolbar > Add 2D-Cam**[109]

If the new 2D-CAM is one from a pair of **Conjugate-Cams**, or it is one flank of a **Groove-Cam**

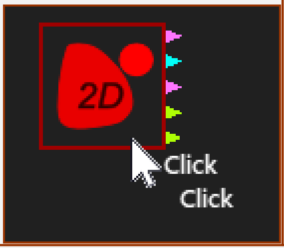
1.a. Add a CONJUGATE-CAM FB - **see Machine elements toolbar > Add Conjugate Cam FB**[139]

1.b. Edit the CONJUGATE-CAM FB to select least two 2D-CAMS - **see Conjugate-Cam dialog**[353].

2. Select a 2D-CAM or a CONJUGATE-CAM FB as the Power Source for the Follower - **see Configure-Power Source**[480]
3. Review the 2D-CAM : **Display, Properties, Roller-Life, Cam-Life, ...** - **see 2D-Cam dialog**[327]
4. Add a CAM-DATA FB - **see Kinematic FBs > Add Cam-Data FB**[171]
5. Edit the CAM-DATA FB to link it to the 2D-CAM- - **see Cam-Data dialog : Cam Analysis**[357]
6. Connect wires from the CAM-DATA FB to a GRAPH FB to analyse 5 parameters - **see Cam-Data dialog : Cam Analysis**[357]
7. Edit the CAM-DATA FB again to calculate the Cam's Coordinates - **see Cam-Data dialog : Cam-Coordinates**[361]

Link the Cam-Data FB to a 2D-Cam

To link a CAM-DATA FB to a 2D-CAM, you must open the CAM-DATA DIALOG:



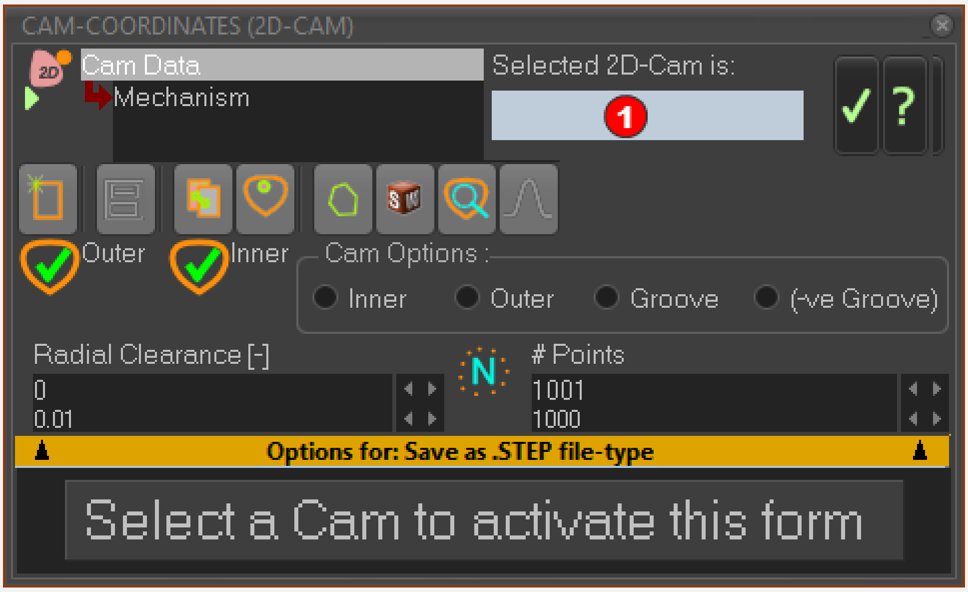
1. Double-click the CAM-DATA FB to open the CAM-DATA DIALOG

OR

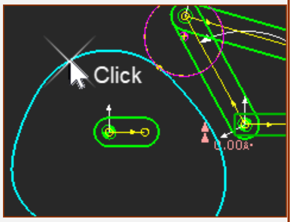
1. **See How to open a dialog**[513]

The CAM-DATA DIALOG is now open - **see below**

SELECTED 2D-CAM IS box 1: is Light Blue - a 2D-CAM has not been selected.



Cam-Data dialog - NOT linked to a 2D-Cam

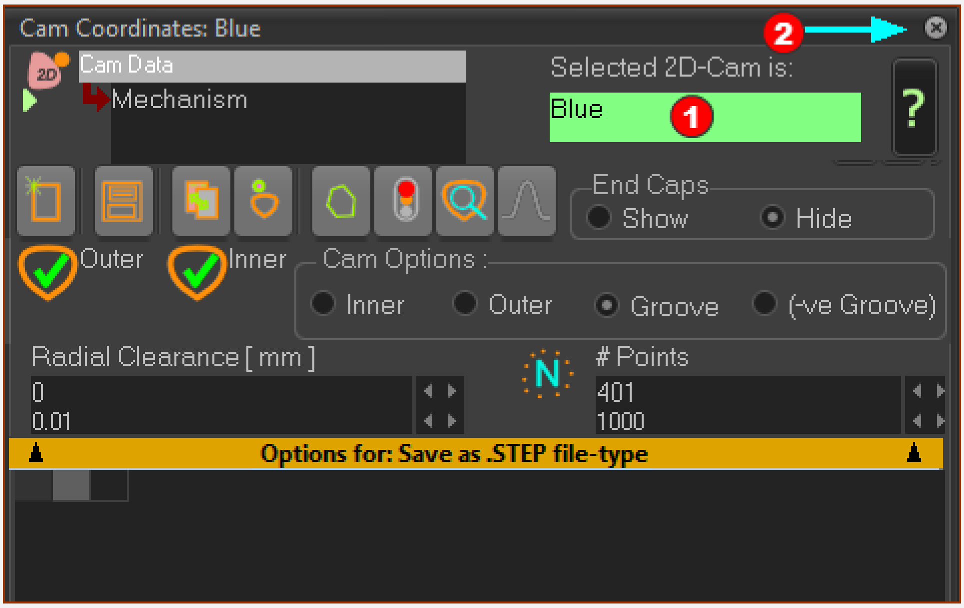


Click to select a 2D-Cam

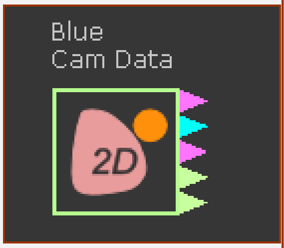
2. Click a **2D-CAM** in the graphic-area or the **ASSEMBLY-TREE**

The **2D-CAM** is now in the Selected 2D-Cam ❶ box - see image below.

3. Click ❷ (⊗) to close the **CAM-DATA DIALOG**



Cam-Data FB linked to 2D-Cam 'Cam Outer'



The caption above the **CAM-DATA FB** is the name of the element: **CAM-DATA FB.** - See rename

The caption above the name, is the name of the **2D-CAM** that has a link with the **CAM-DATA FB**.

TOP-TIP: When there is more than one **2D-CAM** in the **MECHANISM-EDITOR**:

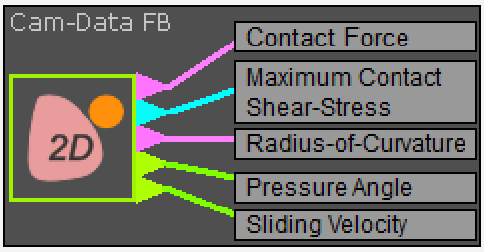
1. Edit the color of the **2D-CAM** - **see** **2D-Cam dialog : Display tab >** [329] **DISPLAY OPTIONS**
2. Rename the **2D-CAM** element to its color - **see** **Rename dialog** [279]

Now, it is easier to see which **2D-CAM** is linked to the **CAM-DATA FB**.

You can now :

- **Analyze 5 Cam parameters - see** **Cam-Analysis Parameters** [358].
- **Calculate the Cam's Coordinates - see** **Cam-Coordinates dialog** [361]

Cam-Data FB: Cam Analysis Parameters



Analysis Parameters from Cam-Data FB

The five output-connectors from the **CAM-DATA FB** provide:

- ***F* : *Contact Force*** : See Note 1 [359]
- ***Ʊ* : *Contact Maximum Shear Stress*** : See Note 2 [359]
- ***ρ* : *Radius-of-Curvature*** : See Note 3 [359]
- ***μ* : *Pressure Angle* :** See Note 4 [359]
- ***Ue* : *Sliding Velocity*** - see Note 5 [360]

Drag wires from the output-connectors of the **CAM-DATA FB** to a **GRAPH FB** [172], a **MATH FB** [396] or a **STATISTICS FB** [179]

Make sure you select the correct cam (**INNER, OUTER, PITCH-CENTER PATH**) in the Y–axis display option [174] of the **GRAPH FB** interface.

Top-Tip:

Connect the ***same*** output-connector and parameter (for example, Pressure-Angle) from ***different*** **CAM-DATA FBS** to the ***same*** **GRAPH FB**. Then you can analyse the same parameter for up to **4** × **2D-CAMS** in one graph plot.

Set the Minimum and Maximum Values to be the same for all graphs. Then it is even easier to compare the same parameter for each **2D-CAM**.

Note 1:

Make sure to do Configure the Power Source [480].

If **CONTACT-FORCE** is 0.0N in the **GRAPH FB**, and the **Configure Power Source** is correct, change the Cam from **INNER** to **OUTER**, or vice versa, in the **GRAPH FB > Y-AXIS OPTIONS**.

Note 2:

Make sure to do Configure the Power Source [480].

If **SHEAR-STRESS** is 0.0Nm^{-2} in the **GRAPH FB**, and the **Configure Power Source** is correct, change the Cam from **INNER** to **OUTER**, or vice versa, in the **GRAPH FB > Y-AXIS OPTIONS**.

The **Maximum Shear Stress** is not on the surface of the Cam - it is a little below its surface. The **Maximum Shear Stress** is a function of the material properties of the contacting bodies

Material Properties:

- **Cam-Follower**: $E = 210GPa, \nu = 0.3$
- **Cam**: taken from the Steel Selection in the Cam-Life tab of the 2D-Cam dialog [347].

There calculations have assumptions. The most significant assumption is that friction is zero, and the bodies are perfectly smooth.

Note 3:

By convention, the **RADIUS-OF-CURVATURE (ROC)** of a Roller is positive.

When the cam is convex relative to the Roller, the **ROC** of the cam is positive.

When the cam is concave relative to the Roller, the **ROC** of the cam is negative.

RADIUS-OF-CURVATURE of the **Pitch-Curve** is not available. If you need to know the **RADIUS-OF-CURVATURE** of the **PITCH-CURVE**, export the **RADIUS-OF-CURVATURE** of the **INNER** and **OUTER** cams to Excel, and calculate their average.

Note 4:

The PRESSURE ANGLE is through the center of the Cam-Follower. There are two other PRESSURE ANGLES available: the Contact Pressure Angle of the INNER and OUTER Cams - these are labeled as INSIDE CONT. PR, ANG. in the Graph FB > Y-axis display options.

These should be used with a Flat-Faced Follower only.

We cannot calculate for you the PRESSURE ANGLE for a Stationary Cam.

Note 5:

Use **Sliding-Velocity** to calculate the Film Thickness of the lubricating oil between the Cam-Follower and Cam. If you know the surface finish of the cam and cam-follower, then you can also calculate the **Film Thickness Ratio**, λ. This is an important parameter to calculate as it strongly influences the life of the cam. A **Film Thickness Ratio** of less than 1 means that metal asperities of the Cam-Follower and Cam contact each other, and wear occurs.

Sliding Velocity = (Velocity of Cam Surface - Velocity of Cam-Follower Surface)

Entrainment Velocity = (Velocity of Cam Surface + Velocity of Cam-Follower Surface) ÷ 2

Slip-Slide-Ratio = Sliding-Velocity ÷ Entrainment Velocity

Contact Force / Contact Shear-Stress

PREPARATION:

If the cam is **Body-Closed** (for example. a Conjugate-Cam or a Groove-Cam) then you must add 2 × 2D-CAMS and a CONJUGATE CAM FB.

See more : Conjugate Cams FB [139]

You **must** do Configure Power Source [480] to calculate correctly the CONTACT-FORCE and CONTACT SHEAR-STRESS.

There are four methods to show the Contact-Force between the Cam and the Follower.

Display in the Graphic-Area:

1. **Forces toolbar > Display Force Vectors** [198] button.

   Two equal and opposite Force-Vectors radiate from the Contact-Point between the Cam and Follower. They are

   - the force that ACTS-ON the Follower and Follower-Part
   - the force vector that ACTS-ON the Cam and Cam-Part.

   You must analyze whether the force between the cam and the cam-follower is active or not active.

2. Use: 2D-Cam dialog > Display tab > Cam Display Options > Contact-Force [334].

Plot with a Graph-FB:

3. Add a Force-Data FB [191], edit it select the 2D-CAM as the FORCE ELEMENT, plot with a GRAPH FB.
4. Use the CAM-DATA FB to plot CONTACT FORCE from the top output-connector with a GRAPH FB.

   Contact-Force is **=** 0 N if Contact-Force **≤** 0 N.

   This is the preferred option if you need to know whether the Cam-Force becomes less than zero.

1.10.2.3 Dialog: Function-Block: Cam-Coordinates

Cam-Data FB > Cam-Coordinates

See Add Cam-Data FB 109

Use a CAM-DATA FB to:

- Calculate the Cam-Profile for the Cam's Inner, Outer, and Pitch-Line
- Calculate the Cam-Profile as XY Points or as BiArcs
- Export the Cam-Profile directly to SOLIDWORKS
- Save the Cam-Profile as STP (STEP), TXT, DXF, and SLDCRV file-types

If you save the Cam-Profile as a STP file, you can define for the Cam, the

- Plate Overall Diameter, Plate Thickness, Bore Hole Diameter, and if the Cam is a Groove-Cam, define the Depth of the Cam-Profile

2D-Cam: Work-flow

1. Add a 2D-CAM **- see Machine elements toolbar > Add 2D-Cam** 109

If the new 2D-CAM is one from a pair of **Conjugate-Cams**, or it is one flank of a **Groove-Cam**

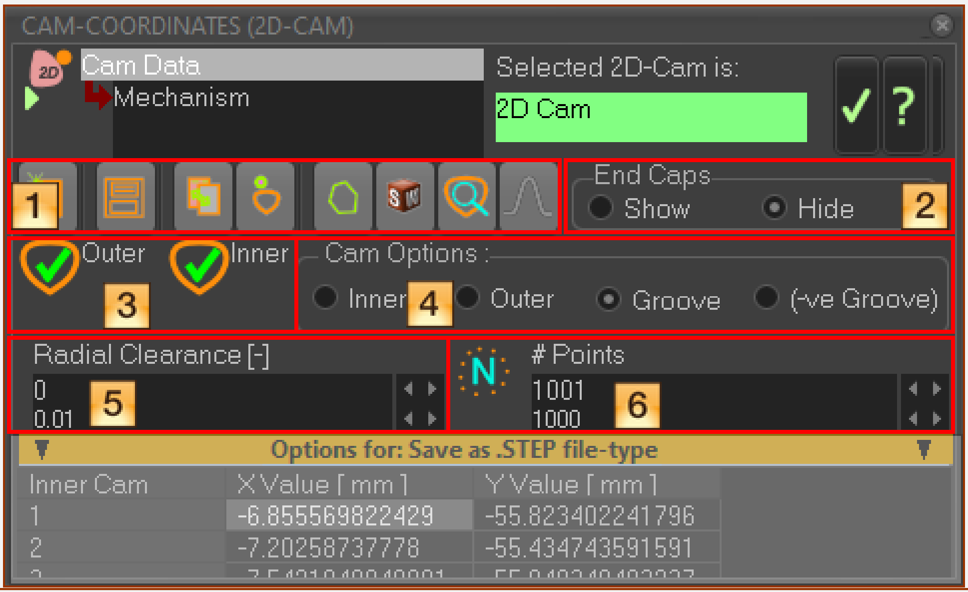
1.a. Add a CONJUGATE-CAM FB **- see Machine elements toolbar > Add Conjugate Cam FB** 139

1.b. Edit the CONJUGATE-CAM FB to select least two 2D-CAMS **- see Conjugate-Cam dialog** 353.

2. Select a 2D-CAM or a CONJUGATE-CAM FB as the Power Source for the Follower - **see Configure-Power Source** 480
3. Review the 2D-CAM : **Display, Properties, Roller-Life, Cam-Life, ...** **- see 2D-Cam dialog** 327
4. Add a CAM-DATA FB - **see Kinematic FBs > Add Cam-Data FB** 171
5. Edit the CAM-DATA FB to link it to the 2D-CAM- **- see Cam-Data dialog : Cam Analysis** 357
6. Connect wires from the CAM-DATA FB to a GRAPH FB to analyse 5 parameters **- see Cam-Data dialog : Cam Analysis** 357
7. Edit the CAM-DATA FB again to calculate the Cam's Coordinates - **see Cam-Data dialog : Cam-Coordinates** 361

Cam-Data dialog: Cam-Coordinates

If necessary, link the CAM-DATA FB with a 2D-CAM **- see Cam-Analysis** 357



1 - **Cam-Coordinates toolbar**

2 - **End-Caps** - this box shows **ONLY** when the first coordinate of the Cam does not equal the last coordinate. For example, a linear Slot-Cam.

3 - **Inner / Outer** - to indicate the status of the Cam-Coordinate calculations and Cams.

4 - **Cam Options** - to calculate, display, and/or save the cam-profile coordinates for the **INNER**, **OUTER**, **GROOVE**, or **NEGATIVE GROOVE** cam-type. The **NEGATIVE GROOVE** cam-type applies to the STEP file-type only.

5 - **Radial Clearance** - to offset the Cam-Profile from the true Cam-Profile

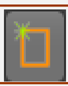
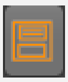
6 - **# Points** to calculate the **XY-Points** (see toolbar [362] 1 **below**)

OR

6 - **BiArc Error** to calculate the **Biarcs** (see toolbar [362] 1 **below**)

1 Cam-Coordinates toolbar:

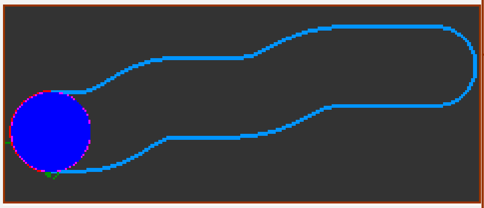
Click icon 5 to switch between XY-Points and Biarcs.

<table>
<tr><td>1</td><td></td><td>Clear ALL Data from the Table</td></tr>
<tr><td>2</td><td></td><td>Save Cam-Coordinate data as:
• STP(STEP)- XY Points 5 only
• DXF
• CSV (Comma Separated Values) - Header, then # Data-Point, X-data, Y-data, Z-data as 0.0
• TXT - Header, then # Data-Point, X-data, Y-data, Z-data as 0.0</td></tr>
</table>

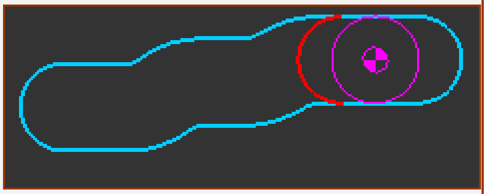
		• SLDCRV (SOLIDWORKS Curve file, tab delimited) - XY Points only , X-data, Y-data, Z-data as 0.0) **Note:** If this icon is not enabled, then make sure the PLATE-RADIUS parameter is larger than the **Maximum Radius** of the **Outer Cam**. See STEP File Options [366].
3		Copy Coordinates to your Clipboard
If CAM OPTION 4 is GROOVE CAM, click to toggle between:		
4		- INNER-CAM
		- OUTER-CAM
		- **Pitch-Curve** (XY-POINTS only)
Toggle how to Calculate the Cam:		
5		XY-POINTS - **Calculate Cam with X-Y Coordinates** Enter the **Number-of-Points** # POINTS. Use 4 to toggle between the Inner-Cam, Outer-Cam, and Pitch-Circle of the Cam Calculate the 2D-Cam at equal increments of the **MASTER MACHINE ANGLE**.
		BIARCS - **Calculate Cam with Biarcs** with a Maximum Chordal Error (**see Biarcs**) Enter the maximum **Chordal-Error** as BIARC ERROR The number of **Biarcs** is a function of the BIARC ERROR and the shape of the **Cam-Profile**. Click Rebuild after you edit BIARC ERROR
6		Before you click this button, make sure that **SOLIDWORKS** is open and the active document is a part document (SLDPRT). **Bi-Arcs** are sent to SOLIDWORKS as **Arc** sketch entities. **XY-Points** are sent to SOLIDWORKS as a list for the SOLIDWORKS **Curve** feature. Use **Convert-Entities** in SOLIDWORKS to convert the **Curve** to a **Spline** sketch entity. **See also Note 4** [370]
		The X and Y coordinates of the Cam-Profile are calculated automatically BIARCS, are also calculated automatically, but it takes more time to complete. If the TRAFFIC-LIGHT **icon remains** RED, click the TRAFFIC-LIGHTS icon again.
7		Display a preview of the cam-profile.
8		Read-only - the Cam is Open or it is Closed

2 - End Caps

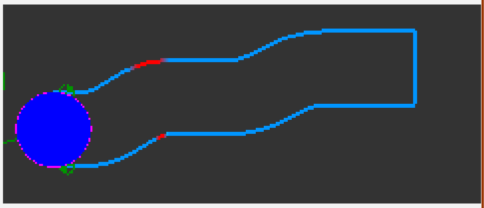
End Caps relate only to **Slot-Cams**, also known as **Linear-Cams**, or **Ramp-Cams**.



End-Caps - Show



Slot-Cam with 'False' End-Cams



End-Caps - Hide

SHOW or HIDE END-CAPS =

End-Caps work well when the independent axis to the Cam-Follower moves in one direction only - **see image "End-Caps - Show"**

If the independent axis reverses its direction in the middle of its travel, you see **False End-Caps** in the middle of the **Slot-Cam**. - **see image: "Slot-Cam with 'False' End-Caps"**.

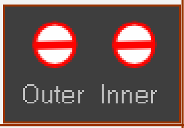
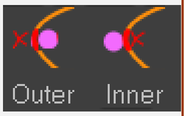
Before you export Cam-Coordinates we recommend you add a ***dummy*** slot-cam - **and move the main axis of the Slot-Cam in one direction at Constant-Velocity, over one machine-cycle.**

To make sure the Constant-Velocity is correct:

1. Connect a wire from a LINEAR-MOTION-FB to a GEARING FB
2. In the GEARING FB, enter a GEARING RATIO = MAXIMUM TRAVEL DISTANCE OF CAM/360).
3. Connect a wire from the GEARING FB to the MOTION-DIMENSION FB as the motion of the Slot-Cam.

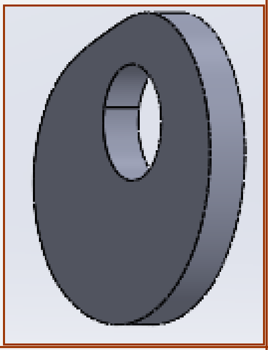
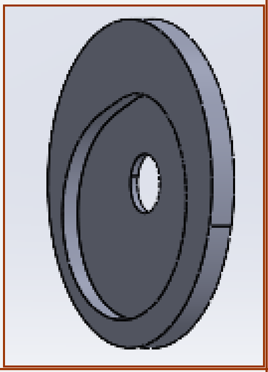
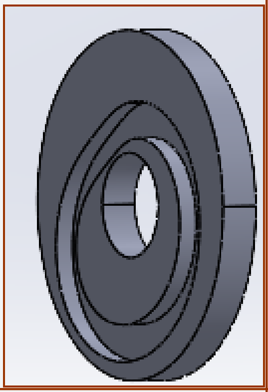
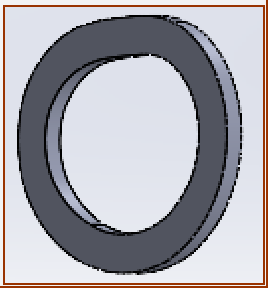
   As the **MMA** increases from 0 to 360, the Cam travels from **0** to **MAXIMUM TRAVEL DISTANCE OF CAM (mm).**

3 - Cam Status

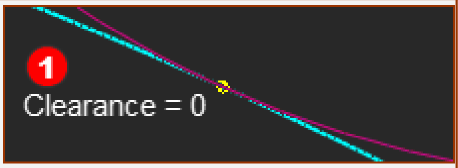
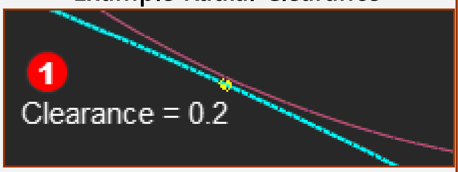
	The cam-coordinates have been calculated. The **Radius-of-Curvature** is more than the CAM ROC SOFT-LIMIT parameter as set in the **2D-Cam dialog** 329
	The cam-coordinates have not been calculated - usually after you edit the # POINTS or the BIARC ERROR.
	It is not possible to calculate the cam-coordinates correctly, possibly because of under-cutting.
	The cam-coordinates have been calculated. The Cam's **Radius-of-Curvature** is **less** than CAM ROC SOFT-LIMIT as set in the **2D-Cam dialog** 329 - **Undercutting is possible.**

4 - Cam Options

Calculate, Display, and/or Save the Cam-Coordinates:

	• **INNER** Cam-Profile ONLY
	• **OUTER** Cam-Profile ONLY
	• **GROOVE-CAM** If you select **GROOVE-CAM** option, AND if you toggle 1 ❺ to **XY-POINTS**, then you can also toggle 1 ❹ to calculate the **PITCH-CIRCLE** coordinates.
	• **NEGATIVE-GROOVE** The **Negative-Groove** option applies only when you want to save the Cam as a STEP file-type - **see** 1 ❷. The STEP file is the space filled by the path of the Cam-Follower along the cam-profile (plus or minus any Radial Clearance).

5 - Radial-Clearance

 Example Radial-Clearance  Cam with 0.2mm Radial Clearance	See also : **2D-Cam dialog > Parameters tab > Cam Range and Radius** 329 **RADIAL CLEARANCE (MM)** Default = 0 , Negative or Positive **Positive Value** - the cam is resized to give a **clearance** between the Cam-Follower and the Cam-Profile. AND: **Negative Value** - the cam is resized to give an **interference** between the Cam-Follower and Cam-Profile.
APPLICATIONS:	

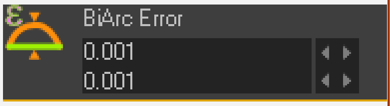
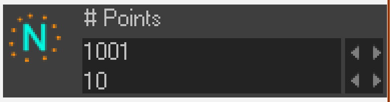
Groove-Cam - add clearance (**Positive Value**) for a Cam-Follower Roller bearing in a Groove cam-type (See **Note**)

Conjugate-Cam - to add a small clearance (**Positive Value**) to compensate for tolerances in the cam assembly.

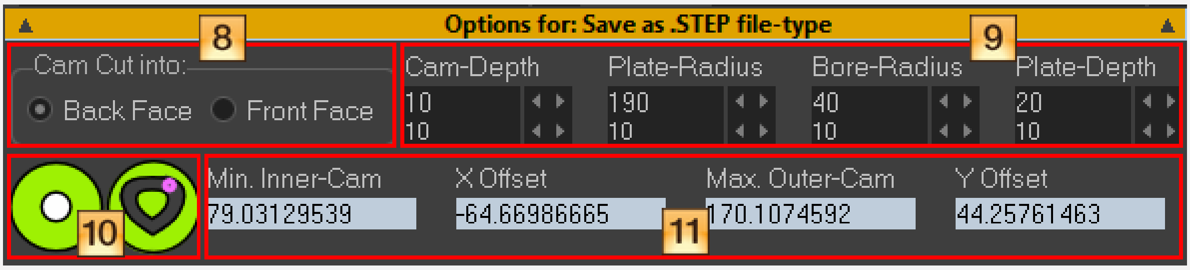
Rough-Cut - to oversize an Inner-Cam or undersize an Outer-Cam for a Rough-cut. RADIAL-CLEARANCE should be a **Negative-Value**

Note: Stud-type Cam-Followers often have a large negative tolerance bias - for example, $0 - 50\mu m$. You may consider that to be enough clearance.

6 - Cam-Coordinates

 Max Chord-Error with BiArcs	1 5 **set to Biarcs** Apply this maximum Chord-Error between the Cam-Profile and the Biarcs.
 Number of XY Points	1 5 **set to XY-Points** Calculate these number of Points for the Cam-Profile at equal increments of the **Master-Machine Angle**.

OPTIONS FOR: SAVE AS .STEP FILE-TYPE



8 - **STEP-ONLY** - To indicate the **Front Face** or the **Back Face** from which to extrude the **Groove-Cam** by CAM-DEPTH **(see also** 9)

9 - **STEP-ONLY** - Dimensions that apply to the Cam-Plate when you save the cam as a STEP file-type (**see** 1 2)

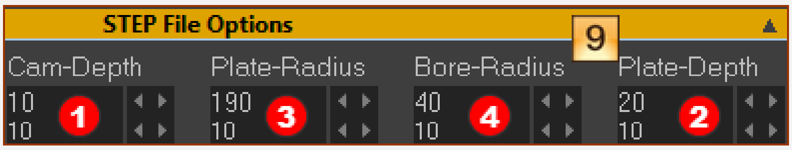
10 - **STEP-ONLY** - Schematics to indicate the Cam type, and if the PLATE-RADIUS and BORE-RADIUS are suitable **(see also** 7)

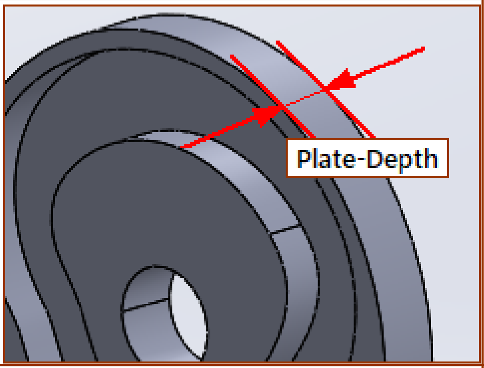
11 - **STEP-ONLY -** (Stationary Cam only) - enter an X-OFFSET and Y-OFFSET to move the center of the Cam-Plate, as required.

11 - MIN. INNER-CAM and MAX. OUTER-CAM. Read-only radius of the Cam-Profiles to help you specify the PLATE-RADIUS and BORE-RADIUS **(see also** 9)

9 Dimensions for STEP file ONLY

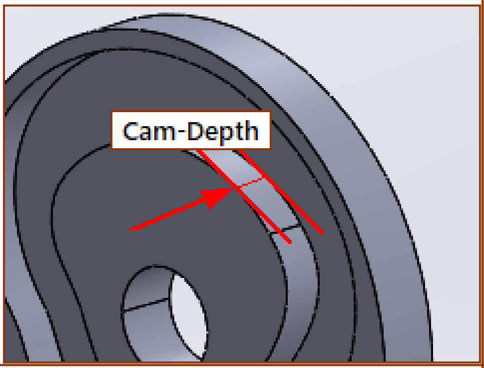
These images are of a Groove cam-type that are saved as the STEP file-type, and then opened in **SOLIDWORKS** to show the features.





❷ PLATE-DEPTH

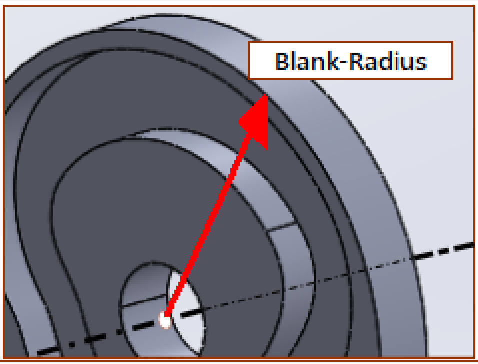
The total depth of the Cam-Plate.



❶ CAM-DEPTH

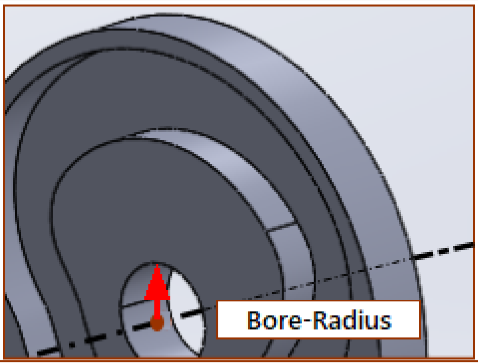
The depth of the Cam cut into the Cam-Plate. The width of the Cam-Flanks.

PLATE-DEPTH – CAM-DEPTH ≥ 2 mm



❸ PLATE-RADIUS

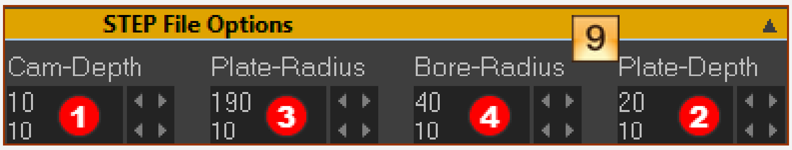
The radius of the Cam-Plate, when you select an **Outer-Cam** or a **Groove-Cam** - see



❹ BORE-RADIUS

The radius of the hole through the center of the Cam-Plate

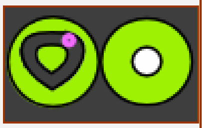
10 Schematics of Cam type and are the Dimensions OK or Not-OK?



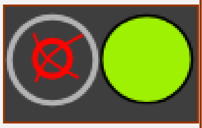
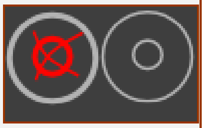
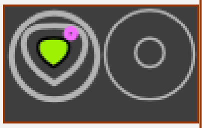
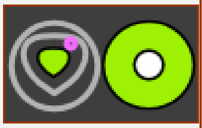
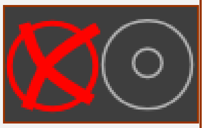
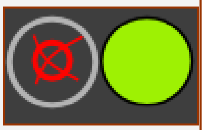
Status of Cam Type, Blank-Radius and Bore-Radius relative to Cam-Size

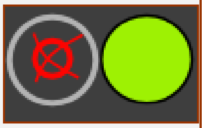
The PLATE-RADIUS ❸ parameter applies to **Outer** and **Groove Cams** only 4

The BORE-RADIUS ❹ parameter applies to all Cam types 4



4 GROOVE-CAM

	❸ PLATE-RADIUS is **OK** - it is greater than the **Maximum Radius of the Outer-Cam** 11 ❹ BORE-RADIUS is **OK** - it is less than the **Minimum Radius of the Inner-Cam** 11
	4 GROOVE-CAM ❸ PLATE-RADIUS is **NOT OK** - it is less than the **Maximum Radius of the Outer-Cam** 11 ❹ BORE-RADIUS is **OK** - it is less than the **Minimum Radius of the Inner-Cam** 11
	4 GROOVE-CAM ❸ PLATE-RADIUS is **OK** - it is greater than the **Maximum Radius of the Outer-Cam** 11 ❹ BORE-RADIUS is **NOT OK** - it is greater than the **Minimum Radius of the Inner-Cam** 11
	4 GROOVE-CAM ❸ PLATE-RADIUS is **NOT OK** - it is less than the **Maximum Radius of the Outer-Cam** 11 ❹ BORE-RADIUS is **NOT OK** - it is greater than the **Minimum Radius of the Inner-Cam** 11
	4 INNER-CAM ONLY ❸ PLATE-RADIUS is **0mm** ❹ BORE-RADIUS is **0mm** - the Cam does not include a Hole through its center
	4 INNER CAM ONLY ❸ PLATE-RADIUS is **OK** - it is greater than the **Maximum Radius of the Outer-Cam** 11 ❹ BORE-RADIUS is **OK** - it is less than the **Minimum Radius of the Inner-Cam** 11
	4 INNER-CAM ONLY ❸ PLATE-RADIUS is **NOT OK** - but, this dimension is ignored with **Inner Cams** 4 ❹ BORE-RADIUS is **OK** - it is less than the **Minimum Radius of the Inner-Cam** 11 You can save the STEP file.
	4 INNER-CAM ONLY ❸ PLATE-RADIUS is **OK** - it is greater than the **Maximum Radius of the Outer-Cam** 11 ❹ BORE-RADIUS is **NOT OK** - it is greater than the **Minimum Radius of the Inner-Cam** 11

	You **can NOT** save the STEP file.
	4 OUTER-CAM ONLY ❸ PLATE-RADIUS is **OK** - it is greater than the **Maximum Radius of the Outer-Cam** 11 ❹ BORE-RADIUS is **OK** - it is less than the **Minimum Radius of the Inner-Cam** 11
	4 OUTER CAM ONLY ❸ PLATE-RADIUS is **NOT OK** - it is less than the **Maximum Radius of the Outer-Cam** 11 ❹ BORE-RADIUS is **OK** - it is less than the **Minimum Radius of the Inner-Cam** 11 You **can NOT** save the STEP file.
	4 OUTER-CAM ONLY ❸ PLATE-RADIUS is **OK** - it is greater than the **Maximum Radius of the Outer-Cam** 11 ❹ BORE-RADIUS is **NOT OK** - it is greater than the **Minimum Radius of the Inner-Cam** 11 You **can** save the STEP file.
	4 NEGATIVE GROOVE-CAM ❸ PLATE-RADIUS is **OK** - it is greater than the **Maximum Radius of the Outer-Cam** 11 ❹ BORE-RADIUS is **OK** - it is less than the **Minimum Radius of the Inner-Cam** 11

11 Minimum/Maximum Radius of Cams, Offset X and Offset Y.

MINIMUM OF INNER-CAM and MAXIMUM OF OUTER-CAM **(see also** 9 **)**

Min. Inner-Cam	Max. Outer-Cam
79.03129539	170.1074592

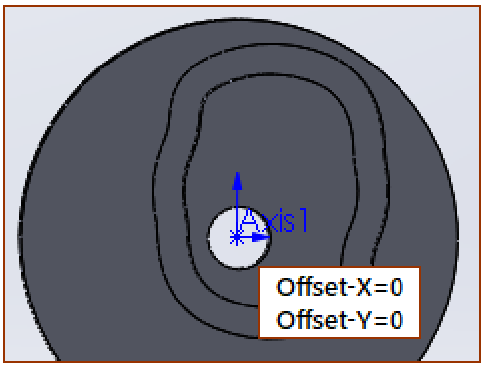
These value are useful to know when you want to make sure there is a minimum wall thickness between the:

- BORE diameter and the minimum radius of the INNER Cam.
- PLATE diameter and the maximum radius of the OUTER Cam.

X-OFFSET and Y OFFSET

When the Cam is stationary, enter an X-OFFSET and Y-OFFSET to move the center of the Cam-Plate relative the 0,0 of the **MECHANISM-PLANE** / **BASE-PART**.

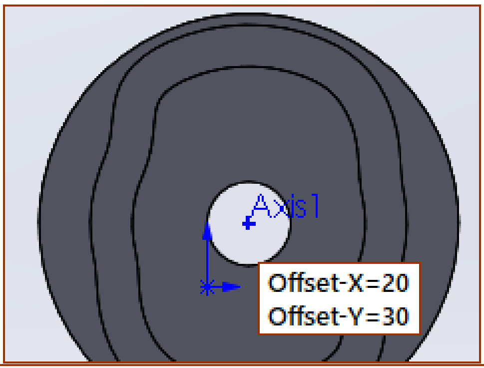
X Offset	Y Offset
-64.66986665	44.25761463



Offset-X = 0, Offset-Y = 0

These are read-only if the Cam is a rotating part.

Image left: the **Blank-Radius** and **Blank-Bore** with **Offset-X=0,-Y=0.**



Centre-X = 20, Center-Y = 30

Image left: the **Blank-Radius** and **Blank-Bore** with Offset-X=20,Y=30.

The maximum radius of the **Blank-Bore** can be increased, and the minimum radius of the **Blank-Radius** can be decreased.

Notes:

Note 1: When a 2D-CAM is a rotating-cam, make sure that the **START-POINT** (origin) of the rotating **Cam-Part** is at the center of rotation.

Note 4: Cam-Data and SOLIDWORKS.

Default Case: The Cam-Profile and SOLIDWORKS Curve feature are coplanar with the Front-View of the SOLIDWORKS part document.

To move the Cam-Profile to a different Plane: Save the Cam-Profile as XY-Points as a SLDCRV file-type. Then in Excel, move the X (or Y) column to the Z column, and the Z column to the X (or Y) column. Save the Data again. Rename it again to **file-name.SLDCRV**

To mirror the cam: Multiply the X-column data by –1, to mirror the data about the Y-axis.

To import the modified data: In a SOLIDWORKS part document, do: **Insert > Curve > Curve Through XYZ Points…** and Browse to the find **file-name.SLDCRV**.

Note5: The Cam as **BiArc Data** has six columns:

1	2	3	4	5	6
Arc Radius	Arc Start Point X Coordinate	Arc End Point X Coordinate	Arc Start Point Y Coordinate	Arc End Point Y Coordinate	Arc Angle Range (degs)

1.10.20 Dialog: Function-Block: Linear-Motion

Linear-Motion FB

See Add Linear-Motion FB 147

When you enable **Run** > **Cycle (ALT+C)**, the default output from the **LINEAR-MOTION FB** is equal to the **MASTER-MACHINE-ANGLE (MMA).**

Use the output from the **LINEAR-MOTION FB** as the machine-angle and the input to other **Kinematic FBs**.

Edit the **LINEAR-MOTION FB** to change the output relative to the **MASTER-MACHINE-ANGLE**.

How to open the Linear-Motion dialog-box

Edit the **LINEAR-MOTION FB**:

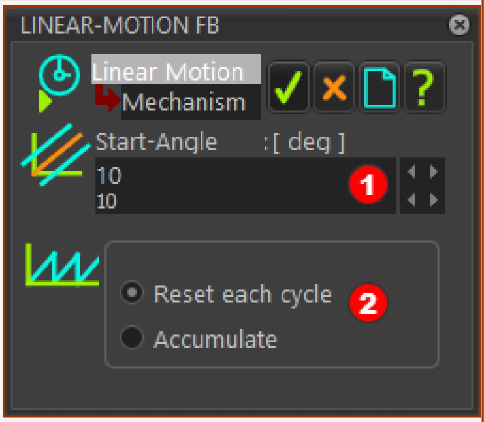
1. Double-click the **LINEAR-MOTION FB** in the graphic-area.

OR

1. **See How to Open a dialog** 513

The **LINEAR-MOTION FB DIALOG** is now open.

Linear-Motion dialog



Linear Motion FB dialog

START-ANGLE: ❶

Enter a real number in the data-box❶

A positive value advances the timing relative to the MMA.

OUTPUT OPTIONS: ❷

⦿ **RESET EACH CYCLE (DEFAULT)**

Output = Master Machine Angle + ❶

⦿ **ACCUMULATE**

Output = (Revs × 360) + Master Machine Angle + ❶

The function of the **LINEAR-MOTION FB** is:

If **RESET**❷: Output = MMA + **START-ANGLE** ❶

OR

If **ACCUMULATE**❷: Output = (Revs × 360) + Master Machine Angle + **START-ANGLE** ❶

1.10.25 Dialog: Function-Block: Gearing

Gearing FB

see **Add Gearing FB** 148

Use GEARING FB to transform motion-data* with a simple linear mathematical function. Open the GEARING FB DIALOG to edit the parameters of the linear function.

The default parameters do not change the motion-data - the output motion-data are equal to those at the input.

* Position, Velocity, and Acceleration, with Linear or Angular motion-units.

How to open the Gearing FB dialog-box

Edit the GEARING FB

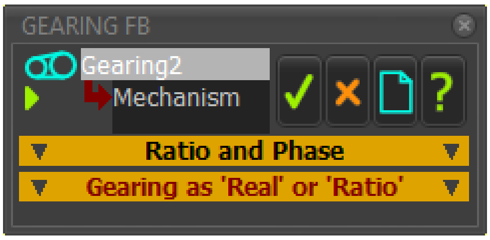
1. Double-click the GEARING FB in the graphic-area.

OR

1. **See How to Open a dialog** 513

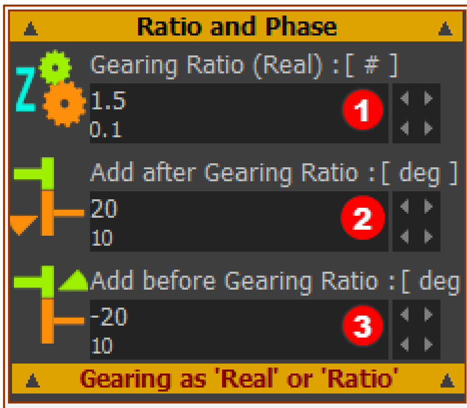
The GEARING FB DIALOG is now open.

Gearing FB dialog



Gearing FB dialog

RATIO AND PHASE



Ratio and Phase

Gearing Parameters

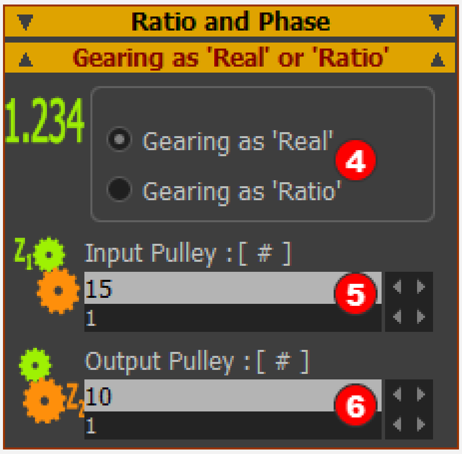
❶ GEARING RATIO (**See** GEARING AS 'REAL' OR 'RATIO' **below**)
❷ ADD AFTER GEARING-RATIO
❸ ADD BEFORE GEARING-RATIO

The Gearing function.

The GEARING FB function is:

Output = { ❶ × (Input + ❸) } + ❷

GEARING AS 'REAL' OR 'RATIO'



Ratio of Pulley Teeth

Enter the **GEARING RATIO** in one of two ways:

❹ **GEARING AS 'REAL'**

❹ **GEARING AS 'RATIO'**

If **GEARING AS REAL** is enabled:

❶ **GEARING RATIO** : ± Real Number - see above

If **GEARING AS RATIO** is enabled:

❶ **GEARING RATIO** = ❻ **OUTPUT-PULLEY** / ❺ **INPUT-PULLEY**

Enter

❺ **INPUT-PULLEY** : ± Integer, and 0

AND

❻ **OUTPUT-PULLEY** : + Integer

Output from the Gearing FB

Example

GEARING-RATIO =1.5

ADD AFTER GEARING-RATIO = 83

ADD BEFORE GEARING-RATIO = 23

INPUT = 37

$$Output = [1.5 \times (37 + 23)] + 83 = 173$$

1.10.26 Dialog: Function-Block: Motion

Motion FB

see **Add Motion FB** 149

A **MOTION FB** is a direct a link to a **Motion**, which is identified by its **Motion name-tab** in **MotionDesigner**.

After you edit a **MOTION FB** to select a **Motion**:

- The motion-data at the input-connector to the **MOTION FB** control the **X-axis** of the **Motion**
- The motion-data at the output-connector from the **MOTION FB** correspond to the **Y-axis** of the **Motion**

Note: The **MOTION FB** selects the left-most **Motion name-tab** as the default motion.

How to open the Motion FB dialog

Edit the **MOTION FB**:

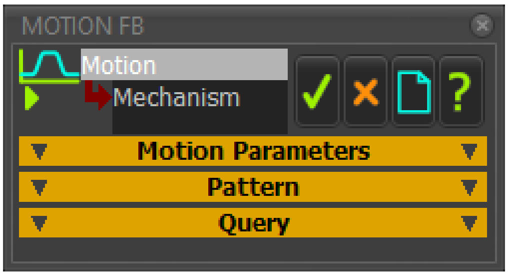
1. Double-click a **MOTION FB** in the graphic-area

OR

1. **See How to Open a dialog** 513

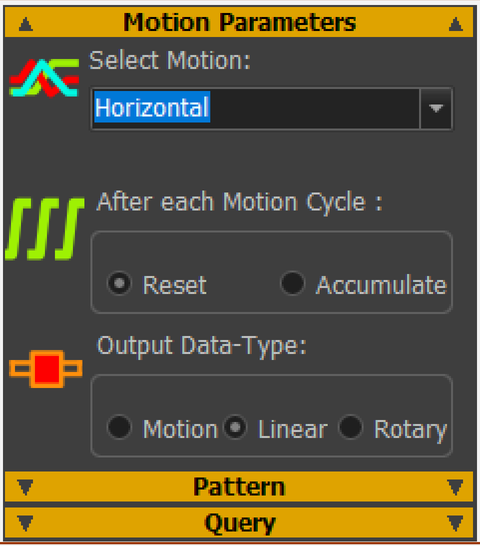
The **MOTION FB DIALOG** is now open.

Motion FB dialog



Motion FB dialog

MOTION PARAMETERS



SELECT MOTION:

Click the drop-down box to select a motion.

The default link is to the left-most **Motion name-tab** and **Motion**.

AFTER EACH MOTION CYCLE:

A **Motion-Cycle** is one revolution of the **Master-Machine Angle** - from 0 to 360.

After each Motion-Cycle, you can:

⦿ RESET OUTPUT: (default)

Start-Position (Machine-Cycle **N**) = Start-Position (Machine-Cycle **1**)

⦿ ACCUMULATE OUTPUT [375]:

Start-Position (Machine-Cycle **N)** = End-Position (Machine-Cycle **N-1)**

OUTPUT DATA-TYPE:

⦿ MOTION : (default, for **ROCKERS** and **SLIDERS**)

⦿ LINEAR *

⦿ ROTARY *

* You **must** select LINEAR or ROTARY if you connect a wire from a **MOTION FB** to a **MOTION-PATH** or a **MATH FB**.

Notes :

Usually, the motion-data at the input to a **MOTION FB** increase at a constant rate from 0 to 360, again and again.

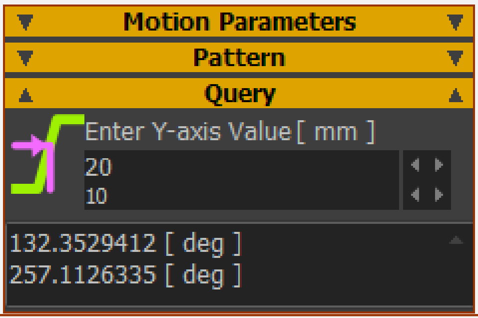
However, the motion-data at the input to the **MOTION FB** can fluctuate in any way.

For example, the input can be from a **MOTION FB**, a **POINT-DATA FB**, or a **MEASUREMENT FB**.

PATTERN

Ignore this separator

QUERY



Enter a Y-axis value

The values in the box (below) are the X-axis values that relate to the Y-axis value for the Selected Motion - see **MOTION PARAMETERS**

Reset or Accumulate after each Motion-Cycle?

This is best explained with an example model:

1. Design an **Indexing Motion** in ***MotionDesigner*** with two Segments:

   Segment 1: Index (X-axis: 0-240; Y-axis: 0-90)

Segment 2: Dwell (X-axis: 240-360, Y-axis: 90).

2. Connect a **LINEAR-MOTION FB** to a **GEARING FB**. Connect the **GEARING FB** to a **MOTION FB**. Connect the **MOTION FB** to a **MOTION-DIMENSION FB** which controls an rotating Motion-Part.
3. Edit the **MOTION FB**.

   Select the **Indexing Motion** you have designed.

   Select **RESET** as the **AFTER EACH MOTION-CYCLE** parameter
4. Edit the **GEARING FB.**

   Edit the **GEARING RATIO**. Enter 4

Result:

As the MMA increases from 0 to 360, the output from the **GEARING FB** increases from 0 to 1440 (**GEARING RATIO** × 360)

As the MMA increases from 0 to 360, the output from the **MOTION FB** increases from 0 to 90, and jumps to 0 again each time the input is an integer multiple of 360.

The indexing-shaft resets its position 4 times.

Now, select **ACCUMULATE OUTPUT** as the **AFTER EACH MOTION CYCLE** parameter

The indexing-shaft indexs 4 times, to make one complete revolution, as the input increases from 0 to 1440.

1.10.20 Dialog: Function-Block: Motion-Dimension

Motion-Dimension FB

see **Add Motion-Dimension FB** 150

A **MOTION-DIMENSION FB**

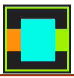
- identifies a **PART** you want to be a **MOTION-PART** - a **Rocker** or **Slider**.
- controls the starting position of the **MOTION-PART**
- controls the positive direction of the **MOTION-PART**

The two icons in the Graphic-Area for a Motion-Dimension FB

The **MOTION-DIMENSION FB** can control an Angular or a Linear dimension.

We use different icons for the **FB**:

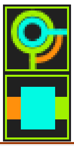
This icon when the **MOTION-DIMENSION FB** controls an **Angular Dimension** (ANGLE) for a **Rocker**



This icon when the **MOTION-DIMENSION FB** controls a **Linear Dimension** (DISTANCE) for a **Slider**

How to open the Motion-Dimension dialog

To open the **MOTION-DIMENSION** dialog:



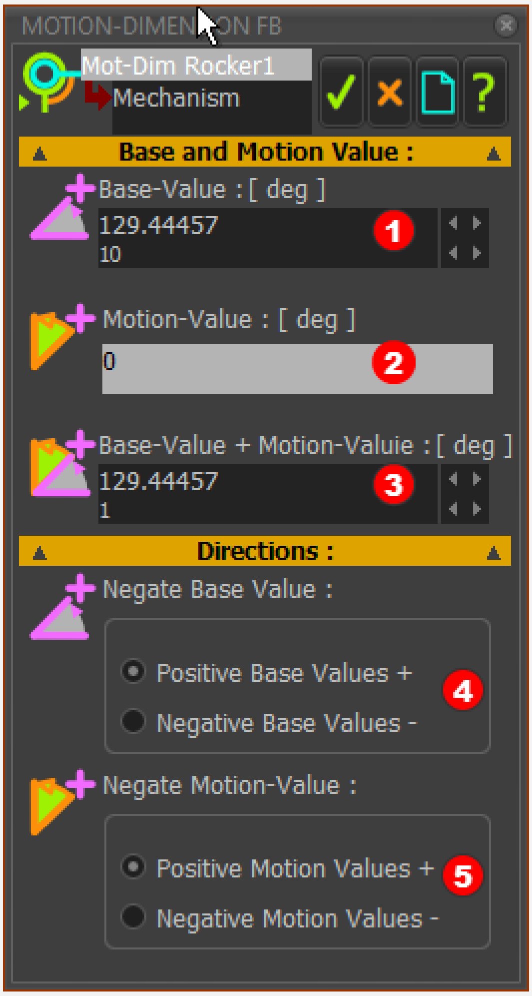
1. Double-click the Linear or Angular **MOTION-DIMENSION FB** in the graphic-area

OR

1. **See How to Open a dialog** 513

The **MOTION-DIMENSION DIALOG** is now open.

Motion-Dimension dialog



Motion-Dimension dialog

The **MOTION-DIMENSION FB** function is:

MOTION-DIMENSION = BASE-VALUE❶ + Input Motion-Value❷

Output Motion-Value❸ = BASE-VALUE❶ + Input Motion-Value❷

Note: UNITS (angular or linear) of the motion-data are defined by the data-type at the input-connector.

BASE AND MOTION VALUES

❶ **BASE-VALUE:**

The **Constant** contribution to the MOTION-DIMENSION

❷ **MOTION-VALUE (AT INPUT) (read-only)**

The MOTION contribution to the MOTION-DIMENSION - equal to the motion-data at the input-connector.

❸ **BASE-VALUE + MOTION-VALUE (Constant + Motion)**

BASE-VALUE + MOTION-VALUE = MOTION-DIMENSION at the **Rocker** or **Slider**.

DIRECTIONS

❹ **NEGATE MOTION-VALUES**

⦿ POSITIVE MOTION-VALUES : MOTION-VALUE × 1 (default)

⦿ NEGATIVE MOTION-VALUES : MOTION-VALUE × –1

❺ **NEGATE BASE-VALUES**

⦿ POSITIVE BASE-VALUE : BASE-VALUE × 1 (default)

⦿ NEGATIVE BASE-VALUE - BASE-VALUE × –1

Notes:

The positive direction of a:

Rocker - counter-clockwise angle

Slider - direction of arrowhead on Slide-Joint - See Positive Direction of Sliders 162

1.10.28 Dialog: Function-Block: Motion-Path

Motion-Path FB

see **Add Motion-Path FB** 164

IMPORTANT:

The **Data-Type** of the motion-data at the input to a MOTION-PATH FB must be **Linear** or **Rotary**.

When the **Data-Type** at the input are:

- **Linear**, the linear position of the MOTION-POINT along the sketch-loop = **Positional value***
- **Rotary**, the linear position of the MOTION-POINT along the sketch-loop = **Angular-Value*** **× (total length of sketch-loop / 360)**

* when you LINEAR OFFSET or PHASE OFFSET parameters are zero - see here 380

See more Data-Type and Motion-Points 384

How to open the Motion-Path dialog

Edit the MOTION-PATH FB:

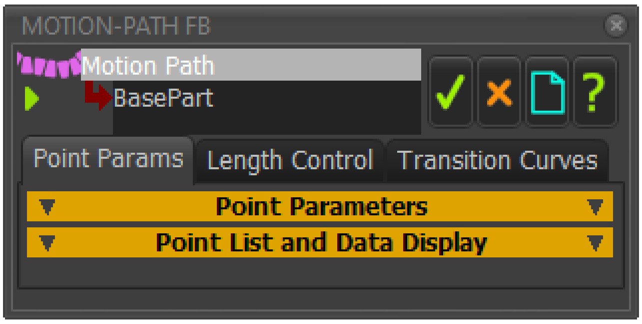
1. Double-click the MOTION-PATH FB in the graphic-area

OR

1. **See How to Open a dialog** 513

The MOTION-PATH DIALOG is now open.

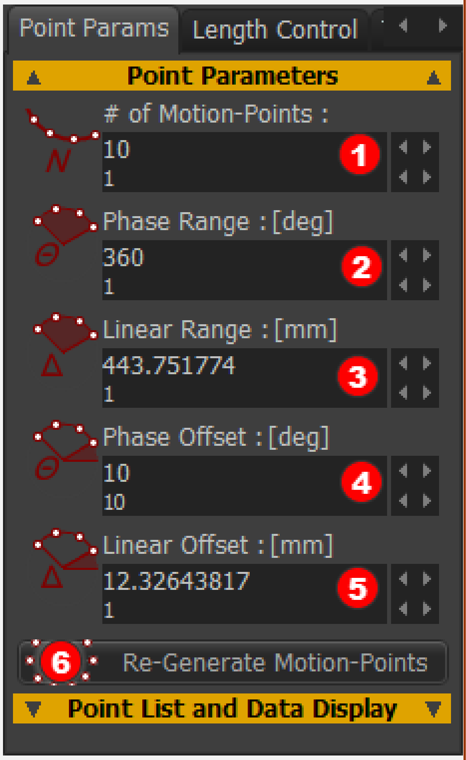
Motion-Path dialog



Motion-Path dialog

Point Parameters tab

POINT PARAMETERS



❶ **# MOTION-POINTS:** | Default = 1

To edit the number of **MOTION-POINTS** along the **sketch-loop**.

1. Increase or decrease the **# OF MOTION-POINTS**❶
2. Click [Re-Generate Motion-Points] button ❻. at the bottom of **POINT PARAMETERS** separator.

MOTION-POINTS distribute equally along the **PHASE-RANGE**❷ and **LENGTH-RANGE**❸ of the **sketch-loop**.

If there is a **PIN-JOINT** between a **PART** and a **MOTION-POINT**, you must add the **PIN-JOINT** again.

---

❷ **PHASE RANGE** | Default = 360 which is equivalent to the total length of the **sketch-loop**

❸ **LINEAR-RANGE** | Default = Length of the **sketch-loop**.

When **# MOTION-POINTS >1**, the **MOTION-POINTS** are distributed equally along the **PHASE-RANGE** and **LINEAR-RANGE** of the **sketch-loop**.

1. Edit the **PHASE RANGE**❷ (deg)

OR

1. Edit the **LINEAR RANGE**❸ (mm)

When you edit **PHASE-RANGE**❷, the **LINEAR-RANGE** updates automatically, and vice-versa.

These two parameters indicate the proportion of the **sketch-loop** over which to distribute **MOTION-POINTS** (when **# MOTION-POINT >1**).

These parameter distribute **MOTION-POINTS** (when **# MOTION-POINTS > 1**) over a range (length) that is different to the total-length of the **sketch-loop**.

The distribution of the **MOTION-POINTS** is best illustrated with examples. We urge you to EXPERIMENT.

EXAMPLE

Say : **Actual length of a sketch-loop** is **120.0mm**

The **NUMBER OF MOTION-POINTS = 3**

Default: **PHASE-RANGE =360** (and **LINEAR-RANGE = 120.0**)

The 3 **MOTION-POINTS** are equally spaced over **120.0mm**. The distance between each **MOTION-POINT = 127.5/3 =40.5mm**

Position of **MOTION-POINT #1 = 0MM, MOTION-POINT #2 = 42.5MM,** and **MOTION-POINT #3 = 85.0MM**

Edit PHASE-RANGE = 120 (and **LINEAR-RANGE = 42.5**)

The 3 **MOTION-POINTS** are distributed over a linear-length of **127.5/3 = 42.5mm**

The distance between each **MOTION-POINTS** is **42.5/(3-1) = 21.25mm**

Position of MOTION-POINT #1 = 0MM, MOTION-POINT #2 = 21.25, and MOTION-POINT #3 = 42.5MM

PHASE OFFSET (DEG)❹ | Default = 0

LINEAR OFFSET (MM)❺ | Default = 0

PHASE OFFSET and LINEAR OFFSET move of ALL of the **MOTION-POINTS** along the **sketch-loop**. The OFFSET value indicate the position of **MOTION-POINT #1** when the motion-value at the input-connector of the MOTION-PATH FB is equal to 0; or when a wire is not connected to its input-connector.

PHASE-OFFSET: equates the length of the **sketch-loop** to **360 degrees of a circle.**

LINEAR-OFFSET: equates the length of the **sketch-loop** to its **actual linear length**.

EXAMPLE - we urge you to EXPERIMENT.

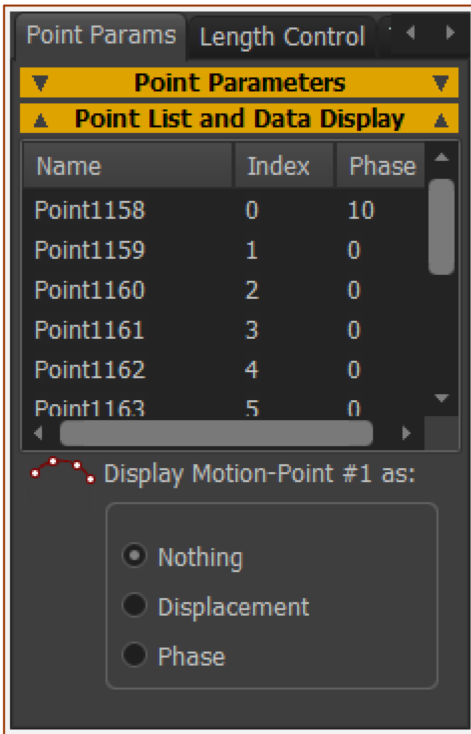
A **sketch-loop** is 127.5mm long. There is **1 MOTION-POINT**.

- A PHASE-OFFSET=40 (deg) moves **MOTION-POINT #1** to 127.5×40/360=14.16mm along the **sketch-loop**. All other **MOTION-POINTS** move the same distance.
- A LINEAR-OFFSET=40 (mm) moves **MOTION-POINT #1** to 40mm along the **sketch-loop**. All other **MOTION-POINTS** move the same distance.

Re-generate Motion-Points ❻ BUTTON

If you change NUMBER OF MOTION-POINTS❶ parameter, you **must** click the button to generate **MOTION-POINTS**.

POINT LIST AND DATA DISPLAY



Headers: Name | Index | Phase (read-only*)*

- ***Name*** is the name to identify the **MOTION-POINT**

  If you click on a **MOTION-POINT** in the list, it shows as ***red*** in the graphic-area.
- ***Index*** is the # **MOTION-POINT**
- ***Phase*** is the relative displacement along the sketch-path, relative to the fist **MOTION-POINT**

DISPLAY MOTION-POINT #1 AS:

You can show the position of **MOTION-POINT #1** on the **sketch-loop**.

⦿ SHOW NOTHING - nothing shows above the **MOTION-POINT #1** in the graphic-area

⦿ SHOW DISTANCE - the linear position of **MOTION-POINT #1** is on the **sketch-loop**

⦿ SHOW PHASE - the 'angular' position of **MOTION-POINT #1** is on **sketch-loop**. 360 is the total-length of the **SKETCH-PATH**.

Length Control tab

CONTROL LENGTH OF PATH

Select a dimension to control the length of the **sketch-loop** that is associated with the MOTION-PATH FB.

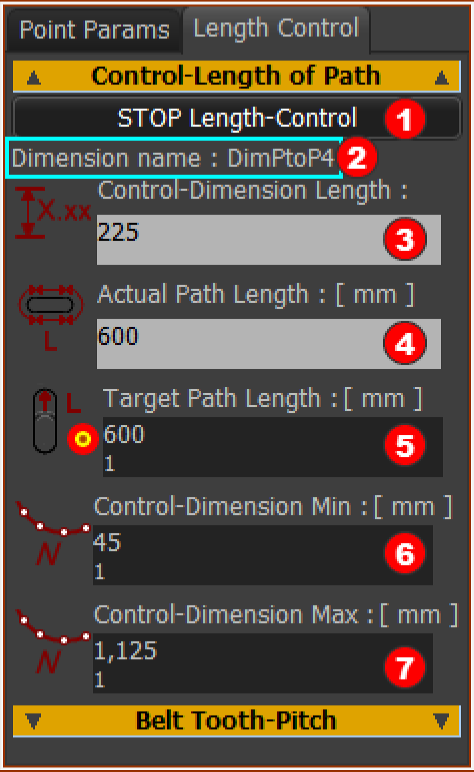
The dimension becomes the CONTROL-DIMENSION - **see Note 1**

IMPORTANT

In the **PART-EDITOR**, the **sketch-loop** should be **fully defined** before you select the CONTROL-DIMENSION.

There should be TANGENT constraints between LINES and ARCS in the **sketch-loop**.

Then, as you edit the CONTROL-DIMENSION, the shape of the **sketch-loop** is more predictable - **see Note 1**



❶ START: Length Control / STOP: Length Control

To control the length of the Motion-Path with a Dimension:

1. Click the START Length Control BUTTON❶

   The **PART-EDITOR** opens.

2. Click a dimension that can control the total length of the **sketch-loop** - **see Note 1**

   The **PART-EDITOR** closes.

The name of the CONTROL-DIMENSION is now under the button❷.

For example, **DimPtoP2**

When you do not want to control the length of the Motion-Path with a dimension:

1. Click STOP Length Control ❶

PARAMETERS

CONTROL-DIMENSION LENGTH - **read-only**❸

The active value of the CONTROL-DIMENSION.

ACTUAL PATH LENGTH - **read-only**❹

The active length of the **sketch-loop** associated with the MOTION-PATH FB

TARGET PATH LENGTH❺

3. Edit the length of the **sketch-loop** associated with the MOTION-PATH FB - **see Note 2**

Note: if the Spin-box tool does not show, double-click in the data-box

CONTROL-DIMENSION MIN❻ / MAX❼ - **see Note 3 below**

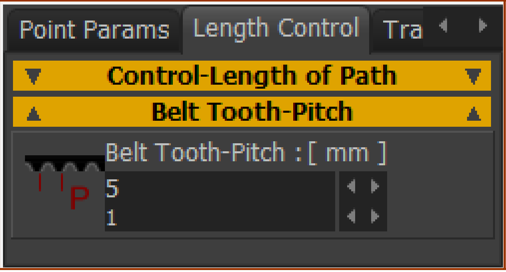
When you select the CONTROL-DIMENSION:

- CONTROL-DIMENSIONS MINIMUM❻ is estimated as 0.5 × CONTROL-DIMENSION - **see Note 3.**
- CONTROL-DIMENSIONS MAXIMUM❼ is estimated as 2.0 × CONTROL-DIMENSION - **see Note 3.**

If required:

	4. Edit the CONTROL-DIMENSION MINIMUM 6 and MAXIMUM 7 limits **- see Note 2.**

BELT TOOTH PITCH



The BELT-TOOTH-PITCH does not change the belt length.

The diameter of a PULLEY is a function of the NUMBER OF TEETH AND BELT TOOTH-PITCH.

See **Pulley dialog** 474

Notes 1,2,3

Note 1: CONTROL-DIMENSION

The CONTROL-DIMENSION should be:

DimPtoP#	the distance between two POINTS
DimL#	the length of a LINE

The CONTROL-DIMENSION should NOT be:

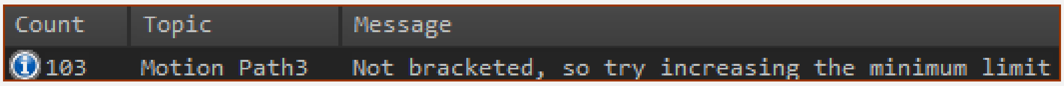
DimPtoL#	the perpendicular distance form a POINT to a LINE
Gray before you select it. For example, it is controlled with a Design-Set	

Note 2: TARGET PATH-LENGTH

Edit the TARGET PATH-LENGTH with small increments. If the ACTUAL PATH LENGTH does not change, **see Note 3**

Note 3: CONTROL-DIMENSION MIN and/or MAX

Message in FEEDBACK-AREA (below the graphic-area)



The default values for the CONTROL-DIMENSION MIN or MAX are such that the ACTUAL-PATH LENGTH can be equal to the TARGET PATH LENGTH.

However, it is possible that the ***Root-Finding Math*** is confused!

To reset the Root-finding Math:

- Increase or decrease the CONTROL-DIMENSION MIN/MAX, then try to edit the TARGET-PATH-LENGTH again.
- You may need to edit these values to be almost equal to the actual length of the CONTROL-DIMENSION.

Transition Curve tab

Information coming with MD16

MOTION-PATH AND DATA TYPES

DATA-TYPE AND MOTION-POINTS

The position of the MOTION-POINTS along the **sketch-loop** is a function of the **Data-Type** at the input-connector to the MOTION-PATH FB. The **Data-Type** at the input-connector must be

LINEAR or ROTARY.

DATA-TYPE = LINEAR

The position of the **MOTION-POINT #1** along the **sketch-loop** has a scale of 1 : 1 with the motion-value at the input-connector to the **MOTION-PATH FB**

∴ the position of the **MOTION-POINT #1** along the **sketch-loop** = motion-value at input-connector(mm).

DATA-TYPE = ROTARY

The position of **MOTION-POINT #1** along the **sketch-loop** has a scale of **360 : 1 × Path-Length.**

∴ the position of the **MOTION-POINT #1** along the **sketch-loop =** Total Length of the **sketch-loop ×** motion-value at input-connector(degrees)**/360**)

1.10.29 Dialog: Function-Block: Graph

Graph FB

see **Add Graph FB** 172

Use a **GRAPH FB** to plot **Kinematic Motion-Data**, **Kinetostatic Force-Data** and **Cam-Data**.

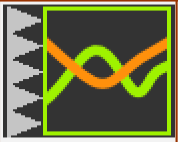
There are:

➔ 4 × **Y-axis input**: the top four(4) input-connectors

➔ 1 × **X-axis input:** the bottom input-connector is the **X-axis** for all of the graphs.

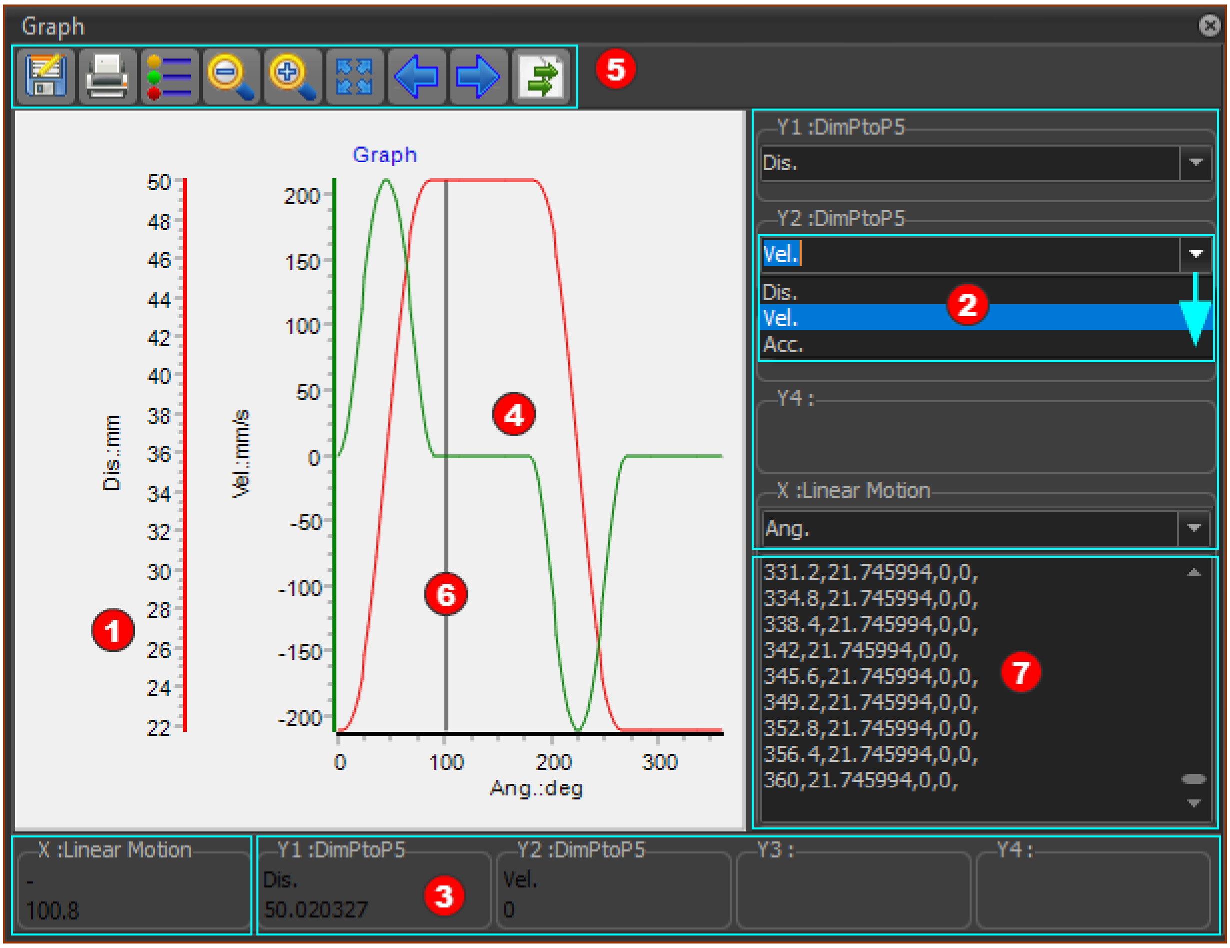
If you do not connect a wire to the X-axis input-connector, the **X-axis** of the graph is one machine-cycle, 0 – 360.

How to open the Graph Interface

	Edit the **GRAPH FB**: 1. Double-click the **GRAPH FB** in the graphic-area. OR 1. **See How to Open a dialog** 513.

The **GRAPH INTERFACE** is now open.

Graph Interface



Grap dialog/interface

❶ Y–axes

There is one Y-axis scale for each wire that you connect to an input-connector. Maximum = 4

From left to right, the Y-axis scales correspond to each wire that you connect to an input-connectors, from top to bottom.

The color of each Y–axis scale is the same as the graph.

1. Edit the colors with the Graph Setting dialog [389].

The minimum and maximum of the Y-axis scale are the minimum and maximum values of the data at the input-connector.

1. Edit the minimum and maximum values for the Y-axis scales with the Graph Setting dialog [389].

❷ **Y–axis Data-Channels (Drop-Down):** See also Y–axis Data-Channel Display Options [388]

Each wire that you connect to an input-connector has 3 Data-Channels.

1. Click the drop-down arrow to select which Data-Channel to plot for each wire at the input-connector.

❸ **Digital Readout (D.R.O.):**

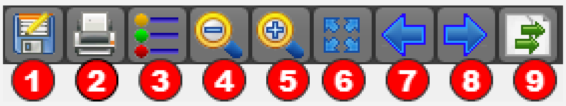
The X–axis and the Y1 to Y4-axis values in the Digital Readout are those values of the graph at the position of the vertical cursor❻

The Number-Format for the X–axis and Y–axis values is configured in Application Settings | Number Format tab. [286]

❹ **Graph Area:**

There is a graph for each wire you connect to a Y-axis input-connector

❺ **Graph toolbar:**



❶ Save data as

❷ Print graphs

❸ Open Graph Settings dialog [389]

❹ Zoom-Out

❺ Zoom-In

❻ Zoom-Extents

❼ Pan Left

❽ Pan Right

❾ Copy Graph Data to Clipboard – see below

❻ **Vertical-Cursor**

When you click or drag your pointer in the graph-area❹:

- the Vertical-Cursor moves with your pointer
- the Digital-Readouts❸ are equal to the Y-axis values of the Vertical-Cursor
- the MASTER MACHINE ANGLE continuously updates to equal the X-axis value of the Vertical-Cursor

- the kinematic-chains that are **kinematically-defined** move to agree with the MASTER MACHINE ANGLE.

7 Graph Data in Clipboard

To list the graph data-values 7,

1. Click the right-most icon in the **Graph toolbar** 9.

   There is one data-point for each Machine Step - see Machine-Settings dialog > Number-of-Steps 291.

You can Paste the data:

- to Excel, or Notepad, ...
- to the ***MotionDesigner*** **Data-Transfer Table**

From the **Data-Transfer Table**, you can paste the data:

- as a dumb Overlay-Trace to compare it with a motion
- to a Position List, Acceleration List, or Z Raw Data segment-type, and link it to a MOTION-PART or MOTION-POINT.

Y-Axis Data-Channels:

Each wire that you connect to a GRAPH FB has three(3) data-channels.

Use the drop-down arrow to select which **Y-Axis Data-Channel** 2 to plot for each **Y-axis.**

The Data-Channels that are available are a function of the Function-Block and wire you connect at the input to the Graph FB.

DATA-CHANNELS:

Kinematic Function-Blocks 142 (Linear-Motion, Gearing, Motion, Motion-Dimension) you can plot:

- LINEAR OR ANGULAR POSITION
- LINEAR OR ANGULAR VELOCITY
- LINEAR OR ANGULAR ACCELERATION

Measurement FB 166 or **Point-Data FB** 169 you can plot:

- LINEAR OR ANGULAR POSITION
- LINEAR OR ANGULAR VELOCITY
- LINEAR OR ANGULAR ACCELERATION

Cam-Data FB 171

- PRESSURE ANGLE.
  You can plot three Pressure Angles – See **Pressure Angle**
- RADIUS-OF-CURVATURE.
  You can plot the Radii-of-Curvature of the Inner and the Outer Cam.
- CONTACT CAM FORCE
- MAXIMUM CONTACT SHEAR STRESS.
- SLIDING-VELOCITY
  Plot the Sliding-Velocity between the Cam-Profile and Cam-Follower for the INNER Cam and the OUTER Cam.

Force-Data FB 191

The output form a FORCE-DATA FB is a function of the element with which it is linked. **See Force-Data FB dialog** [442]

If you link a FORCE-DATA FB to a:

- PIN-JOINT with a Motor, you can plot the **APPLICATION LOAD**.
- SLIDE-JOINT with a Motor, you can plot the **APPLICATION LOAD**
- Joint that does not have a Motor, you can plot:
  - ☑ **TOTAL FORCE** ($F_T = \sqrt{(F_X^2 + F_Y^2)}$)
  - ☑ **FORCE ALONG THE X-COORDINATE OF A POINT:** F_X
  - ☑ **FORCE ALONG THE Y-COORDINATE OF A POINT:** F_Y
- 2D-CAM: you can plot the **CAM CONTACT FORCE**
- SPRING FB: you can plot the **TOTAL FORCE, X FORCE, Y FORCE** acting on the Anchor-Point.

Why are the Y–axis values in the DRO❸ different to the vector-values in the main graphic-area?

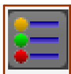
The values may be slightly different because, in the graph, the X–axis and Y–axis values at the D.R.O. must jump from 'machine -step' to 'machine-step'. See Machine Settings dialog > Number-of-Steps [291].

In the graphic-area, the vector values are calculated for the value of the Master Machine Angle, which may not coincide with a machine step.

1.10.30 > Graph Settings

Graph FB > Settings

How to open the Graph-Settings

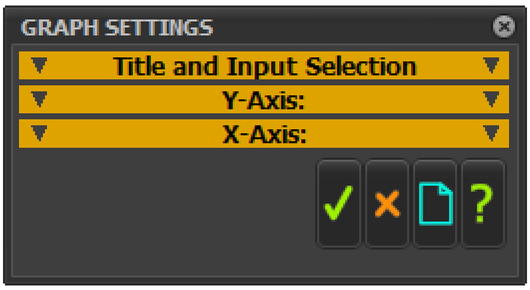


1. Click Graph toolbar > Graph Settings icon [387] in the Graph toolbar to open the GRAPH SETTINGS FORM..

The settings are:

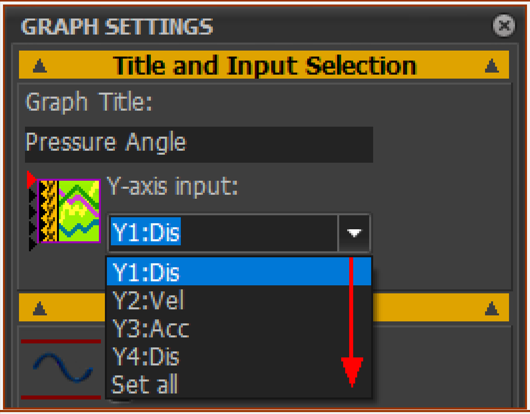
- Graph Name
- Color of each Graph Line
- Maximum and Minimum scale for each **Y-axis**,
- Maximum and Minimum scale for the **X-axis**

Graph-Settings dialog



Graph-Settings dialog

TITLE AND Y-AXIS INPUT SELECTION



GRAPH TITLE

The **GRAPH TITLE** - for example, **Pressure Angle,** shows above the graph plots.

Y–AXIS INPUT

Click the drop-down to select for which **Y-AXIS INPUT** you want to edit

You can select:

- **Y1** to **Y4**

or

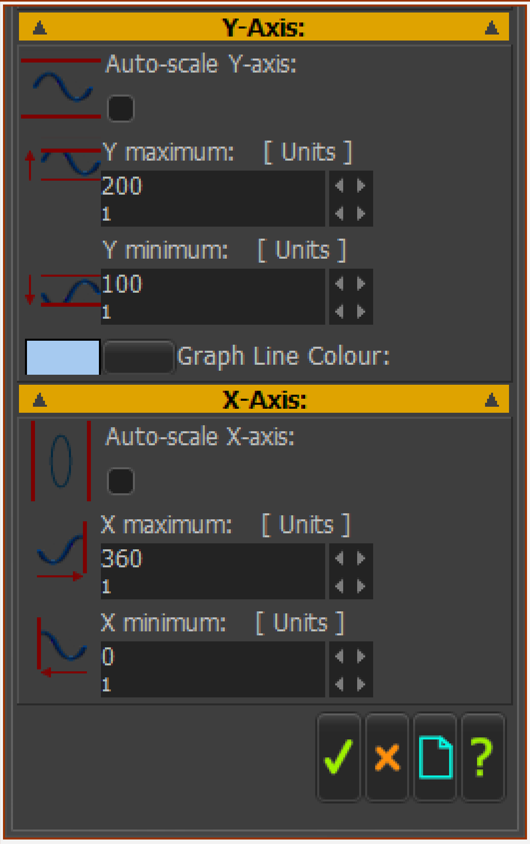
- **SET ALL** : to apply the same **Y-AXIS** and **X-AXIS** settings to **ALL** Y-axis inputs.

Top Tip 1:

As an example, use **SET ALL** if the wires you connect to the input-connectors are **ALL PRESSURE-ANGLES** from different **CAM-DATA FBS**. Then **ALL** **Minimum** and **Maximum** values of the graph-plots are equal for all of the **2D-CAMS**. It is then easier to analyze the **PRESSURE ANGLES** of up to 4 2D-Cams at the same time, and easily see their values.

Top-Tip 2: Use the **Y Maximum** and **Y-Minimum** settings for **RADIUS-OF-CURVATURE.**

Y-AXIS & X-AXIS



The Y-axis parameters apply one the **Y-AXIS INPUT** or **ALL** **Y-AXIS INPUTS-** **see** **Title and Input Selection** 389

Y-AXIS:

AUTO-SCALE Y-AXIS: **check-box**

- ☑ **Auto-scale Y-minimum / Y-minimum** values for the **Y-AXIS INPUT**
- ❑ **Enter the Y-maximum and Y-minimum** values for the Y-axis input

Y MAXIMUM: / Y MINIMUM: Edit when the **AUTO-SCALE Y-AXIS** check box is **Clear**

GRAPH LINE COLOR: click the **GRAPH LINE COLOR** button

Use the Windows® pop-up color picker, to select a different color.

X–AXIS:

The **X-axis values** apply to **ALL** graph plots.

AUTO-SCALE X-AXIS:

- ☑ Auto-scale the **X-MAXIMUM** and **X-MINIMUM** values for the **X-AXIS**.
- ❑ Enter the **X-MAXIMUM** and **X-MINIMUM** values for the **X-AXIS**

1.10.31 Dialog: Function-Block: Point-Data

Point-Data FB

see **Add Point-Data FB** [169]

The **POINT FB** measures motion-values of a **POINT** or **MOTION-POINT** with respect to the **MECHANISM PLANE**.

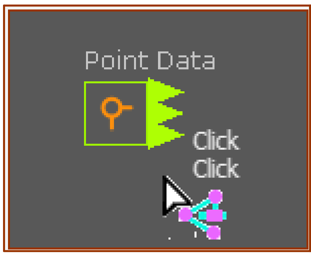
It has three output-connectors. From top to bottom, the data at the output-connectors are:

- ➔ motion-values parallel to the X-axis
- ➔ motion-values parallel to the Y-axis
- ➔ Magnitude of the motion-values equal to:
  - $\sqrt{(X^2 + Y^2)}$

Note: ***Motion-Values*** include the **Position**, **Velocity**, and **Acceleration** of the **POINT** or **MOTION-POINT**.

The **POINT** must be a child to a **PART** that is **kinematically-defined**.

How to open the Point-Data FB dialog



Edit the **POINT-DATA FB**:

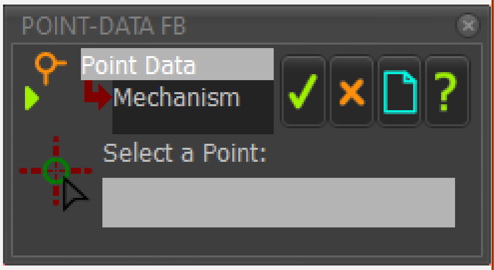
1. Double-click the **POINT-DATA FB** in the graphic-area

OR

1. **See How to Open a dialog** [513]

The **POINT-DATA DIALOG** is now open.

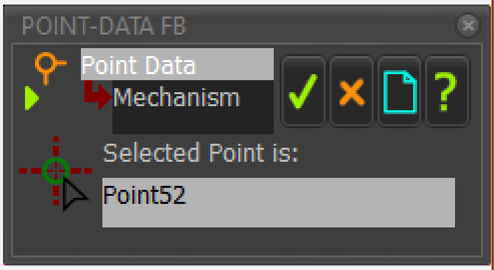
Point-Data dialog



Point-Data dialog; without Point

SELECT A POINT:

1. Click a **POINT** in the graphic-area



Point-Data dialog; with Point

When the correct **POINT** is in the selection-box.

2. Click ✓ to close the **POINT-DATA DIALOG**

To use the Point-Data FB

How to Use :	Connect a wire from its output-connector to the input-connector of a **GRAPH FB** [172] - plot the Position, Velocity and Acceleration of the **POINT** in the X and Y–axis directions - ***see also: Top-Tip***

	Connect a wire from its output-connector to the input-connector of a **MOTION FB** - use the motion of the **POINT** as the independent (X-axis) for a **Motion**. Connect a wire from its output-connector to the input-connector of a **MOTION-DIMENSION FB** or a **MOTION-PATH FB** - the Motion-Values moves the **MOTION-PART** or **MOTION-POINT** - it is best to review the data-values with a Graph first.
Top-Tip :	To plot Position, Velocity, **AND** Acceleration of the same **POINT** in one **GRAPH FB**: **1.** Drag ***three wires*** from one output-connector of the **POINT-DATA FB** to ***three different input-connectors*** of a **GRAPH FB** **2.** Use the Y–axis display options 388 to select each motion-derivatives.
See also :	**Connecting FBs** 144

1.10.32 Dialog: Function-Block: CAD Control

CAD Control FB

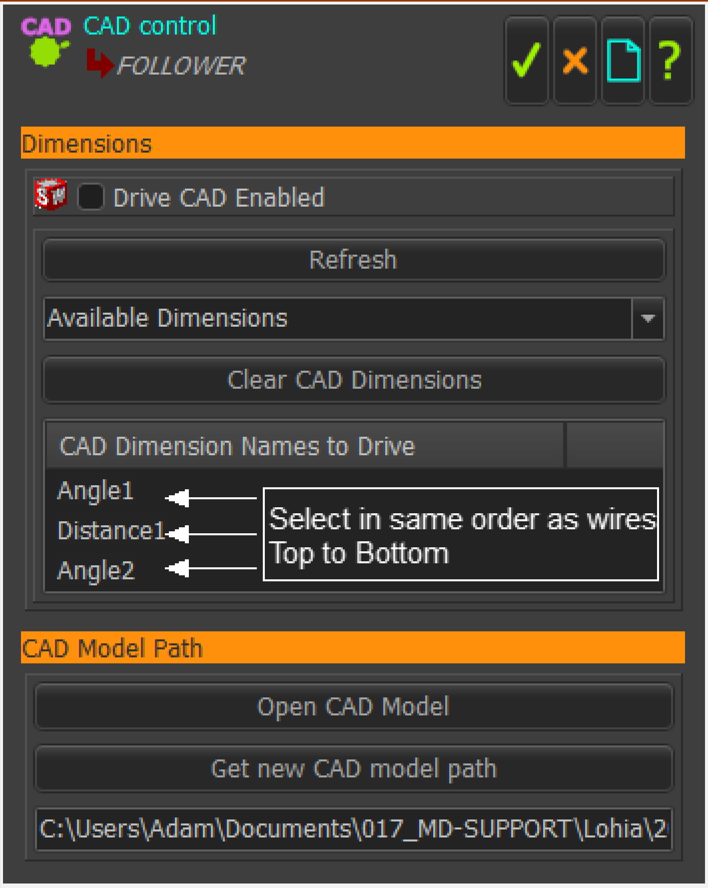
see Add CAD-Control FB 186.

Use a **CAD CONTROL FB** to synchronize **Distance** or **Angle** mates in a SOLIDWORKS® Assembly model with the motion of **PARTS** in a **MechDesigner** model.

You can control up to four(4) SOLIDWORKS® mates with one **CAD-CONTROL FB**.

Why use a **CAD-CONTROL FB**? You can use the powerful Evaluation / Validation Tools in SOLIDWORKS® to check clearance on Cams, for example.

See Getting Started Tutorial 6C3: Examine 3D-Cam Clearances



CAD-Control dialog

STEP 1: Connect wires to the input-connectors of the CAD-Control FB from other FBs in your model.

The motion-values at the output of a FB you connect should control a **Motion-Dimension** and be equivalent to a **Distance** or **Angle** Mate in SOLIDWORKS that you want to control.

If you want to control a Distance (or Angle) Mate, then make sure Output Data Type in the Motion FB dialog(s) 374 are set to Linear (or Rotary).

STEP 2: Open the SOLIDWORKS Assembly.

STEP 3: Double-Click a **CAD-CONTROL FB** in the graphic-area, to open this dialog

If necessary, click **DIMENSION** and **CAD MODEL PATH** to expand the dialog.

STEP 4: *Click the* **Refresh** *button.*

When you click the **Refresh** button, We find for you all of the **Dimension** and **Angle** mates in your SOLIDWORKS® Assembly model.

STEP 5: Click the **Available Dimensions** down-arrow button to see the mates that have been found in the SOLIDWORKS® Assembly model.

STEP 6: Click, in the **Available Dimension** list, each **Dimension** or **Angle** mate you want to control.

Repeat to select all of the mates you want to control - to a maximum of 4.

Note: You **must** select the mates in the same order (sequence) as the wires that are connected to the input-connectors of the FB.

The mates show in the **CAD-Dimension Names to Drive** box.

If the mates are not in the same sequence as the wires you can click the **Clear CAD Dimensions** button.

You can select the mates again.

STEP 7: Click the Drive CAD Enabled check-box at the top of the dialog.

DO NOT CLOSE THE DIALOG **- YOU CAN ONLY CONTROL THE MATES WITH THIS DIALOG OPEN.**

The position of the Parts in the SOLIDWORKS® Assembly model move* when you move the your model in **MechDesigner**.

STEP 8: Click ✅ at the top of the dialog when you have finished using the evaluation and analysis tools in SOLIDWORKS®.

* Only use the **Home, Step Back** or **Step-Forward** buttons in the Run toolbar. **Do not use the Cycle button.**

1.10.30 Dialog: Function-Block: Math (with Calculator)

Math FB (with Calculator)

See Add Math FB 183 **See also :** *Tutorial 18: Math FB*

Use a **MATH FB** to add new math functions.

Use a **MATH FB** to change the Data-Type and units . For example, change Motion units to Force units, and connect a **MATH FB** to a Spring FB 194 .

A **CALCULATOR** interface can help you select different functions.

How to open the Math FB dialog

To open the **MATH FB**:

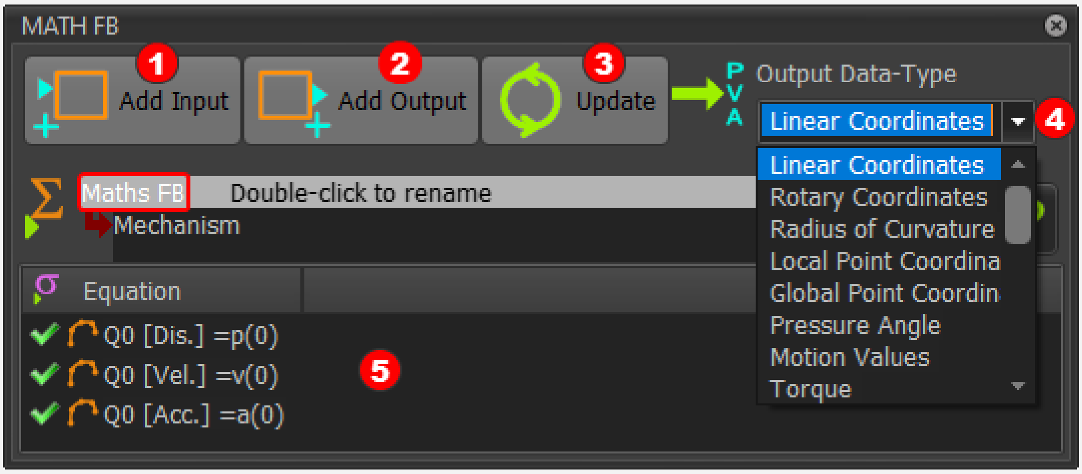
1. Double-click the **MATH FB** in the graphic-area

OR

1. **See How to Open a dialog** 513

The **MATH FB DIALOG** is now open.

Math FB dialog



Math FB dialog

Math FB Interface

ADD INPUT button❶

1. Click the Add Input button to add an input-connectors to the **MATH FB**.

   You need one input-connector for each variable or parameter in your equation.

ADD OUTPUT button❷

1. Click the Add Output button to add an output-connector from the **MATH FB**.

   You need one output-connector for each output.

UPDATE button❸

1. Click the Update button **when you:**
   - Change the **OUTPUT DATA-TYPE**

- Edit an equation

OUTPUT DATA-TYPE❹

The OUTPUT DATA-TYPE applies to ALL of **output-connectors** from the MATH FB.

To select a different OUTPUT DATA-TYPE

1. Click the drop-down box❹
2. Select the OUTPUT DATA-TYPE

Notes: Inside the MATH FB, all units are **SI units**.

A change to the DATA-TYPE may change the value at the output-connector of the MATH FB.

Equations❺

There are three(3) EQUATIONS for each **output-connector**

See below for more explanations.

Output Data-Type. Output-Connectors, Data-Channels

The OUTPUT DATA-TYPE defines the base units for **ALL** of the equations and results of the equations at the output-connectors.

Each **output-connector** has **three(3) Data-Channels**.

You can enter one(1) equation for each **Data-Channel.**

Therefore, you can enter three(3) equations for each **output-connector**

For example:

If you select LINEAR COORDINATES as the OUTPUT DATA-TYPE then ALL equations use **Linear Units**

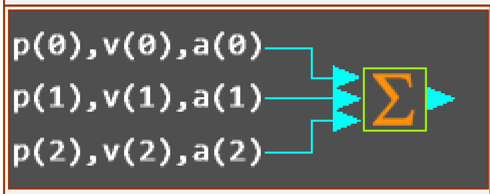
If you have two(2) **output-connectors**, there are 6 equations.

- The units for **Data-Channels** 0 & 3 are Dis, with units of m
- The units for **Data-Channels** 1 & 4 are 'Vel', with units of $m.s^{-1}$
- The units for **Data-Channels** 2 & 5 are 'Acc', with units of $m.s^{-2}$
- Data-Channels/Equations 0, 1, 2 are for output-connector 1
- Data-Channels/Equations 3, 4, 5 are for output-connector 2

Wire Numbers / input-connectors and Data-Channels

The image below shows three(3) **Wires** connected to three(3) **Input-Connectors**.

Each **Wire** has three **Data-Channels**.



3 Wires, 3 Input-Connectors
Each wire has 3 Data-Channels

WIRE-NUMBERS - refer to the image above.

Wire numbers start at 0

- **Input-Connector 1, Wire 0 - connected to the TOP input-connector**
- **Input-Connector 2, Wire 1**
- **Input-Connector 3, Wire 2 - connected to the BOTTOM input-connector**

Each wire has 3(three) data-channels.

Each data-channel is designated with a letter:

- *p = Data-Channel 1*
- *v = Data-Channel 2*
- *a = Data-Channel 3*

The format to reference a Wire-Number and Data-Channel in an equation is

DATA-CHANNEL (WIRE-NUMBER)

EXAMPLE - Entries in an Equation:

p(0) : **p** refers to **Data-channel 1** ; and **(0)** refers to **input-connector 1**

a(2) : **a** refers to **Data-channel 3** ; and **(2)** refers to **input-connector 3**

v(1) : **v** refers to **Data-channel 2** ; and **(1)** refers to **input-connector 2**

HOW MANY INPUT AND OUTPUT-CONNECTORS?

As an example, to calculate **Power** :

$$P = \tau . \omega$$

Power (N.m./sec) = Torque(N.m.) × Angular Velocity (rad/sec)

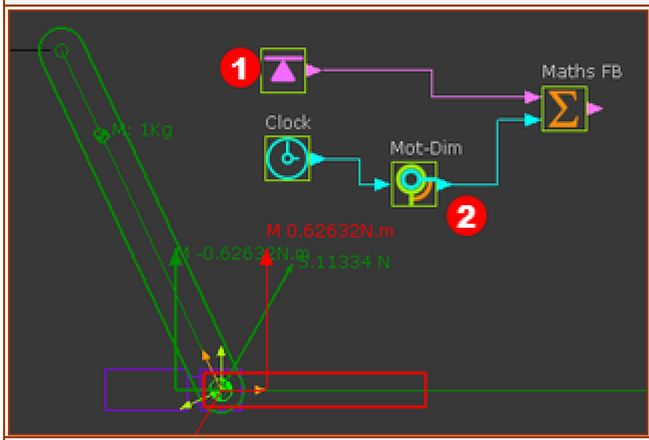
Torque data and **Angular-Velocity data** are inputs to the equation.

Power is the output.

You need **two input-connectors** and **one output-connector**.

Edit the MATH FB to open the MATH FB DIALOG.

1. Add two Input-Connectors with the **ADD-INPUT** button
2. Select **Power** for the **OUTPUT DATA-TYPE**.
3. Click **UPDATE** button



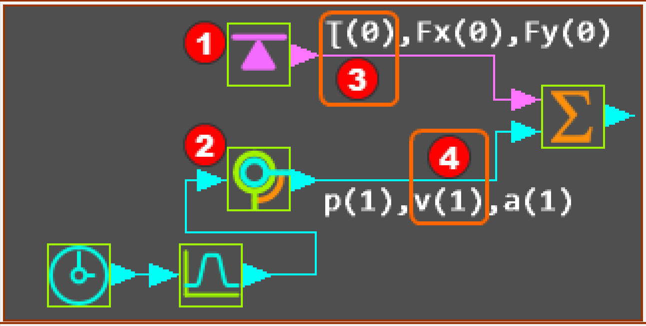
Prepare the model (see above)

1. Add a ROCKER **- see Tutorial 1**
2. Edit the CAD-LINE of the ROCKER, to open the CAD-LINE DIALOG.

In **CAD-Line dialog** > Mass Properties tab [304] > USER MASS PROPERTIES .
Enter a value for the **Mass.**

Enter X and Y values to move the **Center-of-Mass** away from the START-POINT (origin) of the CAD-LINE.

3. Add a FORCE-DATA FB, open the FORCE-DATA FB DIALOG, select the PIN-JOINT, and select the POINT in the BASE-PART to find the **Torque** that acts-on the BASE-PART.



See image above - the DATA-CHANNELS on the WIRE of the FORCE-DATA FB❸ and the MOTION-DIMENSION FB❹

The MATH FB uses SI units for all data 'inside'.

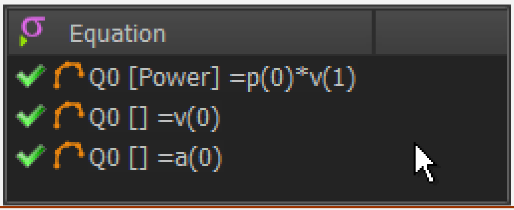
Enter the Equation

Click the first equation in the MATH FB

The **EQUATION-EDITOR** opens - see below.

You need to enter this equation

Q0 [Power] = p(0)*v(1)



p(0) = Wire-Number 0 (top input-connector), Channel 1 (p = Torque)

×

v(1) = Wire-Number 1 (input-connector below top), Channel 2 (v = Angular Velocity)

SYNTAX and VALID Equations

Click the UPDATE after you edit the equation to find out if the syntax is correct

For example, it may not have correct number of parentheses. The reason is given as a message in the Feedback-Area [270].

The ✅ at the left of each equation shows if its syntax is correct.

IMPORTANT

Data from Motion FBs

If you connect a wire from a MOTION FB to the input of a MATH FB, Set the OUTPUT DATA-TYPE in the **MOTION FB** to LINEAR or ROTARY.

MORE ABOUT UNITS 'inside' the Math FB

There are 4 possible sources of confusion with respect to 'units' when you use the MATH FB.

SI UNITS

Inside the MATH FB, the units are **SI.**

Example:

- If the MOTION FB is set to ***Rotary***, and the output from it is 90º, then the value internal to the MATH FB is 1.57radians.

- If the **MOTION FB** is set to ***Linear***, and the output from it is 100mm, the value internal to the **MATH FB** is 0.1m.

If you multiply these together with the **MATH FB**, the internal result is 0.157 - of course.

SI UNITS ARE CONVERTED TO ENGINEERING UNITS

The **SI units** and values that are internal to the **MATH FB** are converted back to Engineering Units [291] of the **OUTPUT DATA-TYPE**.

Example continued...

- If the **OUTPUT DATA-TYPE** is **LINEAR COORDINATES**, then **0.157** becomes **157mm** at the output-connector when the Engineering Units in the **MACHINE SETTINGS DIALOG** are **mm.**
- If the **OUTPUT DATA-TYPE** is **ROTARY COORDINATES**, then **0.157** becomes **9º** at the output-connector when the Engineering Units in the **MACHINE SETTINGS DIALOG** are **degrees**.

  The reason? **0.157rads (inside the Math FB)** = **9º (outside the Math FB).**

Linear-Coordinates to a Rocker, Rotary-Coordinates to a Slider

Example continued...

Rotary units connected to a Linear Slider - that is probably not intended

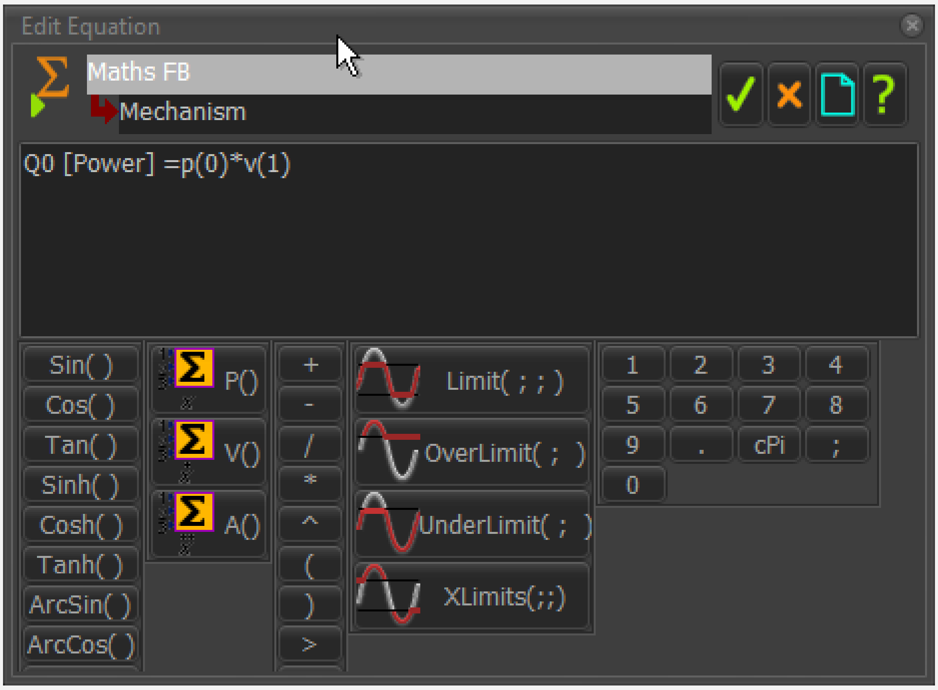
In the **MATH FB**, if you set the **OUTPUT DATA TYPE** to **ROTARY COORDINATES**, then **0.157** becomes **0.157rads** internally. This becomes **9º** at the output, when the Engineering-Units are **degrees**.

If you connect the **MATH FB** to a **MOTION-DIMENSION FB** to move a **Slider**, then the *9º* becomes *9mm*.

Linear Units connected to a Rotary Rocker - that is probably not intended

In the **MATH FB**, if you set the **OUTPUT DATA TYPE** to **LINEAR COORDINATES**, then **0.157** is **0.157m** internally. This becomes **157mm** at the output when the Engineering-Units are **mm**.
If you connect the **MATH FB** to a **MOTION-DIMENSION FB** to move a **Rocker**, then the Rocker rotates **157º**.

Equation-Editor with Calculator



Equation-Editor with Calculator

The **EQUATION-EDITOR** provides a few of the common mathematical functions you can add to each equation.

Note 1:

When **open and close brackets** show within a function button, you must enter a constant value or an variable value at an input-connector.

Example 1: Sin( ),

You can enter a constant - **Sin(3.14)**

You can also enter a variable - **Sin(p(1))** - to calculate the Sine of the continuously varying position values of the second input-connector and wire.

Note 2:

- **p(0)** is data on data-channel 1, at the top, or input-connector **#1,**
- **p(1)** is data on data-channel 1, at input-connector **#2**,
- **p(2)** is data on data-channel 1, at input-connector **#3**, ...

Note 3:

ALL data 'inside' the **MATH FB** are SI units.

Calculator Functions

1. **Position, Velocity, and Acceleration Values for an linear or angular input**
   - P( ) ; V( ) ; A( )
2. **Standard Algebra**
   - Arithmetic: +, –, /, *
   - Power: ^
   - Brackets: ( )
3. **Boolean**
   - **>** : <output> =1 if <value 1> **>** <value 2>, else 0

- **<** : <output> =1 if <value 1> **>** <value 2>, else 0

4. Trigonometry (all inputs in radians)
   - Standard: Sin( ), Cosine( ), Tangent( )
   - Hyperbolic: Sinh( ), Cosh( ), Tansh( )
   - Inverse: ArcSin( ), ArcCos( ) , ArcTan2(;)
5. Limit (input ; maximum value ; minimum value) :

   <output> = <input> if <input> is less than maximum value AND more than minimum value.

   <output> = <maximum-value> if <input> is greater than maximum

   <output> = <minimum-value> if <input> is less than minimum
6. OverLimit (input ; limit-value) :

   <output> = <input> if <input> is 'greater than limit-value'

   <output> = <limit-value> if <input> is 'less than limit-value'
7. UnderLimit (input ; limit value) :

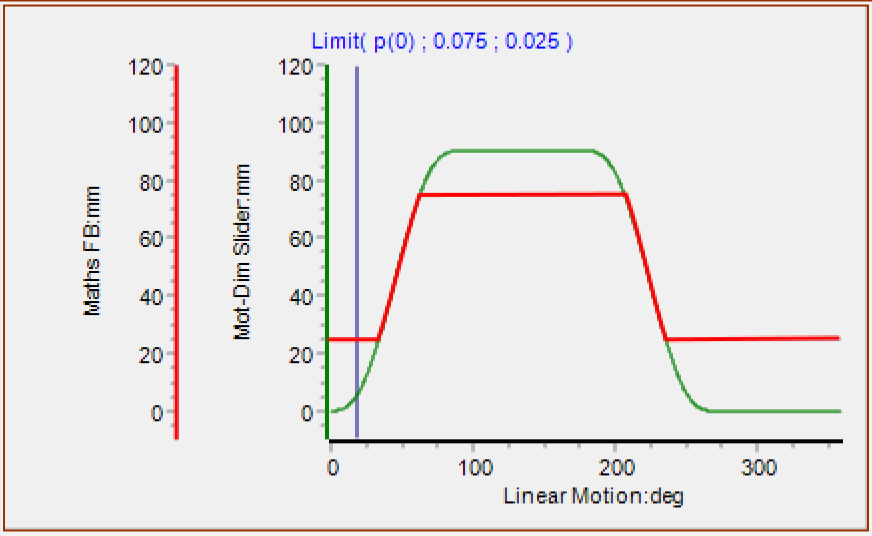
   <output> = <input> if <input> is 'less than limit-value'

   <output> = <limit-value> if <input> is 'greater than limit-value'
8. XLimits (input ; maximum value ; minimum value) :

   <output> = <input> if <input> is 'greater than maximum value' OR less than minimum value'.

   <output> = <maximum-value> if <input> is 'less than maximum' AND 'more than minimum-value

   <output> = <minimum-value> if <input> is 'greater than minimum-value AND 'less than maximum-value'
9. Numerical Keypad
10. 1...9, 0, Π.
11. Abs( )
12. Mag(;) (assumes the two values are 'orthogonal' and uses Pythagoras to find the 'resultant')
13. Sqrt( );
14. DegToRad( )
15. RadToDeg( )



Limit Function

Limit (input ; maximum value ; minimum value) :

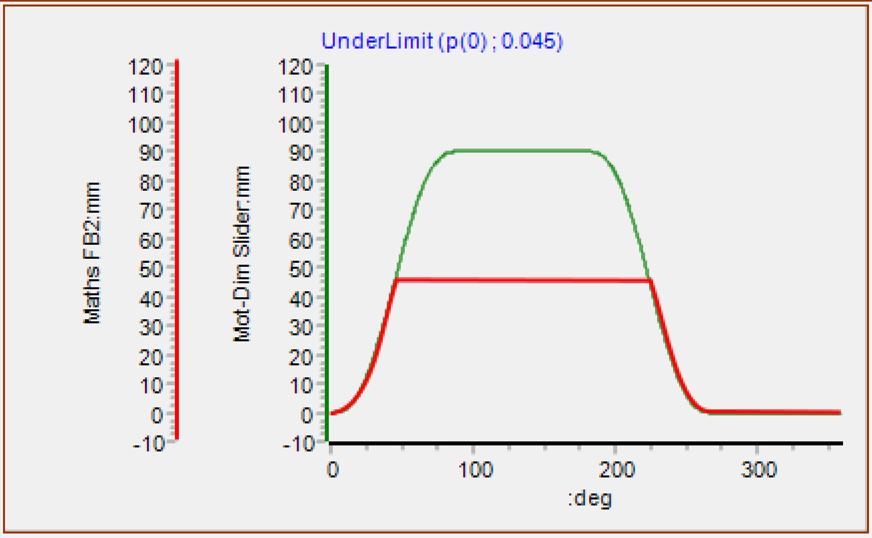
<output> = <input> if <input> is less than maximum value AND greater than minimum value.

<output> = <maximum-value> if <input> is greater than maximum

<output> = <minimum-value> if <input> is greater than minimum

Note that the arguments in the Limit Function are SI units for the units in the model.

For example, 75 millimeters in the model is 0.075 meters in the function.

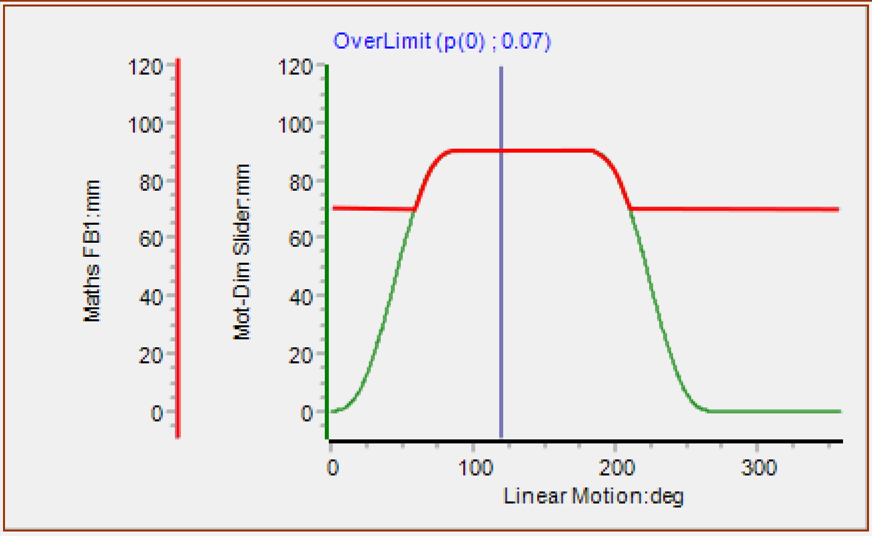


Under Limit Function

UnderLimit (input ; limit value) :

<output> = <input> if <input> is 'less than limit-value'

<output> = <limit-value> if <input> is 'greater than limit-value'



Over Limit Function

OverLimit (input ; limit-value) :

<output> = <input> if <input> is 'greater than limit-value'

<output> = <limit-value> if <input> is 'less than limit-value'

1.10.30 Dialog: Function-Block: Statistics

Statistics FB

See **Add Statistics FB** [179]

If you can connect a wire to a GRAPH FB to plot its data over a complete machine-cycle ...

... then you can connect a wire to a STATISTICS FB to find statistical data for the complete machine-cycle.

Compare with **Element Properties dialog** [498], which displays the data-values at the MASTER MACHINE ANGLE only.

There are two steps you need to do to get Statistics for a Data-Channel on a wire.

STEP 0: Connect a wire to the input of the Statistics FB.

A wire that you connect from the output-connector of a FB to the input-connector of the STATISTICS FB has three Data-Channels.

For example, the three Data-Channels at the output-connector of a MOTION-DIMENSION FB are the three motion-derivatives of the MOTION-DIMENSION

STEP 1: Select the Data-Channel

STEP 1. Open the Statistics dialog

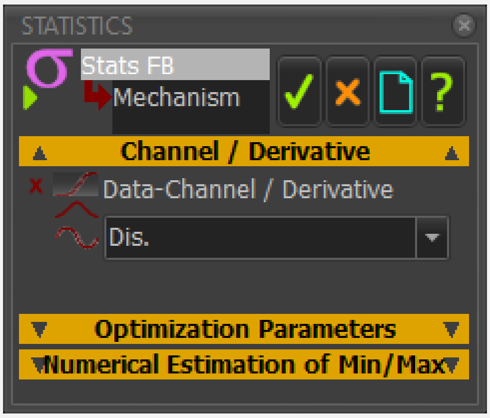
Edit the STATISTICS FB:

1. Double-click the STATISTICS FB in the graphic-area

OR

1. **See How to Open a dialog** [513]

The STATISTICS FB DIALOG is now open.



Statistics FB dialog

CHANNEL / DERIVATIVE

STEP 2. Click the Derivative drop-down and select the derivative for which you want Statistics

Derivative	≡ Data-Channel
Dis.	Linear Displacement or Angular Displacement; or Torque
Vel.	Linear Velocity or Angular Velocity; or Force X
Acc.	Linear Acceleration or Angular Acceleration; or Force Y

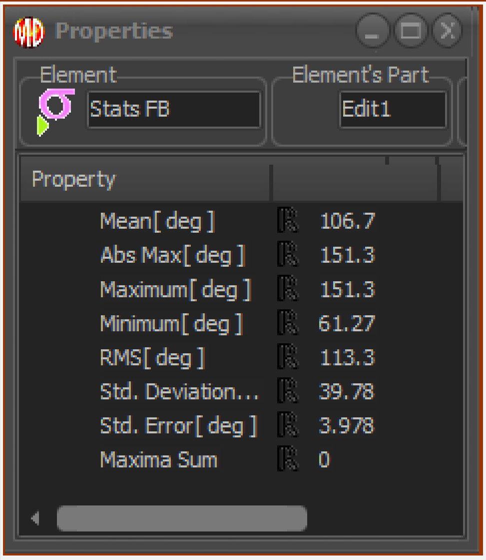
* The **Data-Channels** in the drop-down box are **always**: DIS, VEL, and ACC , even if the actual units are different. The actual units:

- For a KINEMATIC-FB are **three(3) motion-derivatives**
- For a FORCE-DATA FB are **Total Force/Torque, Force X, Force Y.**

Ignore the other SEPARATORS in the dialog.

STEP 3. Click ✓ to close the STATISTICS DIALOG

STEP 2: Show the Statistics Properties



Statistic Properties dialog

STEP 1. Open the Properties box

1. CTRL + Click the STATISTICS FB in the graphic-area

The PROPERTIES **box is now open.**

Read the **Statistical Properties** for the **Data-Channel** of the wire that is connected to its input-connector.

Mean

Ans Max

Maximum

Minimum

RMS

Standard Deviation

Standard Error

Maximum Sum

Note: to edit the NUMBER-FORMAT for the **Statistics Properties**, use:

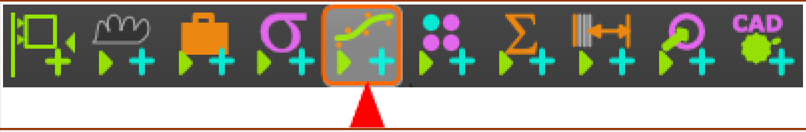
APPLICATION-SETTINGS > NUMBER-FORMAT > DATA-OUTPUT FORMAT > DIGITS and PRECISION

1.10.35 Dialog: Function-Block: Polynomial-Fit

Polynomial-Fit FB

See also: **Add Polynomial Fit FB** [180]

About Polynomial FB

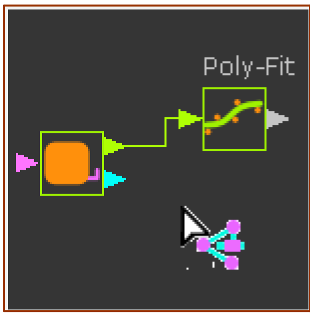


The **POLYNOMIAL FIT FB** finds a concatenated series of Polynomials (5th order) that are approximately equal to the data at its input-connector.

After you do a Polynomial-Fit, you can:

- Transfer Polynomials directly to ***MotionDesigner*** as a new **Motion name-tab**.
- Connect a wire from the output-connector of the **POLYNOMIAL-FIT FB** to the input-connector of a different **FB**.
- Export the motion-values that you calculate as a text-file.
- Export the values for position, velocity, and acceleration at the beginning and end of each polynomial segment as a CSV file.
- Export the polynomials segments in a format suitable for the Schneider-Electric® EPAS4 servo-controller.

How to open the Polynomial-Fit FB dialog



Edit the **POLYNOMIAL-FIT FB**:

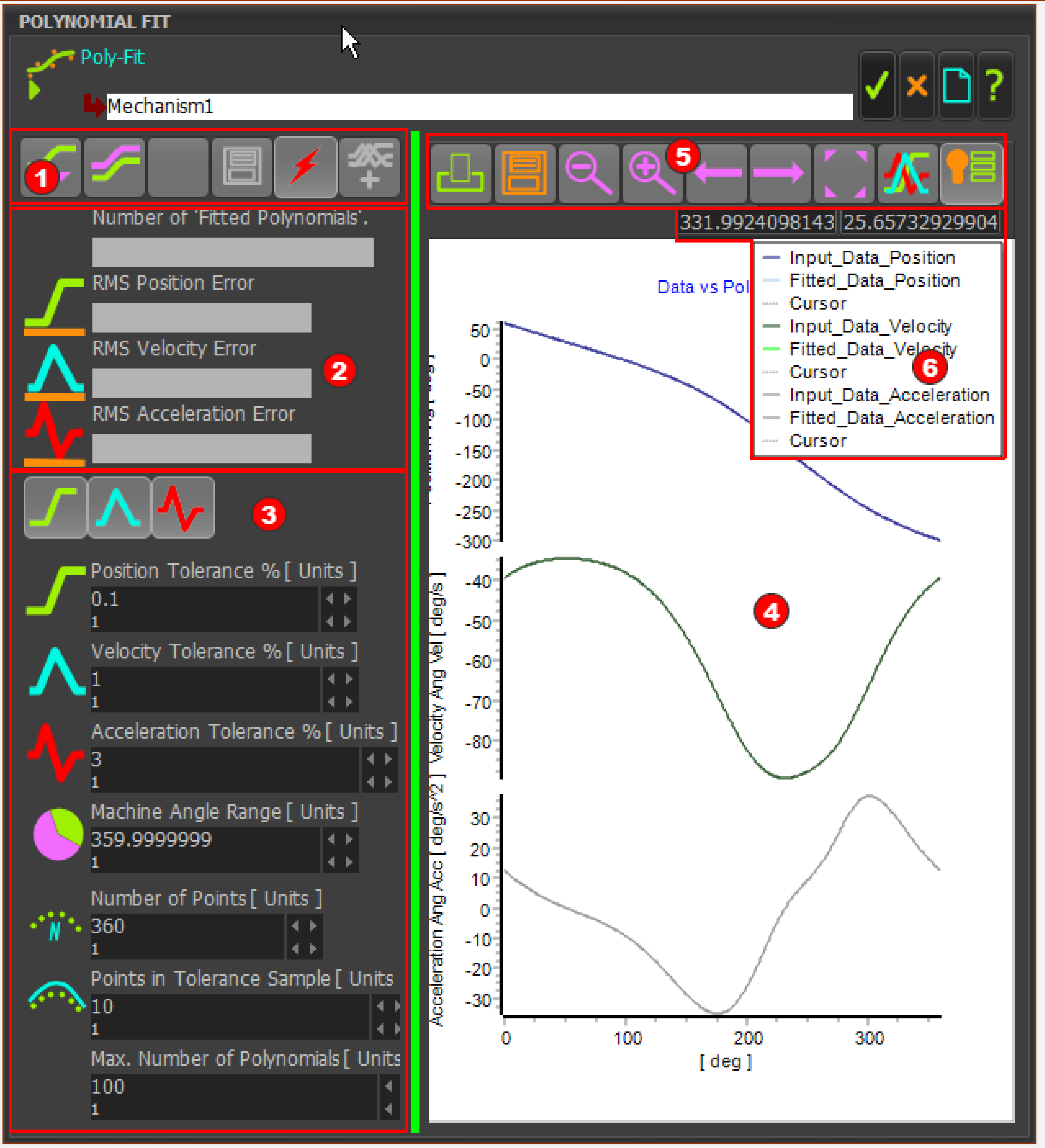
1. Double-click the **POLYNOMIAL-FIT FB** in the graphic-area

OR

1. **See How to Open a dialog** [513]

The **POLYNOMIAL-FIT DIALOG** is now open.

Polynomial-Fit dialog



Polynomial-Fit FB dialog/interface.

The Polynomial-Fit dialog:

❶ **Acquire and Save Toolbar** - acquire and save results.

❷ **Analysis of Results** - RMS errors and the number of Polynomials.

❸ **Settings for Polynomial Fitting Algorithms** - tolerances, limits, and range for polynomial fitting.

❹ **Graph Plots**: for the input-data and the calculated data from the Polynomials.

❺ **Graph toolbar** - view and save the graph-data in different ways.

❻ **Graph Legend**. - graph plot color scheme for original data and fitted Polynomials, for each motion-derivative

1 Acquire and Save toolbar

Acquire and Save Results toolbar

The toolbar buttons are:

1 **Acquire** (Capture, Get One Cycle) **:**

Click this button to acquire data at the input-connector for one machine-cycle - this is the ***Raw-Data***.

2 **Fit Polynomials to Data** :

Click to **Fit Polynomials** to the **Raw-Data**. - see Settings for Polynomial Fitting Algorithms 408.

3 **Stop Algorithms : (see also Auto-Run)**

The algorithms usually fit polynomials within a few seconds. If you do not see the results after a few seconds, then click this button to stop the algorithms.

4 **Save Coefficients** :

Click to save the results to a CSV file, or a EPAS4 file that is suitable for a Schneider Electric servo-controller.

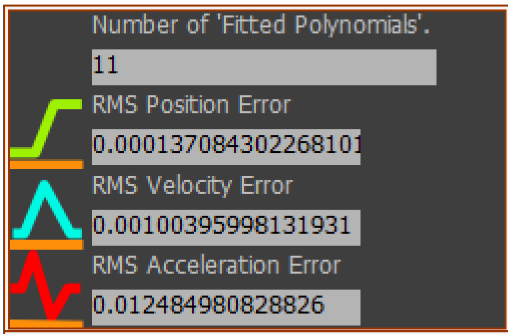
5 **Auto-Run** :

Immediately allow the algorithms to find polynomials fit the data to the accuracy as specified, and with less or equal to the maximum number of polynomials.

6 **Send Polynomials to MotionDesigner** :

Add a new Motion name-tab, and send the Polynomials as a new motion.

2 2 Analysis of Results



Number of Fitted Polynomials in Fit (read-only)

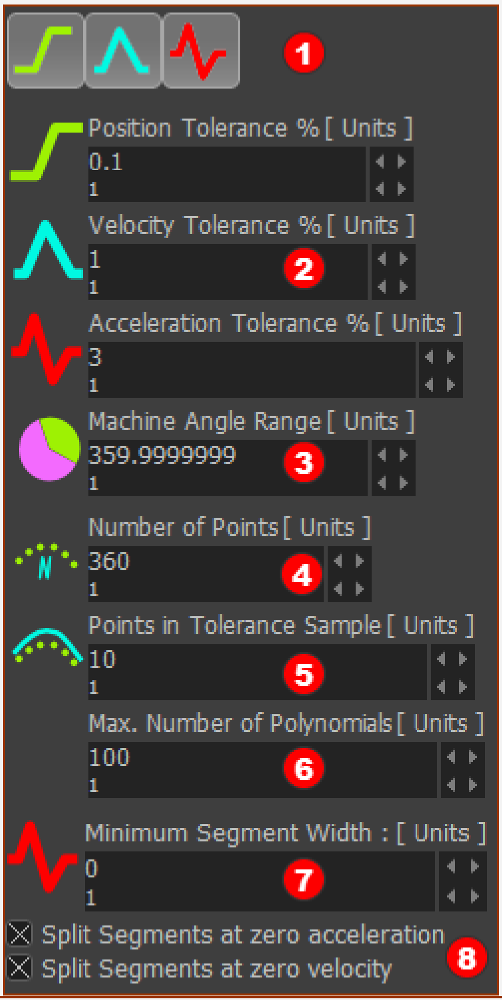
The number of Polynomials in the numerical solution.

See also: **Maximum Polynomial Count**. in the **Settings for Polynomial Fitting Algorithms.** 408

RMS Position Error, RMS Velocity Error, RMS Acceleration Error (read-only).

They give the RMS (Root, Mean, Square) error for each motion-derivative between the Polynomial-Fit and the data at the input-connector as a percentage.

3 Settings for Polynomial Fitting Algorithms



Parameters to enter before you click **Fit Polynomials to Data** (button ❷ in the Acquire and Save toolbar [408], above)

❶ **Position, Velocity, Acceleration buttons.**

Click each button to identify the motion-derivatives to which you want to **Fit Polynomials**.

❷ **Position, Velocity, Acceleration Tolerance %**

Enter the tolerance, as a %, for each motion-derivative, within which the **Fitted Polynomials** should represent the data at the input-connector.

❸ **Machine Angle Range**

Optionally, reduce the range for which you want to calculate Polynomials

❹ **Number of Points**

The number of data points that are calculated from the Polynomials, to plot the graphs.

❺ **Point in Tolerance Sample of Points**

The number of points that the algorithm uses to fit each polynomial to the data at the input-connector.

❻ **Maximum number of Polynomials.**

Upper limit of the number of **Fitted-Polynomials** in the data.

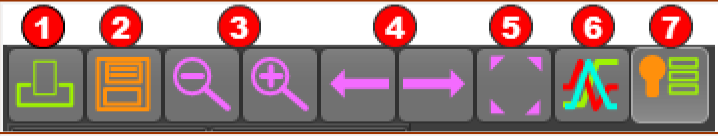
❼ **Minimum Segment Width.**

Lower Limit for the width of a Segment

❽ **Split Segment at Zero Acceleration / Velocity check-boxes**

Force the solutions to split the motion (add a new Segment) at which the Acceleration / Velocity crosses zero - positive to negative and vice versa.

5 Graph toolbar



The toolbar buttons are:

1 **Print Graphs** : Print the graphs immediately to the default printer

2 **Save Graph Data:** Save the Data to a text file.

3 **Zoom Out** and **Zoom In** to the Graph Details

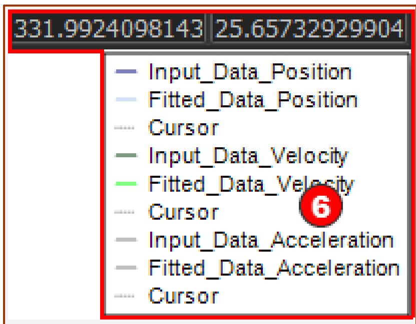
4 **Pan Left and Pan Right** : Click to save the results to a CSV file, or a EPAS4 file that is suitable for a Schneider-Electric servo-controller.

5 **Zoom Extents**

6 **Toggle display: Plot three separate graphs** OR **Plot three stacked on-top of each other**

7 **Toggle display: Do show Graph Legend** OR **Do NOT show Graph Legend**

6 Graph Legend



Graph plot color scheme for

- original, or input, data
- fitted data calculated from the fitted polynomials
- each motion-derivative
- Cursor - you can click ON a graph of each motion-derivative to show a cursor.

  The X-Y data-values of the active cursor is above the graphs.

1.10.30 Dialog: Function-Block: Parameter-Control

Parameter-Control FB

See: Add Parameter-Control FB (184)

Motion-values at the input-connector to a PARAMETER CONTROL FB can control the values of these parameters:

- DIMENSION when the **Driven Parameter** is a DIMENSION
- EXTRUSION DEPTH when the **Driven Parameter** is a PROFILE
- EXTRUSION-OFFSET when the **Driven Parameter** is an EXTRUSION.

Add one PARAMETER-CONTROL FB to control elements with a one Motion.

How to open the Parameter-Control dialog

Open the PARAMETER-CONTROL DIALOG

1. Click the PARAMETER-CONTROL FB in the graphic-area or the ASSEMBLY-TREE.

The PARAMETER-CONTROL FB is now in the SELECTION-WINDOW

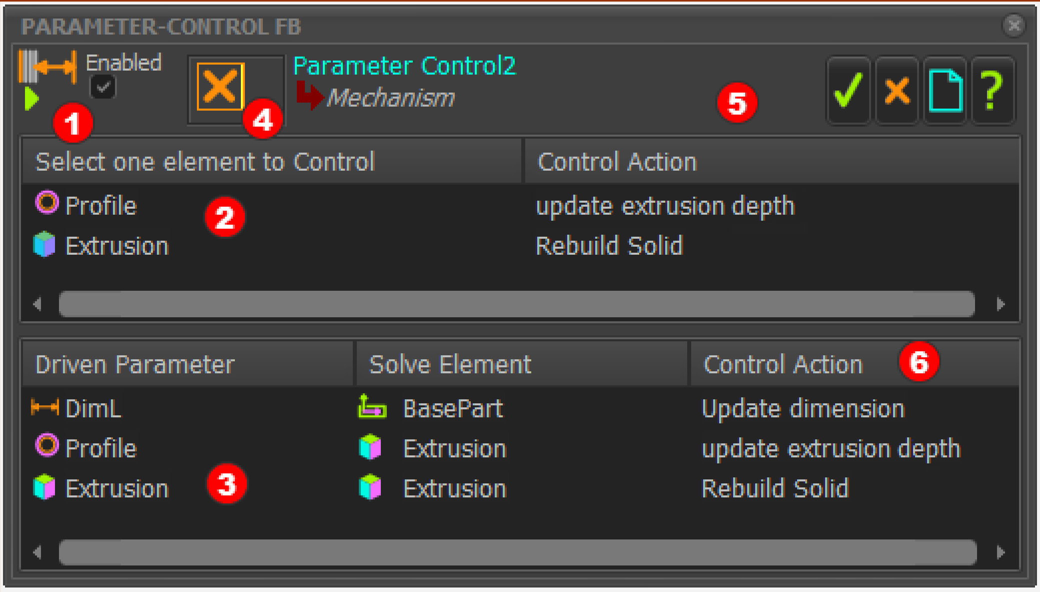
2. Right-click the PARAMETER-CONTROL FB element in the SELECTION-WINDOW
3. Click **Edit element** in the shortcut menu

OR

See How to Open a dialog (513).

The PARAMETER-CONTROL DIALOG is now open.

Parameter-Control dialog



Parameter-Control dialog

The controls in the PARAMETER CONTROL DIALOG are:

1 ENABLE / DISABLE check-box

2 TOP-BOX - PROFILES and EXTRUSIONS are here when you select them in the graphic-area.

You must then click which of the elements to control in the **Top-Box** to put it in the **Bottom-Box**

❸ **BOTTOM-BOX** - Parameters and Control Action that are updated by the motion-values at the input to the FB

❹ **DELETE** - to remove an element from the dialog - preselect the element.

❺ **OK**, **CANCEL**, **CANCEL** and **UNDO ALL CHANGES**, **HELP** icons.

❻ **CONTROL ACTIONS** - the action that the **PARAMETER-CONTROL FB** completes.

How to use Parameter-Control dialog

To enable the FB

☑ Enabled

1. Click the check-box to enable the dialog.

To control a Dimension

1. **MECHANISM-EDITOR**: Click the sketch-element that has a **DIMENSION** (which you have added with the **PART-EDITOR**)

OR

2. **PART-EDITOR**: Click a **DIMENSION**, or the sketch-element with a **DIMENSION**.

In the Bottom-Box

- The **DIMENSION** shows as the **Driven-Parameter**
- The **PART** shows as the **Solve-Element**
- **Update Dimension** shows as the **Control-Action**

To control the Extrusion-Depth of a Profile/Extrusion:

1. **MECHANISM-EDITOR**: Click a **PROFILE**.

The **PROFILE** and **EXTRUSION** show in the **Top-Box**.

In the **PARAMETER-CONTROL DIALOG :**

2. Click the **PROFILE** in the **Top-Box**.

In the Bottom-Box:

- The **PROFILE** shows as the **Driven-Parameter**
- The **Extrusion** shows as the **Solve-Element**
- **Update Extrusion-Depth** shows as the **Control-Action**

To control the Extrusion-Offset of a Profile/Extrusion:

1. **MECHANISM-EDITOR**: Click the **PROFILE** (the Pink contour of a sketch-loop).

The **PROFILE** and **EXTRUSION** show in the **Top-Box**.

In the **PARAMETER-CONTROL DIALOG:**

2. Click the **EXTRUSION** in the **Top-Box**.

In the Bottom-Box:

- The **EXTRUSION** shows as the **Driven-Parameter**
- The **Extrusion** shows as the **Solve-Element**
- **Rebuild Solid** shows as the **Control-Action** - this is actually the **EXTRUSION-OFFSET**.

To delete an element:

1. Click an element in one of the boxes.
2. Click ❹

The **ELEMENT** and **PARAMETER** are removed from the **PARAMETER-CONTROL DIALOG**.

1.10.30 Dialog: Function-Block: Pattern

Pattern FB

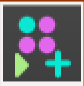
See Add Pattern FB [182]

If you use regular CAD, you know about circular and linear arrays. In **MechDesigner**, there is more complexity because PARTS have complex motions.

- A PATTERN FB makes **Copies** of PROFILE/EXTRUSIONS and CAD-SOLIDS - see SOLIDS,
- The PATTERN FB anticipates the motion of each **Copy** and puts each **Copy** at a different phase of the machine-cycle.
- The PATTERN FB can show or hide each **Copy** for the machine-cycle.
- The PATTERN FB can show or hide each **Copy** as it passes through different phases of the machine-cycle.

Use Visibility toolbar > Show Solids in Mechanisms [51] to see the **Copies** of the SOLIDS that are generated by the PATTERN FB.

How to open the Pattern FB dialog



Open the PATTERN FB DIALOG:

1. Click the PATTERN FB in the ASSEMBLY-TREE.

The PATTERN FB is now in the SELECTION-WINDOW

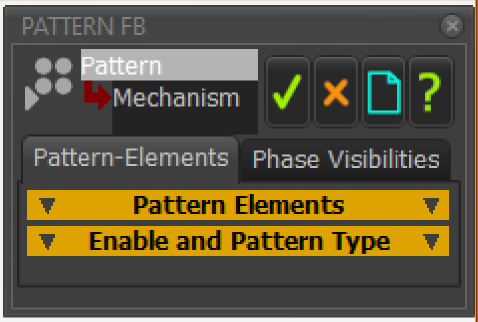
2. Right-click the PATTERN FB element in the SELECTION-WINDOW
3. Click **Edit element** in the shortcut menu

OR

1. **See How to Open a dialog** [513].

The PATTERN ELEMENTS DIALOG is now open.

Pattern dialog



Pattern FB dialog

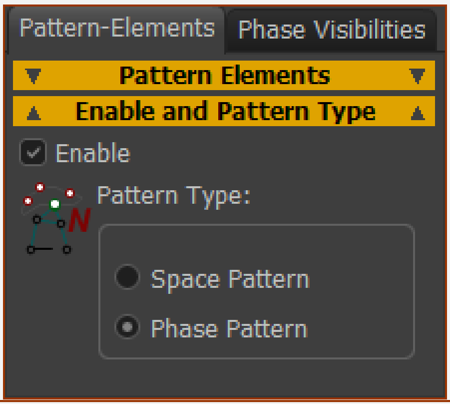
There are two tabs in the PATTERN DIALOG:

Pattern Elements tab

Phase Visibilities tab

Pattern Elements tab

ENABLE AND PATTERN TYPE



1. Click **ENABLE** check-box to make the **PATTERN** dialog active
2. Click **SPACE PATTERN** or **PHASE PATTERN**.

   These two options take a **snap-shot** of the path of a **SOLID**. The Pattern shows copies of the **SOLID** at equal increments of its path in the machine-cycle.

   ⊙ **SPACE PATTERN** (**Note:** I rarely use this option!)

   When you cycle the model:

   a) Each copy of the **SOLID** moves with the **PART**

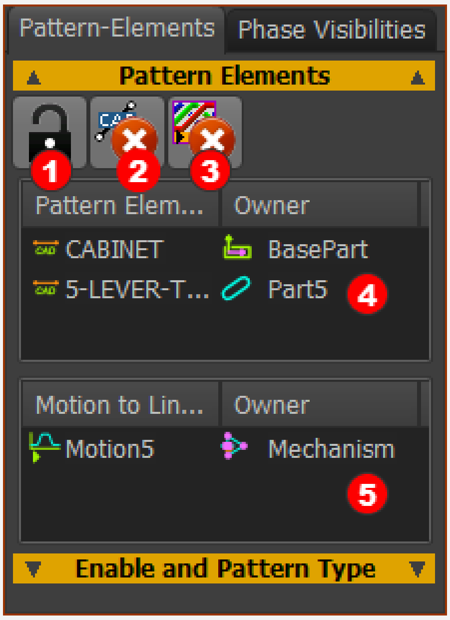
OR

   ⊙ **PHASE PATTERN**

   a) Each copy of the **SOLID** moves along the normal path of the **SOLID**.

   b) Each copy of the **SOLID** is at an advanced position in the machine-cycle.

PATTERN ELEMENTS



Pattern-Elements

PATTERN-ELEMENTS ❹

SOLID elements from which we make for you **copies** for the **Space Pattern** or **Phase Pattern**.

There are two categories of **SOLID** elements you can add as **PATTERN ELEMENTS**:

- **MD-SOLIDS** - **PROFILE/EXTRUSIONS** - you **MUST** select the **EXTRUSION**

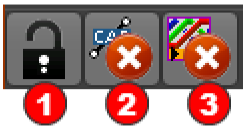
- **CAD-SOLIDS** - **SOLIDS** you import onto a **CAD-LINE** - you can select the **CAD-SOLID** or the **CAD-LINE**.

MOTION TO LINEARIZE❺

It is intended that a **MOTION FB** linearize the motion of the **SOLIDS** in a **PATTERN-ELEMENTS** ❹. EXPERIMENT!

- If the **SOLIDS** in the **PATTERN-ELEMENTS**❹ move as you expect, **do not add** a **MOTION FB**.
- If the **SOLIDS** in the **PATTERN-ELEMENTS**❹ do **not** move as you expect, **do add** a **MOTION FB** - but this may not solve the problem!

Add SOLIDS to the **PATTERN-ELEMENTS** box❹



Pattern-Elements toolbar

1. **Pattern-Elements toolbar**: Click the **Padlock**❶ to unlock the **toolbar**

2. Graphic-Area: **SHIFT+CLICK PROFILE** elements to add **EXTRUSION** elements that **you cannot see**

OR

2. Graphic-Area: **CLICK EXTRUSION** elements that **you can see** to add **EXTRUSION** elements

OR

2. Graphic-Area or **ASSEMBLY-TREE**: Click **CAD-LINES** onto which a **CAD-SOLID** has been imported.

3. **Pattern-Elements toolbar**: Click the **Padlock**❶ to lock the **toolbar** and box.

SOLID elements are now in the **PATTERN ELEMENTS** box as **EXTRUSIONS** and/or **CAD-LINES**.

To add a MOTION FB to the MOTION TO LINEARIZE box❺

Pattern-Elements toolbar

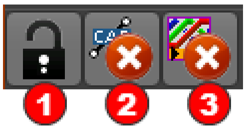
1. **Pattern-Elements toolbar**: Click the **Padlock**❶ to unlock the **Pattern-Elements toolbar**.

2. Graphic-Area or **ASSEMBLY-TREE:** Click a **MOTION FB**

3. **Pattern-Elements toolbar**: Click the **Padlock**❶ to lock the **Pattern-Elements toolbar** and box.

A **MOTION FB** is now in the **MOTIONS TO LINEARIZE** box

To remove a SOLID❹ or MOTION FB❺



Pattern-Elements toolbar

1. **Pattern-Elements toolbar**: Click the **Padlock❶** to unlock the **Pattern-Elements toolbar**.

---

2. Click to select a **PATTERN-ELEMENT** or a **MOTION FB**
3. Click the **REMOVE SOLID❷** or the **REMOVE MOTION FB❸** button in the **Pattern-Elements toolbar**

---

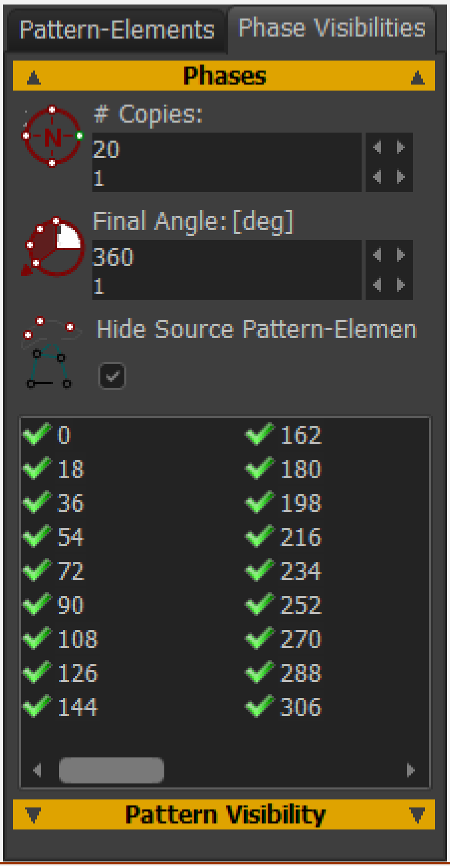
4. **Pattern-Elements toolbar**: Click the **Padlock❶** to lock the **Pattern-Elements toolbar** and box.

Phase Visibilities tab

To help you understand what the **Phase-Visibilities** can do, it is helpful to think of a chain around a circular 12-hour clock. The chain moves continuously around the clock. One of the chain-links is the Source **PATTERN-ELEMENT** (**SOLID**).

- Use **PHASES** to decide how many chain-links you want in the chain - called **# COPIES**
- Use **PHASES** to hide one or more chain-links (the Copies) permanently, **ALL** the way around the clock.
- Use **PATTERN VISIBILITY** to hide each chain-link, one link after the next link, as it moves between hours, for example, between 6 and 9 o'clock.

PHASES



PHASES are copies of **PATTERN-ELEMENTS**.

 # COPIES: from 1 to 100 : The number of copies of the PATTERN-ELEMENTS to be in the **Pattern**

The format of the **PHASES** separator is slightly different for the SPACE PATTERN and the PHASE PATTERN.

IF SPACE PATTERN:

START ANGLE, FINAL ANGLE:

- The **COPIES** show at equal intervals from the START ANGLE to the FINAL ANGLE.
- The **PHASE VISIBILITY** separator does **not** show.

IF PHASE PATTERN:

FINAL ANGLE = ?

The concept of a **Motion-Period** becomes important.

If FINAL ANGLE is 360, then the **Motion-Period** is equal to **1 Machine-Cycle**

If FINAL ANGLE is, for example, 20º, the **Motion-Period** is 20º.

- The **# COPIES** show at regular intervals within one **Motion-Period**, from HOME ANGLE (**MMA=0**) to FINAL ANGLE.
- The PHASE PATTERN repeats one time for each **Motion-Period** or 36 ⁄ 20 = 18 times in 1 Machine-Cycle
- You can show or hide each copy within each **Motion-Period** - see **PHASE VISIBILITY** separator.

HIDE SOURCE PATTERN ELEMENT:

SOURCE PATTERN ELEMENTS are those **SOLIDS** that are you select in the model as PATTERN-ELEMENTS , and show in the **PATTERN ELEMENTS** separator of the Pattern-Elements tab (415).

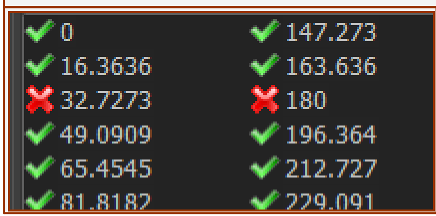
Nearly always, hide the SOURCE PATTERN-ELEMENT. This is because the SOURCE PATTERN-ELEMENT and the first COPY (at 0°) are at the same position. It can be become confusing if you intend to hide the Copy (at 0°) but the SOURCE PATTERN-ELEMENT remains in the model.

1. Click a PATTERN-ELEMENT in the PATTERN ELEMENTS (415) separator and Pattern-Elements tab (415)

It should show in the PHASE-VISIBILITIES (418)

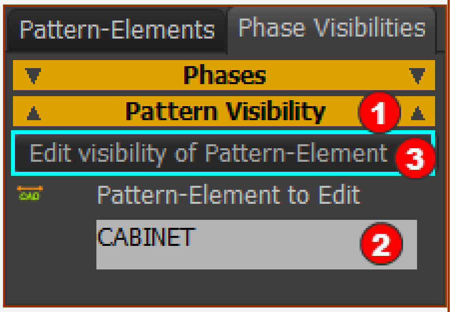
2. Click the HIDE SOURCE PATTERN ELEMENT check-box

COPIES BOX



- To **hide** a copy for the **Motion-Period**, click a ✓ to show a ✗.
- To **show** a copy for the **Motion-Period**, click the ✗ to show a ✓.

PATTERN VISIBILITY



Phase-Visibilities > Pattern-Visvibility

Note: PATTERN VISIBILITY ❶ shows only if you select PHASE PATTERN - **see** **Enable and Type.** 414

Use PATTERN VISIBILITY to show or hide a PATTERN-ELEMENT for different periods within the **Motion-Period.**

To edit the **Visibility** of a PATTERN-ELEMENT:

1. Click an EXTRUSION or CAD-LINE in the PATTERN ELEMENTS box - **see** **Pattern-Elements tab** 414

The element - EXTRUSION or CAD-LINE - must show in the PATTERN-ELEMENT TO EDIT❷ box.

2. Click the **EDIT VISIBILITY OF PATTERN-ELEMENT...** button❸.

The **Pattern Visibility** interface is now open.

You can now edit the Visibility of the element that is in the PATTERN-ELEMENT TO EDIT❷ box

How to use the Pattern Visibility interface

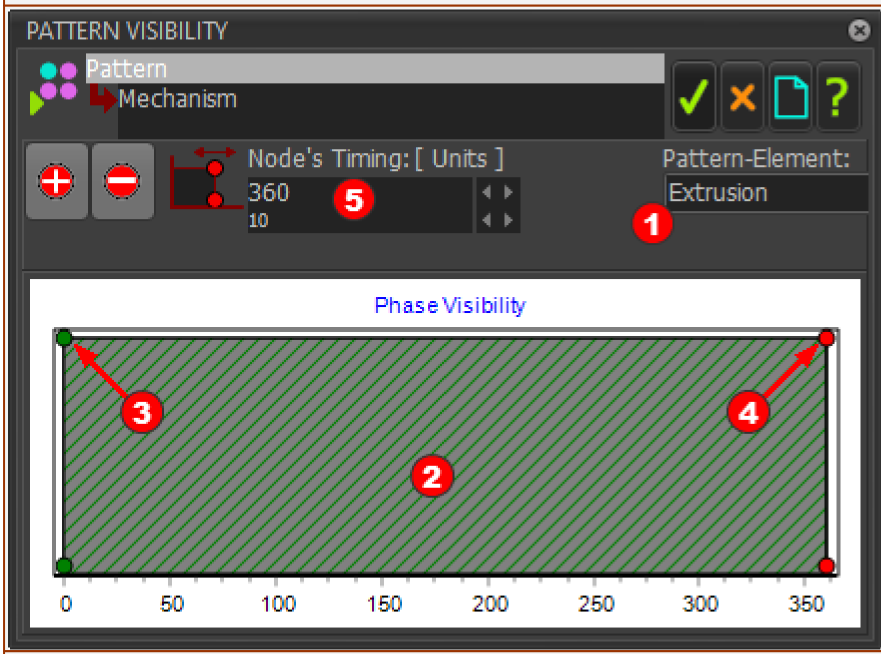
Refer to the image below:

❶ **Pattern-Element box (top-left)-** Identifies the PATTERN-ELEMENT 414 you are editing.

❷ **Phase Visibility Timing Chart -** this chart controls the **Visible-Period** of the PATTERN-ELEMENT (**SOLID**) in the Motion-Period.

As an example

In the image below, the chart is **High** from 0º to 360º. This means the **Visible-Period** for the PATTERN-ELEMENT is a complete motion-period.



There are two(2) **NODES** that identify and specify the timing of each **Visible-Period** in the chart.

- Node at the Start - **Start-Node** ❸
- Node at the End - **End-Node** ❹

To edit a **Visible-Period** for the PATTERN-ELEMENT you must edit the timing of the **Start-Node** and/or **End-Node.**

STEP 1. (Example) Edit the Start-Node and End-Node to define a Visible-Period.

1. Click the **Start-Node**❸.

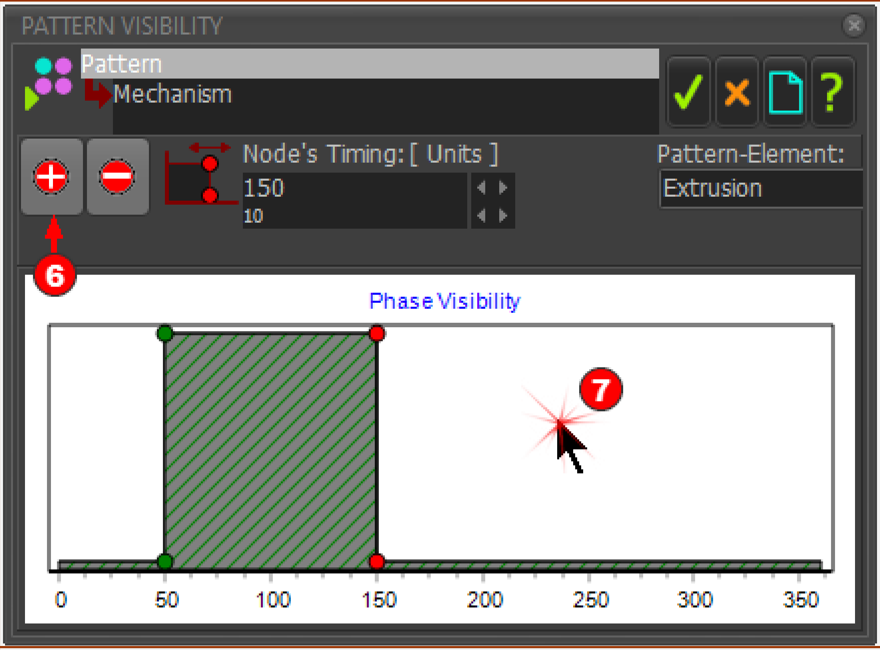
When a **Start-Node** is Red ...

2. Edit the NODE'S TIMING❺

Do **1** and **2** again, but click the **End-Node**❹, to define the **Visible-Period** for the PATTERN-ELEMENT

The image below shows the new timing of the **Visible-Period** for the PATTERN-ELEMENT (EXTRUSION).

It starts at 50 and end at 150 in the Machine-Cycle.



STEP 2. (Example continued...) Add a new Visible-Period

- see image above.

1. Click ADD NODE button ❻

2. Click the PHASE VISIBILITY CHART in a free area❼ that is not within the **Visible-Period** 50 to 150 - see cursor in the image above

There is now a new **Visible-Period** in the PHASE-VISIBILITY CHART - see image below.

STEP 3. Edit the Start-Node and End-Node to define the timing of the new Visible-Period.

1. Click the **Start-Node**❸ or **End-Node**❹ of the new **Visible-Period**

When a **Node** is Red ...

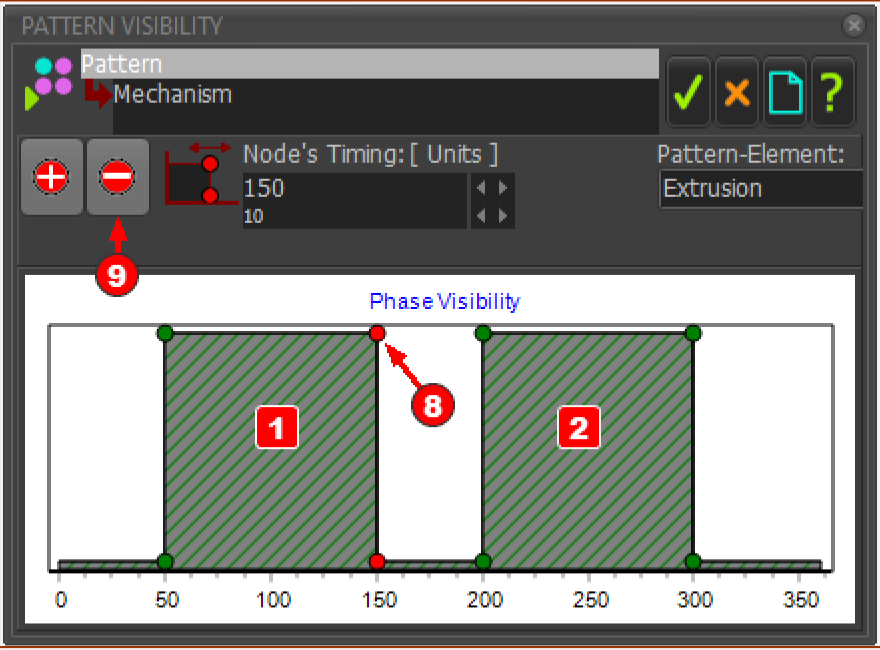
2. Edit the NODE'S TIMING❺

Do **1** and **2** again to define the new **Visible-Period** for the PATTERN-ELEMENT

The image below shows two **Visible-Periods** for the PATTERN-ELEMENT (EXTRUSION)

Visible Period 1 - Starts at 50 and ends at 150

Visible-Period 2 - Starts at 200 and ends at 300



STEP 4. (Example continued...) Delete a Visible-Period

1. Click the **Start-Node** or **End-Node** of the **Visible Period** you want to delete

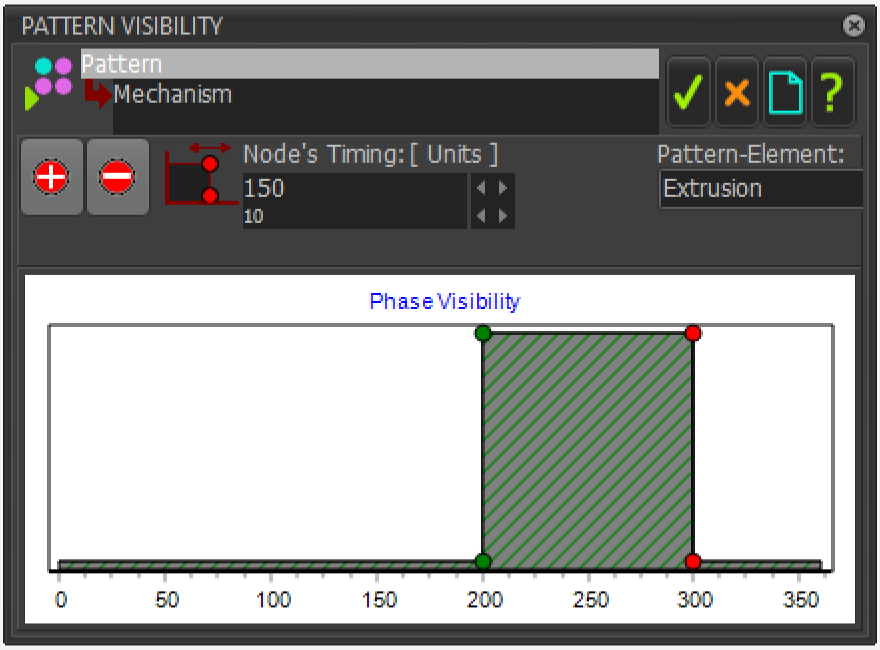
In the image above the **End-Node**8 of **Visible-Period** 1 is Red ...

2. Click **DELETE-NODE** 9

The **Visible-Period** 1 is removed for the PATTERN-ELEMENT (EXTRUSION)

The image below shows one **Visible-Period** for the PATTERN-ELEMENT (EXTRUSION)

You can now Click to close the **Pattern Visibility interface**.



1.10.38 Dialog: Function-Block: Briefcase

Briefcase FB

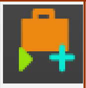
See: Add Briefcase [178]

Use a **BRIEFCASE FB** to hide other **Function-Blocks** from the graphic-area.

You can use one **BRIEFCASE FB** to hide all of the **Function-Blocks**. However, I usually add one **BRIEFCASE FB** for each machine function.

You can also hide a **BRIEFCASE FB** inside a different **BRIEFCASE FB.**

How to open the Briefcase FB dialog



Open the **BRIEFCASE FB** dialog:

1. Double-click the **BRIEFCASE FB** in the graphic-area

OR

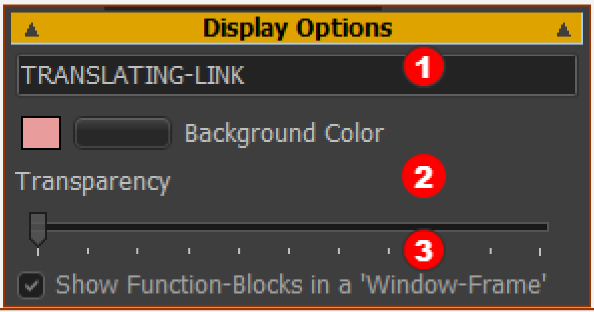
1. **See How to Open a dialog** [513]

The **BRIEFCASE DIALOG** is now open.

Briefcase FB dialog

Briefcase FB dialog

DISPLAY OPTIONS



❶ << WRITE A LABEL >> (For example TRANSLATING-LINK)

After you add **Function-Blocks** to the **BRIEFCASE FB**, you can enter a **name** to remind you of their function in the model. The **name** shows as a label above the **BRIEFCASE FB** in the graphic-area.

❷ BACKGROUND COLOR and TRANSPARENCY

Use the BACKGROUND COLOR button to edit the color of the **BRIEFCASE DISPLAY WINDOW - see example below.**

Use the TRANSPARENCY slider to edit the transparency/opacity of the **BRIEFCASE DISPLAY WINDOW**.

❸ SHOW-FUNCTION-BLOCKS IN A WINDOW FRAME

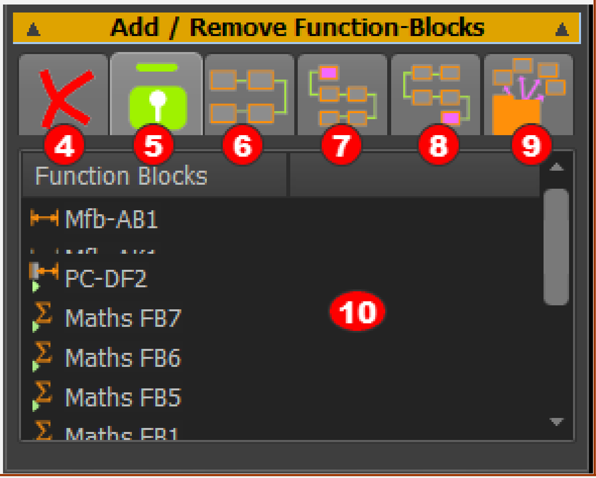
☑ Enable SHOW-FUNCTION-BLOCKS IN A WINDOW FRAME to show the BRIEFCASE DISPLAY WINDOW

❑ Disable SHOW-FUNCTION-BLOCKS IN A WINDOW FRAME to hide the BRIEFCASE DISPLAY WINDOW

SHORT-CUT - To toggle SHOW-FUNCTION-BLOCKS IN A WINDOW FRAME

1. **Move your mouse above a** BRIEFCASE FB
2. **CTRL+Click**

ADD / REMOVE FUNCTION-BLOCKS



Add FBs to the Briefcase

1. Click the PADLOCK❺ to unlock the BRIEFCASE

The PADLOCK❺ icon becomes unlocked - see image.

2. Click FBS in the graphic-area

   When you click a FB, it moves from the graphic-area to the FUNCTION-BLOCKS❿ box in the BRIEFCASE.

3. Click the PADLOCK❺ again to lock the BRIEFCASE.

Remove FBs from the Briefcase:

1. Click the PADLOCK❺ to *unlock* the BRIEFCASE
2. Click a FB that is in the FUNCTION-BLOCKS❿ box to make the active FB.
3. Click the RED CROSS❹ button.

   The FB returns to the graphic-area, outside of the BRIEFCASE DISPLAY WINDOW.
4. Do **1 - 3** again to remove a different FBS.
5. Click the PADLOCK❺ again to lock the BRIEFCASE

Arrange FBs in the Briefcase Display Window.

1. Click SHOW ELEMENTS check-box❸ - see above.
2. Click the ARRANGE FBS❻ button

The FBS are now arranged in a **linear array** in the graphic-area.

You may need to drag FBS to arrange the FBS within the **linear array.**

Note: SHORT-CUT: CTRL+CLICK a BRIEFCASE FB to show then hide the **Briefcase Display Window**

Add all FBs that are connected with wires 'upstream' of a FB

1. Click a FB in the FUNCTION-BLOCKS box❿ in the BRIEFCASE
2. Click the ADD ALL UPSTREAM icon ❼

The FBS that are connected to its input-connector are now in the FUNCTION-BLOCK box.

Add all FBs that are connected with wires 'downstream' of a FB

1. Click a FB in the FUNCTION-BLOCKS box❿ in the BRIEFCASE
2. Click the ADD ALL DOWNSTREAM icon❽

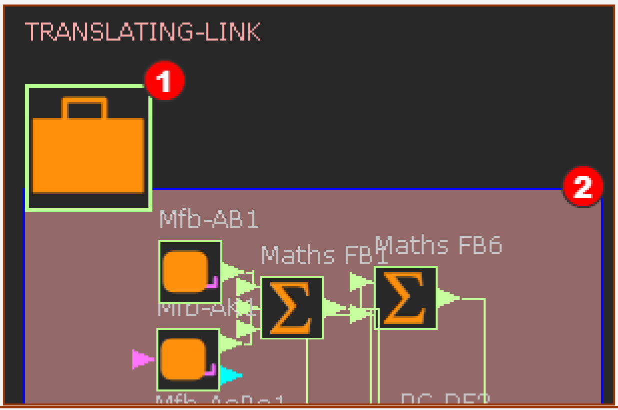
The FBS that are connected to its output-connector are now in the FUNCTION-BLOCK box.

Remove all FBs from the Briefcase

1. Click the REMOVE ALL icon❽

The FBS are now in the graphic-area

EXAMPLE:



❶ BRIEFCASE FB and label: **TRANSLATING-LINK**.

❷ BRIEFCASE DISPLAY WINDOW is a colored box, with a **blue** border.

- The window re-size as you drags, add, or remove FUNCTION-BLOCKS.
- Drag the BRIEFCASE FB❶ to move the BRIEFCASE DISPLAY WINDOW❷

1.10.39 Dialog: Function-Block: Point-Cloud

Point-Cloud FB

See **Add Point-Cloud** 177

Do two(2) steps in the POINT CLOUD DIALOG:

STEP 1: IMPORT-DATA (the POINT-CLOUD, TXT file-type)**.**

Use the

Frequently, the POINT-CLOUD are coordinates of a cam-profile, which are measured with a **Coordinate Measurement Machine** (CMM).

It is possible to use numerical techniques to calculate the motion-derivatives from the POINT-CLOUD.

STEP 2: FIT-A-CURVE TO THE DATA

Use the POINT-CLOUD DIALOG to fit a **Curve** to the POINT-CLOUD.

The **Curve** is a continuous-function that we symbolically differentiate for you to calculate the motion-derivatives.

How to open the Point-Cloud dialog

Open the POINT-CLOUD DIALOG:

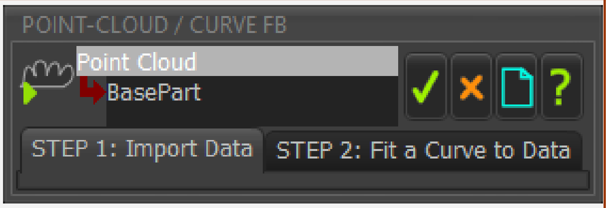
1. Double-click a POINT-CLOUD FB in the graphic-area

OR

...see How to open a dialog 513

The POINT-CLOUD DIALOG is now open.

Point-Cloud dialog



Point-Cloud dialog

There are two tabs:

STEP 1: Import Data

STEP 2: Fit Curve to Data

STEP 1: Import Data

Do A, B, AND C.

Import and Browse buttons

A. BROWSE...

Click the BROWSE... button to select the data-file.

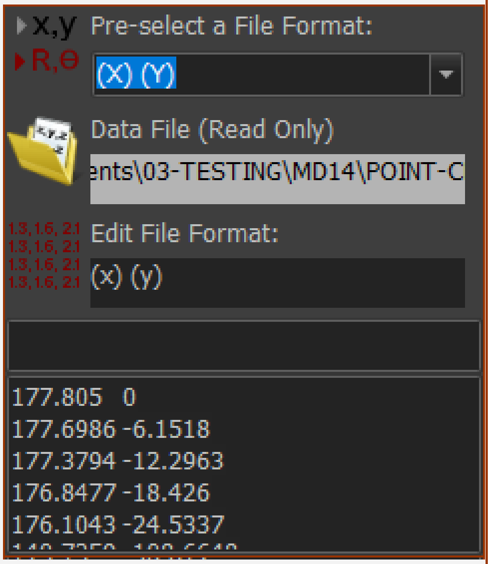
The data-file must by an ASCII, or TXT file-type.

The file-name shows in the DATA-FILE (READ-ONLY) box. A preview of the data-file is in the box below.

B. EDIT FILE-FORMAT

Header text in the file is not imported.

The data should be rows of data-points - see **Data-Points** below



Example File Format

You can remove extra **data** from columns/rows with the EDIT FILE-FORMAT box. See File Formats[426] below.

C. IMPORT BUTTON

Click the **IMPORT** button to import the POINT-CLOUD data-file.

The POINT-CLOUD is a continuous line in the graphic-area.

See also: Messages[427]

Data-Points

The **File Format** ***must*** include at least these terms:

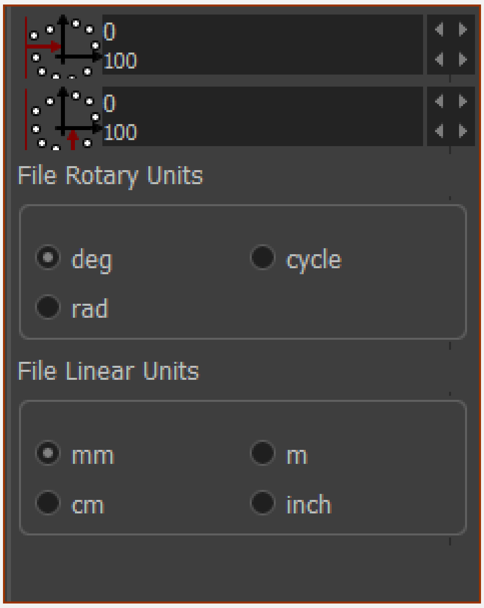
- **(x) (y)** : if each data-point is an **X** and **Y** coordinate

OR

- **(Radius) (Angle)** : if each data-point is a **Radius** and **Angle** coordinate.

Characters that are ignored in the data-file:

- Header lines, when entered as text.
- ASCII characters that are NOT: **. , -, 0–9,** in all lines.



Optional: Data Offset and Scale option

Options:

X , Y

When you add a POINT-CLOUD, you select a PART. The PART has an **ORIGIN** - its **START-POINT** - where its XY coordinates are (0,0)

When the POINT-CLOUD has a measurement-offset of (**X, Y**), you can remove the offset. Enter (-**X**, -**Y**), which is the negative of the offset.

Or, to offset the POINT-CLOUD, by (X,Y), enter (X,Y) values.

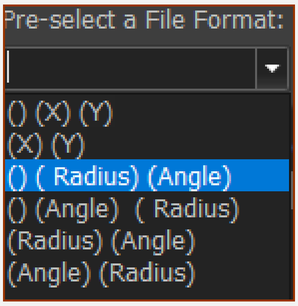
FILE ROTARY UNITS ; FILE LINEAR UNITS

Select the units that match those of the **original** POINT-CLOUD data-file

File Formats:

PRE-SELECT A FILE FORMAT

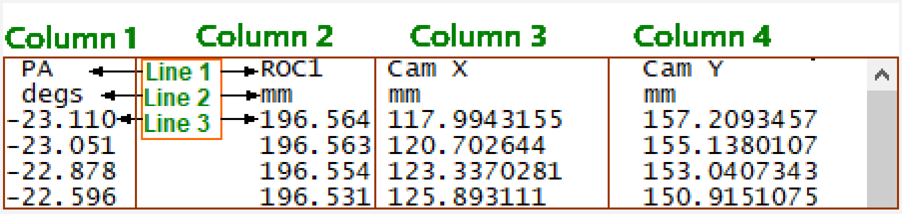
To help you with the formatting, you can use the PRE-SELECT A FILE FORMAT from the drop-down list-box.



When any of these formats do not represent the format of the data in your data-file, then edit the format in the **EDIT FILE FORMAT** box directly.

Example: Data-Files & File-Formats and the Enter File Format text.

EXAMPLE 1: Data-File:



Enter File Format: () () (x) (y)

Why?

Line 1	Is Not imported - text only (A, B, C, … Z)
Line 2	Is Not imported - text only (A, B, C, … Z)
Line 3 Column 1 & Column 2:	**Is Not imported** because of () () in the File Format () () (x) (y) removes these two Columns
Line 3 Column 3 & Column 4	**Is Imported** because the (x) (y) in the File Format () () (x) (y) identify that these Columns as x, y data
Lines 4, 5, 6, …	Equal to Line 3

EXAMPLE 2: Data-File

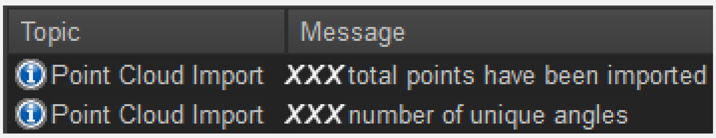
Data Format: N100 G00 X10 Y25 Z27

Enter File Format: () () (x) (y) ()

Why?

N100 G00 X10 Y25 Z27	Letters not imported (A, B, C, … Z)
~~N100~~ ~~G00~~ ~~X~~10 ~~Y~~25 ~~Z27~~	Column 1, Column 2, and Column 5 - not imported - the () in the **File Format** () () (X) (Y) ()
10 25	Imported - the (X) and (Y) in the **File Format** () () (X) (Y) ()

Feedback Messages:

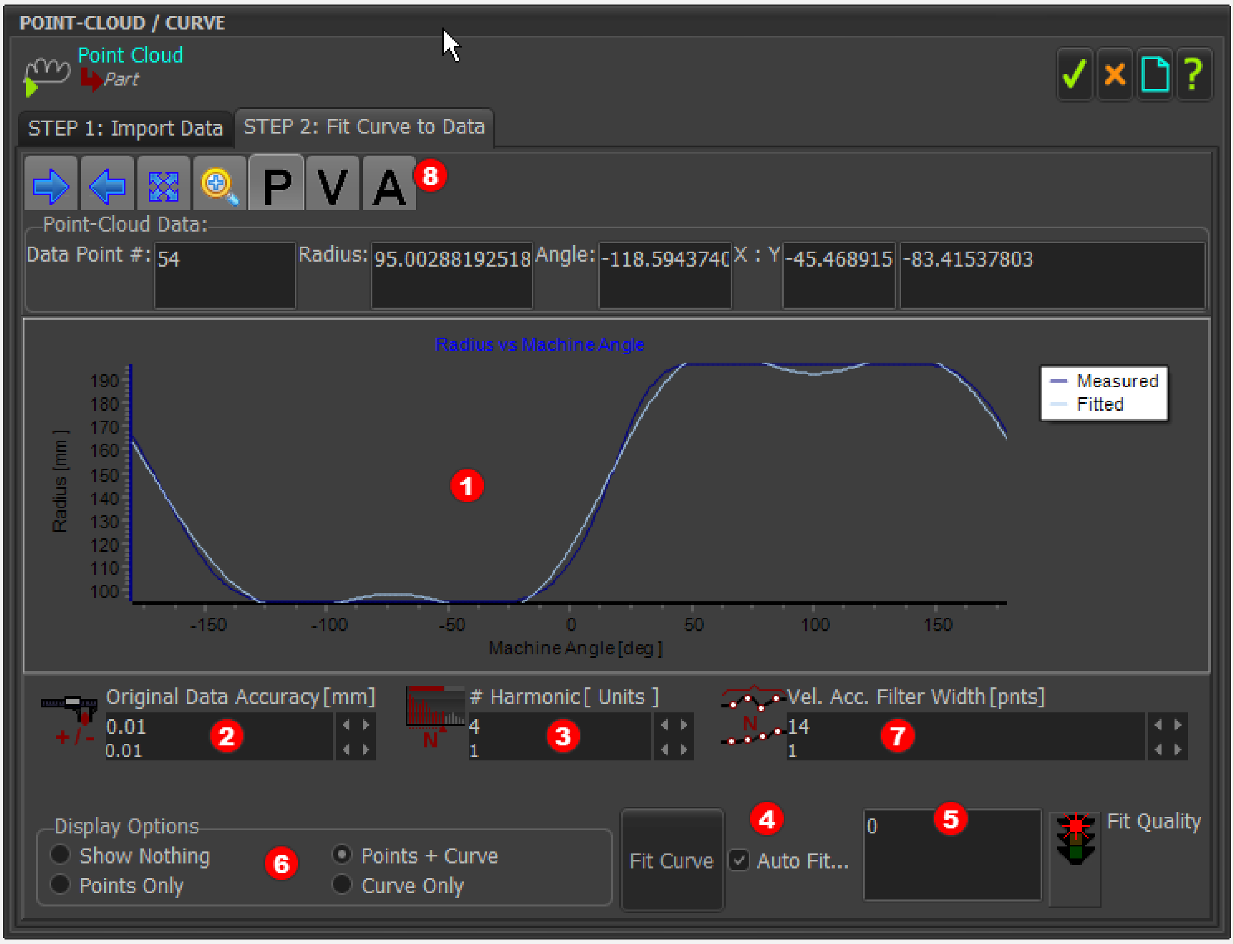


STEP 2: Fit Curve to Data

When you import **Point-Cloud data**, the chart❶ shows the **Point-Cloud** data in (Cyan) and the **Continuous Curve** in (Pink).

In the chart below, the **Continuous Curve** does *not* follow the **Point-Cloud data** - this is intended and for illustration ONLY.

The two charts show as one chart after you successfully **Fit the Continuous Curve** to the **Point-Cloud data**.



Point-Cloud dialog > Fit Curve to Data tab

Parameters

❷ ORIGINAL DATA ACCURACY

This is a judgment you must make. Start with 0.1, then reduce it to approximately 0.001 - but see below 429 for more thoughts.

❹ AUTO-FIT CHECK-BOX AND FIT CURVE BUTTON

Usually, click the AUTO-FIT button to make it active - see options

❸ INCREASE THE # HARMONICS

Edit the # HARMONICS❸

The **Continuous-Curve** becomes nearer to the **Imported Data** as you increase the **# Harmonic**❸.

❺ FIT-QUALITY =0 >> 1

When FIT QUALITY TRAFFIC LIGHT becomes Green, ...

or

When the number in the FIT-QUALITY NUMBER BOX❺ is near to, or equal to, one(1), ...

... STOP - do not increase the number of Harmonics.

❻ DISPLAY OPTIONS

Click one of the four radio-buttons

- O SHOW NOTHING
- O POINTS ONLY
- O POINTS AND CURVE
- O CURVE ONLY

The view of the **Continuous-Curve** and the POINT-CLOUD in the Graphic-Area match the P, V, or A❽ button.

Steps to do:

A. Edit the # Harmonics

☑ AUTO-FIT Check-box

The CURVE updates immediately each time you edit the # HARMONICS

❑ AUTO-FIT check-box is clear.

The CURVE only updates when you click FIT CURVE❹

B. Do A again

Do until the FIT-QUALITY and the TRAFFIC-LIGHT becomes **Green** or **Orange**

VELOCITY AND ACCELERATION GRAPHS AND FILTER WIDTH❼

The default plot is for Position. If you want to look at the plot of the Point-Cloud as Velocity or Acceleration (❽), then click the **V** or the **A** icons.

If the plot of the Velocity or Acceleration data is noisy you can smooth the noise (remove the noise) with the Filter-Width❼. This may help you identify the original Motion-Law.

Original Data Accuracy❷

Thoughts:

- Consider an **Original Data Accuracy** of 0.1, then reduce it to approximately 0.01.
- Do not enter a value that is more accurate than the original accuracy of the data. For example, if the data has 2 d.p. , then 0.05 OK, and 0.001 is not a sensible value. However, experiment, as you find that more accuracy gives you better results.
- Is the skill of the person that measured the Cam-Profile a factor?
- Is the cam clean? Is the cam worn? Is the cam damaged?

Machine Measurement Accuracy Approximations: per 100mm Diameter of Cam Diameter:

- Temperature-Controlled (18-22ºC) CMM approximately 0.2 - 5μm
- Non-temperature-controlled CMM approximately 2 - 10μm (~h4 for a Ø100mm shaft)
- Portable CMM approximately 5 - 20μm (~h5-h6 for a Ø100mm shaft)
- Test Rigs are variable, but you should aim for approximately 20 - 100μm (~h8-h9 for a Ø100mm shaft)

1.10.4 Dialog: Function-Block: Design-Set

Design-Set FB

See Add Design-Set 185

Use DESIGN-SETS to edit a number of dimensions and parameters in one place - with the DESIGN-SET DIALOG.

Add to a DESIGN-SET those dimensions and parameters that you believe to be important to the outcome of a design-objective. You can add more than one DESIGN-SET to the model.

Question:

Why use a **DESIGN-SET?**

Answers: Three(3) good reasons:

A. Frequently, you edit many times the same dimensions and parameters to improve a design objective. With a DESIGN-SET, you can add them to the **Design Set interface**, and edit them continually.

B. If you need to open the model after a period of time, the DESIGN-SET reminds you of which dimensions and parameters are important to your design. Also, you can give your model to a different engineer, and tell him to ONLY edit the dimensions and parameters in the DESIGN-SET.

C. When dimensions are in a DESIGN-SET, you cannot edit them in the model. A DESIGN-SET can prevent accidental changes to the model. Also, you can add those dimensions to a DESIGN-SET specifically to stop you editing them.

IMPORTANT

After you add a DIMENSION to a DESIGN-SET, you CAN ONLY use the DESIGN-SET to edit that DIMENSION.

To edit a DIMENSION in the PART-EDITOR again, you must delete the DIMENSION from the DESIGN-SET.

A DIMENSION that is in the DESIGN-SET is **gray** in the PART-EDITOR - to indicate you cannot edit it.

TOP-TI[S:

1. Rename each dimension to a function-name, to help you recognize it in the DESIGN-SET.
2. **Rename** each DESIGN-SET to remind of its design-objective.

See SHOW DIMENSION-NAMES 286 , and SHOW FUNCTION-BLOCK NAMES 287

How to open the Design-Set FB dialog

Edit the DESIGN-SET FB

1. Double-click the DESIGN-SET FB in the graphic-area

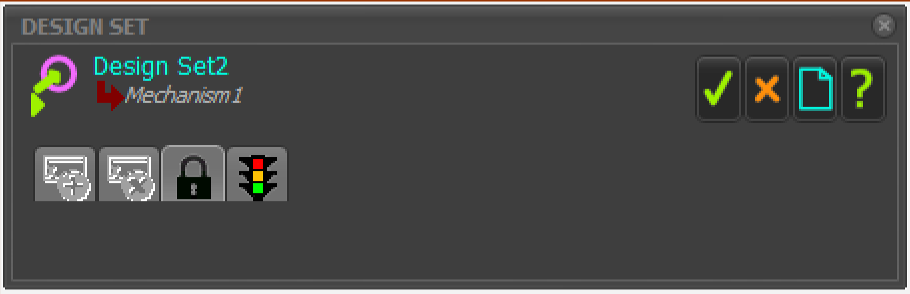
OR

1. **See How to Open a dialog** 513

The **DESIGN-SET DIALOG** is now open.

Design-Set dialog

Design-Set toolbar



Design-Set dialog - LOCKED

Design-Set dialog - UNLOCKED

When you open the **DESIGN-SET DIALOG**, the toolbar is locked. It is ***NOT*** active.

The Design-Set toolbar icons - when active -are:

❶ : ADD ELEMENT-ROW

❷ : DELETE ELEMENT-ROW

❸ : PADLOCK : LOCK / UNLOCK TOGGLE

❹ : REBUILD MODEL

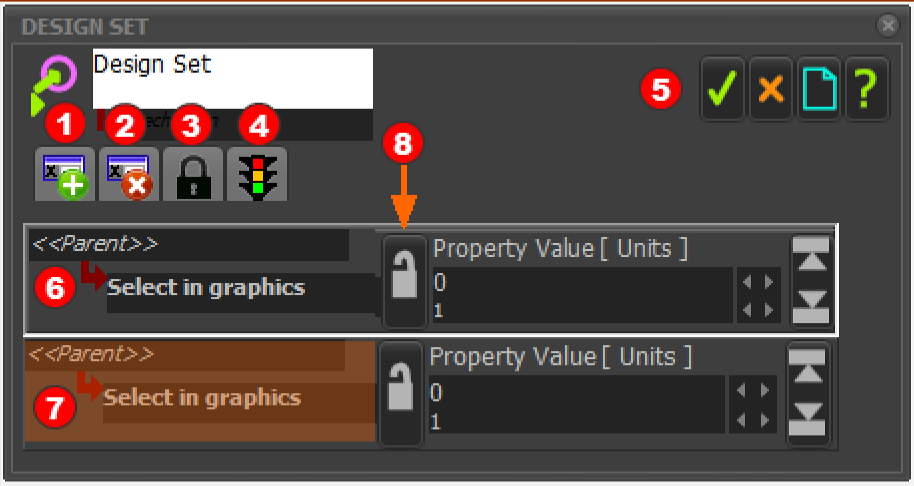
❺ : ✓ × 🗋 ? OK, CLOSE, UNDO EDITS, HELP

To enable the Design-Set toolbar icons

1. Click the **PADLOCK** icon❸.

The icons and the **DESIGN-SET** in the toolbar become ***colorized*** and ***active***.

Add or Delete Element-Rows



Design-Set - 2 Element Rows

To add Element-Rows

1. Click the **PADLOCK**❸ - the toolbar is now colorized.
2. Click **ADD ELEMENT-ROW**❶ one time for each dimension or parameter you want to add to the **DESIGN-SET**.

   In the image above, there are two **Element-Rows**❻❼.

The active Element-Row.

The **active Element-row**❻ has a white box around its perimeter.

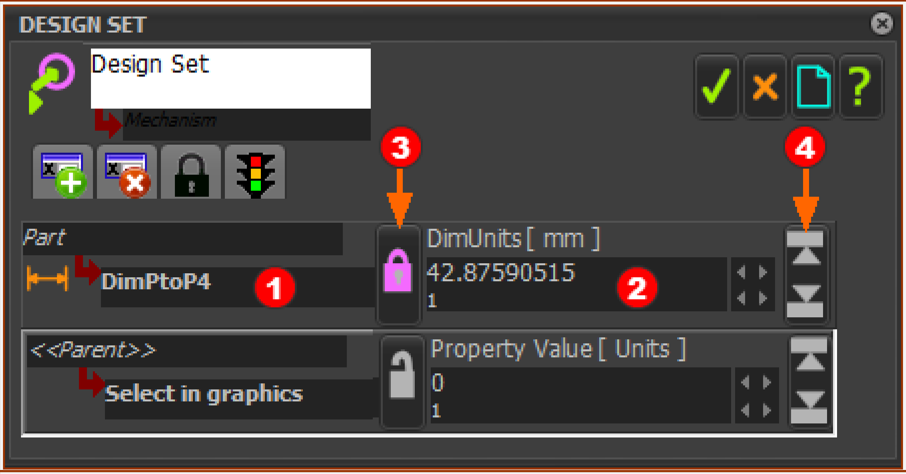
To make a different **Element Row** the **active Element-Row**:

1. Click in the ***Select in graphics***❼ label **-** or the parameter name

To delete an Element-Row

1. Click the ***Select in Graphics*** label to make it the **active Element-Row**
2. Click **DELETE ELEMENT-ROW**❷

Link a Dimension to an Element-Row



1. If necessary, open the **DESIGN-SET DIALOG**
2. Click the **Select in graphics** label of an **Element-Row**, so that it becomes the **active Element-Row**.
3. Edit a **PART**
4. Click the arrowhead of a **DIMENSION** in the **PART-EDITOR**

Result:

The **Dimension name**❶ replaces the **Select in graphics** (DimPtoP4)

The Dimension Units name replace the Property Value (DimUnits)

The **Dimension value**❷ replaces the **Property Value** (42.87580515)

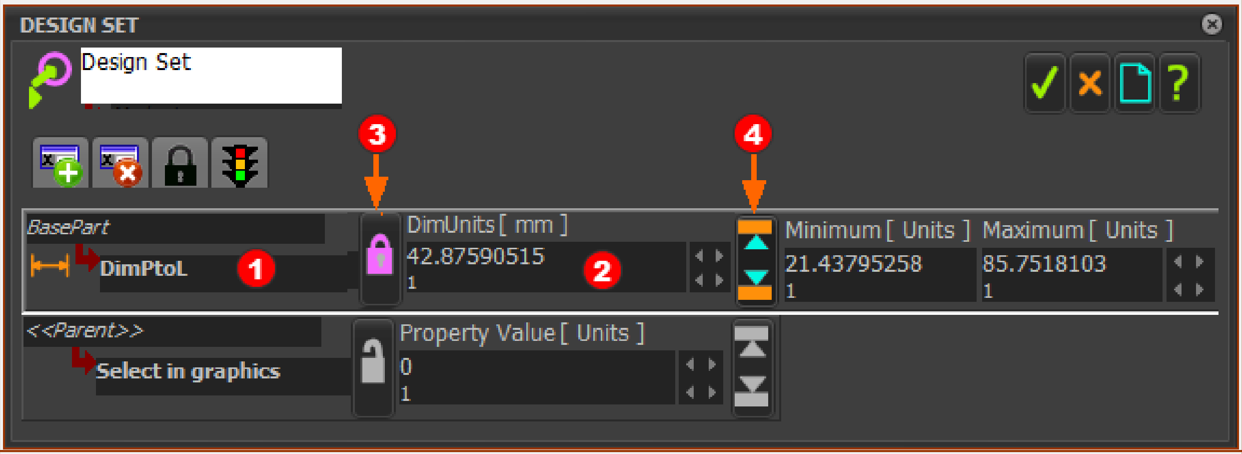
The **Locked Padlock**❸ becomes colorized. In this state, you **cannot overwrite** this **Element-Row.**

The next **Element-Row** becomes the **Active Element-Row** (a white box is around the **Element-Row**)

Maximum and Minimum 'Hard-Limits'

Maximum and Minimum 'Hard-Limits'

It is frequently useful to set maximum and minimum limits for each dimension or parameter in the **Design-Set**.



To set the Maximum and Minimum limits:

1. Click the up/down arrow **Min-Max button❹ - see image above**

   The **Minimum** and **Maximum** data-boxes becomes active.
2. Edit the **Minimum** and **Maximum** values.
3. Continue to edit the Dimension❷ - but now it cannot be less-than or more-than the **Minimum** or **Maximum** values, respectively...

Replace a Dimension or Parameter

To replace a Dimension or Parameter in an **Element-Row** with a different Dimension or Parameter.

1. Click the **Dimension name** to make it the **Active Element-Row**.
2. Click the Locked Padlock❸ so that it becomes Grey Unlocked Padlock.
3. Click a different Dimension in the graphic-area.
4. Click the Grey Unlocked Padlock so it be becomes the Locked Padlock❸ again.

Parameter Lists in different Element-Types:

In addition to dimensions, you can link parameters from dialogs for different element types.

For example: Parameters for FBs, Parameters for a Gear-Pair, Parameters for a CAD-Line, and more.

Link a Parameter (from a dialog-box) with an Element-Row

Select the Element-Type:

1. Open the DESIGN-SET DIALOG
2. Click the **Element-Row** to make it the **Active Element-Row**
3. Click the PADLOCK icon, to unlock the DESIGN-SET.
4. Click the ELEMENT-TYPE in the graphic-area or the ASSEMBLY-TREE

If the **Element-Type** has only one(1) PARAMETER

The PARAMETER passes immediately into the **Element Row**.

If the **Element-Type** has two(2) or more PARAMETERS:

A box opens with a list of **PARAMETERS** that you can select.

See below: 'Parameter Lists in different-elements' below. If you cannot identify the **PARAMETER** from its name, then do not use it.

1. Click the **PARAMETER** in the list
2. Click the ✓ button to close the **PARAMETER** box

   The **PARAMETER** name and its value shows in the **Element-Row** in the **Design-Set**.

AVAILABLE ELEMENT-TYPES AND THEIR PARAMETERS

Linear-Motion Function-Block

When you click a **LINEAR MOTION FB**, you add:

- **LINEAR MOTION** as the **Element Name**
- **OUTPUTSTART_UNITS** as the parameter.

OUTPUTSTART_UNITS ≡ **START-ANGLE** parameter in the **LINEAR-MOTION DIALOG**.

Motion-Dimension Function-Block

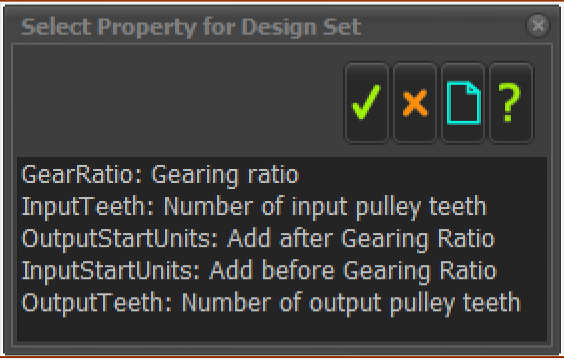
When you click a **MOTION-DIMENSION FB**, you add:

MOT-DIM ROCKER (or **SLIDER**) as the **Element Name**

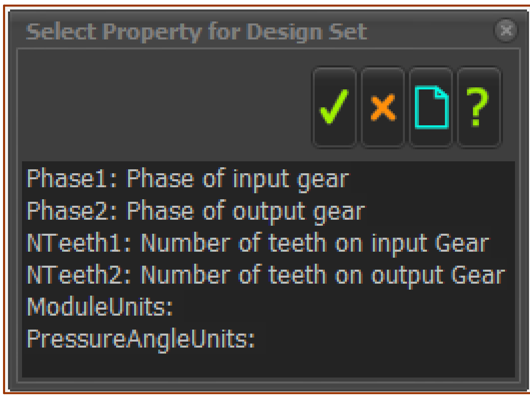
BASEUNITS as the parameter

BASEUNITS ≡ **BASE-VALUE** in the **MOTION-DIMENSION DIALOG**.

Gearing Function-Block

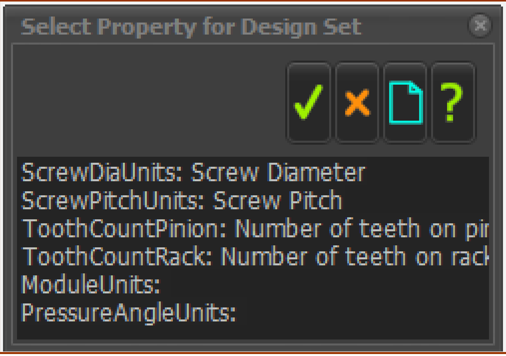


Gear-Pair

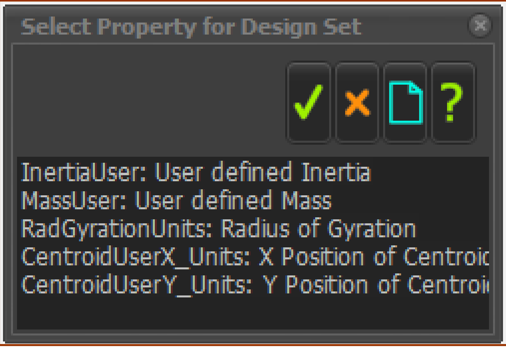


Do not select Phase1 or Phase2 for the Design-Set.

Rack-Pinion / Ball-Screw



CAD-Line



Pulley

When you click a PULLEY, you add

- PATH JOINT as the **Element Name**
- TOOTH-COUNT as the parameter.

TOOTH-COUNT ≡ NUMBER OF PULLEY TEETH parameter in the PULLEY DIALOG.

1.10.4 Dialog: Function-Block: Continuous-Crank

Continuous-Crank

See also **Add Continuous-Crank** [187]

It is often the case, in a multi-part mechanism, that the motion is designed for the Tool-Part (the Part with the tooling).

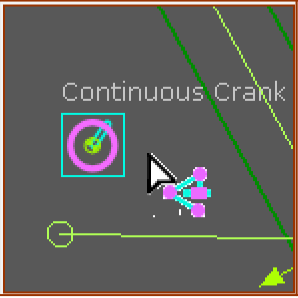
The Tool-Part is often driven by a different Part in the mechanism, which is the Driving-Part. We derive for you the motion of the Driving-Part from the Tool-Part by inverse-kinematics.

Usually, with a Cam mechanism, the Driving-Part is an oscillating Cam-Follower that is moved by a Cam.

However, there is also the case in which the Driving-Part rotates fully, but with a non-uniform speed. This is a special-case. To make sure this can happen, the Driving-Part must be exactly the correct length, and its motion, also derived by inverse-kinematics from the motion of the Tooling, is more complex.

This is one application of the CONTINUOUS-CRANK FB.

How to open the Continuous-Crank FB dialog



Edit the CONTINUOUS-CRANK FB:

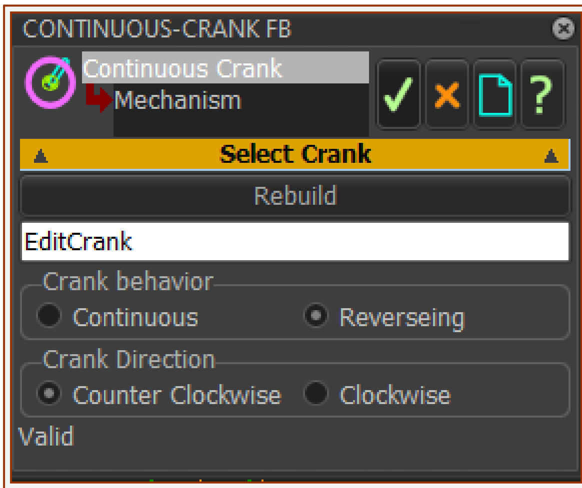
1. Double-click the CONTINUOUS-CRANK FB in the graphic-area

OR

1. **See** **How to Open a dialog** [513]

The CONTINUOUS-CRANK DIALOG is now open.

Continuous-Crank dialog



Continuous-Crank dialog - not enabled

STEP 1: Select Crank Behavior and Direction

CRANK BEHAVIOR

CONTINUOUS - the PART rotates continuously.

REVERSING - the PART rotates to ~180º, and then return back to ~0º.

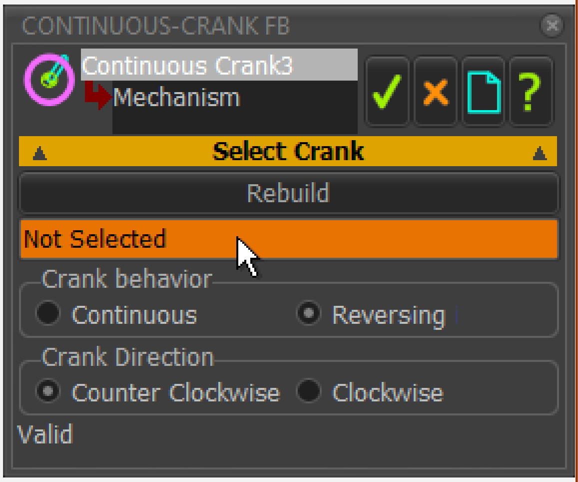
CRANK DIRECTION

If CRANK BEHAVIOUR is CONTINUOUS, then:

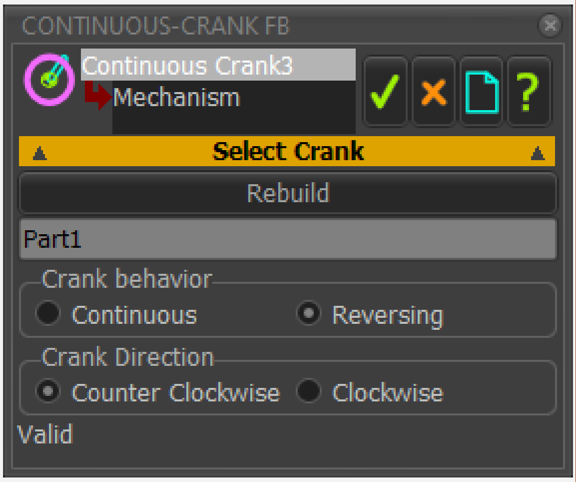
COUNTER-CLOCKWISE - the PART rotates in the Counter-Clockwise direction

CLOCKWISE - the PART rotates in a Clockwise direction

If CRANK BEHAVIOUR is REVERSING, to change Crank closure, see Change-Dyad Closure [101].



Continuous-Crank dialog- enabled



Continous-Crank dialog

STEP 2: Activate the Not Selected box

1. Click in the box labeled **Not Selected**

THE BOX IS NOW ORANGE

You can now select a PART.

STEP 3: Click a Part that you want to be the Continuous-Crank

1. Click a PART in the graphic-area or **ASSEMBLY-TREE.**

The Continuous-Crank PART must have a PIN-JOINT that joins it to the BASE-PART.

The PART immediately changes its length so that it can rotate continuously or oscillate by ~180º. If there is a problem, see **Messages at the bottom of the dialog**.

STEP 4: Confirm the behavior is as expected

1. Do **Run toolbar > Cycle Continuously** (**ALT+C**)

STEP 5: Click to close the dialog.

Messages at the bottom of the dialog:

When you select a PART as the Continuous Crank, there is be a message at the bottom of the dialog.

Message	Meaning
Valid	The PART can be a continuous Crank
Failed assign pivot-Point	We could not find for you the rotating-axis.
Impossible crank length: Geometry is inconsistent	For example, the **MOTION-PART** does not move, and thus the length of the Crank must zero(0).

1.10.42 Dialog: Function-Block: Spring

Spring FB

See Add Spring FB[194]

Use a SPRING FB to apply a force between two POINTS. The name that we use for the two POINTS is ANCHOR-POINTS.

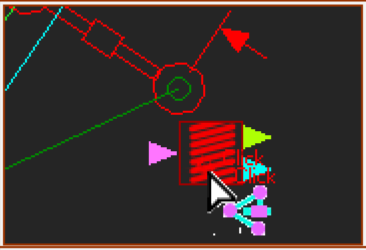
Force units are defined in the **Machine-Settings dialog > Engineering Units**[292] - default Force units are Newtons (N).

See also:

Machine Settings > Engineering Units[292]

Configure Power Source[480]

How to open the Spring FB dialog



In Mechanism-Editor:

1. Double-click the SPRING-FB in the graphic-area

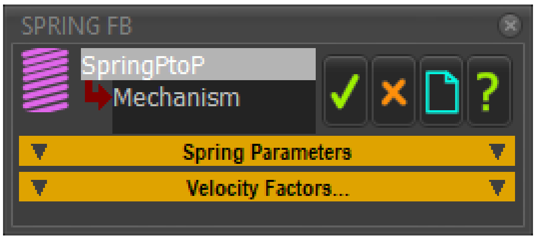
OR

1. **See How to Open a dialog**[513]

Note: It is **not** necessary to connect a wire to the input-connector or from the output-connectors of the SPRING FB. The input and output-connectors provide extra functionality - see Here[440]

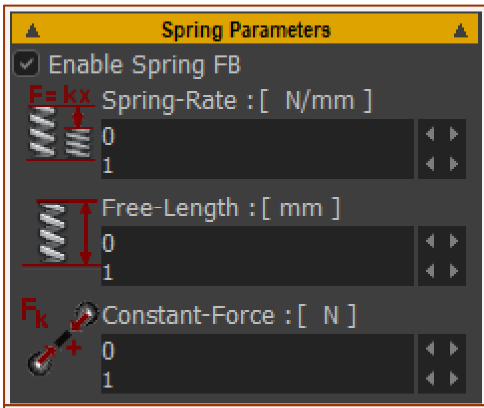
The SPRING DIALOG is now open.

Spring dialog



Spring FB dialog

SPRING PARAMETERS



☑ Check-box to enable the SPRING FB.

SPRING-RATE:

Force Contribution:

$= \text{SPRING-RATE} \times abs \left| \text{SPRING-LENGTH} - \text{FREE-LENGTH} \right|$

SPRING-LENGTH is the distance between the two anchor-points.

The Force always tries to move the anchor-point to the FREE-LENGTH (They cannot, of course).

FREE-LENGTH:

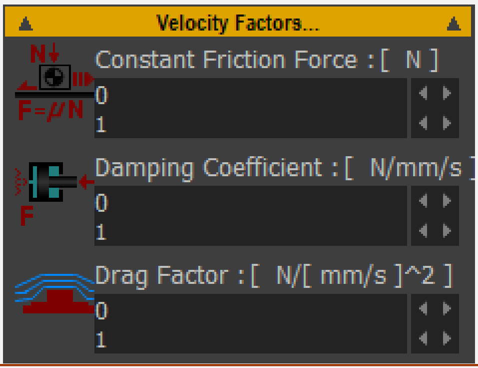
The natural length of the **Spring**, when it is not joined to the ANCHOR-POINTS.

CONSTANT-FORCE:

A positive force PULLS the ANCHOR-POINTS together (**o>>> + <<<o**)

A negative force PUSHES the ANCHOR-POINTS apart (**o<<< — >>>o**)

VELOCITY PARAMETERS



COULOMB (CONSTANT) FRICTION FORCE: (Ff = µ.Fn)

A **CONSTANT FORCE** that is opposite to the direction of the motion.

You can use it to approximate a Friction-Force.

Note: Friction Force is NOT constant in general. Friction-Force is proportional to the Contact-Force between two bodies, and the Contact-Force is not generally constant.

VISCOUS COEFFICIENT: (Fv α V (N/(mm/s))

A Force that is proportional to the relative velocity between the POINTS.

The Force is opposite to the direction of motion.

For example, if the DAMPING COEFFICIENT = 4 and the **VELOCITY IS +100MM/S**, (anchor points moving away from each other), the force pulling the points together is a force of 400N.

DRAG FACTOR: (Fd α V2) (N/((mm/s)^2))

A Force that is proportional to square of the relative velocity between the POINTS. For example, air resistance.

The Force is opposite to the direction of motion.

For example, if the DRAG FACTOR = 4, and the **VELOCITY = +100MM/S** (anchor-points are moving apart), then the force pulling the anchor-points together is a force of 40000N.

INPUT AND OUTPUT-CONNECTORS

OUTPUT-CONNECTORS

The SPRING-FB has two output-connectors.

TOP :	**Distance, Velocity, Acceleration** of the dimension between the ANCHOR-POINTS.
BOTTOM :	**Total Force, Force-X, Force-Y** acting on ANCHOR-POINTS. OR **Driving-Force = Linear Motive Force**, if you configure the SPRING FB as a LINEAR MOTOR - see Configure Power Source[480]

INPUT-CONNECTOR - **see more at Spring FB dialog**[440]

The SPRING-FB has one input-connector.

Connect a wire to the input-connector of a SPRING FB to add a **Force-Function** to the **Total-Force** that is exerted by the SPRING-FB.

The units of the data-values at the input-connector must be **Force,** for example, **Newtons**.

If necessary, use a MATH FB to convert values and units to **Force units.**

For example: To convert **100mm** at the input to a MATH FB to **100N**, you must:

1. **Change the Output Data Type to Force (N) - see Math FB dialog > Output data-type**[397]

   AND

2. **Multiply the input-value by 1000 to convert 0.1m to 100N**

1.10.43 Dialog: Function-Block: Force-Data

Force-Data FB

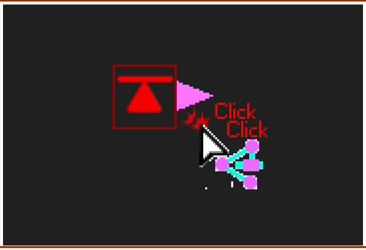
See: Add Force-Data FB [191] **, Configure Power Source** [193]

Use the FORCE-DATA FB to measure the Force that acts on a POINT at a Joint, Spring, or the Contact-Point of a 2D-CAM.

To use the FORCE-DATA FB correctly, you must also do CONFIGURE POWER SOURCE [480].

How to open a Force-Data dialog

To open a FORCE-DATA DIALOG:



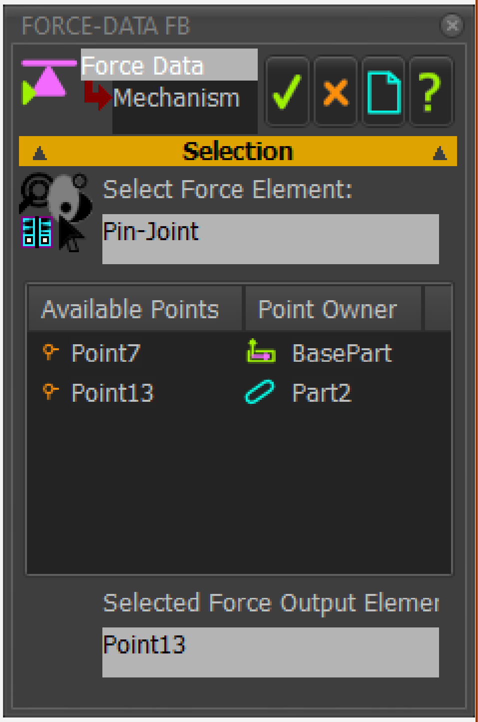
1. Double-click the FORCE-DATA FB in the graphic-area.

OR

2. **See How to Open a dialog** [513]

The FORCE-DATA DIALOG is now open. –

Force-Data dialog



Force-Data dialog

SELECTION

STEP 1. Select a Force Element:

A Force-element is a: JOINT, CAM, GEAR, RACK, BALL-SCREW **or a** SPRING

1. Click a **Force element** in the graphic-area or the ASSEMBLY-TREE.

The **Force element** should now be in the SELECT FORCE ELEMENT box

<<< in the image: **Pin-Joint** is the **Force element**

In the box below:

A list of AVAILABLE POINTS, each with a **Point Owner**

Each AVAILABLE POINT is a child to the **Force element**

STEP 2. Select a Point:

1. Click a POINT in the AVAILABLE-POINTS box, above

A POINT should now be in the SELECTED FORCE OUTPUT ELEMENTS box

The output from the FORCE DATA FB is the Force ACTING ON the selected POINT.

List of AVAILABLE-POINTS for each type of **Force element**

- **PIN-JOINT** - there are 2 **AVAILABLE POINTS**
  Each **POINT** that makes the **PIN-JOINT**.
- **SLIDE-JOINT** - there are 4 **AVAILABLE POINTS**
  Two **POINTS** on each **LINE** that make the **SLIDE-JOINT**.
- **2D-CAM** 109 - there are 0 **AVAILABLE POINTS**
  The **CONTACT POINT** is assumed.
- **SPRING FB** 194 - there are 2 **AVAILABLE POINTS**
  The **POINT** that anchors each end of the **SPRING FB**.

1.10.40 Dialog: Point Properties

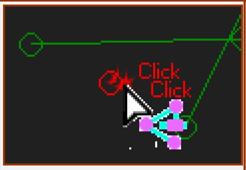
Point Properties

Use the **POINT-PROPERTIES DIALOG** to:

- Show the **Velocity Vectors** and **Acceleration Vectors** of a **POINT***.
- Find the exact X and Y coordinates of a **POINT*** in the XY-Plane of the **MECHANISM-EDITOR**.
- Find the exact x, y and z coordinates of a **POINT*** relative to the xy-axes of a **PART**.
- Edit the x, y (and z) positions of a **POINT***, in a **PART** - ONLY if the position of the **POINT*** does not have ANY dimensions or constraints.

* **POINT**, **START-POINT**, **END-POINT**, **CENTER-POINT**.

How to open the Point Properties dialog



To open the **POINT PROPERTIES DIALOG**

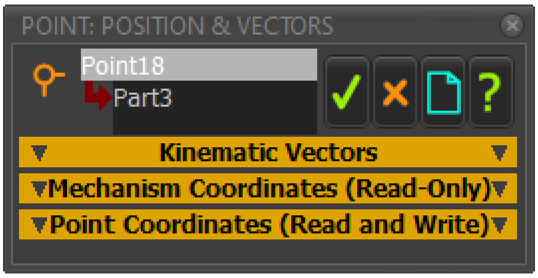
1. Double-click the **POINT** in the graphic-area

OR

1. **See How to Open a dialog** 513

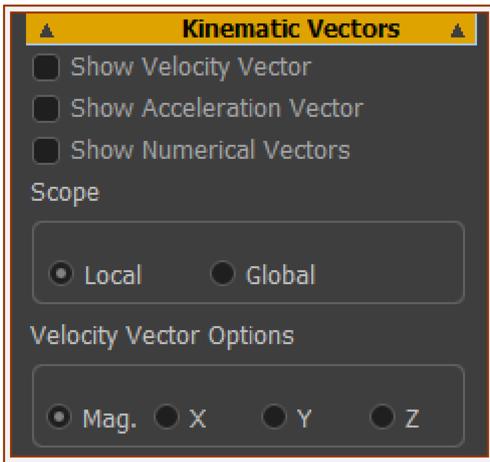
The **POINT PROPERTIES DIALOG** is now open.

Point Properties dialog



Point-Properties dialog

KINEMATIC VECTORS



VELOCITY AND ACCELERATION VECTOR check-boxes

- ❑ **SHOW VELOCITY VECTOR** - evaluated at each machine step with closed-form algorithms
- ❑ **SHOW ACCELERATION VECTOR** - evaluated at each machine step with closed-form algorithms
- ❑ **SHOW NUMERICAL VECTORS** - calculated numerically with a simple difference algorithm.

NOTE: Usually, do **not** show the **NUMERICAL VECTORS**. If you do show them, you will need to jog the MMA back and forward by a small angle so that we can calculate the values correctly. If you see there is a difference between the numerical vectors and the Kinematic Vectors, please email psmotion!

If you show **VELOCITY** and **ACCELERATION VECTORS**, or both:

- The **length** of the **VECTORS** are proportional to the **POINT'S** instantaneous **Velocity** and **Acceleration**.
- The **direction** of the **VECTORS** are in the direction of the **Velocity** and **Acceleration**
- The **magnitude** of the **VECTORS** are at the Vectors' arrowhead.

See **Edit menu >Application-Settings > Graphics > Display Colors** to edit the **color** of the vectors.

SCOPE

Calculate and display the **Velocity and Acceleration Vectors** relative to the:

- **LOCAL XYZ AXES** - **MECHANISM AXES** (default)
- **GLOBAL XYZ AXES** - **MODEL AXES**.

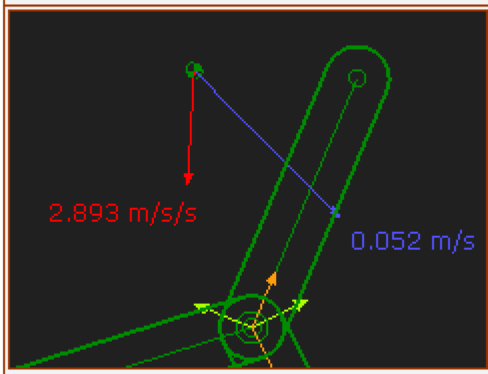
VELOCITY VECTOR OPTIONS

Display **Velocity Vectors** as:

- **MAGNITUDE** (default) of the **Velocity**, **or**
- **X COMPONENT** only of the **Velocity**, **or**
- **Y COMPONENT** only of the **Velocity**, **or**
- **Z COMPONENT** only of the **Velocity**

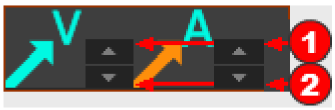
Use only the X, Y, and Z Velocity component to compare a X, Y, or Z Velocity component of **POINT** in different mechanisms.

For example, to compare the X (or Y or Z) velocity of a **POINT** on a Transfer-Tool with the velocity of a conveyor.



Scale and Length of Vectors

Edit the length of the vectors, use the buttons in the Feedback-Area [270]:

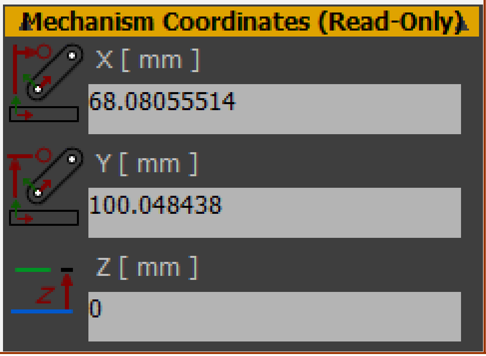


V Click ❶❷ to increase / decrease the length of the **Velocity Vector**

A Click ❶❷ to increase / decrease the length of the **Acceleration Vector**

See: App-Settings > Graphics tab [287] > **DISPLAY COLORS** | **VEL/ACC VECTORS**

MECHANISM COORDINATES (READ-ONLY)

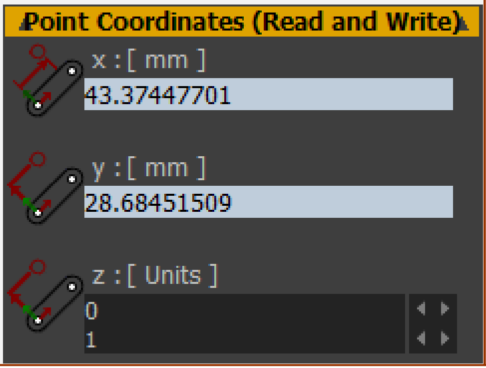


The POINT COORDINATES (X, Y, Z) relative to the **ORIGIN** of the **MECHANISM-PLANE** and BASE-PART.

The X, Y, Z dimensions are read-only.

POINT COORDINATES: READ-WRITE

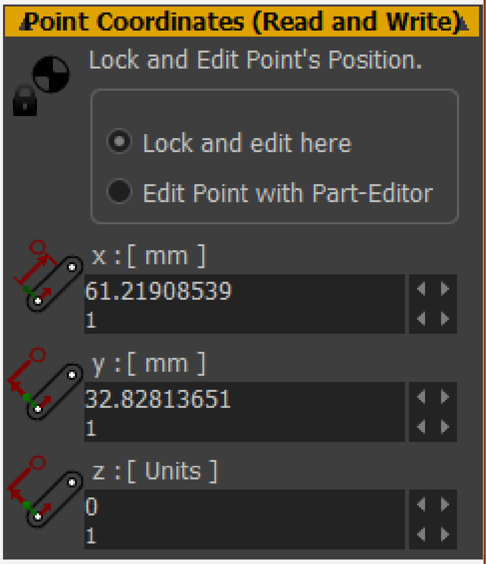
Point with Constraints



If the POINT has one or more constraint (for example, Vertical Constraint, or a Dimension), then you cannot edit its x, y coordinates with this dialog.

You can edit its z coordinates.

Point without Constraints



If the POINT does not have a constraint (or dimension) in the **PART**, then it is possible to edit the position of the POINT in this dialog

LOCK AND EDIT POINT'S POSITION:

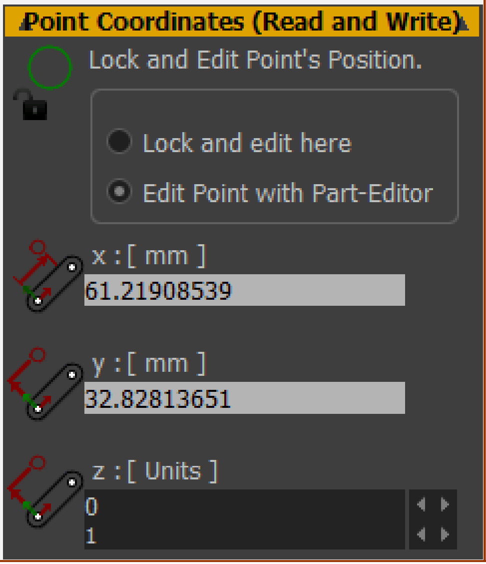
The POINT COORDINATES **(x , y, z)** are relative to the PART'S axes:

⊙ LOCK AND EDIT HERE

Edit the X, Y and/or Z coordinate of the POINT

Note:

If you click LOCK AND EDIT HERE check-box, you cannot edit the position of the POINT with the **PART-EDITOR**.



⦿ **EDIT IN PART-EDITOR**

You can edit the position of the **POINT** with the **PART-EDITOR**.

1.10.45 Dialog: Dimension

Dimension dialog

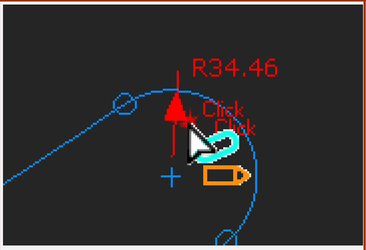
The **DIMENSION DIALOG** opens when you add a **DIMENSION** to a sketch-element.

You can, of course, edit the **DIMENSION** later.

IMPORTANT: You **cannot** edit a **DIMENSION** when a command is active - even if **Add Dimension** is active.

Deselect all commands to edit a **DIMENSION**.

How to open the Dimension dialog (in the Part-Editor)



Edit a dimesnion

To open the **DIMENSION DIALOG**:

1. De-select **ALL** commands
2. Double-Click the **arrowhead** of a **DIMENSION**

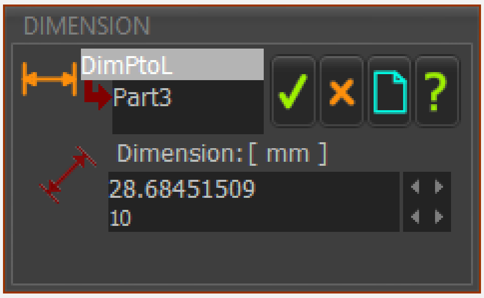
OR

1. **See How to Open a dialog** 513

Note: Nothing happens if you click the **number** - **R34.46** in the image to the left.

The **DIMENSION DIALOG** is now open.

Dimension dialog



Dimension dialog

To edit the **DIMENSION** parameter-value in the **DIMENSION DIALOG**, do one or all of these:

- Enter a **value**, and press the **enter** key (↵) on your keyboard, **OR**
- Enter an **equation**, and press the **enter** key (↵) on your keyboard, **OR**
- Use the **Spin-Box** tool, **OR**
- Right-click the data-box, and use the **Zero / Round / Copy / Paste** shortcut menu

See How to edit a parameter in a dialog 517

1.10.46 Dialog: Blend-Curve

Blend-Curve

See also **Add Blend-Curve** 236

About the Blend-Curve

A **BLEND-CURVE** is a sketch-element that has a **START-POINT** and an **END-POINT**.

Use the **BLEND-CURVE DIALOG** to edit geometric-properties **AT** its **START-POINT** and **AT** its **END-POINT**. The properties you can edit are its:

- ANGLE
- CURVATURE
- CURVATURE RATE

Typically, you merge the **START-POINT** and/or **END-POINT** of the **BLEND-CURVE** with other sketch-elements.

The default geometric-properties ensure geometric continuity 452 of the **BLEND-CURVE** with adjacent sketch-elements.

CASE 0 : You do NOT merge the **BLEND-CURVE** with another sketch-element. You can edit these geometric-properties at its **START-POINT** and **END-POINT**:

- ANGLE (default = 0)
- CURVATURE (default = 0)
- CURVATURE RATE (default = 0)

CASE 1 : You do merge the **START-POINT** and/or **END-POINT** of the **BLEND-CURVE** with a **LINE.**

- ANGLE is equal to the angle of the **LINE**
- CURVATURE is equal to zero
- CURVATURE RATE is equal to zero

CASE 2 : You do merge the **START-POINT** and/or **END-POINT** of the **BLEND-CURVE** with an **ARC.**

- ANGLE is equal to the tangent of the **ARC**
- CURVATURE is equal to 1 / Radius of the Arc (m)
- CURVATURE RATE - is equal to zero

CASE 3 : You do merge the **START-POINT** and/or **END-POINT** of the **BLEND-CURVE** with a **BLEND-CURVE.**

Edit the **END-POINT** of a **BLEND-CURVE** that you merge with the **START-POINT** of an adjacent, or following, **BLEND-CURVE**.

In the **BLEND-CURVE DIALOG**, edit the:

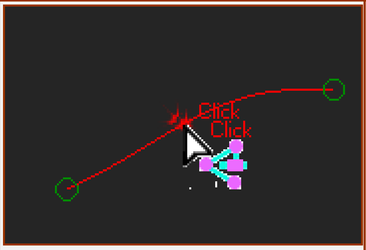
- ANGLE : Disable the **END-POINT** ANGLE check-box: edit the ANGLE - the angle of the adjacent **BLEND-CURVE** with equal the angle you enter.
- CURVATURE : Disable the **END-POINT** CURVATURE check-box: edit the CURVATURE - the curvature of the adjacent **BLEND-CURVE** with equal the curvature you enter.
- CURVATURE-RATE : Disable the **END-POINT** CURVATURE-RATE check-box: edit the CURVATURE-RATE - the curvature-rate of the adjacent **BLEND-CURVE** with equal the curvature-rate you enter.

You can edit the geometric-properties

- **Exactly** : with the **Blend-Curve dialog** 450
- **Approximately** : with the **Blend-Curve drag-handles** 451.

Edit Exactly

Open the Blend-Point dialog



In the PART-EDITOR or the MECHANISM-EDITOR:

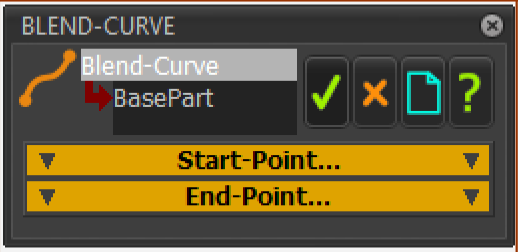
1. Double-click the BLEND-CURVE in the graphic-area

OR

1. **See How to Open a dialog** 513

The BLEND-CURVE DIALOG is now open.

Blend-Curve dialog



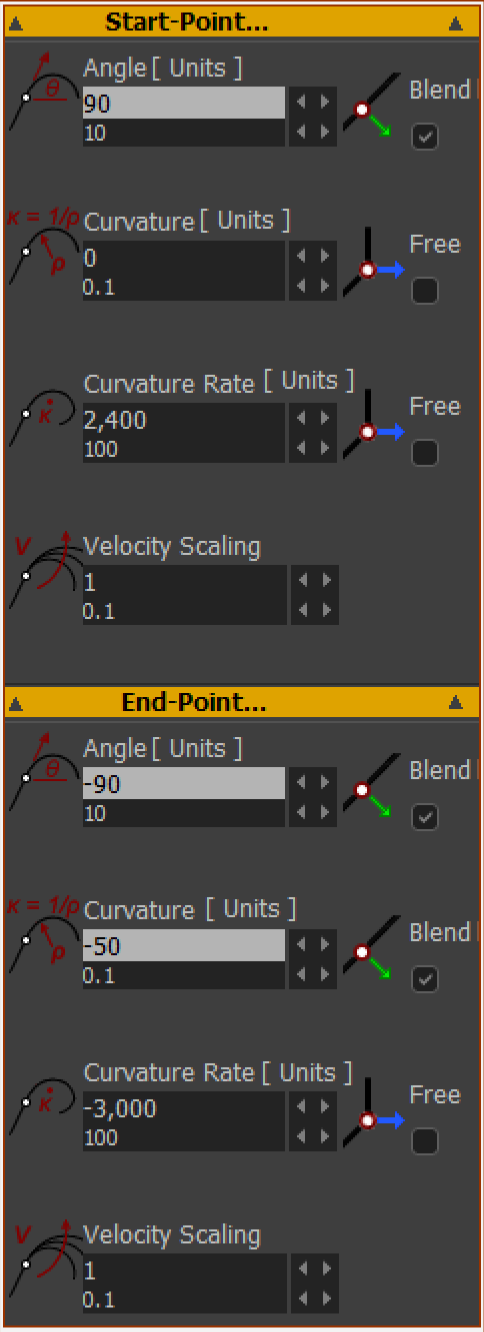
Blend-Curve dialog

There are two separators:

START-POINT...

END-POINT...

The parameters are the same for the two separators.



START POINT ... / **END POINT ...**

ANGLE (UNITS: °):

BLEND / FREE check-box:

☑ (DEFAULT) The **ANGLE** of the **BLEND-CURVE** (at its **START-POINT / END-POINT**) is equal to the **ANGLE** of the adjacent sketch-element

❑ Edit the **ANGLE** of the **BLEND-CURVE** (at its **START-POINT / END-POINT**).

CURVATURE K (UNITS:1 / METER):

BLEND / FREE check-box:

☑(DEFAULT) The **CURVATURE** of the **BLEND-CURVE** (at its **START-POINT / END-POINT**) is equal to the **CURVATURE** of the adjacent sketch-element.

❑Edit the **CURVATURE** for the **BLEND-CURVE** (at its **START-POINT / END-POINT**).

CURVATURE-RATE DK / DS (UNITS: 1 / M^2):

BLEND / FREE check-box:

☑ The **CURVATURE-RATE** of the **BLEND-CURVE** (at the **START-POINT / END-POINT**) is equal to the **CURVATURE-RATE** of the adjacent sketch-element

❑ (DEFAULT) Edit a **CURVATURE-RATE** for the **BLEND-CURVE** (at its **START-POINT / END-POINT**).

VELOCITY SCALING (NO UNITS)

Increase **VELOCITY-SCALING** to **inflate** the balloon!

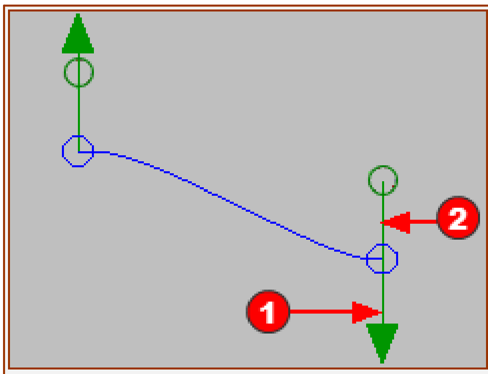
Decrease **VELOCITY-SCALING** to **deflate** the balloon!

VELOCITY SCALING does not have units.

Default = 1 ; Maximum = 10 ; Minimum 0.01

Edit Approximately

Enable / disable Blend-Curve drag-handles



Drag handles in the Graphic-Area

To show the Drag-Handles

1. Click the **BLEND-CURVE** one time

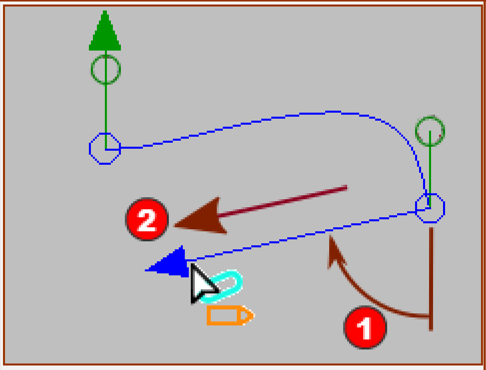
Now, you can see the **Drag-Handles** at the **START-POINT** and the **END-POINT**

There are two(2) Drag-Handle types:

❶ **Arrowhead Drag-Handle**

❷ **Circle Drag-Handle** (not easy to see)

Blend-Curve drag-handles



Arrowhead Drag-Handle

Arrowhead Drag-Handles - this controls the ANGLE and the VELOCITY-SCALING FACTOR

1. Double-click an **Arrowhead Drag-Handle** to enable it.

   The **Drag-Handle** should now be blue - it is enabled

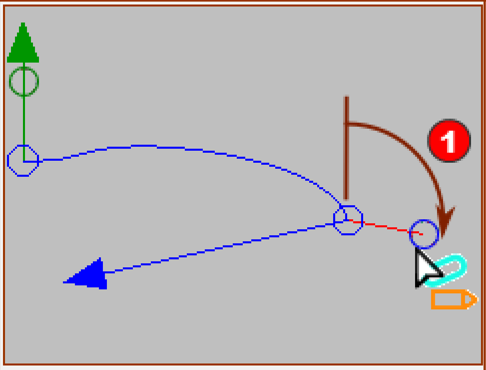
2. Drag the blue arrowhead **relative to** the **START- (OR END-)POINT:**

❶ **Rotate the arrowhead** to control the ANGLE of the BLEND-CURVE.

❷ **Move the arrowhead away** to increase the VELOCITY-SCALING FACTOR

3. Double-click the **Arrowhead Drag-Handles** again.

   The **Drag-Handle** should now be green - it is disabled.



Circle Drag-Handle

Circle Drag-Handle - this controls the CURVATURE

1. Double-click a **Circle Drag-Handle (not easy to see!)**

   The **Drag-Handle** should now be blue **(not easy to see!)** - it is enabled.

   At the default position (vertically-up), the CURVATURE = 0 (a straight-line).

2. **Rotate the** blue circle around the **START (OR END-)POINT:**

❶ In a clockwise direction to increase the **+VE** CURVATURE

❶ In a counter-clockwise direction to increase the **–VE** CURVATURE

When the blue circle is vertically-down the CURVATURE is a large +ve or –ve value.

3. Double-click the **Circle Drag-Handle** again.

   The **Drag-Handle** should now be green - it is disabled.

Note on Curvature and Curvature-Rate

Curvature and Curvature-Rate

CURVATURE (k) = 1 / Radius of Curvature

The **Radius-of-Curvature** and the **Curvature** of a simple sketch-elements is constant. For example:

- Radius of Curvature of a Line = ∞, Curvature of a Line = 0
- Radius of Curvature of an Arc or Circle = Radius, Curvature of a Circle = 1 / Circle Radius,

For example, if a circle has a radius of 20mm, its curvature = 1/0.02m = 50/m.

However, the **Radius-of-Curvature** continually changes along a general curve.

Thus, the **Curvature** also continually changes along a general curve.

CURVATURE-RATE = dk/ds

Curvature-Rate = rate-of-change of **curvature (k)** with respect to the **displacement (s)** along the curve.

1.10.47 Dialog: Import SOLIDWORKS Sketch FB

Import SOLIDWORKS Sketch FB

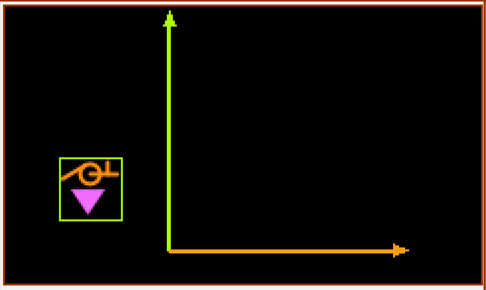
See also : Add Import SOLIDWORKS Sketch FB [239]
See also: CAD-Line > DXF tab [306]

Use the **IMPORT SW SKETCH FB** to import sketches from a SOLIDWORKS document.

Limitations:

- Cannot import Blocks, Parabolas, Ellipses, ... Safe to say, you can import Lines and Arcs.
- Limited to perhaps 20 sketch-elements.
- Construction and center-lines are imported as Lines.
- Constraints are not imported
- By default, **END-POINTS** and **CENTER-POINTS** of Lines and Arcs are Locked. To unlock, see Point Properties dialog [446].

How to open the Import SOLIDWORKS Sketch dialog



Import SOLIDWORKS Sketch FB in PART-EDITOR

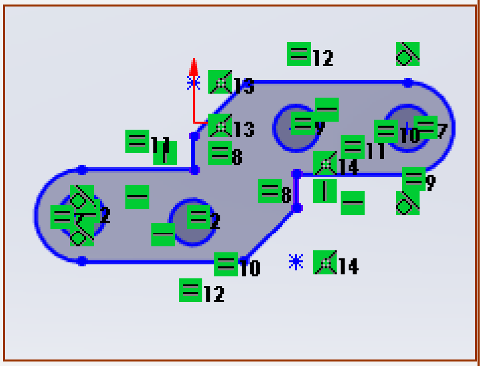
1. **MECHANISM-EDITOR**: Edit the **PART** to which you have added the **IMPORT SW SKETCH FB**
2. **PART-EDITOR**: Double-click the **IMPORT SW SKETCH FB**

OR

2. **See How to Open a dialog** [513]

The **IMPORT SOLIDWORKS SKETCH DIALOG** is now open.

Import SolidWorks Sketch dialog



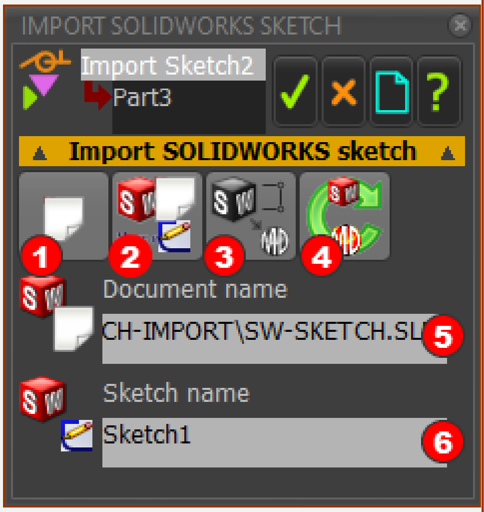
STEP 1: In SOLIDWORKS.

1. Edit a SOLIDWORKS sketch to make it **active**

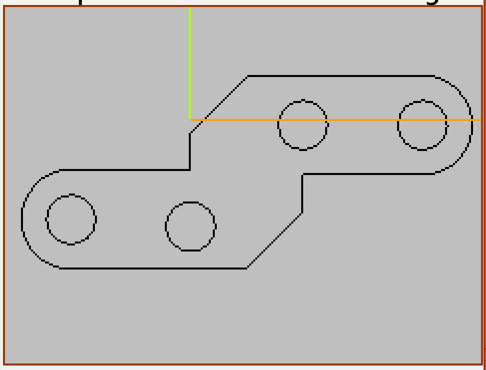
This image shows the active sketch in SOLIDWORKS.

IMPORTANT:

To import the sketch, it must be **active** in SOLIDWORKS.



Import SolidWorks sketch dialog



STEP 2. **Import the active in SOLIDWORKS**

In the **IMPORT SOLIDWORKS SKETCH DIALOG**:

1. Click button ❷.

   Document name ❺ shows the active SOLIDWORKS document (SLDPRT) name

   Sketch name ❻ shows the active sketch name, as in the SOLIDWORKS feature-tree

   Button ❸ becomes colorized.

2. Click button ❸

   WAIT until the SOLIDWORKS sketch shows in your **PART-EDITOR**.

   Button ❹ becomes colorized.

If you edit the sketch in **SOLIDWORKS**

3. Click button ❹ to Refresh, or Update the sketch

Note: To use the same **IMPORT SW SKETCH FB** to import a different **SOLIDWORKS** sketch:

4. Click button ❶ to clear the Document name and the sketch-name from ❺ & ❻

This image shows the active sketch in the **MechDesigner** graphic-area.

1.10.40 Dialog: Ball-Joint

Ball-Joint

See: Add Ball-Joint (97). Tutorial 12: Design a Spatial Mechanism

Use the BALL-JOINT for spatial mechanisms.

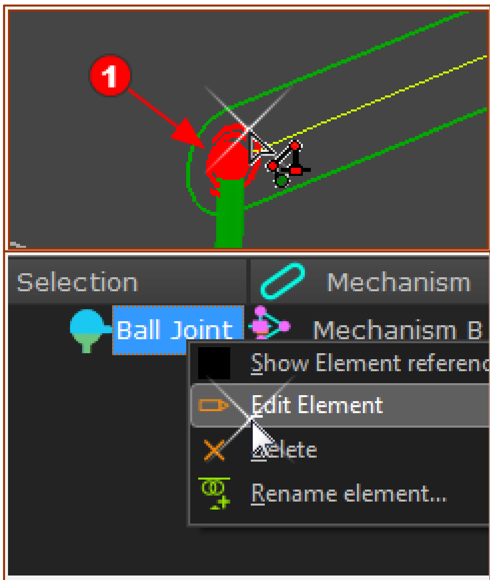
You must add a BALL-JOINT to each end of one PART.

Connecting-Part : the derived name of a PART that has a BALL-JOINT at each end.

One of the two BALL-JOINT joins the **Connecting-Part** to a PART in a different MECHANISM-EDITOR.

The other BALL-JOINT, at the other end of the **Connecting-Part**, joins it to a PART in the active MECHANISM-EDITOR.

How to open the Ball-Joint dialog



Ball-Joint in the Selection-Window

There are two BALL-JOINTS, with one at each end of a **Connecting-Part**.

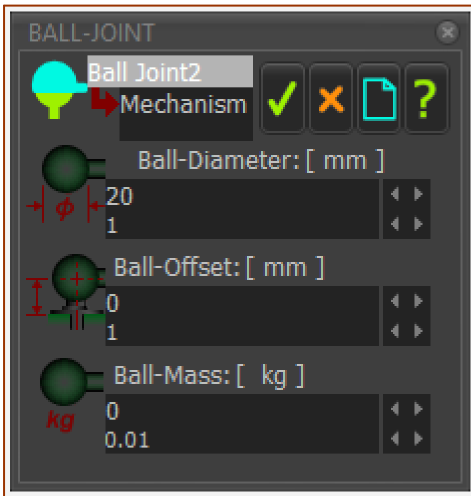
The BALL-JOINT element symbol may be ***inside*** the symbol of the **Connecting-Part**.

1. Move your mouse-pointer above the end of the **Connecting-Part** until the symbol for the BALL-JOINT is *red* - see image.
2. Click the symbol for the BALL-JOINT 1.

   The BALL-JOINT element should be in the SELECTION-WINDOW
3. Right-click the BALL-JOINT in the SELECTION-WINDOW
4. Click **Edit Element** in the shortcut menu.

The BALL-JOINT DIALOG is now open.

Ball-Joint dialog



Ball-Joint dialog

BALL DIAMETER

Diameter of the Ball symbol in graphic-area.

BALL-DIAMETER does NOT effect the Kinematic-Analysis.

BALL OFFSET (±)

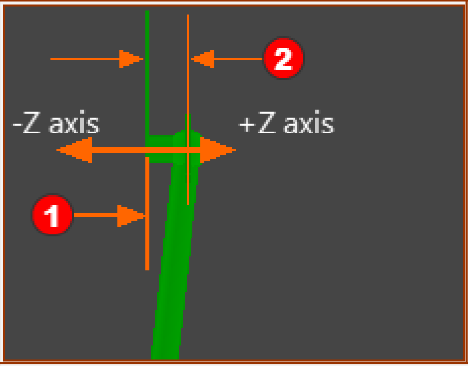
Perpendicular distance (±) from the MECHANISM PLANE to the center of the BALL-JOINT.

BALL-OFFSET does effect the Kinematic-Analysis.

BALL MASS

Mass of the BALL-JOINT.

Its center-of-mass is coplanar with the MECHANISM-PLANE. even if the BALL-OFFSET ≠ 0.



Example of Ball-Offset : Z = 10mm

<<< the image shows the model from the left view - **View toolbar > View Left** (**Shortcut : F7**).

❶ BALL-JOINT : on the **MECHANISM-PLANE** when BALL-OFFSET = 0mm.

❷ In the image the BALL-OFFSET = +**10mm** in the +Z-axis direction.

When to use the BALL-OFFSET parameter?

The symbols of all other Kinematic elements are on the **MECHANISM-PLANE**. However, the center of a BALL-JOINT is frequently ***not*** on the **MECHANISM-PLANE**. The kinematic-analysis is correct only when the distance to the center of the BALL-JOINT from the **MECHANISM-PLANE** is also correct.

See also: Ball-Joint Configurations

1.10.49 Dialog: Magnetic-Joint

Magnetic-Joint

A **Magnetic-Joint** pulls a **Circular** PROFILE to be in continuous contact with an **Irregular** PROFILE / CURVE*. The **Circular** PROFILE and the **Irregular** PROFILE / CURVE are in different PARTS and **KINEMATIC-CHAINS**.

After you add the MAGNETIC-JOINT, the motions of the two **KINEMATIC-CHAINS** are related by the contact between the **Circular** PROFILE and **Irregular** PROFILE / CURVE.

* The shape of the **irregular** PROFILE can be:

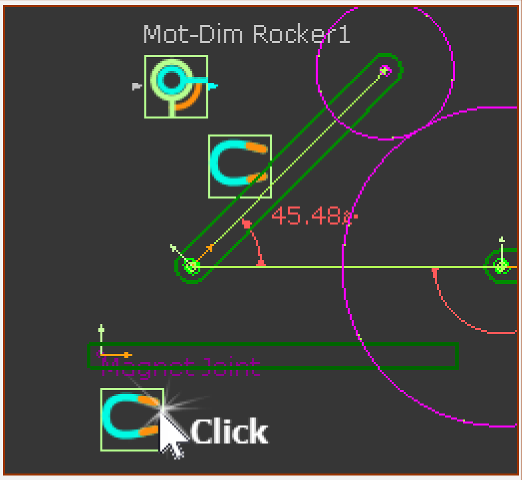
- a PROFILE that you add to a **SKETCH-LOOP**

or

- a CURVE that we calculate for you from a POINT-CLOUD 177

How to open the Magnetic-Joint dialog

See also: Add Magnetic-Joint 82.



Before you edit a MAGNETIC-JOINT:

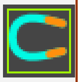
1. Click the **HOME** key on your keyboard to move the **MMA** to zero.
2. Click a MAGNETIC-JOINT FB in the graphic-area (see image to the left)

The MAGNETIC-JOINT element is in the **SELECTION-WINDOW**

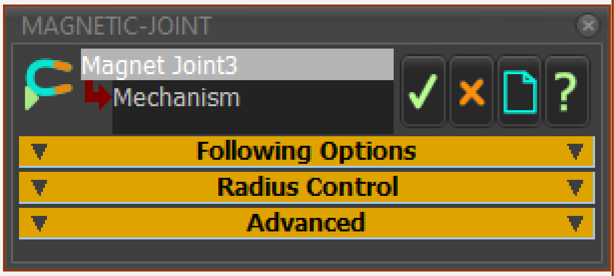
3. Right-click the MAGNETIC-JOINT element in the **SELECTION-WINDOW**
4. Click **Edit element** in the shortcut menu.

OR

1. **See How to Open a dialog** 513

Note: The icon of the MAGNETIC-JOINT has three states in the Graphic-Area.	
	Enabled Magnetic-Joint - FOUND A SOLUTION
	Disabled Magnetic-Joint - not active
	Enabled Magnetic-Joint - FAILED TO FIND A SOLUTION

Magnetic-Joint dialog

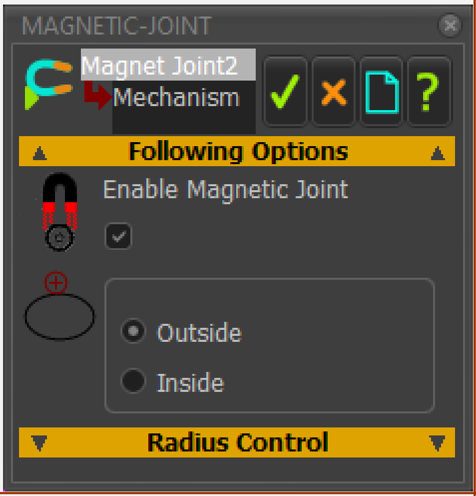


Magnetic-Joint dialog

The **Magnetic-Joint dialog** has three separators:

- FOLLOWING OPTIONS
- RADIUS CONTROL
- ADVANCED

FOLLOWING OPTIONS

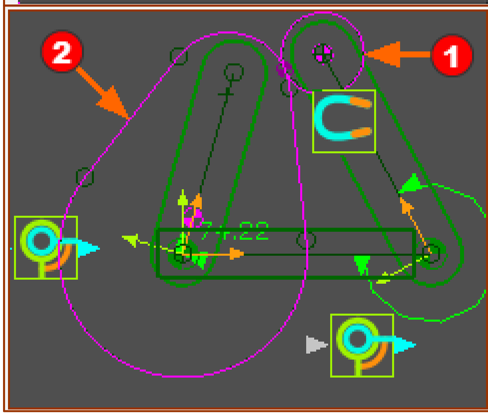


☑ ENABLE MAGNETIC-JOINT check-box

The circular PROFILE is in continuous contact with the irregular PROFILE or POINT-CLOUD [177].

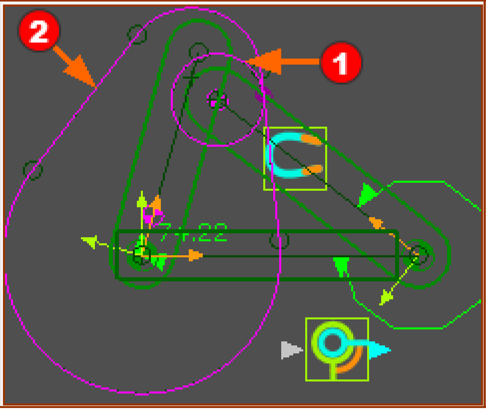
❑ DISABLE MAGNETIC-JOINT check-box.

The circular PROFILE moves to the BASE-VALUE (Linear or Angular position) as specified by the MOTION-DIMENSION FB that was in continuous contact with the irregular PROFILE.



OUTSIDE

The image shows the circular PROFILE❶ in contact with the ***out-side*** of the irregular PROFILE❷.



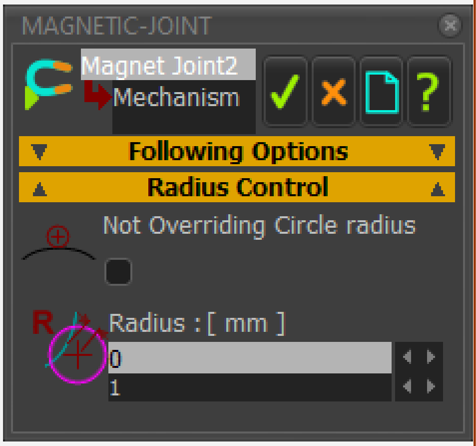
INSIDE

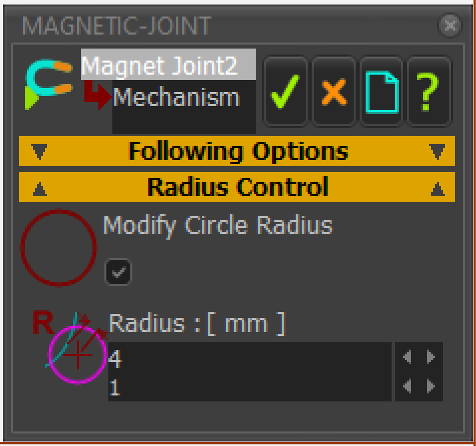
The image shows the circular **PROFILE** ❶ in contact with the ***in-side*** of the irregular **PROFILE**❷.

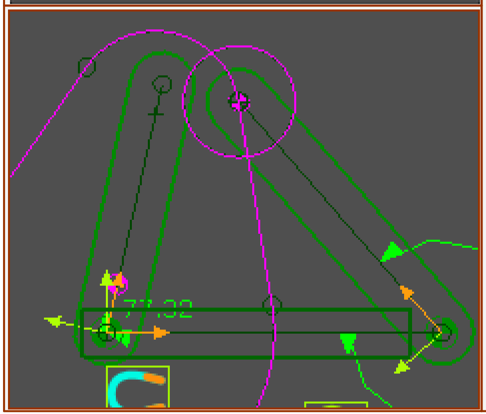
Note:

The **MAGNETIC-JOINT INSIDE** option does not operate as successfully as the **OUTSIDE**

RADIUS CONTROL







Optionally, edit the effective radius of the **Circular PROFILE**.

- ❑ **NOT OVERRIDING CIRCLE RADIUS** (default)
- ☑ **MODIFY CIRCLE RADIUS**

Enable the check-box if you want the **Circular PROFILE** to overlap the **Irregular PROFILE**.

For example:

RADIUS = 0MM

When the **RADIUS** is 0mm, the center of the **Circular PROFILE** moves along the **Irregular Profile** as normal.

RADIUS = 3MM

When the **RADIUS** is 3mm (for example), the center of the **Circular Profile** is 3mm inside the **Irregular Profile**.

This is useful if you use a CMM machine with a probe diameter that is different to the Cam-Roller, and the CMM software does not have Probe Compensation.

When the CMM data does not compensate for the Probe Diameter, you would enable **MODIFY CYCLE RADIUS** to recreate the contact between the **irregular-Profile** and the **Cam-Roller**.

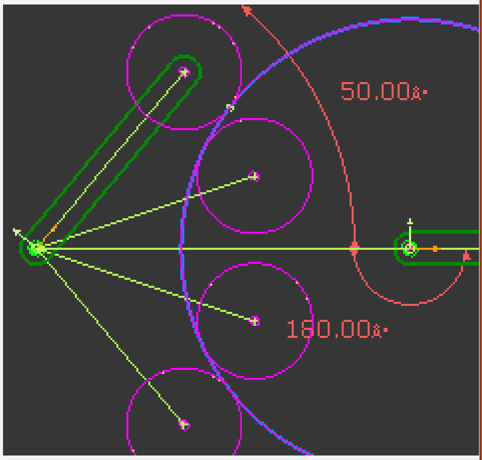
Offset Profile Radius = Radius of Actual Cam-Roller – Radius of CMM Probe

$R = R_{cf} - R_p$

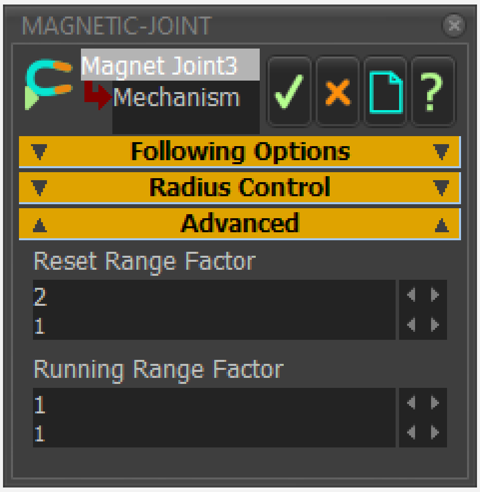
R_{cf} = **Cam-Roller Radius;**

R_p = **CMM Probe Radius.**

ADVANCED



4 Possible Contact Solutions between Circular-Profile and a simple Irregular-Profile



When you add a MAGNETIC-JOINT, the **Circular Profile** uses numerical techniques to find a contact with the **Irregular Profile**.

To humans, it is easy to see the contact that you want. But mathematically, even when the **Irregular-Profile** is a simple shape, there are usually a minimum of 4 solutions - see image to the left. **MechDesigner** may find the solution you want, or a solution you do not want, or even fail to find a solution at all.

> BEFORE **YOU ADD THE** MAGNETIC-JOINT - it is best to edit the BASE-VALUE parameter in the MOTION-DIMENSION FB, to put the **Circular Profile** within **0.5 x Radius of Circular-Profile.**

To Edit the starting position of the Circular-Profile

1. Run menu > Home **(or use the Alt+H keyboard shortcut)** to put the **MMA at 0.**
2. Open the MOTION-DIMENSION DIALOG [377] to be controlled by the MAGNETIC-JOINT.
3. Edit the BASE-VALUE to position the **Circular-Profile** to be **≤ 0.5 × Radius of the Circular-Profile** from the **Irregular-Profile.**

The factors in ADVANCED help us to find for you the solution you want.

RESET RANGE FACTOR (DEFAULT = 2)

This factor limits the range of values within which to search for a solution for the MOTION-DIMENSION, when the model is at the **HOME** position (**MMA** = **0**).

A RESET RANGE FACTOR = 1 is equal to the **1 × Radius of the Circular Profile**.

The smaller RESET RANGE FACTOR, the nearer you should edit the BASE-VALUE to put the **Circular Profile** to the **Irregular-Profile** before you add the MAGNETIC-JOINT.

RUNNING RANGE FACTOR (DEFAULT = 1)

MechDesigner must find a solution for contact between the Circular and Irregular-Profiles of the MAGNETIC-JOINT contact at all steps in the machine-cycle. There are frequently at least 4 solutions - see image, left and top.

However, the solution at any step in the machine-cycle should not be hugely different from the previous solution.

Use the RUNNING RANGE FACTOR to limit the range of MOTION-DIMENSION values within which to search for

	a solution from the previous solution, at each step of the solution cycle. RUNNING RANGE FACTOR = 1 is equivalent to **1 × Radius** of **Circular-Profile** that is contact with the **Irregular-Profile**. Reduce the RUNNING RANGE FACTOR if the **Circular-Profile** does not remain in contact with the **Irregular-Profile**, as you cycle the model.

1.10.50 Dialog: Gear-Pair

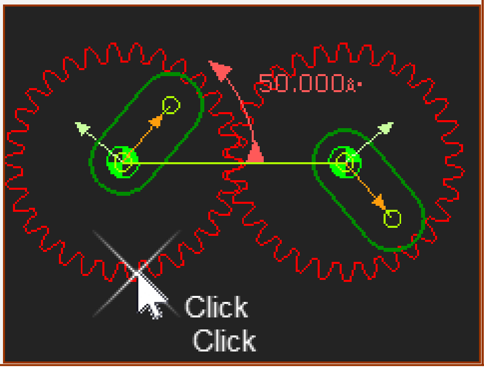
Gear-Pair

See: Add Gear-Pair 119

Terminology and definitions:

GEAR MESH:	Inter-locking gear-teeth that allow a torque and motion to be transmitted from one shaft to a different shaft.
EXTERNAL GEAR MESH :	The gears have teeth that engage and diverge out from their centers-of-rotation.
INTERNAL MESH :	One of the gears has teeth that converge in to its center-of-rotation.
SIMPLE GEAR-PAIR :	Two gears that rotate about fixed centers.
EPICYCLIC GEAR-PAIR :	One gear orbits around the center of the other gear.

How to open the Gear-Pair dialog



Edit the GEAR-PAIR:

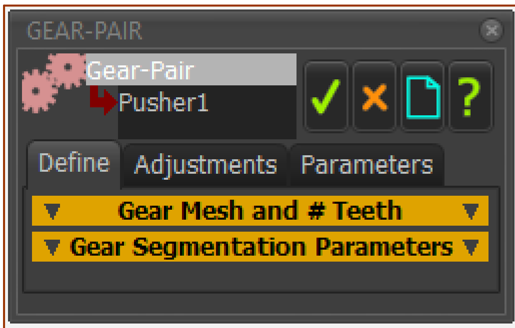
1. Double-click a GEAR-PAIR in the graphic-area or ASSEMBLY-TREE.

OR

1. **See How to Open a dialog** 513

The GEAR-PAIR DIALOG is now open.

Gear-Pair dialog



Gear-Pair dialog

There are three tabs in the GEAR-PAIR DIALOG.

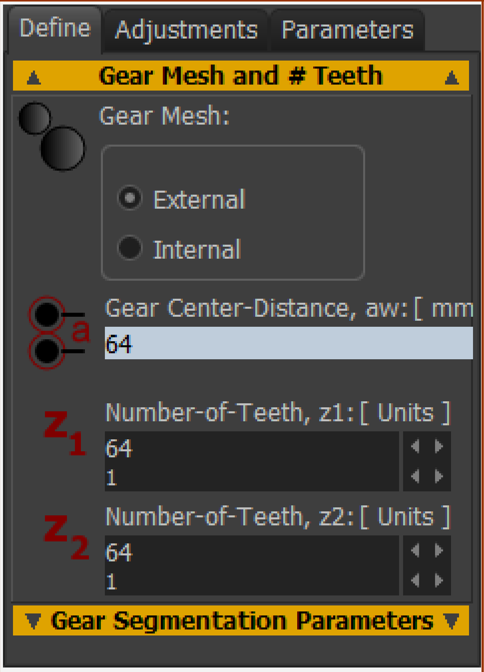
Define tab

Adjustments tab

Parameters tab

Define tab

GEAR MESH AND # TEETH



MESH - see Terminology above

- **EXTERNAL**

OR

- **INTERNAL**

Read-only

CENTER DISTANCE -

See also **CENTER-DISTANCE CALCULATION** in Adjustments tab[463]

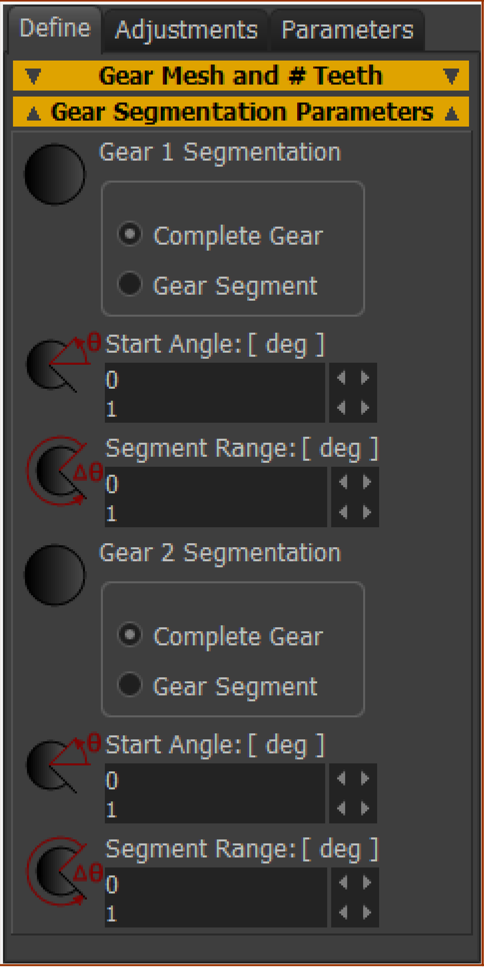
NUMBER-OF-TEETH, Z_1 (Minimum =5)

The **NUMBER OF-TEETH** on the input-gear - called the Driving-Gear

NUMBER-OF-TEETH, Z_2 (Minimum = 5)

The **NUMBER OF-TEETH** on the output gear - the Driven-Gear in the **GEAR-PAIR**.

GEAR SEGMENTATION PARAMETERS



Notes:

A **GEAR-SEGMENT** is a gear that must oscillate.

Gear 1 and/or **Gear 2** can be a **GEAR-SEGMENT**

GEAR # SEGMENTATION

- ⊙ **COMPLETE GEAR** - a normal full gear
- ⊙ **GEAR SEGMENT** - a gear that is not complete, it has fewer number-of-teeth.

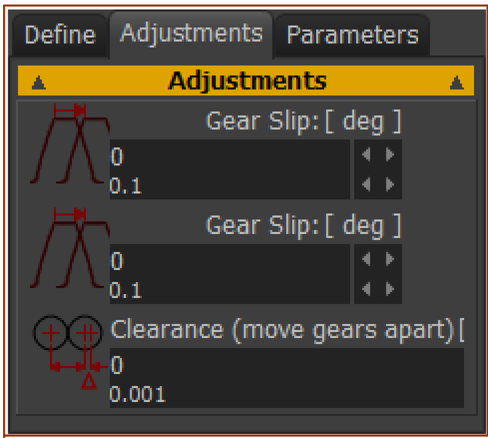
To reduce the number-of-teeth of Gear 1 or Gear 2:

1. Click to enable ⊙ **GEAR-SEGMENT**
2. Edit the **START ANGLE** : **Minimum = 0, Maximum = 360**
3. Edit the **SEGMENT RANGE** : **Minimum = 0; Maximum 360**

Each Gear has an integer number-of-teeth.

Adjustments tab

ADJUSTMENTS



GEAR-SLIP (DEGREES)

These parameters move ***Gear 1*** and ***Gear 2*** around at the PIN-JOINT.

This parameter rotates the gear teeth, not the PART.

TOP-TIP: To keep the gears in mesh:

Gear-Slip 2 = –Z1

(Z2 × Gear-Slip 1)

CLEARANCE (MOVE GEARS APART)

This parameter changes the length of the **Line-of-Centers** that is between the two Gears.

Usually, CLEARANCE is a +ve value for External Gears, and a –ve value for Internal Gears. See Notes on recommended backlash.

An alternative way to change the clearance is to reduce (by machining) the size of the gear-teeth and not to change the center-distance. We do ***not*** include for you a parameter with this deign option. However, it is the standard method to provide backlash in commercial gear-boxes. Talk with the machinist.

CENTER-DISTANCE CALCULATION

If EXTERNAL MESH, then CENTER-DISTANCE = **Clearance + (Module×(Number-of-Teeth Gear1+Number-of-Teeth Gear2)÷2)**

If INTERNAL MESH, then CENTER-DISTANCE = **Clearance + (Module×(Number-of-Teeth Gear1+Number-of-Teeth Gear2)÷2)**

Notes: Recommend Backlash/Clearance.

- Minimum: 0.006 × (center-Distance)$^{0.5}$
- Maximum: 0.024 × (center-Distance)$^{0.5}$

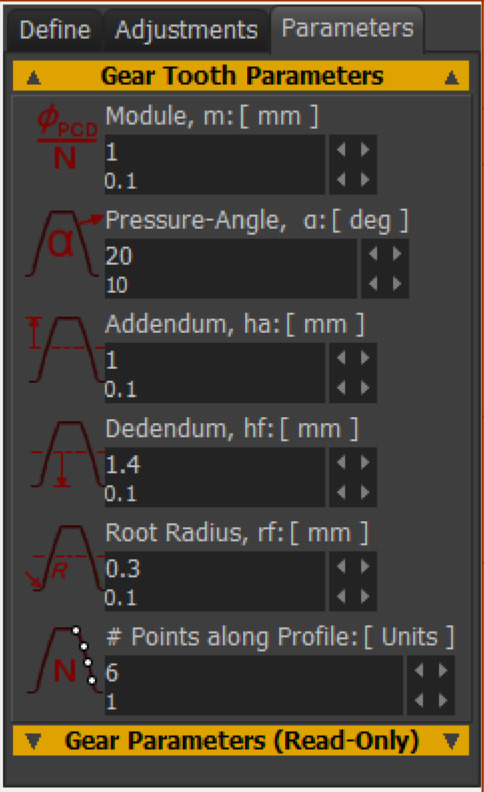
Also, I have read that:

- Minimum normal backlash = 0.03×module + 0.05 mm

If the torque reverses each machine cycle, then you should aim for the minimum recommended backlash.

Parameters tab

GEAR TOOTH PARAMETERS



Use GEAR TOOTH PARAMETERS to define the size and shape of the gear teeth.

MODULE m = P.C.D (in mm) / Number-of-Teeth.

PRESSURE-ANGLE α - default = 20º

Standard gears are 20º.

Other standards are:

- 14º, 17.5º (weaker, quieter),
- 22.5º and 25º (stronger, noisier).

ADDENDUM, ha - default = m.

The radial height of the gear tooth from the Pitch Circle to the top of the tooth.

DEDENDUM, hf - default = m × 1.25.

The radial depth of the gear tooth below the Pitch Circle to the root of the tooth.

The DEDENDUM is usually larger than the ADDENDUM to give clearance for the teeth.

If 0.25<m<1, Dedendum is usually = m × 1.4

If m>1, Dedendum is usually = m × 1.25

ROOT-RADIUS, rf - Default is 0.3 × m

The small fillet between the Flank and the Root of the Gear Tooth.

Note: In reality, if the gear is manufactured from a Hob/Rack Cutter, then the root of the gear is a Trochoid.

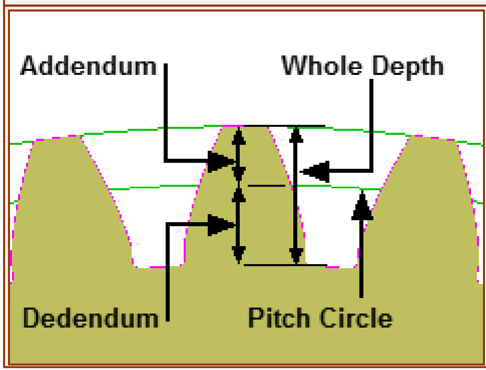
 # POINTS ALONG PROFILE

The number of 'facets' along the gear tooth flank (and around the two Root Radii).

To display Gears more accurately, increase the number of points.

Otherwise, 4 is adequate, 10 is good.

There is a CPU overhead when drawing Gear-Pairs, do not increase the number of facets without a good reason.



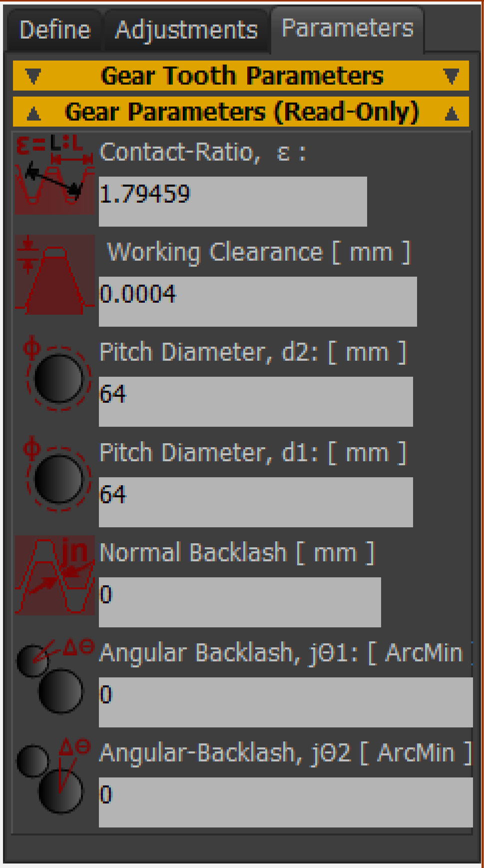
FOR INFORMATION:

Working Clearance = Dedendum - Addendum

Working Depth = Addendum × 2

Total Depth = Addendum + Dedendum

GEAR PARAMETERS



ALL READ-ONLY

CONTACT-RATIO: ε

Contact-Ratio should ***not*** be less than 1.1. **Contact-Ratio** should be a minimum of 1.2 for working gears.

WORKING CLEARANCE = DEDENDUM - ADDENDUM

The distance between the top land of a gear tooth and the bottom land of the gear with which is meshed.

GEAR 1 PCD & GEAR 2 PCD

PCD = Pitch Circle Diameter.

PCD = Module × Number-of-Teeth, (m×z)

NORMAL BACKLASH (MM)

This is the gap (backlash) between the gear-flanks you can measure with a 'feeler gauge'.

NORMAL BACKLASH is a function of the CENTER DISTANCE ADJUSTMENT [463] parameter ONLY.

ANGULAR BACKLASH - GEAR 1 (DEG) jΘ1

This parameter gives the maximum rotation of Gear 1 if you do not move Gear 2.

ANGULAR BACKLASH is a function of the CENTER DISTANCE ADJUSTMENT [463] parameter ONLY.

ANGULAR BACKLASH - GEAR 2 (DEG)

This parameter gives the maximum rotation of Gear 2 if you do not move Gear 1.

ANGULAR BACKLASH is a function of the CENTER DISTANCE ADJUSTMENT [463] parameter ONLY.

CONTACT-RATIO

The **Contact-Ratio** gives the number-of-teeth that are in contact, on average, as they pass through the meshing point. A **contact ratio** between 1 and 2 means that contact alternates between one and two pairs of teeth at any one time. Gears that have a high **contact-ratio** are smoother and quieter. The **contact-ratio** of an Internal Gear-Pair is higher than that of a similar External Gear-Pair, even greater than 2.

If the **Contact-Ratio** is too low, then consider these options:

- Decrease the pressure-angle
- Increase the number-of-teeth
- Increase the working-depth

USEFUL GEARING CALCULATIONS AND EQUATIONS

Gearing Equations

TO OBTAIN	FROM KNOWN	USE THIS FORMULA
Pitch Diameter	Module, *m*, number-of-teeth, *N*	$D = mN$
Circular Pitch	Module	$p_c = m.\pi = D.\pi \,/\, N$

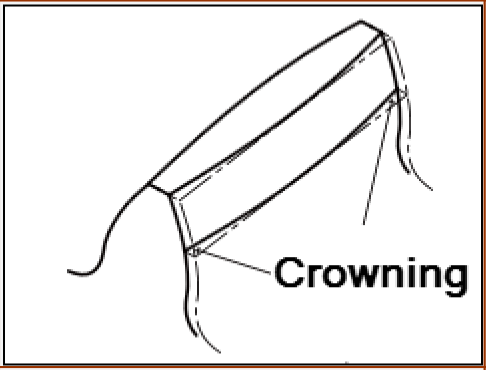
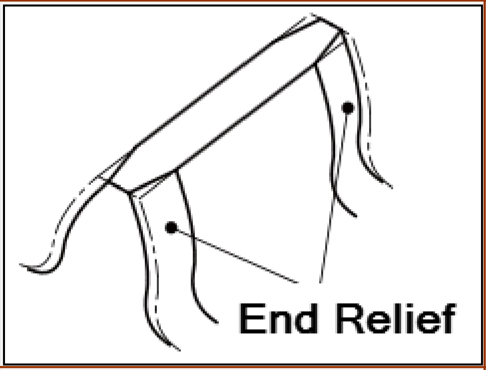
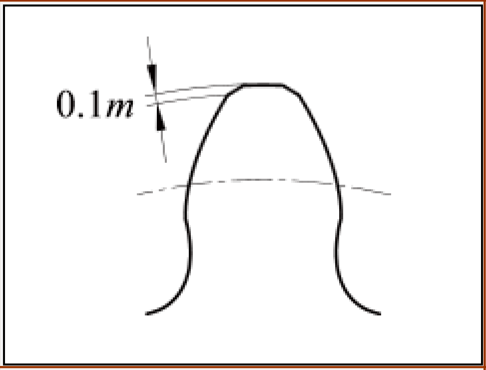
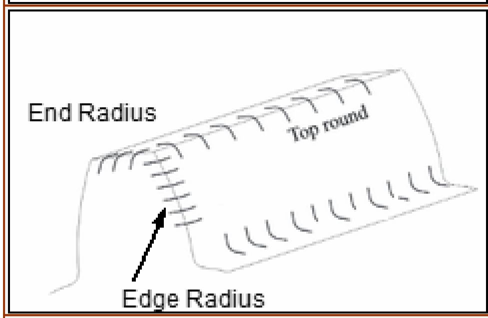
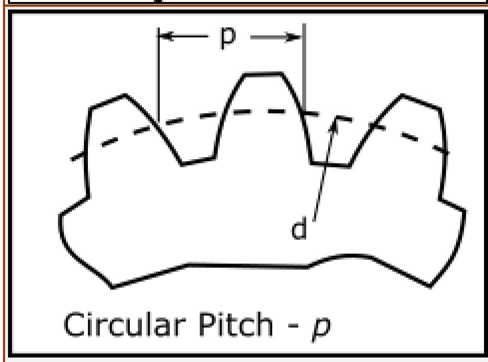
TO OBTAIN	FROM KNOWN	USE THIS FORMULA
Module	Diametrical Pitch, *Pd*	$m = 25.4 / P_d$
Number-of-Teeth	Module, *m*, Pitch Diameter, *D*	$N = D / m$
Addendum	Module, *m*	$a = m$
Dedendum	Module, *m*	$b = 1.25m$
Outside Diameter	Module, *m*, Pitch Diameter, *D*, or number-of-teeth, *N*	$D_o = D + 2m = m(N + 2)$
Root Diameter	Pitch Diameter, *D*, Module, *m*	$D_R = D - 2.5m$
Base Circle Diameter	Pitch Diameter & Pressure Angle	$Db = D \cos \mu$
Base Pitch	Module, *m, &* Pressure Angle, *μ*	$p_b = m \pi \cos \mu$
Tooth Thickness at Standard Pitch Diameter	Module, *m*	$T_{std} = \pi . m / 2$
center Distance	Module, *m*, number-of-teeth, *N*	$C = m . (N_1 + N_2)/2$
Contact Ratio for Spur Gears (1 < CR < 2)	Outside Radii, Base Circle Radii, center Distance, Pressure Angle	$CR = (\sqrt{R_{01}^2 - R_{b1}^2} + \sqrt{R_{02}^2 - R_{b2}^2} - C \sin \mu) / m \pi \cos \mu$
Backlash (linear)	Change in center Distance, ΔC	$B = 2(\Delta C)\tan \mu$
Backlash (linear)	Change in Tooth Thickness, ΔT	$B = \Delta T$
Backlash (linear) along Line-of-action	Linear Backlash along Pitch Circle, B	$B_{LA} = B \cos \mu$
Backlash, Angular	Linear Backlash, D	$B_a = 6880\, B/D$ (arc minutes)
Min. No. of Teeth for No Undercutting	Pressure Angle, μ	$N_c = 2/\sin^2 \mu$ Nc (20º) = ~17 Teeth

Useful Gearing Definitions

Term	Definition
Addendum:	the height of the gear tooth above the pitch circle diameter
Backlash:	the angle the output shaft of the gearbox can move without the input shaft moving
Base Circle:	an imaginary circle used in involute gearing to generate the involutes that form the tooth profiles
Bevel Gears:	used for right-angle applications. There are two types of bevel gears which are straight and spiral
center distance:	distance between the axes of two meshed gears - Length of the Line-of centers
Circular Thickness:	the thickness of the tooth on the pitch circle.
Dedendum:	the depth of the tooth below the diameter of the pitch circle.
Diametrical Pitch:	the teeth per inch of the diameter of the pitch circle

Term	Definition
Differential Gear:	a bevel gear which allows two shafts to rotate at a different speed.
Gear:	a wheel with teeth that meshes with another wheel with teeth to translate motion.
Gear center:	the center of the pitch circle.
Gear Train:	two or more gears meshed by their teeth. A gear train generates power speed through the meshed gears rotating
Gear Ratio:	the ratio between the numbers of teeth of meshing gears.
Helical Gear:	gear with the gear teeth cut at angles
Line of Contact:	the line or curve along which two tooth surfaces are tangent to each other
Involute:	the curve which describes a line which is unwound from the circumference of the gear
Pinion:	a small cogwheel which fits into a larger gear or track.
Pitch Circle:	the curve of intersection of a pitch surface of revolution and a plane of rotation
Pitch Diameter:	the diameter of the pitch circle
Pitch Radius:	the radius of the pitch circle
Planetary Gears:	a system that consists of three components: the sun gear, ring gear, and two or more planet gears. The sun gear is in the center, the ring gear is the outermost gear, and the planet gears are the gears surrounding the sun gear inside the ring gear.
Pressure Angle:	the angle between the line-of-action and the normal (90º, perpendicular) to the surface of the tooth
Spiral Bevel Gears:	shafts whose axes are perpendicular (90º) to each other and are used in right-angle applications
Spur Gear:	connect parallel shafts which have involute teeth that are parallel to the shaft
Sun Gear:	a gearwheel that rotates around its own axis and has other gears (planet gears) that rotate around it
Torsional Strength:	the measure of the amount of torque that a radial shaft can sustain during its rotation in a mechanical system
Working Depth:	the max depth a tooth of one gear extends into the tooth gear of a mating gear
Worm Gear:	a gear with one or more teeth with screwed threads

Gear Modifications

Image	Term
	Crowning:
	End-Relief
	Topping and Semi-Topping
	End Radius and Edge Radius/End Relief Top Round/Semi-topping
	Definition of Circular Pitch, p Circular Pitch = pitch circle circumference(pi.d) / number of teeth(z) ; $p = \Pi.d / z$ Module = Pitch Diameter(d) / Number of Teeth(z) $; m = d/z$ Circular Pitch(p) / Module(m) ; $p/m = \Pi$ Pitch Diameter(d) = module(m) × number of teeth(z) ; $d = m.z$

1.10.51 Dialog: Rack-Pinion / Ball-screw

Rack-Pinion / Ball-Screw

See: **Add Rack-Pinion** 116

Use the RACK-PINION DIALOG to edit parameters for a RACK-PINION and a BALL-SCREW.

How to open the Rack-Pinion dialog

Edit the RACK-PINION:

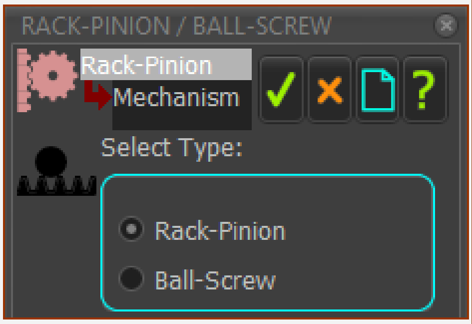
1. Double-click a RACK-PINION in the graphic-area or ASSEMBLY-TREE.

OR

1. **See How to Open a dialog** 513

The RACK-PINION DIALOG is now open.

Rack-Pinion / Ball-Screw dialog



Rack-Pinion / Ball-Screw dialog

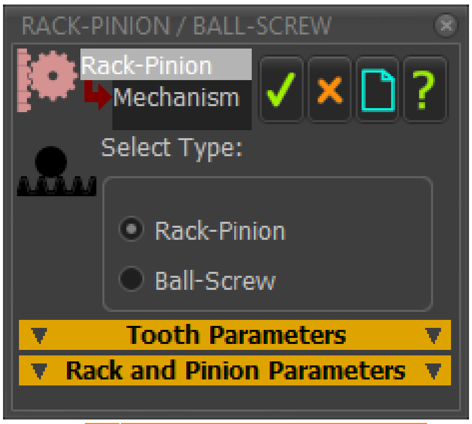
The format of the Rack-Pinion dialog changes when you enable:

- **Rack-Pinion (default)**

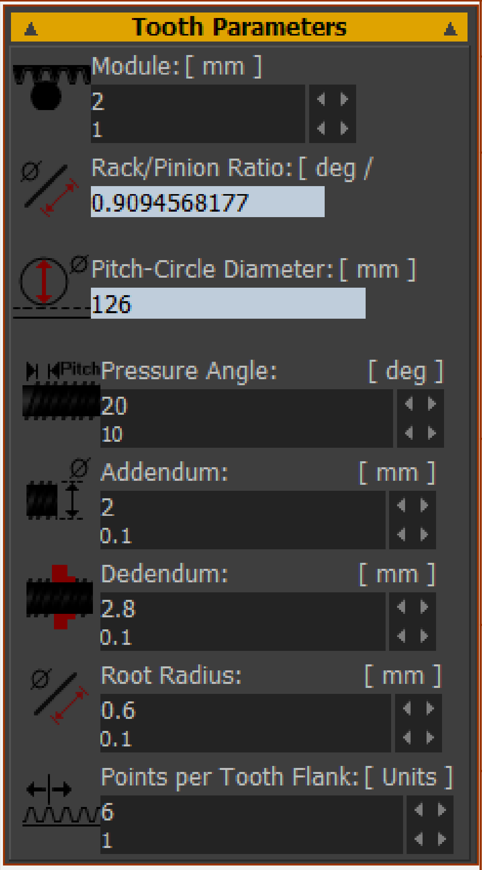
OR

- **Ball-Screw**

Rack-Pinion enabled:



TOOTH PARAMETERS



MODULE = P.C.D. ÷ **NUMBER-OF-TEETH** on **PINION**

Rack/Pinion Ratio (degrees), per Rack Displacement (mm).

Read-only:

RACK/PINION RATIO (deg/mm) - number of degrees the Pinion must rotate to move the Rack by 1mm.

PITCH CIRCLE DIAMETER (mm) - the diameter of the Pitch Circle of the Pinion. The abbreviation is P.C.D.

P.C.D. (mm) = Number-of-Teeth × Module

PRESSURE ANGLE: (Default = 20º).

The Pressure Angle of gears is usually 20º. Other standard Pressure-Angles are: 14º, 17.5º (weaker, quieter), 22.5º and 25º (stronger, noisier).

ADDENDUM: (Usually = Module).

The height of the gear tooth from the Pitch-Circle to the tip of the tooth.

DEDENDUM: (Usually = Module × 1.25 (Module>1)).

The depth of the gear tooth below the Pitch-Circle to the root of the tooth.

The **DEDENDUM** is larger than the **ADDENDUM** to give clearance for the tooth-tip of the two gears that engage with each other.

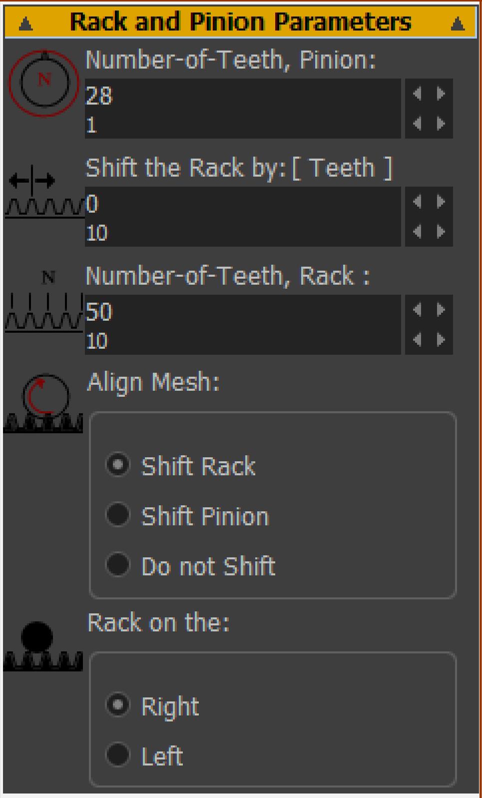
ROOT RADIUS: (Usually = 0.3 × Module)

The small fillet between the Flank and the Root of the Gear Tooth.

POINTS PER TOOTH FLANK:

The number of points on each Involute of each gear tooth. 6 is OK. More points make your model slower to edit.

RACK AND PINION PARAMETERS



SHIFT THE RACK BY:

Move the RACK along its pitch-line by # number-of-teeth.

NUMBER-OF-TEETH, RACK:

The NUMBER-OF-TEETH on the RACK.

Edit the NUMBER-OF-TEETH on the RACK so that it stays in mesh with the PINION for a machine-cycle

 # NUMBER-OF-TEETH, PINION:

The number-of-teeth on the PINION.

As you increase the number-of-teeth, the PCD increase, unless you also edit the Module.

ALIGN MESH:

When you add two(2) RACKS to one(1) PINION, you usually see two(2) PINIONS on the same axis. If you want to see one(1) PINION, click one of these options:

- ⦿ SHIFT RACK
- ⦿ SHIFT PINION
- ⦿ DO NOT SHIFT

RACK ON THE:

The positive-direction of **Rack-Pinion** and **Ball-Screw** model are specified when you select RIGHT or LEFT.

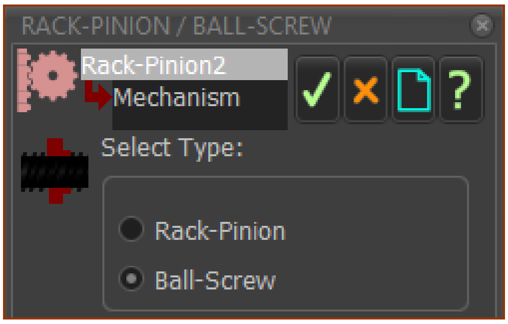
⦿ RIGHT (default)

When the PINION (Ball-Screw Shaft) rotates **Counter-Clockwise**, the RACK (Ball-Screw Nut) move in the **Positive-Direction** of the SLIDE-JOINT.

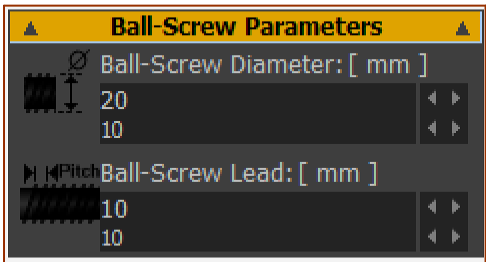
⦿ LEFT

When the PINION (Ball-Screw Shaft) rotates **Clockwise**, the RACK (Ball-Screw Nut) move in the **Negative-Direction** of the SLIDE-JOINT.

Ball-Screw enabled :



BALL SCREWS PARAMETERS



The symbol in the graphic-area for a BALL-SCREW is a **Helix/Screw.**

There are two parameters:

- BALL-SCREW LEAD (mm) - distance the SLIDING-PART moves after one(1) rotation of the ROTATING-PART.

	• **BALL-SCREW DIAMETER** (mm) - diameter of the **BALL-SCREW** symbol in the graphic-area. The length of the **BALL-SCREW** is equal to the length of the **LINE** in the **SLIDING-PART** - see Add Rack-Pinion [116]. The **Nut** (**SLIDING-PART**) of the **BALL-SCREW** moves in the **Positive-Direction** of the **SLIDE-JOINT** when the **RACK-SIDE** [471] is set to **RIGHT** - **see General tab** [471]

1.10.52 Dialog: Pulley

Pulley

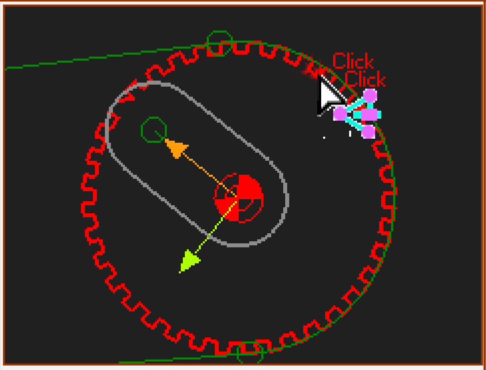
See Add Pulley [135]

A PULLEY is a wheel with teeth that engage with a toothed-belt. The belt wraps around the PULLEY.

The path of the belt is defined by a **sketch-loop** and MOTION-PATH FB.

Use the PULLEY DIALOG to edit the NUMBER-OF-TEETH on the PULLEY.

How to open the Pulley dialog



Double-Click a Pulley

To edit the PULLEY:

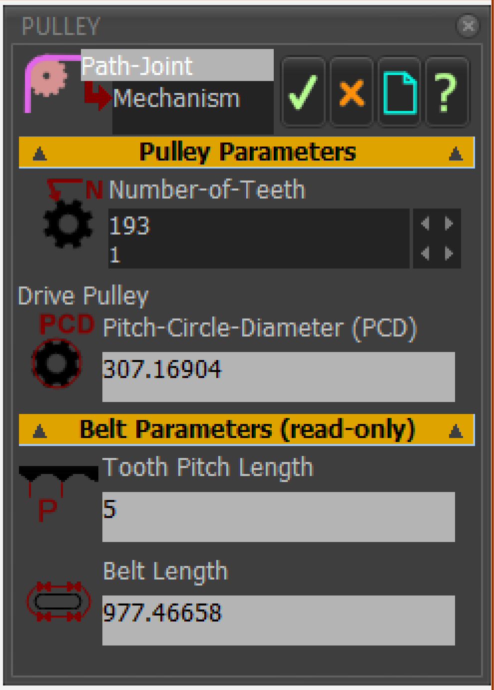
1. Double-click a PULLEY in the graphic-area or ASSEMBLY-TREE.

OR

1. **See How to Open a dialog** [513]

The PULLEY DIALOG is now open.

Pulley dialog



Pulley dialog

PULLEY PARAMETERS

NUMBER-OF-TEETH

The NUMBER-OF-TEETH around the circumference of the PULLEY.

The radius of the ARC in the sketch-path, and the P.C.D of the PULLEY updates ass you edit the NUMBER-OF-TEETH.

PITCH-CIRCLE-DIAMETER (PCD) - **read-only**

The diameter of the Pitch-Circle , which is the neutral line of the Belt.

$$PCD = (\#\,Teeth\ on\ Pulley \times Tooth - Pitch)/\pi$$

BELT PARAMETERS

TOOTH PITCH - **read-only**

The linear distance between each tooth on the belt - read-only in this dialog.

Edit the TOOTH-PITCH in the MOTION-PATH DIALOG [384].

BELT LENGTH - **read-only**

	The total length of the belt. It is equal to the length of the **sketch-loop** that wraps around the pulleys. You can control the **BELT-LENGTH** in the **PART-EDITOR**, or you can control the **BELT-LENGTH** with the MOTION-PATH DIALOG 384.

Sprocket Parameters (Apply with Transition-Curve)

Details coming soon...

1.10.50 Dialog: Feedscrew [Scroll]

Feedscrew (Scroll)

See Add Scroll 127

We have labeled the dialog as *Feedscrew* and *Scroll* because they are both common terms.

Other terms include: *Worms, Infeed Screws, Gusanos, and Tortillas sin fin.*

Use the *Feedscrew (Scroll)* dialog to:

- Set the number-of-points along the feedscrew surface for each 'rim'
- Set the Start and End - the '*range*' - of the 3D-Cam
- Edit the display of the feedscrew in the graphic-area
- Export the feedscrew to SOLIDWORKS®
- Rebuild the feedscrew with the current or active settings.
- Save the feedscrew data that defines the surface mesh.

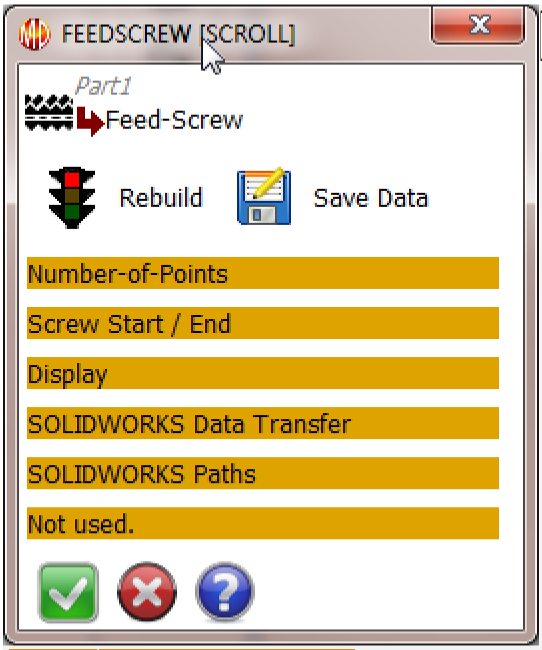
To Open the Feedscrew dialog

1. *Double-click* a FEEDSCREW in the **GRAPHIC-AREA**

OR

1. **see How to open a dialog** 513

Feedscrew dialog

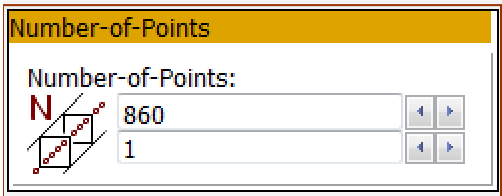


<u>Buttons at top of dialog</u>

<u>Rebuild</u> and *<u>Save Data buttons</u>*

- Use the *Rebuild* button to
  - re-calculate the points along each 'rim' when you change a parameter
  - before you use the Save Data button
  - before you transfer the Feedscrew to SolidWorks
- Use the *Save Cam* button to save the feedscrew mesh data as text files.

NUMBER OF POINTS



Preamble:

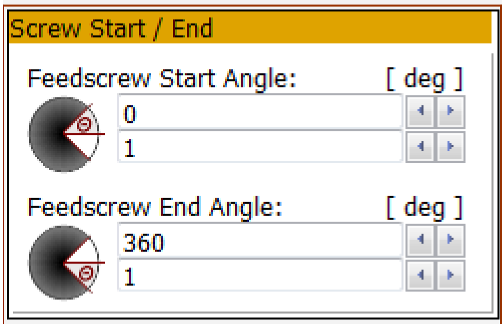
- We specify a Feedscrew with a surface.
- There are a number of lines, that we call *Rims*, along the feedscrew surface.
- There is one Rim for each Motion-Point as defined by the Motion-Path FB.
- There are a number-of-points along each *Rim*.

PARAMETERS:

NUMBER-OF-POINTS:

Number-of-points along each *Rim*. - 860 in dialog to left

SCREW START/END



Preamble:

When you add a Feedscrew, we calculate for you the surface for a complete cycle of the MMA : 0 – 360

Use this separator to edit the range of the MMA for which you want to calculate the feedscrew.

The NUMBER-OF-POINTS along each rim does not change.

PARAMETERS:

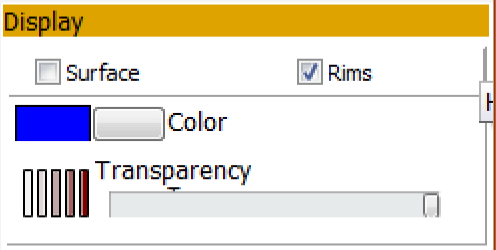
FEEDSCREW START-ANGLE

- Edit the *Start-Angle*

FEEDSCREW END-ANGLE

- Edit the *End-Angle*

DISPLAY



Preamble:

The '**MechDesigner** Display' edits how the surface of the Feedscrew shows in the graphic-area.

Controls:

'Surfaces' ; 'Rims' check-boxes

Use the check-boxes to Show or Hide *Surface*, and Show or Hide *Rims*.

Solids: show or hide, with the color in the 'color' control, and with the Transparency given by the 'Slider'.

Rims: show or hide the Rims along the Feedscrew.

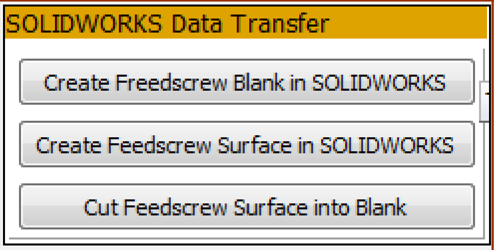
COLOR

- Use the Windows color picker to select a *color* for the Feedscrew and Rims.

TRANSPARENCY

- Use the slider to change the *Transparency* of the Feedscrew.

SOLIDWORKS DATA TRANSFER



Preamble: When you have set the correct parameters in this dialog: Number-of-Point and Start/End Angles, then you can export the Feedscrew to SOLIDWORKS. To do this

1. Open SOLIDWORKS
2. Add and Save a new Part
3. Make sure the new Part is the 'active' document in SOLIDWORKS

Buttons:

Create Feedscrew Blank in SolidWorks

- We export for you the sketch of the Feedscrew Blank and its Axis-of-Rotation'
- We instruct SOLIDWORKS for you to use the Feedscrew Blank and Axis to add a 'Revolved' feature

Create Feedscrew Surface in SolidWorks

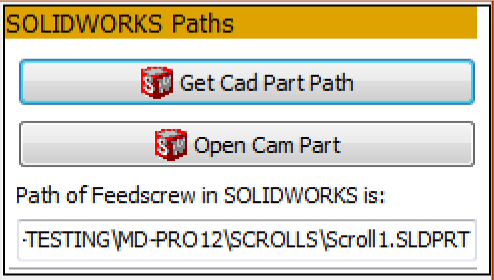
- We export for you the XYZ data of each Rim to a '*Curve*' feature
- We instruct SOLIDWORKS for you to use the *Curves* to Insert a *Boundary Surface* feature.

Cut Feedscrew Surface into Blank.

- We instruct SOLIDWORKS for you to cut the Revolved feature with the Boundary Surface

See Troubleshooting below 478.

SOLIDWORKS PATHS



Preamble:

After you have sent the Feedscrew to SOLIDWORKS, it is often useful to link the its file-name in SOLIDWORKS with the Feedscrew element that is in **MechDesigner.** At a later date, when you open this Feedscrew dialog again, you can reopen the part in SOLIDWORKS.

Buttons:

Get Cad Part Path button: Click this button to link the CAD file-name in SOLIDWORKS with the Feedscrew.

The file-name, with its full path, is put into the 'Path of Feedscrew in SOLIDWORKS is :

Open Cam Part button: Click this button to open the part with the file-name in the 'Path of Feedscrews in SOLIDWORKS is' box.

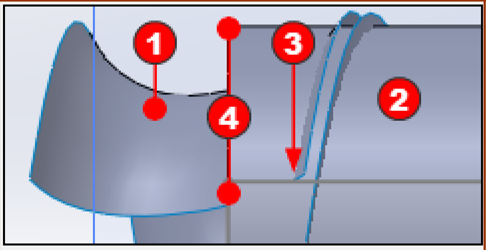
Before this is possible:

1. You have previously transferred a Feedscrew to SolidWorks and saved the file,

AND

2. You have already clicked the button 'Get Cad Part Path' button.

Trouble-shooting the Feedscrew in SOLIDWORKS

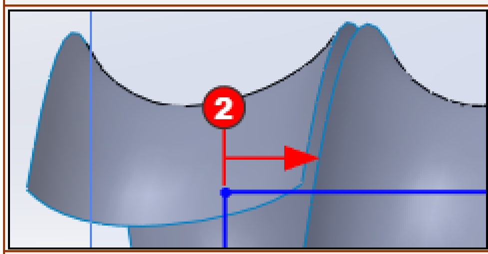


The last command: '*Cut Feedscrew Surface into Blank*' ❸ may not work.

CHECK 1: In SOLIDWORKS, inspect the length of the *Scroll Blank*❷ relative to the *Scroll-Surface*❶.

In the image to the left, the left of the *Scroll-Blank*❹ is to the left of the end of the *Scroll-Surface*❸.

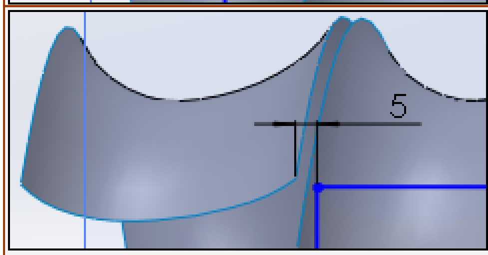
In this case, the '*Cut Scroll Surface from Blank*' (button ❸ in dialog) does not work.



You must, in SOLIDWORKS, or **MechDesigner**

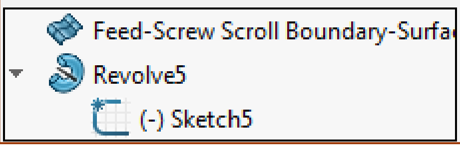
1. Edit the sketch of the Scroll-Blank❷

I prefer to edit the sketch in SOLIDWORKS.

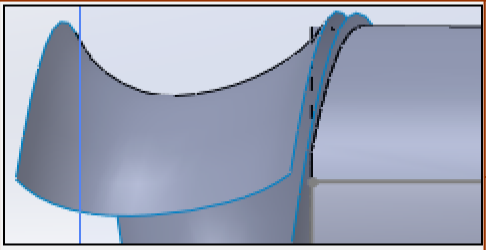
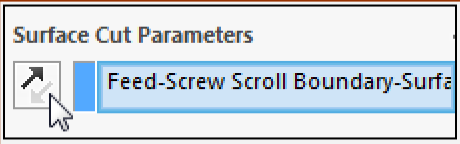
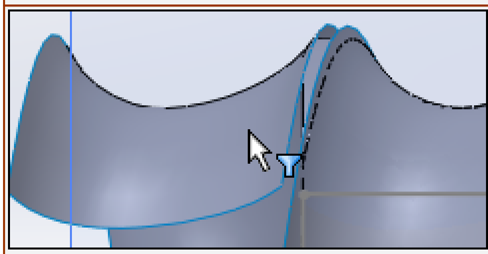
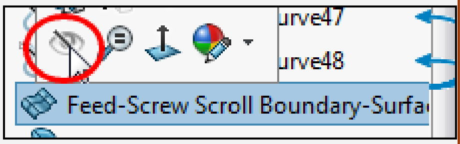
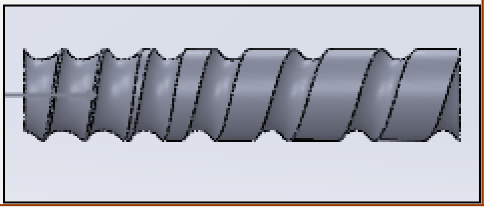


2. Move the end of the sketch to the right of the end of the Scroll-Surface**.**

In this sketch, I have added a dimension between the end of the Scroll-Surface and the left of the Sketch-Blank.



To help, I have, in SOLIDWORKS, dragged the '*Revolve*' feature to

<table>
<tr><td></td><td></td><td>below 'Feed-Screw Scroll Boundary-Surface'.
Then, as I edit the sketch for the Sketch-Blank I can see the end of the Scroll-Surface.</td></tr>
<tr><td rowspan="2"></td><td colspan="2">CHECK 2: After you close the sketch and rebuild the model. the Surface-Cut feature in SOLIDWORKS might cut to the incorrect side of the Scroll-Surface - see image to the left. In this case:</td></tr>
<tr><td></td><td>1. Edit the Feed-Screw Scroll Boundary-Surface feature in SOLIDWORKS
2. Click the direction arrow - see image- to reverse the direction of the Cut.</td></tr>
<tr><td rowspan="2"></td><td colspan="2">In the image, you can see the Scroll-Blank has been cut correctly by the Scroll-Surface.
However, the Feed-Screw Scroll Boundary-Surface feature is still shown in the model</td></tr>
<tr><td></td><td>CHECK 3: Edit the Feed-Screw Scroll Boundary-Surface feature to hide it from the display.
See image.</td></tr>
<tr><td></td><td colspan="2">Finally, the scroll is correct.</td></tr>
</table>

1.10.54 Dialog: Configure Power Source

Configure Power Source

Each degree-of-freedom in a kinematic-chain must have a **Power-Source**. The default **Power-Source** is a motor at the **MOTION-PART**.

However, frequently, the **Power-Source** is a not a **MOTION-PART**. For example, it is a **2D-Cam** that drives a **Cam-Follower** and its kinematic-chain.

You use **Configure Power Source** to make sure each **MOTION-DIMENSION** (and **degree-of-freedom)** receives its **Power** from the correct element.

When the **Power-Source** for each kinematic-chain is correct, the **Application-Load** and the force at each joint is correct.

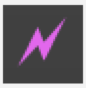
Power-Source elements can be:

- PIN-JOINT - we calculate for you the **Application-Load** and also size a Motor and Gearbox - **see Gearbox and Servo-Sizing** [483].
- SLIDE-JOINT - we calculate for you the **Application-Load** for a Linear-Motor.
- 2D-CAM - we calculate for you the **Contact-Force**, the **Cam-Life**, and the **Roller-Life** - **see 2D-Cam dialog** [327]
- CONJUGATE CAMS - we calculate for you the **Contact-Force** between at least 2 x 2D-CAMS**,** the **Cam-Life**, and the **Roller-Life**
- SPRINGS - we calculate for you the **Application-Load** for a Linear-Motor.

How to open the Configure Power Source dialog

There are three(3) methods to open the CONFIGURE POWER-SOURCE DIALOG

METHOD 1



1. Click the **KINEMATICS-TREE** in the Element-Explorer [261]
2. Click the **Kinematic-Chain** for which you want to specify the **Power-Source**

The Kinematic-Chain should be Blue.

3. Click Force toolbar [187] > Configure Power Source icon

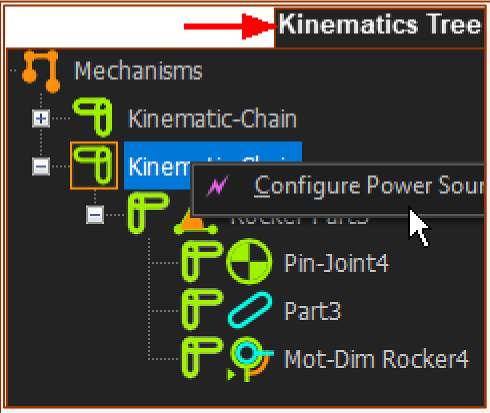
METHOD 2

In the **Kinematics-Tree:**

1. Click the **KINEMATICS-TREE** in the Element-Explorer [261]
2. Click the **Kinematic-Chain** for which you want to specify the **Power-Source**

The Kinematic-Chain should be Blue.

3. Right-click the Kinematic-Chain
4. Click **Configure Power Source** in the shortcut menu



Configure Power Source from Kinematics-Tree

METHOD 3

1. Click the **KINEMATICS-TREE** in the Element-Explorer [261]
2. Click the **Kinematic-Chain** for which you want to specify the **Power-Source**
3. Press the **ALT+P** keyboard shortcut on your keyboard.

The CONFIGURE POWER SOURCE DIALOG is now open.

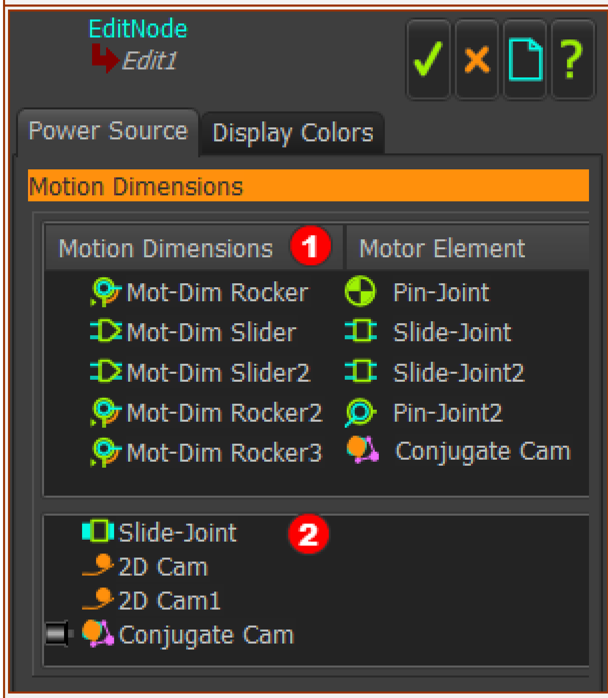
Configure Power Source dialog

You must **Configure the Power Source** for each **Kinematic-Chain**.

There are two tabs in the dialog:

Power Source tab

MOTION DIMENSIONS



Configure Power Source dialog > Power tab

MOTION-DIMENSIONS and MOTOR ELEMENTS **Box** ❶

In box ❶, even if the MOTOR ELEMENT to the right of a MOTION-DIMENSION is correct, please do **1** and **2**

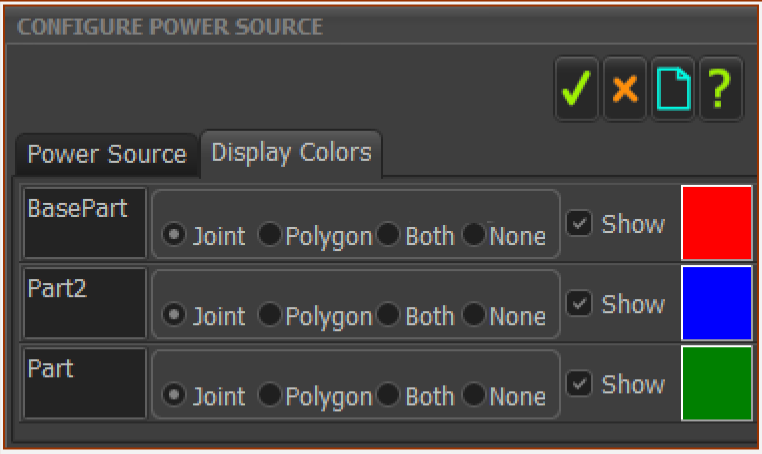
1. In box ❶ - Click a MOTION-DIMENSION - ROCKER or SLIDER - to make it **active**

2. In box ❷ - Click a MOTOR ELEMENT

The MOTOR ELEMENT in box ❷ is now the **POWER-SOURCE** for the **active** MOTION-DIMENSION that is in box ❶ .

See also : Tutorial 13 - Forces: introduction.
See also : Machine elements toolbar > Add Conjugate-Cam FB 139.

Display Colors tab



Configure-Power Source dialog > Display-Colors tab

For **EACH** PART **you can:**

Change the color of the **PART-OUTLINE** and **FORCE-VECTORS** that act on the PART.

Display the **FORCE-VECTORS** that act on the PART:

- At the Joint
- As a Polygon of Forces
- At the Joint and as a Polygon of Forces
- No Force-Vectors

These options do not help me much!

OR

- Hide the **FORCE-VECTOR** that act on the PART

1.10.5 Dialog: Servo-motor and Gearbox Sizing.

Kinetostatic Torque and Speed

Note: The options for a Linear Motor with a SLIDE-JOINT will become available in a later release.

Use the **Kinetostatic Torque and Speed dialog** to:

- Select a **Planetary Gearbox** from the major gearbox manufacturers
- Select a **Servo-motor** from the major manufactures
- Plot the **Application Load Torque-Speed** curve, in four quadrants
- Plot the **Gearbox Load Torque-Speed** curve, in four quadrants
- Plot the **Servo-motor Torque and Speed** curve; in four quadrants, find the Maximum and RMS Torque values
- Calculate the Duty-Cycle as a percentage of the machine-cycle

Note: Use this tool with one degree-of-freedom kinematic-chains. Email PSMotion if the kinematic-chain has two or more degrees-of-freedom, each with a servo-motor.

How to open the Kinetostatic-Torque and Speed dialog

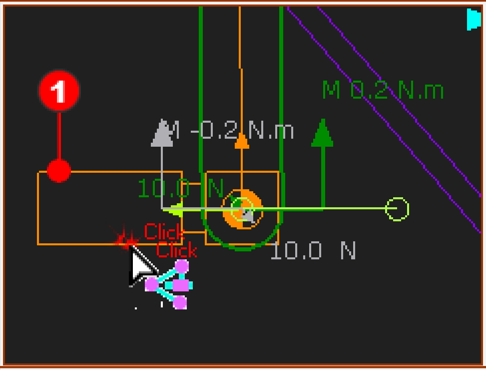
STEP 1: Enable:

- Force toolbar > Force Vectors: Calculate 190
- Force toolbar > Force Vectors: Display 198

You must be able to see the **Motor-Symbol** at a joint❶ - see image below. A **Motor Symbol** is at the PIN-JOINT that is a Power Source.

STEP 2: Move the Power Source to the correct joint

- **Force toolbar > Configure Power Source** 193



STEP 3: Open the Kinetostatic Torque-Speed dialog:

1. Click the **Motor Symbol**❶ in the graphic-area

The PIN-JOINT shows in the SELECTION-WINDOW.

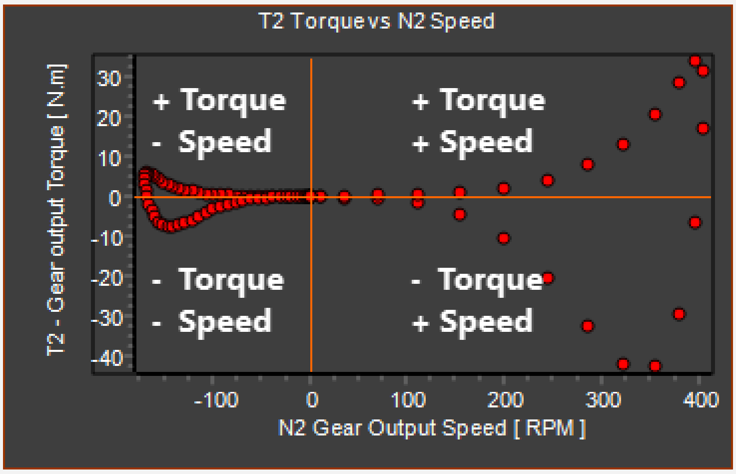
2. Right-Click the PIN-JOINT
3. Click **Edit element** in the shortcut menu.

OR

See How to open a dialog 513

The KINETOSTATIC TORQUE-SPEED DIALOG is now open.

Application (Load) Torque vs Speed plot



FOUR QUADRANT: TORQUE(LOAD) vs SPEED

Before you select a Gearbox or Servomotor, the plot is **Torque vs Speed** of the **Application Load**, in four-quadrants.

The is the **load** that the **Servo-motor and Gearbox** must drive. It does NOT include the load to drive the inertia and friction of the Gearbox and Servo-motor.

Four-Quadrant - Why 4 Quadrants?

When inertia is the dominant load, we characterize the load for you as:

❶ + Torque, + Speed - when accelerating the load in the positive direction

❷ – Torque, + Speed - when decelerating the load in the positive direction

❸ – Torque, – Speed - when decelerating the load in the negative direction

❹ + Torque, - Speed - when decelerating the load in the negative direction

STEP 1: Auto-Filter and Range-Factor

Auto-Filter check-box

The **Auto-Filter check-box** is at the top and right of the dialog.

❑ **Auto-Filter OFF**

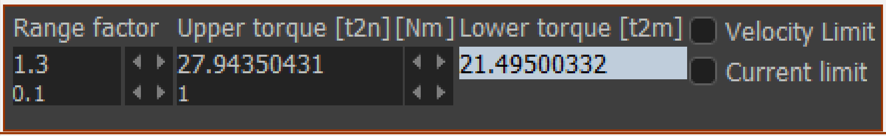
When Auto-Filter is off, you can select from 1000s of Gearboxes.

☑ **Auto-Filter ON**

We recommend that you enable the Auto-Filter. When Auto-Filer is enabled, the Gearboxes presented are limited to those that have a:

- **Lower-Torque Capacity** that is greater than the Torque required to move the Application Load.
- **Upper-Torque Capacity** that is determined by the **Range-Factor**.

The **Range-Factor** is at the top of the dialog.



Thus, the capacity of the Gearboxes that are listed for you are summarized by:

Application-Load ≤ Gearbox Capacity ≤ Application-Load × Range-Factor

Example: Auto-Filter ON, Range-Factor: 1.3

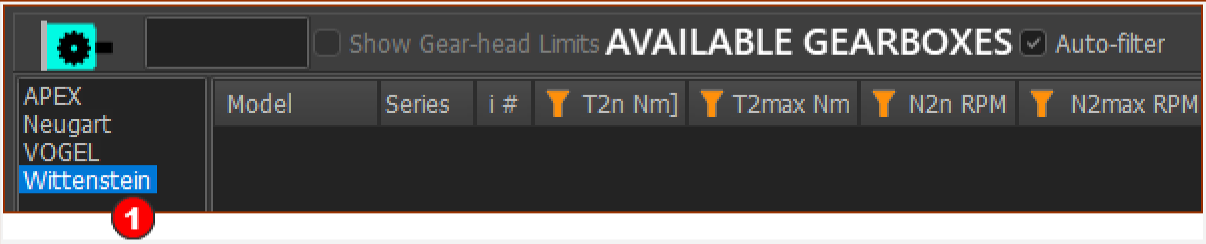
Lower Gearbox Torque Capacity = Equivalent Application (Load) Torque (21.495 N.m.)

Upper Gearbox Torque Capacity = Range Factor × Equivalent Application (Load) Torque (1.3 × 21.495 = 27.944 N.m.)

STEP 2: Select a Gearbox Manufacturer

Select a Gearbox Manufacturer ❶

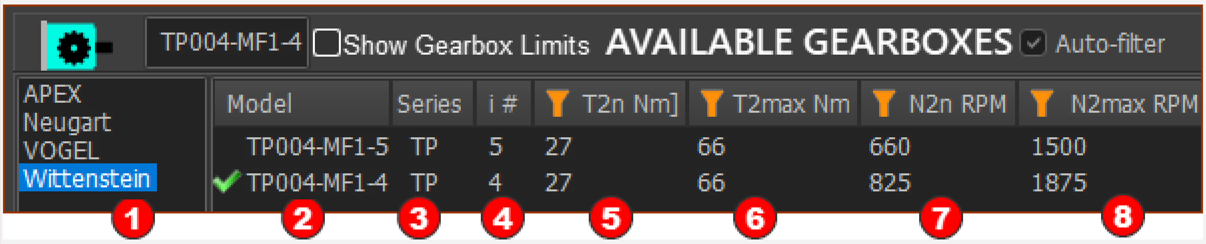
There are four Gearbox manufacturers from which you can choose: APEX, Neugart, Vogel, and Wittenstein



Note: If you need us to list a different Gearbox Manufacturer, then please email us.

STEP 3: Select a Gearbox Model

Select a Gearbox Model ❷



Gearbox manufacturer and Gearbox model part numbers

When you select a **Gearbox Manufacturer**, and **Auto-Filter is ON**, and you have entered a **Range-Factor**, the list of **Gearboxes** is limited to those whose capacity can satisfy these in-equalities:

❶	**Manufacturer**		The Selected Gearbox Manufacturer	
❷	**Model**		The Selected the Gearbox Model Part Number Each manufacturer has a different Model Part-Number format.	
❸	**Series**		Series - from the selected Model	
❹	*i*	#	Gearbox Ratio	from the Model
❺	T2N	Nm	Rated Torque referred to Output Shaft	- data sheet -
❻	T2MAX	Nm	Max. Torque referred to Output Shaft	- data sheet -
❼	n2N	RPM	Rated Speed referred to Output Shaft	- data sheet / *i* -
❽	n2MAX	RPM	Max. Speed referred to Output Shaft	- data sheet / *i* -

$T2_{max} \leq T2_{MAX} \leq RF \times T2_{max}$

Maximum Application (Load) Torque ≤ Maximum Output Torque of the Gearbox ≤ Range-Factor × Maximum Application (Load) Torque

$T2_m \leq T2_N \leq RF \times T2_m$

Equivalent Application (Load) Torque ≤ Rated Output Torque Capacity of the Gearbox ≤ Range-Factor × Equivalent Application (Load) Torque

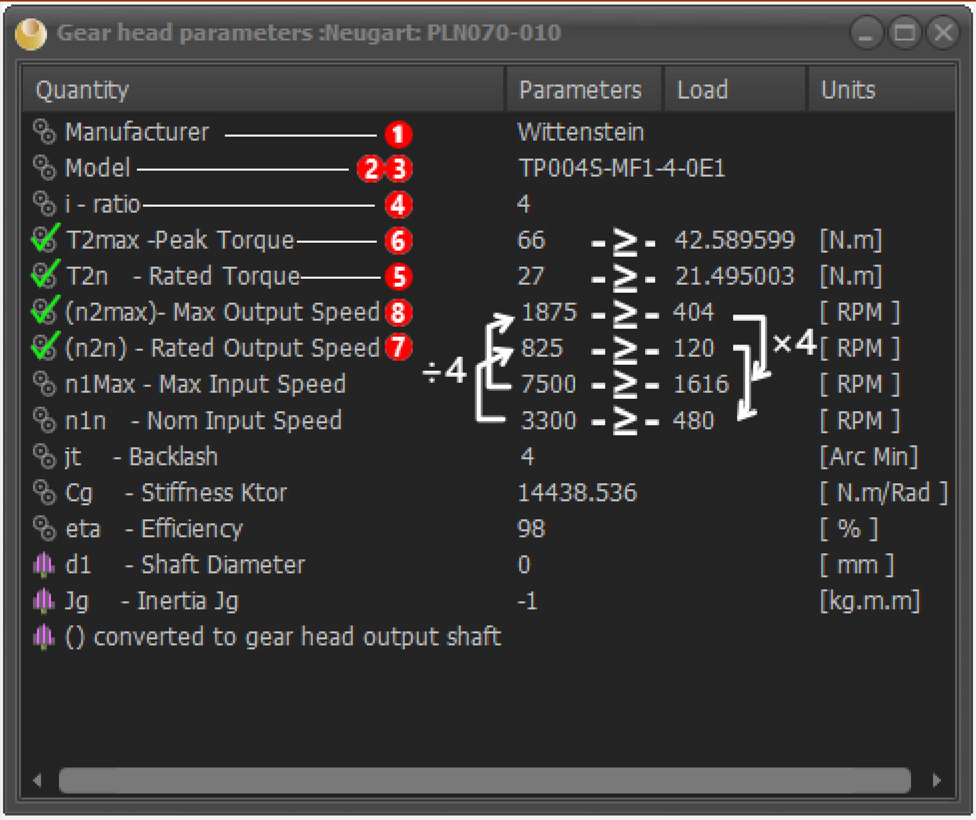
$n2_{max} \times i \leq n1_{MAX}$

Maximum Application (Load) Speed × *Gearbox Ratio (i)* ≤ Maximum Input Speed of the Gearbox

$n2_n \times i \leq n1_N$

Average Application (Load) Speed (or Equivalent, Mean) × *Gearbox Ratio (i)* ≤ Rated Input Speed of the Gearbox

Gearbox Parameters Check List.



Gearbox Parameters Summary Sheet

After you select a Gearbox model, its **Gearbox Parameters Summary Sheet** show. This form is read-only.

On the left of the sheet, there should be four(4) × ✓ to indicate that the Gearbox has the capacity to drive the Application Load and satisfies the 4 requirements.

6 Peak Torque ≥ Maximum Application Torque

5 Rated Torque ≥ Equivalent Application Torque

8 (Max Output Speed) ≥ Maximum Application (Load) Speed

7 (Rated Output Speed) ≥ Equivalent Application (Load) Speed

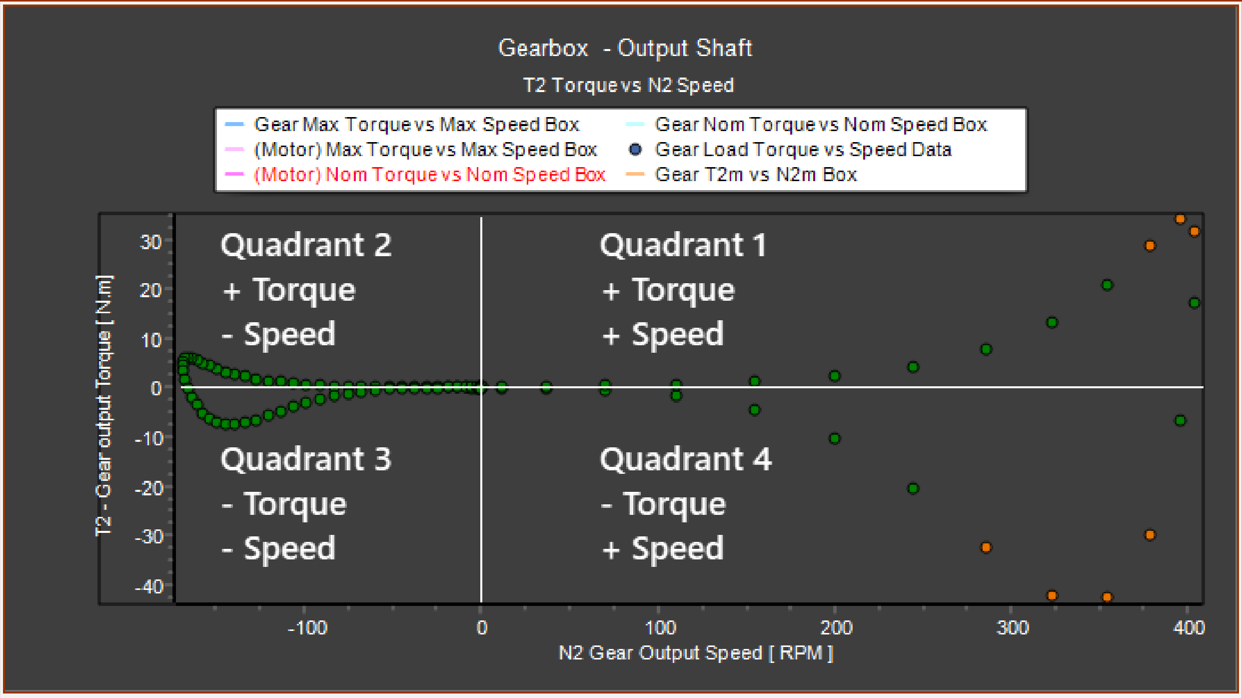
Notes: Speed parameters in brackets **( )** are referred to the output-shaft. They are calculated from the Gear-Ratio of the Gearbox.

Other Parameters include:

- Backlash
- Stiffness
- Efficiency
- Shaft Diameter - the input shaft diameter, which is not in the list until you select a Servo-motor.
- Inertia - referred to the input, which is not in the list until you select a Servo-motor.

Note: Not all Gearbox manufacturers provide all of these parameters.

Gearbox Output Torque vs Speed plot

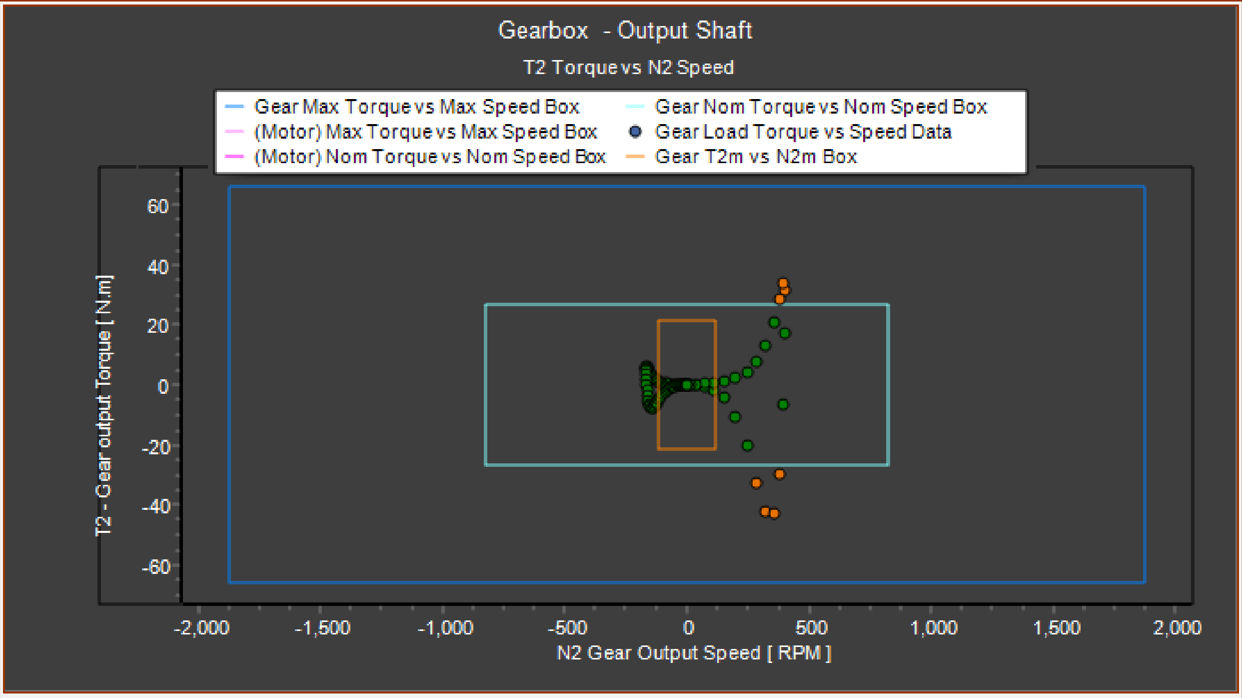


Torque-Speed plot at Output of Gearbox.

When you select a Gearbox, the Torque (N.m.) vs Speed (RPM) plot changes to indicate whether the Gearbox has the capacity to drive the Application Load.

The color code is :

- Green : Torque AND Speed of the Load ≤ Rated Gearbox Capacity
- Amber : Torque OR Speed of the Load ≥ Rated Gearbox Capacity ≤ Maximum Gearbox Capacity
- Red : Torque OR Speed of the Load ≥ Maximum Gearbox Capacity



Torque-Speed at Gearbox at Output with Gearbox Limits

Gearbox Torque and Speed Limits.

The **Show Gearbox Limits** check-list box is located above the Gearbox selection box.

❑ **Show Gearbox Limits ENABLED**

The axes are auto-scaled to the Maximum Torque and Speed, at the output of the Gearbox.

☑ **Show Gearbox Limits DISABLED**

The axes are auto-scaled to the Maximum Torque and Speed Capacity, referred to the output of the Gearbox.

This helps you compare the Torque and Speed Capacity of the Gearbox with the Application Load.

The Limits of Torque and Speed are shown as boxes.

Before you select a servo-motor, the limit boxes and their color codes indicate the:

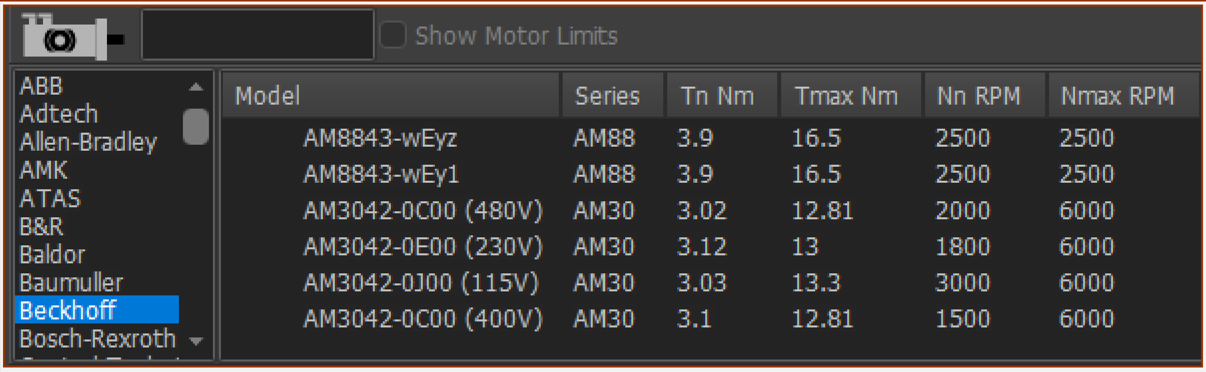
- Dark Blue : Maximum Torque and Speed capacity of the Gearbox
- Light-Blue : Rated (or Nominal) Torque and Speed capacity of the Gearbox
- Light-Brown : Equivalent Torque and Speed of the Application Load

STEP 4: Select a Servo-motor Manufacturer

Select a Servo-motor Manufacturer ❶

There are many Servo-motor manufacturers from which to select.

When you select a Servo-motor Manufacturer, we find for you those Servo-motors that have the Torque and Speed Capacity required by the Application Load.

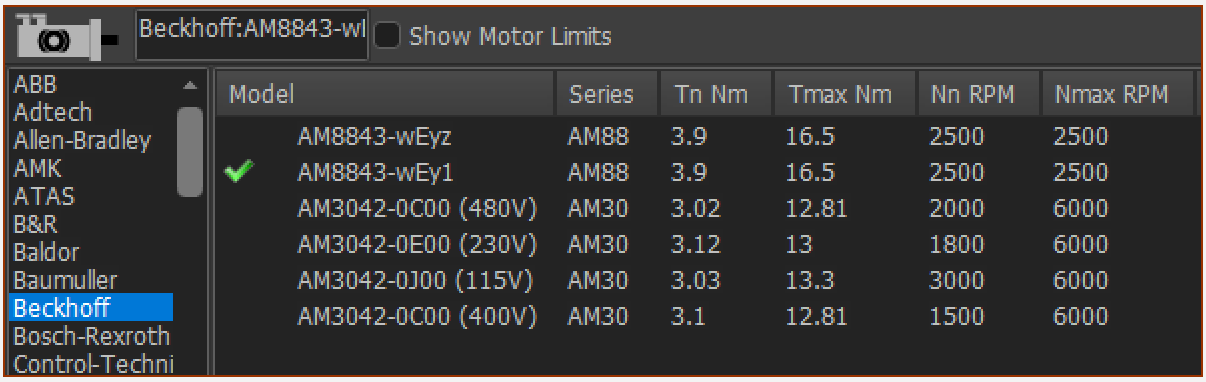


Servo-motor manufacturers and Servo-motor part numbers

Note: If you need us to list a different Servo-motor Manufacturer, then please email us.

STEP 5: Select a Servo-motor Model

Select a Servo-motor Model



When you select a **Servo-motor Manufacturer**, and **Auto-Filter is ON**, and you have entered a **Range-Factor**, the list of **Servo-motor** is limited to those whose capacity can satisfy these in-equalities:

1	**Manufacturer**		The Selected Manufacturer	
2	**Model**		The Selected the Model. Each manufacturer has a different model format.	
3	**Series**		Series - from the Selected Model	
4	***d1***	**mm**	Motor Shaft Diameter	- data sheet -
5	**Tn**	**Nm**	Rated Torque	- data sheet -
6	**Tmax**	**Nm**	Max. Torque	- data sheet -
7	**Nn**	**RPM**	Rated Speed	- data sheet -
8	**Nmax**	**RPM**	Max Speed	- data sheet -

$d1_{max} \leq D \leq \ d1_{min}$

The Servo-motor Shaft Diameter, D, must fit directly into the Gearbox, d1.

Note: Precision Planetary Gearboxes can accept a range of shaft diameters at their input. The Gearbox inertia, which is referred to its input, is different for each shaft diameter.

$T_{max} \leq T_{mot\ max} \leq RF \times T_{max}$

Maximum Application Load Torque referred to the Gearbox input-shaft

≤ Maximum Servo-motor Torque Capacity ≤

Range-Factor × Maximum Application Load Torque referred to the Gearbox input shaft

$T_m \leq T_N \leq RF \times T_m$

Equivalent Application (Load) Torque referred to the Gearbox input-shaft

≤ Rated Servo-motor Torque Capacity ≤

Range-Factor × Equivalent Application (Load) Torque referred to the Gearbox input shaft

$n1_{max} \leq N_{max}$

Maximum Application Speed referred to the Gearbox input-shaft ≤ Maximum Servo-motor Speed

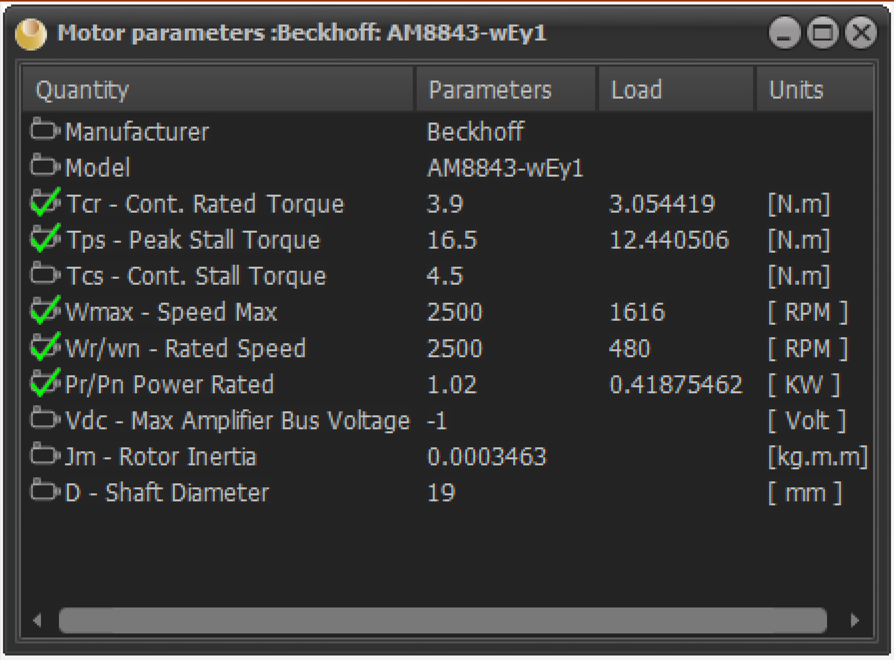
$n1_N \leq N1_N$

Average Application Speed (or Equivalent, Mean) ≤ Rated Servo-motor Speed

$Pn \leq Pr$

Application Power < Servo-motor Power Capacity

Servo-motor Parameters Check List.


Motor parameters :Beckhoff: AM8843-wEy1

Quantity	Parameters	Load	Units
Manufacturer	Beckhoff		
Model	AM8843-wEy1		
Tcr - Cont. Rated Torque	3.9	3.054419	[N.m]
Tps - Peak Stall Torque	16.5	12.440506	[N.m]
Tcs - Cont. Stall Torque	4.5		[N.m]
Wmax - Speed Max	2500	1616	[RPM]
Wr/wn - Rated Speed	2500	480	[RPM]
Pr/Pn Power Rated	1.02	0.41875462	[KW]
Vdc - Max Amplifier Bus Voltage	-1		[Volt]
Jm - Rotor Inertia	0.0003463		[kg.m.m]
D - Shaft Diameter	19		[mm]

Servo-motor Parameters Summary Sheet

When you select a servo-motor model, its **Servo-motor Parameters Summary Sheet** show. This form is read-only.

To the left of the sheet, there should be five(5) × to indicate that the Servo-motor has the capacity to drive the Application Load and also satisfies those 5 × requirements.

Peak Servo-motor Torque ≥ Maximum Application (Load) ÷ i

Rated Servo-motor Torque ≥ Equivalent Application (Load) Torque ÷ i

Max Servo-motor Speed ≥ Maximum Application (Load) Speed × i

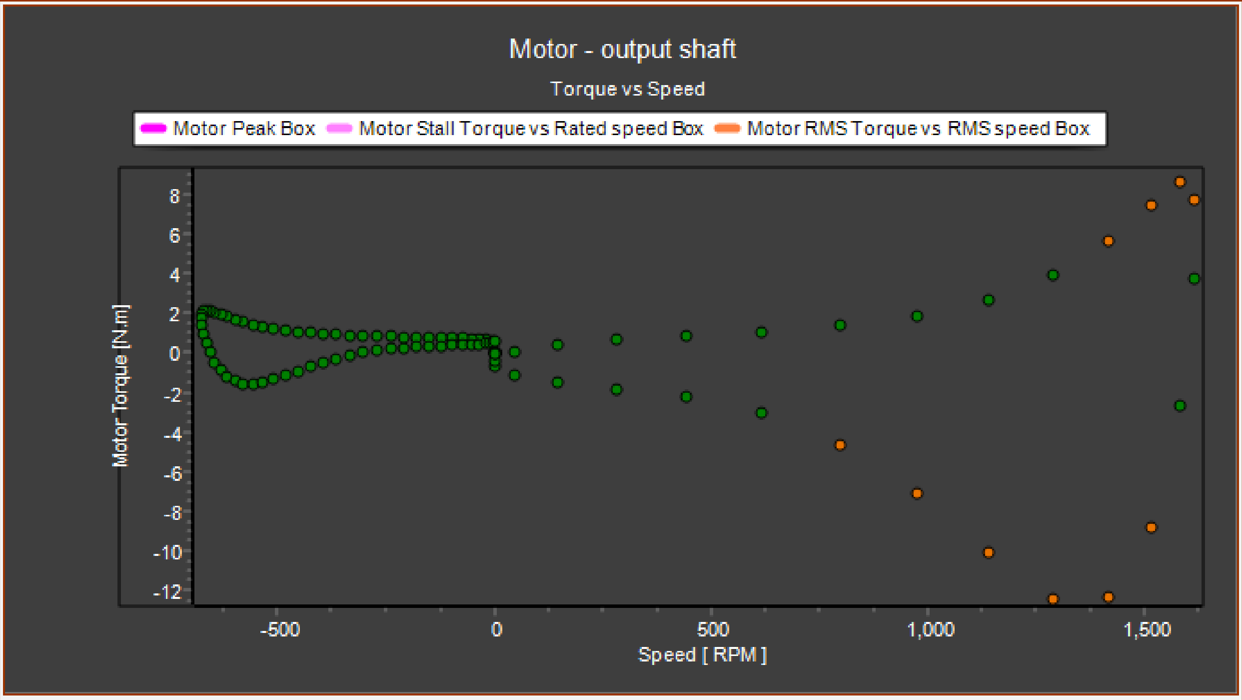
Continuous Servo-motor Speed ≥ Equivalent Application (Load) Speed × i

Rated Power > Nominal Load Power.

Other Parameters include:

- Vdc - Bus Voltage for Drive
- Jm - Servo-motor Inertia
- D - Servo-motor shaft diameter.

Servomotor Torque vs Speed plot

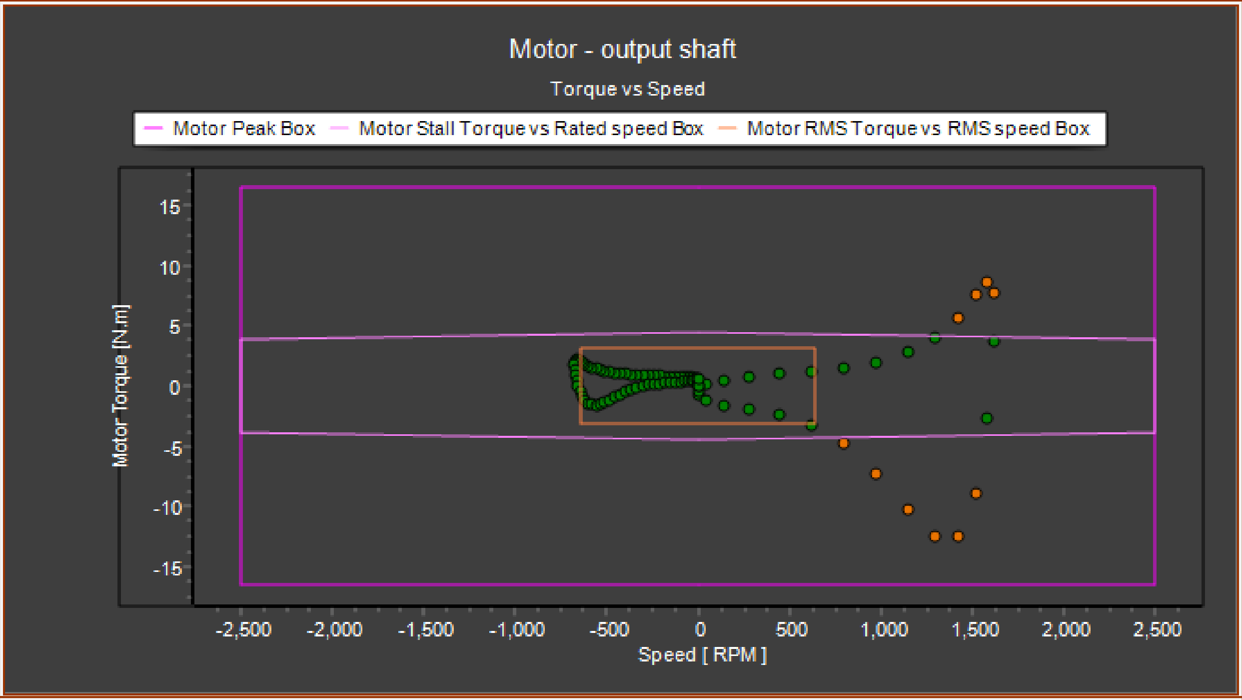


Torque-Speed plot of Servo-motor

When you select a Servo-motor, colors of the dots for the **Torque (N.m.) vs Speed (RPM)** plot changes to indicate whether the Servo-motor has the Capacity to drive the Application Load or not.

The color code is :

- **Green** : Torque AND Speed of the Load ≤ Rated Servo-motor Capacity
- **Amber** : Torque OR Speed of the Load ≥ Rated Servo-motor Capacity ≤ Maximum Servo-motor Capacity
- **Red** : Torque OR Speed of the Load ≥ Maximum Servo-motor Capacity



Motor Torque and Speed Limits.

The Show Motor Limits check-list box is located above the Motor selection box.

❑ **Show Motor Limits ENABLED**

The axes are auto-scaled to the Maximum Torque and Speed, at the Servo-motor Drive Shaft.

☑ **Show Motor Limits DISABLED**

The axes are auto-scaled to the Maximum Torque and Speed Capacity of the Selected Servo-motor

This helps you compare the Torque and Speed Capacity of the Servo-motor with the Application Load referred to its input-shaft.

The Limits of Torque and Speed are shown as boxes.

- Purple : Maximum Torque and Speed capacity of the Servo-motor
- Pink: Rated (or Nominal) Torque and Speed capacity of the Servo-motor
- Light-Brown : Equivalent Torque and Speed of the Application Load

Why is the Motor Torque is different to the Mechanism Torque

The Torque we calculate for you when you Display Force Vectors 49 is the torque to move the mechanism only. This is the Application Torque. The Application Torque does not include the Torque to move the motor and gearbox. It is necessary to add the Torque to accelerate the motor and the gearbox to the Torque in the graphic-area.

Clearly, the Torque to move motor depends on the inertia of the motor and gearbox as well as the gearbox ratio.

The higher the gear ratio, the less Application Torque the motor 'sees'. But the motor must run faster and accelerate more. This influences the overall power, as well as system efficiency.

More on Torque vs Speed Curves

The performance characteristics of a brushless servo motor (motor/drive combination) are described by a torque/speed operating envelope.

As shown below, the colored areas of the curve identify the Exceeded Duty, Continuous Duty, and Intermittent Duty working areas of the system.

Exceeded Duty

To maximum speed and/or the maximum torque, of the gearbox and/or Servo-motor must be exceeded. Usually, to make sure this cannot happen, there is a Current (amps) Limit on the Power Drive.

Continuous Duty Zone (S1)

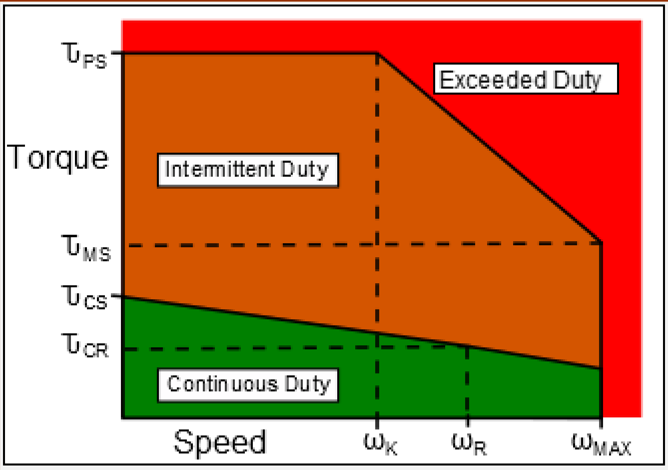
The continuous duty zone is bounded by the maximum continuous stall torque up to the intersection with the intermittent duty line. The continuous torque line is set by either the motor's maximum rated temperature, or the drive's rated continuous current output, whichever is less.

The system maximum continuous or 'voltage line' is set by the voltage rating of the drives, the line voltage supplied, and the motor winding.

The system can operate on a continuous basis anywhere within this area, assuming the ambient temperature is 40°C or less.

Intermittent Duty Zone (S5)

The intermittent duty zone is bordered by the peak-stall torque and the system voltage line. The peak torque line is set by either the drives' peak current rating, which the drive can give for a limited time, or the maximum rated peak current for the motor, whichever is less. Higher torque levels may be achievable at higher power levels.



Typical Torque / Speed Duty Capability of a Brushless Servomotor.

Peak Torque: (τ_{PS}) The Peak Torque the Motor and Gearbox at Stall Speed

Continuous Stall Torque: (τ_{CS}). The Continuous Torque the Motor and Gearbox can give Continuously at Stall Speed.

Maximum Speed: (ω_{MAX}) Maximum possible speed of the Motor and Gearbox. Not attainable when the voltage is limited by the drive.

Knee Speed: (ω_K) The Speed at 'knee' in peak envelope that is the intersection of the Peak Torque with the Voltage Torque/Speed Limit Line.

Continuous Rated Torque:(τ_{CR}). The Continuous Torque at the Speed of the Rated Power.

Rated Speed: (ω_R) The Rated Speed or Speed at Rated Power. The motor can operate at this speed with the supply voltage.

How we calculate the Motor Torque and Motor Speed

The Reflected Inertia at the Motor shaft usually continuously changes in a machine-cycle.

With **Constant Inertia Mechanical Systems** it is easy to calculate **Inertia Torque**.

With mechanisms, the Torque is dependent on reflected inertia that is a function of Acceleration, Velocity, and Position.

PSMotion has developed algorithms to calculate these, which give a true indication of the reflected inertia at a motor shaft for even the most complex mechanisms.

The equations below, are calculated at every instant in a machine-cycle.

Usually, you aim to make the **Reflected Load Inertia = (Motor + Gearbox) Inertia.**

However, when the **Load Inertia** is not constant, it is more difficult to select the Motor and Gearbox.

Speed:

$\omega_m = N \times \omega_L$

$\alpha_m = N \, . \, \alpha_L$

Torque

$T_m = - \alpha_m (J_G + J_M) + sign(T_{VD})$

$T_L = (T_{MD} \,/\, N) \,/\, \eta$

$T_T = T_m + T_L$

N = Gear Ratio
ω_m = Motor Angular Velocity
ω_L = Load Angular Velocity
T_T = Total Torque
T_m = Motor Torque
T_L = Load referred to Motor Shaft
T_{VD} = Viscous 'Drag' Torque. It is always opposite to the direction of motion
T_{MD} = Torque derived by your model at the Motor Shaft ($f\{P_L, \omega_L, \alpha_L\}$)
J_G = Inertia of Gearbox
J_M = Inertia of Motor
α_m = Motor Acceleration
α_L = Load Acceleration
η = Gearbox Efficiency

1.10.50 Dialog: Make Movie

Make Movie

Background:

Commercial video-recording software may not record each step in a machine-cycle, as the model cycles in **MechDesigner**.

Make-Movie solves these problems.

About Make Movie

Use **Make Movie** to record :

- a movie as a GIF file-type.

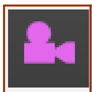
OR

- a sequence of images as PNG, JPG, or BMP file-types.

The file-name of each image is appended with a number.

Also: You can take a **Single Snap-Shot** of the model at its active Machine Angle.

How to open the Make-Movie dialog



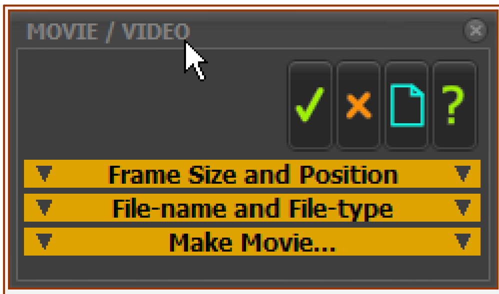
1. Click **File toolbar > Make Movie**

OR

1. Click **File menu > Movie**

The **MAKE-MOVIE DIALOG** is now open

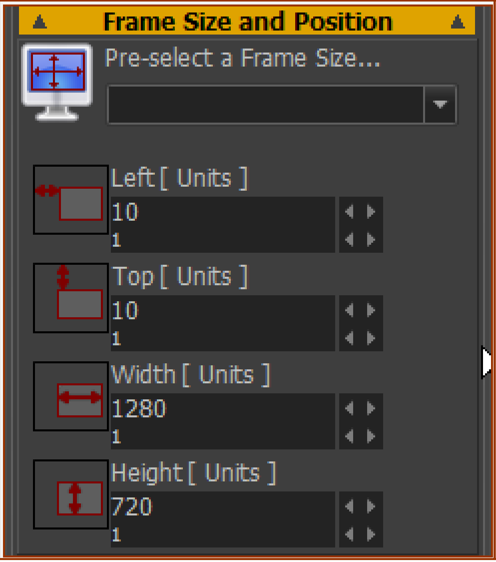
Make-Movie dialog



Make-Movie dialog

Top-Tip - Before you open the dialog, do **ALT+H** to move the **MASTER MACHINE ANGLE** to zero(0).

FRAME SIZE AND POSITION



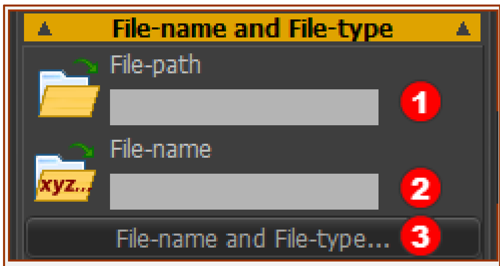
You should see a rectangle in the Graphic-Area. This is the **FRAME** that you record. Usually, edit the size and position of the **FRAME**.

1. Use the **PRE-SET FRAME SIZE**...drop-down list-box to select a typical frame size.

OR

1. Edit **WIDTH** and **HEIGHT** to the frame.
2. Edit the **LEFT** and **TOP** of the frame.

FILE-NAME AND FILE-TYPE



1. Click the **FILE-NAME AND FILE-TYPE...** button ❸ (at the bottom of this separator)

In the **Windows® Save as** dialog:

a. Browse to a **File-path** and enter a **File-name (see Top-Tip, below)**

b. Use the **Save as type** list-box to select a file-type.

Note: different file-types have different results - see **FILE-TYPES** below

c. Click the **Save** button

The **FILE-PATH** and **FILE-NAME** show in the boxes. ❶❷

Top-Tip: Do **not** enter a **file-name** that ends with a number.

FILE-TYPES

GIF file-type.

To make a movie with the GIF file-type.

GIF files can be large when the frame-size is large.

PNG, JPG, JPEG, BMP file-types.

To save one image for each frame - **see** **Number-of-Frames** 497

We append for you **001**, **002**, **003**, ... to the name of each image. For example, **HINGE-A000.PNG, HINGE-A001.PNG, HINGE-A002.PNG, ... "HINGE-A"&"NUMBER-OF-FRAMES".** The numbers help you and other software sort the images.

To compile **PNG** images as a movie, I use **Avidemux 2.6** - a free tool you can download from the internet. My lawyer tells me I cannot recommend it.

In **Avidemux 2.6**:

1. Open the first image in the sequence of images - for example, **HINGE-A000.PNG**

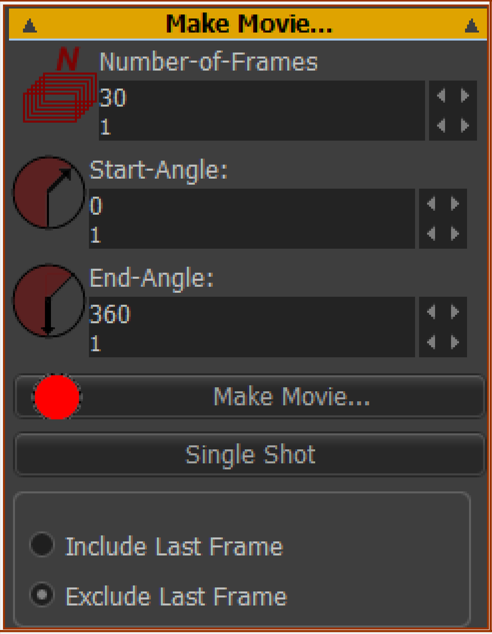
Open the first image only. **AVIDEMUX2.6** automatically imports the other images in the numbered sequence.

2. Save the movie with the **MP4** movie format.

Save the Movie file with the same frame-size as the recorded images.

MP4, with the x264 codec, is the recommended format for **YouTube®**.

MAKE MOVIE...



NUMBER-OF-FRAMES

1. Edit the **NUMBER-OF-FRAMES** box (Default = 30 Frames, Minimum=5 ; Maximum=10000)
2. Edit the **START ANGLE** (Default = 0 ; Minimum > 0, Maximum ≤ Machine End-Angle)
3. Edit the **END ANGLE** (Default = 360 ; 720 ≥ Maximum > Minimum)
4. Select
   - **INCLUDE LAST FRAME** , or
   - **EXCLUDE LAST FRAME**

Experiment! View the movie in your Video Player to see if there is a stutter at the end of the movie.

MAKE MOVIE... button

5. Click the Make Movie button

When you click the Make Movie button, the model moves the model to the **START-ANGLE** and compiles the movie (GIF file-type) or save the images (PNG, JPG, BMP file-types)

Please wait as the file(s) are saved to your PC.

OR

SINGLE SHOT... button

5. Click the Single Shot button

When you click the Single Shot button, you will save a Single Snapshot of the model at its active **Master-Machine Angle**.

1.10.50 Dialog: Element Properties

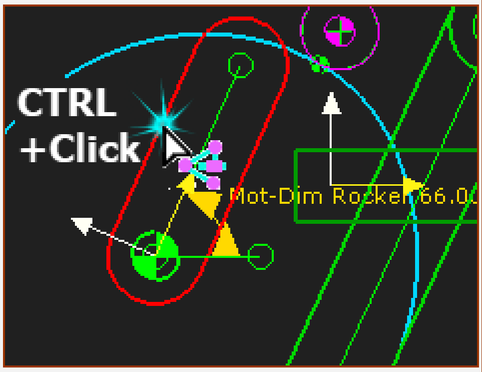
Element Properties dialog.

The **ELEMENT PROPERTIES DIALOG** provides, for an element, the motion and force data, which is evaluated at the **MASTER MACHINE ANGLE**.

Elements Properties include:

- **P** : a Position, or lnear value
- **V** : a Vector
- ℝ : a Real Number
- **|*n*|** : Absolute Real Number
- X, Y, ΔX, ΔY, Θ, ω, α, ΔΘ, Δω, Δα

How to open the Element Properties dialog



In the graphic-area

+ Click an element

For example: a **CTRL+Click** a wire between two Kinematic FBs.

The wire has 3 Data-Channels

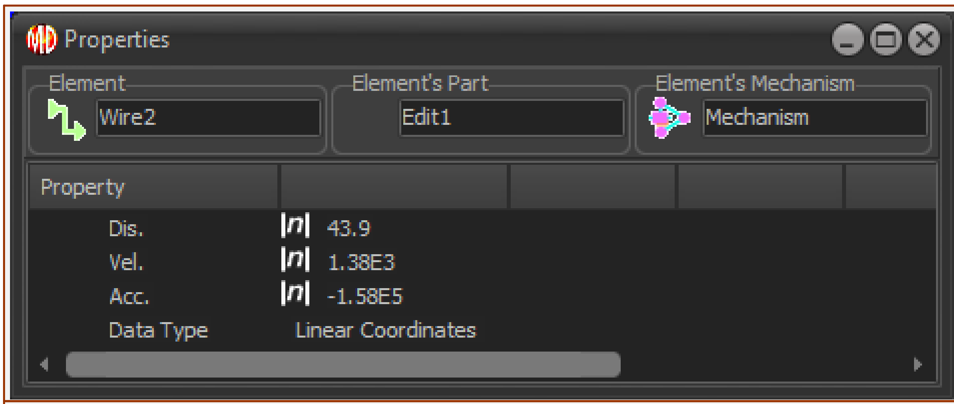
The Data-Channels are Linear or Angular Position, Velocity and Acceleration.

Element Properties dialog (read-only)

At the top of the dialog you can see the **Element's Name**, **Element's Part**, and **Element's Mechanism**.

The Element is a child to the **Element's Part**, which is a child to the **Element's Mechanism**

Wire Element Properties dialog



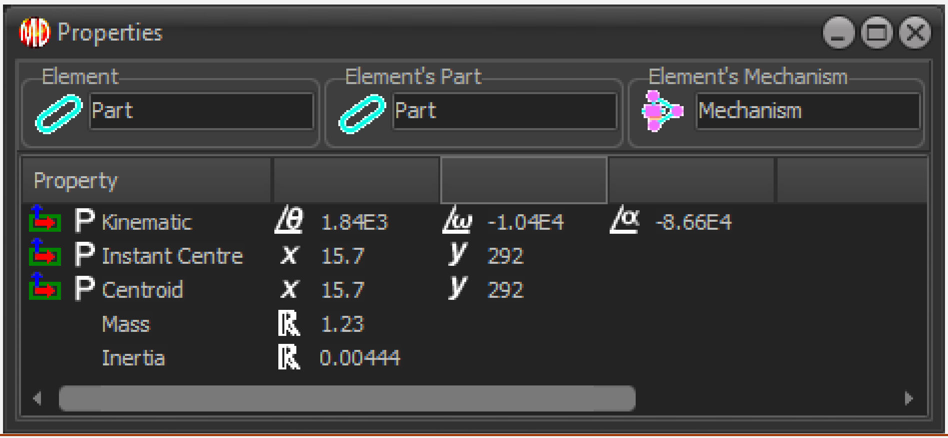
CTRL + Click a wire

A wire has a 3 Data-Channels

In the example: the three Data-Channels are Displacement(Position), Velocity, and Acceleration

The Data Type is Linear.

Part Element Properties dialog



CTRL + Click a Part-Outline

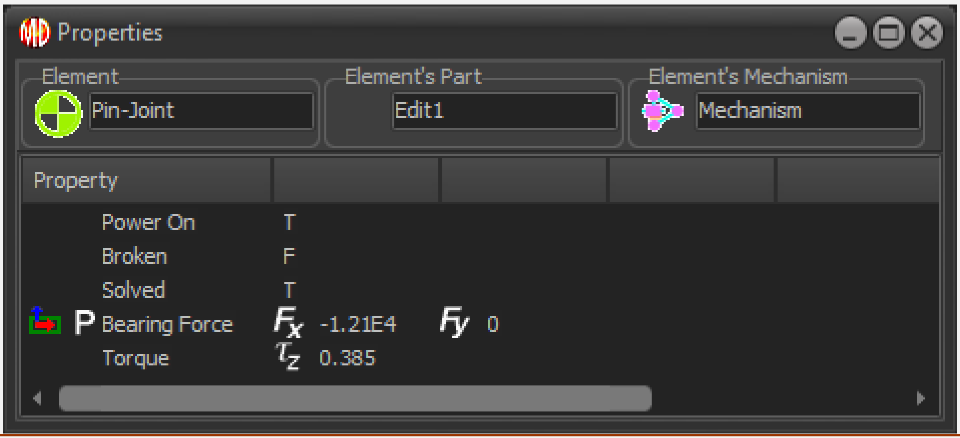
At the top of the dialog you can see the Element's Name, the Element's Part, and the Element's Mechanism.

PROPERTIES:

P - Position

- **Kinematic**: Angle of **PART** in Mechanism Plane, Angular Velocity or Part in Mechanism Plane
- **Instant Centre**: The **PART** has an instant center of rotation: X and Y coordinates of the instant-center in Mechanism Coordinates
- **Centroid**: X and Y coordinates of the Center of Mass
- **Mass**: Value
- **Inertia**: Value

Pin-Joint Element Properties dialog



CTRL + Click a Pin-Joint

At the top of the dialog you can see the Element's Name, the Element's Part, and the Element's Mechanism.

Property:

- **Power On**: True or False (T or F) - if the Pin-Joint has the Power Source - see Configure Power Source [480]
- **Broken**: True or False (T or F) - if the Joint does not solved (Broken) [262] at this instant in the machine-cycle
- **Solved:** True or False (T or F) - if the Joint is a child to a kinematic-chain that is **kinematically-defined**
- **Bearing Force**
    - Fx and Fy ; Vectors in Mechanism Coordinates

- **Torque**
    - Tz ; Vector normal to Mechanism Plane (if the Pin-Joint is the Power Source)

Slide-Joint: Element Properties dialog

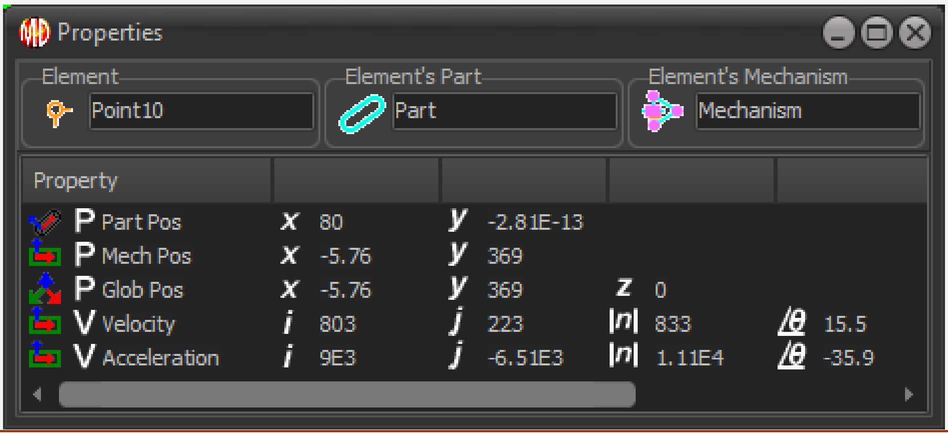
CTRL + Click a Slide-Joint

At the top of the dialog you can see the Element's Name, the Element's Part, and the Element's Mechanism.

Property:

- Power On: True or False (T or F) - if the Power Source is given to this Pin-Joint - see Configure Power Source [480]
- Broken: True or False (T or F) - if the Joint does not solved Broken) [262] at this instant in the machine-cycle
- Solved : True or False (T or F) - if the Joint is a child to a kinematic-chain that is **kinematically-defined**
- Drive Force
    - Fx ; Force along positive direction of Slide-Joint (if the Power Source is the Slide-Joint

Point: Element Properties dialog



CTRL+Click a Point

At the top of the dialog you can see the **Element's Name**, **Element's Part**, and **Element's Mechanism**.

The Element is a child to the **Element's Part**, which is a child to the **Element's Mechanism**

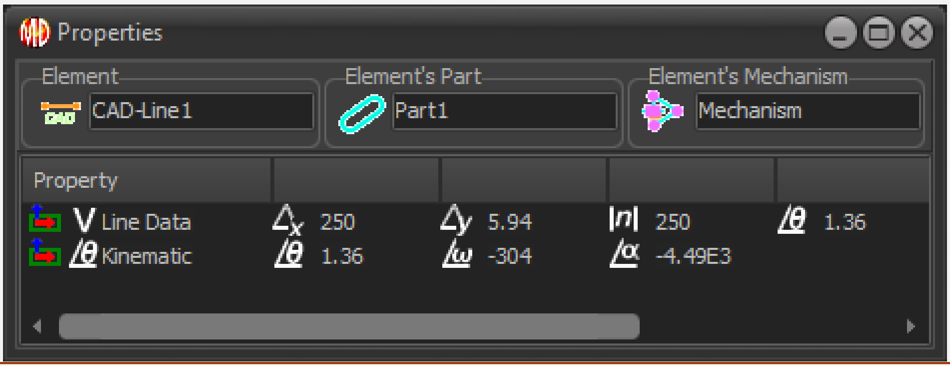
PROPERTIES:

(P - Position ; V- Vectors)

- **P Part Pos**: Position of the Point; Part Coordinates
- **P Mech Pos**: Position of the Point; Mechanism Coordinates

- **P Glob Pos**: Position of the Point; Global Coordinates (Z does not show!)
- **V Velocity, V Acceleration**
  - **i,j:** the coordinates of the Velocity / Acceleration Vector, X and Y directions; Mechanism Coordinates
  - **|n|**: the magnitude of the Velocity / Acceleration Vector
  - **Θ:** the direction of the Velocity / Vector Vector

Line: Element Properties dialog



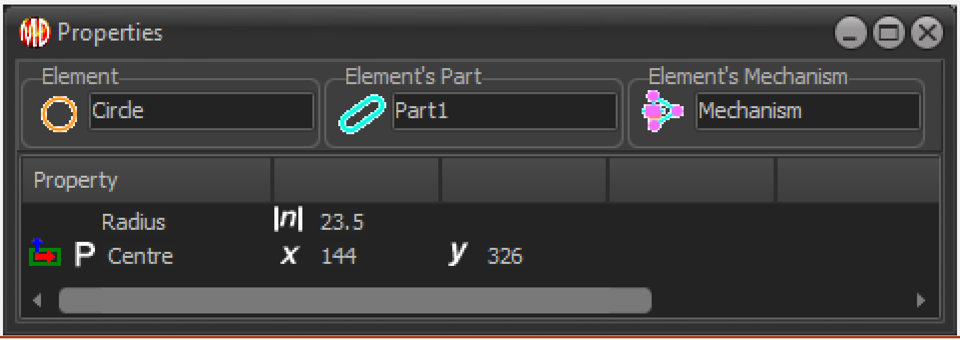
CTRL + Click a Line.

At the top of the dialog you can see the Element's Name, the Element's Part, and the Element's Mechanism.

Property:

- Line Data:
  - Δx, Δy : the change in x and y from the Line's start-Point and end-Point
  - |n|: the length of the Line
  - Θ: the angle of the Line relative to the Mechanism Coordinates
- Kinematic:
  - ω : angular velocity of the Line
  - α : angular acceleration of the Line

Circle: Element Properties dialog



CTRL + Click a Circle.

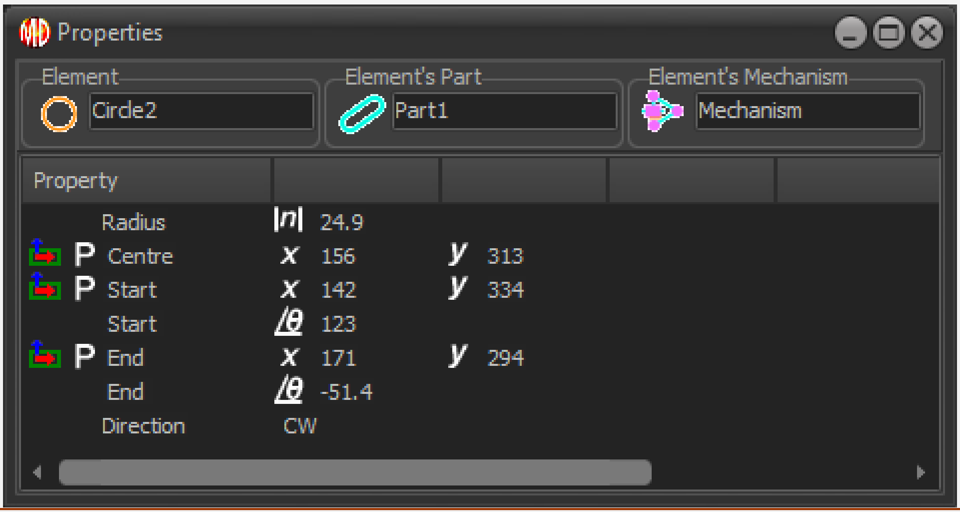
At the top of the dialog you can see the Element's Name, the Element's Part, and the Element's Mechanism.

Property:

- **Radius:**
  - **|*n*|**: the Radius of the Circle

- center:
  - *x & y* : the position of the center of the Circle in Mechanism Coordinates

Arc: Element Properties dialog



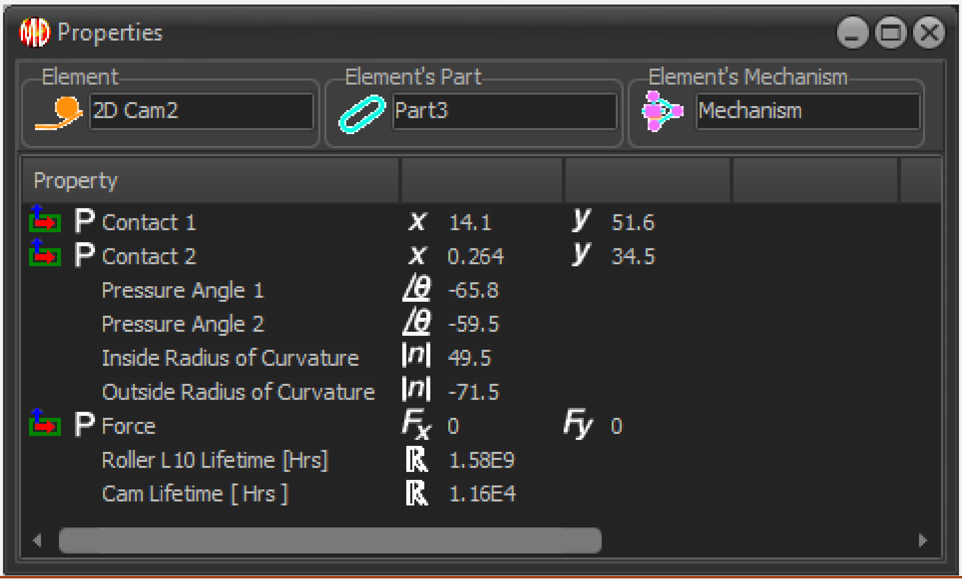
CTRL + Click an Arc.

At the top of the dialog you can see the Element's Name, the Element's Part, and the Element's Mechanism.

Property:

- **Radius:**
  - **|n|**: the Radius of the Circle
- **center:**
  - **x & y :** the position of the center of the Arc in Mechanism Coordinates
- **Start**
  - **x & y** : the position of the start-Point of the Arc in Mechanism Coordinates
  - **Θ** : the angle of the start-Point from the center of the Arc
- **End**
  - **x & y**, the position of the end-Point of the Arc in Mechanism Coordinates
  - **Θ** : the angle of the end-Point from the center of the Arc

Cam: Element Properties dialog



CTRL + Click a 2D-Cam.

At the top of the dialog you can see the Element's Name, the Element's Part, and the Element's Mechanism.

Property:

Contact 1 & 2:

- ***x & y:*** the coordinates of the contact between the Inner and Outer Cam-Profiles and the Follower-Profile.

Pressure Angle 1 & 2:

- ***Θ :*** the Pressure Angle of the Cam-Follower-Roller with the Inner and Outer Cam-Profiles

Curvature 1 & 2:

- **|*n*|** ***:*** the Radius of Curvature of the Inner and Outer Cam-Profiles

Force

- ***Fx & Fy :*** the Contact Force of the Cam on the Roller, given in the X and Y directions, with Mechanism Coordinates

---

ONLY if you have selected a Cam-Follower and Steel Type with the **2D-CAM DIALOG**

Roller L10 Lifetime [Hrs] - The Lifetime in Hours of the Cam-Follower Roller, Lubrication Conditions, and Factor-of-Safety that you have selected with **2D-CAM DIALOG** 336

Cam Lifetime [Hrs] - the lifetime in hours of the Cam Profile (Outer or Inner) , with the Steel Type, Quality, Hardness, and Factor-of-Safety that you have entered in the **2D-CAM DIALOG**

1.10.58 Dialog: Select Elements

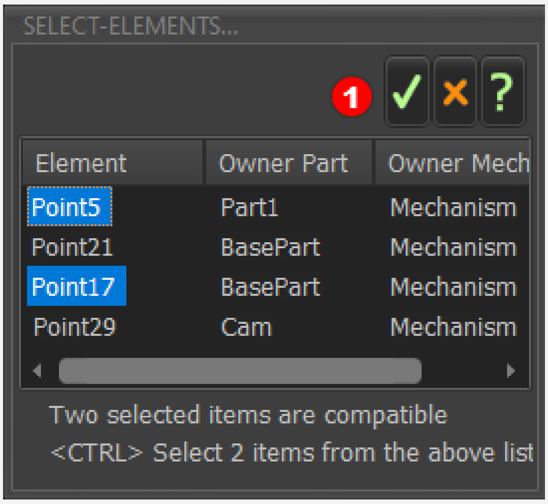
Select Elements

If, when you select elements to do a command, there is ambiguity as to which **ELEMENT(S)** to actually use for the command, the **SELECT ELEMENTS DIALOG** opens **automatically**.

It lists all of the elements that you select, including the **POINTS** you want to select.

You must **CTRL+Click** the elements to do the command.

Select-Elements dialog



Select-Elements dialog

In the **SELECT-ELEMENTS DIALOG**:

1. Ctrl + **Click ALL** of the elements you need to do the command.

The ✓ becomes colorized only after you select the elements that are compatible with the active command.

2. Click ✓ to close the **SELECT-ELEMENTS DIALOG**

If the **COMMAND-MANAGER** is also active:

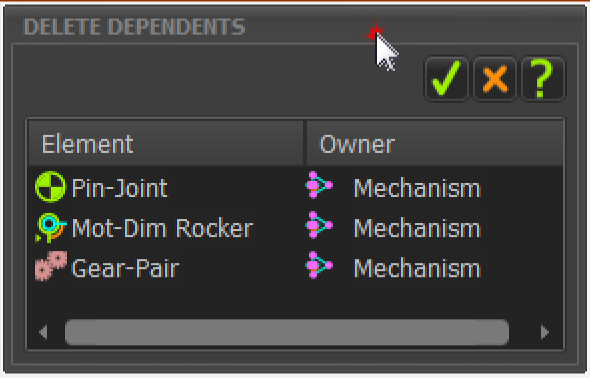
3. Continue to select the elements, as required.

and / or

4. Click ✓ in the **COMMAND-MANAGER** to complete the command.

1.10.59 Dialog: Delete Dependent Elements

Delete Dependent Elements



Warning - When you delete the 'element' you will also delete these elements.

Many elements **depend on (need) other elements**

The **Delete Dependents dialog** is a warning to you that, if you want to delete an element, you will also delete one or more other elements.

You can:

- **Delete element and Dependents**- click the button

OR

- **Cancel Delete** - click the button

For example:

When you add a **PIN-JOINT**, you select two **POINTS** in two different **PARTS**.

If you want to delete the one of the **POINTS** that needs the **PIN-JOINT**, you are warned that you also delete the **PIN-JOINT**.

1.10.6 Dialog: View References

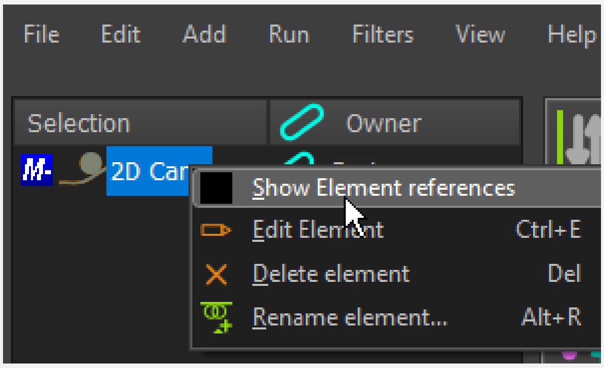
Show Element References

See also: Delete Dependent Elements 505

Many elements reference, or need, other elements.

Show Element References lists those elements that a more complex element needs to build.

For example, when you add a **PIN-JOINT**, you select two **POINTS** in two different **PARTS**. Therefore, the Reference Elements of the **PIN-JOINT** are the two **POINTS**.



Selection--Window: Right-Click

To see the **Element References** for an element:

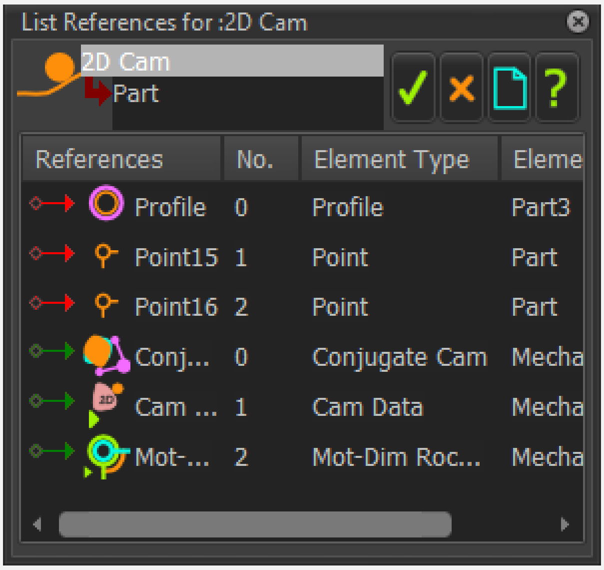
1. Click an **ELEMENT** in the graphic-area or the **ASSEMBLY-TREE**

   The element should be in the **SELECTION-WINDOW**

   Right-click the **ELEMENT** in the **SELECTION-WINDOW**

2. Click **Show Element References** in the shortcut menu

   The **List References for: element-name** shows.



EXAMPLE: Element References for 2D-Cam

The example shows the:

- **Reference Element name**
- **Number(?)**
- **Reference Element Type**
- **Reference Element Owner**

1.10.6 Dialog: Tutorials

Download Tutorials

To open the Download Tutorials dialog

Click File toolbar > Download Tutorial Videos

OR

Help menu > Download Tutorial Videos

Download Tutorials dialog

When the Download Tutorials dialog opens:

1. Click a video on the left of the dialog.

If this is the first time you have downloaded that video. please wait until it downloads to your computer

If, at a different time, you click to play the same video, it should play immediately from your hard-disk.

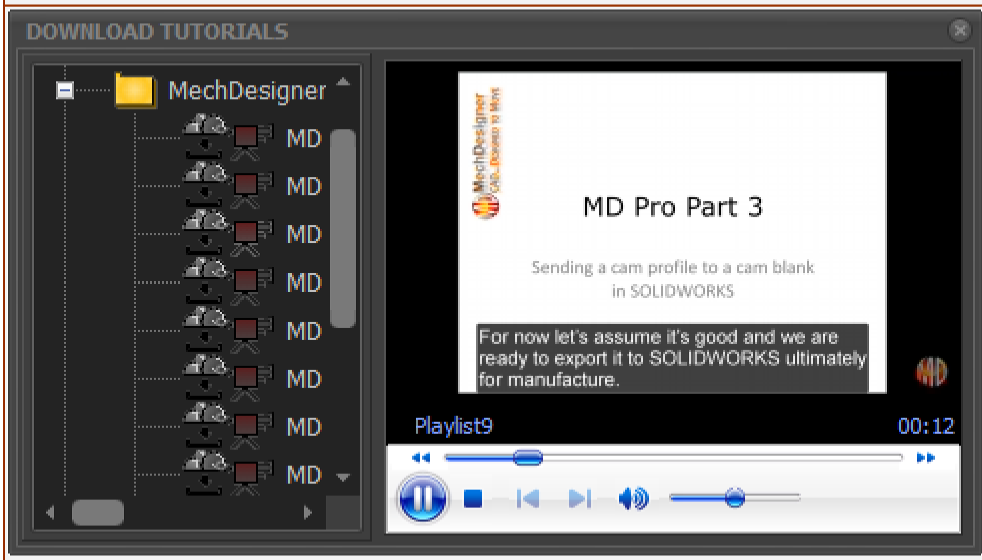
Video display too small?

If the video dialog is too small, drag the right and bottom sides of the dialog to increase its size.

Where do the Videos download to?

The default path is **<LocalAppData> \ Tutorials\ *.mp4**

To change the download path, see Application-Settings > General tab > File Options separator > Tutorial Video Directory [284].



Download Tutorials dialog

1.11 FAQs and 'How to...?'

FAQs and How to...?

General FAQs and How to ...?

Model-Editor: FAQs and How to ...?

Mechanism-Editor: FAQs and How to ...?

Part-Editor: FAQs and How to ...?

FAQs

1.11.1General FAQs

General FAQs

1.11.1. How to ? 1

1.11.1.1.1 How do I edit, save, and move my Styling?

Settings, Styles, and Themes...?

We call the **Style**, **Theme**, and **Settings** the **Styling**.

Each time you exit **MechDesigner**, we save for you the **Styling** to **MechDesigner.INI** and **MechDesigner.XML.**

MechDesigner uses the **Styling** automatically the next time it starts.

Styling Definitions:

Theme :	**Border, Icon, and dialog colors.** Use **Application Settings : General tab** > **THEME SETTINGS** > **Target Theme** Select one of these **Themes**: Windows, Charcoal Dark Slate, Aqua Light Slate, Windows10 Dark, Tablet Dark, Slate Classico, Windows10 Slate Gray, Windows 10. Note, this help uses Charcoal Dark Slate.
Style :	**Load a Style** Click the Load button to open a Style **Dark.XML** - is intended to be used with dark themes, for example, Charcoal Dark Slate. **Light.XML** - is intended to be used with light theme, for example, Aqua Light Slate.

	Save a Style Click the Save button: Save any changes you have made to Style file-name Do NOT overwrite Dark.XML or Light.XML.
Application Settings :	Edit icon-sizes, number-formats and colors for different elements: • **Application-Settings** > **Graphics tab** > DISPLAY COLORS There are many other settings you can make to suit your preferences. Note: You can also save and load these with the Save and Load buttons

Save the Styling

You may want to save your personal styling, especially if different people use **MechDesigner** on the same PC.

1. **File menu** > **Application-Settings** :
2. Click the **Save** button at the bottom of the **APPLICATION-SETTINGS DIALOG**.

   The default path for your Styling should open **<CommonAppData>\PSMotion\Style \ Comma** or **\Stop**

   You see **Dark.XML** and **Light.XML** files (and others).

   DO NOT OVERWRITE these files.

   - Save the Styling to **...style\Comma\My-style-name.XML** if your number system uses a comma **','** as the decimal symbol.
   - Save the Styling to **...style\Stop\ My-style-name.XML** if your number system uses a stop **'.'** as the decimal symbol.

Move your Styling

You may want to move your styling to a different computer for your own use, or to distribute the Styling to all Workstations.

1. Save the active **Styling** to a ***My-file-name.XML*** - see preceding section [511].

   Remember, DO NOT OVERWRITE the default **Dark.XML** and **Light.XML** files.
2. Exit **MechDesigner**
3. Use File-Explorer to navigate to:
   - **<CommonAppData>\PSMotion\style\Comma** if your number system uses a comma ',' as the decimal symbol.
   - **<CommonAppData>\PSMotion\style\Stop** if your number system uses a stop '.' as the decimal symbol.

Note 1: **<CommonAppData>** is usually **C:\ProgramData**

Note 2: If you cannot see **C:\ProgramData** ... then in **Windows File Explorer:**

a) Click **View** tab

b) Click **Hidden Items** check-box in the **Show/hide** group-box.

---

4. Copy **My-style-name.XML**, to a **USB** stick, for example.

You also need to make copies of the **MechDesigner.INI** and **MechDesigner.XML** files

5. Use File-Explorer to navigate to path: <LocalAppData> C:\users\<user-name>\AppData\Local\PSMotion\IniFiles****
6. Copy **MechDesigner.INI** and **MechDesigner.XML** to a **USB** stick.

On the other Workstation:

7. **Replace the MechDesigner.INI** and **MechDesigner.XML** files on the workstation, with the files on your **USB** stick
8. **Copy My-style-name.XML from your stick into the path used for styling -** see preceding sections 511.
9. Start **MechDesigner** on the workstation
10. Click **Application-Settings** > **Load button**
11. Load **My-style-name.XML** from the USB stick as the new style.
12. Click **Save** button

    Save the **Styling** to **<CommonAppData>** is usually **C:\ProgramData**

The **Theme, Style** and **Application-Settings** are saved automatically as defaults. They load the next time you start **MechDesigner**.

1.11.1.1.2 How to open a dialog

Open an element's dialog

To build a model:

STEP 1:	Add an ELEMENT	- see Menus and Toolbars [12]
STEP 2:	Open the element's dialog	- see below
STEP 3:	Edit the ELEMENT'S PARAMETERS	- see How to edit a Parameter in a dialog [517]

How to open a dialog

There are many methods you can use to open an element's dialog.

With all methods, you must be able to select the element. You can select the element in three places:

- Graphic-Area
- Assembly-Tree
- Selection-Window

It may be helpful to use a **Selection-Filter** to isolate for the type of element you want to edit.

See Selection-Filters [57]

To open an element's dialog, do one of these methods:

Method 1: Use the Selection-Window

CLICK

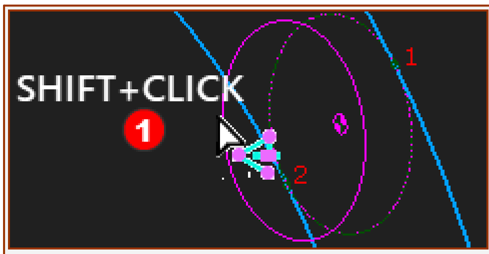
1. Click the ELEMENT (see also the SHIFT-CLICK exception below) in the graphic-area or the ASSEMBLY-TREE

   The element is now in the SELECTION-WINDOW
2. Right-click the ELEMENT in the SELECTION-WINDOW

   A shortcut menu shows next to your pointer
3. Click **Edit element** in the shortcut menu

   The element's dialog opens.

SHIFT+CLICK



SHIFT+CLICK Profile

An EXTRUSION is a child to a PROFILE. It is possible to show or hide the EXTRUSION from the graphic-area. When the EXTRUSION is hidden from the graphic-area:

1. SHIFT+CLICK the PROFILE contour that is the parent to the extrusion in the graphic-area

   The PROFILE AND EXTRUSION are now in the SELECTION-WINDOW.
2. Right-click the EXTRUSION

   In the shortcut menu:
3. Click **Edit element** to open the EXTRUSION DIALOG.

The EXTRUSION DIALOG opens.

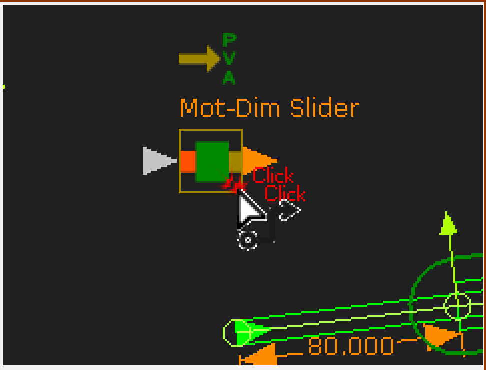
Method 2: Double-Click the element

Summary:

You open the dialog **only if you** double-click **one ELEMENT**.

Double-click one **ELEMENT** in the:

- Graphic-Area
- **ASSEMBLY-TREE**
- **SELECTION-WINDOW**



Double-click to open the element's dialog-box

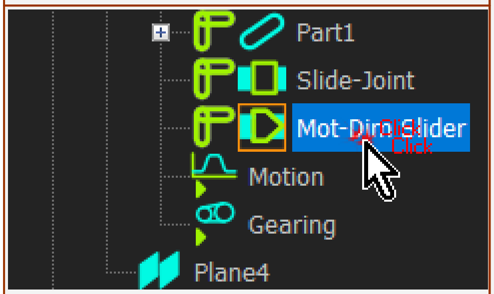
EXAMPLE: EDIT A MOTION-DIMENSION FB

Double-Click in the Graphic-Area

1. Move your mouse-pointer above the **ELEMENT** so that **only** the **ELEMENT** changes color to the Selection Color (usually, it becomes **red**).
2. Double-click the **ELEMENT** in the graphic-area

When it is difficult to double-click **only** the **ELEMENT** in the graphic-area:

- Edit the Cursor Selection size - See Edit toolbar > Application Settings > Accessibility tab [289].
- Use Selection Filters [57] to make it possible to select only the **ELEMENT** type you want to edit.
- Use Display Filters [54] to hide other **ELEMENTS**.

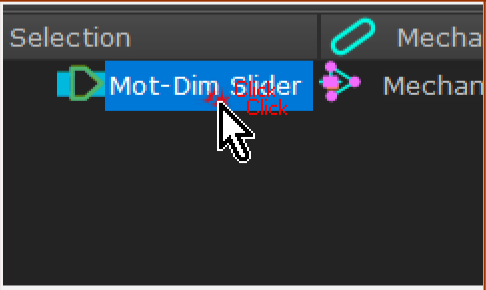


Double-Click element in Assembly-Tree

OR

Double-click in the Assembly-Tree

1. Double-click the **ELEMENT** in the **ASSEMBLY-TREE**.



Double-Click element in Selection-Window

OR

Double-click in the Selection-Window

1. First, click the **ELEMENT** in the graphic-area or the **ASSEMBLY-TREE** one time so that it shows in the **SELECTION-WINDOW**.

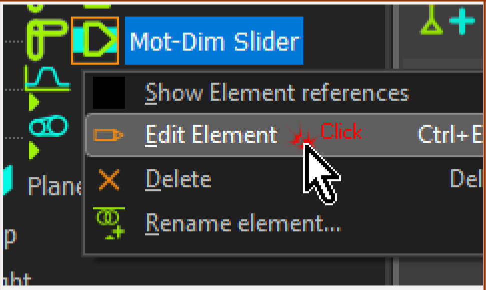
When the **ELEMENT** is in **SELECTION-WINDOW**,

2. Double-click the **ELEMENT** in the **SELECTION-WINDOW**.

Method 3: Right-Click the element

Summary:

1. Right-click the **ELEMENT** in the **SELECTION-WINDOW**, the **ASSEMBLY-TREE** and graphic-area
2. Click **Edit-Element** in the shortcut menu.

<table>
<tr>
<td>

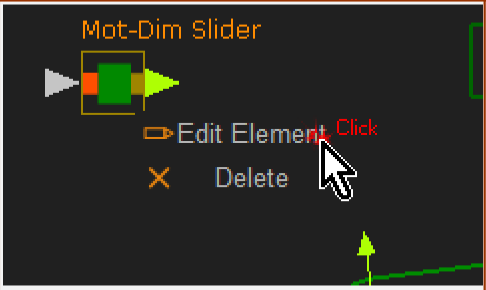
Click, then Right-Click in Assembly-Tree</td>
<td>

EXAMPLE: EDIT A MOTION-DIMENSION FB

In the Assembly-Tree:

1. Click to select the ELEMENT in the ASSEMBLY-TREE
2. Right-click the ELEMENT

In the shortcut menu:

3. Click **Edit Element**
</td>
</tr>
<tr>
<td>

Right-click in the graphic-area</td>
<td>

In the Graphic-Area:

1. Click the graphic-area to deselect all elements
2. Hover + Right-click an ELEMENT.

The shortcut menu shows.

In the shortcut menu:

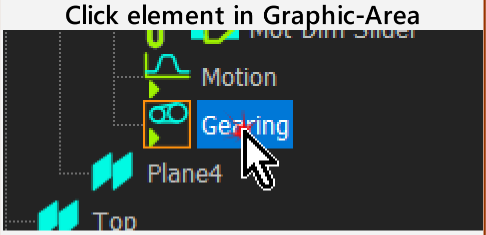
3. Click **Edit Element**
</td>
</tr>
</table>

Method 4: Edit toolbar > Edit Element tool

<table>
<tr>
<td colspan="2">

Do:

1. Click the ELEMENT in the SELECTION-WINDOW, ASSEMBLY-TREE or Graphic-Area
2. Click **Edit toolbar > Edit Element icon**
</td>
</tr>
<tr>
<td>

Click element in Graphic-Area


Click element in Assembly-Tree</td>
<td>

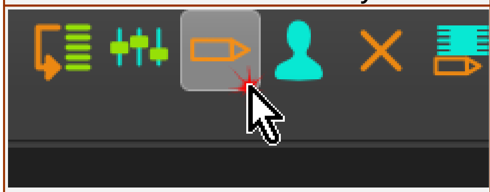
In the Graphic-area, Assembly-Tree, or Selection-Window:

1. Click the GEARING FB in the SELECTION-WINDOW

OR

1. Click the GEARING FB in the graphic-area

OR

1. Click the GEARING FB in the ASSEMBLY-TREE
</td>
</tr>
<tr>
<td>
Click 'Edit-Element' icon in Edit menu</td>
<td>

2. Click **Edit toolbar > Edit Element**

The GEARING-FB DIALOG is now open.
</td>
</tr>
</table>

Special Cases:

To edit a dimension in the Part-Editor

<table>
<tr>
<td>

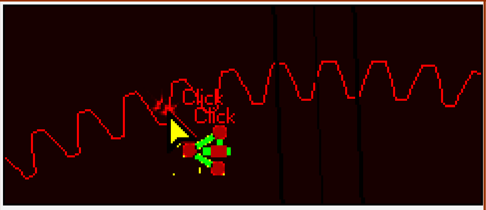
Double-click Arrowhead</td>
<td>

De-select other commands. Most usually, **Add Dimension** is active.

1. Double-Click the arrowhead of a DIMENSION to open the DIMENSION DIALOG

If you try to click the dimension number - **34.46** in the image to the left - **MechDesigner** does not respond.
</td>
</tr>
</table>

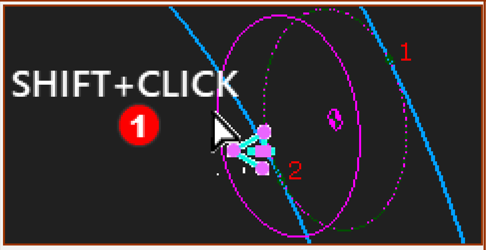
To edit a Gear-Pair or 2D-Cam



When you click a **GEAR-PAIR** or **2D-CAM** it is possible to click **four** **ELEMENTS**:

- the **GEAR-PAIR** (or **2D-CAM**) - select it in the **SELECTION-WINDOW**
- a **POLYLINE** - you cannot edit a **POLYLINE**
- a **PROFILE** and **EXTRUSION** - you can only edit the **EXTRUSION**.

To edit an Extrusion



Shift-click a Profile

To edit an **EXTRUSION** that you cannot see, but you can see its **PROFILE** or **AUTO-PROFILE.**

1. SHIFT-CLICK the PINK **PROFILE ELEMENT**

The **PROFILE** AND **EXTRUSION ELEMENTS** are now in the **SELECTION-WINDOW**.

2. Right-click the **EXTRUSION ELEMENT** in the **SELECTION-WINDOW**
3. Click **Edit element** in the shortcut menu.

Reasons you cannot edit an element

You cannot edit an **ELEMENT** when:

- A different dialog is open. **Close all other dialogs.** (ALT+F4 if you cannot see the dialog).
- You **double-click** two or more elements. You open none or one dialog.
- A command is active, in the **COMMAND-MANAGER**. You must complete or cancel that command.
- You cannot edit the element - for example, a **SLIDE-JOINT**

1.11.1.1.3 How to edit a Parameter in a dialog

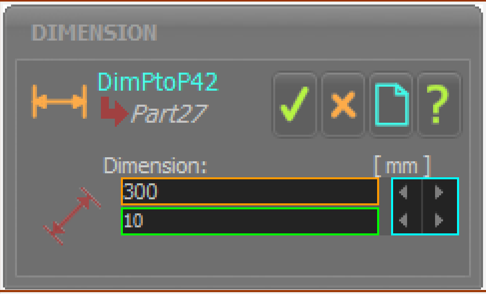
Edit Parameters.

To build a model:

STEP 1:	Add an ELEMENT	- see Menus and Toolbars [12]
STEP 2:	Open the element's dialog	- see How to open a dialog [513]
STEP 3:	Edit the element's PARAMETERS	- see How to edit a Parameter in a dialog [517]

A parameter-value in a dialog

In this example, the parameter is DIMENSION in the DIMENSION DIALOG, and the parameter-value is 300.



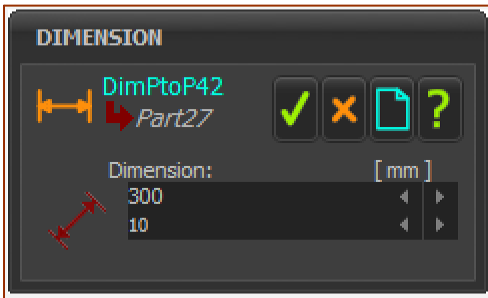
In the image, I have added three colored boxes

300	PARAMETER-VALUE - the value, in the data-box, to edit and apply to the element.
10	Spin-Increment - the change to the PARAMETER-VALUE each time you click a **Spin-Box arrow** - see Spin-Box tool [518]
	**Spin-Box tool**: see more Spin-Box tool [518] **Note**: If you cannot see the **Spin-Box tool:** Double-click with your mouse in the PARAMETER-VALUE box

How to edit parameters in a dialog.

There are three different methods you can use to edit a value in the parameter box.

Method 1: Keyboard

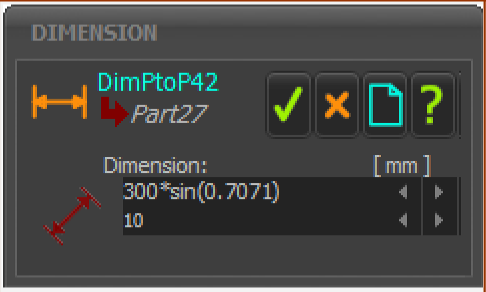


Dimension dialog-box

Enter a value directly:

1. Use your keyboard to enter a value in the data-box
2. Press the Enter key (↵) on your keyboard to update the parameter to the new value

   You must press the Enter key.



Equation in Dimension box

Enter a value as a symbolic equation:

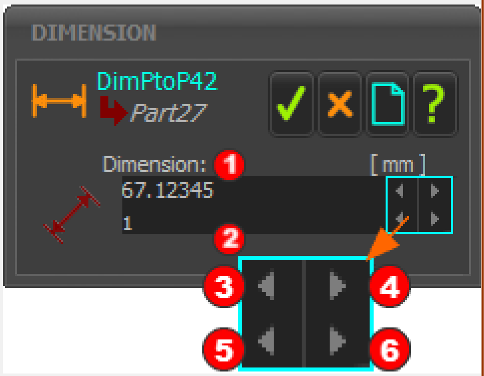
1. Use your keyboard to enter a symbolic-equation in the data-box
2. Press the Enter key (↵) on your keyboard to update to the new value

You must press the Enter key.

SYMBOLIC PARAMETERS FOR AN EQUATION:

- **Simple Arithmetic**: +, –, *, /
- **Indices**: ^, Sqrt()
- **Trigonometric** (Angles are Radians): pi, Sin(), Cos(), Tan(), Sinh, Cosh, Tansh, ArcSin(), ArcCos(), ArcTan2(;)

Method 2: Spin-Box tool



Spin-Box tool

Spin-Box arrowhead buttons (in the CYAN-BOX)

❸ Top & Left arrowhead : to subtract (**–**) the PARAMETER-VALUE❶ by the SPIN-INCREMENT❷

❹ Top & Right arrowhead : to add (**+**) the PARAMETER-VALUE❹ by the SPIN-INCREMENT❷

❺ Bottom & Left arrowhead : to divide (÷) by the SPIN-INCREMENT❷

❻ Bottom & Right arrowhead : to multiply (**×**) by the SPIN-INCREMENT❷

EXAMPLE:

Edit the PARAMETER-VALUE

- **Click the Top & Left❸ arrowhead**: Subtract the Spin-Increment(1)❷ from the PARAMETER-VALUE❶
  PARAMETER-VALUE after one click is 67.12345 – Spin-Increment = 66.12345
- **Click the Top & Right❹ arrowhead two times:** Add the Spin-Increment(1)❷ to the PARAMETER-VALUE❶ 2 x
  Parameter-value after one click is **66.12345 + 2 × Spin-Increment = 68.12345**

Edit the Spin-Increment

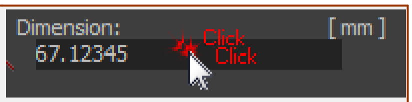
- **Click the Bottom & Left❺ arrowhead**: Divide the ***active* Spin-Increment** by 10❷
  The **active Spin-Increment** value after one click is **1 ÷ 10 = 0.1**
- **Click the Bottom & Right❻ arrowhead two times:** Multiply the ***active* Spin-Increment** by 10❷ 2 x
  The **active Spin-Increment** value after two clicks is **0.1 × 10 × 10 = 10.**

WARNING: The model re-builds each time you click the top arrowhead buttons in the **Spin-Box tool**. If the model is complex, there is a long time to update the model.

Method 2A: Show / Hide the Spin-Box

Double-click to hide Spin-Box

< >
Double-Click

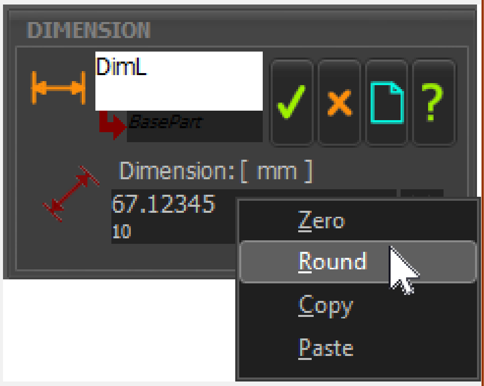


Double-click to hide Spin-Box

Double-click, with your mouse-pointer **in** the data-box, to hide the SPIN-BOX.

Double-click the data-box ***again*** to show the SPIN-BOX.

Method 3: Zero / Round / Copy / Paste



To see the Zero / Round / Paste / Copy shortcut menu:

1. Right-click with your mouse-pointer inside the parameter box
2. **Click:**

- **Zero** - the parameter-value becomes zero(0), or the smallest possible value for the parameter.
- **Round -** the parameter-value becomes the nearest value that is exactly divisible by the **Spin-Increment.**

Note: The **Spin-Increment** is below the data-box. It is ten(10) in the image to the left - see **Spin-box** tool

- **Copy -** to copy the parameter-value to your clipboard.
- **Paste** - to paste your clipboard to the parameter-value.

1.11.1.1.4 How do I rename an element?

See

Rename dialog [279]

1.11.1.2 FAQs

FAQs

What are the System Requirements? [520]

Why is it taking a long time to add or edit elements? [521]

1.11.1.2.1 What are the system requirement?

System Requirements

Operating System :	Windows 10, 11 64-bit **only**
PC Memory (RAM) :	Minimum 8GB Recommended: 16GB and more
Hard Drive Space :	Minimum: 20GB of available space Recommended: 100GB or more - keep your installation disk healthy. Recommended: SSD - Solid State Disk
Graphics Card :	High-performance Graphics-Card GPU OpenGL 3.5+ Minimum: 2GB VRAM Recommended: 16GB VRAM.
Screen Resolution :	Minimum : 1920 × 1080 Recommended : 4K (3840 × 2160)
Windows DPI Setting :	If Screen Resolution is 1920 × 1080 , then set to 100% If Screen Resolution is 4K, then set to Windows DPI to 150%. Shortcut for Display-Settings: Right-click Windows Desktop, select Display settings.

Windows Permissions

Installation :	**Administrator Rights.**
Download Help : **Download Tutorials :**	**Administrator Rights** to download the **Local Help** and **Tutorials** with the **Help menu** 1. Right-click the **MD** application open 2. Click **Run as Administrator** in the shortcut-menu
General Running :	**Administrator Rights** are not needed. However, you do need an internet connection. ***MechDesigner*** occasionally checks your license with CopyMinder.

Software Protection

We use **CopyMinder**® for our software protection.

When you buy ***MechDesigner***, we email you a **Product Key.**

You need the **Product-Key** to run ***MechDesigner*** the first time, or to move MechDesigner to a different computer.

If you have a **Network-License** (also called a **Floating-License**), your I.T. department should install ***MechDesigner***. I.T. should keep the **Product Key**.

See also: CopyMinder Software Protection

1.11.1.2.2 Why is it taking a long time to add elements?

Tips for Working with Large Models:

- **Disable Auto-Rebuild** [37]. Then experiment. You can continue to add many elements successfully when rebuild is inactive**.** Use Rebuild-Now [37] every 4-5 commands to rebuild.
- Number-of-Steps [291] : Make 90 steps, or even less. You do not need many steps as you build the model.
- Disable Show Solids in Mechanisms [51] : You do not need to show the **SOLIDS** to add elements, to review the kinematics, or to calculate Forces.
- Image Quality [287] : If you must show **SOLIDS**, then experiment with the Image-Quality. Do this when there are many Solids in the model.
- Spin-Box tool [518]. Do **not** use the **SPIN-BOX TOOL**. The model re-calculates with each click of the Spin-Box. Use your keyboard to enter new values in each data- box.

1.11.1.2.3 Cam-Terminology

Cam-Terminology:

Cam Terminology :	Definitions
Cam :	the PART that supports the **Cam-Profile** shape. Its motion is the input to the Cam system.
Cam-Shaft :	a Cam that rotates, usually with constant angular velocity.
Cam-Profile :	a surface that is in continuous contact with the **Follower-Profile** (the 2D-CAM). Also called **Cam-Flank.**
Cam-Track :	a groove that is machined into a **Cam-Blank** with an Outer and an Inner **Cam-Profile**.
Cam-Follower :	the PART that supports the **Follower-Profile**. Its motion is the output from the Cam system.
Follower-Profile :	the surface that interacts with the **Cam-Profile**
Cam-Follower Roller :	a **Cam-Follower** with a **Follower-Profile** that has a cylindrical or barrel shaped surface
Flat-faced Cam-Follower :	a **Cam-Follower** with a **Follower-Profile** that has a flat surface
Translating Cam-Follower :	a **Cam-Follower** that slides along a straight axis.
Rotating Cam-Follower :	a **Cam-Follower** that rotates about a fixed axis.
Progressive Motion :	a motion that moves the **Cam-Follower** generally in one direction.
Non-Progressive Motion :	a motion that returns the **Cam-Follower** to its original position after each machine cycle.
Reciprocating :	a non-progressive, translating motion.
Oscillating :	a non-progressive, rotating motion - also called Swinging.
Indexing :	a progressive, usually rotating motion.
Indexer :	a device whose input is a **Cam-Shaft** and output is a **Progressive Motion**. After a number of machine cycles the output advances by 360º.
Cam-Blank :	the material that you need before you cut the shape of the Cam-Profile.
Conjugate-Cams :	a **Body-Closed Cam** with typically two **Cam-Profiles** that rotate on one axis/shaft, which are in continuous contact with two **Cam-Follower Rollers** that oscillate or index a **Cam-Follower**, which is on a different shaft.
Body-Closed Cams :	**Conjugate-Cam** in which it is impossible for the **Cam-Follower** to move independently of the **Cams**.
Force-Closed Cams :	a **Cam** in which an external force maintains the contact between the **Cam-Profile** and the **Follower-Profile**. If the external force is overcome, the **Follower-Profile** loses contact with the **Cam-Profile** and the **Cam-Follower** moves independently of the **Cam**.

1.11.2Model Editor: FAQS, How to...?

Model-Editor FAQs

- How do I stop the solids turning red?[523]

1.11.2.1 How do I stop Solids turning red?

How to stop Solids turning Red as your mouse-pointer moves over them.

In the **MODEL-EDITOR**, **SOLIDS** become ***red*** as you mouse-over them (default color in Application-Settings > Graphics tab[41])

To stop this, filter for an element that is ***not*** in the **MODEL-EDITOR**.

For example, **Filter for Points**. **POINTS** are not in the **MODEL-EDITOR**, and therefore **SOLIDS** do not become ***red*** as you mouse-over them.

See Filters menu (Filters toolbar) > Filter Points[57].

1.11.3Mechanism Editor: FAQs, How to...?

Mechanism-Editor: FAQs and How to ...?

1.11.3. How to...?
1

How to...?

1.11.3.1.1 ... show Velocity & Acceleration Vectors?

See

Point Properties dialog. 444

1.11.3.1.2 ... edit a 2D Cam, Gear-Pair, or Profile?

To edit a 2D-Cam, Gear-Pair, Profile, Extrusion.

There are certain elements that are coincident with each other in the graphic-area.

For example: A **POLYLINE** element may be coincident with a **2D-CAM** and also a **PROFILE/ EXTRUSION.**

When you want to edit one of these elements, you may find you open a dialog that you do not want to open, or you fail to open any dialog.

To edit the element that you want to edit **2D-CAM, POLYLINE, PROFILE**, or **EXTRUSION**

There are two methods:

Method 1 - Selection-Window (preferred)

1. Click the element you want to edit the graphic-area

All of the elements you click show in the Selection-Window 257

2. Right-click the element you want to edit
3. Click **Edit Element** in the shortcut menu

Method 2 - Assembly-Tree

1. Find the element in the Assembly-Tree
2. Click the element in the Assembly-Tree to select it.
3. Right-click the element you want to edit
4. Click **Edit Element** in the shortcut menu

1.11.3.1.3 ... edit Profiles and Extrusions?

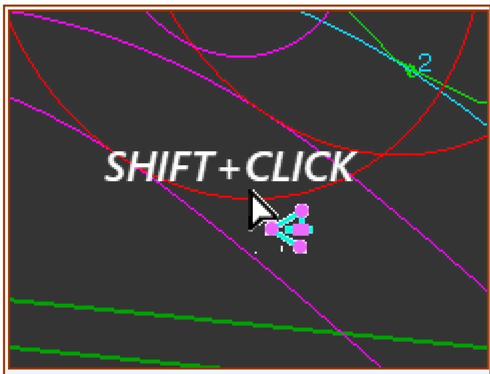
How to edit a Profile and Extrusion element?

NOTE:

- You can ONLY EDIT **EXTRUSIONS**
- You can ONLY DELETE **PROFILES**.

If you delete a **PROFILE**, you also delete the **EXTRUSION**.

Edit the Extrusion



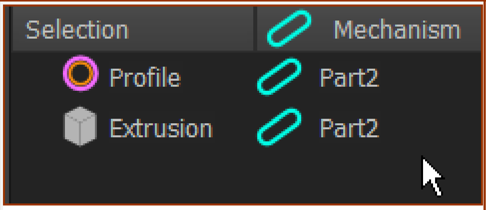
EDIT THE EXTRUSION

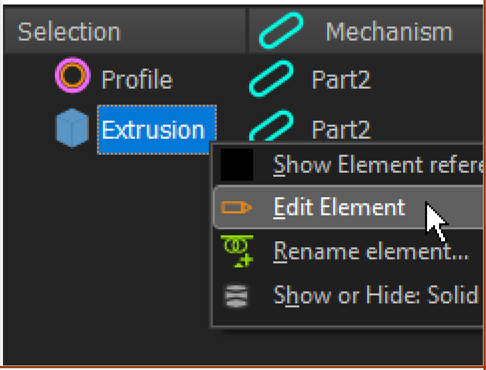
When the **EXTRUSION** (**SOLID**) is not in the Graphic-Area

GRAPHIC-AREA - OPTION 1:

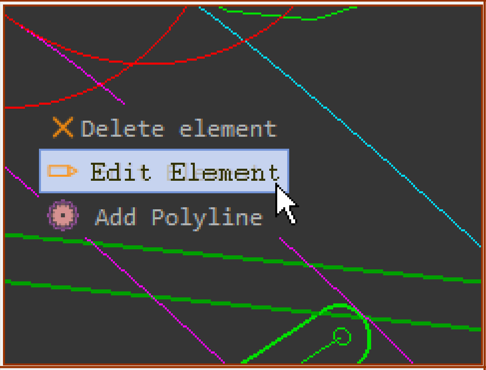
The image to the left shows **PROFILES** contours (Pink is the default color) for two elements in the graphic-area

1. **MECHANISM-EDITOR:** Click a Mechanism **name-tab** to make it the active Mechanism-Editor
2. **MECHANISM-EDITOR:** Spin the View to show the Pink **PROFILE** contours in the graphic-area
3. SHIFT+ Click a **PROFILE**



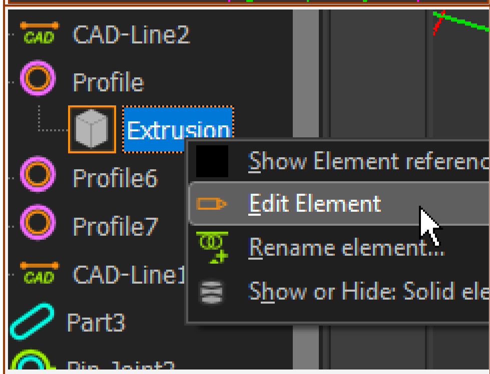


4. **SELECTION-WINDOW**: Make sure the **PROFILE** and **EXTRUSION** show in the **SELECTION-WINDOW**
5. **SELECTION-WINDOW**: Right-click the **EXTRUSION** in the **SELECTION-WINDOW**
6. **SELECTION-WINDOW**: Click **Edit element** in the shortcut menu



GRAPHIC-AREA - OPTION 2:

1. **MECHANISM-EDITOR**: Hover + Click one **PROFILE/EXTRUSION**
2. **MECHANISM-EDITOR**: Right-Click the **PROFILE/EXTRUSION** in the graphic-area.
3. **MECHANISM-EDITOR**: Click the **Edit Element** in the shortcut menu.



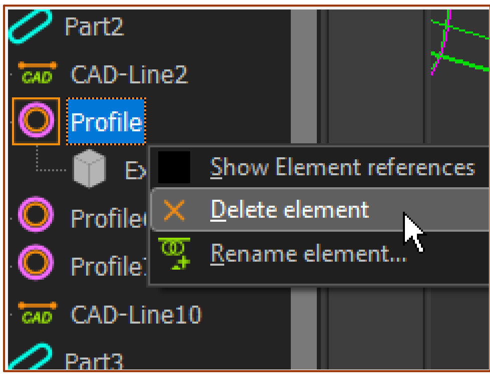
ASSEMBLY-TREE - OPTION 3:

1. **MECHANISM-EDITOR**: In the graphic-area, Click the **PART** that is the parent to the **PROFILE**.

In the **ASSEMBLY-TREE**, the **PART** element shows with a square around the Element icon

2. **ASSEMBLY-TREE**: Expand the tree for the **PART** to show the **PROFILE** and **EXTRUSION** elements
3. **ASSEMBLY-TREE**: Right-click the **EXTRUSION**
4. **ASSEMBLY-TREE**: Click **Edit element** in the shortcut menu

Delete a Profile?



Delete the Profile *(and the Extrusion)*

1. **MECHANISM-EDITOR**: Click a **PROFILE** in the **ASSEMBLY-TREE** or the graphic-area

The **PROFILE** shows in the **SELECTION-WINDOW**

2. Right-Click the **PROFILE**
3. Click **Delete element** in the shortcut menu

1.11.3.1.4 ... change the assembly configuration of a mechanism?

See : Change the Closure of a Dyad [101]

1.11.3.1.5 ... move a Motion-Dimension (not the FB)?

How do I move a Motion-Dimension - (Rocker and Slider)?

CTRL + Drag the **MOTION-DIMENSION FB** to move the Motion-Dimension.

1.11.3.1.6 ... export a Cam directly to SOLIDWORKS

Export a smooth Cam

Calculate the coordinates of the cam profile with a Cam-Data FB [361].

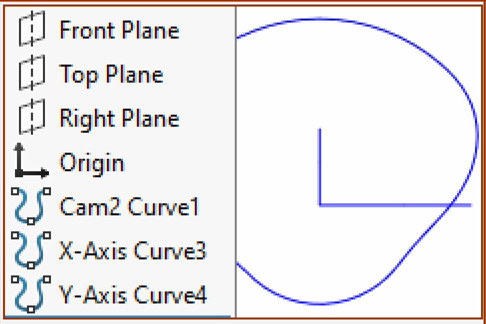
You can calculate the cam profile as a series of Points or as a series of Biarcs.

To export the cam profile data to SOLIDWORKS from MechDesigner as a series of Points

Open a CAM-DATA DIALOG:

1. Calculate the Cam-Coordinates at **EQUAL INCREMENTS** of the Master Machine Angle - say 360 for a typical plate size of cam
2. Make sure the SOLIDWORKS is open, and a Part document, existing or new, is the active document.
3. Click the send SOLIDWORKS button in the toolbar.

The data in SOLIDWORKS is a Curve feature.



The Curve feature is on the XY-Plane, which is the Front View in SOLIDWORKS.

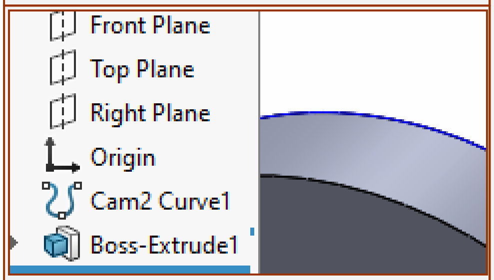
Note: To move the Curve feature to the XZ or the YZ Plane. In the CAM-COORDINATED DIALOG:

1. Save the Data as a TXT file-type
2. Edit the data in Excel®, and swap the X-data, or Y-data, with the Z-data
3. Save the data to NotePad®

In SOLIDWORKS:

4. Click Insert menu > Curve > Curve Through XYZ Points
5. Import the data, and close the Curve feature dialog
6. Select a Plane that is coplanar or parallel to the Curve feature
7. Start a sketch on the Plane
8. Use ***Convert Entities*** in SOLIDWORKS® sketch editor
9. Extrude the Curve.

You can see the Cam does not have any facets.



To import into other CAD

Open Excel®

1. Open the file, with **file-type as...** *.TXT

2. Select delimiter, as a ',' or 'tab'.

You want the 'X' and 'Y' Cam Points in separate columns.

Experiment with the import options if necessary.

3. Tidy up the data, remove headers etc.
4. Move all the data so the first X Point starts at Cell A1, and the first Y Point at B1.
5. Add a Column for the Z-Axis, and fill it with Zeros (0).
6. If necessary, make the first and last point to be identical.

Experiment as necessary.

7. If necessary, scale the Cam-Data to the units you are working in your CAD
8. Save your data as an Excel® spreadsheet, for back-up purposes.
9. Save your Data to a NotePad® file.

You use the file that you save in Excel® or NotePad® to import into your CAD.

Import Cam Data into Solid Edge

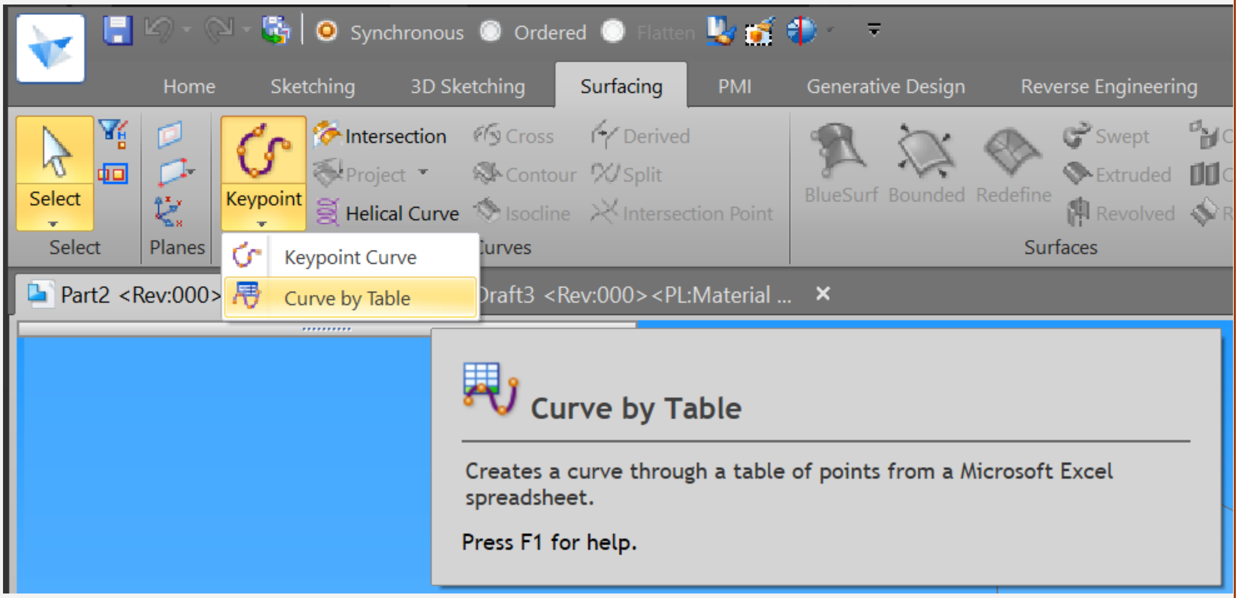
Insert Object / Curve by Table

Select:

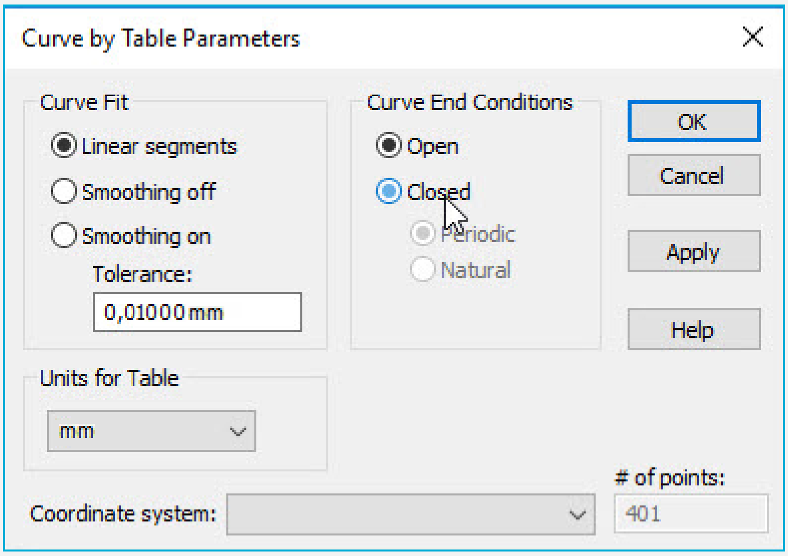
1. Surfacing tab → Curves group → Keypoint → Curve By Table
2. On the Insert Object dialog, set the **Create from File** option and click OK.

Browse to and Open your Excel® spreadsheet

Curve by Table uses an Excel spreadsheet to define a construction curve. The spreadsheet, which is embedded in the Solid Edge document, allows you to more easily import your data.



Solid Edge : Keypoint > Curve by Table



Solid Edge: Curve by Table Parameters dialog

Curve by Table Parameters

Curve Fit

- Linear Segment

Creates line segments between the data points in the table.

- Smoothing Off

Applies the direct curve fit method to the data points in the table. Using this method, the curve passes directly through the points.

- Smoothing On

Applies the 'least squares' curve fit method to the data points in the table. When you select this option, you enable the tolerance drop list. Using this method, the curve path is controlled by the tolerance value and does need to touch the points.

- Tolerance

Allows you to specify a tolerance for the Smoothing On option. It is the maximum distance the curve can deviate from the points.

Curve End Conditions

- Open

To specify an open curve. For example, the first and last data points do not touch each other or any part of the curve.

- Closed

To specify a closed curve. For example, the first and last data points can touch one another or any part of the curve. When this option is set, the Periodic and Natural options become available.

- Periodic

Connects the first data point with the last data point to make a smooth tangent closed curve.

- Natural

Creates a closed curve without a tangency condition.

Note: The **Curve by Table** feature fails if the first and last data points in the spreadsheet are coincident. If you select 'Closed', delete the last point in the table if it is coincident with the first point.

Coordinate System

Allows you to select coordinate systems to offset the curve data to. Make the coordinate system prior to creating the curve through table of points.

Number-of-points

Records the total number-of-points in spreadsheet.

Possible Problems with Curve by Table

The following are possible error messages resulting from Curve by Table:

- Invalid Geometry: Edit Feature Inputs

  The data points in the spreadsheet are incorrect. Check the spreadsheet and make certain you have input at least two rows of X,Y,Z coordinates, have not skipped any cells, and have not entered data points that define a curve that runs back through itself either in 2D or 3D.

- Curve Is Self Intersecting

  The curve runs back through itself in 2D or 3D. Change the data points in the spreadsheet.

- Units Set Out Of Range

  The units are too large. Make sure the diameter of the model is less than five kilometers.

- Curve By Table Feature Failed

  Closed and Periodic options are set on the Curve Table Parameters dialog, and the first and last data points in the spreadsheet are coincident. Change one of the coincident data points.

1.11.3.1.7 ... model a Pin in a Slot?

See INTERNET LINK MechDesigner: Getting Started Tutorial 2A: Four-Bar Kinematic-Chains STEP 2.8 Kinematic-Chains: Model a Pin in a Slot.

1.11.3.2 FAQs

FAQs

Troubleshoot.

1.11.3.2.1 What are Part Outlines and their Colors?

What are Part-Outlines

A **PART-OUTLINE** is the symbol for a **PART**. You can change the size of the **PART-OUTLINE** in the Edit menu > Application Settings | Accessibility tab 289.

You cannot edit a **PART-OUTLINE**.

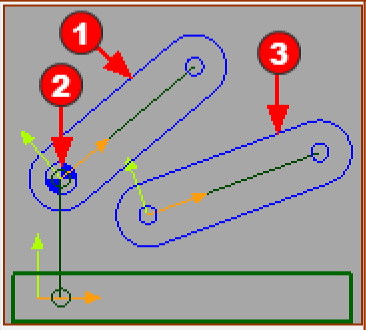
- Double-click a **PART-OUTLINE** to edit the **PART** in the **PART-EDITOR**.
- The color of the **PART-OUTLINE** helps you identify if the **PART** is **kinematically-defined.**

Why do Part-Outlines have different colors?

You can edit the colors of the Parts that are kinematically-defined and not kinematically-defined in the

Edit menu > Application-Settings > Graphic tab > Display Colors 284.

Look for **PART SOLVED** and **PART NOT SOLVED.**



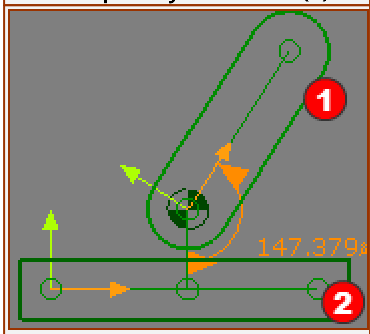
Free Part(1) ;
Completely Free Part(3)

Blue Part-Outlines - or near to Blue.

The **PART** is NOT **kinematically-defined.**

Other names for **Blue PARTS** are:

- **Completely-Free**❸ - without a Joint
- **Free**❶ - with a Joint, but not **kinematically-defined**
- **Part Not Solved**



Kinematically-Defined Parts
Rocker & Base-Part

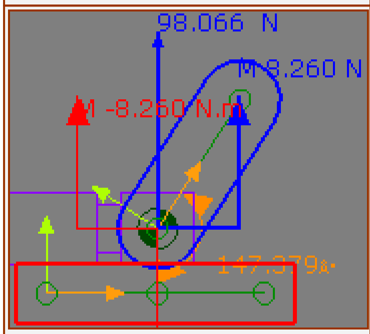
Green Part-Outlines - or near to Green

The **PART** is **kinematically-defined.**

Other names for **Green PARTS** are:

- **kinematically-defined**❶❷
- **Part Solved**❶❷

Note: The **BASE-PART** is always a **PART** that is **kinematically-defined, and therefore near to Green.**



Part-Outlines with Display Forces

Multi-colored Part-Outlines

Each **PART** has a random color when, in the Forces Toolbar 187, you:

- Enable Calculate Forces 190

AND

- Display Force Vectors 49

1.11.3.2.2 What is the hierarchy of Extrusions?

The Parent to Child hierarchy is as follows

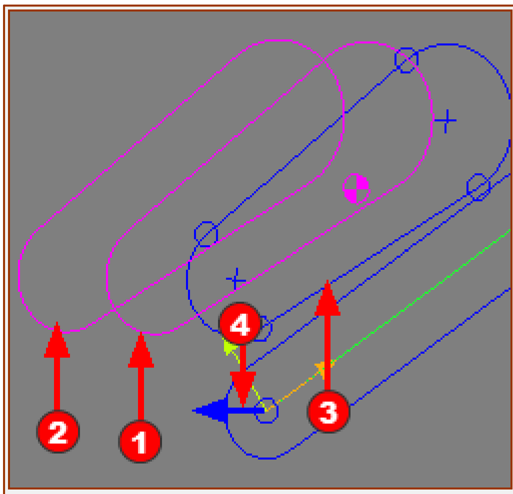
MODEL > PLANE > MECHANISM > PARTS > (sketch-loop) > (Auto) PROFILE > EXTRUSION > HOLE

There are a number of elements that must 'co-exist' to create Extrusions for Solid Modeling. It is helpful to understand the:

- Commands that you need to use
- Actions that **MechDesigner** adds to the model for you.

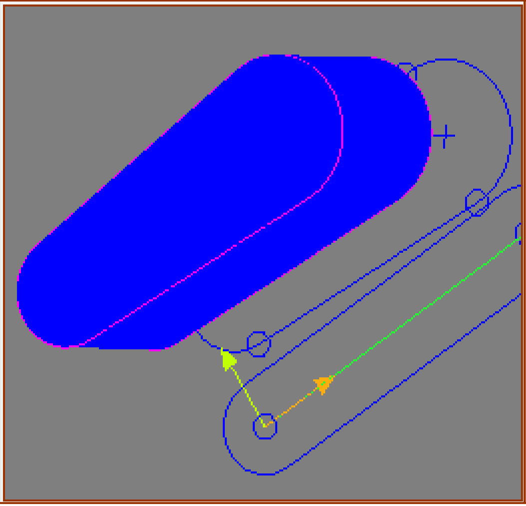
Command 1 :	**Add sketch-elements** to **LINES**, **CAD-LINES**, **ARCS**, **CIRCLES** to define the shape of a sketch-loop OR **Add Polyline** to a **2D-CAM** or a **GEAR-PAIR**				
MechDesigner :	➔	***Sketch-Loops:*** ...we recognize for you a sketch-loop			
Command 2 :		➔	**Add Profile:** Select a sketch-element that one sketch-element in a sketch-loop		
MechDesigner :			➔	***Extrusions:*** ... are added with a Add-Profile	
Command 3 :				➔	**Add Holes:** Through Extrusions

See Tutorial 4 How to add sketch-loops 545



Information:

- A Sketch-Loop is series of sketch-elements❸, joined end-to-end, or a Circle, added to a Part.
- The Sketch-Loop cannot have branches - 'T-bars'
- The Sketch-Loop cannot cross itself - 'Figure-of-Eights'.
- Sketch-Elements and Sketch-Loops do NOT show in the Assembly-Tree.
- To add a Profile, select a Sketch-Loop❸, to define the shape - called the contour - of the Profile
- The Profile is two contours: the Primary Contour❶ and the Secondary Contour❷.
- The Profile contours are offset from each other, they can only be offset along the Z-axis of the Part; the Secondary Contour is always at a more positive Z-axis value (it is in-front of the Primary Contour when viewed from the 'Front' view).
- The Extrusion❹ simply fills the space between the Primary and Secondary Contours of the Profile.
- Extrusions have a few more properties: visibility, density, color, opacity, etc.



- The location of the Profile contours are edited with the Extrusion dialog.
- You can sketch many Sketch-Loops in each Part - hence Add many Profile and Extrusions
- You can supply many Profiles from one Sketch-Loop - each with different offsets
- You can use a sketch-loop to cut a hole through an Extrusion

1.11.3.2.3 What does kinematically-defined mean?

Kinematic states of PARTS

There are two kinematic-states of **PARTS**.

Not kinematically-defined :	The state of a **PART** when it has **one** or **more** degrees-of-freedom.
kinematically-defined :	The state of a **PART** when it has **zero** **Mobility**. Its position, velocity, and acceleration are fully specified.

Mobility of a Kinematic-chain =

Degrees-of-Freedom – # Motion-Dimensions.

 # Degrees-of-Freedom $= 3.(N-1) - 2.J$

N = Number of **PARTS**

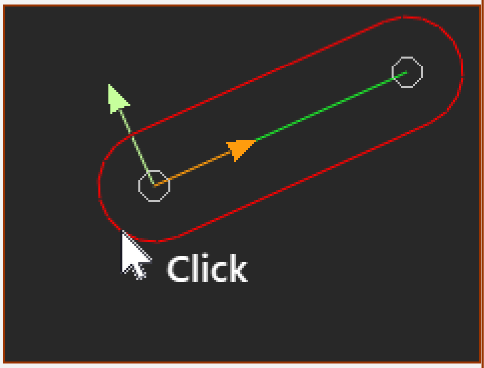
J= Number of **JOINTS**

All **PARTS** in the model must be **kinematically-defined** **before** you should analyse any motion and force data.

1.11.3.2.4 I cannot select the Part-Outline! What to do?

Occasionally, with some graphic-card, especially on Laptops, you cannot select the **PART-OUTLINE** easily.

Try these different methods to see which works best for you.



METHOD 1: Select near start-Point.

METHOD 1:

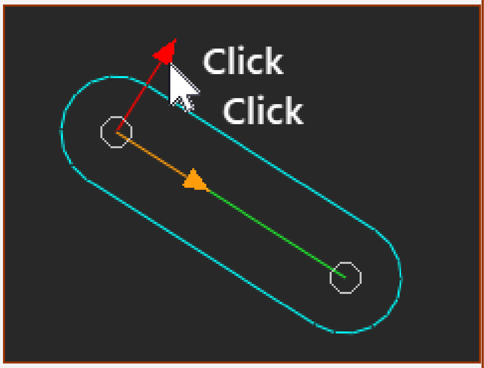
1. Move your pointer above the ***arc*** of the **PART-OUTLINE** - see image
2. Double-Click your mouse

or

2. Click your mouse

The **PART** should be in the **SELECTION-WINDOW**

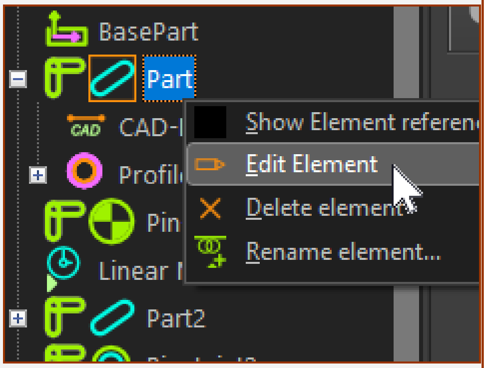
3. Right-Click the **PART**
4. Click **Edit element** in the shortcut menu



METHOD 2: Double-click the Y-axis

METHOD 2:

1. Double-click the **Y-axis** to edit a PART,



METHOD 3: Select in the Assembly-Tree

METHOD 3:

In the ASSEMBLY-TREE:

1. Right-Click the PART
2. Select **Edit element** in the shortcut menu

1.11.3.2.5 What are the different 2D-Cam elements

2D-Cam: Elements, Properties, Parameters, Analysis, Coordinates

These are the different elements and Function-Blocks you need to know, and how they are related.

Usually, do these steps

1. Add a 2D-CAM - **see** **Add 2D-Cam** [109]

If the new 2D-CAM is one of a **Conjugate-Cam pair**, or one Flank of a **Groove-Cam** (BODY-CLOSED CAM) then:

1.a. Add a CONJUGATE-CAM FB

1.b. Edit the CONJUGATE-CAM FB to add the inner and/or outer flanks of two 2D-CAMS - **see** **Conjugate-Cam dialog** [353].

Then:

2. Select the 2D-CAM or the CONJUGATE-CAM as the **Power Source** for the **Cam-Follower** - **see** **Configure-Power Source** [480]

Then:

3. Open the 2D-CAM DIALOG [327] to review 2D-CAM'S PROPERTIES.

2D-CAM PROPERTIES include:

- Display the INNER OR OUTER, INNER AND OUTER, PITCH-CIRCLE
- Display Cam-Profile as PROFILE-ONLY, PRESSURE ANGLE, CONTACT-FORCE, SHEAR-STRESS
- Indicate at Contact-Point the PRESSURE ANGLE, CONTACT-FORCE, SHEAR-STRESS, RADIUS-OF-CURVATURE
- Display COLOR, LINE THICKNESS of CAM-PROFILE

- **ROLLER LIFE**: Edit the Roller manufacturer, Roller model, Oil/Grease Lubrication Temperature, Oil/Grease Properties, Oil/Grease Contamination
- **CAM LIFE**: Edit the Cam's Steel Category, Steel Quality, Steel Heat-Treatment, Steel Hardness.

Then:

4. Add a CAM-DATA FB - **see** **Add Cam-Data FB** [171]
5. Link a 2D-CAM to the CAM-DATA FB - **see** **Link the Cam-Data FB to a 2D-Cam** [357]

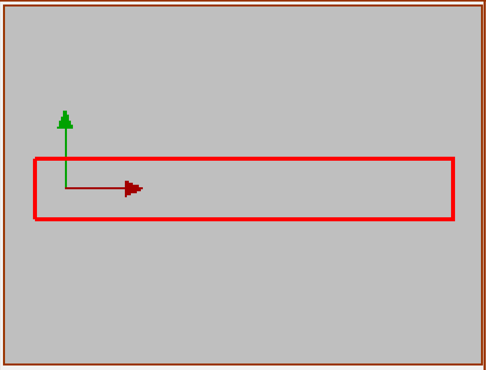
Then:

6. Connect wires from the CAM-DATA FB to a GRAPH FB to analyse 5 different parameters of the 2D-CAM - **see** **Cam-Analysis Parameters** [358]
7. Calculate the Cam-Coordinates - **see** **Cam-Coordinates dialog** [361]

Output-Connectors from CAM-DATA FB:	Calculate Cam-Coordinates [361]
➔ CONTACT-FORCE ➔ CONTACT-SHEAR-STRESS ➔ RADIUS-OF-CURVATURE ➔ PRESSURE-ANGLE ➔ SLIDING VELOCITY	➔ Save as DXF, TXT, STEP ➔ Transfer Cam directly to SOLIDWORKS ➔ Save as: XY-Coordinates, Biarcs, STEP

1.11.3.2.6 Why can I not see the Base-Part properly?

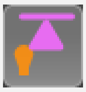
Why is the Base-Part not Green when I had a Mechanism-Editor?



Base-Part with Forces ON

The Base-Part is a Random color.

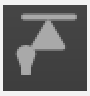
If the BASE-PART is a not Green:



Forces toolbar > Force Vectors: Display to display the Force Vectors.

When this icon is enabled to display Force Vectors, each PART is given a random color.

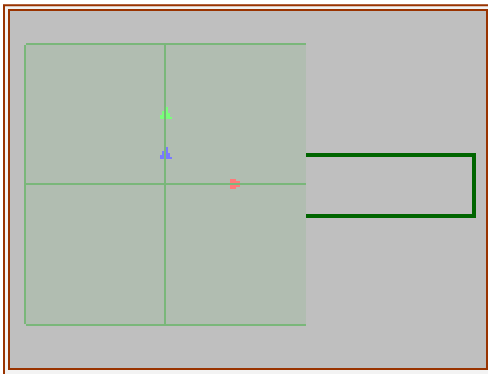
To make the PART Green, you should:



Click the Force Vectors: Display to not display the Force Vectors

When the icon is disabled the BASE-PART should be Green.

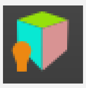
Why can I not see the Base-Part properly?



Base-Part partially hidden by a Plane.

The Base-Part is hidden by a Plane.

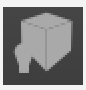
This is because:



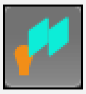
Visibility toolbar > Show Solids in Mechanisms is enabled

When you show the **SOLID** elements in **MECHANISM-EDITORS**, we also show the PLANES.

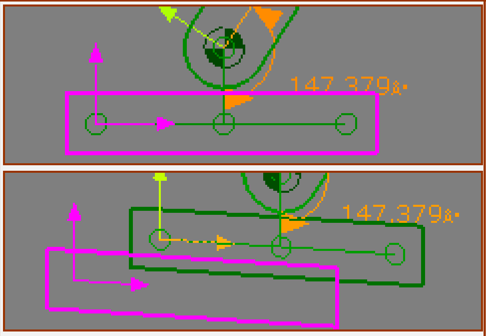
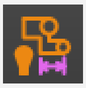
To hide the PLANES:



Click Visibility toolbar > Show Solids in Mechanisms disabled.

	OR  Click **Display toolbar > Display Planes to Hide Planes** (54)

Why is the Base-Part Pink?

	 Visibility toolbar > Show other Kinematic and Sketch elements If this Visibility Toggle is active, and if another MECHANISM-EDITOR is in front of the active MECHANISM-EDITOR, you see the Pink* **Other Base-Part** (* Default Color) If you spin the model you can see the BASE-PART of the active MECHANISM-EDITOR. * See Application-Settings > Graphics tab > Display Colors > Background Sketch

1.11.3.2.7 I cannot add an Angle (or Linear) Motion Dimension FB. Why?

Motion-Dimension: Commands and Elements.

You must select three elements when you add a MOTION-DIMENSION FB for a ROCKER. They are:

- A PIN-JOINT...
- ...followed by two LINE sketch-elements.

However, you may find you cannot select one of the LINES to complete the command.

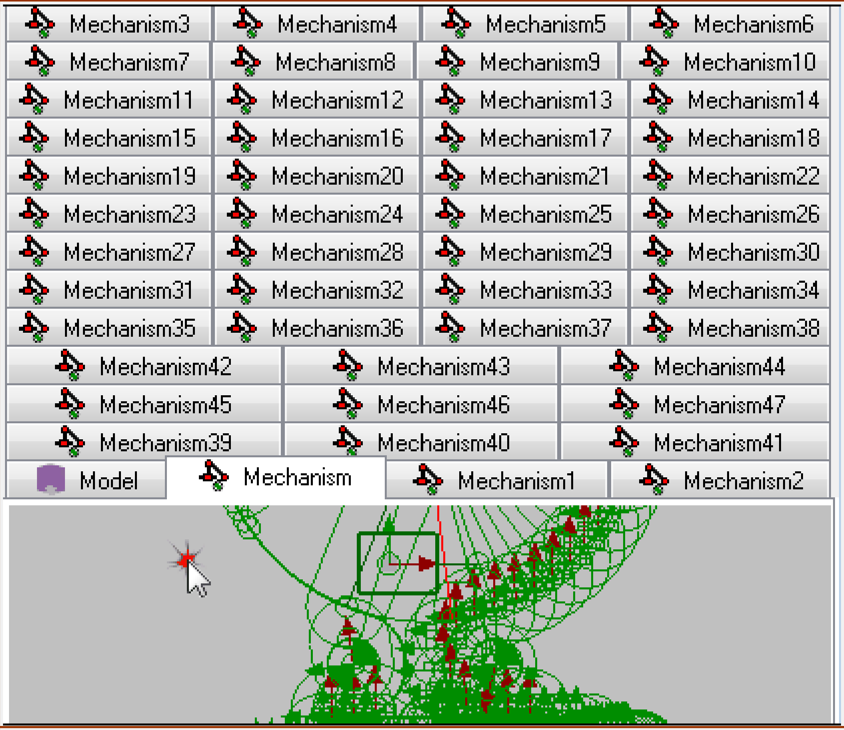
This can happen when:

- there is already a MOTION-DIMENSION FB at the PIN-JOINT
- the POINT at the PIN-JOINT is not a child to the LINE that radiates from the PIN-JOINT.
- the PIN-JOINT that you select is not a PIN-JOINT between the two LINES you want to select.

Read this topic: **Add Motion-Dimension** (154).

1.11.3.2.8 How many Mechanism Editors can I add?

How many Mechanism-Editors/Tabs can I add?



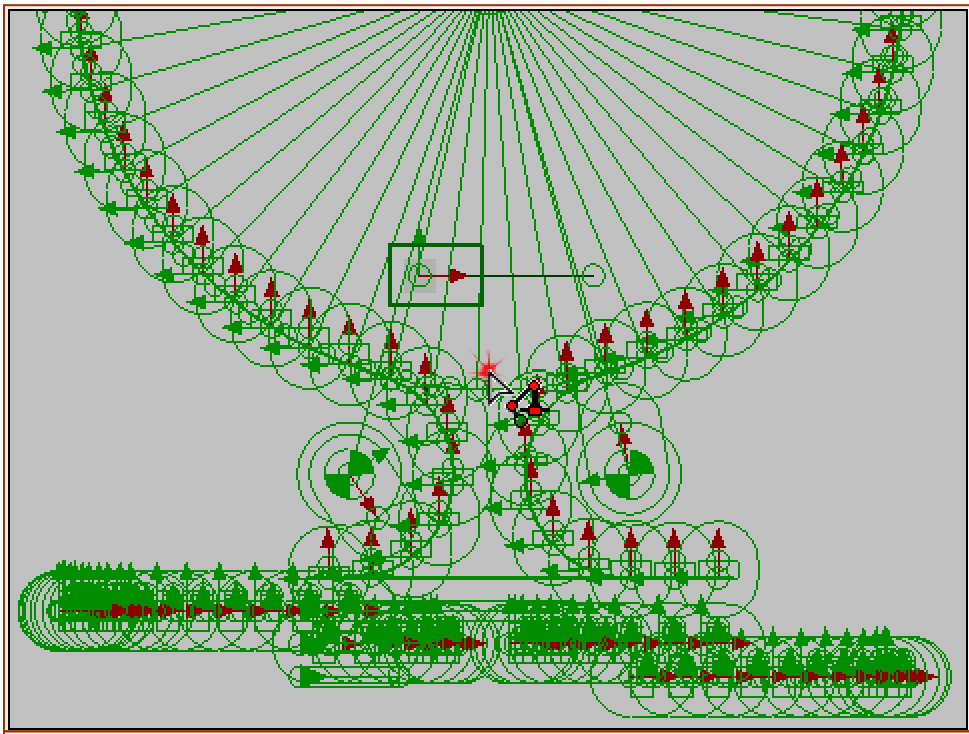
You can add as many Mechanism-Editors and Tabs to as many Planes as you like.

It is possible to design complete machines in one Mechanism-Editor. Alternatively, you can model a machine with many Mechanisms Editors, each contributing a kinematic-chain.

The model to the left shows 47 Mechanism-Editors.

1.11.3.2.9 How many Kinematic-Chains in a Mechanism Editor?

How many Kinematic-Chains can I add to each Mechanism-Editor?



You can add as many Kinematic-chains as you need to model your machine. Within the capacity of your PC and RAM.

For example, in the image to the left, I have modeled 160 kinematic-chains on one Mechanism-Editor - all Sliders with a Motion

1.11.3. Troubleshooting 3

PROFILES are not above the sketch-loop?

To correct:

1. Home the model
2. Edit a **PART**, then close the **PART-EDITOR** immediately.
3. Rebuild the model.

1.11.4Part Editor: FAQs and How to...?

Part-Editor: FAQs and How to ...?

How to...?

FAQs

1.11.4.1 How to...?

How to...?

1.11.4.1.1 ... start the Part-Editor

How to start (open) the Part Editor

See also: **How to close the Part-Editor** [217]
See also: **I cannot select the Part-Outline. What to do?** [533]

Notes:

The **PART-EDITOR** can edit ***one*** PART at a time.

To edit a different PART, you must close the **PART-EDITOR** to return to the **MECHANISM-EDITOR**, and then edit the different PART.

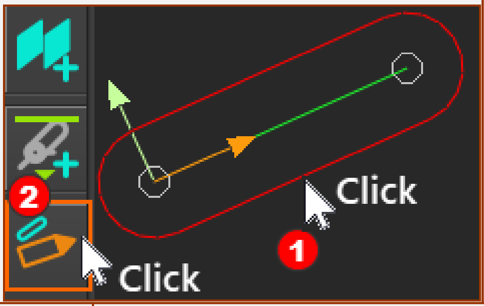
Video: **How to start the Part-Editor:**

Double-click to watch

How to start the Part-Editor - more details

To start (open) the **PART-EDITOR**, do one these methods.

METHOD 1: Edit-Part icon



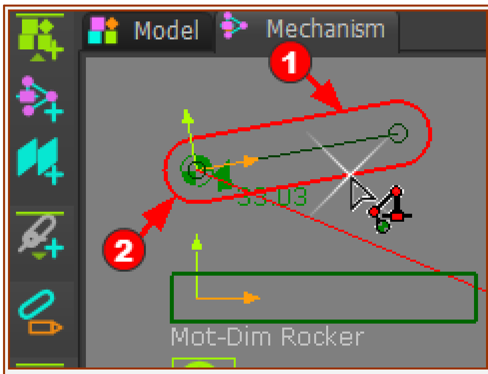
STEP 1: Click a **PART-OUTLINE** ❶ in the graphic-area
STEP 2: Click **Edit toolbar > the Edit Part in Part-Editor** ❷

The **PART-EDITOR** is now open.

Note: The **Edit Part** icon is to the left of the graphic-area.

See also: **How to close the Part-Editor: Method 1** [217]

METHOD 2: Selection-Window



STEP 1: Click a Part-Outline

STEP 1: **Select the Part**

Graphic-area:

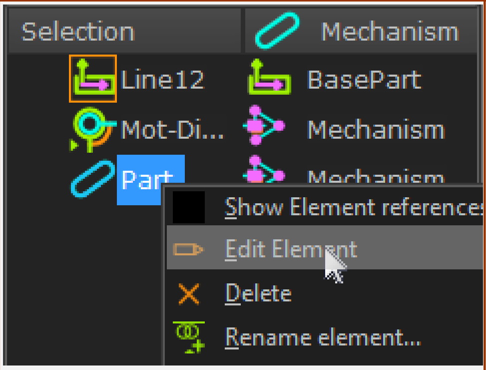
1. Click the **PART-OUTLINE** ❶ (or ❷ with some graphic-cards)

OR

Assembly-Tree:

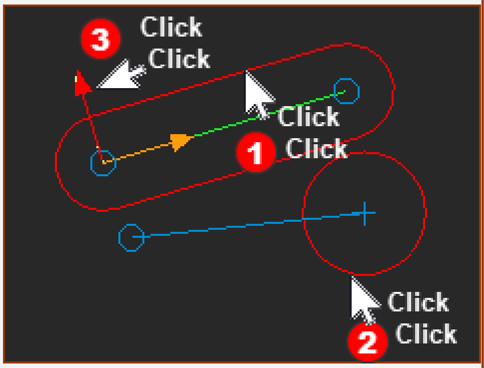
1. Click the PART element

STEP 2: **In the Selection-Window**

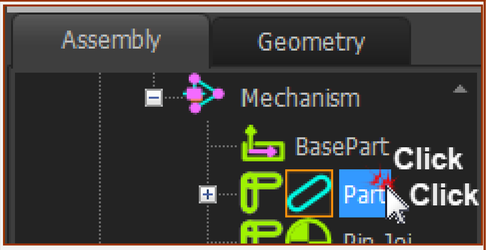


STEP 2: Right-click the Part in the Selection-Window, Click Edit element

METHOD 3: Double-Click



Double-click a Part-Outline in the graphic-area



Double-click a Part in the Assembly-Tree

STEP 1: Double-click ...

... the Part-Outline in the Graphic-area:

1. Double-click the PART-OUTLINE❶

OR

1. Double-click a LINE*, ARC*, or CIRCLE* SKETCH-ELEMENT❷ that you have added to the PART

OR

1. Double-click the Y-AXIS❸ of the PART

The PART is now open in the PART-EDITOR.

OR

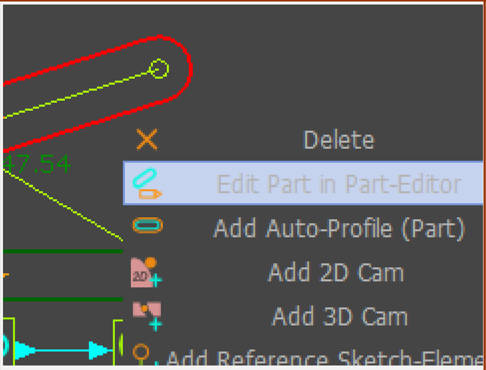
... the Part element in the Assembly-Tree:

1. Double-click the PART element

The PART is now open in the PART-EDITOR.

* If you double-click a:	
CAD-LINE	... the CAD-LINE DIALOG 301 opens
BLEND-CURVE	... the BLEND-CURVE DIALOG 449 opens
POINT	... the POINT PROPERTIES DIALOG 444 opens
More than one element	... nothing happens.

METHOD 4: Right-Click



Hover+ Right-click a Part-Outline in the graphic-area

In the graphic-area:

1. Hover so the **PART-OUTLINE** becomes ***red***

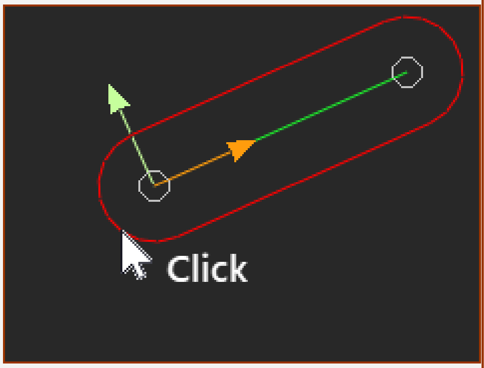
Note: With some graphic-cards, you must move your mouse-pointer to the arc of the Part-Outline near to the Origin❷.

2. Right-Click the **PART-OUTLINE**
3. Click **Edit Part Geometry** from the shortcut menu.

The **PART** is now open in the **PART-EDITOR**.

Occasionally, there is a problem with a Graphic-Card

With some graphics cards, you cannot select the **PART-OUTLINE** to edit a **PART**. Try these methods:



Select the Part-Outline near start-Point.

Method 1: Graphic-area

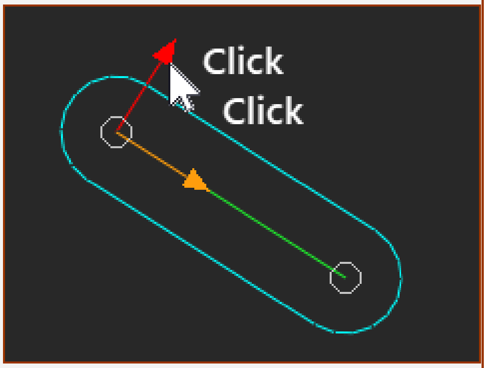
1. Move your mouse-pointer above the ***arc*** of the **PART-OUTLINE**
2. Double-click your mouse

Method 2: Selection-Window

1. Move your pointer above the ***arc*** of the **PART-OUTLINE** - see image
2. Click your mouse one time.

Then, in the **SELECTION-WINDOW**:

3. Right-Click the **PART**
4. Click **Edit element**

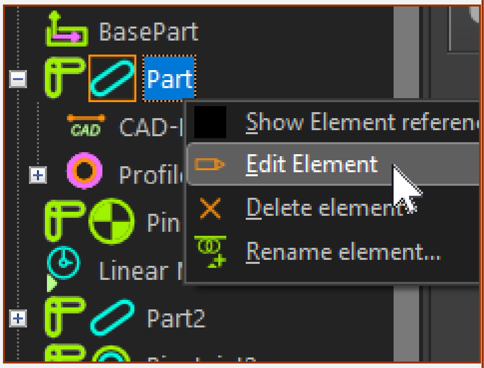


OR Double-click the Y-axis

Method 3:

OR

1. Double-click the **Y-axis** to edit a **PART**,



Select a Part in the Assembly-Tree

Method 4:

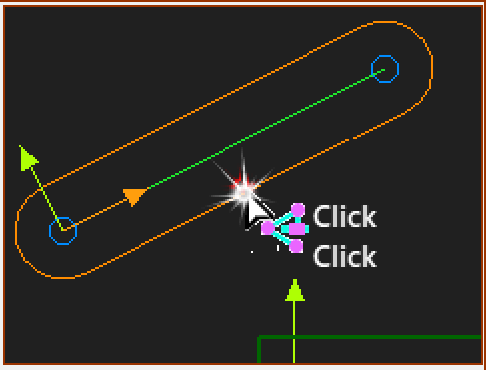
OR

In the **ASSEMBLY-TREE**

1. Right-Click the **PART**
2. Select **Edit element** in the shortcut menu.

1.11.4.1.2 ... edit the Length of a Part?

How to edit the Length of a Part, or to edit a Dimension



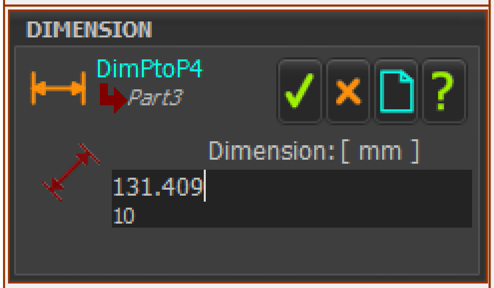
Part in Mechanism-Editor

Start the Part-Editor.

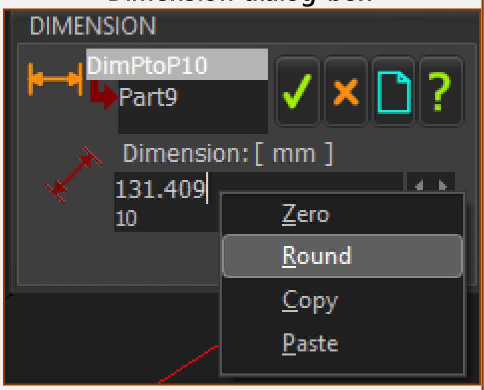
STEP 1: Double-Click the Part-Outline of the Part you want to edit

You are now using the Part-Editor.

Double-click arrowhead of dimension



Dimension dialog-box



Dimension dialog-box - contextual menu

The length of the Part is the dimension between the start-Point and end-Point:

STEP 2: Double-Click the arrowhead of the dimension

The DIMENSION DIALOG opens.

STEP 3: Enter a new dimension

There are four(4) methods you can use to enter a new value to edit the dimension:

A: Enter a new dimension directly with your keyboard

B: Enter an equation with your keyboard

C: Use the spin-box tool

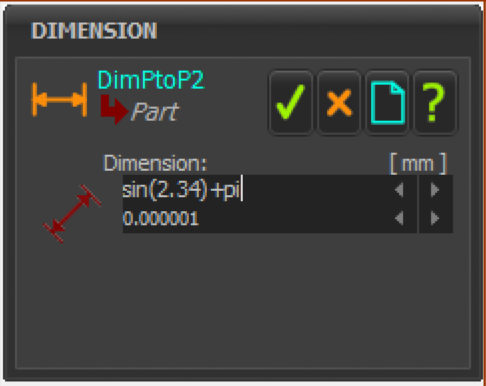
D: Right-click the data-box, then click **Zero, Round, Copy or Paste.**

Notes:

A & B: Direct and Equation: Click the ENTER key (↵) on your keyboard to update the dimension to the new value.

C: Spin-Box: The dimension AND the model updates immediately - if the model is large, there is a delay after each click

D: Round: The Dimension changes to the nearest value that is exactly divisible by the Spin-Increment



Enter an equation for the dimension

More on EQUATIONS:

You can enter:

- **Simple Arithmetic**: +, −, *, /
- **Powers**: ^, Sqrt()
- **Trigonometric (Angle in Radians)**
    - Sin(), Cos(), Tan(),
    - Sihn, Cosh, Tansh
    - ArcSin(), ArcCos(), ArcTan2(;)

1.11.4.1.3 ... delete Sketch-Elements

How to delete Sketch-Elements:

To delete **ONE** sketch-element:

1. Click the element in the graphic-area

   The element shows in the **SELECTION-WINDOW**.

2. Right-click the element in the **SELECTION-WINDOW**
3. Click **Delete** in the shortcut menu

OR

To delete **MULTIPLE** sketch-elements

1. Drag a window to select multiple sketch-elements in the graphic-area.
2. Click the **Delete** key on your keyboard

1.11.4.1.4 ... delete Constraints

How to delete a sketch Constraint

You cannot see Geometric-Constraints in the graphic-area.

1. SHIFT+CLICK a sketch-element in the graphic-area

   The sketch-element AND any constraints that you have added to it show in the **SELECTION-WINDOW**.

2. Right-click the constraint in the **SELECTION-WINDOW**
3. Click **Delete** in the shortcut menu

Note: If a Constraint has been added specifically to a **Point**, (a **POINT**, **sTART-POINT**, **END-POINT** or **CENTER-POINT**) then you must click the **Point**, and not the sketch-element, to see the constraint in the **SELECTION-WINDOW**.

1.11.4.1.5 ... add a Sketch-Loop?

How to add a sketch-loop.

A **sketch-loop** is a continuous 'string' of sketch-elements. The **START-POINTS** and **END-POINTS** are merged into one **POINT**.

A **sketch-loop** can be a **closed sketch-loop** or an **open sketch-loop**.

About Merge-Points and Coincident-Constraint

Each **LINE, ARC, BLEND-CURVE** has a **POINT** at its ends. These **POINTS** are the **START-POINT** and **END-POINT**.

To join sketch-elements, into a continuous string, you can:

- Add a Coincident Constraint between the **END-POINT** of the one sketch-element with the **START-POINT** of another sketch-element

  See Constraints toolbar > Add Coincident [255]

  Add Coincident does not remove a **POINT**.

OR

- Merge the **END-POINT** of a one sketch-element with the **START-POINT** of another sketch-element

See **Geometry toolbar** > Merge-Points[240]

Add Merge-Points does remove a **POINT**

Why Merge-Points?

You can add a **PROFILE** to a sketch-loop.

You can add a **MOTION-PATH FB** (with **MOTION-POINT**) to a an open or closed sketch-loop. This means you can add a Belt, guide the motion-path for a Robot, ...

Sketch-Loops with the Hover Technique and Merge-Points Tool

There are two ways to merge two **POINTS** into one **POINT**:

- Dynamically, as you add the sketch-elements, with the **HOVER TECHNIQUE**
- Later, with **Geometry toolbar** > **Merge-Points**.

Video of Hover Technique[241]

Why do Sketch-Loops fail?

Condition 1: **COINCIDENT POINTS, AND NOT MERGED**

POINTS that are coincident are not recognized as part of a sketch-loop.

Condition 2: **BRANCHES**

You CANNOT merge three **POINTS** into one **POINT** - for example, the ends of three **LINES** or **ARCS** - you have a '**BRANCH-POINT**'. A **BRANCH-POINT** is not O.K.

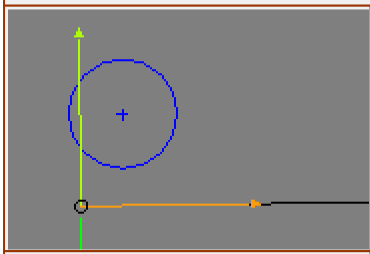
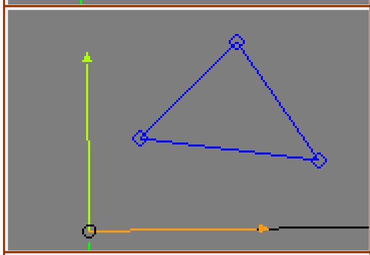
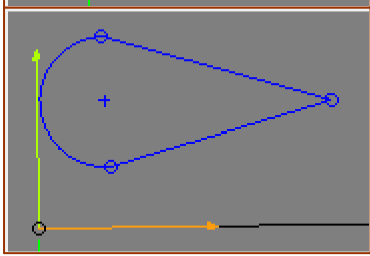
Condition 3: **FIGURES=OF-EIGHT**

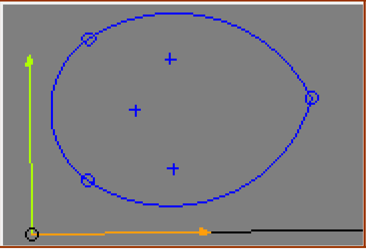
A sketch-element can not cross another sketch-element in the same sketch-loop.

A sketch-element cannot touch, another sketch-element in the same sketch-loop.

Example sketch-loops

Video of Hover Technique[241]

	**Circle** The **CIRCLE** is a sketch-loop. A **CIRCLE** is frequently used for a Cam-Follower.
	**Lines** **LINES** joined, with **POINTS** merged.
	**Lines and Arcs** **LINES** and **ARCS**, with **POINTS** merged.



Arcs and Arcs

ARCS and **ARCS**, with **POINTS** merged.

1.11.4. FAQs 2

FAQs

1.11.4.2.1 How many Parts can I edit at one time?

How Many Parts can I edit at a time

You can edit **one** **PART** at a time.

To edit a different **PART**, you must:

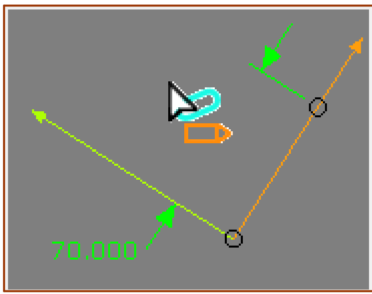
1. Close the **PART-EDITOR** to return to the **MECHANISM-EDITOR**
2. Select a different **PART** to edit in the **PART-EDITOR**

1.11.4.2.2 Why edit a Part?

Why Edit a Part?

There are many reasons why you may want to edit a **PART**.

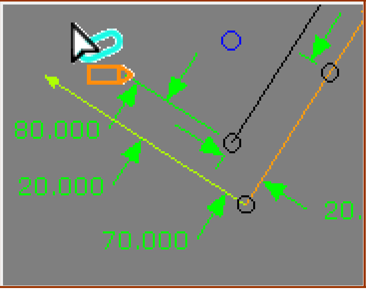
Edit the Length of Parts



You must edit the **PART** to specify its exact length.

See also: Edit the Length of a Part, Edit Dimension 543

Pin-Joints, Slide-Joints and Ball-Joints

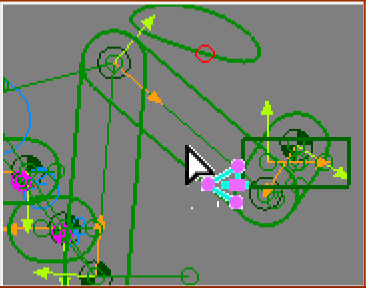


In a **PART**, you can use the CAD-Line for a Slide-Joint and its **START-POINT** and **END-POINTS** for Pin-Joints

To add a joint between a **PART** and the **BASE-PART**, you MUST add a minimum of one sketch-element to the **BASE-PART**.

You can also add new **LINES**, **CAD-LINES**, and **POINTS** for joints at other locations in any **PART**.

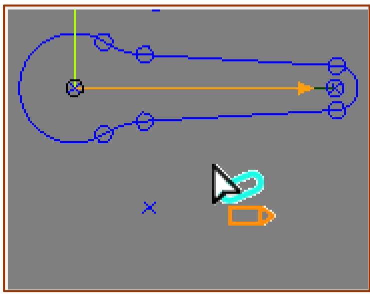
Trace Points



You can add a **POINT** to a **PART** and then view the trace of that **POINT** on the **MECHANISM-PLANE** during a machine-cycle.

See Add Trace-Point [104].

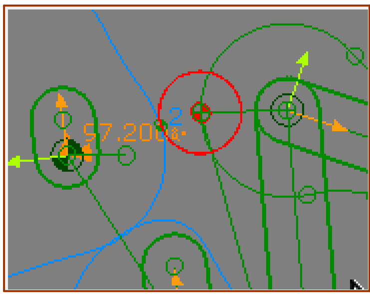
Sketch-Loops



You must sketch a sketch-loop before you can add a **PROFILE / EXTRUSION**. .

Use the **PART-EDITOR** to add the sketch-elements for the sketch-loop.

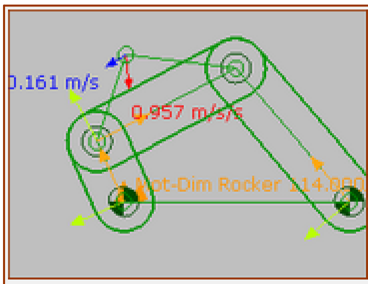
Cam-Follower Profiles



Each Cam needs a Cam-Follower. A Cam-Follower must be sketched in the **PART** as a sketch-loop.

You must also sketch the shape of the 'Cam-Blank' with the **3D-CA**ms.

Plot exact Kinematic Data



For any **POINT** that you add to a **PART**, you can:

- Show the instantaneous kinematic vectors (Velocity and Acceleration) of a **POINT** with the POINT PROPERTIES DIALOG [444]
- Plot motion-values of Position, Velocity and Acceleration throughout the machine-cycle with the POINT-DATA FB [169] or MEASUREMENT FB [166] wired to a GRAPH FB [172]

Add a CAD-Line

You can add **CAD-LINES** to any **PART**. You can use the **CAD-LINE** to import **DXF-DRAWINGS**, SOLIDWORKS parts, or STL file-types.

CAD-LINES can be on top of each other, or at different positions in the **PART**.

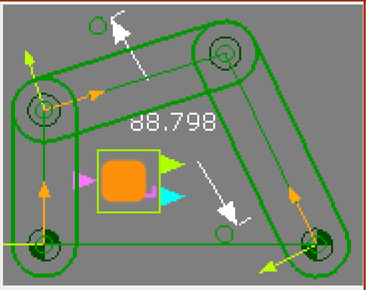
There is also a **CAD-LINE** between the **START-POINT** and **END-POINT** of every **PART** you add to the model.

Constraint Based Editing Tools for Mechanism Synthesis

See Mechanism-Synthesis.

- Three, Four and Five Position Synthesis
- Coupler Curve and Coupler Point Synthesis
- Function Generation

To add sketch-elements for a Measurement FB Dimension



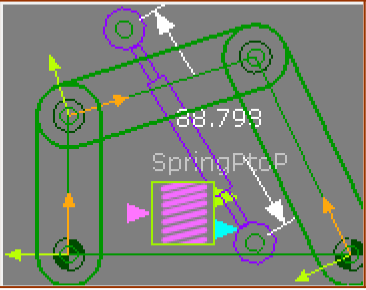
Add a **MEASUREMENT FB** between two sketch-elements. See Add Measurement FB [166]

The output from a **MEASUREMENT FB** is the dimension, velocity, and acceleration between the two sketch-elements.

The output from a **POINT-DATA FB** is position, velocity and acceleration of a **POINT**.

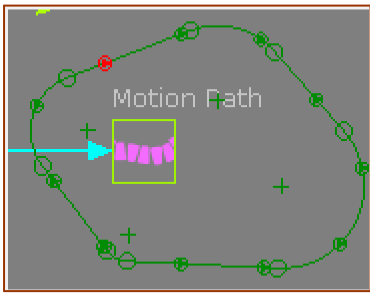
You can use the output of a **MEASUREMENT FB** or **POINT-DATA FB** as the independent variable (X-axis) of a different **FB**.

Add a Point or Line for a Spring FB



Add a **POINT** or **LINE** added to anchor a **SPRING FB**.

Add a sketch-loop for a Motion-Path.

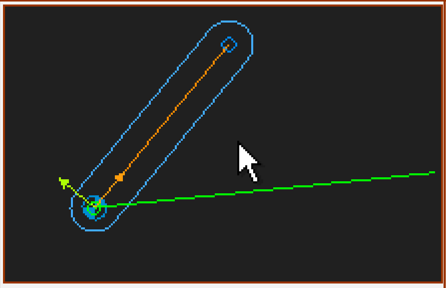


A **MOTION-PATH FB** needs an open or closed sketch-loop.

You must add a **MOTION-PATH FB** to add a **PULLEY** - the sketch-loop is the path of the **BELT**.

1.11.4.2.3 How do I know I am using the Part-Editor?

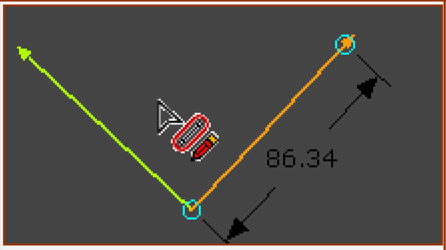
How do I know I am using the Part-Editor and not the Mechanism-Editor?



Part in Mechanism-Editor

Mechanism-Editor:

- You see the Part-Outlines
- The XY-axes are small

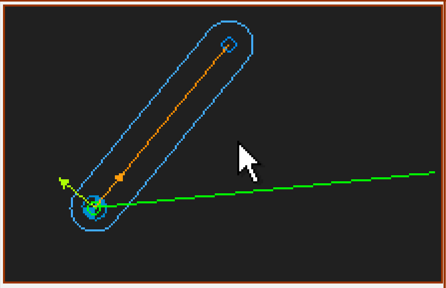


Part in Part-Editor

Part-Editor:

- You cannot see the **PART-OUTLINE**
- The XY-axes are relatively large
- You can see the **DIMENSION** for the length of the **CAD-LINE** from the **START-POINT** to the **END-POINT**.

1.11.4.2.4 Can I edit (and other questions relating to) the Part-Outline?



Part-Outline in the Graphic-Area

Can I edit the Part-Outline?

NO. The **PART-OUTLINE** is the symbol for the **PART** element.

What do I do with a Part-Outline?

You also click, or double-click, the **PART-OUTLINE** to edit the **PART** in the **PART-EDITOR**

It is the symbol of the **PART** that you select to complete other commands; for example: Add **2D-CAM**.

Can I change the size of the Part-Outline symbol?

YES.

Use the Application Settings > Accessibility tab > Graphic > Symbol Display Size [289]

Must I display Part-Outlines in the graphic-area?

NO.

To hide **PART-OUTLINES**, see Display Filters > Show/Hide Part-Outlines [54].

You can double-click the **Y-axis** to edit a **PART**

You must display **PART-OUTLINES** to display **Force-Vectors** [198]

Can I hide individual Part-Outlines?

No.

1.11.4.2.5 Why is the dimension 'negative' in the Dimension dialog'?

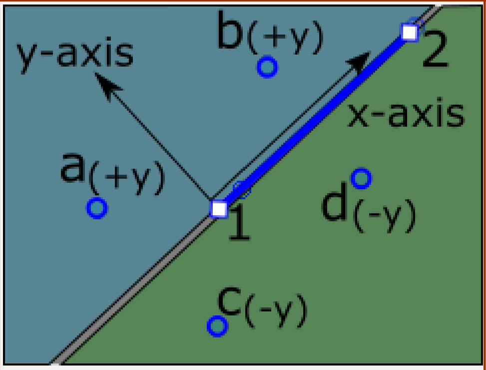
Why is my dimension negative?

Negative Dimensions.

The **– sign** appears if the coordinate of a dimension is **negative**.

Generally, do not delete the **– sign**, but edit the dimension in the normal way.

More Details

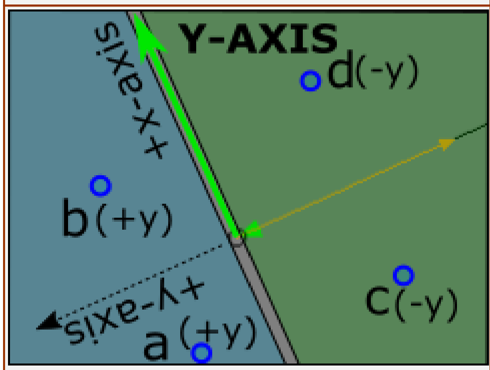
<table>
<tr>
<td>
</td>
<td>LINES have an X-Y coordinate-system.

The Origin of a Line is at its START-POINT
The +X-axis of a Line is from its START-POINT(1) to its END-POINT(2)
The +Y-axis of a Line is at +90º from Line.

When you add a dimension that is a perpendicular-distance from the LINE, the

dimension is Positive if in +y-axis half of the LINE (blue area in the image)
dimension is Negative if in the –y-axis of the LINE (green area in the image)

In the top image:

Dimensions from the LINE(1 to 2) to Points a and b are positive
Dimensions from the LINE to Points c and d are negative
</td>
</tr>
<tr>
<td>
</td>
<td>The X & Y-AXES of a PART each has its own coordinate-system.
The image to the left shows the Coordinate-System of the Y-AXIS
The local +x-axis of the Y-AXIS of a PART is along the Y-AXIS!
Thus, the local +y-axis of the Y-AXIS of a PART is at +90º from Y-AXIS
Thus a dimension that is a perpendicular-distance from the Y-AXIS to a Point in the green area are negative ...
... and this is in the +X AXIS direction of the PART!</td>
</tr>
</table>

1.11.4.2.6 Why do I need to add Geometry to the Base-Part?

Why do I need to add geometry to the Base-Part?

1. Why is there not a CAD-LINE in the BASE-PART?

We do not want you to automatically add Joints to the Origin of the BASE-PART.

However, you are welcome to do this.

2. Why?

For the **BASE-PART**, we have found it to be useful to add **POINTS** and **LINES** that are not at the Origin or along the X-axis or Y–axis. We then add dimensions to give the position of the Points and Lines relative to the origin of the **BASE-PART**.

3. Why?

You can edit the dimensions later, and also add them to a Design-Set[185].

4. Why edit **POINTS** and **LINES** in the **BASE-PARTS**?

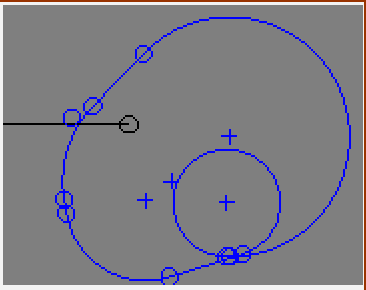
You have the design flexibility to:

1. Move the Joints, if the **POINTS** or **LINES** are used for Joints
2. Move the **PLANE** and **MECHANISM-EDITOR** in the Machine Frame, if a **LINE** is used for a new **PLANE** and then **MECHANISM-EDITOR**.

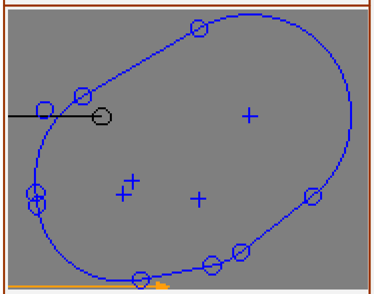
1.11.4.2.7 My sketch has disappeared. What do I do?

Trouble-shooting the sketch-editor.

Where is my Sketch?



Incorrect Arc



Correct Arc - After Undo

If your sketch changes in an unexpected way, and you cannot *Drag* to recover it:

Use **Edit menu > Undo**[43], **or CTRL + Z**

To return the sketch to state it was in before:

Use **Edit menu > Undo**[43] **or CTRL + Z**

Sometimes the Sketch 'Collapses' to a small dot.

Use **Edit menu > Undo**[43] **or CTRL + Z**

Sometimes the Sketch 'Explodes to the size of a planet.

Use **Edit menu > Undo**[43] **or CTRL + Z**

Why is my sketch BLACK when it should be BLUE?

Note:

BLUE means the geometry is **Not Solved**.

BLACK means the geometry is **Fully Solved**.

Sometimes, a sketch-element is **BLACK** but you know it should be **BLUE**. You should be able to add a dimension or constraint to it, but you cannot.

Or, a sketch-element is **BLUE** when it should be **BLACK**.

The best solution is to do one of these actions:

- Delete the sketch-element to add it again.
- Click the sketch-element so it shows in the Selection-Window
- Right-click a constraint, or the sketch-element, and select 'Delete Element'.

OR

- Use **Edit menu > Undo**[43] **or CTRL + Z**

Then add a different constraint.

You can add the same constraint again, or use a different constraint, to see if the sketch-element turns **BLACK**.

In our defense, the Part-Editor is continuously improved.

Why is my sketch Blue, and I cannot add any more constraints?'

Sometimes, a sketch-element is **BLUE** but you cannot add more constraints.

You might be able to drag a sketch-element that is blue, but when you come to add a new constraint, **MechDesigner** does not accept it.

In this case, a new constraint may remove two degrees-of-freedom, but the sketch has only one remaining degree-of-freedom.

Example:

Add Coincident Constraint between the center-Point of a Circle and a Point at the end of a Line.

If: the CENTER-POINT of the CIRCLE has a COINCIDENT CONSTRAINT with the LINE

Then: it is not possible to also add a COINCIDENT-CONSTRAINT with the START-POINT or END-POINT of the LINE

1. SHIFT+CLICK the LINE or the CENTER-POINT of the CIRCLE so the COINCIDENT-CONSTRAINT shows in the SELECTION-WINDOW
2. Delete the COINCIDENT CONSTRAINT using the SELECTION-WINDOW

1.11.4.2.8 What are the color codes of the Part-Editor?

Color-codes of Elements in the Part-Editor

You can specify the Color for the two states of Geometry.

- Geometry Under Defined - we usually describe this color as **Blue** in this help
- Geometry Fully Defined. - we usually define this color as **Black** in this help

... in **Application-Settings > Graphics tab > Display Colors**

Other Colors

- Reference Geometry is **Orange**

These dimensions are **Gray**:

- Dimension that have been added to a Design-Set
- The Dimension for the Line between that is the distance between the two gears of a Gear-Pair
- The Dimension for a Radius of an Arc that is radius of a Pulley
- The Dimension of the Controlling-Dimension that controls the length of a Motion-Path

- . -

- 2 -

- 3 -

- 4 -

- A -

- D -

- E -

- F -

- G -

- H -

- I -

- J -

- K -

- L -

- M -

- N -

- O -

- P -

- Q -

- R -

- S -

- T -

- U -

- V -

- W -

- X -