DMU Kinematics Simulator



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Getting Started

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Detecting Clashes Automatically in V5 Checking Joint Limits Measures Measuring Properties Measuring Distances between Geometrical Entities Digital Mockup Review Reviewing Simulations Recording Positions Replaying Simulations Resetting a V5 Mechanism Sequencing Mechanisms with Laws **Managing Kinematics Data in Sub-products** Visualizing and Simulating Mechanisms in Sub-products More about Importing Mechanisms Dressup Importing a Mechanism and its Dressup Importing a Mechanism and its Dressup from a Skeleton Structure Managing the Mechanism Dressup **Defining a Swept Volume Defining a Swept Volume** Defining a Swept Volume from a Mechanism **Defining a Swept Volume from a Moving Reference Filtering Swept volume Positions** More About Swept Volume Working with ENOVIA LCA

Workbench Description

Menu Bar DMU Kinematics Toolbar Simulation Toolbar DMU Joint Toolbar DMU Generic Animation Toolbar DMU Kinematic Update Automatic Clash Detection Toolbar DMU Space Analysis Toolbar Specification Tree

Glossary

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Overview

Welcome to the *DMU Kinematics User's Guide*. This guide is intended for users who need to become quickly familiar with the DMU Kinematics Version 5 product.

This overview provides the following information:

- DMU Kinematics Simulator in a Nutshell
- Before Reading this Guide
- Getting the Most out of This Guide
- Accessing Sample Documents
- Conventions Used in this Guide

DMU Kinematics Simulator in a Nutshell

DMU Kinematics Simulator is an independent CAD product dedicated to simulating assembly motions. It addresses the design review environment of digital mock-ups (DMU) and can handle a wide range of products from consumer goods to very large automotive or aerospace projects as well as plants, ships and heavy machinery.

DMU Kinematics Simulator is a dedicated DMU Navigator workbench and is available on both UNIX and Windows environments.

Before Reading this Guide

Before reading this guide, you should be familiar with basic Version 5 concepts such as document windows, standard and view toolbars. Therefore, we recommend that you read the *Infrastructure User's Guide* that describes generic capabilities common to all Version 5 products. It also describes the general layout of V5 and the interoperability between workbenches. You may also read *DMU Navigator User's Guide*

You may also like to read the following complementary product guides, for which the appropriate license is required:

- Knowledge Advisor User's Guide
- DMU Fitting Simulator User's Guide
- DMU Space Analysis User's Guide

Getting the Most out of this Guide

To get the most out of this guide, we suggest you start reading and performing the step-by-step tutorial Getting Started. This tutorial will show you how to create mechanisms and joints from scratch.

Once you have finished, you should move on to the next section: <u>Basic Tasks</u> dealing with the main capabilities of DMU Kinematics product (mechanism and joints design, Kinematics simulations...) The next section <u>Advanced Tasks</u> focuses on analysis and review. You might be interested in reading the Interoperability section which can be accessed directly from the table of contents using the following

icon III may also be a good idea to take a look at the section describing the menus and toolbars: Workbench Description

Accessing Sample Documents

To perform the scenarios, you will be using sample documents contained in the online $\sc x2\sc and sc and$

When samples belong to capabilities common to different products, those samples will be found in the online $cysa_X2$ samples folder.

*Where X can be C for CATIA or E for ENOVIA.

For more information about this, refer to Accessing Sample Documents in the *Infrastructure User's Guide*.

Conventions Used in this Guide

To learn more about the conventions used in this guide, refer to the *Conventions* section.

Conventions

Certain conventions are used in CATIA, ENOVIA & DELMIA documentation to help you recognize and understand important concepts and specifications.

Graphic Conventions

The three categories of graphic conventions used are as follows:

- Graphic conventions structuring the tasks
- Graphic conventions indicating the configuration required
- Graphic conventions used in the table of contents

Graphic Conventions Structuring the Tasks

Graphic conventions structuring the tasks are denoted as follows:

This icon	Identifies		
_			
\odot	estimated time to accomplish a task		
()	a target of a task		
۲	the prerequisites		
	the start of the scenario		
()	a tip		
	a warning		
(i)	information		
(0))	basic concepts		
	methodology		
(i)	reference information		
(i)	information regarding settings, customization, etc.		
**	the end of a task		
	functionalities that are new or enhanced with this Release.		
	allows you to switch back the full-window viewing mode.		

Graphic Conventions Indicating the Configuration Required

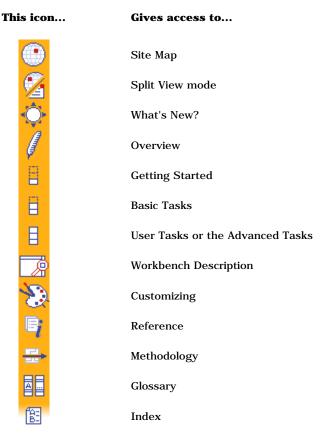
Graphic conventions indicating the configuration required are denoted as follows:

This icon... Indicates functions that are...

P1	specific to the P1 configuration
P2	specific to the P2 configuration
P3	specific to the P3 configuration

Graphic Conventions Used in the Table of Contents

Graphic conventions used in the table of contents are denoted as follows:



Text Conventions

The following text conventions are used:

- ◆ The titles of CATIA, ENOVIA and DELMIA documents *appear in this manner* throughout the text.
- File -> New identifies the commands to be used.
- Enhancements are identified by a blue-colored background on the text.

How to Use the Mouse

The use of the mouse differs according to the type of action you need to perform.

```
Use this
mouse button... Whenever you read...
```

DMU Kinematics Simulator

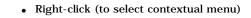
Version 5 Release 13

- Select (menus, commands, geometry in graphics area, ...)
- Click (icons, dialog box buttons, tabs, selection of a location in the document window, \ldots)
- Double-click
- Shift-click
- Ctrl-click
- Check (check boxes)
- Drag
- Drag and drop (icons onto objects, objects onto objects)



.....

- Drag
- Move



What's New?

This section identifies what new or improved capabilities have been documented in the Version 5 Release 13 of DMU Kinematics Simulator User's Guide.

Enhanced Functionalities

All Commands

Working with ENOVIA LCA

Optimal PLM Usability for DMU Kinematics Simulator to ensure that data created in CATIA can be correctly saved in ENOVIA V5

Sensors

Using sensors

In the Sensors dialog box, selecting a clash sensor activates automatically the clash detection (is set to on)

Mechanism Analysis

Analyzing a mechanism

It is now possible to save the information of the Mechanism Analysis dialog box (in .xls, txt and lotus 123 formats) $\$

Managing Kinematics Data in Sub-products

Importing a mechanism and its dressup from a skeleton structure

It is now possible to import the dressup associated to a mechanism from a skeleton structure, please also read More about importing mechanisms dressups and Importing a mechanism and its dressup

Getting Started

Before getting into the detailed instructions for using DMU Kinematics Simulator Version 5, the following tutorials aim at giving you a feel of what you can do with the product. It provides two stepby-step scenarios showing you how to use key functionalities. The main tasks described in this section are:

> Designing a V5 Mechanism Using V4 Kinematic Data

Designing a V5 Mechanism

Entering the Workbench Creating a Mechanism and Revolute Joints Creating Cylindrical Joints Defining a Command Defining a Fixed Part Simulating



These tasks should take about 20 minutes to complete.

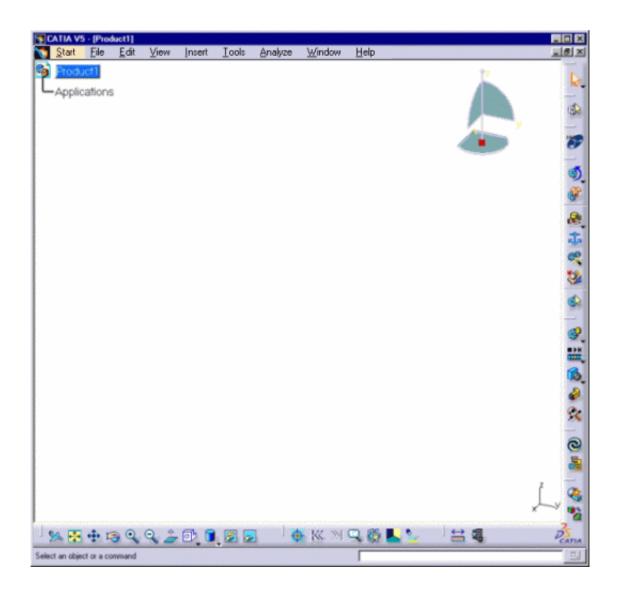
Entering the Workbench

Before starting this scenario, you should be familiar with the basic commands common to all workbenches. These are described in the *DMU Navigator User's Guide*.

This first task will show you how to enter the DMU Kinematics Simulator workbench and select your models.

1. Select Digital Mockup -> DMU Kinematics from the Start menu.

The DMU Kinematics workbench is loaded and an empty document opens:



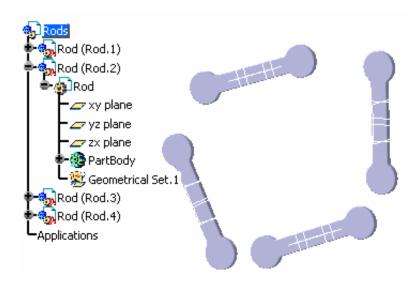
- **2.** Select **File** -> **Open** from the menu bar.
- 3. Select the rods.CATProduct document from the samples folder.

File Selection				? ×
Look jn:	🔄 samples	•	E	
🛛 🚮 PointSurfa	ace_with_Joint.CATProduct		4	rods+1joint.CAT
🛛 🌆 PointSurfa	ace_without_Joint.CATProduct		4	l rods+2joints.CA
🛛 🚳 Ring.CATI	Part		1	🕈 rods+3joints.CA
rod.CATP	art		1	ods+4joints.CA
rods.CATF			_	iods+4joints+crr
rods_with_	_joints.CATProduct		1	l rods+4joints+crr
•				F
File <u>n</u> ame:	rods.CATProduct			<u>O</u> pen
Files of type:	All Files (*.*)		-	Cancel
	🗖 Open as read-only			
	Show Preview			

4. Click Open to open the selected file.

The specification tree is displayed showing all the selected products.

5. Select the products in the tree, then select Edit->Representations->Design Mode. Then expand the tree to show all the design components of the products.



Use the Fit All In icon to position the model geometry on the screen.

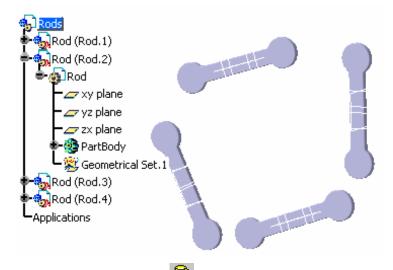


Creating a Mechanism and Revolute Joints

This task will show you how to create a mechanism and revolute joints.

Open the rods.CATProduct document.

Select the product in the specification tree, then select Edit->Representations->Design Mode.
 You can now expand the tree to show all the design components of the products.



 Click the Revolute Joint icon from the Kinematic Joints toolbar. The Joint creation : Revolute dialog box is displayed:

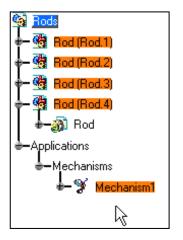
Joint Creation : Re	evolute	? ×
Mechanism :		New Mechanism
Joint name :		
	Curren	t selection :
Line 1 :	Line 2 :	
Plane 1 :	Plane 2 :	🔍 🕙 Null Offset 🔿 Offset = 🛛 🕅 🧮
Plane 3 : 📘	Plane 4 :	O Centered
	Angle driven	
		🥥 OK 🧾 🍛 Cancel

 Click New Mechanism. The Mechanism Creation dialog box is displayed: you can now enter a name of your choice. Click Ok when done. In our example, keep the default name Mechanism. 1

Mechanism Creation	? ×
Mechanism name : Mechanism	.1
💽 ок	Cancel

Joint Creation : Revolu	ite		? ×
Mechanism : Mechanism	1.1	•	New Mechanism
Joint name : Revolute.1			
Line 1 : F Plane 1 : F	Current se Line 2 : Plane 2 :	election :	○ Offset = 🛛 🕅 🚍
Plane 3 : -	Plane 4 : 🔤	Centered	
🗌 Ang	gle driven		
		0	OK Scancel

The Mechanism is identified in the specification tree.

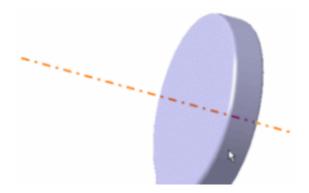


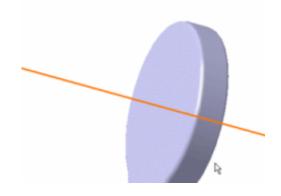
Now you need to select two lines and two planes

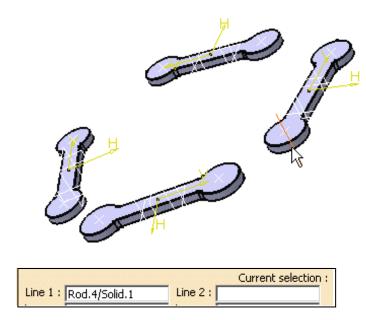
4. Select Line 1 in the geometry area. In our example select a cylinder as shown below.

The dialog box is automatically updated with your selection.

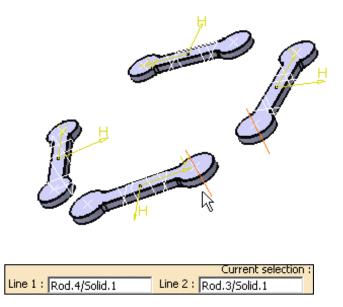
i Zoom in if necessary using the **View**->**Zoom In Out** command and drag (left mouse button) to zoom in progressively.





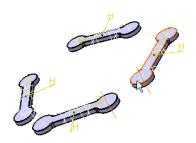


Select Line 2 in the geometry area. Select a second cylinder.
 The dialog box current selection field is automatically updated.

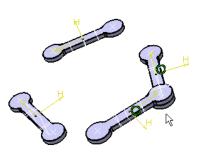


6. Select the planes as shown below.

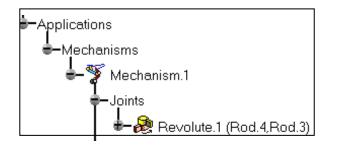
The Current selection field is automatically updated.



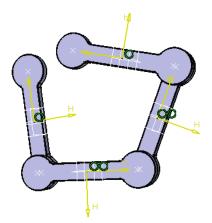




 Click Ok to end the Revolute Joint creation. The Revolute Joint is created. The specification tree is updated.



*P*roceed in the same manner to create Revolute. 2, Revolute. 3 This is what you obtain:



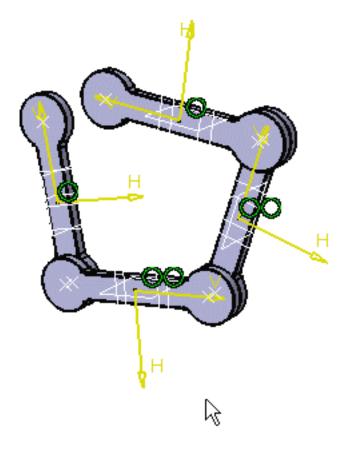
You can also create a new mechanism selecting Insert-> New Mechanism... from the Menu bar. The new mechanism is created and identified in the specification tree.

Insert	<u>T</u> ools	<u>A</u> nalyze	<u>W</u> indow
Q	[bject		
<u>۲</u> ۲	lew Mech	anism	
i N	l <u>e</u> w Joint		× •
ĴμE	ixed Part		

Creating Cylindrical Joints

This task will show you how to create cylindrical joints.

You created a mechanism and 3 revolute joints as shown in the previous task.

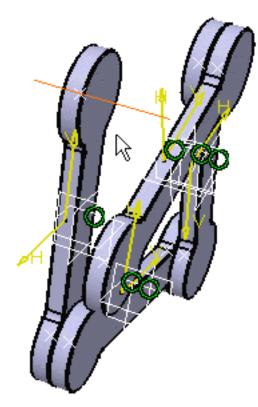




Click the Cylindrical Joint icon
 The Joint Creation: Cylindrical dialog box appears:

Joint Creation : Cylindrical		? ×
Mechanism : Mechanism.1	New Mechanism	
Joint name : Cylindrical.4		_
Cur	rent selection :	
Line 1 :	Line 2 :	
Angle driven	Length Driven	
	🕒 OK 🧕 🥯 Car	ncel

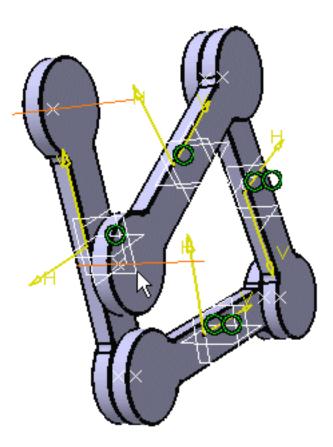
2. Select Line 1 in the geometry area. In our example select a cylinder as shown below:



The dialog box is automatically updated with your selection.

			Current selection :
Line 1 :	Rod.4/Solid.1	Line 2 :	

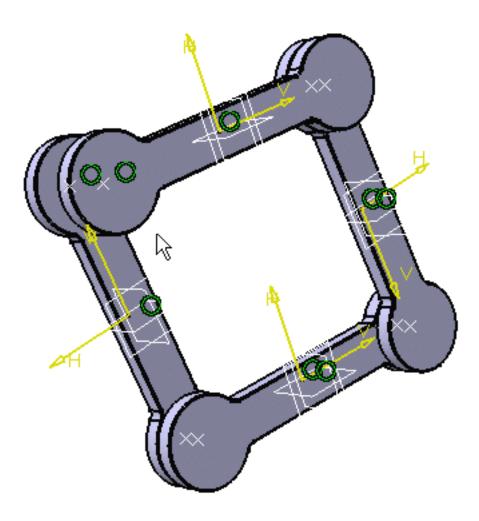
3. Select Line 2 in the geometry area. In our example select a cylinder as shown below:



The dialog box is automatically updated with your selection.

	Current selection :
Line 1 : Rod.4/Solid.1	Line 2 : Rod.3/Solid.1

4. Click OK to end the cylindrical joint creation.



i

The Cylindrical joint is created as well as the constraints. The specification tree is updated.

- You can define commands while creating cylindrical joints:
- Angle driven



Length driven

all you need to do is to check the required option.

Remember that you can at any time modify the command. For this, double-click the joint in the specification tree and edit the settings in the displayed dialog box. For more details, please refer to Editing joints.



Defining a Command

i) You can either define a command after joint creation or during joint creation.

In our example, you will define a command after joint creation.

1. Double-click Revolute. 3 in the specification tree.

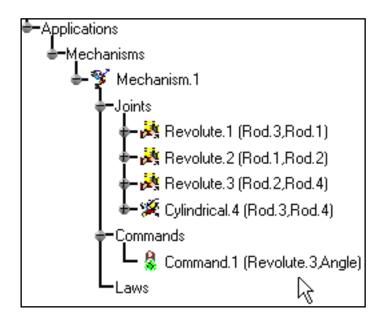
Applications
Mechanisms
🖨 🌱 Mechanism.1
1
≑− Joints
🛊 🖓 Revolute.2 (Rod.1,Rod.2)
💠 🚧 Revolute.3 (Rod.2,Rod 4)
👉 🎉 Cylindrical. 4 (Rod. 3, Rod. 4)
 Commands
Laws

The Joint Edition dialog box is displayed.

Joint Edition : Revolute.3	? ×
Joint name : Revolute.3	
	geometry :
Line 1 : Axis	Line 2 : Axis
Plane 1 : Face	Plane 2 : Face
Angle driven	
Joint Lime	
	🔽 Upper Limit : 360deg 📑
	OK Cancel

 Select the Angle Driven checkbox. The Angle driven option lets you assign the revolute joint an angle type command. **3.** Click **Ok** to confirm your operation.

The command is identified in the specification tree.

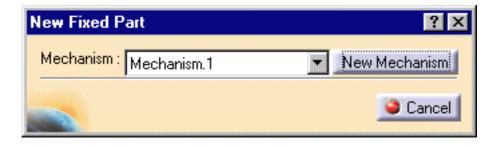




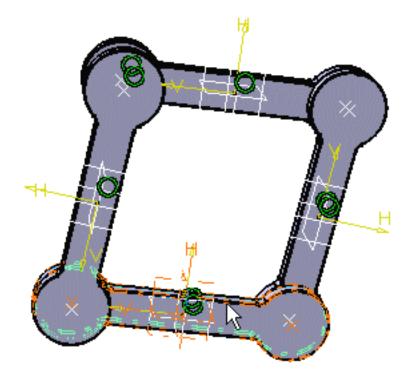
Defining a Fixed Part

This task will show you how to define a Fixed part.

1. Click the Fixed Part icon from the DMU Kinematics toolbar or select **Insert**->**Fixed Part...** from the menu bar. The New Fixed Part dialog box is displayed.

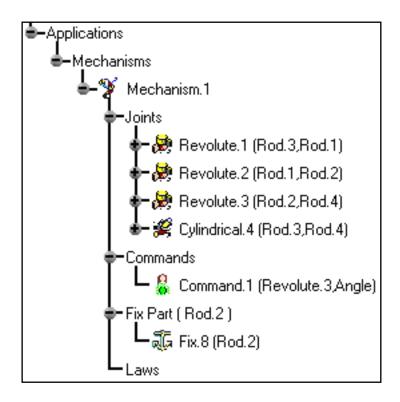


2. Select the Fixed Part either in the geometry area or in the specification tree.



3. The fixed Part is automatically defined.

The fixed part is identified in the specification tree.



👔 At any time you can use the undo command 🎽 to modify your selection.

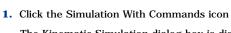




Simulating a V5 Mechanism

💮 This task will show you how to simulate the V5 mechanism you created.

You designed a V5 mechanism as described in the previous steps.





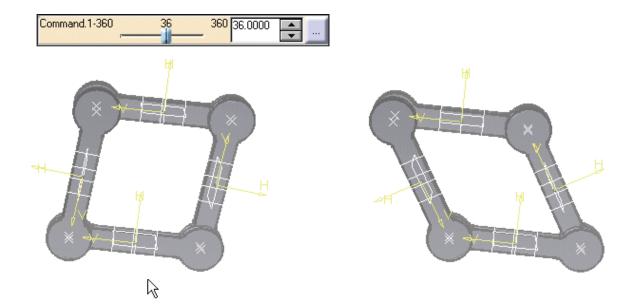
The Kinematic Simulation dialog box is displayed:

Kinematic Simulation - Mechanism.1	? ×
Mechanism : Mechanism.1	•
Command.1 -360 360 0.0000	.
Activate Sensors	······
Reset Analysis	More>>
	Close

The command of the kinematics mechanism is available

2. Manipulate the slider of the command.

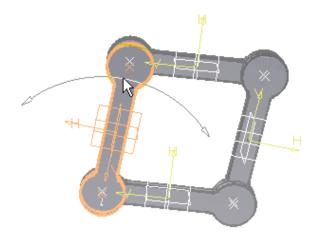
The kinematics mechanism moves accordingly.

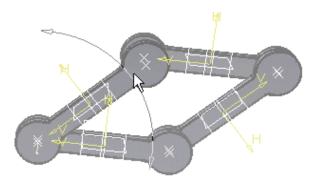


3. or Use the manipulator in the geometry area. For this:

Move the mouse over a joint. The driven joint highlights and the manipulator appears.

Drag the model with the left mouse button.





For more information please refer to Running Simulations and About Joints.

 $m{i}$) You can also enter a value for the command to achieve the same result.

*i*Note that if you click the More >> button, the Kinematics Simulation dialog box expands. The **immediate** option is set by default. For more information about the **On request** option, please refer to Simulating on Request



Using V4 Kinematics Data

Entering the Workbench Browsing the Mechanism Simulating With Commands Simulating With Laws



These tasks should take about 20 minutes to complete.

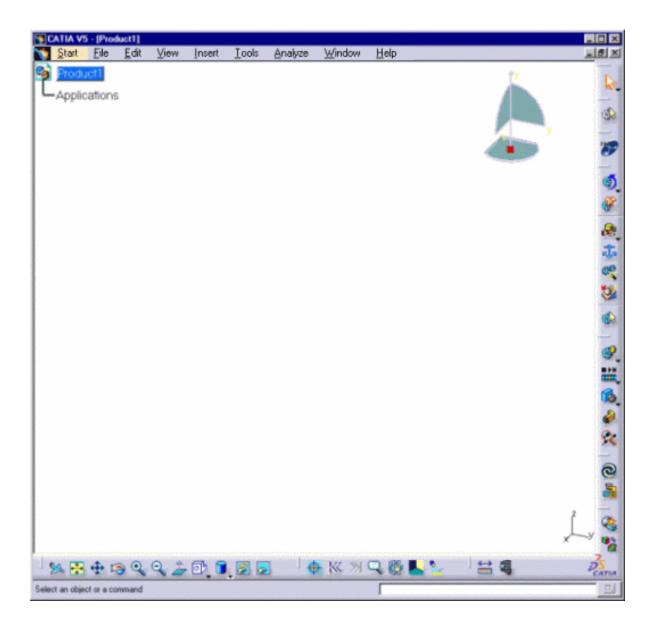
Entering The Workbench

Before starting this scenario, you should be familiar with the basic commands common to all workbenches. These are described in the DMU Navigator User's Guide.

This first task will show you how to enter the DMU Kinematics Simulator workbench and select your models.

1. Select Digital Mockup->DMU Kinematics from the Start menu.

The DMU Kinematics workbench is loaded and an empty document opens:

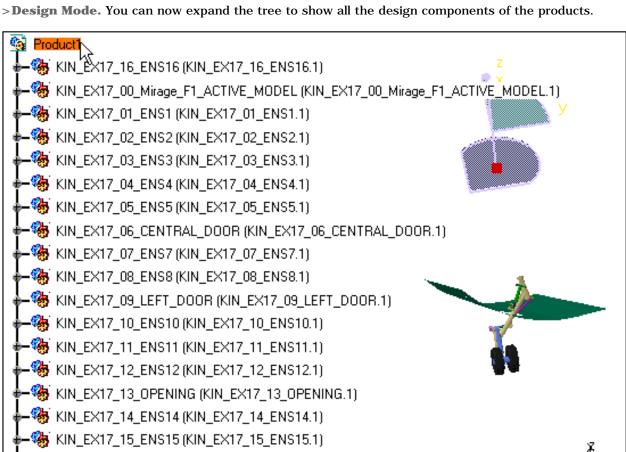


- 2. Select Insert -> Existing Component... from the menu bar.
- 3. Select the desired Kinematics model files by clicking the first one then shift-clicking the last one you want.
- 4. Click **Open** to open the selected files.

The specification tree is displayed showing all the selected products.

Insert an Exi	sting Component		? ×
Look jn:	🔄 InputData 📃		
FRT_LEF1	[_SPRINGREF.model	🔊 KIN	_EX17_03_ENS3.m
🛛 🔊 FRT_LEF1	[_WHEELREF.model	📓 KIN	_EX17_04_ENS4.m
KIN_EX17	_00_FIXE.model	📓 KIN	_EX17_05_ENS5.m
🛛 📉 KIN_EX17	_00_Mirage_F1_ACTIVE_MODEL.model	📓 KIN	_EX17_06_CENTR4
🛛 📉 KIN_EX17	_01_ENS1.model	📓 KIN	_EX17_07_ENS7.m
KIN_EX17	_02_ENS2.model	📓 KIN	_EX17_08_ENS8.m
•			F
File <u>n</u> ame:	"KIN_EX17_02_ENS2.model" "KIN_EX1	7_00_	<u>O</u> pen
Files of <u>type</u> :	model(*.model)	•	Cancel
	Open as read-only		

5. Select the products in the tree containing kinematics objects, then select **Edit** -> **Representations**-



Remember that DMU Kinematics Simulator exploits CATIA Version 4 multi-model sessions that have been prepared with one or more kinematic mechanisms.

) Use the Fit All In icon 🔛 to position the model geometry on the screen.

KIN_EX17_00_FIXE (KIN_EX17_00_FIXE.1)

Applications

÷

Browsing the Properties of the Kinematics Mechanism

This task will show you how to browse the properties of the selected kinematics mechanism.

Insert the KIN_EX17* .model files from the samples folder.

If you work with the Cache System, please make sure you are in Design mode (select **Edit**->**Representations**->**Design Mode.**). for more detailed information, please refer to the *DMU Navigator user's Guide* - *Task: Viewing the Cache Content.*

- **1.** Select KIN_EX17_00_F1_ACTIVE and expand the tree.
- Right-click the kinematics mechanism in the specification tree or select the Edit->Properties... from the menu bar.
- 3. In the first case, select **Properties** from the contextual menu displayed.

operties		Ŷ
Current selection : LANDING GEAR		7
Kinematic		
- General Properties		
Mechanism name :	LANDING GEAR	
Mechanism can be simulated:	Yes	
Number of joints :	21	
Number of commands :	5	
Degrees of freedom without command	5	
Degrees of freedom with commands :	0	
Fixed part :	REF	
		More

- 4. Click OK.
- displayed as shown.

5. Click the Mechanism Analysis icon . The General Properties of the kinematics mechanism are

6. You can select another mechanism using the Mechanism name combo.

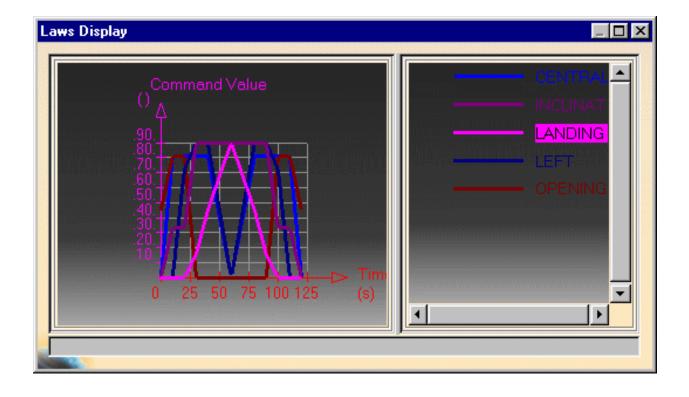
echanism Analy -General Propertie							?
Aechanism name :	.5		LAN	DING GEAR			
				DING GEAR			
Aechanism can be	simulated :		Yes				
Number of joints :			21				
lumber of commar	ids :		5				
egrees of freedor	n without cor	mmand :	5				
egrees of freedor	n with comma	ands :	0				
ixed part :			REF				
ixed pare i			IKEP				
🔾 Show joints 🔎	Hide joints				Save		Laws
Joint Command	Туре	Part 1		Geometry 1	Part 2	Geometry 2	Part 3 🔺
	Revolute	ENS10		DRT35	FIXE	DRT35	
	Prismatic	ENS8		DRT33	ENS12	DRT33	
LANDING	Revolute	ENS1		DRT27	FIXE	DRT27	
	Revolute	ENS11		DRT37	FIXE	DRT37	
	Revolute	ENS7		DRT36	ENS11	DRT36	
	Spherical	ENS3		PT 30	ENS7	PT 30	
	Revolute	ENS1		DRT12	ENS3	DRT12	
	Spherical Revolute	ENS3 ENS4		PT 27 DRT19	ENS5 ENS5	PT 27 DRT19	
	Revolute	EN54 EN54		DRT13	ENS2	DRT13	
	1.CYOIGC5	END I		DATIO		DICTIO	112
lechanism dressup	information:						
Part 1	Part 2	Part	3				
							Close

7. If you check the Show joints, this is what you obtain:

ĺ



If you click the Laws... button, you access to a graphic representation of the laws associated to each command. It is represented by a colored curve. When you pass the cursor along the curve, information about the law is displayed in the status bar.



For more detailed information please refer to Analyzing A Mechanism.

For more detailed information about laws, please refer to Simulating With Laws.



Simulating With Commands

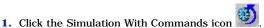
This task will show you how to run a kinematics simulation with commands.

Insert the KIN_EX17* .model files from the samples folder.

If you work with the Cache System, please make sure you are in Design mode (select Edit->Representations->Design Mode.). for more detailed information, please refer to the DMU Navigator user's Guide - Task: Viewing the Cache Content.

 \vec{l} In our sample document, there is only one mechanism. If you work with a product containing more than one mechanism, it is strongly recommended to select the mechanism you need before starting the simulation with commands.

1



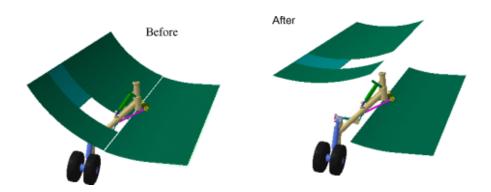
The Kinematic Simulation dialog box is displayed:

Kinematic S	imulation - LAN	DING GEA	R	? ×
Mechanism	LANDING GEAP	3		•
CENTRAL	-360	360	0.0000	
LEFT	-360	360	0.0000	.
OPENING	-350	370	10.0000	
INCLINAT	-360	360	0.0000	
Check J	oint Limits			
Reset		Analysis		More>>
				Close

The commands of the kinematics mechanism are available as shown

2. Manipulate the slider of a command. For instance select the LEFT.

The corresponding part of the kinematics mechanism moves accordingly



i Note that if you click the More >>> button, the Kinematic Simulation dialog box expands. The **immediate** option is set by default. For more information about the **On request** option, please refer to Simulating on Request.

Kinematic S	Simulation	- LANDIN	IG GEAR	? ×
Mechanism		G GEAR		-
CENTRAL	-360 ,	_1_	360 0.0000	
LEFT	-360	_1_	360 0.0000	
OPENING	-350	_ <u>i</u> _	370 10.0000	
INCLINAT	-360		360 0.0000	
4				
🗌 Check J	oint Limits			
Reset		Ana	alysis	(KKLess)
- Simulation	1			
🥥 Immedia	ite O On i	request		
	KI			
	Nu	mber of step	os: <mark>40</mark> 🔽	
				Close

(i) You can use the slider, enter a value or manipulate the geometry directly to achieve the same result.

3. Manipulate the other commands in the same way.

Running a Simulation With Laws



This task will show you how to run a kinematics simulation with laws that are already defined on the mechanism.



Insert the KIN_EX17* .model files from the samples folder. The Kinematics Simulation dialog box is displayed as described in the previous task.



1. Click the Simulation With Laws icon **W**. The Kinematic Simulation dialog box appears

Kinematic Simulation -	LANDING GI	AR ? 🛛
Mechanism : LANDING	GEAR	•
Start 0	- 120 0.000	.
Number of steps:	M	Analysis
Check Joint Limits		
		Close

2. Set the **Number of steps** to 10, then click the Play button.



You can use the other buttons to run the simulation again in different modes (backward, step by step, and so on).



Notice that you cannot record simulations within the Simulation With Laws functionality. If you need to record such a simulation or several simulations, please refer to Recording Positions.



Basic Tasks

The table below lists the tasks you will find in this section.

Setting Up Your Session Designing a V5 mechanism Designing Joints Fixed Parts and Commands Converting Constraints into Joints Using the Update Command Moving Constrained Components Using the Compass Running Simulations

Setting Up Your DMU Kinematics Simulator Session

DMU Kinematics Simulator provides easy methods to simulate mechanisms previously defined using the CATIA Version 4 KINEMAT and KINEMUSE functions.

You may find it useful to refer to your CATIA Version 4 Kinematics User's Reference Manual.

Prepare CATIA Version 4: transfer the solid and surface geometry that represents the moving parts into separate models (1 part per model). The model containing the kinematics mechanism should only be a stick model (that is, wireframe plus the definition of the mechanism). Use KINEMUSE function's DRESSUP item to define set/model relationships. Save all models and, if needed, the session.

Convert V4 Kinematic Data into DMU Kinematic V5: open the model containing the kinematics mechanism. In the specification tree where the Version 4 kinematics model is displayed, select the mechanism you wish to copy into the Kinematics Simulator Version 5. Put the data you have selected in the clipboard, then select Application in the specification tree and paste.

Open Version 5: enter the DMU Kinematics workbench, then select **Insert**->**Existing Component** in order to select the desired models.

Preparing a Multi-Model Session in CATIA Version 4



This task shows how to prepare a CATIA Version 4 kinematics mechanism for use in DMU Kinematics Simulator Version 5.



 Transfer the solid and surface geometry that represents the moving parts into separate models (1 part per model).



The model containing the kinematics mechanism should only be a stick model (that is, wireframe plus the definition of the mechanism).

- **2.** Use KINEMUSE function's DRESSUP item to define set/model relationships.
- **3.** Save all models and, if needed, the session.



Converting Version 4 Kinematics Data into Kinematics Version 5 Data

This task shows how to convert CATIA Version 4 kinematics data into DMU Kinematics Simulator Version 5.Data

Insert the KIN_EX17* .model files from samples folder.

If you work with the Cache System, please make sure you are in Design mode (select **Edit**->**Representations**->**Design Mode.**). for more detailed information, please refer to the *DMU Navigator user's Guide - Task: Viewing the Cache Content.*

The following task shows how kinematics data is pasted from an existing Version 4 model to an existing Version 5 document alongside V5 data. You can of course also insert the V4 data into a new Version 5 document.

1. Open the model containing the kinematics mechanism.

Open the Kinematics Simulator workbench if necessary.

(i)

The model containing the kinematics mechanism should only be a stick model (that is, wireframe plus the definition of the mechanism).

2. In the specification tree or in the geometry area where the Version 4 kinematics model

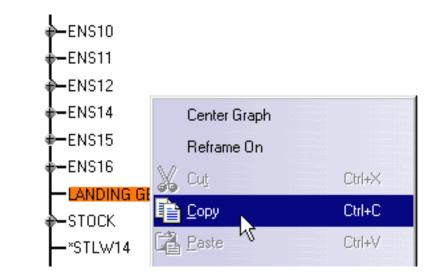
is displayed, select the mechanism you wish to copy into the Kinematics Simulator

Version 5. In our example, select KIN_EX_00_F1_ACTIVE and LANDING GEAR.

Product1
KIN_EX17_16_ENS16 (KIN_EX17_16_ENS16.1)
+-% KIN_EX17_00_FIXE (KIN_EX17_00_FIXE.1)
- 🗞 KIN_EX17_01_ENS1 (KIN_EX17_01_ENS1.1)
+-% KIN_EX17_02_ENS2 (KIN_EX17_02_ENS2.1)
+-% KIN_EX17_03_ENS3 (KIN_EX17_03_ENS3.1)
♣–% KIN_E×17_04_ENS4 (KIN_E×17_04_ENS4.1)
+-% KIN_EX17_05_ENS5 (KIN_EX17_05_ENS5.1)
← 🍓 KIN_EX17_06_CENTRAL_DOOR (KIN_EX17_06_CENTRAL_DOOR.1)
+-% KIN_EX17_07_ENS7 (KIN_EX17_07_ENS7.1)
- 🗞 KIN_EX17_08_ENS8 (KIN_EX17_08_ENS8.1)
- KIN_EX17_09_LEFT_DOOR (KIN_EX17_09_LEFT_DOOR.1)
+-% KIN_EX17_10_ENS10(KIN_EX17_10_ENS10.1)
+-% KIN_EX17_11_ENS11 (KIN_EX17_11_ENS11.1)
- 🗞 KIN_EX17_12_ENS12 (KIN_EX17_12_ENS12.1)
- 😽 KIN_EX17_13_OPENING (KIN_EX17_13_OPENING.1)
+-% KIN_EX17_14_ENS14 (KIN_EX17_14_ENS14.1)
- 🗞 KIN_EX17_15_ENS15 (KIN_EX17_15_ENS15.1)
- 🗞 KIN_EX17_00_F1_ACTIVE (KIN_EX17_00_F1_ACTIVE.1)
L_Applications K

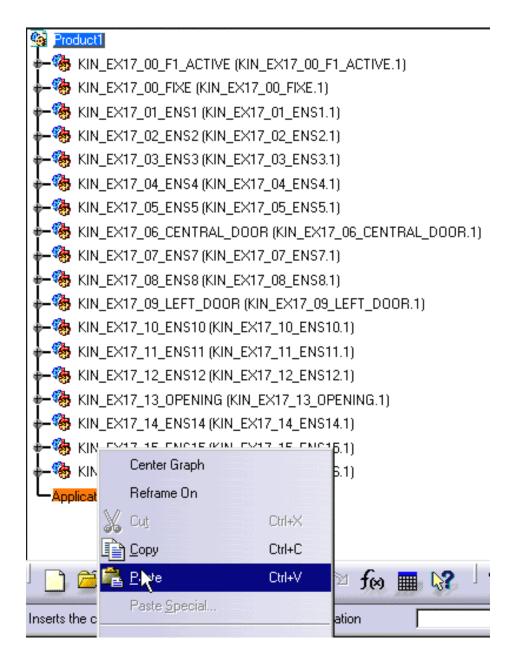
Product1
— *AXS9
-*TRA2
*AXS120
∲—ENS1
ENS2
∲ —ENS3
₽ —ENS4
₽ —ENS5
₽-ENS7
∲ —ENS8
∲—FIXE
₽-ENS10
₽-ENS12
∲—ENS14
€ ENS15
∲−ENS16
- LANDING GEAB
р−стоск Цд

- **3.** You can also use the drag &drop capability.
- Put the data you have selected in the clipboard. To do this, either click the Copy icon, select the Edit->Copy command or select the Copy command in the contextual menu.



- **5.** Select Application in the specification tree.
- Now either click the Paste icon, select the Edit->Paste command or select the Paste command in the contextual menu.

This operation recovers the data previously put in the clipboard

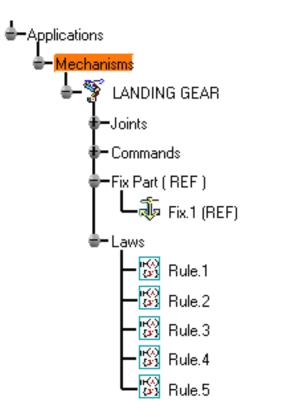


i You may want to click the Fit All In icon to fit all data in the window.

The dressup is maintained when you perform a copy/paste within the same document.

Notice that the toolbars change depending on whether a Version 4 model or a DMU Kinematic Version 5 document is selected.

The result should look something like this: V4 laws are converted in a V5 mechanism.



Kinematics Simulator fully supports V4 mechanisms (2D/3D) conversion into 3D mechanisms version 5.

What About the Elements You Convert?

To make sure the elements you need to handle in your session are those you expected, here is a list presenting the CATIA V4 Kinematics data supported when converted into a Kinematics Version 5 document:

V4 Data Type	V5 Data Type
Mechanism Structure	Result
2D / 3D	3D
mechanism	V5 mechanism
joint	V5 joint
(revolute, cylindrical, spherical, planar,	
prismatic,rigid,pt/crv,roll/crv,slid/crv,	
gear,rack,cable,screw)	
command	V5 command
fix	V5 fixed part
model	CATProduct
	sub-product + associated part
set	Geometry contained in the set
V4 dressup	V5 dressup
•	•
Outerante	
Outputs	a minuterationality (na according)
numerical (angles/distances)	equivalent functionality (no conversion) are not converted
speed, acceleration	
traces	equivalent functionality (no conversion)
clashes	equivalent functionality (no conversion)
distances	equivalent functionality (no conversion)
Laws	

numeric laws

geometric laws

are not converted

Knowledgeware rules



Opening Your DMU Kinematics Simulator Document in Version 5

This task recalls how to open a DMU Kinematics Simulator Version 5 document.

1. Enter the DMU Kinematics workbench, then select **Insert**->**Existing Component** in order to select the desired models.

Please refer to Entering the DMU Kinematics Workbench and Selecting Models

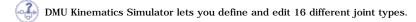
2. Activate the desired kinematics products in the specification tree.



Designing a V5 Mechanism

About Joints Creating a Mechanism and Revolute Joints Creating Joints Editing Joints Deleting Joints

About Joints



The tables below describe the joint types and their characteristics:

DMU Kinematics Simulator lets you define the following joints using axis systems:

- u joint
- prismatic
- revolute
- cylindrical
- spherical

V4 NAME	JOINT TYPE	DEGREES OF FREEDOM	COMMAND TYPE	DIRECT MANIPULATION
revolute	Revolute	훥 1 Rotation	Angle	YES / Left-mouse button
prismatic	Prismatic	1 Translation	Length	YES / Left-mouse button
		1 Rotation	Length + Angle	Length: Left-mouse button Angle: Left-Mouse
		1 Translation	AND/OR	button + Middle- Mouse button
actuator	Cylindrical	1	Angle or Length	YES / Left-mouse
				button
pt/pt	Spherical	🗳 3 Rotations	_	NO
planar	Planar	2 Translations 1 Rotation	_	NO
rigid	Rigid	d –	_	NO
roll/crv	Roll Curve	1 Rotation 1Translation	Length	NO
slid/crv	Slide Curve	2 Rotations 1 Translation	_	NO
pt/crv	Point Curve	3 Rotations 1 Translation	Length	NO
pt/surf	Point Surface	2 Translations 3 Rotations	_	NO
u jnt	U Joint	1 Rotation	_	NO
gear	Gear joint	1 Rotation	Angle1 or Angle2 (exclusive)	YES / Left-mouse button
rack	Rack Joint	1 Rotation or 1 Translation	Length1 or Angle2 (exclusive)	YES / Left-mouse button
cable	Cable Joint	1 Translation	Length1 or Length2	YES / Left-mouse button
screw	Screw Joint	1 Rotation or 1 Translation	Angle or Length (exclusive)	YES / Left-mouse button
cv joint	CV Joint	S -	_	NO

JOINT TYPE			SI	ELECTION	5			RATIO	* CONDITIONS
Revolute		sel.1	sel.2	sel.3	sel.4	sel.5	sel.6		
	8	Line	Line	Plane	Plane	Plane	Plane	-	(1)(2)(3)(6)
Prismatic	6	Line	Line	Plane	Plane	-	_	_	(1)(2)(4)
Cylindrical	×	Line	Line	_	_	_	_	_	(1)
Spherical	÷	Point	Point	_	_	_	_	_	(1)
Planar	R	Plane	Plane	_	_	_	_	_	(1)
Rigid	Z	Product	Product	_	_	_	_	_	(1)
Roll Curve	20	Curve	Curve	_	_	_	_	_	(1)
Slide Curve	-æ	Curve	Curve	_	_	_	_	_	(1)
Point Curve	₹	Curve	Point	_	_	_	_	_	(1)
Point Surface	+↓⊭	Surface	Point	_	_	_	_	_	(1)
Universal Joint	<u></u>	Line	Line	Line	_	_	_	_	(1)(5)
Gear Joint	6 2	Revolute	Revolute	_	_	_	_	Ratio	(7)
Screw Joint	۵	Line	Line	_	_	_	_	Ratio	(1)
Cable Joint	2	Prismatic	Prismatic					Ratio	(7)
Rack Joints	283	Prismatic	Revolute					Ratio	
CV Joints	\$	U joint	U joint						(7)(8)

* Conditions between selections:

- (1) selection 1 in another product than selection 2
- (2) selection 3 in either first selections' product, selection 4 in the other
- (3) line orthogonal to plane of same part
- (4) line lying in plane of same part
- (5) line 'selection 3' must be in either first selections' product and cross selection
- (6) selection 5 and 6 are optional ('centered case'); selection 5 in either first selections' product, selection 6 in the other
- (7) compound joints are based on basic joints selection or on-the-fly creation: a part is shared by the two joints
- (8) requires equal input and output angles

 \oplus

Creating a Mechanism and Revolute Joints

This task shows how to create a kinematics mechanism to use in DMU Kinematics Simulator Version 5.

Open the rods.CATProduct document.

Make sure you are in Design mode. If not, select the product in the tree, then select Edit->Representations->Design Mode.

If the menu item cannot be selected, right-click product1 in the specification tree.

- 2. Click the Revolute Joint icon from the DMU Simulation Toolbar. The Joint Creation: Revolute dialog box is displayed:
- 3. Click New Mechanism.

The Mechanism Creation dialog box is displayed: This dialog box lets you enter a meaningful name for the mechanism. Click Ok when done.

(i)

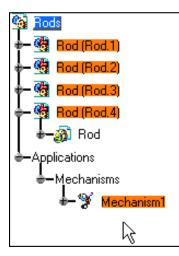
You can also create a new mechanism selecting Insert-> New Mechanism... from the Menu bar.

Mechanism Creation	? ×			
Mechanism name : Mechanism.1				
🥌 💽 ок 💌	Cancel			

In our example, keep the default name Mechanism.1.

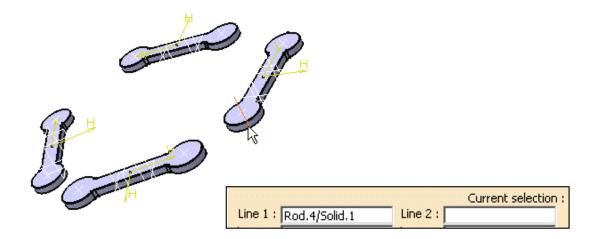
Joint Creation : Revolu	te		? ×
Mechanism : Mechanism.	1	•	New Mechanism
Joint name : Revolute.1			
	Current s	election :	
Line 1 :	Line 2 :		
Plane 1 :	Plane 2 :	🔍 🎯 Null Offset	🔿 Offset = 🛛 🕅 🚍
Plane 3 : 💽	Plane 4 : 💽	Centered	
🗌 Ang	le driven		
			OK Cancel

The Mechanism is identified in the specification tree.



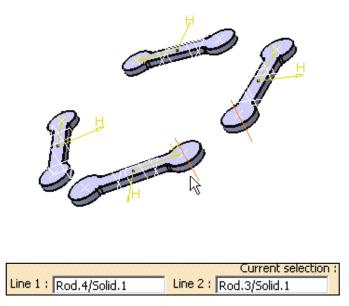
Now you need to select two lines and two planes

4. Select Line 1 in the geometry area. In our example select a cylinder as shown opposite: The dialog box is automatically updated with your selection.

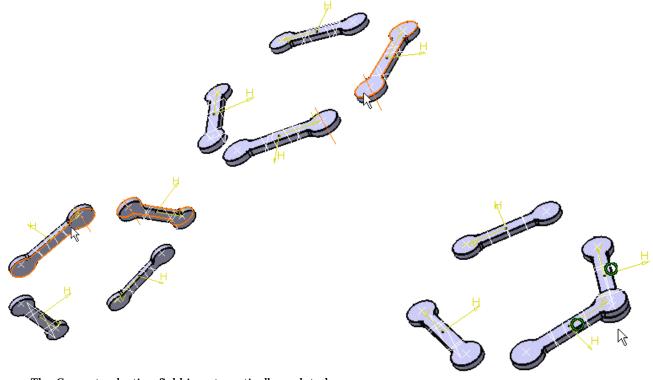


5. Select Line 2 in the geometry area. Select a second cylinder.

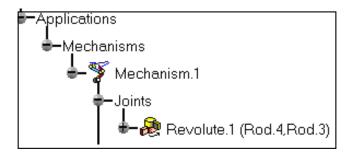
The dialog box current selection field is automatically updated.



6. Select the planes as shown below.



The Current selection field is automatically updated. The specification tree is updated.



- 7. Click Ok to end the Revolute Joint creation
- 8. Proceed in the same manner to create other joints

 $lap{ll}$ Do not forget to define a command and at least one fixed part within your mechanism.



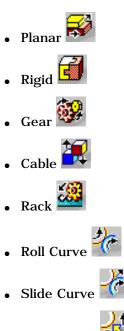
Creating Joints



This task shows how to create joints in a V5 mechanism. You can now create 16 joint types from the following list:

- Revolute 🊵
- Prismatic
- Cylindrical
- . <u>4</u>
- Spherical
- Universal 鄻

These joint can now be created using axis systems. See Creating Joints Using Axis Systems



- Point Curve 🖆
- Point Surface



• Screw 🎾



Open the rods+3joints.CATProduct document.



When you create joints, you can define the mechanism within the same dialog box. Remember though, that you create a mechanism independently from the joints by selecting **Insert**->**New Mechanism...** from the menu bar.

- **1.** Click the Revolute Joint icon from the DMU Kinematics toolbar (Revolute joint is the default joint type)
- **2.** Click the arrow within the icon and undock the Kinematics Joints toolbar.

ä D 🚦 ø ල

The DMU Kinematics toolbar is displayed:

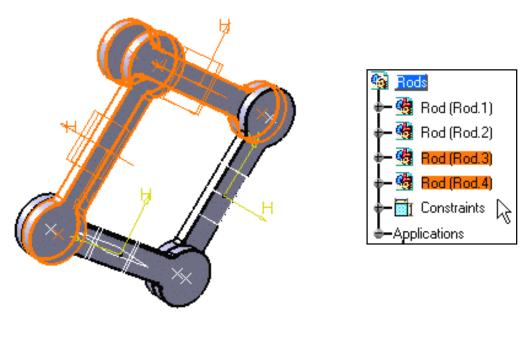


- **3.** Select the joint type of your choice.
- **4.** For instance click the rigid joint icon **Solution**. The Joint Creation: Rigid dialog box is displayed.

Joint Creation : Rigid	? ×
Mechanism : Mechanism.1	New Mechanism
Joint name : Rigid.4	
Curre	nt selection :
Part 1 :	Part 2 :
	OK Cancel

i The term Rigid corresponds to "Fully restricted" in the standard Kinematics terminology.

5. Select the parts either in the geometry area or in the specification tree.

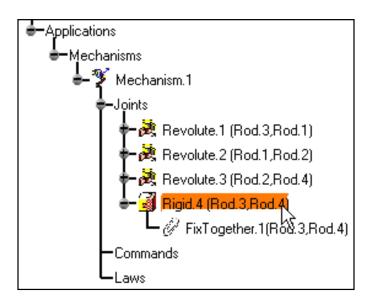


	Current selection :
Part 1 : Rod (Rod.3)	Part 2 : Rod (Rod.4)

ı

6. Click Ok to confirm your operation.

The Rigid Joint is identified in the specification tree.



For more information, please refer to About Joints and Creating Mechanisms and Joints.



Editing Joints

NDMU Kinematics Simulator lets you easily edit joints. Editing joints means you can modify:

- 🥖 🔹 its name
 - deactivate the command
 - modify its specifications (curve joints specifications: point curve, slide curve, roll curve joints)

either editing the existing geometry joint or replacing the existing geometry with a new one. The V5 joint mechanism is updated accordingly.

Please refer two the following scenarios: Replacing Curve Joint Specifications Editing Curve Joint Specifications

This task shows you how to do so.

Open the rods+4joints.CATProduct document.

1. Double-click the joint to be edited in the specification tree. For instance Revolute.1. The Joint Edition dialog box appears:

Joint Edition : R	Revolute.1 (Revolute)	? ×
Joint name : Rev	volute.1	_
	Joint geometry :	
Line 1 : Axis	Line 2 : Axis	
Plane 1 : Face	Plane 2 : Face	
🗌 Ang	gle driven	
Joint Limits		
Lower Limit :	unset 🔄 🗌 Upper Limit : 🛛 unset	÷
	🥥 OK 🥥 Car	ncel

2. In the name field enter a meaningful name: Revolute. 1-3 for instance.

Joint Edition : Revolute.1 (Revolu	te) ? 🗙
Joint name : Revolute.1-3	
Joir	nt geometry :
Line 1 : Axis	Line 2 : Axis
Plane 1 : Face	Plane 2 : Face
Joint Limits	
Cover Limit : -360deg	🛃 🔽 Upper Limit : 360deg 📑
	🕒 OK 🧕 🥯 Cancel

3. Click the Angle driven checkbox to apply an angle driven command

You can check the command positive orientation and invert it if necessary (either during joint or command edition). Note though, for Roll curve and Point curve joints you can only check the command orientation but not change it.

The command orientation is defined by a green arrow in the geometry area

- The parts which are not involved in the joint creation are displayed in low light (to easily locate the joint you are working on):
- $_{\odot}~$ Pass the cursor over the green arrow to launch a short animation
- o Click the arrow to reverse the command orientation if necessary

(1) The positive orientation of a command does not indicate an absolute movement of the parts involved (in the joint which is assigned the command) but the intrinsic movement of the second part with respect to the first part involved in the joint.

4. Set Joints limits if needed. Please refer to Setting Joint Limits and Checking Joint Limits for more detailed information.

3. Click the Angle driven checkbox to apply an angle driven command

You can check the command positive orientation and invert it if necessary (either during joint or command edition). Note though, for Roll curve and Point curve joints you can only check the command orientation but not change it.

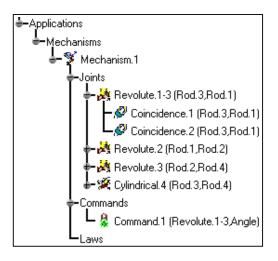
The command orientation is defined by a green arrow in the geometry area

- $_{\circ}~$ The parts which are not involved in the joint creation are displayed in low light (to easily locate the joint you are working on):
- $_{\circ}~$ Pass the cursor over the green arrow to launch a short animation
- $_{\circ}~$ Click the arrow to reverse the command orientation if necessary

The positive orientation of a command does not indicate an absolute movement of the parts involved (in the joint which is assigned the command) but the intrinsic movement of the second part with respect to the first part involved in the joint.

- 4. Set Joints limits if needed. Please refer to Setting Joint Limits and Checking Joint Limits for more detailed information.
- 5. Click OK to confirm your operation.

The Joint is updated and identified in the specification under its new name. The angle command assigned to Revolute.1-3 is also identified.



Note: you can edit the mechanism name. All you need to do is right-click the mechanism in the specification tree and select **Properties** from the contextual menu displayed.



Deleting Joints

Whenever you have to delete joints, you not necessarily have to delete the associated constraints. The Deletion capability lets you define what you really want to delete.

This task shows how to delete a joint and what this operation involves.

Open the Jack.CATProduct document.

 Right-click Prismatic.6 in the specification tree and select the delete item from the contextual menu displayed. The Delete dialog box appears:

 $m{i}$ Note: The Delete all children option is set by default

Delete	? ×
Selection Jack\CATCompoundCont\Mechanisms\Mec	chanism.1\joir
	F
Parents	
Delete exclusive parents	
Children	
🧧 Delete all children	More >>
С ОК	Cancel

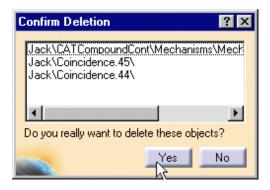
2. Click More>>

Additional options and the elements affected by the deletion are displayed. You can delete the constraints associated with the joint.

^{3.} Click the Delete/Undelete All button.

Delete		?	х
Selection Jack\CATCompoundCont\Mechanisms\Mechanism.1\joir	Advanced Children Management- Replace:	With:	_
	Replace	With	
Parents Delete exclusive parents			
Children Ckildren CK Less			
	I Elements Affected by Deletion		
	Elements	Status	
	Jack\Coincidence.45\ Jack\Coincidence.44\	Delete Delete	
	Delete/Undelete Delete/Und	delete All	
		OK SCance	əl

- 4. Click Ok
- 5. Click Yes in the Confirm Deletion dialog box displayed:



i

Keep in mind you can apply the ${\bf Undo}$ command if you inadvertently deleted a joint

Designing Joints

More about Joints and Constraints Designing Lower Pair Joints Creating Joints Using Axis Systems More About Resulting Constraints Designing Higher Pair Joints

Version 5 Release 13

More About Joints and Constraints

DMU Kinematics Simulator lets you define and edit 17 different joint types. .,?,

We can classify these joints under 4 different categories depending on the way they are defined. Of course one specific joint can belong to several categories:

- Joints using assembly constraints (i.e. a revolute joint is defined by two constraints (coincidence between two lines) and an offset between two planes
- Joints using topological or geometrical elements (i.e. a point curve joint is defined by a point and a curve)
- Compound joints using other joints (i.e. a gear joint is defined with two revolute joints)
- Joints defined with axis systems (i.e. universal joint)

DMU Kinematics Simulator lets you define the following joints using axis systems (

- u joint
- prismatic •
- revolute •
- cylindrical
- spherical •

The table below describes the joint types with respect to the categories they belong to (the way they are defined)

JOINT TYP	PE		WITH		WITHOUT ASSEMBLY CONSTRAINTS
		General case	sub-compound joints	axis systems	With Geometry
Revolute	2	Х		Χ	
Prismatic	6	X		Х	
Cylindrical	Q	Х			
Spherical	Ŷ	Х		Χ	
Planar	R	X			
Roll Curve	<u>*</u> 7				Х
Slide Curve	ł				Х
Point Curve	╶╏╴				Х
Point Surface	*				Х
U Joint	3			Χ	X
Gear joint	69	X	X		
Rack Joint	<u>.</u>	Х	Х		
Cable Joint	2	X	Х		
Screw Joint	٩	Х			
CV Joint	%		<i></i>		Х
			100		

Designing Lower Pair Joints

Creating Revolute Joints Creating Prismatic Joints Creating Cylindrical Joints Creating Planar Joints Creating Gear Joints Creating Rack Joints Creating Cable Joints Creating Screw Joints Creating Spherical Joints Creating Rigid Joints Creating Universal Joints Creating CV Joints

Creating Revolute Joints (Beginner's Mode)

This task shows how to create revolute joints in a V5 mechanism.

Open the Create_Revolute.CATProduct document.

Automatic switch to Design mode:

If you work with the cache system in visualization mode, you no longer need to use **Edit->Representations->Design Mode** beforehand as the switch to design mode is automatic (an eye appears as you point the product in the geometry or specification tree). All you need to do is click on the object.

You can now use axis systems to create revolute joints. Select the joint from axis icon kinematic toolbar. Please refer to Creating Joints Using Axis

- 1. Click the Revolute Joint icon from the Kinematic Joints Toolbar. The Joint Creation: Revolute dialog box is displayed
- 2. Click New Mechanism.

The Mechanism Creation dialog box is displayed: **Note:** this new dialog box lets you enter a meaningful name for the mechanism. Click Ok when done



Create_Revolute
Create_Revolute
Wheel (Wheel.1)
Greate_Hinge (Hinge.1)
Constraints
Applications
Chechanisms
Checha

- In our example, keep the default name Mechanism.1. The Mechanism is identified in the specification tree.
- 3. The Null Offset option is set by default (radio button). Keep it as it is.

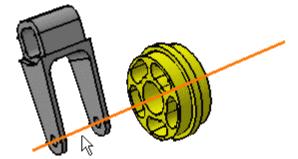
Joint Creation : Reve	olute	? ×
Mechanism : Mechani	sm.1	New Mechanism
Joint name : Revolute.	1	
Line 1 :	Current	selection :
Plane 1 :	Plane 2 :	🗢 🔍 Null Offset 🔿 Offset = 🛛 🕅 🚍
Plane 3 : 💽	Plane 4 :	O Centered
□ A	angle driven	
		OK Cancel

DMU Kinematics Simulator

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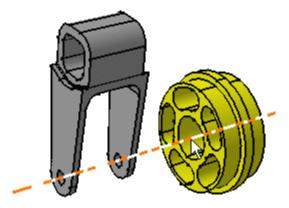
Remember you can use the preselection navigator, it can be helpful to select the geometry.Refer to Selecting Using the Preselection Navigator in the *Infrastructure User's Guide*

4. Select Line 1 in the geometry area. In our example select the hinge axis as shown below:

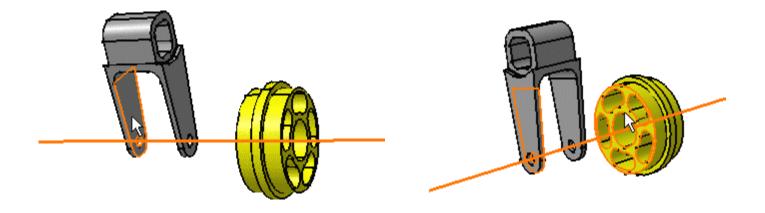


5. Select Line 2 in the geometry area. Select the wheel axis:

The dialog box current selection field is automatically updated.



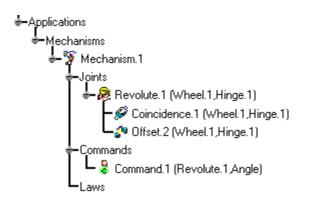
- **6.** Select the planes as shown below:
 - Plane 1: select the left inner hinge plane
 - Plane 2: select the left wheel surface

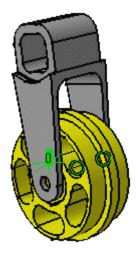


7. Assign the Angle driven command if needed.

Joint Creation : Revolute		?×
Mechanism : Mechanism.1		New Mechanism
Joint name : Revolute.1		
	Current selection :	
Line 1 : Hinge.1/Solid.1	Line 2 : Wheel.1/Solid.1	
Plane 1 : Hinge.1/Solid.1		: O Offset = Omm
Plane 3 : -	Plane 4 :O Centered	
📮 Angle driven		
		OK Cancel

8. Click Ok to end the Revolute Joint creation. The specification tree is updated





9. Open the Create_Coincidence_Revolute.CATProduct to check your result.

Please refer to Creating Revolute Joints with Offset (Advanced mode)

ĺ



Creating Prismatic Joints

This task shows how to create prismatic joints in a V5 mechanism.

Open the Prismatic.CATProduct document.

Automatic switch to Design mode:

If you work with the cache system in visualization mode, you no longer need to use **Edit**->**Representations**->**Design Mode** beforehand as the switch to design mode is automatic (an eye appears as you point the product in the geometry or specification tree). All you need to do is click on the object.

You can now use axis systems to create prismatic joints. Select the joint from axis icon kinematic toolbar. Please refer to Creating Joints Using Axis

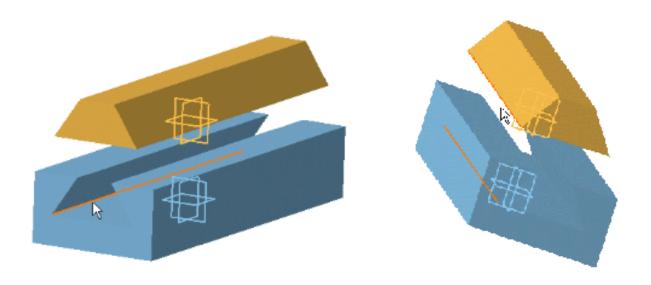
- Click the Prismatic Joint icon from the DMU Kinematics Joints Toolbar or select Insert -> New Joint -> Prismatic from the Menu bar. The Joint Creation: Prismatic dialog box appears.
- **2.** Click New Mechanism. The Mechanism Creation dialog box is displayed:

Note: this new dialog box lets you enter a meaningful name for the mechanism. Click Ok when done.

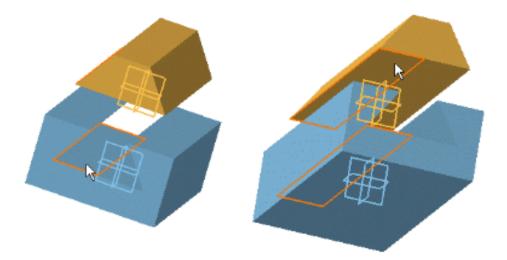


In our example, keep the default name Mechanism.1. The Mechanism is identified in the specification tree. *Now you need to select two lines and two planes*

- **3.** Select Line 1 in the geometry area. In our example select an edge (fix.1)
- 4. Select Line 2 in the geometry area. Select a second edge (slot.1)

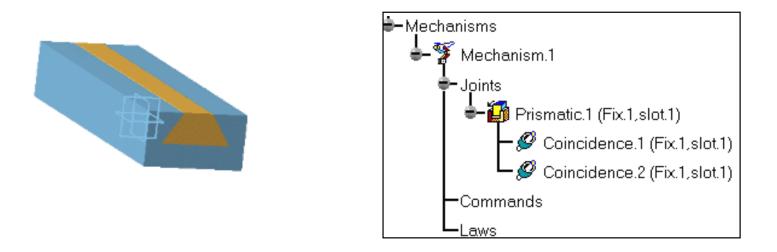


5. Select Plane 1 and Plane 2 as shown below:



6. Click **Ok** to end the prismatic joint creation.

Joint Creation : Prismatic	<u>? ×</u>
Mechanism : Mechanism.1	New Mechanism
Joint name : Prismatic.1	
Current	t selection :
Line 1 : Fix.1/Pad.1	Line 2 : slot.1/Pad.1
Plane 1 : Fix.1/Pad.1	Plane 2 : slot.1/Pad.1
Length Driven	
	OK Gancel



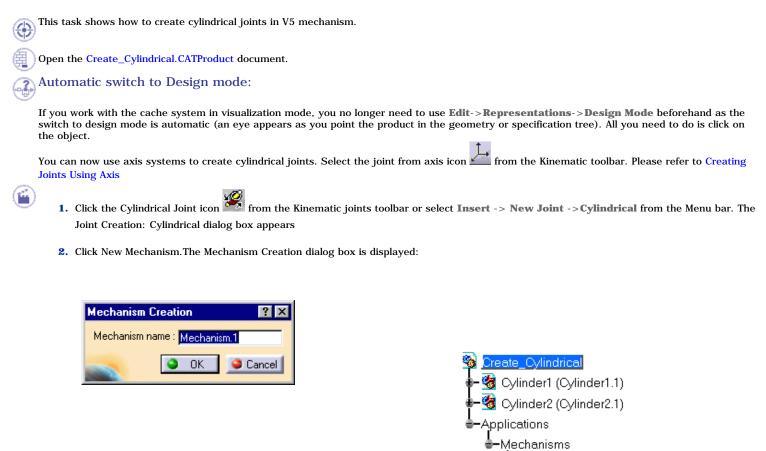
The prismatic joint is created and identified in the specification tree



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Creating Cylindrical Joints



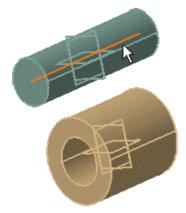
In our example, keep the default name Mechanism.1. The Mechanism is identified in the specification tree.

-Joints -Commands Laws

Mechanism.1

Now you need to select two lines

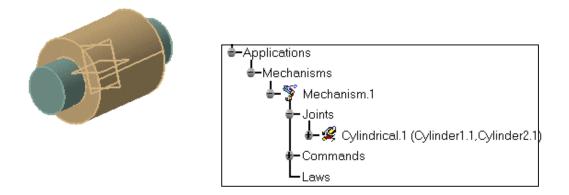
- 3. Select Line 1 in the geometry area. In our example select the Cylinder 1 axis
- 4. Select Line 2 in the geometry area. Select the Cylinder 2 axis



- 5. Select the Angle Driven and Length Driven check boxes
- 6. Click Ok to end the cylindrical joint creation.

Joint Creation : Cylindrical	<u>? ×</u>
Mechanism : Mechanism.1	New Mechanism
Joint name : Cylindrical.1	
Cur	rrent selection :
Line 1 : Cylinder1.1/Pad.1	Line 2 : Cylinder2.1/Pad.1
Angle driven	🔎 Length Driven
	OK Sancel

The joint is created and identified in the specification tree



7. Define a Fixed part, for this click the Fixed Part icon and select the object (Cylinder 2). The mechanism can be simulated

8. Open the Create_Cylindrical_Result.CATProduct document to check your result.



Creating Planar Joints

This task shows how to create planar joints in a V5 mechanism.

Open the Create_Planar.CATProduct document.

Automatic switch to Design mode:

If you work with the cache system in visualization mode, you no longer need to use **Edit->Representations-**>**Design Mode** beforehand as the switch to design mode is automatic (an eye appears as you point the product in the geometry or specification tree). All you need to do is click on the object.



1. Click the Planar Joint icon from the DMU Simulation Toolbar or select Insert -> New Joint -> Planar from the Menu bar. The Joint Creation: Planar dialog box appears.

2. Click New Mechanism. The Mechanism Creation dialog box is displayed:

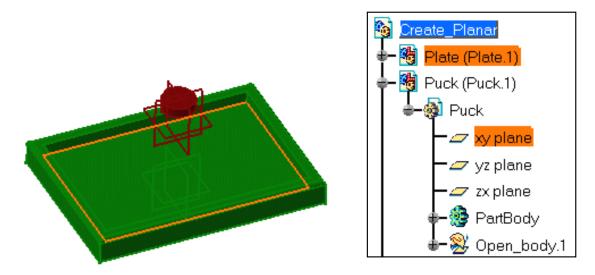
Mechanism Creation		
Mechanism name : Mechanism.1		
🥌 💽 ок 🏓	Cancel	

In our example, keep the default name Mechanism.1

Joint Creation : F	Planar	? ×
Mechanism : Mec	hanism.1	▼ New Mechanism
Joint name : Plana	ar.1	
	Current selection	:
Plane 1 :	Plane 2	2:
	L	OK Cancel

The Mechanism is identified in the specification tree. *Now you need to select two planes.*

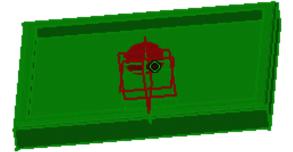
- 3. Select Plane 1 in the geometry area (plate inner face)
- **4.** Select Plane 2 in the specification tree (Puck xy plane)

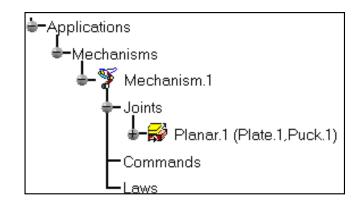


5. Click **Ok** to end the planar joint creation.

Joint Creation : Planar	<u>? ×</u>
Mechanism : Mechanism.1	New Mechanism
Joint name : Planar.1	
Current s	election :
Plane 1 : Plate.1/Solid.1	Plane 2 : Puck.1/xy plane
	OK Sancel

The planar joint is created and identified in the specification tree







14

Creating Gear Joints

This task shows how to create gear joints in a V5 mechanism.

Open the Create_Gear.CATProduct document.

Automatic switch to Design mode:

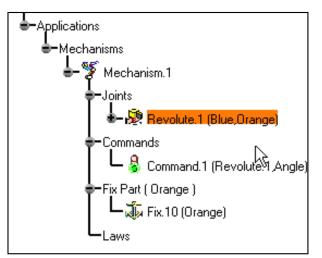
If you work with the cache system in visualization mode, you no longer need to use **Edit->Representations->Design Mode** beforehand as the switch to design mode is automatic (an eye appears as you point the product in the geometry or specification tree). All you need to do is click on the object.

1. Click the Gear Joint icon **i** from the Kinematic Joints Toolbar or select **Insert** -> **New Joint** -> **Gear...** from the Menu bar. The Joint Creation: Gear dialog box appears.

Joint Creation : Gear	? ×
Mechanism : Mechanism 1	New Mechanism
Joint name : Gear.2	
	Current selection
Revolute Joint 1 :	Create Revolute Joint 2: Create
Ratio : Define	Rotation directions 🥑 Same 🔿 Opposite
Angle driven for Revolute 1	Angle driven for Revolute 2
	OK Cancel

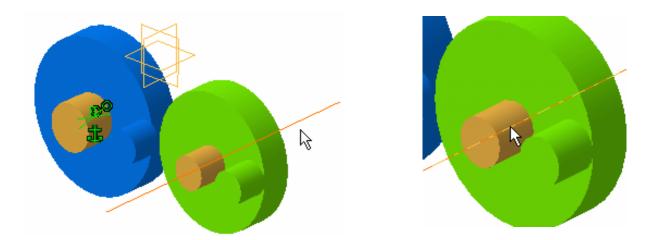
The Mechanism is identified in the specification tree. *Now you need to select two Revolute joints.*

2. Select Revolute.1 either in the specification tree or in the geometry area.

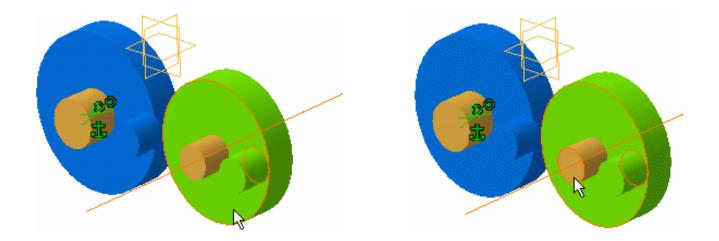


Joint Creation : Revo	olute	? ×
Mechanism : Mechanis	sm. 1	New Mechanism
Joint name : Revolute.	2	
	Current	selection :
Line 1 :	Line 2 :	
Plane 1 :	Plane 2 :	🔮 Null Offset 🔿 Offset = 🛛 🕅 🚍
Plane 3 :	Plane 4 :	O Centered
	ngle driven	
		🕒 OK 🥥 Cancel

Select line .1 (green cylinder axis) and line.2 (orange cylinder axis) as shown below in the geometry area:



Select plane.1 (green cylinder face) and plane.2 (orange cylinder face) either in the specification tree or in the geometry area:



4. Check the Offset option and keep the default value. When done, click **Ok**

Joint Creation : Revolute			? ×
Mechanism : Mechanism.1	_	New Mechanism	
Joint name : Revolute.2			
Current select Line 1 : Green/Solid.1 Line 2 : Orange/Pad.1 Plane 1 : Green/Solid.1 Plane 2 : Orange/Pad.1		Offset = -20mm	
Plane 3 : Plane 4 :	O Centered	6	
Angle driven		OK 🧕	Cancel

5. Assign a command, check for instance Angle driven for Revolute1 option.

Joint Creation : Gear	? ×
Mechanism : Mechanism.1	New Mechanism
Joint name : Gear.2	
	Current selection
Revolute Joint 1 : Revolute.1	Create Revolute Joint 2: Revolute.2 Create
Ratio : 1	Define Rotation directions 🕑 Same 🔿 Opposite
Angle driven for Revolute 1	Angle driven for Revolute 2
	OK Sancel

About Ratio definition

Two methods are available to define the ratio parameter:

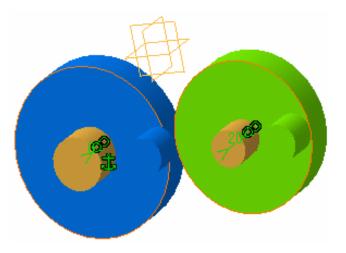
 modifying the formula (in this case the ratio is a knowledge parameter) for this, right-click in the ratio field and use the Edit Formula contextual menu displayed



- \circ using the Define option in the joint creation dialog box to calculate the ratio automatically
- 6. Click Define ... to define the ratio parameter automatically. The Gear Ratio Definition dialog box is automatically displayed

ľ	Gear Ratio	Definition	? ×
	Select two	circles	
	Radius 1 :		Radius 2 :
	Ratio :		
			Cancel

7. Select the two circles in the geometry area



8. Click Ok when done

Gear Ratio Definition	? 🗙	
Select two circles Radius 1 : 45 Ratio : 1.125	Radius 2 : 40	
	OK OCancel	
	the Joint Creation: Gear dialog box	
Joint Creation : Gear		? ×
Mechanism : Mechanism 1		New Mechanism
Joint name : Gear.2		
	Current selection	
Revolute Joint 1 : Revolute.1	Create Revolute Joint 2	Revolute.2
Ratio : 1.125	Define Rotation directions 🥥 San	ne O Opposite
Angle driven for Revolute 1		Angle driven for Revolute 2
		OK Cancel

9. Change the rotation direction option if needed.

The default is Same (positive)

Opposite is negative

i) Note: simulate your mechanism with commands to check the direction is the one you want

10. Click **Ok** when done

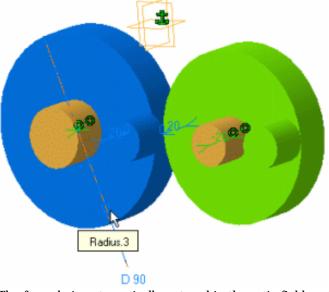
Now, if you use the formula editor, (a formula is already defined in our sample), from step 6

Right-click in the ratio field and use the Edit Formula contextual menu displayed



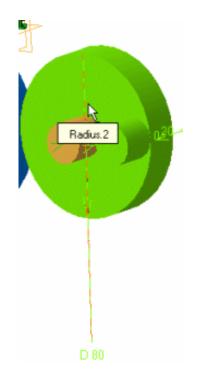
The Formula Editor: Ratio is automatically displayed.

Click Gear1 product in the specification tree and select the radius as shown below:



The formula is automatically entered in the ratio field, enter a / (division symbol)

Click Gear 2 and select radius.2 as shown below:



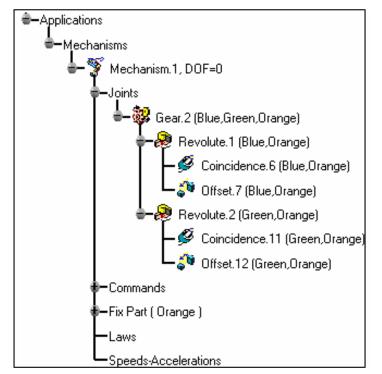
Formula Editor : Ratio		? ×
Incremental		0
Ratio		=
Gear1\Body.2\Sketch.2\R	adius.3\Radius /Gear2\PartBody\S	iketch.1\Radius.2\Radius
Dictionary	Members of Parameters	Members of All
Parameters Point Constructors Surface Constructors Law Operators Line Constructors Wireframe Constructors List	All Renamed parameters Length Boolean CstAttr_Mode	Gear2\PartBody\Sketch.1\Radius.2\Radius Gear2\PartBody\Sketch.1\Radius.2\Activity Gear2\PartBody\Sketch.1\Radius.2\mode
Gear2\PartBody\Sketch.1\	Radius.2\Radius = 40mm	OK Gancel

11. Click Ok when done. The ratio is updated

Joint Creation : Gear	? ×
Mechanism : Mechanism.1	New Mechanism
Joint name : Gear.2	
Current	selection
Revolute Joint 1 : Revolute.1	
Ratio : 1.125 f (x) Define Rotation	directions 🥑 Same 🔿 Opposite
🔎 Angle driven for Revolute 1	Angle driven for Revolute 2
	OK Scancel

The gear joint is created and identified in the specification tree.

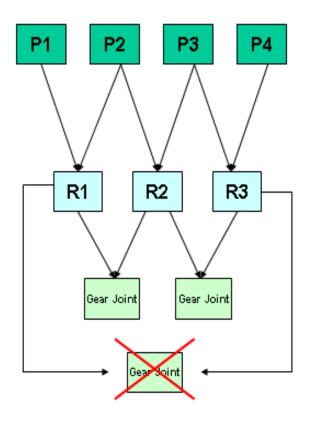
Now expand the gear joint you just created, the embedded leaf joints are displayed. **Note:** the joints involved in a compound joint can be neither edited nor deleted directly.



12. Open the GearV5_Result.CATProduct to check your result

Note: To create a gear joint, the two revolute joints involved in the gear joint must rely on a same support part.

See picture below: (P stands for Part, R for Revolute)





14

Creating Rack Joints

This task shows how to create rack joints in a V5 mechanism.

Open Create_Rack.CATProduct document.

Automatic switch to Design mode:

If you work with the cache system in visualization mode, you no longer need to use **Edit->Representations->Design Mode** beforehand as the switch to design mode is automatic (an eye appears as you point the product in the geometry or specification tree). All you need to do is click on the object.

1. Click the Rack Joint icon from the Kinematic Joints toolbar or select **Insert** -> **New Joint** -> **Rack...** from the Menu bar.

The Joint Creation: Rack dialog box appears.

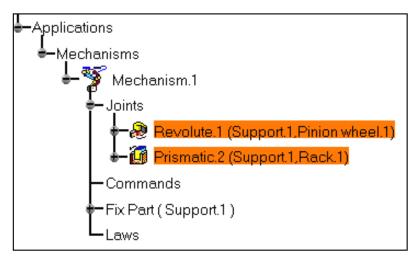
The default ratio is 1 turn per mm

Joint Creation : Rack		? 🗙
Mechanism : Mechanism.1	•	New Mechanism
Joint name : Rack.3		
	Current selection	
Prismatic Joint :	Create Revolute Joint :	Create
Ratio : 1mm_turn	Define	
Length driven for Prismatic		Angle driven for Revolute
		🕒 OK 📔 🥥 Cancel

2. Select Prismatic joint.2 in the specification tree.

If the prismatic and the revolute are not created yet, use the create button. The corresponding joint creation dialog box automatically appears. For more detailed information, see Creating Gear joints, Creating Prismatic Joints and Creating Revolute Joints

3. Select Revolute.1 the specification tree



4. Assign a command, for instance select Angle driven for Revolute check box

Joint Creation : Rack		? ×
Mechanism : Mechanism.1	Ŧ	New Mechanism
Joint name : Rack.3		
	Current selection	
Prismatic Joint : Prismatic.2	Create Revolute Joint :	Revolute.1 Create
Ratio : 1mm_turn	Define	
Length driven for Prismatic		Angle driven for Revolute
		OK Cancel

5. Click **Ok** to end the rack joint creation. The rack joint is created and identified in the specification tree. Your mechanism can be simulated: a warning message is displayed.

Now expand the rack joint you just created, the embedded leaf joints are displayed

i Note: the joints involved in a compound joint can be neither edited nor deleted directly.

- Applications
-Mechanisms
- 🌮 Mechanism.1
Joints
🖝 Commands 🛛 🔓
Fix Part (Support.1)
Laws

- **6.** Double-click Mechanism. 1 to launch the simulation with commands functionality.
- 7. Open the Rack_Result.CATProduct document to check your result

Now let's modify the ratio

8. Double-click Rack.3 in the specification tree

The Joint Edition dialog box is displayed

Joint Edition : Rack.3 (Rack)	? ×
Joint name : Rack.3	
Joints :	
Revolute Joint 1 : Prismatic. 2 Revolute Joint 2 : Revolute.1	
Ratio : 1mm_turn Define	
Length driven for the Prismatic 1 🧏 📮 Angle driven for the Revolute 2	
Joint Limits	
🖬 1st Lower Limit : 🛛 -100mm 📑 🖬 1st Upper Limit : 🛛 100mm	
🖾 2nd Lower Limit : 📴 360deg 🔚 🖾 2nd Upper Limit : 360deg	
	Cancel

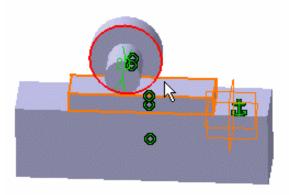
DMU Kinematics Simulator

9. Click the **Define** button

The Rack Ratio Definition dialog box appears:

Rack Ratio Definition	? ×
Select a circle	D-K-
Radius :	Ratio :
	OK Scancel

10. Select a circle in the geometry area



Rack Ratio Definition		? ×
Select a circle		
Radius : 20	Ratio : 125.664	
	OK I	Cancel

11. Click Ok

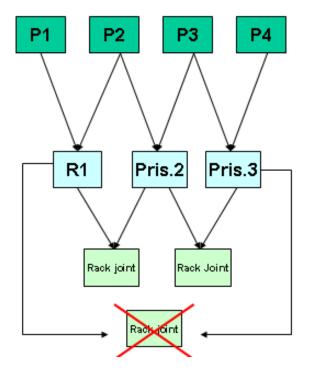
The ratio is automatically calculated

Joint Edition : Rack.3	(Rack)				? ×
Joint name : Rack.3					
		J	oints :		
Revolute Joint 1 : Prism	natic.2		Revolute Joint 2 : R	evolute.1	
Ratio: 125.664mm_turi	n De	efine			
📁 Length driven for the	e Prismatic 1		Angle driven for t	he Revolute 2	
Joint Limits					
🖾 1st Lower Limit :	-100mm		🚦 🖾 1st Upper Limit :	100mm	
2nd Lower Limit :	Unset	E	🗧 🗌 2nd Upper Limit :	Unset	E
-					
				I OK	🥯 Cancel



Notes: 1. To create a rack joint, the prismatic and revolute joints involved in the rack joint must rely on a same support part.

See picture below: (P stands for Part, R for Revolute, Pris. for Prismatic)



2. It is possible to set a negative value for the ratio. For example, if the translation direction is not consistent with the rotation direction, change the sign of the ratio .



Creating Cable Joints

This task shows how to create cable joints in a V5 mechanism.

Open the Create_Cable.CATProduct document.

Automatic switch to design mode:

If you work with the cache system in visualization mode, you no longer need to use **Edit->Representations->Design Mode** beforehand as the switch to design mode is automatic (an eye appears as you point the product in the geometry or specification tree). All you need to do is click.

1. Click the Cable Joint icon 🙀 from the Kinematic Joints Toolbar or

select Insert -> New Joint -> Cable from the Menu bar.

The Joint Creation: Cable dialog box appears.

Joint Creation : Cable	? ×
Mechanism : Mechanism.1	echanism
Joint name : Cable.3	
Current selection	
Prismatic Joint 1 : Create Prismatic Joint 2 :	Create
Ratio :	
Length driven for Prismatic 1 Length driven for Prism	atic 2
ОК	Sancel

Now you need to select two prismatic joints

2. Select Prismatic Joint 1 and Prismatic Joint 2 in the specification tree

D If the prismatic joints are not created yet, use the create button. The Joint creation: Prismatic dialog box automatically appears. For more detailed information, see Creating Gear joints and Creating Prismatic Joints

3. Assign a command, for example select the Length driven for Prismatic 2 check box

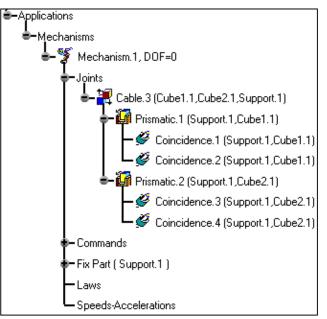
Joint Creation : Cable	? ×
Mechanism : Mechanism.1 New Mechanis	m
Joint name : Cable.3	
Current selection	11111111111
Prismatic Joint 1 : Prismatic.1 Create Prismatic Joint 2 : Prismatic.2	Create
Ratio : 1	
Length driven for Prismatic 1	
	1.1.1
	Cancel

4. Click Ok to end the cable joint creation.

The mechanism can be simulated

The cable joint is created and identified in the specification tree. Now expand the cable joint you just created, the embedded leaf joints are displayed. **Note:** the joints involved in a compound joint can be neither edited nor deleted directly.

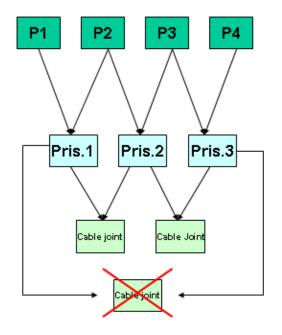
DMU Kinematics Simulator



- 5. Double-click Mechanism.1 in the specification tree to launch the Simulation with Commands or click the Simulation with Commands icon
- 6. Open the Cable_Result.CATProduct document to check your result.

Note: To create a cable joint, the two prismatic joints involved in the cable joint must rely on a same support part.

See picture below: (P stands for Part, Pris. for Prismatic)





Creating Screw Joints

This task shows how to create Screw joints in a V5 mechanism.

Open the Create_Screw.CATProduct document.

When you create joints, you can define the mechanism within the same dialog box. Remember though, that you create a mechanism independently from the joints by selecting **Insert**->**New Mechanism...** from the menu bar.

Automatic switch to Design mode:

If you work with the cache system in visualization mode, you no longer need to use **Edit**->**Representations**->**Design Mode** beforehand as the switch to design mode is automatic (an eye appears as you point the product in the geometry or specification tree). All you need to do is click on the object.

1. Click the arrow within the Revolute Joint icon from the DMU Kinematics toolbar (Revolute joint is

the default joint type)

2. Undock the Kinematics Joints toolbar:

Kinematic joints										×
l 😣 🚺 🗶 🍹	9 🥰 🗱	**	6 20	*	2	\$	69	:	2	4
	6.00									

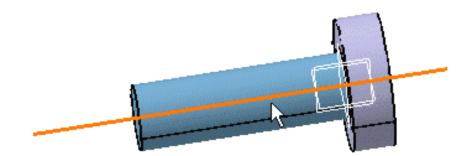
- **3.** Select the Screw Joint icon **Screw** Joint dialog box is displayed.
- 4. Click on New Mechanism. The Mechanism Creation dialog box is displayed:

Mechanism Cr	eation	? ×
Mechanism nar	ne : <mark>Mechani</mark>	sm.1
	🌖 ОК	Cancel

In our example, keep the default name Mechanism.1.

Joint Creation : Screw	? ×
Mechanism :	New Mechanism
Joint name :	
Cu	rrent selection :
Line 1 :	Line 2 :
Angle driven	Length Driven
Pitch : 1	
	OK Scancel

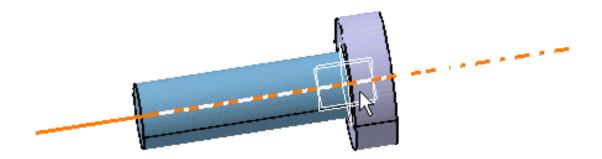
5. Select Line 1 either in the geometry area or in the specification tree. In our example, select the screw cylinder axis as shown below:



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The current selection field is automatically updated.

6. Select Line 2, the Part2 cylinder axis:



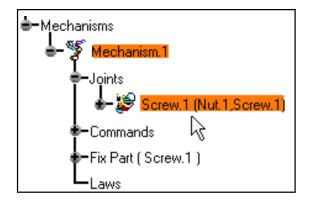
7. Enter 10 in the pitch field and select the **Length Driven** check box.

Note: for all joints: the name of parts involved in the joint creation appears in the joint creation dialog box

Joint Creation : Screw	? ×
Mechanism : Mechanism 1	New Mechanism
Joint name : Screw.1	
C	urrent selection :
Line 1: Pad 1	Line 2 : Solid.1
Angle driven	🖾 Length Driven
Pitch: 1	
	OK Scancel

8. Click **Ok** to end the Universal joint creation.

The specification tree is updated



9. Open the Screw_Result.CATProduct to check your result. (In this sample document, we added a fixed part which means the mechanism can be simulated)

10. Double-click Mechanism1 or click the Simulation With Commands icon 🕮

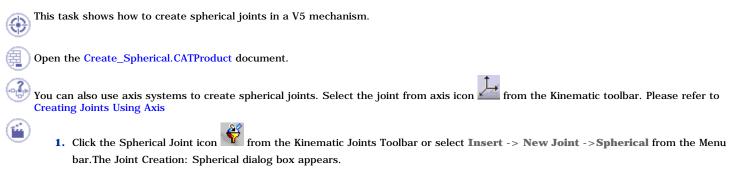


For more information, please refer to About Joints and Creating Mechanisms and Joints.



Version 5 Release 13

Creating Spherical Joints



2. Click New Mechanism. The Mechanism Creation dialog box is displayed:

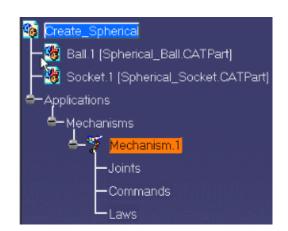
Mechanism Creation	? ×
Mechanism name : Mechanism.1	
🥌 💽 ок 💽	Cancel

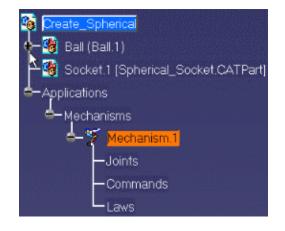
In our example, keep the default name Mechanism.1.

Joint Creatio	on : Spherical		? ×
Mechanism :	Mechanism.1		New Mechanism
Joint name :	Spherical.1		
	Current s	election :	
Point 1 :		Point 2 :	
		•	OK Scancel

Automatic switch to Design mode:

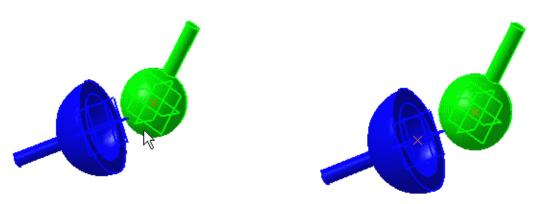
If you work with the cache system in visualization mode, you no longer need to use **Edit->Representations->Design Mode** beforehand as the switch to design mode is automatic (an eye appears as you point the product in the geometry or specification tree). All you need to do is click on the object.





The Mechanism is identified in the specification tree. *Now you need to select two points.*

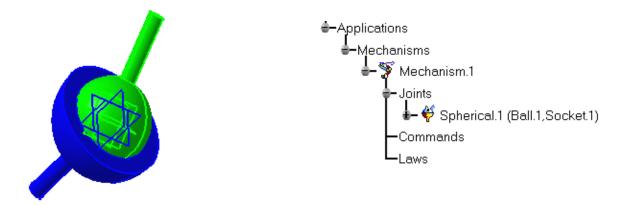
- 3. Select Point 1 in the geometry area. In our example select the ball extremity
- 4. Select Point 2 in the geometry area. Select a second point (socket extremity)



5. Click Ok to end the spherical joint creation.

Joint Creation : Spherical	<u>? ×</u>
Mechanism : Mechanism.1	New Mechanism
Joint name : Spherical.1	
Current s	election :
Point 1 : Ball.1/Shaft.1	Point 2 : Socket.1/Shaft.1
	OK Cancel

The spherical joint is created and identified in the specification tree



Creating Rigid Joints

This task shows how to create rigid joints in a V5 mechanism.

Open the Create_Rigid.CATProduct document.

Automatic switch to Design mode:

If you work with the cache system in visualization mode, you no longer need to use **Edit->Representations-**>**Design Mode** beforehand as the switch to design mode is automatic (an eye appears as you point the product in the geometry or specification tree). All you need to do is click on the object.

(***)

1. Click the **Rigid Joint** icon from the **Kinematic Joints** Toolbar or select **Insert** -> **New Joint** -> **Rigid** from the Menu bar. The Joint Creation: Rigid dialog box appears.

2. Click New Mechanism. The Mechanism Creation dialog box is displayed:

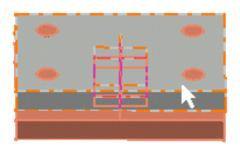
Mechanism Creation	? ×
Mechanism name : Mechanism.1	
🥌 💽 ок 🔎 с	Cancel

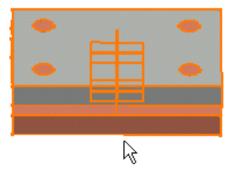
In our example, keep the default name Mechanism.1

Joint Creatio	n : Rigid		? ×
Mechanism :	Mechanism.1	-	New Mechanism
Joint name :	Rigid.1		
	Current selection	on :	
Part 1 :	Part 2	2:	
		۵ (IK Sancel

The Mechanism is identified in the specification tree. *Now you need to select two parts.*

- 3. Select Part 1 either in the specification tree or in the geometry area. In our example, select Plate1.
- **4.** Select Part 2 either in the specification tree or in the geometry area. Select Plate2.

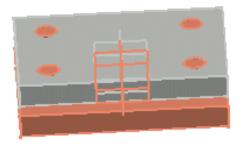


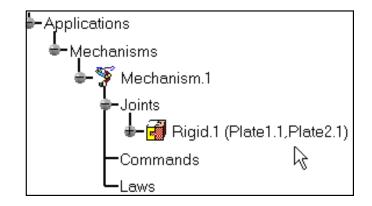


5. Click Ok to end the rigid joint creation.

Joint Creation : Rigid	? ×
Mechanism : Mechanism.1	New Mechanism
Joint name : Rigid.1	
Curren	t selection :
Part 1 : Plate1.1	Part 2 : Plate2.1
	QK QCancel

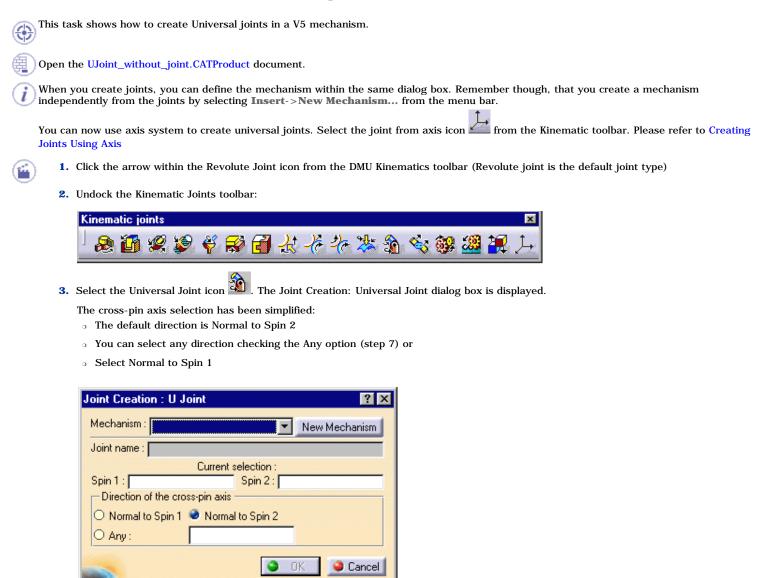
The rigid joint is created and identified in the specification tree.







Creating Universal Joints

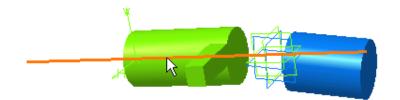


4. Click on New Mechanism. The Mechanism Creation dialog box is displayed:



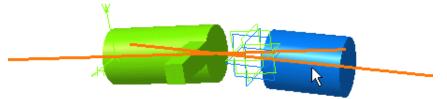
In our example, keep the default name Mechanism.1.

5. Select Spin 1 either in the geometry area or in the specification tree. In our example, select the green cylinder axis as shown below:

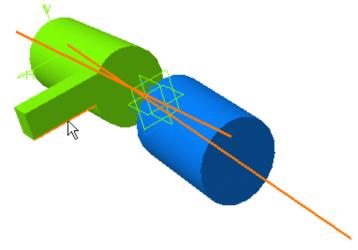


The current selection field is automatically updated.

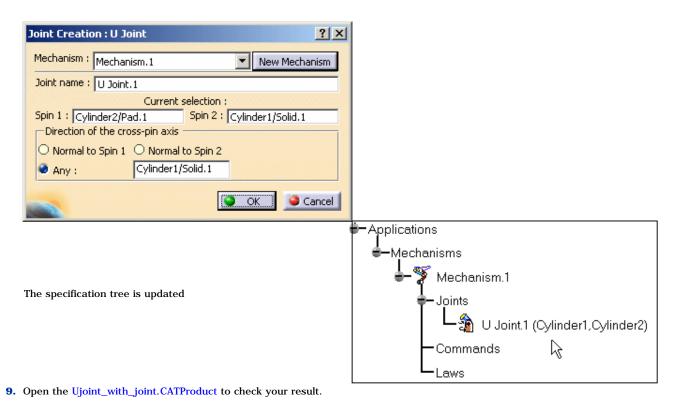
6. Select Spin 2, for example the blue cylinder axis:



7. Select the cross-pin axis direction. Select the **Any** option. In this case, you need to select the direction of the cross-pin axis (which has to be perpendicular with one of the two axis previously selected). In our example, select an edge of the green cylinder.



8. Click Ok to end the Universal joint creation.



For more information, please refer to About Joints and Creating Mechanisms and Joints.

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Creating CV Joints

This task shows how to create Point Surface joints in a V5 mechanism.

Open the Create_CVjoint.CATProduct document.

When you create joints, you can define the mechanism within the same dialog box. Remember though, that you create a mechanism independently from the joints by selecting **Insert**->**New Mechanism...** from the menu bar.

1. Click the arrow within the Revolute Joint icon from the DMU Kinematics toolbar (Revolute joint is

the default joint type)

2. Undock the Kinematic Joints toolbar:

Kinematic joints		×
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- **3.** Select the **CV Joint** icon **Solution**. The Joint Creation: CV Joint dialog box is displayed.
- **4.** Click on New Mechanism. The Mechanism Creation dialog box is displayed:

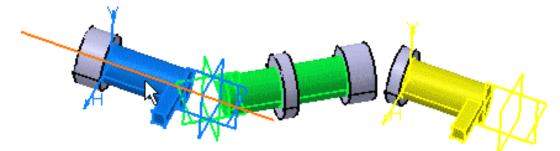


In our example, keep the default name Mechanism.1.

Joint Creation : (CV Joint	? ×
Mechanism : Mec	chanism.1	New Mechanism
Joint name : CV J	oint.1	
Cu	rrent selection :	
Spin 1 :	Spin 2 :	Spin 3:
		OK Scancel

5. Select Spin 1 either in the geometry area or in the specification tree. In our example, select the blue cylinder axis as shown below:

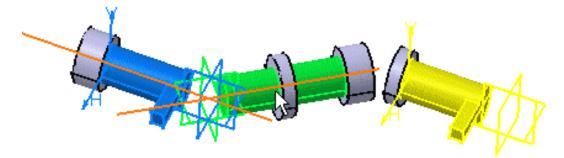




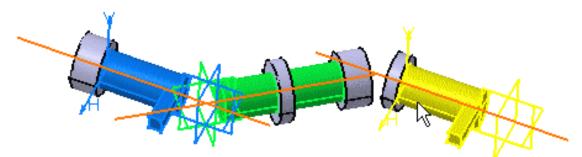


The current selection field is automatically updated.

6. Select Spin 2, for example the green cylinder axis:



7. Select Spin 3, the yellow cylinder axis.

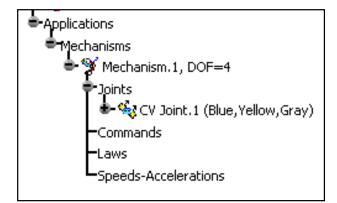


8. Click **Ok** to end the CV joint creation.

Joint Creation : CV Joint	<u>? ×</u>
Mechanism : Mechanism.1	New Mechanism
Joint name : CV Joint.1	
Current selection :	
Spin 1 : Blue/Solid.1 Spin 2 : Gray/Solid.1	Spin 3 : Yellow/Solid.1
	OK Cancel

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The tree is updated



Note, you need to add a fixed part and to create revolute joints (with at least one command) to simulate this mechanism.

9. Open the CVjoint_Result.CATProduct document to check your result and simulate the mechanism (double-click mechanism.1 to display the Simulation with command dialog box).

For more information, please refer to About Joints and Creating Mechanisms and Joints.



Creating Joints Using Axis Systems



This task shows how to create joints using the axis systems

Open the Ujoint_axis_without_kin.CATProduct document.

Automatic switch to Design mode:

If you work with the cache system in visualization mode, you no longer need to use **Edit**->**Representations**->**Design Mode** beforehand as the switch to design mode is automatic (an eye appears as you point the product in the geometry or specification tree). All you need to do is click on the object.



1. Click the Joint from axis icon from the Kinematic Joints toolbar.

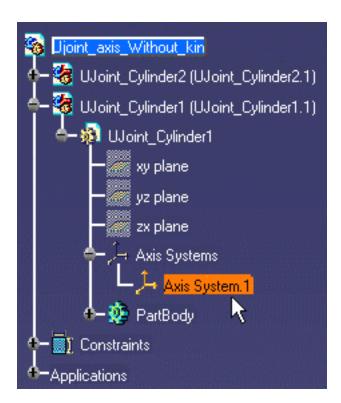
The Joint Creation using axis dialog box appears:

Joint Creation using Axis	? ×
Mechanism : Mechanism.2	New Mechanism
Joint name : U Joint 1	
Joint Type : U Joint	
Cur	rent selection
Axis 1 :	Axis 2 :
Ratio : 1	
Command 1	Command 2
	🕒 OK 🥥 Cancel

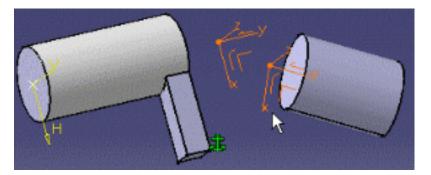
- **2.** Define the joint type you want from the Joint Type drop-down list:
 - o **u joint**
 - o prismatic
 - o revolute
 - o cylindrical
 - o spherical

In our example we keep the default type: u joint

3. Click the axis1 selection field and select Ujoint_Cylinder1 Axis System.1 either in the geometry or in the specification tree:



4. Click the axis2 selection field and select Ujoint_Cylinder2 Axis System.1 either in the geometry or in the specification tree



5. Click **Ok** to end the u joint creation.

Joint Creati	on using Axis		? ×
Mechanism :	Mechanism.2	-	New Mechanism
Joint name :	U Joint.1		
Joint Type :	U Joint 🔡	-	
		Current selection	η
Axis 1 :	Axis System.1	Axis 2 : Ax	is System.1
Ratio :	1		
Comman	d 1		Command 2
			DK 🥥 Cancel

The u joint is created and identified in the specification tree:



Constraints are created:

- $_{\odot}$ $\,$ coincidence constraint between the axis systems origins
- \circ angle (perpendicular constraint = 90deg) between: x axis from first axis system and z axis from the second axis system

The interest of this creation mode lies in the ability to keep associativity even if you move one part involved in the joint. Before, you could not update the joint.

i.e. If you move a part involved in the u joint, use the update command to make sure the u joint remains associative with the parts involved.

6. Open the Ujoint_axis_with_kin.CATProduct document to check your result

For more detailed information about constraints created using the axis system mode, see More About Resulting Constraints

Note that it is impossible to create joints using axis from V4 models.



More about Resulting Constraints

i DMU Kinematics Simulator lets you define and edit 17 different joint types.

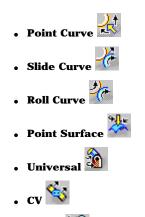
The table below describes the constraint types created when using the axis mode to create the following joints:

V4 NAME	JOINT TYPE	RESULTING CONSTRAINT TYPES
u jnt	U Joint	 coincidence (between axis systems origins) angle 90deg (x axis1/z axis2) Notice you can define u joints without constraints (See Creating Universal Joints)
revolute	Revolute	 coincidence (z axis1/z axis2) coincidence (xy plane1/ xy plane2)
prismatic	Prismatic	 coincidence (z axis1/z axis2) coincidence (yz plane1/yzplane2)
actuator	Cylindrical	 coincidence (z axis1/z axis2)
pt/pt	Spherical	• coincidence (between axis systems origins)
screw	Screw Joint	 coincidence (z axis1/z axis2) (+ pitch)



Designing Higher Pair Joints

The following list shows these 7 particular joint types which do not associate assembly constraints during creation:





The conditions under which you can create these joints are the following:

Point Curve and Point surface: the point has to be on the curve. **Slide and Roll curves:** the two curves are in contact and tangent in this point.

Note: to create these four kinematics joints, the parts involved in the joint creation must be well positioned. Please refer to Tips for Curve or Surface Joints Creation



Create Point Curve joints: select **Insert**->**New Mechanism...** from the menu bar or click the Point Curve Joint icon, then click New Mechanism in the dialog box displayed. Select one curve and one point and click Ok.

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Create Point Surface joints: select Insert->New Mechanism... from the menu bar or click the Point Surface Joint icon, then click New Mechanism in the dialog box displayed. Select one surface and one point, when done click Ok.



Create Roll Curve joints: select Insert->New Mechanism... from the menu bar or click the Roll curve Joint icon, then click New Mechanism in the dialog box displayed. Select two curves and click Ok.



Create Slide Curve joints: select Insert->New Mechanism... from the menu bar or click the Slide Curve Joint icon, then click New Mechanism in the dialog box displayed. Select two curves and click Ok.

You can easily modify Curve Joint specifications either **editing** the existing geometry (**positioning** or **definition**) or **replacing** the existing geometry with a new one. The V5 joint mechanism is updated accordingly.

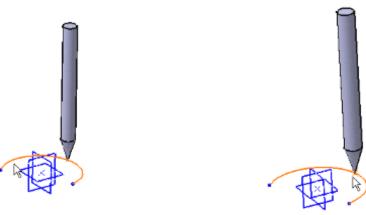
Please refer the following Chapter:

Editing Joints -Introduction and read the three corresponding step-by-step scenarios

Creating Point Curve Joints



- 5. Select Curve 1 in the geometry area. The current selection field is automatically updated with your selection
- 6. Select Point 1 in the geometry area. In our example, select Point.1 either in the geometry or in the specification tree



Joint Crea	tion : Point Curve	? ×
Mechanism	Mechanism.1	New Mechanism
Joint name	Point Curve.1	
	Curre	nt selection :
Curve 1 :	Curve.1/Circle.1	Point 1 : Pencil.2/Point.1
	Length Driven	
		OK OK Cancel

You can check the command orientation defined by a green arrow in the geometry area and in this example (command assigned to a revolute) but you cannot change its orientation

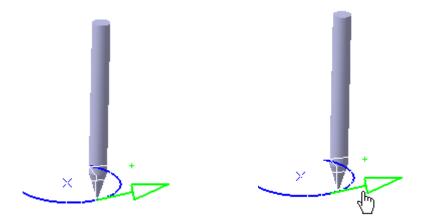
- $_{\odot}~$ The parts which are not involved in the joint creation are displayed in low light (to easily locate the joint you are working on)
- Pass the cursor over the green arrow to launch a short animation

Let's say you forgot to assign a command, double-click the Point Curve joint in the specification tree:

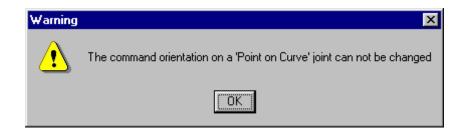
The following dialog box appears: check the 'Length Driven' option

Note: in edition mode you can see the command orientation identified by a green arrow:

Joint Edition : Point Curve.1 (Point Curve)	? ×
Joint name : Point Curve.1	
Joint geometry :	
Curve 1 : Curve.1/Circle.1 Point 1 : Pencil.2/Point.1	
Joint Limits	
🖾 Lower Limit : 🛛 -9.737mm 📑 🖾 Upper Limit : 190.263mm	
	Cancel

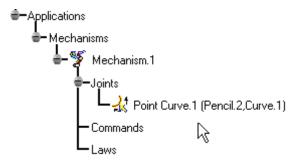


M If you try inadvertently to change the command orientation on a point curve joint, the following warning message is displayed



7. Click Ok to end the point curve joint creation

The specification tree is updated:





It is impossible to create point curve joints if the parts involving in the joint are not well positioned. For more detailed information, please refer to Tips for Curve or Surface Joints Creation



Creating Point Surface Joints

This task shows how to create point surface joints in a V5 mechanism.

Open the PointSurface_without_Joint.CATProduct document.

Automatic switch to Design mode:

If you work with the cache system in visualization mode, you no longer need to use **Edit**->**Representations**->**Design Mode** beforehand as the switch to design mode is automatic (an eye appears as you point the product in the geometry or specification tree). All you need to do is click. When you create joints, you can define the mechanism within the same dialog box. Remember though, that you create a mechanism independently from the joints by selecting **Insert**->**New Mechanism...** from the menu bar.

- **1.** Click the arrow within the Revolute Joint icon from the DMU Kinematics toolbar (Revolute joint is the default joint type).
- **2.** Undock the Kinematic Joints toolbar:

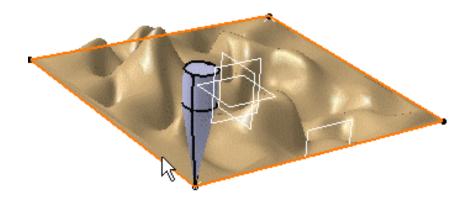
Kinematic joints						×
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		-				

- **3.** Select the **Point Surface Joint** icon The Joint Creation: Point Surface dialog box is displayed.
- 4. Click on New Mechanism. The Mechanism Creation dialog box is displayed:



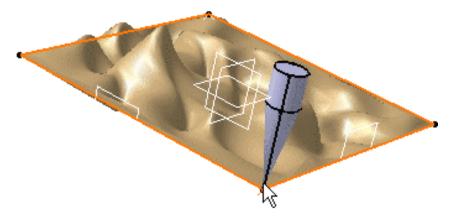
In our example, keep the default name Mechanism.1.

5. Select Surface 1 either in the geometry area or in the specification tree.



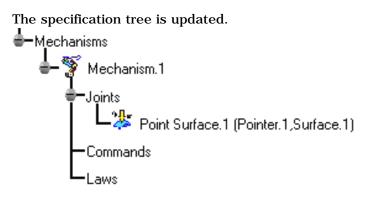
The current selection field box is automatically updated with your selection.

6. Select Point 1 either in the geometry area or in the specification tree.



7. Click **Ok** to end the point surface joint creation.

Joint Creatio	n : Point Surface		? ×
Mechanism :	Mechanism.1	•	New Mechanism
Joint name :	Point Surface.1		
	Current se	election :	
Surface 1 :	5urface.1/Loft.1	Point 1 : Poir	nter.1/Extremity
		e c	K Cancel



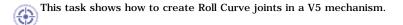
8. Open the PointSurface_with_Joint.CATProduct document to check your result.

It is impossible to create point surface joints if the parts involving the joint are not well positioned. For more detailed information, please refer to Tips for Curve or Surface Joints Creation

For more information, please refer to About Joints and Creating Mechanisms and Joints.

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Creating Roll Curve Joints



Open the RollCurve_without_kin.CATProduct document.

*W*hen you create joints, you can define the mechanism within the same dialog box. Remember though, that you create a mechanism independently from the joints by selecting **Insert**->**New Mechanism...** from the menu bar.

- 1. Click the arrow within the Revolute Joint icon from the DMU Kinematics toolbar (Revolute joint is the default joint type).
- 2. Undock the Kinematic Joints toolbar:

Kinematic joints				l.
l 😣 🚹 🗐 🌮	🧳 💕 🛃	なやや	🎠 🏠 🐝	🈻 🌌 🛃 🔎

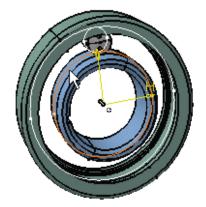
- **3.** Select the **Roll Curve Joint** icon *****. The Joint Creation: Roll Curve dialog box is displayed.
- **4.** Click on New mechanism. The Mechanism Creation dialog box is displayed:

Mechanism Creation 🔗 🏾						
Mechanism name : Mechanism.1						
🔷 ОК	Cancel					

Joint Crea	ion : Roll Curve			? ×	
Mechanism	C Mechanism.1		▼ New	Mechanism	
Joint name	Roll Curve.1				
Current selection :					
Curve 1 :		Curve 2 :			
C	Length Driven				
) OK	Cancel	

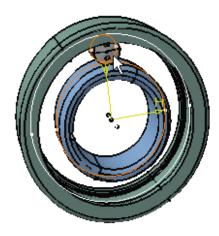
In our example, keep the default name Mechanism.1.

5. Select Curve 1, in our example, select the inner ring in the geometry area. The current selection field is automatically updated with your selection.



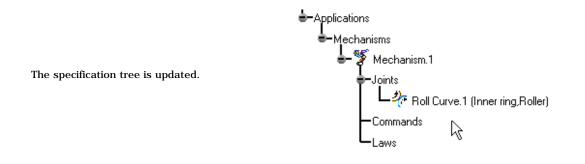
DMU Kinematics Simulator

6. Select Curve 2, for instance select the roller as shown below:

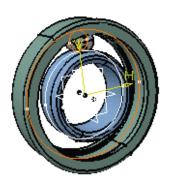


7. Click **Ok** to end the roll curve joint creation.

Joint Creatio	on : Roll Curve		? ×			
Mechanism :	Mechanism.1	•	New Mechanism			
Joint name :	Roll Curve.1					
Current selection :						
Curve 1 : In	ner ring/Sketch.2	Curve 2 : Rol	ler/Sketch.1			
	Length Driven					
			Cancel			



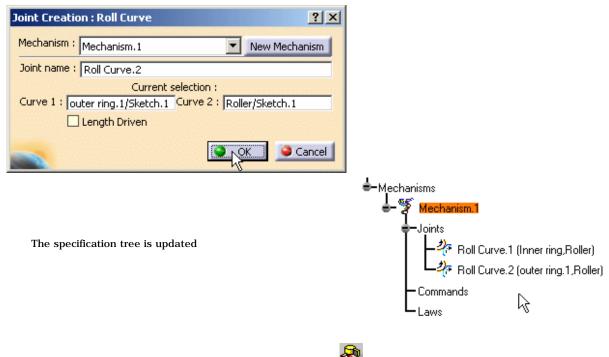
- Now, create Roll Curve. 2. For this select the Roll Curve Joint icon again. The Joint Creation: Roll Curve dialog box is displayed.
- 9. Select the outer ring as curve 1 and the roller as curve 2 in the geometry area



DMU Kinematics Simulator

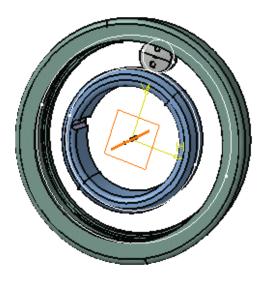
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- **11.** Now, create Revolute. 3. For this select the Revolute Joint icon The Joint Creation: Revolute dialog box is displayed.
- **12.** Select the lines and the planes:
 - $_{\circ}$ $\,$ Inner ring axis for line 1 $\,$
 - \circ outer ring axis for line 2
 - zx plane (inner ring)
 - zx plane (outer ring)

Remember you can use the preselection navigator, it can be helpful to select the planes. Refer to Selecting Using the Preselection Navigator in the *Infrastructure User's Guide*



Joint Creation : Revolute				<u>? ×</u>
Mechanism : Mechanism.1		•	New Mechani	ism
Joint name : Revolute.3				
	Current selection :			
Line 1 : Inner ring/Solid.1	Line 2 : outer ring.1/Shaft.1			
Plane 1 : Inner ring/zx plane	Plane 2 : outer ring.1/zx plane	Null Offset	Offset = Omm	
Plane 3 : -	Plane 4 : [(Centered		
📮 Angle driven				
			٢	OK Gancel

In Edition mode only:

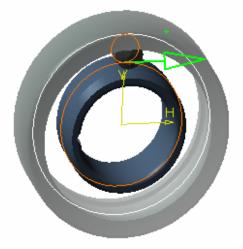
you can check the command orientation defined by a green arrow in the geometry area and in this example (command assigned to a revolute), you can change its orientation:

- The parts which are not involved in the joint creation are displayed in low light (to easily locate the joint you are working on): **Picture. 1**
- $_{\odot}~$ Pass the cursor over the green arrow to launch a short animation
- Click the arrow to change the command orientation if necessary: Picture. 2

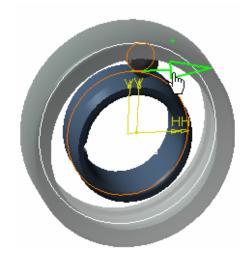
Picture .1	Picture .2

Now, if you delete the command and assign it to Joint.1 for instance, all you can do is check the orientation, you cannot change the command orientation.

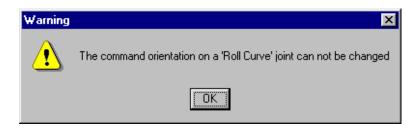
Joint Edition : Roll Curve.1 (Roll Curve)		? ×
Joint name : Roll Curve.1		_
	eometry :	
Curve 1 : Inner ring/Sketch.2	Curve 2 : Roller/Sketch.1	
🔎 Length Driven		
Joint Limits		
Lower Limit : 2.102mm	Upper Limit : 202.102mm	
	🇿 ОК 🕒 Арріу 📔 Са	incel



Pass your cursor over to launch the animation:



If you try inadvertently to change the command orientation on a roll curve joint, the following warning message is displayed



14. Click **Ok** to end the revolute joint creation.

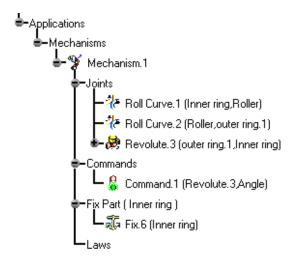
The specification tree is updated.

15. Click the Fixed Part icon from the Simulation toolbar or select **Insert**->**Fixed Part**... from the menu bar. The New Fixed Part dialog box is displayed.

New Fixed Part	? ×
Mechanism : Mechanism.1	New Mechanism
	Sancel

16. Select the inner ring as Fixed Part either in the geometry area or in the specification tree.

The specification tree is updated and the mechanism can be simulated.



Open the RollCurve_with_kin.CATProduct document to check your result.

It is impossible to create roll curve joints if the parts involving the joint are not well positioned. For more detailed information, please refer to Tips for Curve or Surface Joints Creation

i For more information, please refer to About Joints and Creating Mechanisms and Joints.



Editing Curve Joints- Introduction

About Joint Modification:

It may prove useful to change a mechanism behavior or to improve its design. To do so, you need to modify joints specifications. Once you performed your modifications, the parts involved in the mechanism need to be reassembled.

For kinematics joints based on assembly constraints (i.e. Revolute Joint, Spherical Joint,

Cylindrical Joint, Gear Joint, Rack Joint, Cable Joint, Screw Joint, Axis Joint), this capability is

already available as you can modify the underlying constraints. You can now modify the curve

joints specifications

Dealing with **mechanisms which can be simulated**:

you can edit point curve, roll curve, point surface or slide curve joints definition, modifying their underlying geometric elements (point or curve). There are three procedures to perform such a modification:

- you keep the geometric element itself but you change its position. The mechanism can no longer be simulated:
 - click the **Update Positions** icon **Sec.** The parts involved in the joint are reassembled
 - simulate your mechanism using either the Simulation With laws or Simulation With Commands.
 The mechanism is updated
- you keep the geometric element itself but you change its definition. The mechanism can no longer be simulated.
 - click the Update Positions icon . The parts are reassembled, the mechanism can be simulated
 - simulate your mechanism using either the Simulation With laws or Simulation With Commands.
 The mechanism is updated

1

Please refer to Using the Update Command

- You do not keep the geometric element, you change it by another one, using the joint definition command:
 - $_{\odot}~$ double-click the curve joint to be modified.
 - in the Edit dialog box displayed, select the curve/point to be replaced.
 - then, select a new curve/point in the geometry area and click Apply.
 - when done click OK.

Editing Point Curve Joints (modifying geometry position) Editing Point Surface Joints (modifying joints definition) Replacing Slide Curve Joint Specifications



Editing Point Curve Joints

(modifying geometry position)

This task shows how to edit point curve joints in a V5 mechanism modifying the elements involved in the joint position

Open the Edit_PointCurve.CATProduct document.

Automatic switch to Design mode:

If you work with the cache system in visualization mode, you no longer need to use **Edit**->**Representations**->**Design Mode** beforehand as the switch to design mode is automatic (an eye appears as you point the product in the geometry or specification tree). All you need to do is click on the object.



1. Check the mechanism can be simulated, for this: click the Simulation With Commands

icon 🕮 from the Simulation toolbar

The Kinematics Simulation dialog box appears:

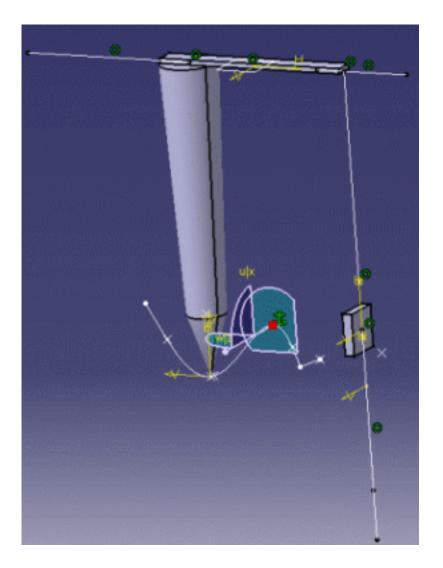
Note: the state of the dialog box depends on your settings (expanded or collapsed) The command of the kinematic mechanism is available as shown below.

nism.1	-
300 100.0000	.
Analysis	More>>
	300 100.0000

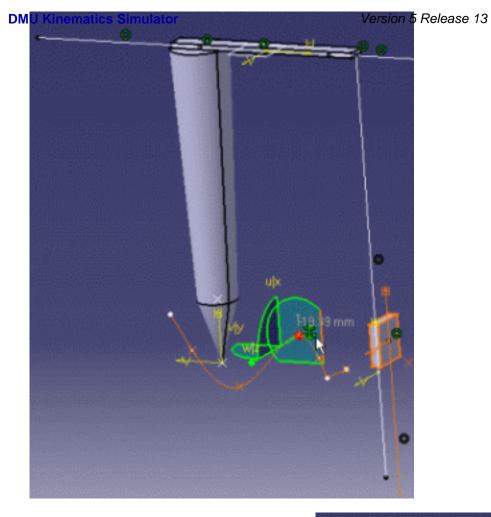
- **2.** Run your simulation using the slider of the command.
- **3.** Click the **Reset** button and when done, click **Close**.
- 4. Modify the geometry position: in our example, you are going to move the curve.Rightclick the 3D compass and select Snap Automatically to Selected Object item from the contextual menu displayed:

Lock Current Orientation Lock Privileged Plane Orientation Parallel to Screen
Make XY the Privileged Plane
Make YZ the Privileged Plane
$Ma\underline{k}e \times Z$ the Privileged Plane
Mak <u>e</u> Privileged Plane Most Visible
Snap Automatically to Selected Object
E <u>d</u> it

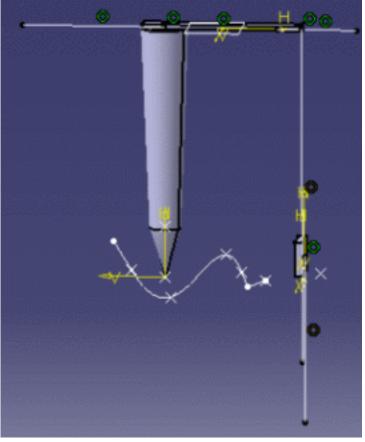
Select the curve either in the specification tree or in the geometry area. The 3D compass is automatically snapped onto the curve object



6. Drag the compass as shown below:



7. Detach the 3D compass:

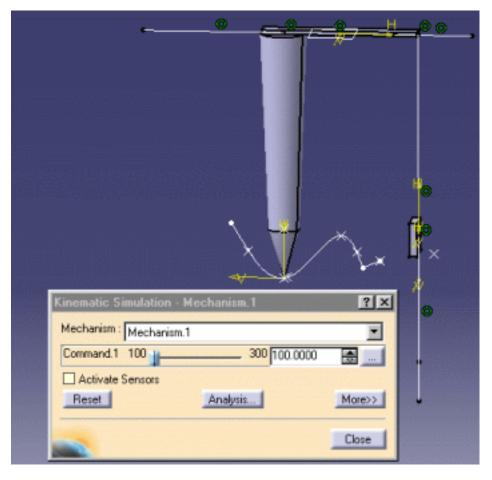


Click the Simulation With Commands again from the Simulation toolbar. The mechanism can no longer be simulated. A warning message can be displayed (it is not always the case as sometimes, DMU Kinematics Simulator performs an automatic update).

Warning	×
٩	Some joints of this mechanism reference constraints which are not verified. Do you want to perform an update before simulating ?
	<u>Yes</u> <u>N</u> o

- **9.** Click **Yes**. The Mechanism is updated automatically:
 - the parts involved in the mechanism are reassembled
 - the mechanism can be simulated

The Kinematic Simulation dialog box appears:



For more information, please refer to About Joints and Creating Mechanisms and Joints.



Editing Point Surface Joints

(modifying joints definition)

This task shows how to edit point surface joints in a V5 mechanism modifying the elements involved in the joint position

Open the PointSurface.CATProduct document.

Automatic switch to Design mode:

If you work with the cache system in visualization mode, you no longer need to use Edit->Representations->Design **Mode** beforehand as the switch to design mode is automatic (an eye appears as you point the product in the geometry or specification tree). All you need to do is click.

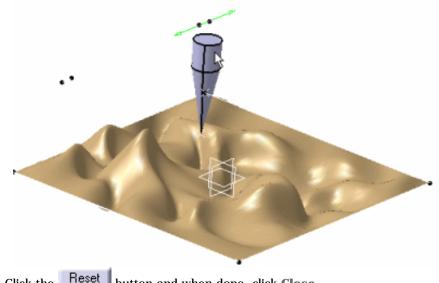
1. Check the mechanism can be simulated, for this: click the Simulation With Commands icon from the Simulation toolbar. The Kinematics Simulation dialog box appears:

Note: the state of the dialog box depends on your settings (expanded or collapsed)

The commands of the kinematics mechanism is available

Kinematic Simulation - Mechanism.1	? ×
Mechanism : Mechanism.1	•
Command.1 0 100 48.5624	
Command.2 0 100 73.9296	
Activate Sensors	
Reset Analysis	More>>)
	Close

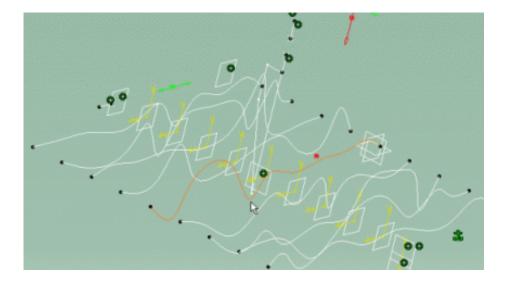
2. Run your simulation either using the sliders or the manipulators in the geometry area:



- Reset button and when done, click Close. 3. Click the
- 4. Modify the geometry: in our example, you are going to modify one line pertaining to the

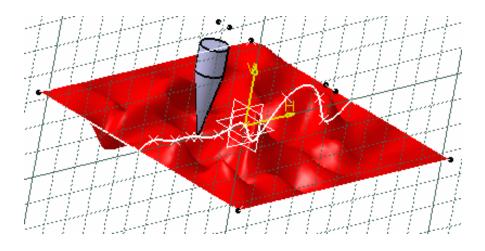
Surface_PointSurface.CATPart

- **5.** Click the **Swap visible space** icon to display hidden objects
- **6.** Double-click one curve as shown below:

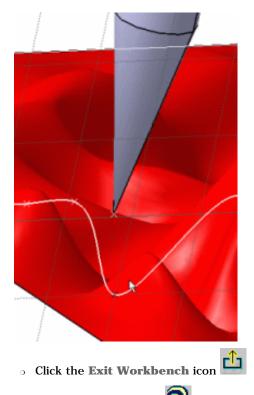


The Sketcher workbench is automatically displayed

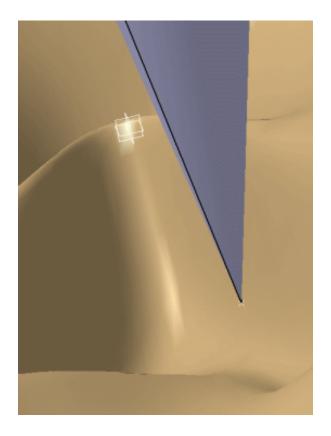
7. Click the **Hide/Show** icon first and then the **Swap visible space** icon to display the curve in the show space:



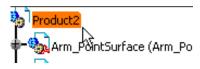
8. Modify the curve as shown below in the Sketcher workbench.



- Click the **Update** icon if necessary
- $_{\odot}\,$ The surface and pointer are disassembled: the mechanism can no longer be simulated.



9. Double-click Product.2 in the specification tree



Your are back in Kinematics Simulator workbench

- 10. (Optional) Click the Update positions icon S.
 Keep in mind that DMU Kinematics Simulator performs an automatic update.
 The Mechanism is updated automatically: the parts involved in the mechanism are reassembled
- 11. Simulate your mechanism again with the design changes. For this, all you need to do is click the Simulation

With Commands 999 from the Simulation toolbar.

12. Click **Close** when satisfied.



Replacing Slide Curve Joints Specifications

Open the Edit_SlideCurve.CATProduct document.

When you create joints, you can define the mechanism within the same dialog box. Remember though, that you create a mechanism independently from the joints by selecting **Insert**->**New Mechanism...** from the menu bar. Automatic switch to Design mode:

If you work with the cache system in visualization mode, you no longer need to use **Edit**->**Representations**->**Design Mode** beforehand as the switch to design mode is automatic (an eye appears as you point the product in the geometry or specification tree). All you need to do

۲

is click.

1. Check the mechanism can be simulated, for this: click the Simulation With Commands



from the Simulation toolbar

The Kinematic Simulation dialog box appears:

Note: the state of the dialog box depends on your settings (expanded or not)

The command of the kinematics mechanism is available as shown below.

Kinematic Simulation - Me	? ×	
Mechanism : Mechanism.1		•
Command.1 100	300 100.0000	.
Activate Sensors		
Reset	Analysis	More>>
		Close

- **2.** Run your simulation using the slider of the command.
- 3. Click the Reset button and when done, click Close.
- In the specification tree, double-click the joint to be modified.
 In our example, double-click Joint.1

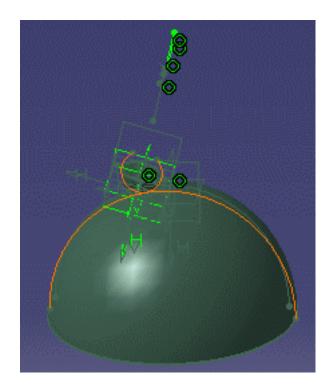
The joint Edition dialog box is displayed:

Joint Edition : Joint.1 (Slide Curve)								
Joint name : Joint.1								
	Joint geometry :							
Curve 1 : Sketch.1	Curve 2 : Sketch.1							
	OK Apply Cancel							

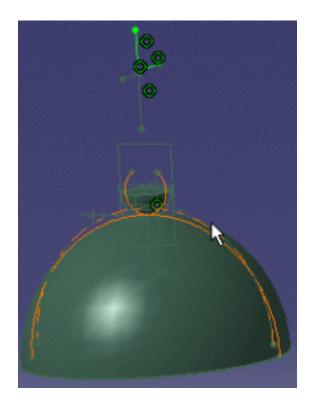
5. In the joint Edition dialog box, select the curve you want to be replaced

Joint Edition : Joint.1	(Slide Curve)	? ×
Joint name : Joint.1		
	Joint geometry :	
Curve 1 : Sketch.1	Curve 2 : Sketch.1	
		13
	OK Apply	Cancel
Country of		

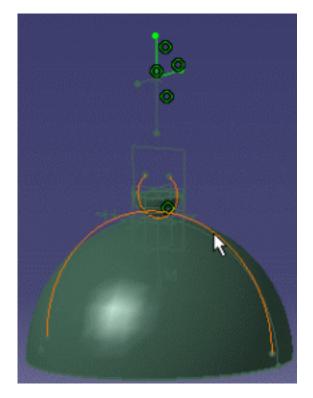
The geometry involved in the slide curve joint is low-lighted

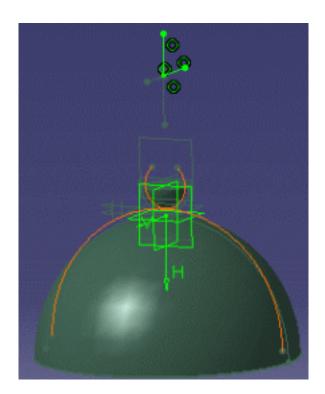


6. Select a new curve for your slide curve joint in the geometry area

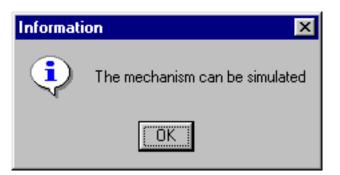


7. Click Apply to check the new joint. The mechanism parts are automatically reassembled





8. Click **OK** to confirm you operation. The mechanism can be simulated, an information message is displayed:



i For more information, please refer to About Joints and Creating Mechanisms and Joints.



Creating Slide Curve Joints

This task shows how to create slide curve joints in a V5 mechanism.

Open the SlideCurve_without_kin.CATProduct document.

When you create joints, you can define the mechanism within the same dialog box. Remember though, that you create a mechanism independently from the joints by selecting **Insert**->**New Mechanism...** from the menu bar.

- **1.** Click the arrow within the Revolute Joint icon from the DMU Kinematics toolbar (Revolute joint is the default joint type).
- 2. Undock the Kinematic Joints toolbar.

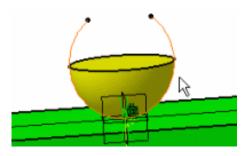
Kinema	tic jo	oints														×
] 😹	6	Q,	٩	Ŷ	R		낪	-tê	や	*	\$	\$	6 9	:	22	Ŀ
						12										

- **3.** Select the Slide Curve Joint icon The Joint Creation: Slide Curve dialog box is displayed
- 4. Click on New Mechanism. The Mechanism Creation dialog box is displayed:

Mechanism Creation	? ×	
Mechanism name : Mechanism.1		
🥌 💽 ок 🍳	Cancel	

In our example, keep the default name Mechanism.1.

5. Select Curve 1 either in the geometry area or in the specification tree. In our example, select the yellow sphere arc as shown below:



6. Select Curve 2 either in the geometry area or in the specification tree. In our example, select the line on the green part as shown below:

DMU Kinematics Simulator

	Joint Creation : Slide Curve	? ×
	Mechanism : Mechanism.1	New Mechanism
	Joint name : Slide Curve.1 Curren Curve 1 : Join.1	it selection : Curve 2 : Line.1
↓H		OK Cancel

7. Click Ok to end the slide curve joint creation

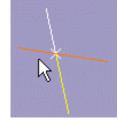
Now create a second joint (revolute joint) and a third joint (prismatic joint)

i) For more information, please refer to About Joints, Creating Revolute Joints and Creating Prismatic Joints.

Revolute.2:

- select line .1 (right white lines)
- select line.2 (left white lines)



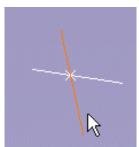


- select xy plane (green) (plane 1)
- select xy plane (white lines) (plane2)
- when done click OK

Prismatic.3

- o select line.1
- select line.2 as shown below:

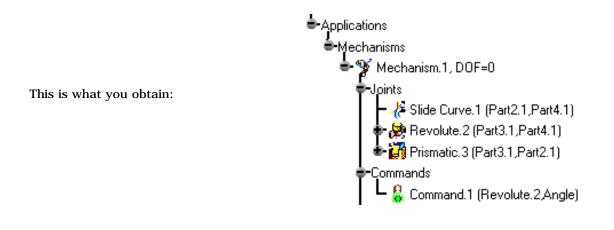




- select xy plane (white lines) (plane 1)
- select xy plane (yellow sphere) (plane2)
- when done click OK

- **8.** You forgot to assign the Command:
 - Double-click Revolute. 2 (revolute joint) in the specification tree
 - $_{\odot}~$ check the Driven Angle option in the Joint Edition : Joint. 2 dialog box displayed
 - When done, click **Ok**

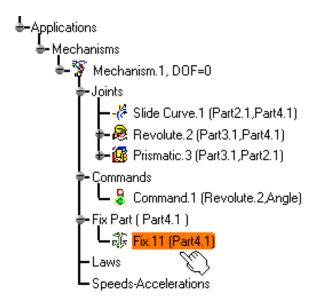
Joint Edition : Revolute.2 (Revolute)		? ×
Joint name : Revolute.2		
	eometry :	
Line 1 : Sketch.1	Line 2 : Line.2	
Plane 1 : xy plane	Plane 2 : xy plane	
🧧 Angle driven		
Joint Limi		
Joint Limi	Upper Limit : 360deg	Ð
	🕒 OK 🧕 Can	cel



9. Click the Fixed Part icon from the Simulation toolbar or select **Insert**->**Fixed Part**... from the menu bar. The New Fixed Part dialog box is displayed.

New Fixed Part	? ×
Mechanism : Mechanism.1	▼ New Mechanism
	Cancel

10. Select the Fixed Part either in the geometry area or in the specification tree. Here, select the Green sphere. The specification tree is updated.



The mechanism can be simulated

11. Double-click Mechanism. 1 in the specification tree

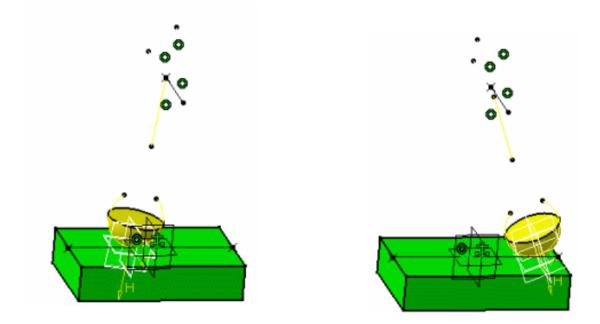
The Kinematic Simulation - Mechanism 1 dialog box is automatically displayed.

i **Note:** if there are laws defined in the mechanism, the simulation with laws functionality will be launched automatically.

Kinematic Simulation - Mechanism.1	? ×
Mechanism : Mechanism.1	•
Command.1 -360 360 0	.0000
Activate Sensors	
Reset Analysis	More>>
	Close

Manipulate the slider of the command





Open the SlideCurve_ with_kin.CATProduct to check your result.

It is impossible to create slide curve joints if the parts involving the joint are not well positioned. For more detailed information, please refer to Tips for Curve or Surface Joints Creation



Tips for Curve or Surface Joints Creation

It is impossible to create the four new kinematics joints (Point curve , Roll curves, Slide curves, Point surface) if the parts involving the joint are not well positioned.

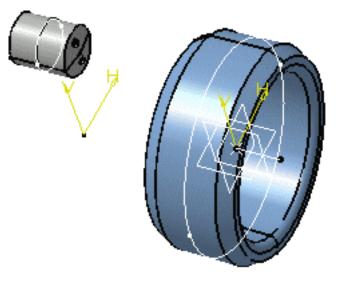
No sample document is provided.

Tips: to position 2 parts involving a Roll Curve joint: create a point and a line tangent to curve on this point for each part. In the Assembly Design Workbench, snap the two points and the two lines.

in ENOVIA context: If the parts involved in the joint are not well positioned, create the mechanism on positioned assemblies.

Remember: to create **Roll Curve and Slide curve joints**, the two curves must be coincident and tangent in one point.

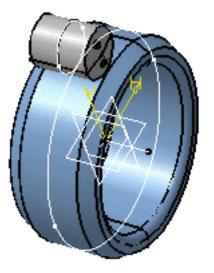
Scenario 1: Create a Roll curve joint



- Select the Roll Curve joint icon from the Kinematics Joints toolbar The Joint creation: Roll curve dialog box is displayed
- You cannot select the Curve 2 in the geometry area. It is impossible to create the roll curve joint because the parts are not well positioned:

Joint Creation : Roll Curve
Mechanism : Mechanism.1
Joint name : Roll Curve.1
Current selection :
Curve 1 : Inner ring/Sketch.2 Curve 2 :
Length Driven
OK Gancel

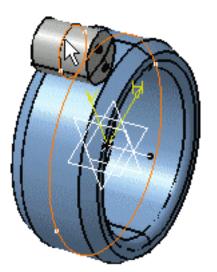
- **3.** Click **Cancel** to exit the Joint creation functionality.
- 4. Select Digital Mockup->DMU Navigator from the Start menu.
- **5.** Reposition the parts using the Snap command. This is what you obtain:



6. Select the Roll Curve joint icon from the Kinematics Joints toolbar.

The Joint creation: Roll curve dialog box is displayed. This time you can create the roll

curve joint: the parts are correctly positioned.



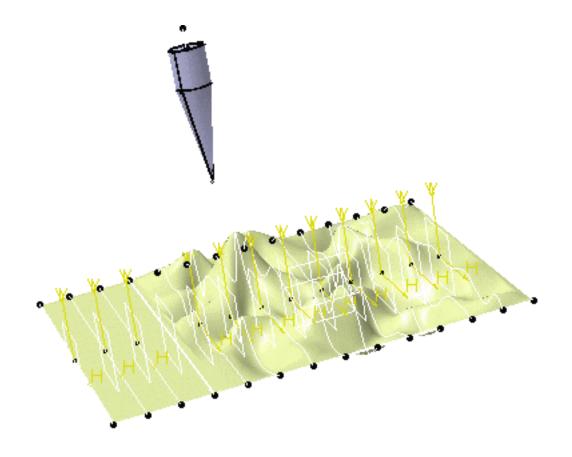
<u>? ×</u>
New Mechanism
selection :
Curve 2 : Roller/Sketch.1
OK Gancel

7. Click **Ok** to end the joint creation



Remember: to create **Point surface and Point curve joints**, the point must be on the curve.

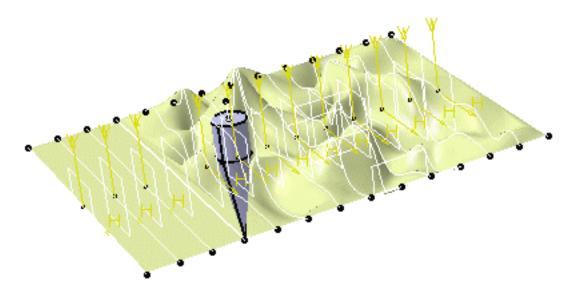
Scenario 2: Create a Point surface joint



- Select the Point Surface joint icon from the Kinematics Joints toolbar The Joint creation: Point Surface dialog box is displayed:
- You cannot select Point 1 in the geometry area. It is impossible to create the point surface joint because the parts are not well positioned:

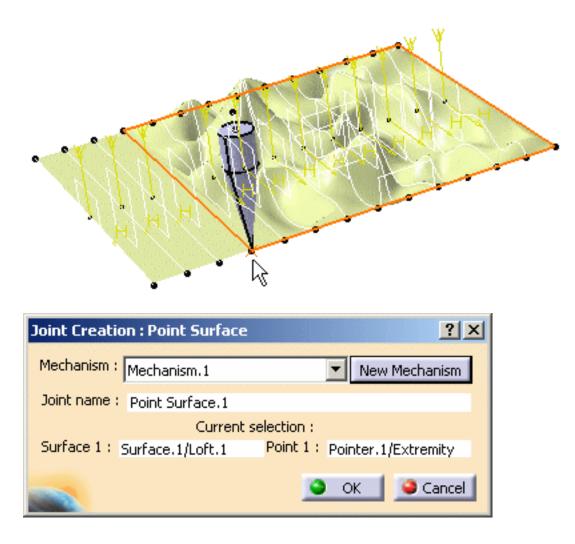
Joint Creation : Point Surface
Mechanism : Mechanism.1
Joint name : Point Surface.1
Current selection : Surface 1 : Surface.1/Loft.1 Point 1 :

- 3. Click Cancel to exit the Joint creation command
- 4. Select Digital Mockup->DMU Navigator from the Start menu
- **5.** Reposition the parts using the Snap command. This is what you obtain:



6. Select the Point surface joint icon from the Kinematics Joints toolbar.

The Joint creation: Point surface dialog box is displayed. This time you can create the point surface joint because the parts are correctly positioned.



7. Click **OK** to end the joint creation.

Fixed Parts and Commands



Defining a Fixed Part Defining Commands

Defining a Fixed Part

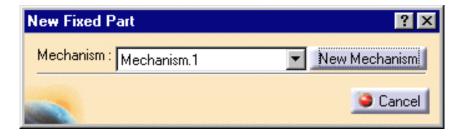
This task will show you how to define a Fixed part.



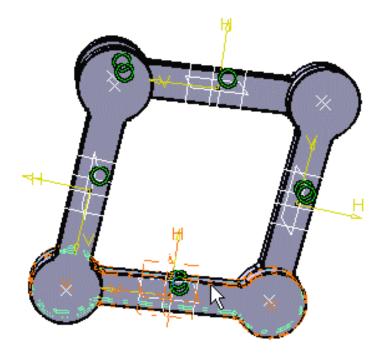
Open the rods+4joints+cmd.CATProduct document.



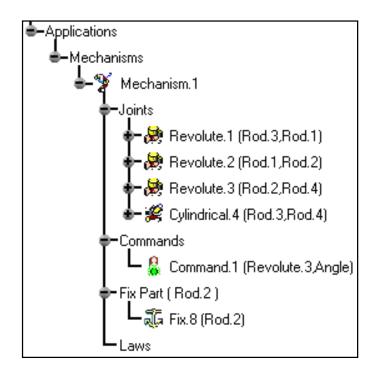
 Click the Fixed Part icon from the Simulation toolbar or select Insert->Fixed Part... from the menu bar. The New Fixed Part dialog box is displayed.



2. Select the Fixed Part either in the geometry area or in the specification tree.



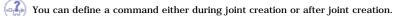
3. The fixed Part is automatically defined. The Fixed part is identified in the specification tree.



 $\ref{eq: constraint}$ Keep in mind you can apply the Undo command $\ref{eq: constraint}$ to modify your selection.



Defining Commands



This task shows how to define a command on a cylindrical joint during its creation.

Open the rods+4joints.CATProduct document. You created a mechanism.

1. Double-click Cylindrical. 4 in the specification tree

Applications
Mechanisms
🚔 뛝 Mechanism.1, DOF=1
🗣 Jpints
🖶 👰 Revolute.1 (Rod.3,Rod.1)
💠 🕀 Revolute.2 (Rod.1,Rod.2)
🖶 👰 Revolute.3 (Rod.2,Rod.4)
🖶 🥰 Cylindrical. 4 (Rod. 3, Rod. 4)
-Commands 🖏
-Laws

The joint Edition dialog box is displayed

i) For more information about commands, see About Joints

2. Select the Angle driven checkbox.

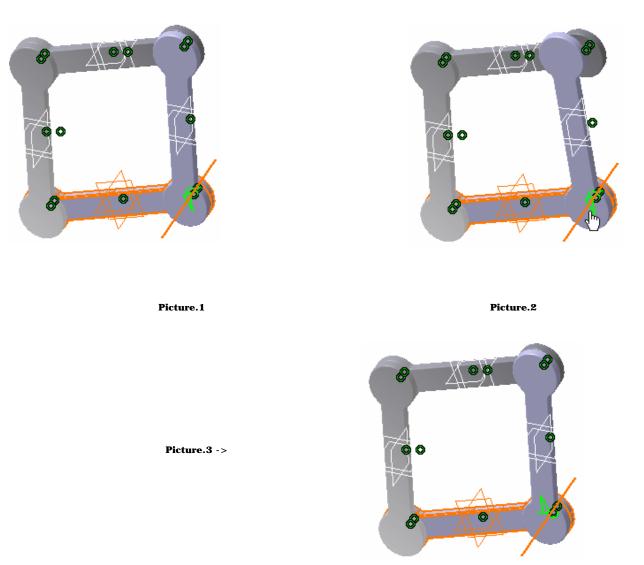
Joint Edition : Cylindrica	al.4 (Cylindrical)	? ×
Joint name : Cylindrical.4		
1.	Joint geometry :	
Line 1 : Axis	Line 2 : Axis	
🏹 Angle driven	Length Driven	
Joint Likys		
🔲 1st Lower Limit : 🛛 🛄	set 📑 📄 1st Upper Limit :	unset 🚍
🖾 2nd Lower Limit : 📑	60deg 📑 🖾 2nd Upper Limit :	360deg
		OK Scancel

You can now check the command positive orientation and change it if necessary (either at joint or command edition)

Note though, for Roll curve and Point curve joints you can only check the command orientation but not change it.

The command orientation is defined by a green arrow in the geometry area and in this example (command assigned to a cylindrical), you can change its orientation:

- The parts which are not involved in the joint creation are displayed in low light (to easily locate the joint you are working on): Picture.1
- Pass the cursor over the green arrow to launch a short animation **Picture.2**
- Click the arrow to reverse the command orientation if necessary: Picture.3



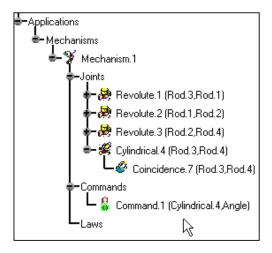
The positive orientation of a command does not indicate an absolute movement of the parts involved (in the joint on which is assigned the command) but the intrinsic movement of the second part with respect to the first part involved in the joint.

3. Click Ok to confirm your operation.

 (Δ)

i

The command is identified in the specification tree.



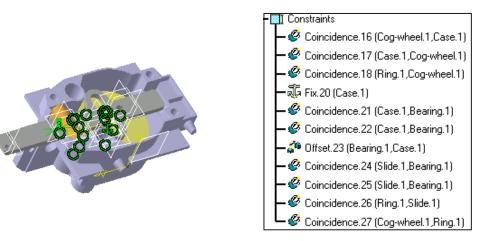
You can also create the command while creating a joint.

Version 5 Release 13

Converting Constraints into Joints (Beginner's Mode)

This task shows you how to convert Assembly constraints into V5 joints.

Open the jigsaw_with_constraints.CATProduct document. The constraints are visible both in the geometry area and in the specification tree



If you work with the Cache System, please make sure you are in Design mode (select Edit->Representations->Design Mode.). for more detailed information, please refer to the DMU Navigator user's Guide - Task: Viewing the Cache Content

1. Make sure you are in Design Mode (Edit->Representations->Design Mode).

2. Click the Assembly Constraints conversion icon icon from the DMU Kinematics toolbar. The Assembly Constraints Conversion dialog box appears:

Assembly Constraints Conversion	? ×
Mechanism :	New Mechanism
Auto Create	More >>
	Unresolved Pairs :
	🕒 OK 📔 🥯 Cancel

- 3. Click on the New mechanism button. When done click OK.
- **4.** Click on the Auto Create button to launch the operation.

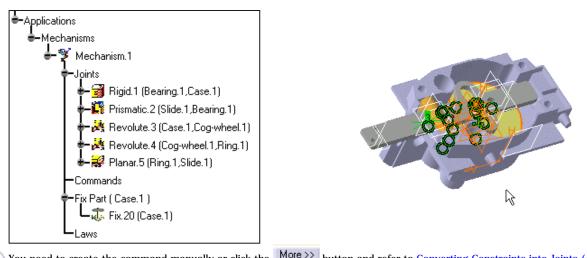
You can see that there are 5 unresolved pairs of products

Assembly Constraints Conversion		? X
Mechanism : Mechanism.1	New Mechar	nism
Aute Create		More >>
-1/2	Unresolved Pairs :	575
	ок ј	Cancel

The constraints are converted into V5 joints. The 5 joints are identified in the specification tree and highlighted in the geometry area

DMU Kinematics Simulator

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- You need to create the command manually or click the More >>> button and refer to Converting Constraints into Joints (Advanced Mode)
- 5. Click Ok to confirm your operation. Now, let's create the command
- 6. Double-click Revolute. 4 in the specification tree. The Edit Joint Edition dialog box is displayed

Joint Edition : Revolute.4 (Revolute)	? ×
Joint name : Revolute.4	
Joint g	jeometry :
Line 1 : Axis	Line 2 : Axis
Plane 1 : Face	Plane 2 : Face
🖾 Angle driven	
Joint Limits	
🖾 Lower Limit : 🔤 -360deg	Upper Limit : 360deg
	OK Scancel

7. Select the 'Angle driven' checkbox and then OK to create the command. The command is created and identified in the specification tree

-Applications	
-Mechanisms	
🖨 🏋 Mechanism. 1	
≑− Jpints	
🗰 🙀 Rigid.1 (Bearing.1,Case.1)	
🖝 🅼 Prismatic.2 (Slide.1,Bearing.1)	
🖝 🙈 Revolute.3 (Case.1,Cog-wheel.1)	
💠 👰 Revolute.4 (Cog-wheel.1,Ring.1)	
- 6 Coincidence.27 (Cog-wheel.1,Ring.1)	
Coincidence.18 (Ring.1,Cog-wheel.1)	
🖆 😹 Planar.5 (Ring.1,Slide.1)	
🛉 - Commands	
🗕 🌡 Command.1 (Revolute.4,Angle)	
Fix Part (Case.1)	
「「「Fix.20 (Case.1)」	
Laws	
(i) An information message is displayed, your mechan	ism can now be simulated
Information 🔀	
The machine and he similar d	
The mechanism can be simulated	
OK	



Using the Update Command



This task shows you how to use the Update Positions command \bigotimes a very powerful tool which lets you keep the Assembly workbench and the Kinematics Simulator workbench synchronized. It means the modifications done are taken into account and the joints or constraints are respectively updated.

Note the synchronization between Assembly and Kinematics workbenches is relevant and complete only for joints with constraints.

Please also read Replacing Curve Joint Specifications and Editing Curve Joint Specifications

in the *"Designing Higher Pair Joints "* section.

A new capability for the Update command is provided:

It replaces the previous Sub-Mechanism import dialog box. When you need to re-import a mechanism, just click on the "Update Positions" icon, then select the imported mechanism you want to re-import as shown below

Update Mec	hanism	? ×
Mechanism :	SUB_PRODUCT_MECHANISM.2\Mechanism.1	-
Take curr	SUB_PRODUCT_MECHANISM.2\Mechanism.1 SUB_PRODUCT_MECHANISM.1\Mechanism.1	
	la construction de la construction	ancel

Note: the behavior of this Update Positions command (described in the step-by-step scenario below) remains the same with non-imported mechanisms.

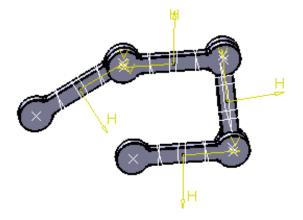
For more detailed information on how to use the Import and Update commands, read Visualizing and Simulating in Subproducts

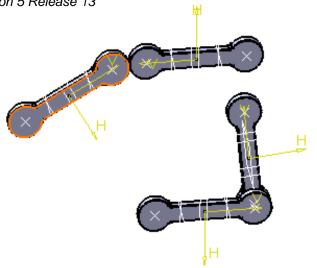
What is taken into account ?

- moving parts in the geometry area
- · deleting or modifying assembly constraints
- editing curve joints specifications

Open the rods_with_joints.CATProduct document.

- 1. Move the Rod.2. and Rod.1 for this:
 - Point to the compass manipulation handle
 - $_{\odot}~$ Drag and drop the compass onto the rod.4 in the geometry area
 - Move the rod.2.
- 2. Reposition the 3D compass as it was.





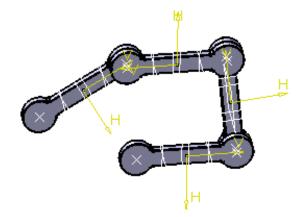


The Update Mechanism dialog box is displayed:

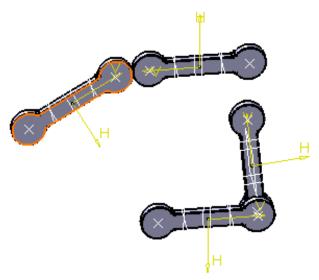
The 'Take current positions for rigid joints' option lets you take into account the new position.

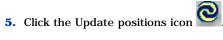
Update Mechanism 🔗 🗙		
Mechanism :	Mechanism.1	•
🗌 Take curr	ent postions fo	or Rigid Joints
	ок	Cancel

4. Click Ok to confirm your operation. The mechanism is updated and the part is back to its initial position.

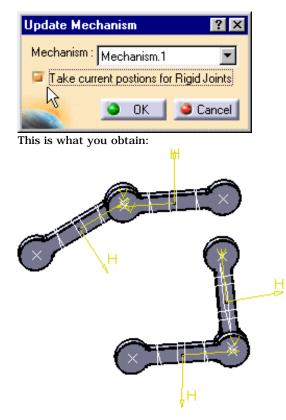


Now, move Rod.2 and Rod.1 in the same way.

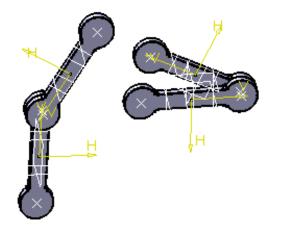




6. Select the Take current positions for rigid joints checkbox.



Now simulate the mechanism. Please refer to Simulating With Commands.



Kinematic Si	mulation - Me	echanism.1	? ×
Mechanism :	Mechanism.1		-
Command.1	-360 ,	360 57.6000	.
Command.2	-360 ,	360 0.0000	.
Activate S	ensors		
Reset		Analysis	More>>
			Close

The current position has been kept for Rigid.2 (Rod.4, Rod.1) *Now delete an assembly constraint.*

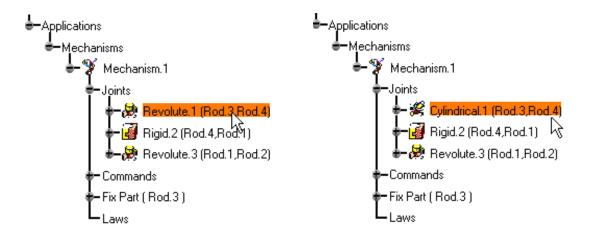
- **7.** If you need information about this particular constraint: double-click Coincidence.2 (Rod.3, Rod.4) in the specification tree to display the Constraint Definition dialog box.
- 8. Right-click Coincidence.2 (Rod.3, Rod.4) in the specification tree.
- 9. Select **Delete** from the contextual menu displayed.

10. Click the Update positions icon **C**. The Update Mechanism dialog box appears.

Update Mechanism 🛛 ? 🗙		
Mechanism :	Mechanism.1	•
Take current postions for Rigid Joints		
	🗿 ОК	Cancel

11. Click **OK**. The joints within the mechanism are updated.

Revolute.1 (Rod.3, Rod.4) is converted into a Cylindrical joint (Cylindrical.1) as shown below:





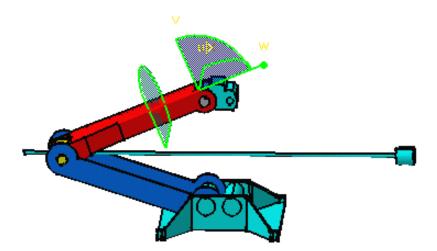
Moving Constrained Components Using the Compass



This task consists in manipulating the components in a V5 mechanism to check if the components react the way we want.



 Select the compass manipulation handle and drag it onto CRIC_BRANCH_1. For details about how to use the compass, please refer to *Infrastructure User's Guide Version 5.* As the compass is snapped to the component, you can manipulate the component.



2. Now, if you press and hold down the Shift key, select v/z axis on the compass, then drag and drop the component up and down, you can see that three components are moving. This is an example of what you can get:



- Repeat the operation as many times as you wish. The product reacts correctly.
 CRIC_FRAME does not move because it is fixed. The other three components can move.
- **4.** Release the left mouse button before releasing the Shift key.
- **5.** Drag the compass away from the selected object and drop it.



Running Simulations

DMU Kinematics Simulator provides easy methods to run kinematics simulations and detect collisions during simulations. See: Mechanism Analysis in the *Advanced Tasks section*

> Simulating With Laws Simulating With Commands Simulating On Request Leaving Simulation in Modified Position Simulating After Having Moved Constrained Components

By default the **new position is kept** when exiting the simulation commands. To restore the initial product position and before leaving the simulation commands \mathbf{I}

you need to click:

- Reset (Simulation With Commands)
- (Simulation With Laws)

To restore the initial product position when you already quit the simulation commands:

you need to click the Reset Positions icon

Please read Resetting a V5 Mechanism

Manipulator symbols are displayed for either translating or rotating the mechanism whenever its joints have associated commands.

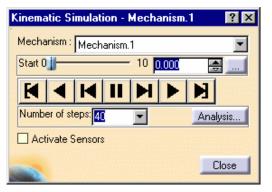
- For a joint with a linear command, a linear manipulator symbol is displayed. To translate the mechanism just drag it using the left mouse button.
- For a joint with an angular command, a circular manipulator symbol is displayed. To rotate the mechanism just drag it using the left mouse button.
- For a joint with linear and angular commands, a linear manipulator symbol is displayed. To translate the mechanism just drag it using the left mouse button.
- To access the circular manipulator for rotating the mechanism you must use the left and middle mouse buttons together and drag as before.

Simulating With Laws

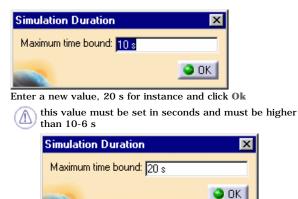
This task will show you how to run a kinematics simulation with laws that are already defined on the mechanism.

Open the Jack.CATProduct document

1. Click the Simulation With Laws icon 🔯 in the DMU Kinematics toolbar



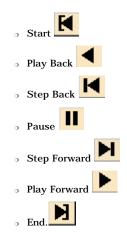
2. Change the range of Kinematics time parameter directly in this dialog box using the Edit Time range button . The Simulation Duration dialog box is displayed



The maximum time bound value is automatically updated in the dialog box:

Kinematic Simulation - Mechanism.1 💦 🔀							
Mecha	nism :	Mech	anism	.1			•
Start 0				20	0.000		
K	◀	M	II	M			
Numbe	er of st	eps: 🜆		-		Ana	lysis
Acti	ivate S	ensors	1				
			1				lose
			ļ				
ſ	New	time	valu	е			

3. Set the desired Number of steps, then run the simulation using the Simulation buttons:



The kinematics mechanism moves according to the pre-defined laws.

You can switch between any of the simulation modes at any time.

U You can also enter a time value to visualize the position of the mechanism at that time.

 Click Analysis if you need to detect interferences or distances while simulating (you need to create interference or distance objects first) The Edit Analysis dialog is displayed.

Edit Analysis	? ×
No Name	
Brov	Wse,
	Close

5. Select the interference if you defined one and set the combo to on.

 $m{i}$) For more details, please refer to Detecting Interferences and Detecting Distances.

- 6. Run the simulation.
- 7. If you check the Activate Sensors option, the Sensors dialog box is automatically displayed.

This functionality lets you retrieve detailed information during simulation operations (With laws and With commands) about:

- joint values (with commands or not)
- measure values (see Measure distances and angles between geometrical entities and points)
- joint limits if previously defined
- speed and accelerations
- V5 Mechanisms

Note: the Check Limits option is available only in the Sensors dialog box accessed with the Activate Sensors check button. Set the required mode (on, stop) using the radio buttons. For detailed information, read Using Sensors V4 Mechanisms

Note: The Check Limits option is still available through the Kinematics simulation commands (with laws, with commands). Please read Checking Joint Limits

Sensor		Unit	Observed 4
Mechanism.1\Joints\Cylindrical.2	2\Length	Millimeter	No
Mechanism.1\Joints\Cylindrical.2	2\Angle	Degree	No
Mechanism.1\Joints\Revolute.3		Degree	No
Mechanism.1\Joints\Revolute.4		Degree	No
Mechanism.1\Joints\Cylindrical.5		Millimeter	Yes
Mechanism.1\Joints\Cylindrical.5		Degree	No
Mechanism. 1\Joints\Prismatic. 6\		Milimeter Milimeter	No
Mechanism.1\Joints\Cylindrical.7	/ (Length	minimeter	No
Deselect Al		Select	All
Display Options			
All		O Limited	1
Detect Clashes	Check Limits		
Jetett Clasnes		0.0	O Stop
	Off	O On	- nech
Automatic O Interferences	 Off Outputs 	O On	

8. Click Close to confirm your operation

By default the **new position is kept** when exiting the simulation commands. Before leaving the simulation command to go back to the initial position: you need to click:

。 Reset (Simulation With Commands)



。 [Simulation With Laws]

For more detailed information, see Leaving Simulation in Modified Position and Simulating After Having Moved Constrained Components

Notice that you cannot record simulations within the Simulation With Laws functionality. If you need to record such a simulation or several simulations, please refer to Recording Positions.



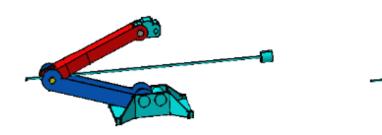
Version 5 Release 13

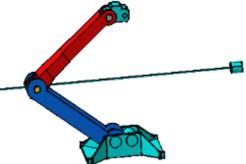
Simulating With Commands

•	This task will s	how you how to run a kinema	atics simulation with c	commands.	
D	Open the Jack.CATProduct document				
	In our sample document, there is only one mechanism. If you are working with a product containing more than one mechanism, it is strongly recommended to select the mechanism you need before starting the simulation with commands.				
	1. Click th	e Simulation With Commands	s icon 🗐. The Kiner	maticS Simula	ation dialog box appears:
	Γ	Note: the state of the dialog l	box depends on your	settings (exp	anded or not)
		Kinematic Simulation - Mec	chanism.1	? ×	
		Mechanism : Mechanism.1	100 0 0000		
	<i>(i)</i>	Command.1 -100 ,	100 0.0000	±	
	\bullet	Activate Sensors			
		Reset	Analysis	More>>	

The command of the kinematics mechanism is available.

2. Manipulate the slider of the command.





The corresponding part of the kinematics mechanism moves accordingly.

i Note that if you click the <u>More</u> *i* button, the Kinematics Simulation dialog box expands. The **immediate** option is set by default. For more information about the **On request** option, please refer to <u>Simulating on Request</u>

Close

Kinematic Simulation - Mechanism.1	? ×
Mechanism : Mechanism.1	•
Command.1 -100 100 0.0000	.
Activate Sensors	
Reset Analysis	(< <less)< td=""></less)<>
Simulation	
Immediate O On request	
Number of steps: 40	
	Close

- i) You can use the slider, enter a value or manipulate the geometry directly to achieve the same result.
- 3. If you set joint limits, use the Activate Sensors check button. In the Sensors dialog box, activate the Check limits option and run

your simulation again. Notice the information about limits will appear in the comments field from the History tab.

For more information, please refer to Using Sensors

D Note: if you are working with a V4 mechanism, the Check Joint Limits button is available in the Kinematics Simulation dialog box, read Checking V4 Mechanism Joint Limits and Simulating With Commands

You can set a command value directly in the spin box. You can also set lowest and highest values for the range of a command by clicking on the button opposite the command and entering values in the displayed pop-up. Click $\mathbf{O}\mathbf{k}$ when done

Slider : Command.	1 🔀
Lowest value:	-100
Highest value:	100
Spin box increments:	5
	<u>Э ок</u>

4. Click Close to confirm your operation.

By default the **new position is kept** when exiting the simulation commands (Simulation With Commands and Simulating With Laws)

You need to click :

ĺ

- 。 Reset (Simulation With Commands context)
- (Simulation With Laws)

For more detailed information, see Leaving Simulation in Modified Position and Simulating After Having Moved Constrained Components

You cannot record your simulation within the Simulation With Commands command. You can record simulations within the Simulation command (please refer to Recording Positions).



1

Simulating On Request

This task shows how to perform a simulation on request.

Open the Jack.CATProduct document

In our sample document, there is only one mechanism. If you work with a product containing more than one mechanism, it is strongly recommended to select the mechanism you need before starting the simulation with commands.

- **1.** Click the Simulation With Commands icon **2**. The Kinematics Simulation dialog box is displayed.
- 2. Click More>>>. The command of the kinematics mechanism is available as shown below.

Kinematic Simulation - Mechanism.1	? ×
Mechanism : Mechanism.1	•
Command.1 -100 100 0.0000	.
Activate Sensors	
Reset Analysis	< <less< td=""></less<>
🥥 Immediate 🔿 On request	
Number of steps: 40 🚽	
	Close

By default, the **Immediate** option is set

- **3.** Activate the **On Request** option.
- **4.** Enter a precise value for the command. For instance 20.
- **5.** Enter the number of steps you need, 20 for example.

Kinematic Simulation - Mechanism.1	? ×
Mechanism : Mechanism.1	•
Command.1 -100 100 20.0000	.
Activate Sensors	
Reset Analysis	< <less< td=""></less<>
Simulation	
🔿 Immediate 🥥 On request	
Number of steps: 20	
	Close

- **6.** Click **Play Forward** The corresponding parts of the kinematics mechanism move accordingly at each step.
- 7. Click Close to confirm your operation

You need to click Reset before leaving the simulation with commands to go back to the initial position.

By default the **new position is kept** when exiting the simulation commands (Simulation With Commands and Simulating With Laws) (keep in mind, in Simulation With Laws you need to click the **Start** button to jump to the initial position)

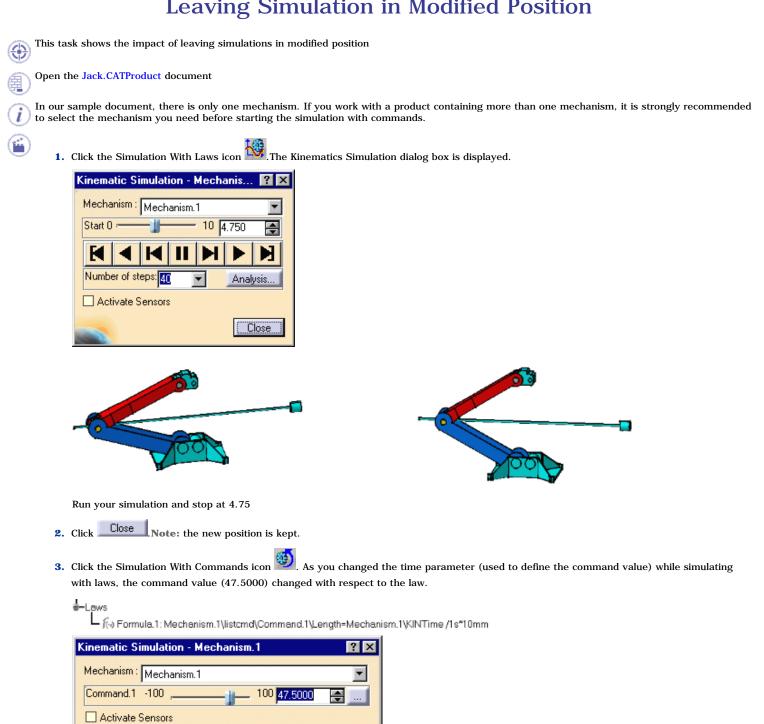
For more detailed information, see Leaving Simulation in Modified Position and Simulating After Having Moved Constrained Components

If there are commands, change at least one command value. You can modify the values of one or more commands for each motion.



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Leaving Simulation in Modified Position



4. Move the slider up to the end.

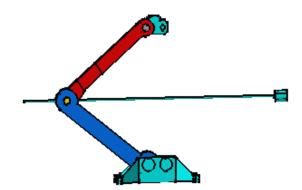
Reset

5. Exit the Simulation With commands. Click **Close**. The position is kept as shown below:

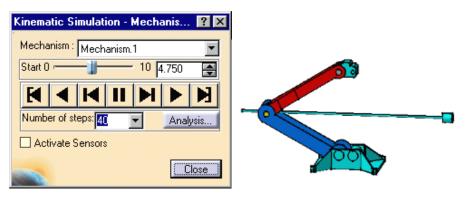
More>>

Close

Analysis.



6. Click the Simulation With Laws icon There is a jump to the command value corresponding to the last time parameter which is 4.750 The command value is automatically recalculated with respect to the law and the time parameter



- 7. Run your simulation again
- 8. Click Close when satisfied



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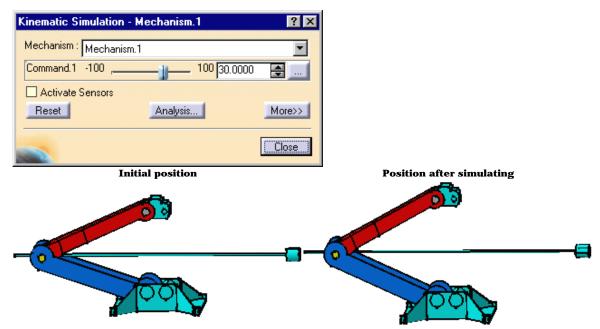
Moving Constrained Components in Simulation With Commands Context

This task shows the impact of moving constrained components in simulation with commands context

Open the Jack.CATProduct document

In our sample document, there is only one mechanism. If you work with a product containing more than one mechanism, it is strongly recommended to select the mechanism you need before starting the simulation with commands.

- 1. Click the Simulation With Commands icon ³⁰. Th
 - With Commands icon ²²². The Kinematics Simulation dialog box is displayed.
- 2. Run your simulation moving the command slider (i.e. until value 30.0000)

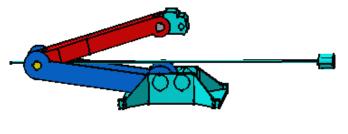


3. Exit the Simulation With Commands command clicking the Close button.

Vou need to click Reset before leaving the simulation command to go back to the initial position By default the new position is kept when exiting the simulation commands (Simulation With Commands and Simulating With Laws)

- 4. Now, use the compass and the Shift key to move CRIC_BRANCH_1.1. For more detailed information, please refer to Moving Constrained Components Using the Compass
- 5. Select the compass manipulation handle and drag it onto CRIC_BRANCH_1.1

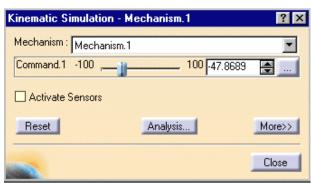
Now, if you press and hold down the Shift key, select v/z axis on the compass, then drag and drop the component up and down, you can see that three components are moving. This is an example of what you can get:



6. Now, click the Simulation With Commands icon 🥮 again.

Note: the last command value was: 30.000 The command value is automatically calculated with respect to this new position. The command value becomes in our example: -47.8689

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- 7. Run your simulation again.
- 8. Click Close.



Advanced Tasks

DMU Kinematics Simulator provides easy methods to detect and analyze collisions and distances between products. It also provides the capacity of generating a swept volume. The DMU Space Analysis Version 5 product must be installed before using these functionalities.



Mechanism Design Mechanism Analysis Digital Mockup Review Working with ENOVIA LCA

Mechanism Design

Creating Revolute Joints with Offset (Advanced Mode) Creating Revolute Joints (Centered Option) Defining Laws in a V5 Mechanism Converting Constraints into Joints (Advanced Mode) Trace Setting Joint Limits ÷

Creating Revolute Joints With Offset (Advanced Mode)

This task shows how to create offset revolute joints or centered revolute joints

Open the Create_Revolute.CATProduct document.

Automatic switch to Design mode:

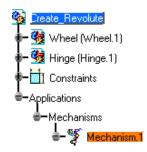
If you work with the cache system in visualization mode, you no longer need to use **Edit->Representations->Design Mode** beforehand as the switch to design mode is automatic (an eye appears as you point the product in the geometry or specification tree). All you need to do is click on the object.

- 🔊
- 1. Click the **Revolute Joint** icon icon from the Kinematic joints toolbar or select **Insert** -> **New Joint** -> **Revolute** from the Menu bar. The Joint Creation: Revolute dialog box appears.
- 2. Click New Mechanism. The Mechanism Creation dialog box is displayed:
 - Enter the name of your choice for the mechanism.
 - Click **Ok** when done



	Joint Creation : Revolute 🔗 🔀
In our example, keep the default name Mechanism.1.	Mechanism : Mechanism.1 Vew Mechanism Joint name : Revolute.1
	Current selection : Line 1 : Line 2 : Plane 1 : Plane 2 : Null Offset O Offset = Om 🚍
	Plane 3 : Plane 4 : Centered

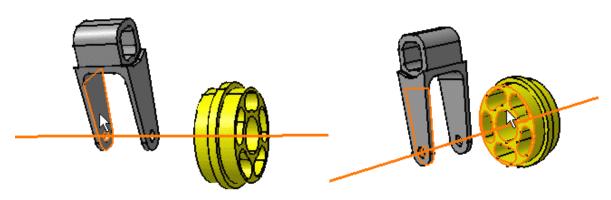
The Mechanism is identified in the specification tree



3. Select the lines in the geometry area:

- Line 1 = hinge axis
- Line 2 = wheel axis
- 4. Select the planes in the geometry area:

- Plane 1 =left inner hinge plane
- Plane 2 = left wheel axis surface



5. Activate the Offset option.

Select the offset value for this, three methods are available:

- enter the required value in the offset field
- use the scrollbar

1

o right-click the field and select the measure item from the contextual menu displayed

In our example keep the default value.

If you perform a right-click in the offset value field, a contextual menu lets you select between two items: measure or change step

Select the measure item: the Measure Between dialog box and measure Tools toolbar appear

Measure Between	? ×
Definition	
Selection 1 mode: Any geometry	
Selection 2 mode: Any geometry	
Other Axis : No selection	
Calculation mode: Exact else approximate	
Results	
Calculation mode:	
Selection 1: Selection 2:	
Minimum distance:	
Angle:	
Customize	e
	ncel

Select two entities. Keep this measure as offset value. A warning message lets you copy the measure you defined.

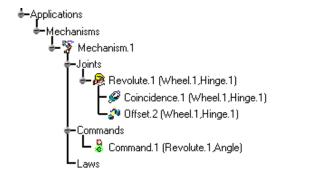


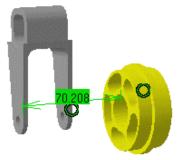
You can also change the step using Change step->new one

i For more information, see Specifying a Parameter Value as a Measure in the Knowledge Advisor User's Guide and Measuring Minimum Distances & Angles between Geometrical Entities and Points in the Space Analysis User's Guide

- 6. Assign the Angle driven command if needed.
- 7. Click Ok to end the Revolute Joint creation.

Joint Creation : Revolute	<u>?</u> ×
Mechanism : Mechanism.1	New Mechanism
Joint name : Revolute.1	
Current se Line 1 : Hinge.1/Solid.1 Line 2 : Wheel.1/Sol Plane 1 : Hinge.1/Solid.1 Plane 2 : Wheel.1/Sol Plane 3 : - Plane 4 : -	lid.1
🖾 Angle driven	
	OK Cancel
The specification tree is updated.	





Open the Create_Revolute_Offset.CATProduct to check your result.

Creating Revolute Joints (Centered Option)

This task shows how to create offset revolute joints or centered revolute joints

Open the Create_Revolute.CATProduct document.

Automatic switch to Design mode:

If you work with the cache system in visualization mode, you no longer need to use Edit->Representations->Design Mode beforehand as the switch to design mode is automatic (an eye appears as you point the product in the geometry or specification tree). All you need to do is click on the object.



1. Click the Revolute Joint icon kinematic joints toolbar or select Insert -> New Joint -> Revolute from the Menu bar. The Joint Creation: Revolute dialog box appears:

Mechanism Creation	? ×
Mechanism name : Mechani	sm.1
🔍 ОК	Cancel

Note: this dialog box lets you enter the name of your choice for the mechanism and click OK.

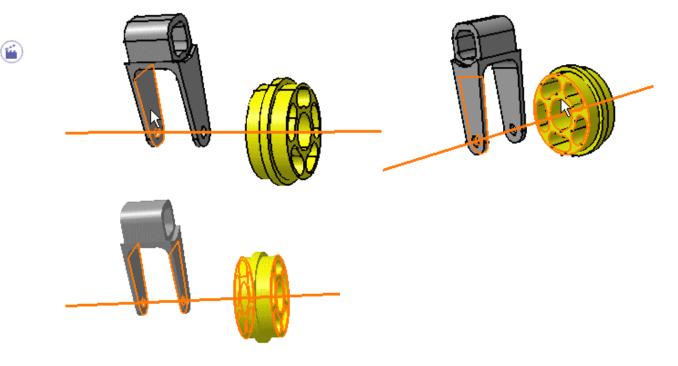
2. Click New Mechanism. The Mechanism Creation dialog box is displayed:

In our example, keep the default name Mechanism.1.

Mechanism : Mechanism.1 New Mechanism Joint name : Revolute.1 Current selection : Line 1 : Line 2 : Plane 1 : Plane 2 : Null Offset O Offset = Ime
Current selection : Line 1 : Line 2 :
Line 1 : Line 2 :
Plane 3 : Figure 4 : Figure 6 Centered
Angle driven
The Mechanism is identified in the specification tree.
Create_Revolute
🛊 📆 Wheel (Wheel.1)
🛊 – 📆 Hinge (Hinge.1)
♥─☐1 Constraints
Applications
Mechanisms
🖶 🌮 Mechanism.1

- 3. Activate the Centered option.
- 4. Select the lines in the geometry area:
 - Line 1 = hinge axis
 - Line 2 = wheel axis

- \circ Plane 1 = left inner hinge plane
- Plane 2 = left wheel axis plane
- Plane 3 = right inner hinge plane
- Plane 4 = outer wheel plane



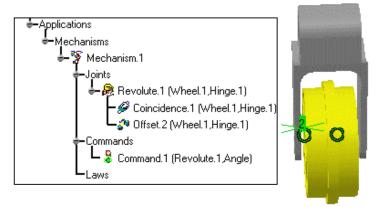
6. Assign the Angle driven command if needed.

7. Click Ok to end the Revolute Joint creation.

Joint Creation : Revolute	×
Mechanism : Mechanism.1	
Joint name : Revolute.1	
Current selection : Line 1 : Hinge.1/Solid.1 Plane 1 : Hinge.1/Solid.1 Plane 2 : Wheel.1/Solid.1 Plane 3 : Hinge.1/Solid.1 Plane 4 : Wheel.1/Solid.1 Current selection : Description: Current selection: Current	3
Angle driven	
	3

The specification tree is updated.

8. Open the Create_Revolute_Centered.CATProduct to check your result.

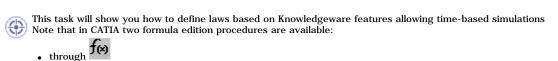






Version 5 Release 13

Defining Laws in a V5 Mechanism



• using the Command edition dialog box (see step3-8)

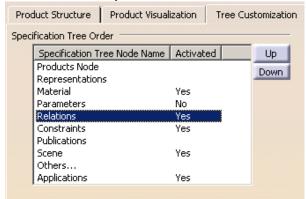
You need a V5 mechanism you can simulate. Please refer to Designing a V5 Mechanism i

Open the DEFNE_LAWS.CATProduct document.

Use the Fit All In icon **the base** to position the model geometry on the screen. (Optional) You can display the relations node in the specification tree.

For this, activate the relations option display:

- Select Tools->Options from the menu bar. • The Options dialog box is displayed
- Expand the Infrastructure category from the tree
- Select Product Structure item in the tree •
- Click the Tree Customization tab
- Activate the Relations option. •



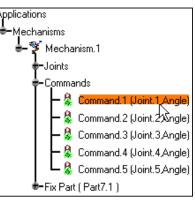
1. Click the Simulation With Laws icon in the DMU Kinematics toolbar. The Kinematic Simulation dialog box appears:

Kinematic Simulation - Mechanism.1	? ×
Mechanism : Mechanism 1	
To simulate with laws add at least a relation between a command and the Time Parameter	
	Close

You need to define at least one relation between a command and the Time parameter, let's create this relation referred to as law throughout this scenario.

2. Click the Close button to exit the dialog box.

You are going to create a law using the existing command. 1 (joint. 1, Angle)



3. IN CATIA ONLY Click the Formula icon from the Knowledge Toolbar. The "Formulas" dialog box is displayed. The 'Incremental' box must be unchecked.

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i Select Mechanism.1 in the specification tree to obtain quickly the parameters specific to your Mechanism document.

Formulas: Mechanism.1			<u>? ×</u>
Incremental Filter On Mechanism.1 Filter Name : Filter Type : All Double click on a parameter to edit it	T		Import
Parameter	Value	Formula	A
Mechanism.1\KINTime	10s		
Mechanism.1\Commands\Command.1\Angle	360deg		
Mechanism.1\Commands\Command.2\Angle	Odeg		
Mechanism.1\Commands\Command.3\Angle	Odeg		
Mechanism.1\Commands\Command.4\Angle	Odeg		
Mechanism.1\Commands\Command.5\Angle	Odeg		
			F
Edit name or value of the current parameter			
Mechanism.1\KINTime		10s	A
New Parameter of type Real	With Single Value	-	Add Formula
Delete Parameter			Delete Formula
		OK C	Apply Gancel

(i) Select Mechanism .1 in the specification tree to obtain quickly the parameters specific to your Mechanism document.

4. IN ENOVIA DMU Double-click Command.1 in the specification tree. Right-click Command value field and select the Edit Formula item from the contextual menu displayed. The 'Formulas' dialog box is displayed.

Command Edition : Command.1	(Angle)	? ×		
Command name : Command.1				
Command value : Ddeg		<u></u>		
,		<u>F</u> ormula	•	Edit
	<u> </u>	Edit Comment		Deactivate
				Delete

The Formula Editor dialog box appears:

5. Select Time in the Members of Parameters list.

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Pa	ae 1	72
	90 .	

Formula Editor : Mecha	nism. 1 \listcmd\Command. 1 \Angle	? ×
Incremental	○ Select Feature 🔮 Filter	0
Mechanism.1\listcmd\Com	mand.1\Angle =	
Dictionary Parameters Operators String Design Table Messages and macro: Attribute Math Attribute Innits	Members of Parameters Members of Time All Renamed parameters Boolean Length Volume Mass Area String Time Angle	
	ОК	Cancel

- 6. Double-click Mechanism.1\KINTime in the Members of time list.
- 7. Enter /1s*36deg after Mechanism. 1\KINTime to complete the formula.

When done click OK to exit the Formula Editor dialog box. In Enovia

Mechanism.1\listcmd\Command.1\Angle	=
Mechanism.1\KINTime /1s*36deg	

The Formulas dialog box is updated,

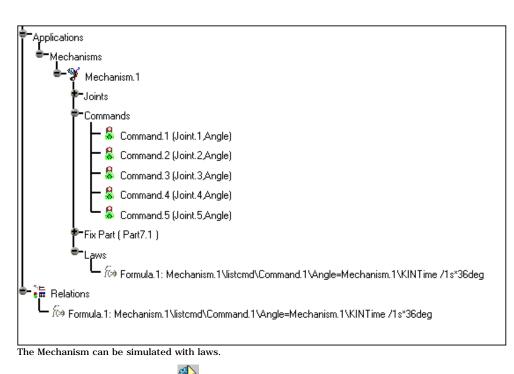
8. Click Ok to end the formula creation.

Formulas: Mechanism.1				? ×
Incremental Filter applied to Mechanism.1 All Double click on a parameter to edit it				Import
Parameter		Value	Formula	Active
Mechanism.1\KINTime		10s	•	
Mechanism.1\listcmd\Command.1\Angle		360deg	= Mechanism.1\KINTime /1s	'36 yes
Mechanism.1\listcmd\Command.2\Angle		Odeg		
Mechanism.1\listcmd\Command.3\Angle		Odeg		
Mechanism. 1 \listcmd \Command. 4 \Angle		Odeg		
Mechanism.1\listcmd\Command.5\Angle		Odeg		
Edit name or value of the current parameter				
New Parameter of type Real	With Single Value	-		Add Formula
Delete Parameter			OK App	Delete Formula

9. IN ENOVIA DMU: the Command Edition: Command.1 (Angle) is updated:

Command Edition : Comm	nand.1 (Angle) ? ×
Command name : Command Command value : 360deg	1.1	f(x)
	I OK	Cancel

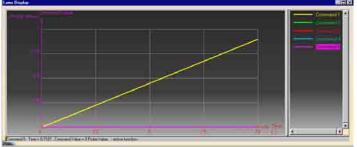
- The relation is created and identified in the specification tree under:
 - Relations item
 - Laws item



10. Click the Mechanism Analysis icon from the DMU Kinematics toolbar.

Click Laws... in the Mechanism Analysis dialog box displayed.

The laws display dialog box appears:



Open USE_LAWS.CATProduct to see another example in which various laws have been defined.

For more information, see the Knowledge Advisor User's Guide.

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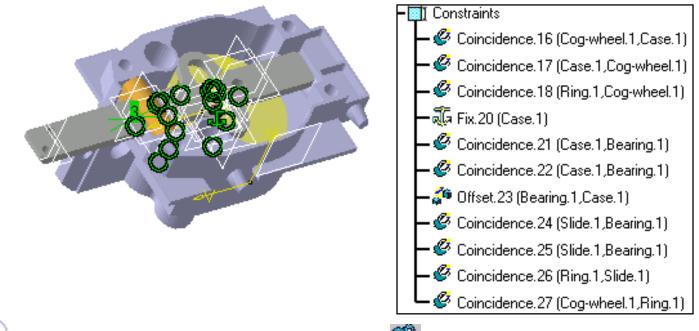


Converting Constraints Into Joints (Advanced Mode)

This task shows how to convert constraints into joints in advanced mode

Open the jigsaw_with_constraints.CATProduct document.

The constraints are visible both in the geometry area and in the specification tree.



1. Click the Assembly Constraints Conversion icon **Second Provide Second Provide Pro**

Assembly Constraints Conversion		?)
Mechanism : 🗾 💌	New Mechar	nism
Auto Create		More >>
	Unresolved Pairs :	
	🗿 ОК 🚺	🔰 Cancel

- **2.** Click on the New mechanism button.
- **3.** Click the More \rightarrow button.

The Assembly Constraints Conversion dialog box expands:

Assembly Constraints Conversion 🔹 🔀
Mechanism : Mechanism.1
Auto Create
Unresolved Pairs : 5/5 Product 1 : Bearing.1 Pair 1/5
Product 2: Case.1
Constraints List Resulting type Joints List
Name Type Elements type Offse Off Plane/Plane Coin Coi Line/Line Coin Coi Plane/Plane
Delete Joint
Fix Constraints List : Delete Fix Current Fixed Part :
Fix.20 (Case.1) Create Fix
OK SCancel

Let's look at this dialog box more carefully:

Unresolved Pairs: 5/5: specifies the status of the product pairs

Product 1 :	Ring.1	Pair 175
Product 2 :	Cog-wheel.1	

Shows the pair comprising of two products (Bearing.1 and Case.1). You are dealing with the first pair (1/5).

The Constraints list displays:

- the name and type of the constraints,
- the elements type and detailed information about the first and second element.

The joints and Fix constraints lists display the same kind of information.

Run the Assembly constraints conversion using one of the VCR buttons:

ilets you step forward.
ilets you go to next unresolved pair.
ilets you go to the last pair.

Joints

.aws

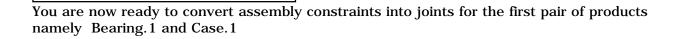
Commands

Fix Part (Case.1)

Fix.20 (Case.1)

4. Click Create Fix to create the fixed part. The fixed part is visible in the Current Fixed Part field.

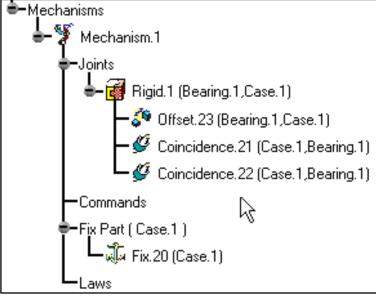




5. Multi-select Offset.23, Coincidence.21 and Coincidence.22 in the Constraints List (use CRTL Key + left mouse button) The Create Joint button is no longer grayed out.

Constraints	List	Resulting type		Joints Li	ist
Name	Туре	Rigid	Name	Туре	Constrair
Offset.23	Offset	Create Joint			
Coincidence.21	Coincid	Add Command :			
Coincidence.22	Coincid	None 💌			
		Delete Joint			
•	Þ		•		F

Click <u>Create Joint</u> (the resulting type is specified in the resulting type field). The Joint list is updated. If you are not satisfied, click the Delete Joint button. The Rigid joint is identified in the specification tree and highlighted in the geometry area



6. Proceed in the same manner to convert the remaining assembly constraints into joints. Use

Go to Next Unresolved button : 🌌

M

You converted the constraints but forgot to create a command. You need to assign the command to Revolute.4 Use **Step Backward** button

7. All you need to do is double-click Revolute.4 in the Joint list and check the Angle driven

option in the Edit joint dialog box displayed.

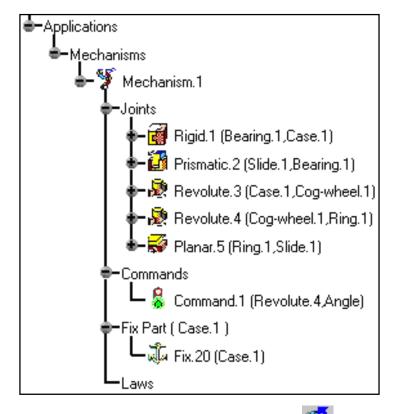
Constraints List	Resulting type	Joints	
Name Type Elements type	C		Type Constra
	Create Joint	Revolute.4	Re Coincid
Joint Edition : Revolute.4 (Re	volute)		? ×
Joint name : Revolute.4			
	Joint geometry :		
Line 1 : Axis	Line 2 : Axi		
Plane 1 : Face	Plane 2 : Far	ce	
🧧 Angle driven			
Joint Limits			
E Lower Limit : -360deg	📑 🖾 Upper Lin	iit : 360deg	
		OK I	Cancel

8. Click **Ok** in the Joint Edition dialog box. When done click OK. The following information message is displayed:



9. Click **OK** to confirm your operation.

The mechanism can now be simulated.



10. Click the Simulation with commands icon **W** from the DMU Kinematics toolbar. Please refer to Simulating With Commands.

Note that when you have several fix constraints in your assembly, you can create a rigid joint between each of them. When this is the case, the following warning is displayed:



Trace

Using the Trace Command Generating a Trace from a V5 Mechanism Generating a Trace from Lines

Using the Trace Command

The Trace and Swept volume functionalities are very useful to design respectively the trajectory of a moving point or the volume swept by a set of moving products during simulation. The main default of these commands is that they accept only a replay object to create the trace or the swept volume. Now these two commands accept also mechanisms. The only condition about these mechanisms is that they can be simulated with laws.



You can use this capability only if you have a Part Design and/or a Generative Shape Design license

This task explain how to use the trace of a point for design purposes. This is very useful in the design process as you can use the resulting trace to design cams.

Open the Create_Trace.CATProduct document. A simulation is recorded and compiled into a Replay object.

Automatic switch to Design mode:

If you work with the cache system in visualization mode, you no longer need to use **Edit**->**Representations**->**Design Mode** beforehand as the switch to design mode is automatic (an eye appears as you point the product in the geometry or specification tree). All you need to do is click on the object. (for more detailed information, please refer to *DMU Navigator User's Guide- Viewing the Cache Content*)



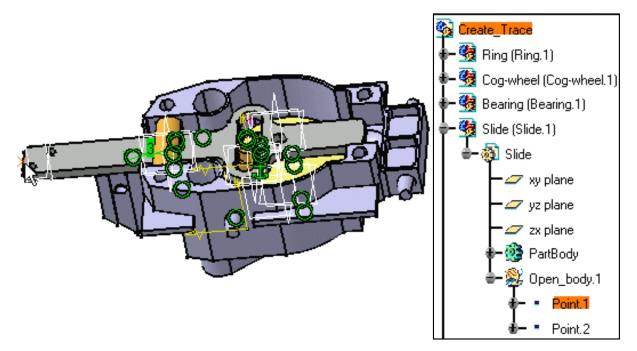


1. Click the Trace icon **Markov** from the DMU Generic Animations toolbar.

The Trace dialog box appears:

Trace	? ×
Object to trace out :	Replay.1
Elements to trace out	(No Selection)
Reference Product :	Create_Trace
Number of steps :	26
Destination of the tr	ace
New Part	O Reference Product
	OK Cancel

2. Select a point to trace either in the geometry area or in the specification tree.

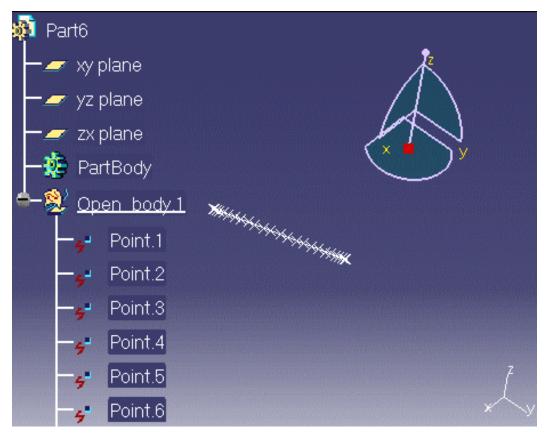


3. Click Ok to end the trace creation.

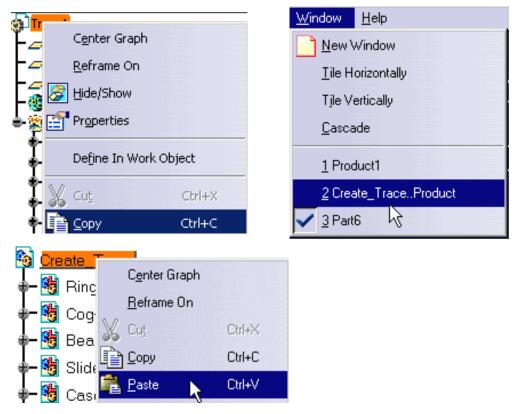
Trace	? ×
Object to trace out :	Replay.1
Elements to trace out	Point.1
Reference Product :	Create_Trace
Number of steps :	26
Destination of the tr	ace
New Part	O Reference Product
	OK Cancel

i If the Reference product is a not a Part, the trace destination is a New Part document as you cannot only write into a part document.

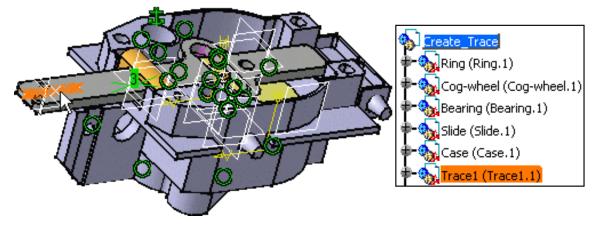
The trace is created in a New Part and looks like this:



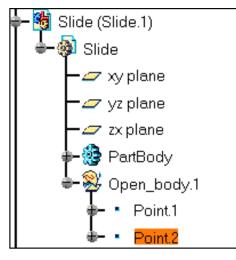
4. Copy your resulting trace into your initial document, for this use the Copy/Paste capability.



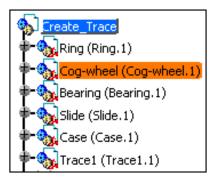
5. The trace is identified both in the specification tree and in the geometry area



- 6. Now, Run the replay step by step. For this: select Replay in the specification tree and double click replay1.
- 7. Click the Trace icon icon again.
- 8. Expand the Slide node and select point 2 in the specification tree as point to trace.

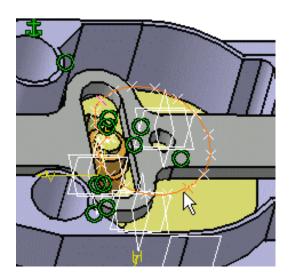


9. Select Cog-Wheel.1 as Reference product



10. Select the trace destination (Reference Product) when done, click Ok.

The trace appears in the geometry area in the part select ($\operatorname{Cog-Wheel}\nolimits.1)$



11. Now, Run the replay step by step. For this: select Replay in the specification tree and double click replay1.



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2...

Generating a Trace from a V5 Mechanism (which can be simulated with laws)

This task explains how to generate the trace of a point from a V5 mechanism which can be simulated with laws.

This is very useful in the design process as you can use the resulting trace to design cams. Open the Use_Sensors.CATProduct document.

Automatic switch to Design mode:

If you work with the cache system in visualization mode, you no longer need to use **Edit->Representations->Design Mode** beforehand as the switch to design mode is automatic (an eye appears as you point the product in the geometry or specification tree). All you need to do is click on the object. (for more detailed information, please refer to *DMU Navigator User's Guide- Viewing the Cache Content*)

1. Click the Simulation With Laws icon 🔯 in the DMU Kinematics toolbar

Kinematic Simulation -	Mechanism.	1 ? ×
Mechanism : Mechanism.	1	•
Start 0	10 <mark>0.000</mark>	.
Number of steps: 40	_	Analysis
Activate Sensors		
		Close

- 2. Run the simulation using the simulation buttons
- 3. Click Close when satisfied.

4. Click the Trace icon in from the DMU Generic Animations toolbar.

The Trace dialog box appears:

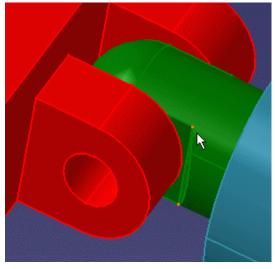
Trace	? ×
Object to trace out :	Mechanism.1
Elements to trace out :	(No Selection)
Reference Product :	Use_Sensors
Number of steps :	41
-Destination of the tr	ace
New Part	O Reference Product
<u> </u>	OK Cancel
About The number of steps:	

It is not possible to edit the *number of points* value directly in the Trace dialog box. By-pass: click the Simulation With Laws icon and change the number of steps value.

If the number of steps value is 100 for example, the number of points traced will be 101 because of the original position. Then the Trace dialog box will appear like this :

Kinematic Simulation - Mechanism.1 🛛 🛛 🔀	
	Trace ? 🗙
Mechanism : Mechanism.1 Start 0 10 0.000 Image: Comparison of the start o	Object to trace out : Mechanism.1
Number of step 100 Analysis	Number of steps : 101 Destination of the trace New Part O Reference Product

5. Select a point to trace either in the geometry area or in the specification tree.

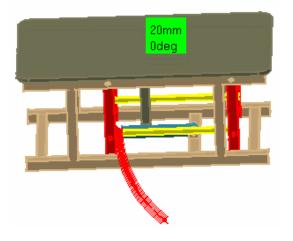


6. Select a reference product, click within the field and select Rear_Moving-Arm.1 in the specification tree

Trace	? ×
Object to trace out :	Mechanism.1
Point to trace out :	Vertex
Reference Product :	Rear Moving Arm.1
Number of Points :	41
Destination of the	trace
O New Part	Reference Product
	🕒 OK 🧾 🍛 Cancel

7. Click **Ok** to end the trace creation.

I f the Reference product is a not a Part, the trace destination is a New Part document as you cannot only write into a part document. The trace is created and looks like this:



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Generating a Trace from Lines

The trace command is very useful to build the trajectory of a moving point of a mechanism or a replay. You can now select lines and several elements (point and /or lines) at a time to generate a line.

This task explains how to generate the trace of a point

Open the Use_Laws.CATProduct document.

Note you can only generate a trace from a replay or a mechanism **which can be simulated with laws**. Remember, a mechanism with no laws associated does not appear in the "Object to trace out" field of the Trace dialog box

Automatic switch to Design mode:

If you work with the cache system in visualization mode, you no longer need to use **Edit->Representations->Design Mode** beforehand as the switch to design mode is automatic (an eye appears as you point the product in the geometry or specification tree). All you need to do is click.

(for more detailed information, please refer to DMU Navigator User's Guide- Viewing the Cache Content)



- 1. Click the Simulation With Laws icon in the DMU Kinematics toolbar
- 2. Change the number of steps value to 120

Kinematic Simulation -	Mechanism	.1 ?×
Mechanism : Mechanism.	1	-
Start 0	25 0.000	
		M
Number of steps: 120	•	Analysis
Activate Sensors		
		Close

3. Run your simulation if necessary

About The number of steps:

It is not possible to edit the *number of points* value directly in the Trace dialog box. By-pass: click the Simulation With Laws icon and change the number of steps value.

If the number of steps value is 100 for example, the number of points traced will be 101 because of the original position. Then the Trace dialog box will take into account the modification: step 4



4. Click the Trace icon **Markov** in the DMU Generic Animations toolbar.

The Trace dialog box appears: The number of steps is 121 (120 + original position)

Trace	? ×
Object to trace out :	Mechanism.1
Elements to trace out :	
Reference Product :	Product1
Number of steps :	121
Destination of the tra	ace
New Part	O Reference Product
<u> </u>	OK Cancel

5. Zoom in, and select a line to trace out as shown below:

Trace ? X
Object to trace out : Mechanism.1 Elements to trace out : Edge Reference Product : Product1 Number of steps : 121 Destination of the trace O Reference Product New Part O Reference Product
Cancel

Note: You can multi-select elements (lines and/or points). In this case, the number of elements you selected appear in the "Elements to trace out" field

For instance, if you selected three elements (two lines + a point, see picture below), three traces will be created. The traces of the three geometrical elements will be created in separate bodies

1) 1)	
1)	
4.1)	
41)	

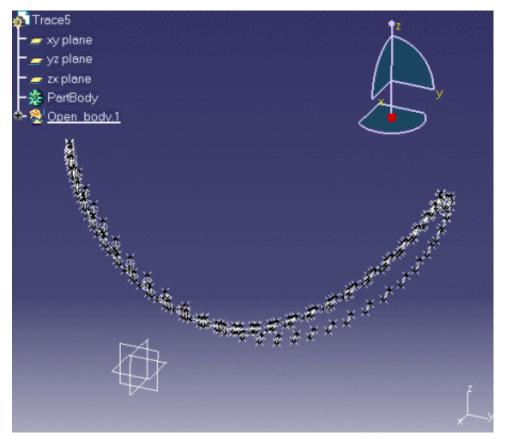
Trace	? ×
Object to trace out :	Mechanism.1
Elements to trace out :	
Reference Product :	Product1
Number of steps :	121
- Destination of the tr	ace
New Part	O Reference Product
	OK Scancel

6. Click Ok to end the trace creation

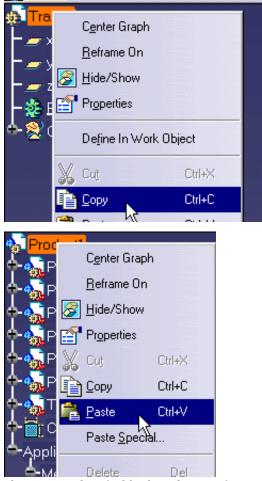
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i If the Reference product is a not a Part, the trace destination is a New Part document as you cannot only write into a part document.

The trace is created in a New Part and looks like this:

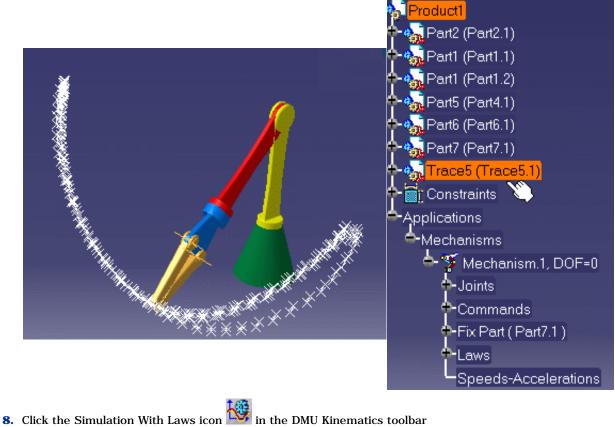


7. Copy your resulting trace into your initial document, for this use the copy/ paste capability.





The trace is identified both in the specification tree and in the geometry area



- 9. Launch your simulation with laws using the Play forward button



Setting Joint Limits



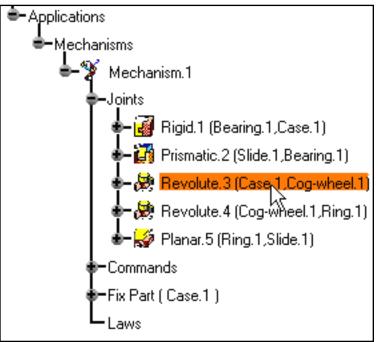
This task consists in setting joint limits.

Limits for joints which can be assigned commands are always set. Those limits are used in kinematics simulation context (please refer to Simulating With Commands)

Open the SETTING_LIMITS.CATProduct document.

•

1. Double-click Revolute.3 in the specification tree.



The Joint Edition: Revolute.3 appears:

Joint Edition : Revolute.3	? ×
Joint name : Revolute.3	
Joint ge	eometry :
Line 1 : Axis	Line 2 : Axis
Plane 1 : Face	Plane 2 : Face
🧧 Angle driven	
Joint Limits	
🖾 Lower Limit : 🛛 -360deg 🛛 😭	🖾 Upper Limit : 360deg
	OK Gancel

1

Because a command is assigned to Revolute.3, the limits are necessarily set. The default values for angle limits are:

- Lower limit -360deg
- Upper limit 360deg

For length command types limits the default values are the following (for Prismatic joints,..)

- Lower limit -100mm
- Upper limit 100mm

Remember you can at any time change the unit using **Tools**->**Options**-> **Parameters and Measures**-> **Units...**

2. Click **Cancel** to exit the Joint Edition dialog box.

You are going to set limits on the prismatic joint which has not been assigned any command

3. Double-click Prismatic.2 in the specification tree.

The Joint Edition: Prismatic.2 is displayed.

Joint Edition : Prismatic.2 (Prismatic)	? ×
Joint name : Prismatic.2	
	cometry :
Line 1 : Edge	Line 2 : Edge
Plane 1 : Face	Plane 2 : Face
Length Driven	
Joint Limits	
Lower Limit : unset	Upper Limit : Unset
	🕒 OK 🥥 Cancel

- 4. Check the Lower and Upper limit buttons and enter the required values:
 - **-10mm**
 - o 10mm

Joint Limits			
🧧 Lower Limit :	-10mm	📮 🔎 Upper Limit :	10mm 📑
1. Second States			

5. Click Ok to confirm your operation.

You are ready to run a simulation with the limits set.



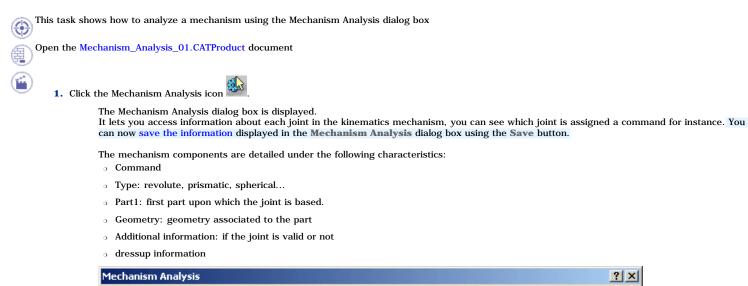
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Mechanism Analysis

DMU Kinematics Simulator provides easy methods to detect and analyze collisions and distances between products. It also provides the capacity of generating a swept volume. The DMU Space Analysis Version 5 product must be installed before using certain functionalities such as swept volume.

Analyzing a Mechanism Sensors Other Analyses Measures

Analyzing a Mechanism



1.	rechams	II Analysis								
	General	Properties —					<u> </u>			
	Mechanism	n name :		Mechanisr	n.1					-
	Mechanism	n can be simul	ated :	Yes						
	Number of	joints :		3						
	Number of	commands :		1						
	Degrees o	f freedom wit	hout comman	d : 1						
	Degrees o	f freedom wit	h commands :	0						
	Fixed part	1		Inner ring)					
	O Show j	joints 🔘 Hide	e joints			Save				Laws
	Joint	Command	Туре	Part 1	Geometry 1	Part 2	Geometry 2	Part 3	Additional info	ormation
	Joint Joint.1	Command	Roll Curve	Inner ring	Edge	Part 2 Roller	Edge		Valid Joint	ormation
	Joint.1 Joint.2		Roll Curve Roll Curve	Inner ring Roller	Edge Edge	Roller outer ring.1	Edge Edge			ormation
	Joint.1	Command Command.1	Roll Curve	Inner ring	Edge	Roller	Edge		Valid Joint	prmation
	Joint.1 Joint.2 Joint.3		Roll Curve Roll Curve Revolute	Inner ring Roller	Edge Edge	Roller outer ring.1	Edge Edge		Valid Joint	prmation
	Joint.1 Joint.2 Joint.3	Command.1	Roll Curve Roll Curve Revolute	Inner ring Roller	Edge Edge	Roller outer ring.1	Edge Edge		Valid Joint	prmation
	Joint.1 Joint.2 Joint.3 Mechanism	Command.1	Roll Curve Roll Curve Revolute	Inner ring Roller outer ring.1	Edge Edge	Roller outer ring.1	Edge Edge		Valid Joint	prmation
	Joint.1 Joint.2 Joint.3 Mechanism	Command.1	Roll Curve Roll Curve Revolute	Inner ring Roller outer ring.1	Edge Edge	Roller outer ring.1	Edge Edge		Valid Joint	prmation
	Joint.1 Joint.2 Joint.3 Mechanism	Command.1	Roll Curve Roll Curve Revolute	Inner ring Roller outer ring.1	Edge Edge	Roller outer ring.1	Edge Edge		Valid Joint Valid Joint	Close

i If you defined a new mechanism, when you delete a part including in the mechanism the corresponding joint is no longer valid. The message invalid joint! appears in the Mechanism Analysis dialog box.

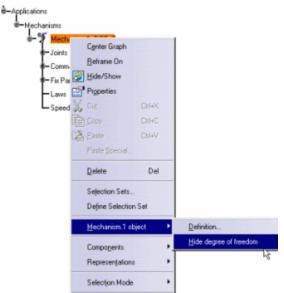
The degree of freedom is displayed by default.

But you can still choose to hide the degree of freedom of the mechanism:

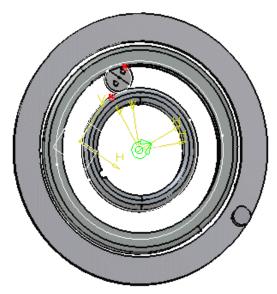
• Right-click mechanism.1 in the specification tree and select hide degree of freedom item form the contextual menu displayed:

Applications
Mechanisms
🚽 🌮 Mechanism 1, DOF=0
🖝 Joints 🛛 🗟
🗰 – Commands
🕶 Fix Part (Inner ring)
- Laws
Speeds-Accelerations

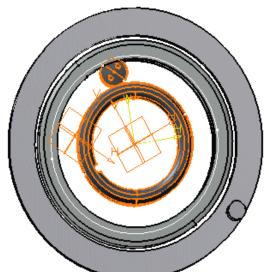
Version 5 Release 13

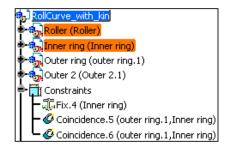


2. Check the Show joints radio button. All joints are visualized in the geometry. (if you select one particular joint, the corresponding joint is visualized)



i Note, a low-light visualization mode is available. You can better visualize the different components involved in joints. For instance, select Joint.1 in the list.





The components involved in the Roll Curve joint are highlighted in the specification tree and in the geometry

3. Select Joint.3 in the list.

The mechanism dressup information is displayed

DMU Kinematics Simulator

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Joint	Command	Туре	Part 1	Geometry 1	Part 2	Geometry 2	Part 3	Additional information
Joint.1		Roll Curve	Inner ring	Edge	Roller	Edge		Valid Joint
Joint.2		Roll Curve	Roller	Edge	outer ring.1	Edge		Valid Joint
Joint.3	Command.1	Revolute	outer ring.1	Axis	Inner ring	Axis		
-								
Mechani	sm dressup info	ormation:		*********				
outer rin	ng.1 Inr	ier ring						
Outer 2.	1							
1								

- 4. Now click Close
- 5. Open the Mechanism_Analysis_02.CATProduct document
- 6. Repeat step 1

Mechan	ism Analysis								<u>? ×</u>
Gener	al Properties -	<u></u>							
Mechan	ism name :		Mechanis	m.1					-
Mechan	ism can be simu	lated :	Yes						
Number	of joints :		3						
Number	of commands :		1						
Degree	s of freedom wit	hout comman	d: 1						
Degree	s of freedom wit	th commands	: 0						
Fixed p	art :		Inner ring)					
O Sho	w joints 🔮 Hid	e joints				Save			Laws
Joint	Command	Туре	Part 1	Geometry 1	Part 2	Geometry 2	Part 3	Additional information	
Joint.1		Roll Curve	Inner ring	Edge	Roller	Edge		Invalid Joint ! minimal distance :	= 4.20149 mm.
Joint.2 Joint.3		Roll Curve Revolute	Roller	Edge	outer ring.1	Edge		Invalid Joint !	
Jointes	Command.1	Revoluce	outer ring.1	Axis	Inner ring	Axis			
Mechan	ism dressup info								
Inner	ring Rolle	r							
1									
									Close

The Mechanism Analysis dialog box appears:

you can benefit from a feedback about the validity of your joints in Additional information field

In our example, two joints are considered as broken. You will need to redesign them.

You can now save the information in various formats: .xls, .txt and Lotus 123 (provided that you have it installed on your machine). To do so:

- 7. Click Save. The Save As dialog box appears:
 - set the appropriate Save as type using the drop-down list (.xls in our example)
 - $_{\circ}$ $\,$ identify the folder in which you want to save the file
 - o enter a file name
 - o click Save

Version 5 Release 13

				? >
🔁 samples		• +	🗈 💣 🎫	
File name:	Mechanism_Analysis		•	Save
Save as type:	Microsoft Excel workshee	ts (*.xls)	-	Cancel
	Lvti_cnf	Le name: Mechanism_Analysis	File name: Mechanism_Analysis	vti_cnf File name: Mechanism_Analysis

- 8. Click Close.
- 9. Open the Mechanism_Analysis.xls file you have just created. You should obtain something like this:

٩M	lechanism_Analysis.xls										_ 🗆 🗵
	A	В	C	D	E	F	G	Н	1	J	K
1	Mechanism name :	Mechanism.1									1
2	Mechanism can be simulated :	Yes									
3	Number of joints :	3	3								
4	Number of commands :	1	1								
5	Degrees of freedom without command :	1	1								
6	Degrees of freedom with commands :	()								
7	Fixed part :	Inner ring									
8											
9											
10	Joint	Command	Туре	Part 1	Geometry	Part 2	Geometry	Part 3	Additional	information	-
11	Joint.1		Roll Curve	Inner ring	Edge	Roller	Edge		Invalid Joir	nt ! minimal	distance
12	Joint.2		Roll Curve	Roller	Edge	outer ring.	Edge		Invalid Joir	nt !	
13	Joint.3	Command.1	Revolute	outer ring.	Axis	Inner ring	Axis				
14											
1E I(+ + +) Mechanism_Analysis						ыČ					



Sensors

Using Sensors Creating Y=f(X) combined sensors curves Measuring Speeds and Accelerations

Using Sensors

About sensors:

Ð

This functionality enables to visualize all joint values (with commands or not), measures and joint limits if defined throughout the simulation process. These different values used as sensors provide useful information to check your mechanism design through both kinematics simulation operations (i.e. simulation with laws and simulation with commands)

Existing distances and interferences specifications are available in the sensors list.

Please follow the step-by-step scenario described below:

This task consists in using sensors to check joint values and measure values during simulation.

Open the Use_Sensors.CATProduct document.

Note: the degree of freedom of the mechanism displayed by default, if you want to hide it all you need to is right-click the mechanism and select Hide degree of freedom item from the contextual menu displayed

<u>M</u> echanism.1 object	×	Definition
Components	•	<u>H</u> ide degree of freedom
-		

1. Click the Simulation With Laws icon 🎬 in the DMU Kinematics toolbar. The Kinematic Simulation - Mechanism. 1 dialog box is displayed:

Kinematic Simulation -	Mechanism	.1 ? 🗙
Mechanism : Mechanism.	.1	
Start 0	10 0.000	.
Number of steps: 40		Analysis
Activate Sensors		
		Close

2. Check the Activate Sensors option. The Sensors dialog box is automatically displayed

This scenario aims at checking your mechanism complies with the bill of material. The required specifications to be checked in our example are the following:

- Table height = 815 mm (see measure already defined)
- Table path = 200 mm approximately (815mm to 1015mm)
- Limits are set on prismatic. 13 (lower limit=0, upper limit = 200mm)
- o There is a law defined corresponding to the jack path
- Minimum distance between the Arm_Joint products and the table + fixation table

In this first try, you are going to check if your Kinematics mechanism is correctly designed using the corresponding sensors during simulation

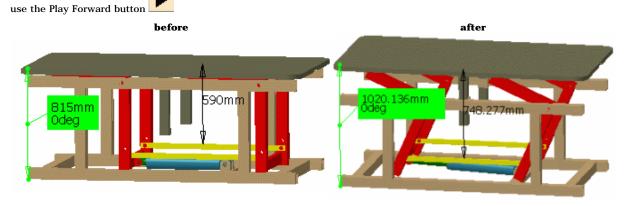
Besides, we added an interference specification.

3. Select the sensors to be observed:

- Prismatic.13\Length (corresponding to the table path)
- MeasureBetween. 369\Length (table height)
- Prismatic.14\Length (corresponding to the jack path)
- Distance Results. 1\Minimal Distance
- 4. Set the Check Limits option to Stop.

ensors		2
Selection Instantaneous Values History		
Sensor	Unit	Observed A
Mechanism.1\Joints\Prismatic.13\Length	Millimeter	Yes
Mechanism 1 Joints Prismatic 14 Length	Milimeter	Yes
Mechanism, 1 Voints \Revolute, 12 \Angle	Degree	No
Mechanism.1\Joints\Revolute.13\Angle	Degree	No
Mechanism.1\Joints\Revolute.14\Angle	Degree	No
Mechanism.1\Joints\Revolute.15\Angle	Degree	No
MeasureBetween. 369\Length	Millimeter	Yes
MeasureBetween. 369\Pt1x	Millimeter	No
MeasureBetween. 369\Pt1y	Millimeter	No
MeasureBetween. 369\Pt1z	Millimeter	No
MeasureBetween.369\Pt2x	Millimeter	No
MeasureBetween. 369\Pt2y	Millimeter	No
MeasureBetween. 369\Pt2z	Millimeter	No
`Distance Results.1\Minimal Distance`	Millimeter	Yes 💌
Deselect All	Select All	
Display Options		Lin
All	O Limited 1	
Detect Clashes Check Limits		\
Automatic O Interferences O Off	O On Sto	
	VIE	/
Graphics Options	File	
		Close

5. Select the **History** tab to visualize the sensors behavior while running your simulation:



- 6. Check the last values for:
 - MeasureBetween.369
 - o Prismatic.13
 - o Prismatic.14

i

o Distance Results.1

Notice the sensors values are valid and correspond (approximately) to the specifications You can re-dimension the jack path to 260mm The measure is now 1020.136mm.

Sensors					<u>? ×</u>
Selecti	ion Instantaneous Va	lues History	1		
Eom	Sensors	a second and	Value	Unit	Comment A
234	Distance Results.1/Mir	nimal Distance`	737.312	Millimeter	
247	Mechanism 1 Joints Pris	smatic.13\Length	195.142	Millimeter	534
247	Mechanism 1 Joints Pris	smatic.14\Length	247	Millimeter	
247	MeasureBetween.369\l	.ength	1010.14	Millimeter	
247	Distance Results.1 Min	nimal Dictance`	742.944	Millimeter	2.4
260	Mechanism, 1 (Joints)Pris	smatic.13\Length	205.136	Millimeter	Upper
260	Mechanism.1\Joints\Pris	smatic.14\Length	260	Millimeter	
260	MeasureBetween.369\l	.ength	1020.14	Millimeter	
260	`Distance Results.1\Mir	nimal Distance`	748.277	Millimeter	
			Clear		
Display	Options				
🛛 All			OL	imited 1	Lines
Detect	Clashes	Check Limits			
Autor	matic O Interferences	O Off	Oon	Stop	
		Outputs			
	6	Graphics	Options	File	
	V	alues to	be checked		Close

You haven't finished yet as we added an interference specification

- 7. Clear the History using the Clear button
- 8. Modify if necessary the Display Options, the default mode is all
- 9. Click the Selection tab and select the Interference Results. 1\Nbclash sensor

The interference is selecte Now, click stop	d by default and set to on
- Detect Clashes O Automatic 🥥 Interferen	ces
💰 💰 👰	5

What happens when you select a interference sensor?

The following table summarizes the various cases and gives the corresponding clash status: Detect Clashes

status	Automatic	Interferences
	Mode is set to interferences and clash detection switches to ON (see	Mode remains set to interferences and clash detection switches to ON
OFF	image below)	(see image below)
	Detect Clashes	Detect Clashes
	O Automatic 🔮 Interferences	O Automatic 🥌 Interferences
	Mode is set to interferences and	Mode remains set to interferences and clash detection
ON	clash detection remains set to ON	remains set to ON
011	(see image below)	(see image below)
	Detect Clashes	Detect Clashes
6	🔿 Automatic 🥥 Interferences	O Automatic 🔮 Interferences
	<u>6</u> 🔊	
	Mode is set to Interferences and	Mode remains set to interferences and clash detection
	clash detection remains set to STOP	remains set to STOP
STOP	(see image below)	(see imae below)
	Detect Clashes	Detect Clashes
%	O Automatic 🥌 Interferences	
	🚯 🚯 🚳	

Two actions clear the interference sensors selection (i.e. ALL interference sensors selected)

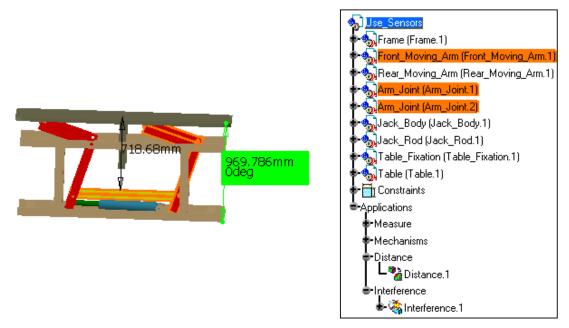
- $_{\odot}~$ When the Interferences mode is selected with the clash detection set to OFF
 - $_{\odot}~$ When you switch from Interferences to Automatic mode
- **10.** Click the **Analysis** button:
 - select Interference.1
 - o click Browse to check the interference specification already defined. See Detecting Clashes if necessary

E	dit A	nalysis	? ×
	No	Name	
	1	MeasureBetween.369	1
	2	Distance.1	
	3	Interference,1	
		Browse	
6		N3	
	-	Clo	se
	1		

11. Run your simulation again (click the Start button **11**). Repeat from step 5

This time, the measure is not valid with respect to the specifications (969.786mm instead of approximately 1015mm) In the previous try it equals 1020.136mm which is correct

The parts in collision are highlighted in the geometry and in the specification tree You will need to redesign the Rear_moving_Arm .CATPart.



12. Once satisfied, click the Graphics button in the outputs area to obtain a graphical representation

Note: You can now plot a sensor according to another sensor using the option button. Please read Creating Y=f(X) combined sensors curves

- Click File to save as a .xls, .txt or Lotus 123 (provided that you have it installed on your machine). Give a name and a path
- **14.** Read your document

📳 u	sesensorsre	esult.xls				_ 🗆 ×
	Α	В	С	D	E	F 🛓
1	Time	Mechanisr	Mechanisr	MeasureBe	etween.369	\Length(Mi
2	0	. 0	0	815		
3	0.25	10.4337	13	825.434		
4	0.5	20.8531	26	835.853		
5	0.75	31.2584	39	846.258		
6	1	41.6497	52	856.65		
7	1.25	52.0265	65	867.026		
8	1.5	62.3883	78	877.388		
9	1.75	72.7344	91	887.734		
10	2	83.0639	104	898.064		
11	2.25	93.3755	117	908.376		
12	2.5	103.668	130	918.668		
13	2.75	113.94	143	928.94		
14	3	124.19	156	939.19		
15	3.25	134.415	169	949.415		
16	3.5	144.615	182	959.615		
17	3.75	154.786	195	969.786		
18						-
RÍ 4	▶ ▶ \She	et1 /		•		<u>ر</u> اط

 (Δ)

 (Δ)

Under UNIX, It is impossible to save your results in . xls format.

Save As	? ×
Savejn: 🔂 Temp	I 🖻 📸 🗐
📄 Bmtmp	🔲 Vbe
Excel8.0	🏝 ~0061649.xls
📄 FrontPageTempDir	💐 ENOVIA_DMU_Navigator_ph
Mse000	💐 fitug-09-26.xls
🔲 🛄 Mse001	💐 Fitugfbp.xls
📄 msoclip1	💐 Fitugfbp0122.xls
•	Þ
File <u>n</u> ame: usesensorsresult	Save
Save as type: Microsoft Excel worksheets	(*.xls) Cancel

About interferences

When an interference is defined in your product, and activated as a sensor The sensor "value" :

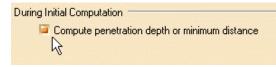
MeasureBetween.1\Diz Interference Results.1\NbClash	-2.35188 0	Millimeter
'Interference Results.1/NbContact'	4 16	
'Interference Results.1\Value`	1.83706	Millimeter

represents either:

- the penetration depth (if there are clashes in the specification results) or
- the clearance value (if there are only clearances in the specification results)

This sensor "value" is valuated only if you checked the Compute penetration depth option in the During Initial Computation clash command setting via Tools->Options->DMU Space Analysis-> DMU Clash tab at interference creation

DMU Kinematics Simulator



(About Interferences, Distances and Measures

- If you create interferences or distances without exiting the Kinematics simulation commands (either with laws or with commands), these new interferences /distances will not be displayed in the sensors list (this list is frozen when entering the simulation commands).
- Note that the distances and measures will not be visible in the geometry area until you activate at least one sensor belonging to these analyses.



Creating Y=f(X) combined sensors curves

About sensors:

This functionality enables to visualize all joint values (with commands or not), measures and joint limits if defined throughout the simulation process.

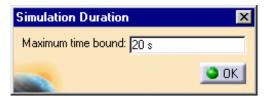
These different values used as sensors provide useful information to check your mechanism design through both kinematics simulation operations (i.e. simulation with laws and simulation with commands)

Within a simulation with laws, you could only plot sensors with respect a time parameter, now you can plot a sensor with respect to another sensor

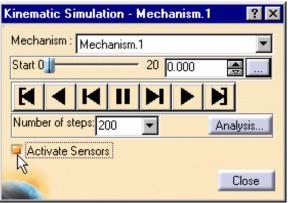
This task consists in using sensors to check joint values and measure values during simulation.

Open the Engine_V4.CATProduct document.

- 1. Click the Simulation With Laws icon in the DMU Kinematics toolbar. The Kinematic Simulation -Mechanism. 1 dialog box is displayed.
- Change the simulation duration, click the Edit Time range button
 The Simulation duration is automatically displayed.
 The default duration is 10 s
- 3. Enter 20 s in the maximum time bound field



- 4. Change the step number to 200
- 5. Select the Activate Sensors check box



This scenario aims at checking the motion of the valve with respect to the crankshaft

You are going to check if your Kinematics mechanism is correctly designed using the corresponding sensors during simulation

6. Select the sensors to be observed in the Sensors dialog box

- Prismatic.25
- Revolute.18
- Revolute.5

Launch the Simulation With laws

7. Click the **History** tab to visualize the sensors behavior while running your simulation:

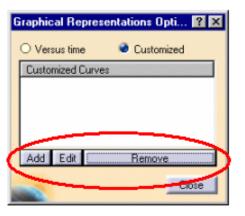
use the **Play Forward** button

8. Click the **Options** button

The Graphical Representation Options dialog box is displayed:

Graphical Represer	ntations Opti ?
Versus time	O Customized
Customized Curves	
Add Edit	Remove
	Close

9. Select the Customized option button: the Add, Edit and Remove buttons become accessible



10. Click Add, the Curve Creation dialog box is displayed

Curve Cre	ation	_ 🗆 🗙
	Curve.1	
Abscissa:	Mechanism.1\Joints\Revolute.5\Angle	•
	Mechanism.1\Joints\Revolute.5\Angle	
		🗿 ОК

11. In the **Abscissa** and **Ordinate** lists, select the required sensors

Curve Cre	eation	_ 🗆 ×
Name:	Curve.1	
Abscissa:	Mechanism.1\Joints\Revolute.5\Angle	
	Mechanism.1/Joints\Revolute.5\Angle	
	Mechanism.1\Joints\Revolute.6\Angle Mechanism.1\Joints\Revolute.7\Angle	
	Mechanism.1\Joints\Revolute.8\Angle	
	Mechanism.1\Joints\Revolute.9\Angle Mechanism.1\Joints\Revolute.10\Angle	_
	Mechanism.1Voints\Revolute.11\Angle	
	Mechanism.1 Voints\Revolute.12\Angle	
	Mechanism.1\Joints\Cylindrical.14\Leng Mechanism.1\Joints\Cylindrical.14\Ang	
	Mechanism.1Voints\Cylindrical.15\Leng	
	Mechanism.1Voints\Cylindrical.15\Angl	

- **12.** Create two customized curves:
 - curve1: prismatic 25 with respect to Revolute18
 - $_{\odot}$ curve2: prismatic 25 with respect to Revolute 5

(Optional)

Give a meaningful name to your new customized curve.

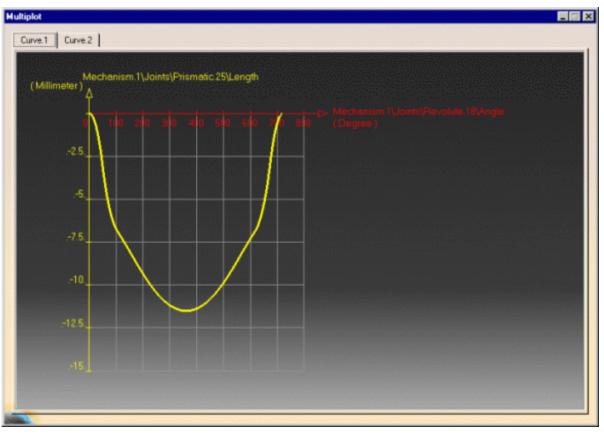
Curve Cre	eation 📃 🗌 🗙	Cu	rve Cre	ation 📃 🗌 🗙
Name:	Curve.1	Na	ame:	Curve.2
Abscissa:	Mechanism.1\Joints\Revolute.18\Angle	At	oscissa:	Mechanism.1Woints\Revolute.5\Angle
Ordinate:	Mechanism.1 Voints \Prismatic.25 \Length	Or	dinate:	Mechanism.1 Joints \Prismatic.25 \Length
	<u> </u>			

13. Click **Ok** when done. The two curves are created:

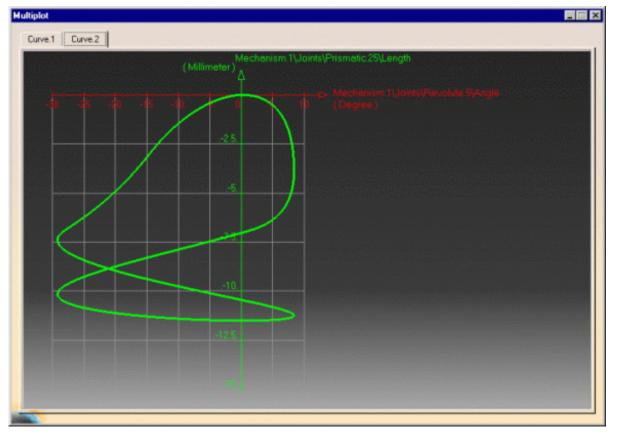
×

14. Click Close.

15. Click the Graphics button from the outputs to obtain a graphical representation. The curve 1 is displayed



16. Click Curve.2 tab:



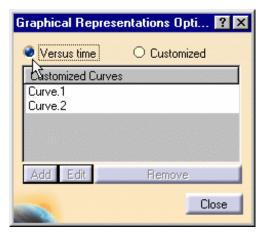
17. Click File to save as a .xls or .txt file.

Give a name and a path

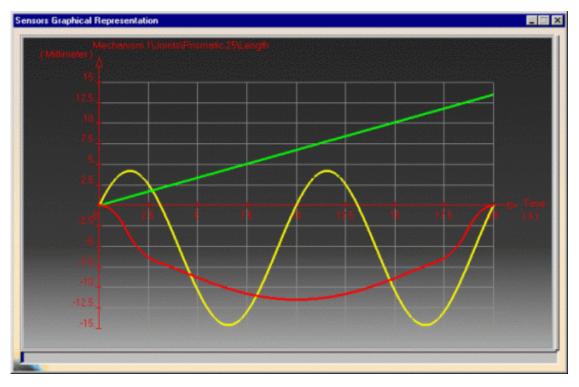
i Note you can Swap to the default mode (curves plotted with respect to time) at any time

DMU Kinematics Simulator

18. Click the **Options** button and select the **versus time** option button



- 19. Click Close
- **20.** Click again the **Graphics** button from the outputs to obtain a graphical representation





Measuring Speeds and Accelerations

About measuring speed and acceleration:

To qualify a mechanism behavior, or to improve its design, it is required to measure speed and accelerations during mechanism operation. Linear Speed and Acceleration calculations are based a point with respect to a reference product, whereas Angular Speed and Acceleration are those of the product to which the point belongs.

You can choose the Cartesian System Axis for the result projection using the Other axis option in the Speed and Acceleration dialog box

This task consists in measuring speeds and accelerations. We want to calculate the reduction ratio of the planetary reducer. We need to measure speeds and accelerations on a point belonging to the output axis.

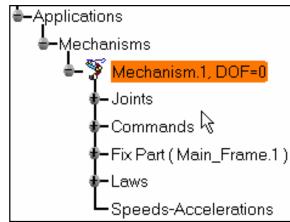
In our example we will define speeds and accelerations:

- on a point belonging to the output shaft (Eccentric_Shaft) with respect to the main frame
- on a point belonging to the exit shaft (Exit_Shaft) with respect to the main frame To simplify the results, we assume the shaft has a translation movement, therefore the results will be projected onto an axis system belonging to the main frame (z axis is co-linear with respect to the exit shaft axis)

Note: this operation can only be performed on mechanisms which can be simulated with laws. Open the MeasureSpeedAcceleration.CATProduct document.

1. Select the mechanism on which you want to define speeds and acceleration specification

i.e. select Mechanism.1 in the specification tree



Note: the degree of freedom of the mechanism displayed by default, if you want to hide it all you need to is right-click the mechanism and select Hide degree of freedom item from the contextual menu displayed

Mechanism.1 object 💦 🕨	<u>D</u> efinition
Components •	<u>H</u> ide degree of freedom
	1



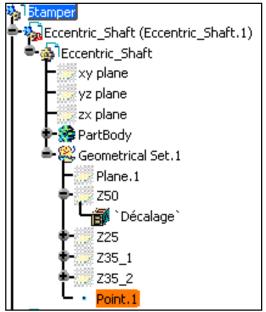
Acceleration dialog box appears

3. (Optional)

Enter a meaningful name. In our example keep the default name which is: Speed-Acceleration1

4. Click once in the **Point selection** field and select a point belonging to the parts involved in the mechanisms.

i.e. select Point.1 under Eccentric_Shaft either in the specification tree or in the geometry area



Note: The speeds and accelerations of this point (sensor) will be calculated with respect to a reference product

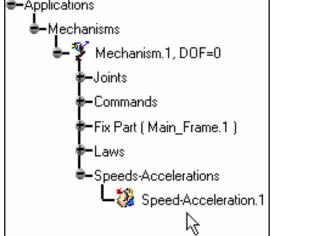
5. Click once in the **Reference product** field and select the reference product of your choice

i.e. select Main_Frame.1 either in the specification or in the geometry area.

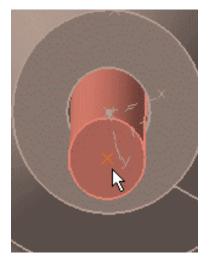


- **6.** Select an axis system for the projection of the result. In our example, keep the default one which is the root product axis system.
- 7. Click **Ok** in the Speed and Acceleration dialog box

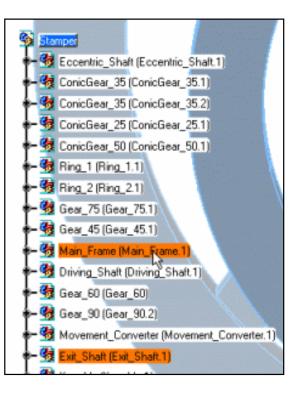
	Speed and Accel	eration		? ×
	Mechanism:	Mechanism.1		-
	Name:	Speed-Accelerati	on.1	
	Reference product:	Main_Frame.1		
	Point selection:	Point.1		
	🥥 Main axis	O Other axis:		
			OK	Cancel
	The Speed-Accelera	tion.1 item is ide	entified ir	n the specific
h	Applications			



- 8. Repeat Step 2.
- **9.** Click once in the point selection field and select Point.2 under Exit_Shaft either in the specification tree or in the geometry area

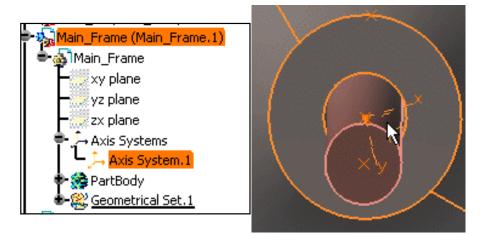


 Click once in the Reference product field and select Main_Frame.1 either in the specification or in the geometry area.



11. Select a Cartesian axis system for the projection results.

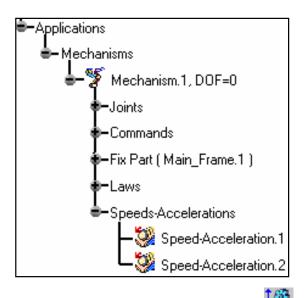
Select the **Other axis** option, click once in the field and select Axis System.1 under Main_Frame.1.



12. Click **Ok** to create the Speed-Acceleration.2 item

Speed and Acceleration	
Mechanism:	Mechanism.1
Name:	Speed-Acceleration.2
Reference product:	Main_Frame.1
Point selection:	Point.2
⊖ Main axis	Other axis: Axis System.1
	Cancel

The Speeds and accelerations are identified in the specification tree



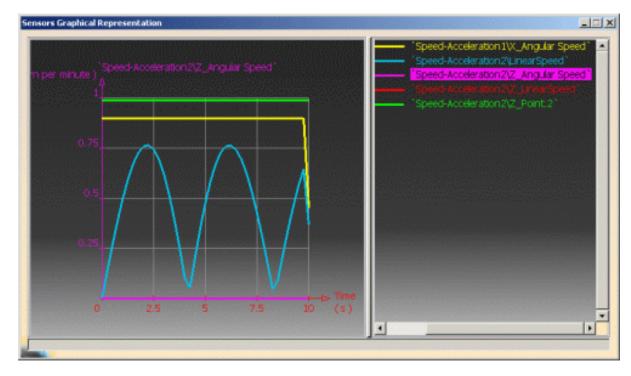
- **13.** Click the **Simulation With Laws** icon ^{ISS} in the DMU Kinematics toolbar
- **14.** Select the **Activate Sensors** option to display all measures during simulation.
- **15.** Select the sensors to be observed:
 - Speed-Acceleration.1\\X_AngularSpeed
 - $\circ \ Speed-Acceleration.2 \ \ Z_Point.2$
 - $\circ \ Speed-Acceleration. 2 \ \ Z_LinearSpeed$
 - \circ Speed-Acceleration.2\\LinearSpeed
 - Speed-Acceleration.2\\Z_Angular Speed

`Speed-Acceleration 1\Y_IRC' Milimeter No `Speed-Acceleration 1\Z_IRC' Milimeter No `Speed-Acceleration 2\X_Point.2' Milimeter No `Speed-Acceleration 2\X_Point.2' Milimeter No `Speed-Acceleration 2\X_Point.2' Milimeter No `Speed-Acceleration 2\X_LinearSpeed' Meter per second No `Speed-Acceleration 2\X_LinearSpeed' Meter per second No `Speed-Acceleration 2\X_LinearSpeed' Meter per second Yes `Speed-Acceleration 2\X_LinearSpeed' Meter per second Yes `Speed-Acceleration 2\X_LinearSpeed' Meter per second Yes `Speed-Acceleration 2\X_Linear Acceleration' Meter per square sec No `Speed-Acceleration 2\X_Angular Speed' Turn per minute No `Speed-Acceleration 2\X_Angular Speed' Turn per minute No `Speed-Acceleration 2\X_Angular Speed' Turn per minute	-
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`Speed-Acceleration.2'/Z_Angular Speed` Turn per minute Yes `Speed-Acceleration.2'/Z_Angular Speed` Turn per minute No `Speed-Acceleration.2'/Z_Angular Acceleration` Radian per square se No	
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"Speed-Acceleration 21X_Angular Acceleration" Radian per square se No	
Speed-Acceleration 21/Y_Angular Acceleration Radian per square se No Speed-Acceleration 21/Z_Angular Acceleration Radian per square se No	
Speed-Acceleration 21/Y_Angular Acceleration Radian per square se No Speed-Acceleration 21/Z_Angular Acceleration Radian per square se No	
"Speed-Acceleration.21/2_Angular Acceleration" Radian per square se No	
"Speed-Acceleration.2%_IRC" Millimeter No	-

- **16.** Run your simulation with laws
- **17.** The Speed and Acceleration result parameters are logged (22 Measures are available, including linear and angular speed and acceleration (their projections on the reference axis chosen + their

magnitude. Besides, the coordinates of the computation point are available too.

18. Click the **Graphics** button from the **outputs** area to obtain a graphical representation



- **19.** Click **Close**.
- **20.** Open MeasureSpeedAcceleration_Result.CATProduct document to check your results.



Other Analyses





Sensors->Check Limits

Calculating Distances Detecting Clashes in V4 Detecting Clashes in V5 Detecting Clashes Automatically in V4 Detecting Clashes Automatically in V5 Checking Joint Limits

Calculating Distances

This task shows how to calculate distances between two products.

Insert the KIN_EX17* .model files from the samples folder. The kinematics document must be already opened. You already defined a simulation. For more information, please refer to Recording Positions.

1. In the specification tree, click KIN_EX17_06_CENTRAL_DOOR then control-click

KIN_EX17_09_LEFT_DOOR.

The two items are selected and highlighted in the specification tree.

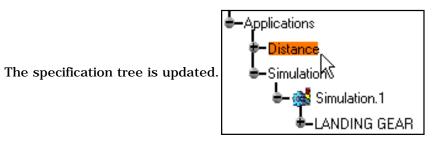
Product1	
— 6 KIN_EX17_16_ENS16 (KIN_EX17_16_ENS16.1)	
KIN_EX17_00_Mirage_F1_ACTIVE_MODEL (KIN_EX17_00_	Mirage_F1_ACTIVE_MODEL.1)
- 6 KIN_EX17_01_ENS1 (KIN_EX17_01_ENS1.1)	
—— 6 KIN_EX17_02_ENS2 (KIN_EX17_02_ENS2.1)	
- 6 KIN_EX17_03_ENS3 (KIN_EX17_03_ENS3.1)	
- 6 KIN_EX17_05_ENS5 (KIN_EX17_05_ENS5.1)	
- KIN_EX17_06_CENTRAL_DOOR (KIN_EX17_06_CENTRAL	_DOOR.1)
	1
- 6 KIN_EX17_10_ENS10 (KIN_EX17_10_ENS10.1)	
- 6 KIN_EX17_11_ENS11 (KIN_EX17_11_ENS11.1)	_
- 6 KIN_EX17_12_ENS12 (KIN_EX17_12_ENS12.1)	\mathcal{A}
- 6 KIN_EX17_13_OPENING (KIN_EX17_13_OPENING.1)	
- 6 KIN_EX17_14_ENS14 (KIN_EX17_14_ENS14.1)	
- 6 KIN_EX17_15_ENS15 (KIN_EX17_15_ENS15.1)	
- 🎭 Product2 (Product2.1)	
⊕—Applications	
••	

- Click the Distance and Band Analysis icon in the DMU Space Analysis toolbar, or select Insert -> Distance from the menu bar to calculate distances. The Edit Distance And Band Analysis dialog box is displayed.
- 3. Ensure that the first Type drop-down list box is set to Minimum and Inside one selection

Edit Dista	nce and Band Ana	lysis	? ×
Definitio	n		
Name:	Distance.1		
Туре:	Minimum	Selection 1:	2 products
	Inside one selection	Selection 2:	No selection
n da da kun kari		Minimum distanc	e: 1mm
Accuracy:	5mm	Maximum distance	ce: 2mm
Results	<u></u>		
Distance	Omm		
Delta	× Omm	Y Omm	Z Omm
Point 1	× 946.077mm	Y -640.908mm	Z -1885.617mm
Point 2	× 946.077mm	Y -640.908mm	Z -1885.617mm
Point 1 on KIN_EX17_09_LEFT_D00R.1			
Point 2 or	KIN_EX17_06_CEN	NTRAL_DOOR.1	
		🕥 ОК	Apply Scancel

i) The default distance analysis is measuring the minimum distance inside one selection.

- 4. Click Apply
- 5. Click OK.



- **6.** Double-click the Simulation.1 in the specification tree. The Edit Simulation dialog box is displayed.
- 7. Click the Edit Analysis button.

Edit Simulation
Name: Simulation.1
Animate viewpoint
Insert Modify Delete Skip
Interference Distance
Off 🔽 Off 🔽
Edit analysis Edit simulation objects
K OK Cancel

The Edit Analysis In Simulation dialog box is displayed:

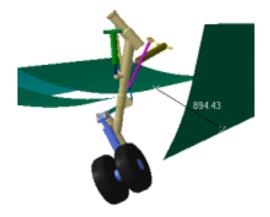
8. Click Add then select **Distance1** from the displayed pop-up.

Select	Edit Analysis in Simulation 🛛 💽 🗙
Distance.1	No Name
OK Cancel	Browse Adr. Remove
	Edit Analysis in Simulation ? × No Name 1 Distance.1
The Edit Simulation dialog box is updated.	Browse Add Remove

9. Set the Distance combo to On in the Edit Simulation dialog box.

The specification tree is updated.

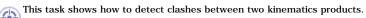
10. In the Kinematics Simulation dialog box, run a step by step simulation using the Use Laws tab. The minimum distance between the two products is displayed at each step.



Please refer to the DMU Space Analysis User's Guide for more information about detecting and analyzing distances between products or between groups.



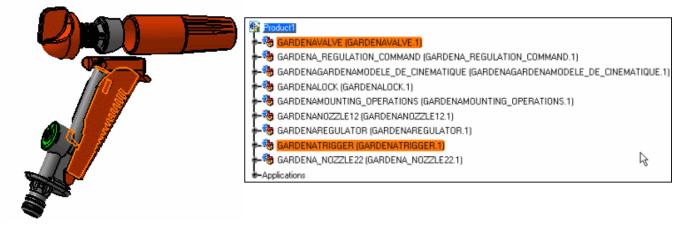
Detecting Clashes In V4



Open CLASH_DETECTION.CATProduct document.

1. In the specification tree, select GARDENAVALVE then control-click GARDENATRIGGER

The two items are highlighted in the specification tree and in the geometry area.



2. Click the Clash icon

The Check Clash dialog box is displayed. Make sure the interference type is set to Contact + Clash and Inside one selection.

Check	Clash		? ×
Def	inition		
Name	Interference.1		
Type:	Contact + Clash	▼ 5mm	Selection 1: 2 products
	Inside one selection	-	Selection 2:
			OK 📔 🌢 Apply 📔 🥥 Cancel

3. Click Apply, when done Ok.

The specification tree is updated.	
-Applications	
-Interference	
L 🍇 Interference.1	

4. Double-click Simulation.1 in the specification tree.

The Edit Simulation and Kinematic Simulation dialog boxes are displayed.

5. Click Edit Analysis in the Edit Simulation dialog box.

The Edit Analysis in Simulation dialog box is displayed.

Edit Analysis in Simulation			
No	Name		
	Browse	Adr	. Remove
		h	3
_		🌖 ОК	Cancel

6. Click Add then select interference 1 from the displayed Select dialog box

Select

Interference

 \mathbb{Q}

	×
1	

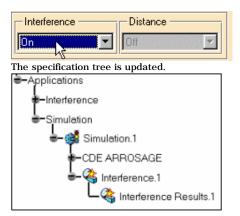
🥥 Cancel

updated:

🕒 ОК |

The Edit Analysis	in Simulatio	
Edit Analysis ir	h Simulation	? ×
No Name 1 Interferer	nce.1	
Browse	Add	Remove
	🎱 ОК	Cancel

- 7. Click OK to confirm your operation. You defined an interference.
- 8. Set the Interference combo to On.



9. To locate the clash position more precisely, set the Interference combo to Stop in the Edit Simulation dialog box

The simulation stops at the position where a collision is detected between GARDENAVALVE and GARDENATRIGGER products.

Edit Simulation ? 🗙
Name: Simulation.1
· · · · · · · · · · · · · · · · · · ·
34.00 🚔 1
Animate viewpoint
Insert Modify Delete Skip
Automatic insert
Interference Distance
Stop
Edit analysis Edit simulation objects
Edit sensors
OK SCANCE

The products in c	ollision are highlighted.
Edit Simulation	

10. Click Edit Analysis in the Edit Simulation dialog box. The Edit Analysis in Simulation appears.



- 11. Click Browse.
- 12. The Check Clash dialog box is displayed. The specification tree is updated

heck (? >
Defini				_						
	Interference.1			-						
Type:	Clash			Omm	Selectio	on 1: 2 proc	lucts			
	Inside one se	lection		•	Selectio	on 2:				
Resul	lts —									
📳 Nu	umber of interf	erences:	1 (Clash:1, Conta	ict:0, Clea	rance:0)					
1.1.1.1.1.1.1.1	t: All types		o filter on value	line in the second	I statuses	T 4				
	a second a second a second	and the second)· ·			ippy more			
List t	by Conflict	List by I	Product							1
No.	Product 1		Product 2		Туре	Value	Status	Кеер	Comment	
1	KIN_EX17	_09	KIN_EX17_01_	ENS1.1	Clash		Not inspect			
										More >>
							۵ ۵	к	Apply	🥥 Cancel

Please refer to the DMU Space Analysis User's Guide for more information about detecting and analyzing interferences between products or between groups.



Detecting Clashes In V5

This task shows how to detect clashes between two kinematics products.

Open the DETECT_CLASH_V5.CATProduct document.

- **1.** Click the Clash icon Section 2. The Check Clash dialog box is displayed. An entry for the interference appears in the specification tree.
- Keep the default computation type (Contact + Clash) and activate the second Type drop-down list box to select between two selections type
- **3.** Select the products to check for interference either in the specification tree or in the geometry area:
 - Selection1: Front_Moving_Arm.1
 - Selection2: Arm_Joint.1 and Arm_Joint.2

C	heck	Clash		? ×
	— Defir	nition		
	Name:	Interference.1		
	Туре:	Contact + Clash	The second secon	Selection 1: 1 product
		Between two selections		Selection 2: 2 products
				🔵 OK 💽 Apply 🚺 🥥 Cancel

4. Click Apply, when done Ok.

The specification tree is updated.



- **5.** Click the Simulation With Laws icon in the DMU Kinematics toolbar. The Kinematic Simulation Mechanism. 1 dialog box is displayed:
- 6. Check the Activate Sensors option. The Sensors dialog box is automatically displayed
- **7.** Click the Selection tab and select the sensor Interference.1\Nbclash for this:
 - Select the Interferences option button
 - Click the **Stop** button



- 8. (optional) Click the Analysis button in the Kinematic Simulation -Mechanism.1 dialog box
 - select Interference1
 - click **Browse** to check the interference specification The Edit Analysis appears:

E	dit A	nalysis	? ×
	No 1 2 3	Name MeasureBetween.369 Distance.1 Interference.1	
		Browse	se

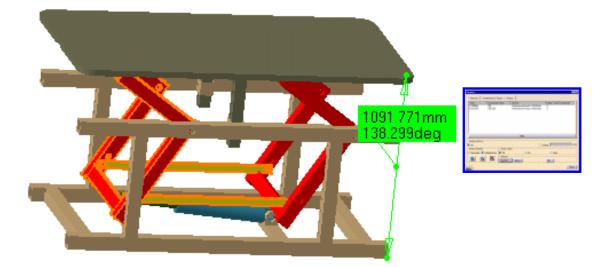
The Check Clash dialog box is displayed

Check	Clash				? ×
Defi	nition				
[1] [2] [2] [2] [2] [2] [2] [2] [2] [2] [2	Interference.1	0.05			
Type:	Contact + Clash	▼ 0mm		Selection 1: 1 p	
	Between two selections	-		Selection 2 2 p	roducts
	umber of interferences: 2 (Clash:2, Contact:0,				
Filter li	st: All types 💽 No filter on value	-	All statuses	<u> </u>	
List	by Conflict List by Product Matrix				
No.		Туре	Value	Status	Comment
1	Front_Moving_A Arm_Joint (Arm Front_Moving_A Arm_Joint (Arm	Clash Clash		Not inspect Not inspect	
2	Pronc_Moving_A Arm_Joint (Arm	Liash		Not inspect	
•					F
				Deselec	t More >>
			OK	Apply	Cancel

 Click Ok, when done, click Close in the Edit Analysis dialog box. Back in the Sensors dialog box, click the History tab to visualize the sensors behavior while running your simulation:



The parts in collision are highlighted in the geometry area and in the specification tree



Please refer to the *DMU Space Analysis User's Guide* for more information about detecting and analyzing interferences between products or between groups.



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Detecting Clashes Automatically in V4

This task shows you how to use the Clash Detection functionality while performing a Kinematics simulation.

Open AUTO_CLASH_DETECTION.CATProduct document.

1. Click the Simulation With Commands icon ¹/₂ from the DMU Kinematics toolbar.

The Kinematic Simulation dialog box is displayed. Select GARDENA as mechanism.

Kinematic Simulation - GARDENA								? ×
Mecha	anism : [GARDENA	4					•
C1	-345			375	15.0000	\$		-
C2	-363			357	-3.0000	-		
C3	-345		-		15.0000	-		
C4	-355		<u> </u>	365	5.0000	-		
Ch	ieck Joir	nt Limits						
Res	et		An	alysis.			Mor	e>>
							Clo	se

- **2.** Click More >> to expand the dialog box.
- **3.** Activate the On request mode.

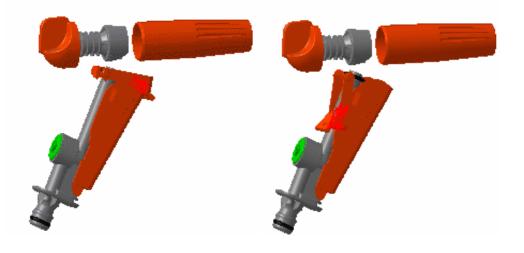
Kinema	atic Si	mulat	ion - l	GARD	ENA				? ×
Mecha	anism :	GAR)ENA						-
C1	-345	i	-1		375	15.0000	-		-
C2	-363		-1		_ 357	-3.0000	-		
C3	-345	·	-1		375	15.0000	÷		
C4	-355		-11-		365	5.0000	-		
ПCh	eck Jo	int Lim	its						
Res	Check Joint Limits Reset Analysis Simulation						.ess		
O Im	mediat	e 🥥 (On req	uest					
		K							
	Number of steps: 40								
								Clo	ise)

4. Click the arrow within the Clash Detection icon from the DMU Generic Animation toolbar. Undock the toolbar if necessary.

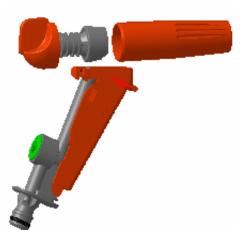


- **5.** Set the Clash detection to on \mathbf{M}
- 6. Move slider to 116 for command 3 (C3)
- 7. Run your simulation

The clash is highlighted in the geometry area



- 8. Now set the clash detection on Stop mode
- 9. Run your simulation. This time, the simulation stops at the first clash detected.





If you need to obtain a finer clash analysis, you need to define a interference, please refer to Detecting Interferences





Detecting Clashes Automatically in V5

This task shows you how to use the Clash Detection functionality while performing a Kinematics simulation.



Open the DETECT_CLASH_V5.CATProduct document.

1. Click the Simulation With Commands icon in the DMU Kinematics toolbar. The Kinematics Simulation dialog box is displayed.

Kinematic Simulation - Mechanism.1					
Mechanism :	Mechanism.1	•			
Command.1	-100	.			
Activate	Sensors				
Reset	Analysis	(More>>			
		Close			

- **2.** Click $\stackrel{\text{More }>>}{\longrightarrow}$ to expand the dialog box.
- **3.** Activate the On request mode.
- 4. Change the number of steps to 20
- 5. Move the slider to the end

Kinematic Simulation - Mechanism.1	? ×
Mechanism : Mechanism.1	•
Command.1 -100351 351.0000	.
Activate Sensors	
Reset Analysis	< <less< td=""></less<>
Simulation	
🔿 Immediate 🥥 On request	
Number of steps: 20	
	Close

6. Check the Activate Sensors option

The Sensors dialog box is automatically displayed

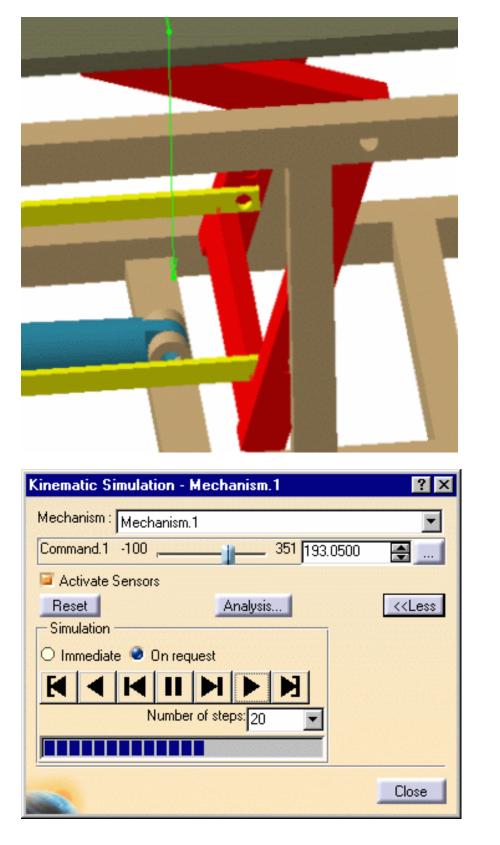
 In the Detect Clashes section, Automatic option is set by default. Set the Clash detection to Stop.

- Detect C	lashes —			
Automatic O Interferences				
		<u>.</u>		

8. Launch your simulation with commands using the Play forward button

The simulation is stopped once a clash is detected





i If you need to obtain a finer clash analysis, you need to define a interference, please refer to Detecting Interferences



Checking Mechanism Joint Limits

- This task consists in checking joint limits during simulation
 Open the CHECKING_LIMITS.CATProduct document. Remember you set joint limits in the previous task
 1. Click the Simulation With commands icon
 from the DMU Kinematics toolbar. The Kinematic Simulation Mechanism.1 dialog box is displayed:
 - **2.** Click the Sensors check button.

simulation commands

- **3.** In the Sensors dialog box displayed set the check limits mode (click the appropriate option button)For instance set the **stop** mode
- **4.** Select the joints to be observed: Prismatic.2 and Revolute.3 (use the **Selection** tab and select the joints in the sensor list)
 - Remember, you set limits on Prismatic.2 in the previous task:
 - Lower limit -10mm
 - Upper limit 10mm

Note: because a command is assigned to Revolute.3, the limits are necessarily set. The default values for angle limits are:

- Lower limit -360deg
- Upper limit 360deg

The clash detection is available within the **Sensors** dialog box.

Sensors				? ×
Selection Instantaneous	Values History			
Sensor	Unit	Observed		
Mechanism.1\Joints\Prismati Mechanism.1\Joints\Revolut Mechanism.1\Joints\Revolut	e.3\Angle Degree	Yes Yes No		
Deselec	t All		Select All	
Display Options				
I All		C	Limited 1	Lines
Detect Clashes	Check Limits			
Automatic O Interferences	O Off	O On	Stop	
6	Outputs Graphics Options	1	File	
				Close

- **5.** Click the **History** tab
- **6.** Manipulate the slider of the command or use the manipulator in the geometry area.

Note: in direct manipulation context (using the manipulators) the simulation is stopped each time a limit is reached.

In both cases (slider manipulation or direct manipulation) the comments column is updated each time a limit is reached.

Event	Commands Value	Sensors	Value	Unit	Comment
2	72	Mechanism.1\Joints\Prismatic.2\Le	6.3	Milli	
2	72	Mechanism.1\Joints\Revolute.3\A	72	Deg	
3	86.4	Mechanism.1\Joints\Prismatic.2\Le	8.8	Milli	
3	86.4	Mechanism.1\Joints\Revolute.3\A	86.4	Deg	
4	93.6	Mechanism.1\Joints\Prismatic.2\Le	10	Milli	Upper Limits Reached
4	93.6	Mechanism.1\Joints\Revolute.3\A	93.6	Deg	Upper Limits Reached
5	122.4	Mechanism.1\Joints\Prismatic.2\Le	14	Mili	Upper Limits Reached
5	122.4	Mechanism.1\Joints\Revolute.3\A	122.4	Deg	Upper Limits Reached
6	172.8	Mechanism.1/Joints/Prismatic.2/Le	19	Milli	
6	172.8	Mechanism.1\Joints\Revolute.3\A	172.8	Deg	
•					•
		Clear			
Display Op	tions				
All			0	Limited	1
Detect Clas	shes	Check Limits			
Automatic	c O Interferences	O Off O On			Stop
		- Outputs			
		Graphics Options		-	le

- 7. Click Clear if needed
- 8. Click Close to exit the command



Measures

Additional tools:



Measuring Properties Measuring Distances between Geometrical Entities

Measuring Properties

The Measure Item command lets you measure the properties associated to a selected item (points, edges, surfaces and entire products).

This section deals with the following topics:

Measuring properties Measuring in a local axis system Customizing the display Editing measures Create Geometry from measure results Exact measures on CGRs and in visualization mode Associative measures Using measures in knowledgeware Measure cursors

Insert the following sample model files: ATOMIZER.model, BODY1.model, BODY2.model, LOCK.model, NOZZLE1.model, NOZZLE2.model, REGULATION_COMMAND.model, REGULATOR.model, TRIGGER.model and VALVE.model.

They are to be found in the online documentation filetree in the common functionalities sample folder cfysa/samples. Restriction: Neither Visualization Mode nor cgr files permit selection of individual vertices.

Note: In the No Show space, this command is not accessible.

Measuring Properties

This task explains how to measure the properties associated to a selected item.

1.Switch to Design Mode (Edit ->Representations ->Design Mode).

2.Set View -> Render Style to Shading with Edges.

Note: You cannot use this command, if Shading only is selected.

³.Click the Measure Item **4** icon.

In DMU, you can also select Analyze -> Measure Item from the menu bar.

The Measure Item dialog box appears.

Measure Item	? ×
	2
Selection 1 mode: Any geometry Selection 2 mode: Any geometry	
Calculation mode: Exact else approximate	
Results	
Keep Measure Create Geometry Custom	nize
	ancel

By default, properties of active products are measured with respect to the product axis system. Properties of active parts are measured with respect to the part axis system.

Note: This distinction is not valid for measures made prior to Version 5 Release 8 Service Pack 1 where all measures are made with respect to the absolute axis system.

Dialog box options

- You can also measure properties with respect to a local V5 axis system.
- The Keep Measure option lets you keep current and subsequent measures as features. This is useful if you want to keep measures as annotations for example.

Some measures kept as features are associative and can be used to valuate parameters or in formulas. In the Drafting workbench, measures are done on-the-fly. They are not persistent. This means that they are not associative and cannot be used as parameters.

- A Create Geometry option in the dialog box lets you create the center of gravity from measure results.
- A Customize... option lets you customize the display of measure results.

Accessing other measure commands

- The Measure Between command is accessible from the Measure Item dialog box. Simply click one of the Measure Between icons in the Definition box to switch commands.
- In DMU, the Measure Thickness command is also accessible from the Measure Item dialog box. For more information, see the appropriate task in the *DMU Space Analysis User's Guide*.

M P1-Only Functionality

In P1, the Measure Tools toolbar appears. This toolbar has two icons:



Between

Chain

Fan

- Measure Dialogs 1 iets you show or hide the associated dialog box.
- Exit Measure $\mathbf{1}$: lets you exit the measure. This is useful when the dialog box is hidden.

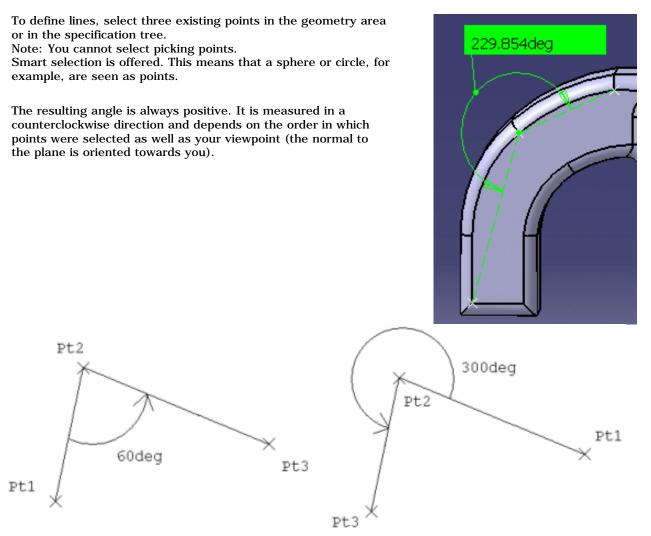
4.Set the desired measure mode in the Selection 1 mode drop-down list box. Defining the Selection 1 Mode

- Any geometry (default mode): measures the properties of the selected item (point, edge, surface or entire product).
- Point only: measures the properties of points. Dynamic highlighting is limited to points.
- Edge only: measures the properties of edges. All types of edge are supported.
- Surface only: measures the properties of surfaces.

In the last three modes, dynamic highlighting is limited to points, edges or surfaces depending on the mode selected, and is thus simplified compared to the Any geometry mode.

- Product only: measures distances between products.
 Products can be specified by selecting product geometry, for example an edge or surface, in the geometry area or the specification tree.
 - Angle by 3 points: measures the angle between two lines themselves defined by three points.

i



- Thickness (DMU only): measures the thickness of an item. For more information, see the appropriate task in the *DMU Space Analysis User's Guide*.
- The Measure Item command lets you access the radius of an exact cylinder or sphere.
- The Measure Item command also recognizes ellipse-type conic sections.
 Description:
 Ellipse in Part1.1
- Using the Other Selection... command in the contextual menu, you can access the axis of a cylinder as well as the center of a sphere to, for example, measure between two cylinder axes.

5.Set the desired calculation mode in the Calculation mode drop-down list box. Defining the Calculation Mode

- Exact else approximate (default mode): measures access exact data and wherever possible true values are given. If exact values cannot be measured, approximate values are given (identified by a ~ sign).
- Exact: measures access exact data and true values are given. Note that you can only select exact items in the geometry area or specification tree. In certain cases, in particular if products are selected, a warning dialog box informs you that the exact measure could not be made.
- Approximate: measures are made on tessellated objects and approximate values are given (identified by a ~ sign).

Note: You can hide the ~ sign using the **Tools** -> **Options** command (**General** ->**Parameters** and **Measure** ->**Measure** Tools).

6.Click to select the desired item.

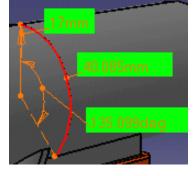
Note: The appearance of the cursor has changed to assist you.

Area=0.003m2
Measure Item ? 🗙
Definition
Selection 1 mode: Any geometry Selection 2 mode: Any geometry
Other Axis : No selection
Calculation mode: Exact else approximate
Results
Calculation mode: Exact Selection: Surface in BODY1.1
Surface area: 0.003m2
Keep Measure Create Geometry Customize
Cancel

The dialog box gives information about the selected item, in our case a surface and indicates whether the result is an exact or approximate value. The surface area is also displayed in the geometry area.

The number of decimal places, the display of trailing zeros and limits for exponential notation is controlled by the Units tab in the Options dialog box (**Tools**-> **Options**, **General**-> **Parameters and Measure**). For more information, see the Infrastructure User's Guide.

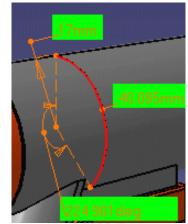
7. <u>Try selecting other items to me</u>asure associated properties.





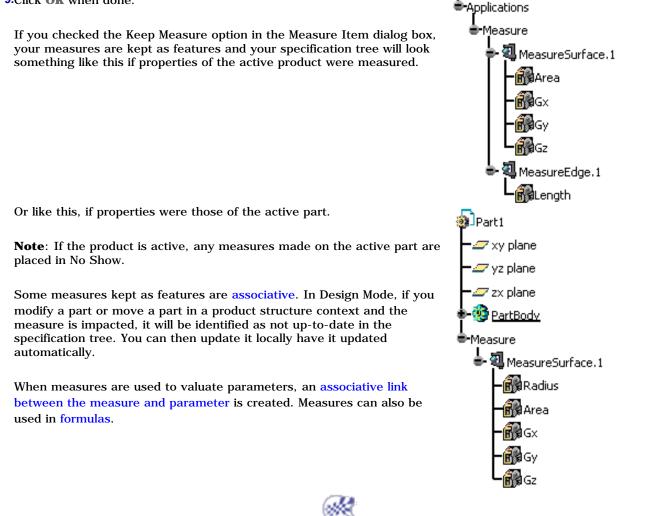
A

8.If necessary, adjust the presentation of the measure: You can move the lines and text of the measure.



The Properties command (Graphics tab) lets you change the fill color and transparency as well as the color, linetype and thickness of measure lines.

Note: You cannot vary transparency properties, the current object is either the selected color or transparent. 9.Click **OK** when done.



Customizing the Display

Customizing lets you choose the properties you want to see displayed in both the geometry area and the dialog box.

1.Click Customize... in the Measure Item dialog box to see the properties the system can detect for the various types of item you can select. By default, you obtain:

Measure	e Item Cu	stomization			×
Point		Edge	Arc	-Surface	Volume
🔎 Des	cription 📔	Description	Description	Description	Description
🧧 Poin	t 🕨	Length	Length	🖼 Area	🔎 Volume
		Point 1	Angle	Center of gravity	🗌 Area
		Point 2	🔎 Radius	🗌 Plane	Center of gravity
		Direction Vector	Diameter	Perimeter	
			Point 1		
			Point 2		
			Center point		
				ОК	Apply Close

Edges

The system detects whether the edge is a line, curve or arc, taking model accuracy into account and displays the properties as set in the Measure Item Customization dialog box.

Results			
Calculation mode:	Exact		
Selection:	Line in BODY1.1		
Length:	106.87mm		
Point 1:	×120.8mm	Y <mark>14.498mm</mark>	Z <mark>-105.735mm</mark>
Point 2:	×75.635mm	Y14.498mm	Z <mark>-8.877mm</mark>
Direction vector:	×-0.422618	<mark>Y</mark> 0	Z <mark>0.906308</mark>

Note: If the angle of an arc is less than 0.125 degrees, only the arc length is displayed in the geometry area. The angle and radius are not displayed.

Surfaces

- **Center of gravity**: The center of gravity of surfaces is visualized by a point. In the case of non planar surfaces, the center of gravity is attached to the surface over the minimum distance.
- **Plane**: gives the equation of a planar face. The equation of a plane is: Ax + By + Cz + D=0.

Results				
Calculation mode:	Exact			
Selection:	Plane in LOCK.1			
Surface area:	2.802e-004m2			
Plane: /	No.	B 1	C <mark>O</mark>	D <mark>-15.55</mark>
Perimeter:	111.061mm			

• Perimeter: Visualization mode does not permit the measure of surface perimeter.

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- Results	
Calculation mode:	Exact
Selection:	Surface in BODY1.1
Surface area:	0.003m2
Perimeter:	285.091mm

2.Set the properties you want the system to detect, then click **Apply** or **Close**.

The Measure Item dialog box is updated if you request more properties of the item you have just selected. **3**.Select other items to measure associated properties.



Measuring Properties in a Local Axis System

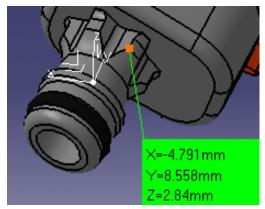
An Other Axis option in the dialog box lets you measure properties in a local axis system.

This type of measure is associative: if you move the axis system, the measure is impacted and can be updated. You will need a V5 axis system.

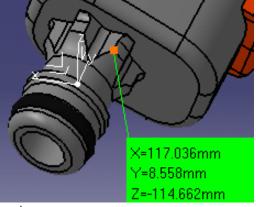
1.Select the Other Axis checkbox in the Measure Item dialog box.

2.Select a V5 axis system in the specification tree or geometry area. 3.Make your measure.

Measure made with respect to local axis system:



Same measure made with respect to absolute axis system:



Note: All subsequent measures are made with respect to the selected axis system. 4. To change the axis system, click the Other Axis field and select another axis system. **5.**To return to the main axis system, click to clear the Other Axis checkbox. 6.Click OK when done.



Editing Measures

In addition to editing the presentation of the measure, you can also edit the measure itself and change the selection on which it was based. This is particularly useful in design mode where you no longer have to redo your measure. You can also change selections that no longer exist because they were deleted.

1. Double-click the measure in the specification tree or geometry area.

2.Make a new selection.

Note:

You cannot change the selection 1 mode. If you selected a curve, you must make a selection of the same type, i.e. another curve.



Measuring Distances between Geometrical Entities

🕋 The Measure Between command lets you measure distance between geometrical entities. You can measure:

- Minimum distance and, if applicable angles, between points, surfaces, edges, vertices and entire products Or.
 - Maximum distance between two surfaces, two volumes or a surface and a volume.

This section deals with the following topics:

Measuring minimum distance and angles Measuring maximum distance Measuring distances in a local axis system Customizing measure between Editing measures Creating geometry from measure results Exact measures on CGRs and in visualization mode Measuring exact angles Associative measures Using measures in knowledgeware Measure cursors

Insert the following sample model files: ATOMIZER.model, BODY1.model, BODY2.model, LOCK.model, NOZZLE1.model, NOZZLE2.model, REGULATION_COMMAND.model, REGULATOR.model, TRIGGER.model and VALVE.model.

They are to be found in the online documentation filetree in the common functionalities sample folder cfysa/samples.

Restriction: Neither Visualization Mode nor cgr files permit selection of individual vertices.

Note: In the No Show space, the Measure Between command is not accessible.

Measuring Minimum Distance and Angles

This task explains how to measure minimum and, if applicable, angles between geometrical entities (points, surfaces, edges, vertices and entire products).

Click the Measure Between 🚔 icon.

In DMU, you can also select Analyze-> Measure Between from the menu bar.

The Measure Between dialog box appears.

Measure Between 🛛 🔋 🗙
Definition
Selection 1 mode: Any geometry
Selection 2 mode: Any geometry
Other Axis : No selection
Calculation mode: Exact else approximate
Results
Calculation mode:
Selection 1:
Selection 2:
Minimum distance:
Angle:
Keep Measure Create Geometry Customize
Cancel

By default, minimum distances and if applicable, angles are measured.

By default, measures made on active products are done with respect to the product axis system. Measures made on active parts are done with respect to the part axis system.

Note: This distinction is not valid for measures made prior to Version 5 Release 8 Service Pack 1 where all measures are made with respect to the absolute axis system.

Dialog box options

- You can also measure distances and angles with respect to a local V5 axis system.
- A Keep Measure option in the dialog box lets you keep the current and subsequent measures as features. This is useful if you want to keep the measures as annotations for example.

Some measures kept as features are associative and can be used to valuate parameters or in formulas. In the Drafting workbench, measures are done on-the-fly. They are not persistent. This means that they are not associative and cannot be used as parameters.

- A Create Geometry option in the dialog box lets you create the points and line corresponding to the minimum distance result.
- A Customize... option opens the Measure Between Customization dialog box and lets you set the display of measure results.

Accessing other measure commands

- The Measure Item command 🔄 is accessible from the Measure Between dialog box.
- In DMU, the Measure Thickness command is also accessible from the Measure Between dialog box. For more information, see the *DMU Space Analysis User's Guide*.



P1-Only Functionality

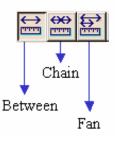
In P1, the Measure Tools toolbar appears.

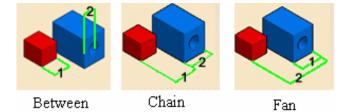
This toolbar has two icons:



- Measure Dialogs **1**: lets you show or hide the associated dialog box.
- Exit Measure 🗋 : lets you exit the measure. This is useful when the dialog box is hidden.
- **2.** Select the desired measure type.

Notice that the image in the dialog box changes depending on the measure type selected.





Defining Measure Types

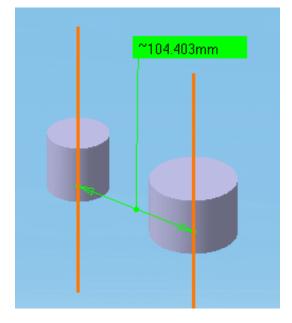
- Between (default type): measures distance and, if applicable, angle between selected items.
- Chain: lets you chain measures with the last selected item becoming the first selection in the next measure.
- Fan: fixes the first selection as the reference so that you always measure from this item.
- 3. Set the desired mode in the Selection 1 and Selection 2 mode drop-down list boxes.

Defining Selection 1 & Selection 2 Modes

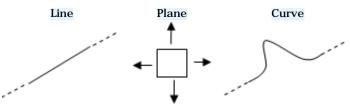
 Any geometry (default mode): measures distances and, if applicable, angles between defined geometrical entities (points, edges, surfaces, etc.).
 Note: The Are center mode is activated in this selection mode.

Note: The Arc center mode is activated in this selection mode.

This mode recognizes the axis of cylinders and lets you measure the distance between two cylinder axes for example.



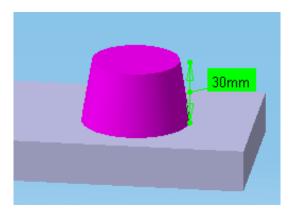
• Any geometry, infinite: measures distances and, if applicable, angles between the infinite geometry (plane, line or curve) on which the selected geometrical entities lie. Curves are extended by tangency at curve ends.

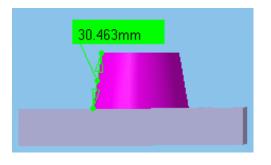


The Arc center mode is activated and this mode also recognizes cylinder axes. For all other selections, the measure mode is the same as any geometry.

Any geometry, infinite

Any geometry



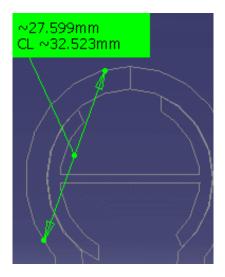


• Picking point: measures distances between points selected on defined geometrical entities. Always gives an approximate measure.

In the DMU section viewer, selecting two picking points on a curve gives the distance along the curve between points (curve length or CL) as well as the minimum distance between points.

Notes:

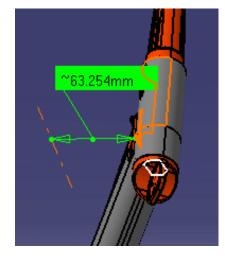
- Both points must be located on the same curve element.
- The minimum distance option must be set in the Measure Between Customization dialog box.



Results	
Calculation mode:	Approximate
Selection 1:	Point on Section.1
Selection 2:	Point on Section.1
Minimum distance:	27.599mm
Curve length:	32.523mm

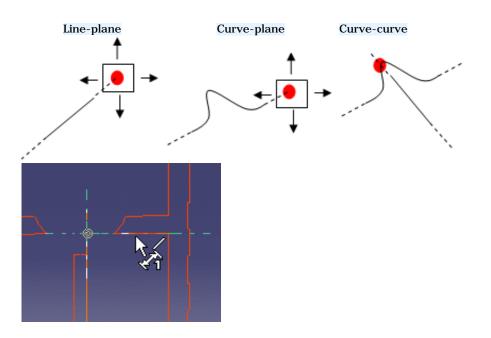
- Point only: measures distances between points. Dynamic highlighting is limited to points.
- Edge only, Surface only: measures distances and, if applicable, angles between edges and surfaces respectively. Dynamic highlighting is limited to edges or surfaces and is thus simplified compared to the Any geometry mode. All types of edge are supported.
- Product only: measures distances between products. Products can be specified by selecting product geometry, for example an edge or surface, in the geometry area or the specification tree.
- Picking axis: measures distances and, if applicable, angles between an entity and an infinite line perpendicular to the screen.

Simply click to create infinite line perpendicular to the screen.



• Intersection: measures distances between points of intersection between two lines/curves/edges or a line/curve/edge and a surface. In this case, two selections are necessary to define selection 1 and selection 2 items.

Geometrical entities (planar surfaces, lines and curves) are extended to infinity to determine the point of intersection. Curves are extended by tangency at curve ends.



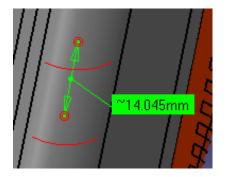
Note: Only intersections which result in points of intersection are managed.



- Edge limits: measures distances between endpoints or midpoints of edges. Endpoints only are proposed on curved surfaces.
- Arc center: measures distances between the centers of arcs.
- Center of 3 points arc: measures distances between the centers of arcs defined by 3 points.

To define arc center, click three points on the geometry.

Note: The resulting measure will always be approximate.



- Coordinate: measures distances between coordinates entered for selection 1 and/or selection 2 items.
- 4. Set the desired calculation mode in the Calculation mode drop-down list box.

Defining the Calculation Mode

- Exact else approximate (default mode): measures access exact data and wherever possible true values are given. If exact values cannot be measured, approximate values are given (identified by a \sim sign).
- Exact: measures access exact data and true values are given. Note that you can only select exact items in the geometry area or specification tree. In certain cases, in particular if products are selected, a warning dialog box informs you that the exact measure could not be made.
- Approximate: measures are made on tessellated objects and approximate values are given (identified by a ~ sign).

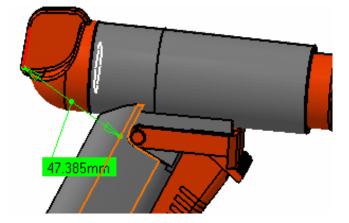
Note: You can hide the display of the ~ sign using the **Tools** -> **Options** command (**General** -> **Parameters and Measure** -> **Measure Tools**).

5. Click to select a surface, edge or vertex, or an entire product (selection 1).

Notes:

- The appearance of the cursor has changed to assist you.
- Dynamic highlighting of geometrical entities helps you locate items to click on.
- 6. Click to select another surface, edge or vertex, or an entire product (selection 2).

A line representing the minimum distance vector is drawn between the selected items in the geometry area. Appropriate distance values are displayed in the dialog box.



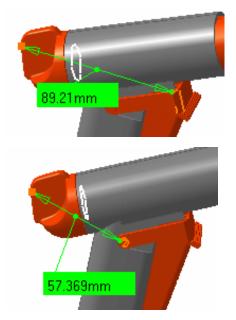
Measure Between 🔋 🗙
Definition
Selection 1 mode: Edge only
Selection 2 mode: Any geometry
Other Axis : No selection
Calculation Mode: Exact else approximate
Results
Calculation mode: Exact
Selection 1: Line on REGULATION_COMMAND.1
Selection 2: Surface in BODY1.1
Minimum distance: 47.385mm
Angle:
Keep Measure Create Geometry Customize
Cancel

By default, the overall minimum distance and angle, if any, between the selected items are given in the Measure Between dialog box.

The number of decimal places, the display of trailing zeros and limits for exponential notation is controlled by the Units tab in the Options dialog box (**Tools** ->**Options**, **General** ->**Parameters and Measure**). For more information, see the *Infrastructure User's Guide*.

- 7. Select another selection and, if desired, selection mode.
- 8. Set the Measure type to Fan to fix the first selection so that you can always measure from this item.
- **9.** Select the second item.

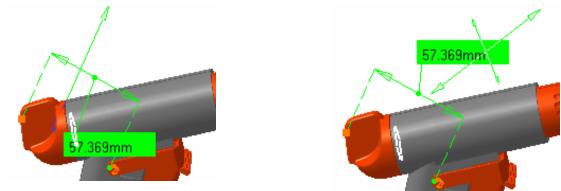
10.Select another item.



Using the Other Selection... command in the contextual menu, you can access the center of spheres. **11.**If necessary, adjust the presentation of the measure:

You can move the lines and text of the measure.

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The Properties command (Graphics tab) lets you change the fill color and transparency as well as the color, linetype and thickness of measure lines.

Note: You cannot vary transparency properties, the current object is either the selected color or transparent.

12.Click **OK** when done.

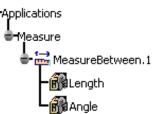
If you checked the Keep Measure option in the Measure Between dialog box, your measures are kept as features and your specification tree will look something like this if measures were made on the active product.

Or like this, if measures were made on the active part.

 $\ensuremath{\textbf{Note}}\xspace$. If the product is active, any measures on parts are placed in No Show.

Some measures kept as features are associative. In Design Mode, if you modify a part or move a part in a product structure context and the measure is impacted, it will be identified as not up-to-date in the specification tree. You can then update it locally have it updated automatically.

When measures are used to valuate parameters, an associative link between the measure and parameter is created. Measures can also be used in formulas.





Sectioning measure results

Having made and kept your measure, select it then click the Sectioning 🧭 icon to section measure results. The plane is created parallel to the direction defined by the measure and sections entities selected for the measure only. All section plane manipulations are available.

Note: You may need an appropriate license to access the Sectioning command.



Customizing Measure Between

1

Customizing lets you choose what distance you want to measure:

- Minimum distance (and angle if applicable)
- Maximum distance
- Maximum distance from 1 to 2.

Note: These options are mutually exclusive. Each time you change option, you must make your measure again.

By default, minimum distances and if applicable, angles are measured.

You can also choose to display components and the coordinates of the two points (point 1 and point 2) between which the distance is measured.

What you set in the dialog box determines the display of the results in both the geometry area and the dialog box.

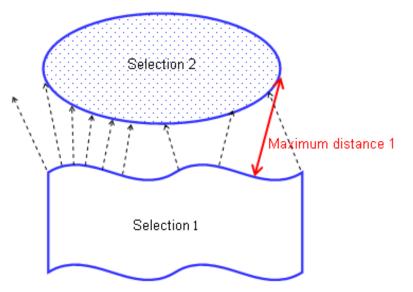
1easure Between Customizati 🗙
Minimum distance
🍯 Angle
Maximum distance
Maximum distance from 1 to 2
Components
Point 1
Point 2
OK Apply Close

Measuring Maximum Distance

You can measure the maximum distance between two surfaces, two volumes or a surface and a volume.

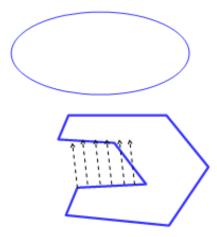
Distance is measured normal to the selection and is always approximate. Two choices are available:

• Maximum distance from 1 to 2: gives the maximum distance of all distances measured from selection 1. Note: This distance is, in general, not symmetrical.



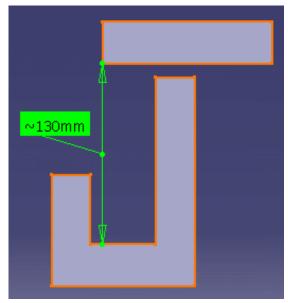
• Maximum distance: gives the highest maximum distance between the maximum distance measured from selection 1 and the maximum distance measured from selection 2.

Note: All selection 1 (or 2) normals intersecting selection 1 (or 2) are ignored.



1. Click Customize... and check the appropriate maximum distance option in the Measure Between Customization dialog box, then click **OK**.

- 2. Make your measure:
 - Select the desired measure type
 - Set the desired selection modes
 - Set the desired calculation mode
 - Click to select two surfaces, two volumes or a surface and a volume.



	Results	
1	Calculation mode:	Approximate
	Selection 1:	Body.2Part2.1
	Selection 2:	PartBodyPart1.1
	Maximum distance:	130mm

3. Click OK when done.

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Measuring Distances in a Local Axis System

An Other Axis option in the dialog box lets you measure distance in a local axis system.

This type of measure is associative: if you move the axis system, the measure is impacted and can be updated.

You will need a V5 axis system.

1. Select the Other Axis checkbox in the dialog box.

2. Select a V5 axis system in the specification tree or geometry area.

.

3. Make your measure.

In the examples below, the measure is a minimum distance measure and the coordinates of the two points between which the distance is measured are shown.

📮 Other Axis :	Axis System.1		
Calculation Mode:	Exact else approx	kimate 🗾	
-Results			
Calculation Mode: Selection 1: Selection 2: Minimum distance:	Exact Arc on REGUL/ Surface in LOCI 50.464mm		AND.1
Angle:			
Point 1:	×-11.395mm	Y2.63mm	Z <mark>140.304mm</mark>
Point 2:	×-48.839mm	Y15.55mm	Z 109.036mm

Same measure made with respect to absolute axis system:

Other Axis :	Axis System.1		
Calculation Mode:	Exact else appro	ximate 🔽	
Results			
Calculation Mode:	Exact		
Selection 1:	Arc on REGUL	ATION_COMMA	ND.1
Selection 2:	Surface in LOC	CK.1	
Minimum distance:	50.464mm		
Angle:			
Point 1:	×115.038mm	Y2.63mm	Z 12.922mm
Point 2:	×77.595mm	Y 15.55mm	Z <mark>-18.346mm</mark>

Note: All subsequent measures are made with respect to the selected axis system.

- 4. To change the axis system, click the Other Axis field and select another axis system.
- 5. To return to the absolute axis system, click to clear the Other Axis checkbox.
- 6. Click OK when done.



Editing Measures

In addition to editing the presentation of the measure, you can also edit the measure itself and change one of the selections on which it was based. This is particularly useful in design mode where you no longer have to redo your measure.

You can also change selections that no longer exist because they were deleted.



1. Double-click the measure in the specification tree or geometry area.

2. Make new selections.

Notes:

You can change selection modes when making new selections. For invalid measures where one selection has been deleted, you only have to replace the deleted selection. For all other measures, repeat all selections.

3. Click OK when done.



1

Digital Mockup Review

Reviewing Simulations Managing Kinematics Data in Sub-products Managing the Mechanism Dressup Defining a Swept Volume

Reviewing Simulations

DMU Kinematics Simulator provides easy methods to record and replay simulations.

Recording Positions Replaying Simulations Resetting a V5 Mechanism Sequencing Mechanisms with Laws

Recording Positions

This task shows how to record positions of a kinematics mechanism. ÷

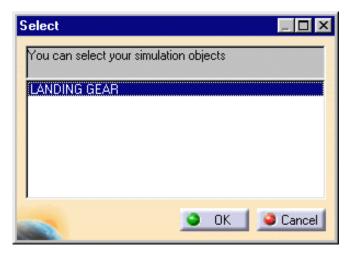
Insert the KIN_EX17* .model files from the samples folder.

At least one kinematics mechanism must be active in the specification tree.

1

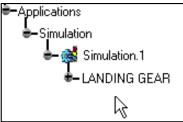
1. Click the Simulation icon

The Select dialog box is displayed.



2. Select LANDING GEAR and click OK

Kinematic Simulation and Edit Simulation dialog boxes appear. A Simulation object is created in the specification tree ..



Insert means that you record and insert positions inside the scenario.

Δ Note: the starting shot (initial position) is automatically recorded.

3. Click the Insert button in the Edit Simulation dialog box

Edit Simulation 🛛 💽 🗙
Name: Simulation.1
<u>u</u>
0.00
Animate viewpoint
Insert Modify Delete Skip
Automatic insert
Interference Distance
Off Off
Edit analysis Edit simulation objects
OK Cancel

Kinematic Simulation	- LANDING GEAR	? ×
Use 5 Commands	Use Laws	
CENTRAL -360	360 <mark>0.0000</mark>	
LEFT -360	360 0.0000	.
OPENING -350	370 10.0000	
INCLINAT -360	360 <u>0.0000</u>	
Reset		

- 4. Move the mechanism (using the manipulators or sliders, for example), then Click the Insert switch again.
- **5.** Record as many positions as necessary.

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6. Use the VCR buttons to replay the recorded positions.

This type of record can be used to simulate several mechanisms simultaneously.



Replaying Simulations

This task shows you how to create a simulation on a geometry of a part.

Insert the KIN_EX17* .model files from the samples folder. See Recording Positions. You then compiled the Simulation created as described in the previous task. Please refer to *Compiling a Simulation* in the *DMU Fitting Simulator User's Guide*



1. Activate the Simulation object in the specification tree.

2. Click the Replay icon

The Replay dialog box is displayed.

Replay ?	×
Name: Replay.1	-
🚺 🛄 💭 🚔 🛛 🗖	
Animate viewpoint	
Edit analysis	
Interference Distance	_

- **3.** Specify the desired speed for instance x 5.
- 4. Click:
 - $_{\odot}~$ the Play button to run a continuous replay of the recorded motion
 - or the Step button to run a step-by-step sequence of the recorded motion.

Each motion is replayed one after the other in the order they were recorded.

You can choose one of the loop modes to re-run the simulation in a continuous way (either in the one direction only or in one direction then the other).



 (\bullet)

Resetting a V5 Mechanism

This task shows how to use the reset command .When exiting the simulation with laws or with commands in DMU Kinematics, the modified

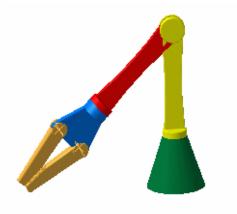
position is kept. You can need to swap to the initial product position, all you need to is click on the Reset Positions icon appropriate option.

In addition, when importing a sub-mechanism, a 'local copy' of the sub-mechanism is created, and if you simulate it, it becomes desynchronized with its reference. The reset command allows to re-synchronize an imported mechanism with its reference. You can also apply a particular state of an imported mechanism to its reference Open the Use_Laws.CATProduct document.

1. Click the simulation with laws icon.

Note: you can also choose to run a simulation with commands

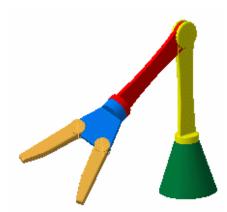
• Initial position when entering the simulation:



- 2. Run your simulation using the Play Forward button
- **3.** Click the pause button.

Kinematic Simulation - Mechanism.1 💦 🔀	
Mechanism : Mechanism.1	
Start 0 - 25 7.500	
Number of steps: Pause Analysis	
Activate Sensors	

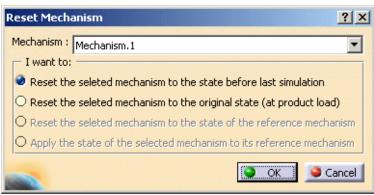
- **4.** Select **File** -> **Save...** from the File menu
- 5. Click Close to exit the Simulation With Laws command. The modified position is kept.



- 6. Click the Simulation With Laws icon. Run your simulation again using the Play Forward button
- 7. Exit the Simulation With Laws command without clicking the Start button **K**. The modified position is kept by default



8. Click the Reset Positions icon ²⁰⁰. The Reset Mechanism dialog box appears:



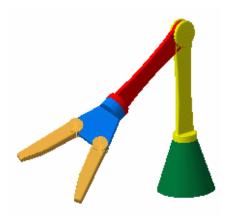
New options buttons appear in the Reset Mechanism dialog box:

 $_{\odot}~$ Reset the selected mechanism to the original state (at product load)

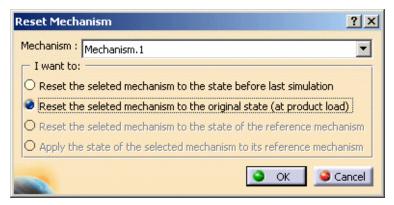
9. Apply the state of the selected mechanism to its reference mechanism (not available here, this option button is available if dealing with imported mechanisms)

The 'Reset the selected mechanism to the state before last simulation' option button is selected by default, keep it as it is.

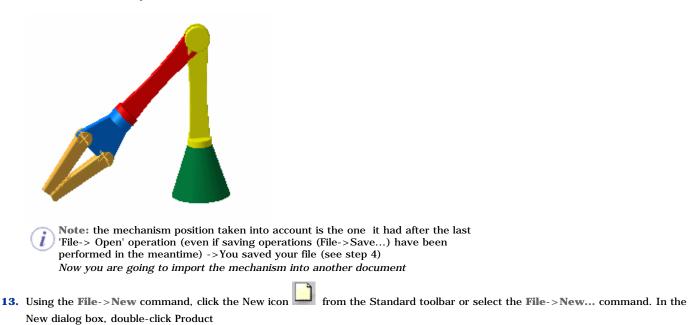
10. Click **Ok**. The mechanism goes back to the position it had before its last simulation:



11. Repeat step 8. and this time, clear the default option button and select 'Reset the selected mechanism to the original state (at product load)' option button.



12. Click **Ok**. This is what you obtain:



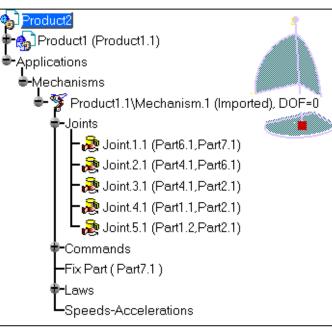
New	? ×
List of Types:	
FunctionalSystem	
Part Process	
ProcessLibrary	
Product	
ZipMill りく Selection:	<u> </u>
Product	
🤍 🎱 OK 🧕 🚇	Cancel
An empty document app	ears

14. Arrange your document windows using Window->Tile Vertically command.

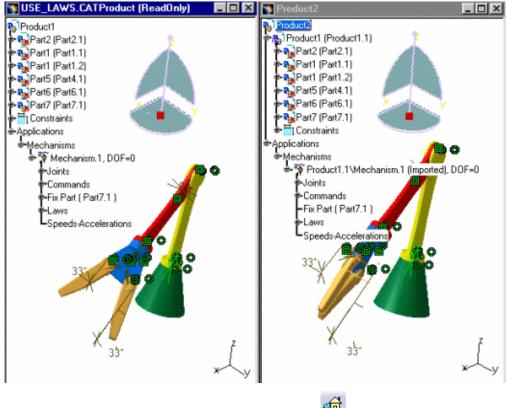
USE_LAWS.CATProduct (ReadOnly)	_ 🗆 X	Product2	_ [] >
Product1 70		Product2	97
🕈 🏡 Part2 (Part2.1)		Applications	
🕈 🏡 Part1 (Part1.1)			
🗣 🏡 Part1 (Part1.2)			
🗣 🏡 Part5 (Part4.1) 💦 🚽			y
🗣 🌄 Part6 (Part6.1)			
Part7 (Part7.1)			
🕈 📺 Constraints			
Applications			
🖶 Mechanisms 🥔			
🕨 👻 🌠 Mechanism.1, DOF=0			
Joints			
Commands			
Fix Part (Part7.1)			
-Laws			
Speeds-Accelerations			
1			

- **15.** Use the Copy/Paste capability to create a new product:
 - $_{\circ}~$ Right-click PRODUCT1 in the left window. Select Copy from the contextual menu displayed.
 - $_{\circ}~$ In the right window, right-click Product2 and select Paste from the contextual menu.
- **16.** You have two possibilities to import the sub-mechanisms:
 - $_{\circ}$ either click the Simulation with laws icon $\stackrel{\scriptstyle \bullet}{\textcircled{}}$ or
 - the Import Sub-Mechanisms icon
 In The import is automatically performed

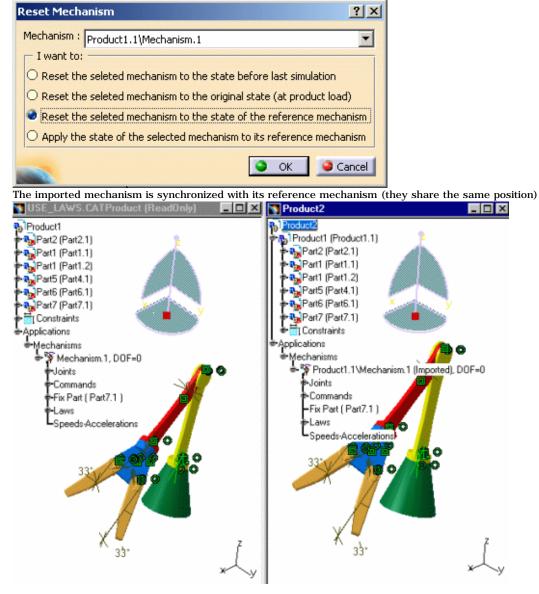
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- 17. Click the Use_Laws.CATProduct window (left in our example) and click the Simulation with laws icon in the DMU Kinematics toolbar
- 18. Run your simulation using the Play Forward button
- **19.** Click the Pause button at a position of your choice:



- 20. Click the Product2 windows and Click the Reset Positions icon
- **21.** Select the imported mechanism (Product1.1\Mechanism.1 the only one in our example)
- **22.** Select the 'Reset the selected mechanism to the state of the reference mechanism' option button

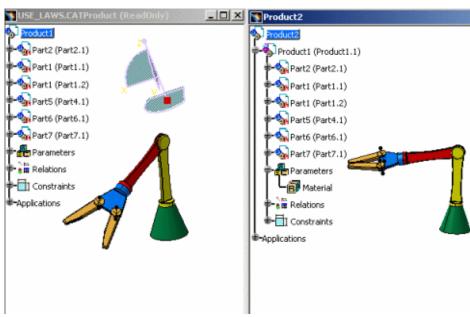


23. Click **Ok** to exit the Reset Mechanism dialog box

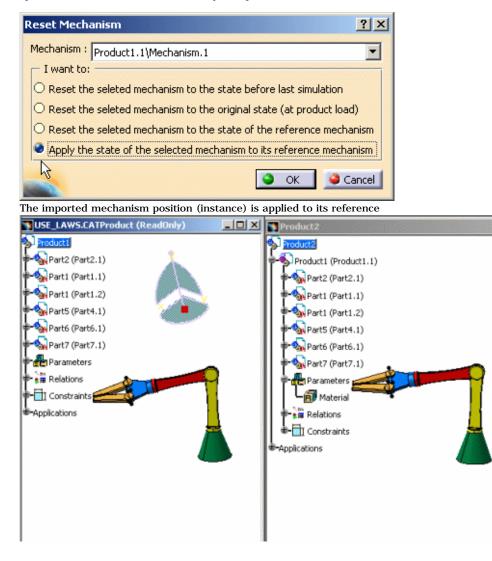
24. Still in the Product2 windows, click Simulation with laws icon

- simulate the imported mechanism
- stop at a given position, for example:

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- 25. Click Close to exit the Simulation With Laws command
- **26.** Click the Reset Positions icon again, this time select 'Apply the state of the selected mechanism to its reference mechanism' option button and click **Ok** to confirm your operation



Sequencing Mechanisms with Laws

The sequence integration allows to follow on or to play simultaneously several mechanisms (if the mechanisms can be simulated with laws).

Some tools will also be added to help the user. First, the Gant Chart command will allows to see the sequence as a Gant Chart. It will be also possible to convert a Simulation object to a Sequence, to convert a Sequence to a Replay or to export a Sequence as an AVI File.

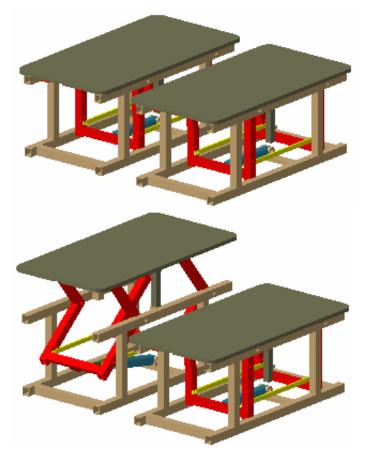
This task shows you how to simulate a mechanism with laws within a sequence

Open the MECHANISM_SEQUENCE.CATProduct document

- 1. Select Use_Sensors.1\Mechanism.1)
- Click the Player icon from the DMU Player toolbar, undock it if necessary using the arrow in the Player icon The player is displayed



3. Use the Play forward button to simulate your mechanism



- 4. Click the Skip to Begin button from the Player
- 5. Click the Play Forward button again
- 6. Repeat from Step 2 selecting this time, Use_Sensors.2\Mechanism.1
- 7. Now you want to sequence the two mechanisms

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Sequencing aims at defining a time frame within which the actions are scheduled.

- **Two sequencing modes are available:** • actions start together (simultaneous mode)
- o actions start right one after the other (consecutive mode)

8. Click the Sequence icon

The Edit Sequence dialog box is displayed

Edit Action Edit Analysis		
Action in session Use_Sensors.1\Mecha Use_Sensors.2\Mecha Mechanism.1	Action in Sequence Step Action	
	Move Up Move Down	Merge Up Merge Down
	Action duration (s)	🚽 default 🗸

9. Select Use_Sensors.1\Mechanism.1Use_Sensors.2\Mechanism.1 in the action in session list and click _____. The two actions are scheduled in simultaneous mode.

Step	Action
1	Use_Sensors.1\Mechanism.1
1	Use_Sensors.2\Mechanism.1
	÷

In fact, you want to play the two mechanisms in consecutive mode (one action starting after the other)

10. Select Use_Sensors.2\Mechanism.1 in the action in Sequence list and click the **Move Down** button

Ec	lit Sequence		? ×
	Edit Action Edit Analysis Action in session Use_Sensors.1\Mecha Use_Sensors.2\Mecha Mechanism.1	Action in Sequence Step Action 1 Use_Sensors.1\Mec	hanism.1
	↑	2 Use_Sensors.2\Mec	
		Move Up	Merge Up
		Move Down	Merge Down
	• •	Action duration (s) 5	🛃 user defined 📃 📃
	Action add mode		
	Create last step and add O Add in last step		
			🕒 OK 🥥 Cancel

11. Now run your sequence using the Player still displayed. You are not satisfied with the Time Step:

Let's customize the Player parameters

- **12.** Click the Parameters icon **EXAMPLE**. The Player Parameters dialog box is displayed:
- **13.** Enter 0.2 s in the Sampling Step field

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Player Parameters	? ×
Sampling Step 0.2 s	_
Temporization 0s	-
	and a second second

Each motion is replayed one after the other in the order they were scheduled.

You can choose one of the loop modes to re-run the simulation in a continuous way (either in the one direction only or in one direction then the other).



Managing Kinematics Data in Sub-products

Visualizing and Simulating Mechanisms in Sub-products More about Importing Mechanisms Dressup Importing a Mechanism and its Dressup Importing a Mechanism and its Dressup from a Skeleton Structure

Visualizing and Simulating Mechanisms in Sub-Products

A This task consists in visualizing and simulating mechanism in sub-products through the Import mechanism capability

Open the SUB_PRODUCT_MECHANISM_LAWS.CATProduct document.

1. Make sure you are in Design Mode if you work with the Cache System (please refer to DMU Navigator User's Guide- Viewing the Cache Content)

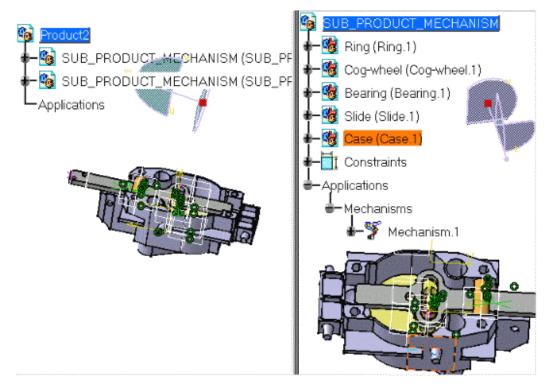
If not, select Edit->Representations->Design Mode from the menu bar...

2. Using the File->New command, click the New icon icon from the Standard toolbar or select the File->New... command. In the New dialog box, double-click Product.



An empty document appears.

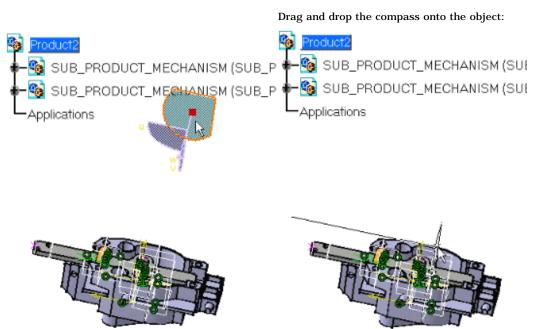
- 3. Arrange your document windows using Window->Tile Vertically command.
- 4. Use the Copy/Paste capability to create a new product:
 - o right-click SUB_PRODUCT_MECHANISM in the right window. Select Copy from the contextual menu displayed.
 - $_{\circ}\;$ in the left window, right-click Product2 and select Paste from the contextual menu.
- 5. Repeat Step 4. This is what you obtain:



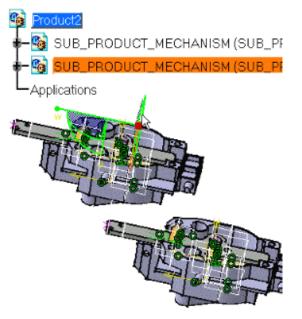
6. In the left window, use the 3D compass manipulation handle as shown below to obtain two different products in the geometry

area:

For more information, see Moving Objects using the 3D Compass in the Infrastructure User's Guide



The compass is snapped to the object selected. The compass changes color. Move the compass to separate the two products as shown below:



The import command has been improved: the mechanism import is automatically performed at the first simulation (with laws or with commands). The Import sub-mechanisms command is useful only for the first import. It has no effect when every mechanism has been imported.

- 7. You have two possibilities to import the sub-mechanisms:
 - $_{\circ}$ $\,$ either click the Simulation with laws icon
 - or the Import Sub-Mechanisms icon
 - The import is automatically done.

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8. Click Ok in the warning message displayed

Sub-Mechanisms Import Result
2 Sub-Mechanisms have been imported successfully.
The sub-mechanisms are imported and identified in the specification tree:
Fix Part (Case. 1) Fix Part (Case. 1) SUB_PRODUCT_MECHANISM. (SUB_PRODUCT_MECHANISM.1) SUB_PRODUCT_MECHANISM (SUB_PRODUCT_MECHANISM.2) Applications SUB_PRODUCT_MECHANISM.1\Mechanism.1 (Imported), DOF=0
Laws Laws Speeds-Accelerations
SUB_PRODUCT_MECHANISM.2\Mechanism.1 (Imported), DOF=0
♥-Joints ♥-Commands
Fix Part (Case.1)
Laws Laws Speeds-Accelerations

The laws are imported. However, knowledgeware rules are not imported. You can modify the laws belonging to a sub-mechanism.

For instance, if you expand the Laws node in SUB_PRODUCT_MECHANISM.1 and double-click the formula:

• In CATIA

The Formula Editor automatically appears, letting you perform your modifications

`SUB_PRODUCT_MECHANISM.1\Mechanism.1\listcmd.1\Command.1.1\Angle` = 180 deg * sin(SUB_PRODUCT_MECHANISM.1\Mechanism.1\KINTime /1s*1rad)						
Dictionary	Members of Parameters	Members of All				
Parameters Point Constructors Surface Constructors Law Operators Line Constructors Wireframe Constructors List	Renamed parameters Boolean Length Angle Plane String Time Real	 Ring\PartBody\Sketch.1\Parallelism.1\activ Ring\PartBody\Sketch.1\Coincidence.4\ac Ring\PartBody\Sketch.1\Perpendicularity.5' Ring\PartBody\Sketch.1\Parallelism.6\activ Ring\PartBody\Sketch.1\Parallelism.7\activ Ring\PartBody\Sketch.1\Parallelism.8\activ Ring\PartBody\Sketch.1\Parallelism.1\activ 				

o In ENOVIA DMU

Double-click Command. 1.1 under Commands item in the specification tree. Click the Formula button in the Command Edition dialog box to display the Formula Editor dialog box

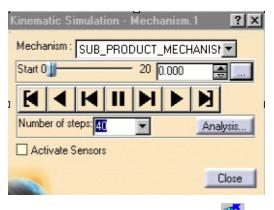
Page 2	78
--------	----

Command Editio	on : Command.1.1 (Angle)	? ×
Command name :	Command.1.1	
Command value :	fixe	
		-13
	I OK	Cancel

9. Select SUB_PRODUCT_MECHANISM (SUB_PRODUCT_MECHANISM.1).

Click the Simulation with laws icon if from the DMU Kinematics toolbar. Please refer to Simulating With Laws.

You can simulate the sub-mechanism with laws.



10. Click the Simulation with commands icon from the DMU Kinematics toolbar. Please refer to Simulating With Commands.

You can simulate the sub-mechanism.

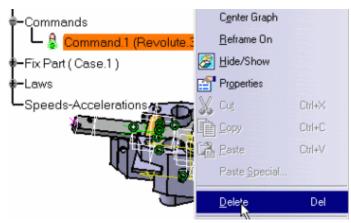
Kinematic Sim	nulation - Sl	JB_PROD	JCT_MECHA	NIS ? 🗙
Mechanism :	SUB_PRODU	CT_MECHA	NISM.1/Mecha	inism.1 💌
Command.1.1			360 0.0000	.
Check Joir	nt Limits			
Reset		Analysis.		More>>
				Close

Note that you can only modify sub-mechanism command and laws. For instance, if you double-click Prismatic 2.1 an information message automatically appears

►¥ SUB_PRODUCT_MECHANISM.2/Me	Informati	on 🔀
 Joints Rigid.1.1 (Bearing.1,Case.1) Rismatic.2.1 (Slide.1,Bearing.1,Case.1) Revolute.3.1 (Case.1)Cog-whete Revolute.4.1 (Cog-wheel.1,Ring.1) 		You can't edit a joint belonging to a Sub-Mechanism

- Now, let's modify SUB_PRODUCT_MECHANISM.CATProduct and use the sub-mechanism import
- delete the existing command (Command.1 (Revolute.3, Angle)
- assign a length command to Prismatic.2
- 11. In the right window (SUB_PRODUCT_MECHANISM.CATProduct), expand the Command item and delete the existing command (Command.1 (Revolute.3, Angle)

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- **12.** Click **Ok** in the information message
- 13. Double-click Prismatic.2 and assign a Length command

Joint Edition : Prismatic.2	? ×
Joint name : Prismatic.2	
Joint ge	eometry :
Line 1 : Edge	Line 2 : Edge
Plane 1 : Face	Plane 2 : Face
Length Driven	
Joint Limft	
Cover Limit : -100mm	🕼 Upper Limit : 100mm
	OK Cancel

14. Click Ok, when done click Ok in the information message displayed

(A) Note this modification results in deleting the existing law in SUB_PRODUCT_MECHANISM.CATProduct

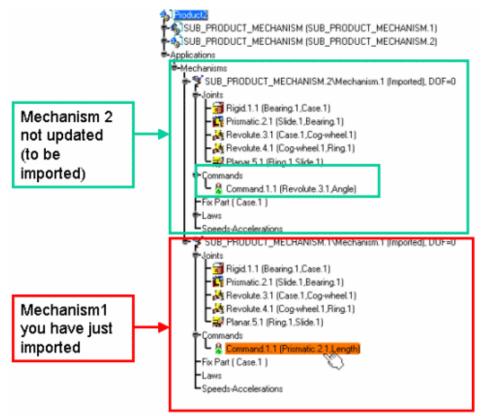
- **15.** Click in the left window and then click the Update Positions icon
- **16.** The Update Mechanism dialog box is displayed:

Update Mechanism	? ×
Mechanism : SUB_PRODUCT_MECHANISM.1\Mechanism.1	-
Take curr <mark>SUB_PRODUCT_MECHANISM.1\Mechanism.1 SUB_PRODUCT_MECHANISM.2\Mechanism.1</mark>	
	Cancel

- 17. Select the mechanism to be imported using the drop-down list box
- 18. Click Ok

The first sub-mechanism is re-imported and updated accordingly (the command is now assigned on the prismatic 2.1 and length driven. The second mechanism remains in its initial state (command assigned to Revolute 3.1)

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Note: the update command imports sub-mechanisms one after the other (i.e. only the one selected has been imported taking into account the modification), you will need to repeat the operation for the second mechanism. The mechanism is re-imported thus displayed in the specification tree in last position.

19. Repeat the step 15 if the Update command is no longer active and select the second mechanism to be imported (the first

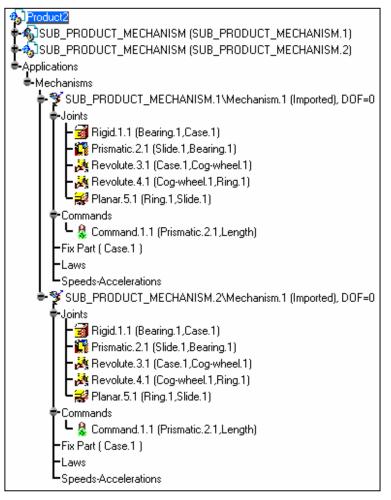
mechanism in the drop-down list because of the inversion valid not only in the specification tree but also in the Update Positions dialog box

Update Mec	hanism	? ×
Mechanism :	SUB_PRODUCT_MECHANISM.2\Mechanism.1	-
Take curr	SUB_PRODUCT_MECHANISM.2\Mechanism.1 SUB_PRODUCT_MECHANISM.1\Mechanism.1	
	SUB_PRODUCT_MECHANISM.1\Mechanism.1 나	
	💁 ок 丨 🛥 с	ancel
		ancel

20. Click **Ok** to validate the operation

Both sub-mechanisms are imported

The modification is taken into account (command change) is taken into account in both sub-mechanisms



[®]More about Importing Mechanisms Dressup

This section provides information about the dressup import.

The Import capability lets you import sub-mechanisms as well as their associated dressups.

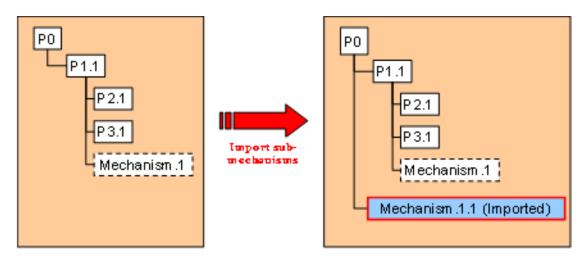
You have two possibilities to import sub-mechanisms:

- 1. either click the Simulation with laws icon
- 2. or the Import Sub-Mechanisms icon

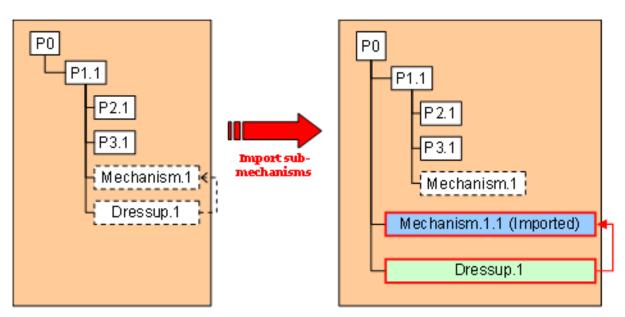
Please refer to the following scenarios Importing a Mechanism and its Dressup and Importing a Mechanism and its Dressup from a Skeleton Structure How does it work?

The Import Sub-Mechanisms command scans all the mechanisms existing in the sub-products. if a mechanism is detected (i.e. candidate to the import operation), several cases are to be studied:

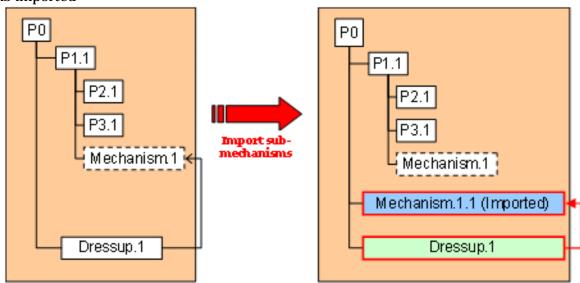
• There is not any dressup associated to the mechanism. Only the mechanism is imported at the root level



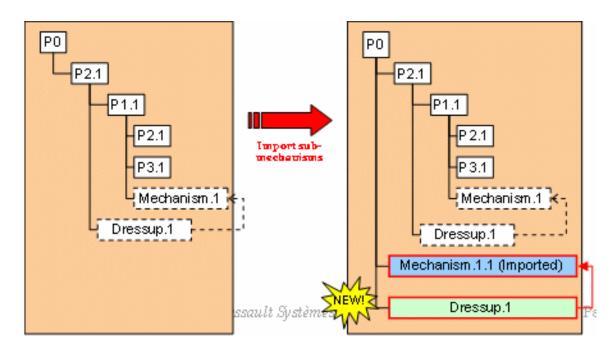
• There is a dressup associated to the mechanism at its level and there is not any other dressup pointing this mechanism elsewhere. The mechanism and its associated dressup are imported at the root level.



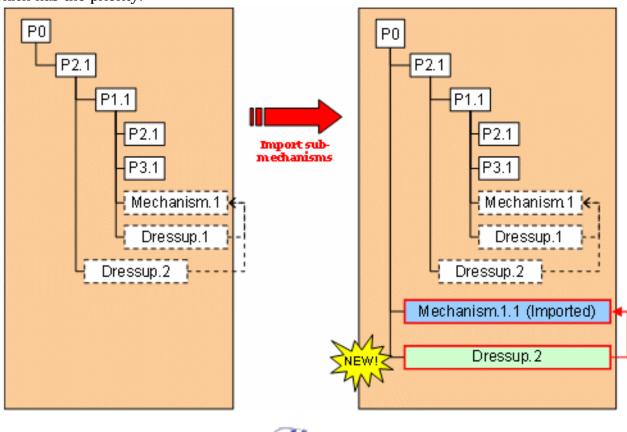
- There is a dressup associated to the mechanism (not at the same level). Two cases:
 - 1. The dressup is already positioned at the root level (i.e integrator level). Only the mechanism is imported



2. The dressup is not positioned at the root level, nor at the mechanism level. The dressup is imported at the root level as well as its associated mechanism.



• Particular case: the mechanism is assigned two dressups: one dressup at the same level and another dressup at another level. During the import operation, this is the highest-level dressup which has the priority:



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Importing a Mechanism and its Dressup

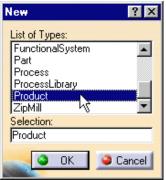
N This task consists in importing a and simulating mechanism with its associated dressup using the Import capability

Open the IMPORT_MECHANISM_DRESSUP.CATProduct document.

1. Make sure you are in Design Mode if you work with the Cache System (please refer to DMU Navigator User's Guide- Viewing the Cache Content)

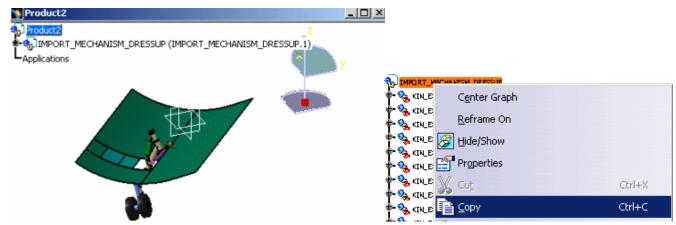
If not, select Edit->Representations->Design Mode from the menu bar...

2. Using the File->New command, click the New icon in from the Standard toolbar or select the File->New... command. In the New dialog box, double-click Product.



An empty document appears.

- 3. Arrange your document windows using Window->Tile Vertically command.
- 4. Use the Copy/Paste capability to create a new product:
 - o right-click IMPORT_MECHANISM_DRESSUP in the right window. Select Copy from the contextual menu displayed.
 - o in the left window, right-click Product2 and select Paste from the contextual menu.



5. In the Product2 window, import the mechansim and its associated dressup, for this:

You have two possibilities :

- either click the Simulation with laws icon
- or the Import Sub-Mechanisms icon

6. Click Ok in the warning message displayed (if you clicked the Import Sub-Mechanisms icon The import operation is performed: The dressup is imported:



7. Select IMPORT_MECHANISM_DRESSUP.1\LANDING GEAR_V5(imported),DOF=0

Click the Simulation with commands icon from the DMU Kinematics toolbar.
 Please refer to Simulating With Commands.

You can simulate the imported mechanism.



^(©)Importing a Mechanism and its Dressup from a Skeleton Structure

This task shows you how to import a mechansim and its associated dressup from a skeleton structure

Skeleton structure:

Consists in defining mechanisms using the skeleton methodology with three product levels:

- an architect level containing a CATProduct document including the master skeleton and its mechanism
- several designer levels (i.e. CATproducts documents containing the 3D solid geometry)
- an integrator level (i.e a CATproduct document which federates the architect level the designer levels and the dressup)

Open the Integrator.CATProduct document.

1. Make sure you are in Design Mode if you work with the Cache System (please refer to DMU Navigator User's Guide- Viewing the Cache Content)

If not, select Edit->Representations->Design Mode from the menu bar...

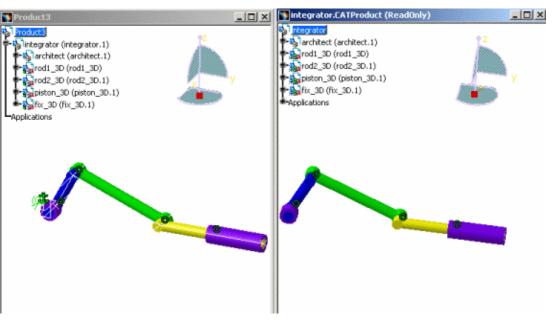
Using the File->New command, click the New icon icon from the Standard toolbar or select the File->New... command. In the New dialog box, double-click Product.



An empty document appears.

- 3. Arrange your document windows using Window->Tile Vertically command.
- **4.** Use the Copy/Paste capability to create a new product:
 - o right-click Integrator in the right window. Select Copy from the contextual menu displayed.
 - in the left window, right-click Productn (Product3 in our example) and select Paste from the contextual menu. This is what you should obtain:

Version 5 Release 13



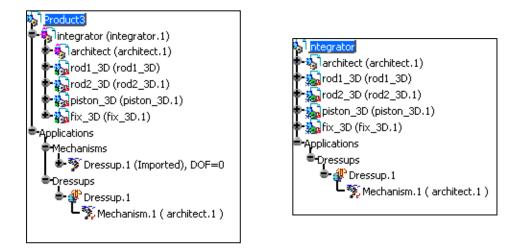
5. Make sure, the Productn (in our example Product3) window is active and click the Import Sub-Mechanisms icon

U Note you can also click the Simulation with laws icon 🏁 to import the mechanism

6. Click Ok in the warning message displayed



The sub-mechanism and its associated dressup are imported and identified in the specification tree: Please also read More about importing mechanisms dressup



Product3 window

Integrator.CATProduct window (Root product)

i The icons change for integrator and architect in the specification tree which become flexible products. For more information, please read Flexible Sub-Assemblies in Assembly User's Guide

7. Select Dressup1 in the specification tree

Click the Simulation with Commands icon ¹⁹⁹ from the DMU Kinematics toolbar. Please refer to Simulating With Commands.

DMU Kinematics Simulator

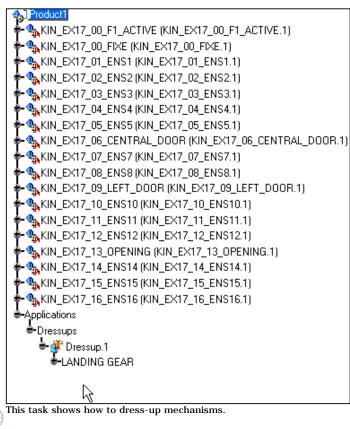
You can simulate the imported dressup

Kinematic Simulation - D)ressup.1	? ×
Mechanism : Dressup.1		▼
Activate Sensors	Analysis	More>>
		Close

×

Managing the Mechanism Dressup

In the perspective of Kinematics integration in ENOVIA VPM (based on a skeleton methodology) (a document will be available in a forthcoming release) the dressup is directly accessible from the specification tree, you can simulate it and it can be saved in ENOVIA VPM



Open the MANAGING_DRESSUP.CATProduct document.

()

(***)

At least one kinematics mechanism must be active in the specification tree.

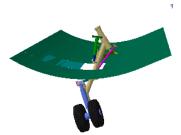
U If you work with V4 data, you no longer need to select Edit->Representations->Design Mode as it is automatically activated. DMU Kinematics Simulator finds the product containing kinematics objects automatically. This capacity is available for all Kinematics commands (simulation...)

1. Click the Mechanism Analysis icon

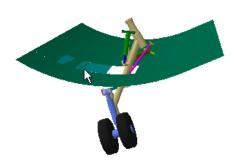
The Mechanism Analysis dialog box appears:

۲		i <mark>sm Analys</mark> ral Propertie:									? ×
		nism name :	2		LAN	DING GEAR					-
	Mechar	nism can be s	imulated :		Yes						-
		r of joints :			21						
		r of command	He i	1000							
ć.		s of freedom		mmand .	5						
	-				1.000						
	-	s of freedom		ands :	0					222222	
	Fixed p	art :			REF		ege tradición				
	O Sha	w joints 🥥	Hide joints				Save			Laws	
	Joint	Command	Туре	Part 1		Geometry 1	Part 2	nor es	Geometry 2	Part 3	
			Revolute	ENS10		DRT35	FIXE		DRT35		
			Prismatic	ENS8		DRT33	ENS12		DRT33		
		LANDING	Revolute	ENS1		DRT27	FIXE		DRT27		
			Revolute	ENS11		DRT37	FIXE		DRT37		
			Revolute	ENS7		DRT36	ENS11		DRT36		
			Spherical			PT 30	ENS7		PT 30		
			Revolute	ENS1		DRT12	ENS3		DRT12		
			Spherical			PT 27	ENS5		PT 27		
			Revolute	ENS4		DRT19	ENS5		DRT19		
	l		Revolute	ENS4		DRT13	ENS2		DRT13		
	Mechan	ism dressup	information	:							
	Part 1		art 2	Part	3	terre deserve					
					-						

Click on the Simulation With Commands icon
 The Kinematic Simulation dialog box is displayed.



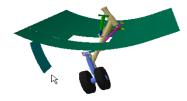
Kinematic S	imulation - LA	NDING GEAR	? ×
Mechanism :	LANDING GE/	AR	-
CENTRAL	-360	360 0.00	00 💽 🛋
LEFT	-360 ,	360 0.00	00 😫
OPENING	-350 ,	370 10.0	000
INCLINAT	-360	1 360 0.00	
Check J	pint Limits		
Reset		Analysis	More>>
			Close



3. Manipulate the slider of the LEFT command.

The corresponding part of the kinematics mechanism namely the **Opening** moves accordingly.

Kinematic Simulation - LANDING GEAR	? ×
Mechanism : LANDING GEAR	•
CENTRAL -360 360 0.0000	
LEFT -360 360 50.4000	.
OPENING -350 370 10.0000	
INCLINAT -360 360 0.0000	
Check Joint Limits	
Reset Analysis	More>>
	Close



4. Click Reset and then Close

Let's attach the left door to the LANDING GEAR mechanism:

 Click the Mechanism Dress-up icon in the DMU Simulation Toolbar. The Mechanism Dress-up dialog box is displayed.

6. Select LEFT DOOR as link

i Note you can select the link directly in the geometry area or in the specification tree, using the the graphic selection option. Though, only one selection is allowed.

Ŧ

Link: 🗊 Graphic selection LEFT DOOR

Note that you can select or deselect attachments directly from the specification tree or geometry area. You can select either the available products or all products. By default the Available products option is set.

Available products
 All products

• Available products: if set, this option lets you visualize the products that are not referenced in any attachment within the mechanism.

- All Products: if set, this option lets you visualize the products that are not attached to the current link (here, LEFT DOOR)
- 7. By default the Available products option is set:

The two products in the left column have not been attached yet.

DMU Kinematics Simulator

Mechanism Dressup			? ×
Dressup :	Dressup.1		New Dressup
Mechanism :	LANDING GEAR		
Link: 🔲 Graphic selection	LEFT DOOR		
Available products	O All products		
Available products KIN_EX17_00_F1_ACTIN KIN_EX17_09_LEFT_DC		Products attached to the link	
		<u> </u>	OK 🥥 Cancel

8. If you set the All products option this is what you obtain

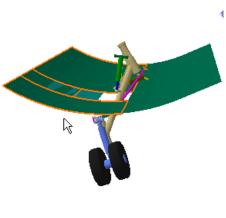
Now, select The KIN_EX17_09_DOOR from the available products list to attach it to the link:

Mechanism Dressup			? ×
Dressup : Mechanism : <mark>Link: □</mark> Graphic selection	Dressup.1 LANDING GEAR LEFT DOOR		New Dressup
O Available products	All products		
Available products KIN_EX17_04_ENS4.1 KIN_EX17_05_ENS5.1 KIN_EX17_06_CENTRAL_ KIN_EX17_07_ENS7.1 KIN_EX17_08_ENS8.1 KIN_EX17_10_ENS10.1 KIN_EX17_11_ENS11.1 KIN_EX17_12_ENS12.1 KIN_EX17_13_0PENING.1 KIN_EX17_14_ENS14.1 KIN_EX17_15_ENS15.1		Products attached to the link KIN_EX17_09_LEFT_DOOR.1	
		9	OK 🧕 🥥 Cancel

The selected product is highlighted in the specification tree and in the geometry area as shown below:

9. Click **OK** to confirm your operation.

S Product1	
- 🎭 KIN_EX17_16_ENS16 (KIN_EX17_16_ENS16.1)	
KIN_EX17_00_Mirage_F1_ACTIVE_MODEL (KIN_EX17_00_Mirage_F1_ACTIVE_MODE	EL.1)
- 🎭 KIN_EX17_01_ENS1 (KIN_EX17_01_ENS1.1)	
- 🎭 KIN_EX17_02_ENS2 (KIN_EX17_02_ENS2.1)	
- 🎭 KIN_EX17_03_ENS3 (KIN_EX17_03_ENS3.1)	
- 🎭 KIN_EX17_04_ENS4 (KIN_EX17_04_ENS4.1)	
- 🎭 KIN_EX17_05_ENS5 (KIN_EX17_05_ENS5.1)	
KIN_EX17_06_CENTRAL_DOOR (KIN_EX17_06_CENTRAL_DOOR.1)	
- 🎭 KIN_EX17_07_ENS7 (KIN_EX17_07_ENS7.1)	
- 🎭 KIN_EX17_08_ENS8 (KIN_EX17_08_ENS8.1)	
—	
- 🎭 KIN_EX17_10_ENS10 (KIN_EX17_10_ENS10.1)	
- 🎭 KIN_EX17_11_ENS11 (KIN_EX17_11_ENS11.1)	

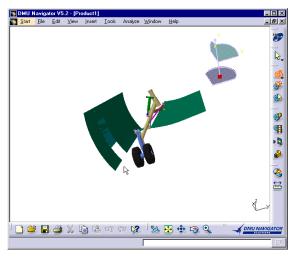


Let's simulate the mechanism with the new dressup

DMU Kinematics Simulator

- 10. Click on the Simulation With Commands icon 🥮 again.
- **11.** In the Kinematic Simulation dialog box, manipulate the slider of the LEFT command.

This time, the corresponding part of the kinematics mechanism moves accordingly.



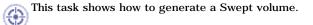
The Simulation With Commands capability is only used to simulate. If you need to record positions use the (Fitting) Simulation functionality.

Defining Swept Volume



Defining a Swept Volume Defining a Swept Volume from a Mechanism Defining a Swept Volume from a Moving Reference Filtering Swept volume Positions More About Swept Volume *******

Defining A Swept Volume



You recorded a simulation in a Simulation object and compiled the Simulation. You obtained a Replay object. Open the KIN_SWEPT_VOL.CATProduct document.

Remember, you can generate a swept volume directly from a V5 mechanism which can be simulated with laws.



The Swept Volume dialog box is displayed.

Ş	wept Volume		? ×
	Definition		
	Selection	Replay.1	-
	Product(s) to sweep	17 products	
	Reference product		
	Filter Positions	Use level of de	etails
	Filtering precision	0.2mm Filtered (%)	
	-Result simplificatio	on	
	Apply Wrapping	Apply Simplification	
	Save	Preview Clos	se

Wrapping, simplification, silhouette and spatial split options are available within the swept volume dialog box if you have a DMU Optimizer license:

wept Volume		_?
Definition		
Selection	Replay.1	•
Product(s) to sweep	17 products	
Reference product	[
Filter Positions	Silhouette	Use level of details
Filtering precision	0.2mm	Filtered (%)
Result simplification	, n	
	Apply Sim	plification 🔲 Spatial Sp
-Wrapping		
Grain 20mm	Offset ratio	0,10
Simplification		
Accuracy 10mm		
-Number of triangl	es —	
Initial :	Result	t :
	Save	Preview Close

i Note: the Filter Positions option is checked by default

- **2.** Click in the Product to sweep spin box, the selection list dialog box lets you select or deselect the bodies you want to sweep.
- 3. Select KIN_EX17_03_ENS3.1

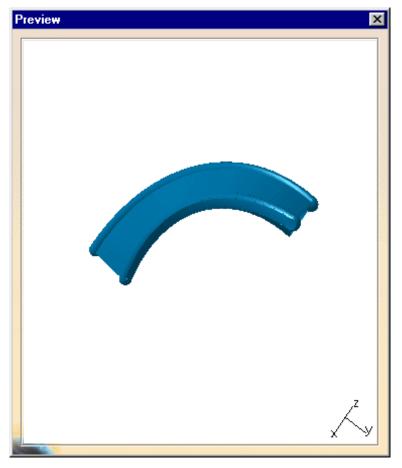
4. Click OK.

Product Multiselection	? ×
KIN_EX17_10_ENS10.1 KIN_EX17_00_FIXE.1 KIN_EX17_08_ENS8.1 KIN_EX17_08_ENS12.1 KIN_EX17_12_ENS12.1 KIN_EX17_01_ENS1.1	1
KIN_EX17_11_ENS11.1 KIN_EX17_07_ENS7.1 KIN_EX17_03_ENS3.1 KIN_EX17_05_ENS5.1	
KIN_EX17_04_ENS4.1	 ок

5. Click Preview. The progress bar is displayed letting you monitor and, if necessary, interrupt (Cancel option) the calculation.

Computation in	progress	×
ð	Computing Swept Volume	
Status :	48% completed	
Estimated time	remaining :18sec	
	(Cancel)	

6. If you check the Use level of details option, this what you obtain:



7. Click Save.

The Save As dialog box appears

 $\left(i
ight)$ you can save your result in various formats, for example in CATIA model file

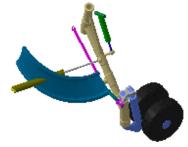
Save As						<u>? ×</u>
Save in:	🔁 samples		•	- 🗈 🗗	•	
History Desktop My Documents My Computer	REGULATION	r cgr cgr IPLIFICATION.cgr _COMMAND.cgr	TANK1.model.co TANK2.model.co TANK3.model.co TANK3.model.co TANK3.model.co TANK3.model.co TANK3.model.co TANK3.model.co TANK3.model.co	gr		
My Network P	File name:		_absoluteresult.cgr	•		Save Cancel
	Save as type:	cgr			-	Cancer
		ogr wrl model stl				

About Save button in swept volume dialog box:

Clicking **Save** keeps the command active and lets you therefore launch the calculation again if needed. When satisfied, click **Save** in the **Save As** dialog box

- 8. Select cgr file and click Save.
- 9. Click Close
- Insert the SWEPTVOLUME_absoluteresult.cgr into Product1, for this right-click Product1 and select Components-> Existing component from the contextual menu displayed.

The Swept volume is identified in the specification tree and in the geometry area



KIN_EX17_13_OPENING (KIN_EX17_13_OPENING.1)
 KIN_EX17_13_OPENING (KIN_EX17_13_OPENING.1) KIN_EX17_14_ENS14 (KIN_EX17_14_ENS14.1)
KIN_EX17_15_ENS15 (KIN_EX17_15_ENS15.1)
- 🗞 KIN_EX17_00_F1_ACTIVE (KIN_EX17_00_F1_ACTIVE.1)
- 🎭 SWEPTVOLUME_absoluteresult (SWEPTVOLUME_absoluteresult.1)



 \oplus

Defining a Swept Volume From a Mechanism

(which can be simulated with laws)

The Swept volume functionality is very useful to design the volume swept by a set of moving products during simulation. Now, this functionality takes into accounts mechanisms.

The only condition about these mechanisms is that they can be simulated with laws. This task shows how to generate a Swept volume from a mechanism

Open the Use_Sensors.CATProduct document.

1. Click the Swept Volume icon

con 🗖.

The Swept Volume dialog box is displayed.

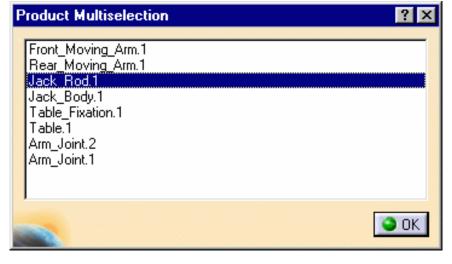
(**i**)1

Note:the Filter Positions option is checked by default

Swept Volume	? ×
- Definition	
Selection	Mechanism.1
Product(s) to sweep	8 products
Reference product	
Filter Positions	Use level of details
Filtering precision	0.2mm Filtered (%)
- Result simplification	on
Apply Wrapping	Apply Simplification
Save	Preview Close

Please also read About the number of steps in Generating a Trace from a V5 Mechanism...

- **2.** Click in the Product to sweep spin box, the selection list dialog box lets you select or deselect the bodies you want to sweep.
- 3. Select Jack_Rod.1



- 4. Click Ok
- 5. Click **Preview** to generate the swept volume

The progress bar is displayed letting you monitor and, if necessary, interrupt (**Cancel** option) the calculation.

Computation in progres	s 🗙
2	Computing Swept Volume
Status : Estimated time remaining	48% completed g:18sec Cancel
This is what you obtain:	
Preview	
	\sim

6. Click Save.

The Save As dialog box appears:

Save As						? ×
Save in:	🔁 samples		•	+ 🗈	-11 *	
History History Desktop My Documents My Documents	REGULATION	r cgr cgr IPLIFICATION.cgr _COMMAND.cgr	ANK1.mode TANK2.mode TANK3.mode TANK3.mode TRIGGER.cgr	l.cgr l.cgr		
	File name:	SWEPTVOLUME.	result.cgr		•	Save
My Network P	Save as type:	cgr			•	Cancel

- 7. Select cgr file and click Save
- 8. Click Close
- 9. Insert the SWEPTVOLUME_result.cgr into Use_Sensors, for this right-click Use_Sensors and select Components-> Existing component from the contextual menu displayed. The Swept volume is identified in the specification tree and in the geometry area



1

Defining a Swept Volume from a Moving Reference

This task shows how to define a swept volume using a moving reference. See the previous task. In our example you need to obtain a finer result to analyze clashes, if any.

Open the KIN_SWEPT_VOL.CATProduct document.

1. Click the Swept Volume icon

The Swept Volume dialog box is displayed.

Swept Volume	? ×
Definition ——	
Selection	Replay.1
Product(s) to sweep	17 products
Reference product	
Filter Positions	Use level of details
Filtering precision	0.2mm Filtered (%)
- Result simplification	on
Apply Wrapping	Apply Simplification
Save	Preview Close

- 2. Uncheck the Filter Positions option as shown below: (the Filter Positions option is checked by default)
- **3.** Click in the Products to sweep spin box, the Product Multiselection dialog box lets you select or deselect the bodies you want to sweep.
- 4. Select KIN_EX17_03_ENS3.1 from the list.
- 5. Click Ok.

Product Multiselection	? ×
KIN_EX17_10_ENS10.1 KIN_EX17_00_FIXE.1 KIN_EX17_08_ENS8.1 KIN_EX17_12_ENS12.1 KIN_EX17_01_ENS1.1 KIN_EX17_01_ENS11.1 KIN_EX17_07_ENS7.1 KIN_EX17_03_ENS3.1 KIN_EX17_05_ENS5.1 KIN_EX17_04_ENS4.1	
	<u> </u>

- **6.** Click in the Reference Product spin box.
- 7. Select KIN_EX17_01_ENS1.1

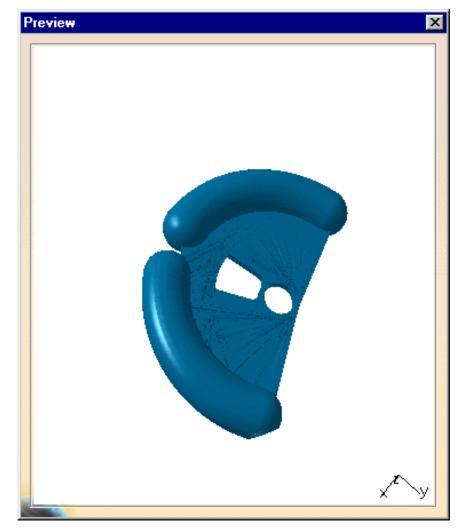
Swept Volume 🛛 🕐 🗙
Definition
Replay Replay.1
Product(s) to sweep 1 product
Filter Positions Use level of details
Filtering precision 1mm
Reference product EX17_01_ENS1.1
Number of positions
Initial : Filtered :
and a second
Save Preview Close

8. Click **Preview** to generate the swept volume.

The Computation is in progress. The progress bar is displayed letting you monitor and, if necessary, interrupt (**Cancel** option) the calculation.

Computation in	progress	×
<i>i</i>	Computing Swept Volume	
Status : Estimated time	48% completed remaining :18sec	
	(Cancel)	

The **Preview** window is displayed

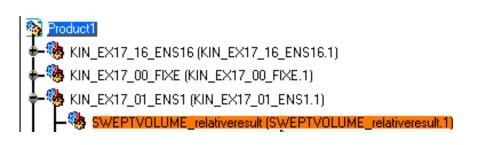


9. Click Save.The Save As dialog box appears automatically

e As					?
Save in:	🔄 samples		• • •	D 💣 🎟 -	
History Desktop Ocuments	SWEPTVOLU	ME_absoluteresult.cgr ME_filterresult.cgr ME_relaresult.cgr ME_relativeresult.cgr ME_result.cgr			
y Network P	File name:	SWEPTVOLUME_relativer	esult.cgr		Save
	Save as type:	cgr		-	Cancel
		egr wrl model stl			

10. Select cgr file from the Save as type drop-down list and click Save.hj,jn

11. Insert the SWEPTVOLUME_relativeresult.cgr into KIN_EX17_01_ENS1, for this right-click and select Components->Existing Component from the contextual menu displayed.The Swept volume is identified in the specification tree and in the geometry area.







Filtering Swept Volume Positions

This task shows how to filter swept volume positions. It can be very useful in terms of calculation performances to retrieve positions in a swept volume. Please refer to Defining a Swept Volume

You recorded a simulation in a Simulation object and compiled the Simulation. You obtained a Replay object. You need this Replay object to define a swept volume.

Open the Open the KIN_SWEPT_VOL.CATProduct document.

1. Click the Swept Volume icon

The Swept Volume dialog box is displayed.

Swept Volume	? ×
Definition ——	
Selection	Replay.1
Product(s) to sweep	17 products
Reference product	
Filter Positions	Use level of details
Filtering precision	0.2mm Filtered (%)
- Result simplification	on
Apply Wrapping	Apply Simplification
Save	Preview Close

- 2. Click in the Products to sweep spin box, the selection list dialog box lets you select or deselect the bodies you want to sweep.
- 3. Select KIN_EX17_03_ENS3.1

Product Multiselection	? ×
KIN_EX17_10_ENS10.1 KIN_EX17_00_FIXE.1 KIN_EX17_08_ENS8.1 KIN_EX17_12_ENS12.1 KIN_EX17_01_ENS1.1 KIN_EX17_01_ENS11.1 KIN_EX17_07_ENS7.1 KIN_EX17_03_ENS3.1 KIN_EX17_05_ENS5.1 KIN_EX17_04_ENS4.1	

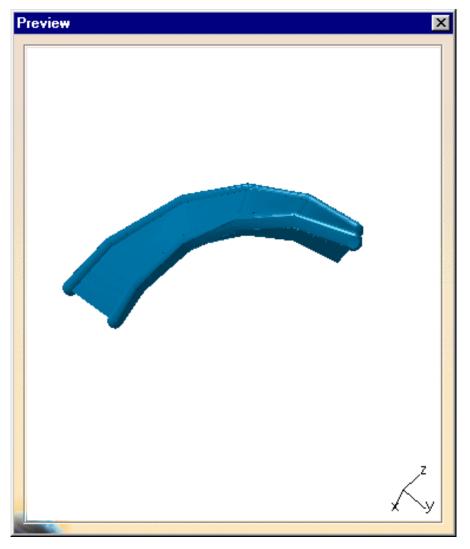
- 4. Click OK.
- 5. Enter 20mm as filtering precision value

Swept Volume	? ×
- Definition	
Replay	Replay.1
Product(s) to sweep	
Filter Positions	Use level of details
Filtering precision	20mm
Reference product	
-Number of position	าร
Initial :	Filtered :
Save	Preview Close

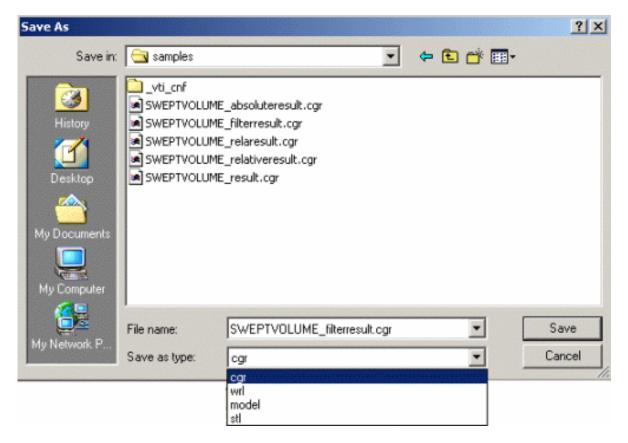
6. Click **Preview** to generate the swept volume. The progress bar is displayed letting you monitor and, if necessary, interrupt (**Cancel** option) the calculation.

Computation in progress 🗙					
ð	Computing Swept Volume				
Status : Estimated time	48% completed remaining :18sec Cancel				

This what you obtain:

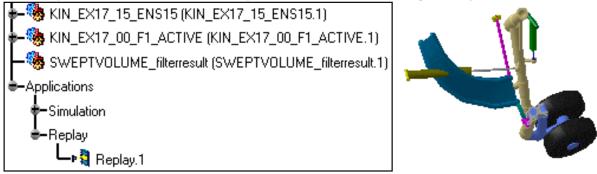


7. Click **Save** if you are satisfied or launch the calculation again with new values. The Save As dialog box appears automatically:



- 8. Select cgr format and click Save
- 9. Click Close
- Insert the SWEPTVOLUME_filterresult.cgr into Product1, for this right-click Product1 and select
 Components->Existing Component from the contextual menu displayed.

The Swept volume is identified in the specification tree and in the geometry area



Please refer to the *DMU Optimizer User's Guide* for more information



More About Swept Volume

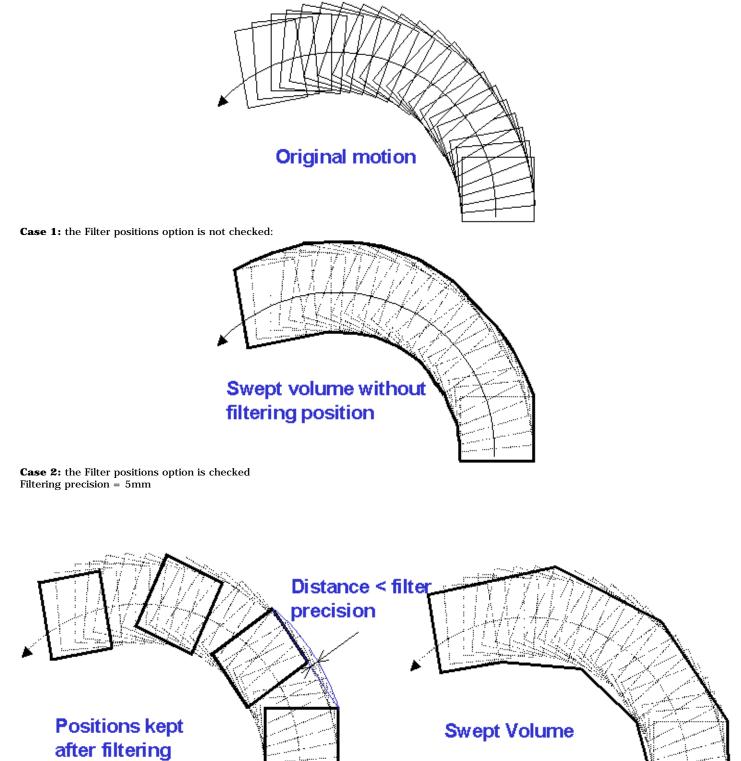


This option can be used to simplify the swept volume computation when the replay object contains many positions or when you know what precision level you need to obtain.

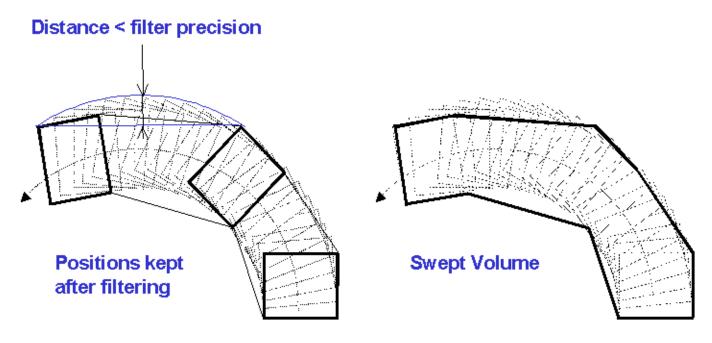
The "filter precision" defines the maximum distance allowed between the simplified trajectory and the initial one (= discretization precision)

1- Filtering swept volume positions

The following example aims at illustrating the impact of the filter positions option on the final result



Case 3: the Filter positions option is checked Filtering precision = 10mm



2- Relative swept volume

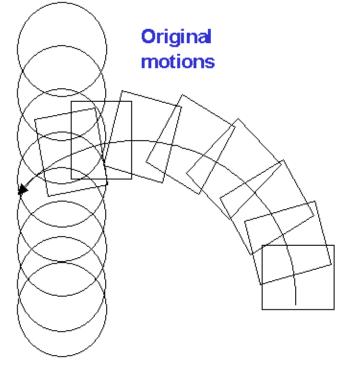
About Relative swept volume:

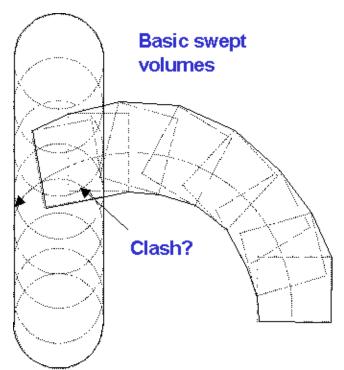
i

You can compute the swept volume of a moving part in the system axis of another moving part. You can use this option when you need to analyze the swept volume of a product versus another product (moving or not)

Example: two moving parts: circle and square With the basic computation of the swept volumes, the clash analysis is not relevant:

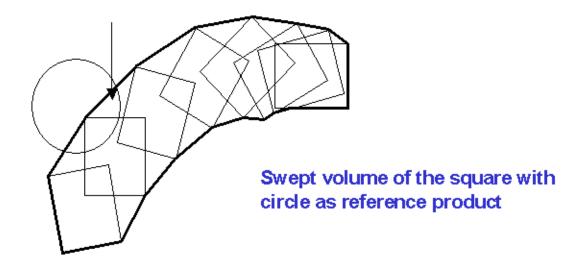
the swept volumes clash but two objects may not be in the same clash area at the same time.





If you use the relative swept volume option and select the circle as the reference product, you can compute the square swept volume in the circle system axis.

The result can now be relevant for clash analysis.



Working with ENOVIA LCA: Optimal PLM Usability for DMU Kinematics Simulator

When working with ENOVIA LCA, the Optimal PLM Usability for DMU Kinematics Simulator ensures that you only create and modify data in CATIA that can be correctly saved in ENOVIA LCA.

ENOVIA LCA offers two different storage modes Workpackage (Document kept - Publications Exposed) and Explode (Document not kept- Structure Exposed).

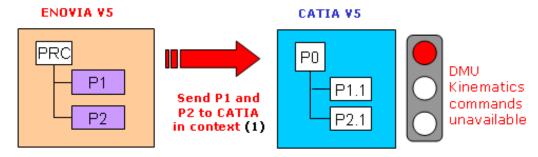
In Kinematics Simulator workbench, it is impossible to create and save Kinematics data in explode mode (structure exposed) in ENOVIA LCA. The Optimal PLM Usability for DMU Kinematics Simulator means that when working in explode mode (structure exposed) all the Kinematics commands are unavailable (i.e grayed).

On the contrary, when working in publication exposed mode (i.e the document you are working on is defined as a workpackage), all the Kinematics commands become available, you can therefore:

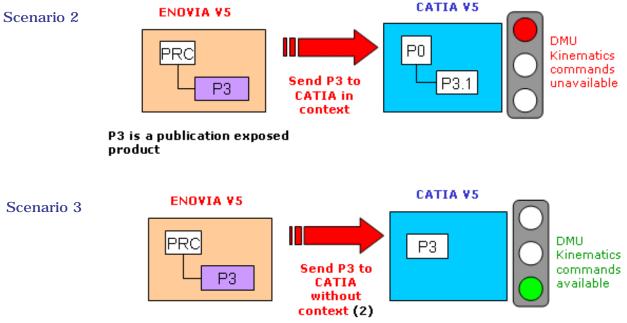
- Create,
- Modify and,
- Save your data in ENOVIA LCA provided that your workpackage is based on the root product (i.e. without context, please refer to Scenario 3).

The three following scenarios aim at explaining the various cases, when working in workpackage mode-publication exposed:





P1 and P2 can be either publication exposed products or structure exposed products (it is not a criteria here)



P3 is a publication exposed product DMU Kinematics Simulator Version 5 Release 13 Page 314 (1) In context: the tree of the selected object is displayed within the appropriate context i.e. with all its parents as far as the PRC.

(2) Without context: the tree of the selected object appears out of context. For example, if you select a CATProduct instantiated on a PRC, the CATProduct opens without the PRC and the associated instances. For more detailed information, please refer to *VPM Navigator User Guide* and *ENOVIA LCA User* guides

Recommended Methodology

The recommended methodology for working with ENOVIA LCA is:

- Send your ENOVIA document to CATIA
- work on your design in CATIA, whether from scratch or modifying an existing design.
- Save your CATIA data in ENOVIA.

To ensure seamless integration, you must have both a CATIA and ENOVIA session running

- **1.** In the Product Structure workbench of CATIA V5, click the **Init Enovia V5 Connection** icon to establish the connection between CATIA V5 and ENOVIA LCA
- 2. In ENOVIA LCA, send your ENOVIA document to CATIA.
 - °
- ^o If your document is structure exposed (explode mode), all the Kinematics commands are grayed.
 - $_{\odot}~$ If your document is in publication exposed (defined as a workpackage), all the Kinematics commands are available, go to the next step
- 3. Work on your design in your CATIA V5 application (Kinematics Simulator)
- **4.** In the Product Structure workbench of CATIA V5, click the **Save Data in ENOVIA V5 Server...** icon vour data in ENOVIA LCA database.

The save in ENOVIA dialog box appears showing objects to be saved and set to the correct save mode and save options.

The dialog box below shows Kinematics Simulator objects.

5. Simply click OK in the dialog box.

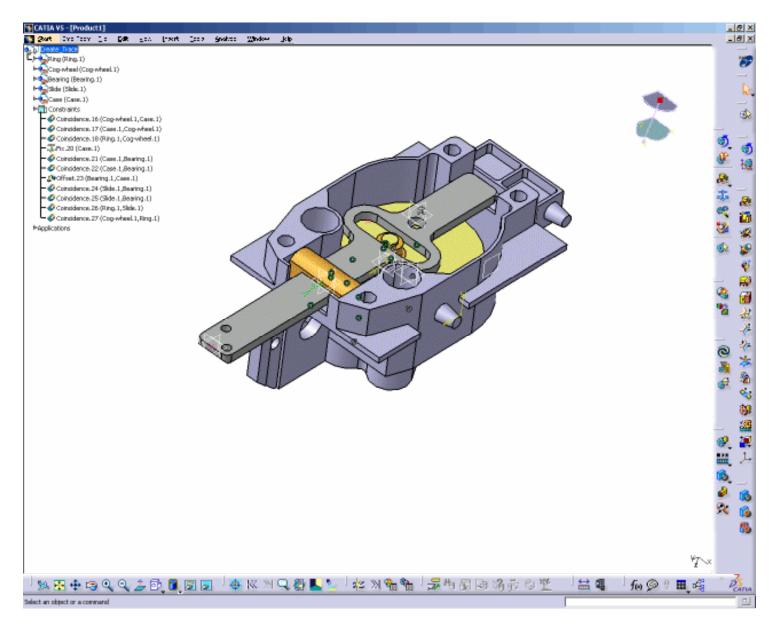
Document Name	Streamed in Vault	Content exposed	-Selected Document
(IN_Engine (ENOVIA V5 Product)	No	Yes	Document Name
exhaust.CATPart	Yes	Yes	Radiator.CATProduct
Radiator.CATProduct	Yes	No	Document Origin
5afety Valve.CATPart	Yes	No	File
Case.CATPart	Yes	No	Storage Mode
Battery.CATPart	Yes	Yes	Document kept in vault
			Publications Exposed
			O Structure Exposed
			Document not kept
			O Structure Exposed
			🖾 Immediate Commit



Workbench Description

This section contains the description of the icons and menus which are specific to the DMU Kinematics Simulator Version 5 workbench.

The DMU Kinematic Simulator window looks like this (click the sensitive areas to see the related documentation):



Menu Bar DMU Kinematics Toolbar Simulation Toolbar DMU Joint Toolbar DMU Generic Animation Toolbar DMU Kinematic Update Automatic Clash Detection Toolbar DMU Space Analysis Toolbar Specification Tree Version 5 Release 13

DMU Kinematics Simulator Menu Bar

Here we will present the various menus and menu commands that are specific to DMU Kinematics Simulator Version 5.

StartFileEditViewInsertToolsAnalyzeWindowsHelp
--

Tasks corresponding to General menu commands are described in the DMU Version 5 Infrastructure User's Guide.

Edit

Edit	⊻iew	Insert	<u>T</u> ools	An	For	Description
න ජ	Selection <u>R</u> epeat		Ctrl+Z Ctrl+Y		Undo	Cancels the last action.
X	Cu <u>t</u>		Ctrl+X		Redo	Recovers the last action that was undone.
P	Сору		Ctrl+C			
ľ	<u>P</u> aste Paste <u>S</u> pec	ial	Ctrl+V		Cut Copy Paste Paste Special	Performs cut copy paste and special paste operations.
	<u>D</u> elete		Del			
H	S <u>e</u> arch		Ctrl+F			
88 88	Selection S	ets	Ctrl+G			
ø	Selection S	ets Editio	on		Delete	Deletes selected geometry.
1	Li <u>n</u> ks P <u>r</u> operties	А	lt+Enter		Search	Allows searching and selecting objects.
	Other Selec	ction			Links	Manages links to other documents.
	Co <u>m</u> ponent: Representa			*	Properties	Allows displaying and editing object properti

Insert

For...

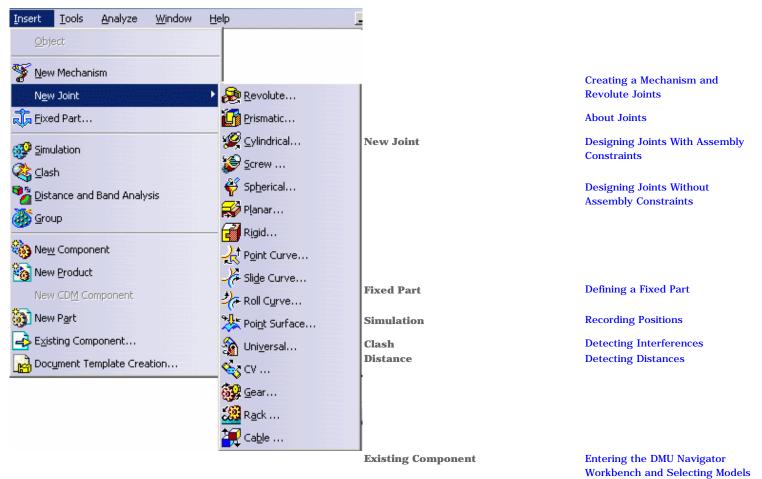
See...

New Mechanism

Creating a Mechanism and Revolute Joints

DMU Kinematics Simulator

Version 5 Release 13



Tools

<u>Fools Analyze</u>	<u>W</u> indow	F	For	See
😠 Eormula				
<u>I</u> mage		۲		
<u>M</u> acro		•		
<u>C</u> ustomize			Generate Video	DMU Fitting documentat
<u>V</u> isualization Fi	lters			
Options				
<u>S</u> tandards				
Conferencing		۲	Generate Replay	
<u>P</u> ublish		•		
Sim <u>u</u> lation		•	<u>G</u> enerate Replay	
			G <u>e</u> nerate Video	

Window

<u>W</u> indow <u>H</u> elp	For	See
New Window	now Contt Chart	
<u>T</u> ile Horizontally	Window	Displaying Gantt Chart in DMU Fitting User's Guide
<u>C</u> ascade		
1 MECHANISMProduct 2 Use_SensorsProduct (ReadOnly)	Camera Window	See Using Camera Capabilities in DMU Navigator User's Guide

DMU Kinematics Toolbar

The DMU Kinematics toolbar contains a number of tools that are useful for DMU Kinematics Simulator.





See Simulating with Commands

See Managing the Mechanism Dressup

See Creating a Fixed Part

See Creating a Mechanism and Revolute Joints See Creating Revolute Joints See Creating Revolute With Offset See Creating Revolute Joints With Centered Option

See DMU Kinematic Joints toolbar

See Converting Constraints into Joints
 See Converting Constraints into Joints (Advanced Mode)



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See: Measuring Speed and Acceleration

See Analyzing a Mechanism

Simulation Toolbar





See Simulating with Commands See Detecting Clashes Automatically

See Setting Joint Limits See Checking Joint Limits

See: Using Sensors



See Simulating With Laws See Defining Laws in a V5 Mechanism

Kinematic Joints Toolbar

The Kinematic Joint toolbar contains the various types of joints you can create in Kinematic Simulation version 5.



See About Joints



See Creating Revolute Joints See Creating Revolute With Offset See Creating Revolute Joints With Centered Option **See Creating Prismatic Joints** See Creating Cylindrical Joints See Creating Spherical Joints See Creating Planar Joints See Creating Rigid Joints See Creating Point Curve Joints See Creating Slide Curve Joints See Creating Roll Curve Joints See Creating Point Surface Joints See Creating Universal Joints See Creating Gear Joints See Creating Screw Joints See Creating Cable Joints See Creating Rack Joints See Creating CV Joints See Creating Joints Using Axis Systems

See More About Resulting Constraints

DMU Generic Animation Toolbar









See Recording Simulations



See Detecting Clashes Automatically



See Defining a Swept Volume See Defining a Swept Volume from a Moving Reference See Filtering Swept Volume Positions

See Using the Trace Command

See Generating a Trace from a V5 Mechansim

DMU Kinematic Update





See Using the Update Command

See Visualizing and Simulating Mechanisms in Sub-products



See Resetting a V5 Mechanism

Automatic Clash Detection Toolbar





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See Detecting Clashes Automatically

DMU Space Analysis Toolbar

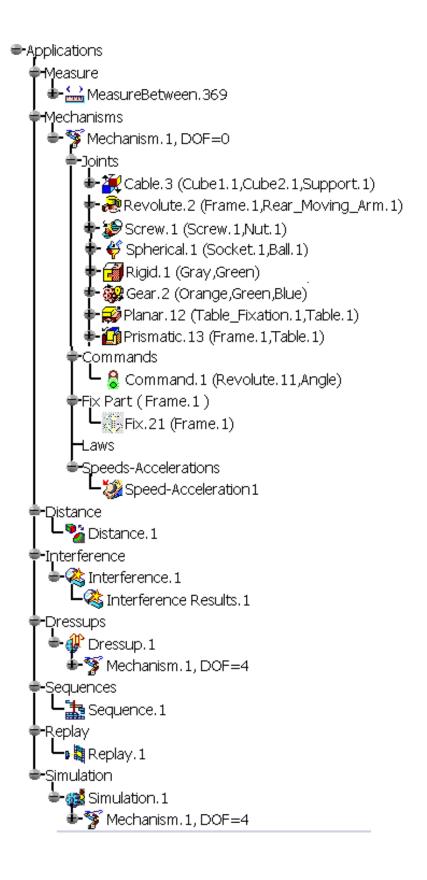




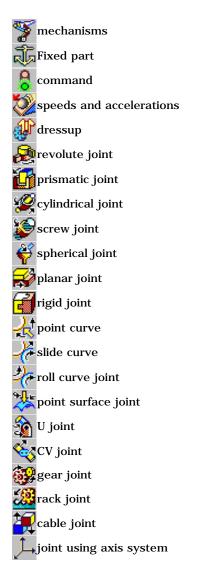
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Specification Tree

Within DMU Kinematics Simulator workbench, you can generate a certain number of features (specific to DMU Kinematics Simulator application or not specific). They are identified in the specification tree as icons under the Application node. The following image is not exhaustive but gives you an idea of what you can obtain:



Icons displayed in the specification tree and specific to the DMU Kinematics workbench identify:



Other icons (which are not specific to DMU Kinematics):

- * Distance and band analysis entries
 - Measures made using the Measure Between command
 - Clash entries
 - *D* For more information about icons specific to DMU Space Analysis workbench please refer to *Specification Tree* section in the *DMU Space Analysis User's Guide*
 - simulation entries ('Fitting' simulation)
- 📭 🚺 Replay entries
 - sequence entries
 - Laws

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DMU Kinematics Simulator

Other icons (which are not specific to DMU Kinematics):

- Distance and band analysis entries

i

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- Distance and band analysis ent
- Measures made using the Measure Between command
- Clash entries
 - For more information about icons specific to DMU Space Analysis workbench please refer to *Specification Tree* section in the *DMU Space Analysis User's Guide*
- simulation entries ('Fitting' simulation)
- Replay entries
 - sequence entries
 - Laws

i) For standard specification tree symbols, see *Specification Tree Symbols* in the *Product Structure User's Guide*.

Glossary © D F G D K D P R S U

С

cable joint	A cable type joint between three products (two products are mobile, the other is a reference). Number of degrees of freedom is 1 (translation).
cylindrical joint	A translation type joint between two products along an axis with a rotation about that axis. Number of degrees of freedom is 2 (1 translation and 1 rotation). This joint was called Actuator in Version 4.
command CV joint	An angular or linear command that drives the kinematics mechanism. A constant velocity joint between two products. Number of degrees of freedom is 4 (comprises two U joints).

D

degrees of freedom dress up	The number of possible independent rotation or translation movements of a joint. A list of models attached to a set of the kinematics model. These models have the same motion as the set.
F	
fixed product	The product that remains stationary when the kinematics mechanism is in motion.
G	
gear joint	A gear type joint between three products (two products are pinions, the other is a reference). Number of degrees of freedom is 1 (rotation).
J	
joint	A constraint between geometric entities of two or three products. There are several types of joint.
joint stop	An imposed limit applied to a joint.
Κ	

kinematics mechanism A mechanism comprising several products that are connected by joints.

kinematics product kinematics simulation	It can be simulated when the number of commands is equal to degrees of freedom (in this case the mechanism is said to be <i>complete</i>). A rigid product defined in a single geometric set that contains all the elements required to describe the kinematics mechanism and its motion. A simulation of the mechanism's motion using commands. Simulation can be immediate (commands are used one by one) or on request (one or more commands are used with a given number of steps).
L	
law	A numeric or graphic representation of the commands applied to a kinematics mechanism as a function of time.
Р	
planar joint	A planar joint between two products. Number of degrees of freedom is 3 (1 rotation and 2 translations).
prismatic joint	A translation joint between tow products along an axis with no rotation about that axis. Number of degrees of freedom is 1 (translation).
PT/CRV joint	A point/curve joint between two products. Number of degrees of freedom is 4 (3 rotation and 1 translation) for a 3D mechanism and 2 (1 rotation and 1 translation) for a 2D mechanism.
PT/SUR joint	A point/surface joint between two products. Number of degrees of freedom is 5 (3 rotations and 2 translations).
R	
rack joint	A gear/rack type joint between three products (one product is the rack, another is the rack, the other is a reference). Number of degrees of freedom is 1 (combined translation and rotation).
revolute joint	A revolute joint about an axis between two products with no translation along that axis. Number of degrees of freedom is 1 (rotation).
rigid joint	A rigid (fully restricted) joint between two products. There are no degrees of freedom associated to this joint.
roll/CRV joint	A rolling type joint between two products that include curves. There is no sliding motion with this type of joint. Number of degrees of freedom is 2 (1 rotation and 1 translation) for a 3D mechanism and 1 (translation) for a 2D mechanism.
S	
screw joint	A screw/nut type joint between two products relative to an axis. Number of degrees of freedom is 1 (combined translation and rotation).

slid/CRV joint	A rolling type joint with a sliding motion between two products that include curves. Number of degrees of freedom is 3 (2 rotations and 1 translation) for a 3D mechanism and 2 (1 rotation and 1 translation) for a 2D
	mechanism.
spherical joint	A spherical joint between two products. Number of degrees of freedom is 3
	(3 rotations) for a 3D mechanism and 1 (rotation) for a 2D mechanism. This joint was called PT/PT in Version 4.
storyboard	A recorded kinematic motion.
TT	
U	
Ujoint	A universal joint between two products. Number of degrees of freedom is 2
- Jomr	(2 rotations).

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