STUDENT GUIDE

Student Notes:

DASSAULT SYSTEMES

Cross-Brand Training

Foils

DMU Kinematics Simulator

Version 5 Release 21
January 2012

EDU_CRB_EN_KIN_FF_V5R21

Student Notes:

Course Presentation

Objectives of the course

Upon completion of this course you will be able to:

- Understand and use the capabilities and the general processes followed in the DMU Kinematics workbench
- Define a mechanism using an existing assembly
- Simulate the mechanism
- Analyze the mechanism for clashes
- Perform kinematic analysis
- Sequence multiple mechanisms

Targeted audience

Designers (CATIA P2 users only)

Prerequisites

Students attending this course should be familiar with DMU Basics and DMU Space Analysis.

8 hours

Student Notes:

Table of Contents (1/3)

Student Notes:

Table of Contents (2/3)

Student Notes:

Table of Contents (3/3)

Student Notes:

Overview of DMU Kinematics

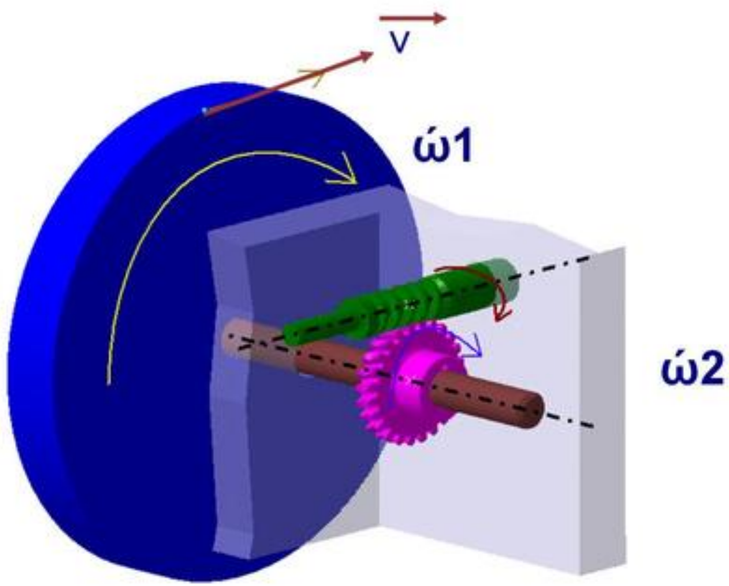
You will be introduced to the DMU Kinematics workbench.

- What is Kinematics?
- Capabilities of DMU Kinematics Workbench
- General Process of Mechanism Design in DMU Kinematics
- To Sum Up

Student Notes:

Overview of DMU Kinematics

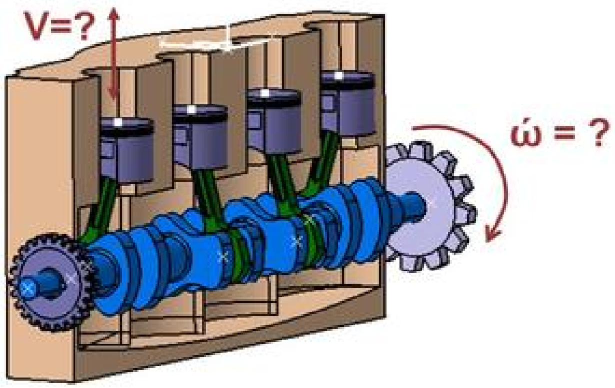
This lesson will give you an overview of the DMU Kinematics workbench capabilities and the general process followed to create a mechanism.

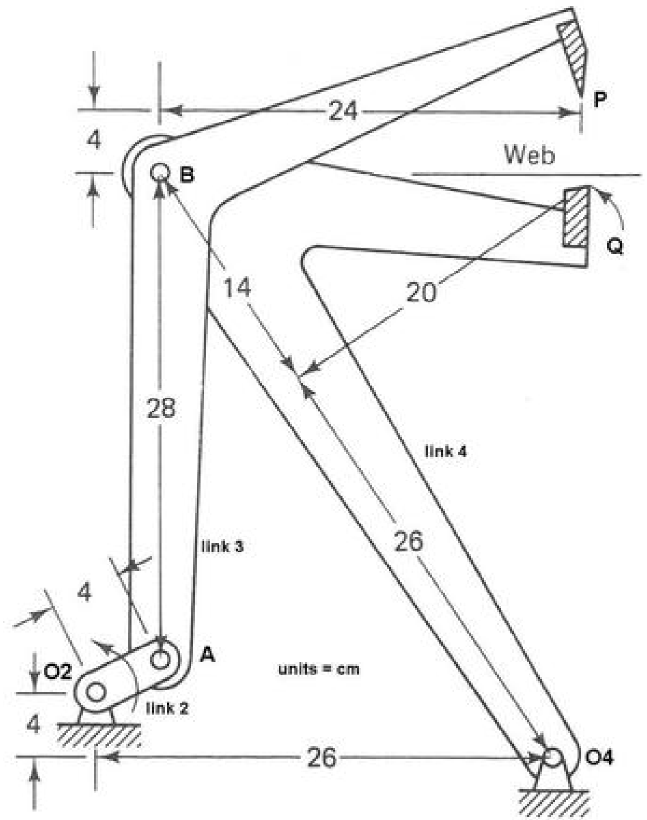


Student Notes:

What is Kinematics?

Kinematics is a branch of motion science, which involves the study of the displacement, velocity and acceleration of a body without taking into account the forces involved in the motion.





Kinematics is required to:

- Determine the direction of motion of the assembled parts with respect to other parts.
- Find the velocity and acceleration of any point on any part, at any specific position (during motion) of the mechanism.
- Analyze the time required for the motion, and ultimately design the mechanism.

Student Notes:

Capabilities of DMU Kinematics Workbench

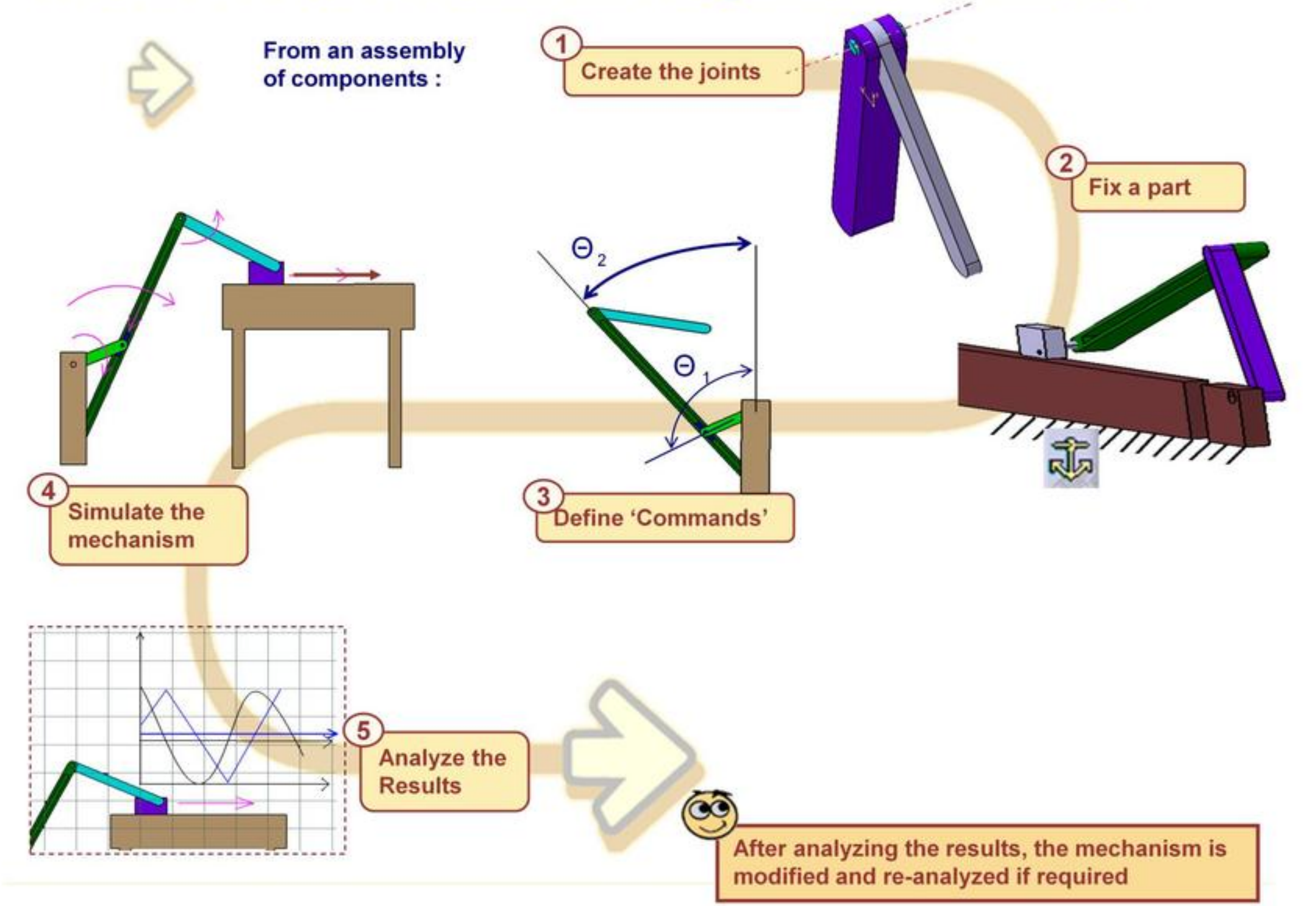
The DMU Kinematics Workbench helps to design mechanisms from the preliminary project concept.

- **Using the 'DMU Kinematics Workbench' you can:**
  - **Directly assemble the parts within a mechanism without using the 'Assembly Design' workbench.**
  - **Give necessary motion instructions to the assembled parts so that they move the way you want them to.**
  - **Analyse the motion and determine any parameters related to motion like velocity, acceleration, distance etc.**
  - **Find the time required for any motion of the mechanism or also get any motion completed in the given time.**
  - **Animate multiple mechanisms in series and/or in parallel.**
  - **Place sensors and perform analysis on the mechanisms.**
  - **Ultimately design the mechanism performing the useful motion.**

STUDENT GUIDE

Student Notes:

General Process of Mechanism Design in DMU Kinematics



To Sum Up

In this lesson, you have seen :

- The capabilities of the DMU Kinematics Workbench.
- The General Process that is involved in DMU Kinematics.

Student Notes:

Defining a Mechanism

You will learn to define mechanisms from the assembly of parts.

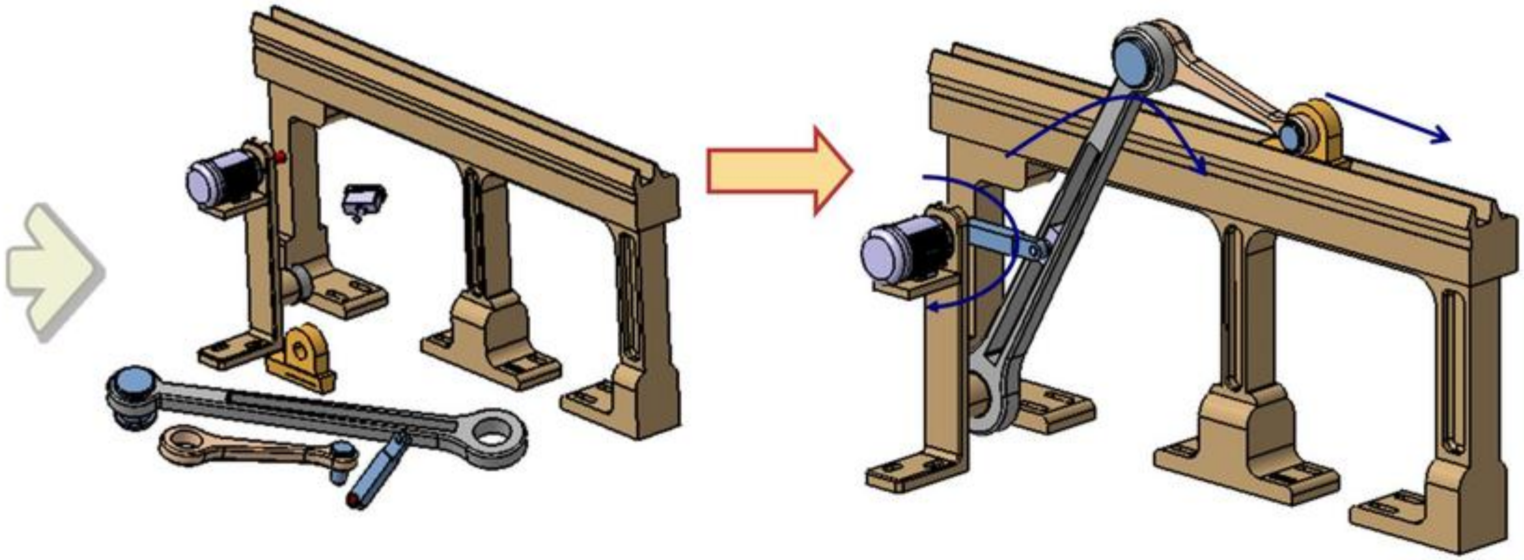
- General Process to Define Mechanisms
- Creating Joints
- How to Fix a Part in the Mechanism
- To Sum Up

Student Notes:

Student Notes:

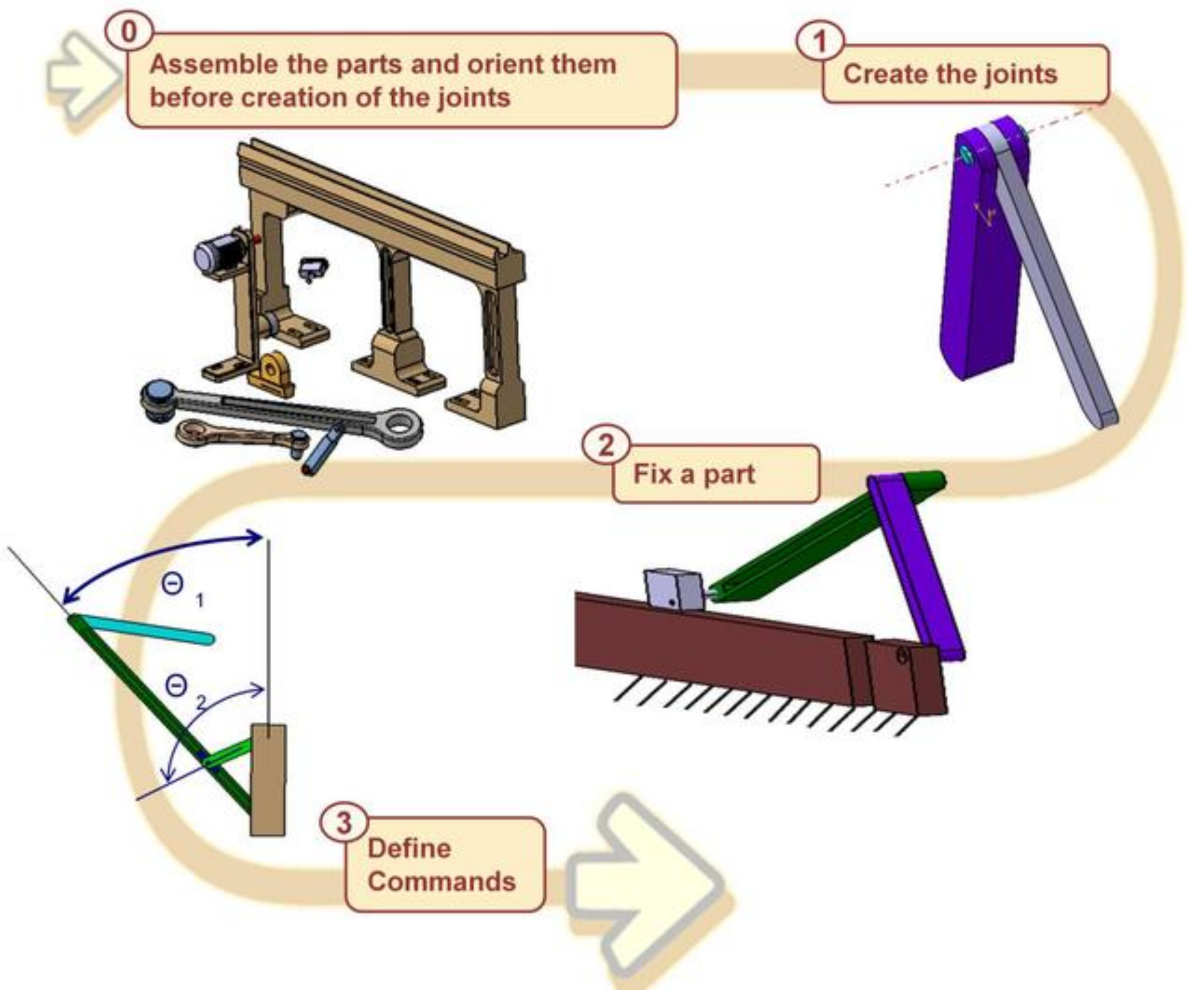
Defining a Mechanism

You will learn to define mechanisms from the assembly of parts.



STUDENT GUIDE

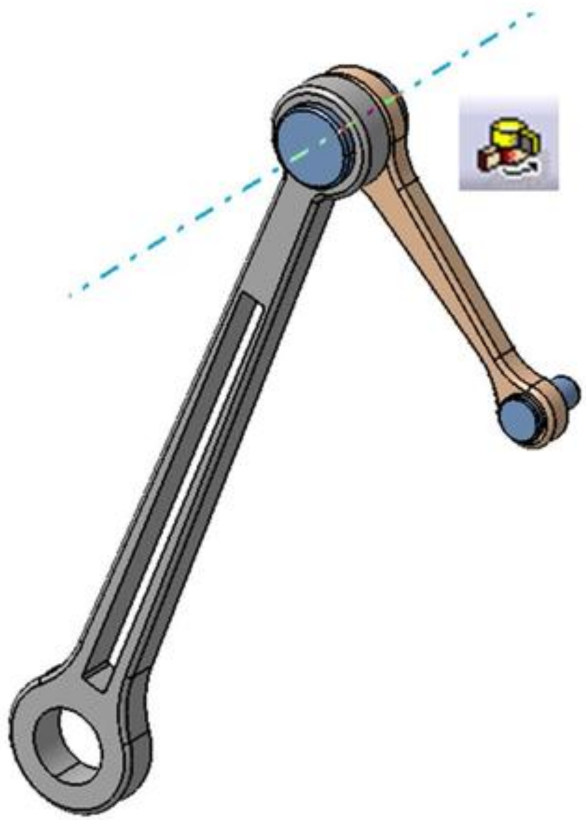
General Process to Define a Mechanism



Student Notes:

Creating Joints

You will learn to define various types of joints.

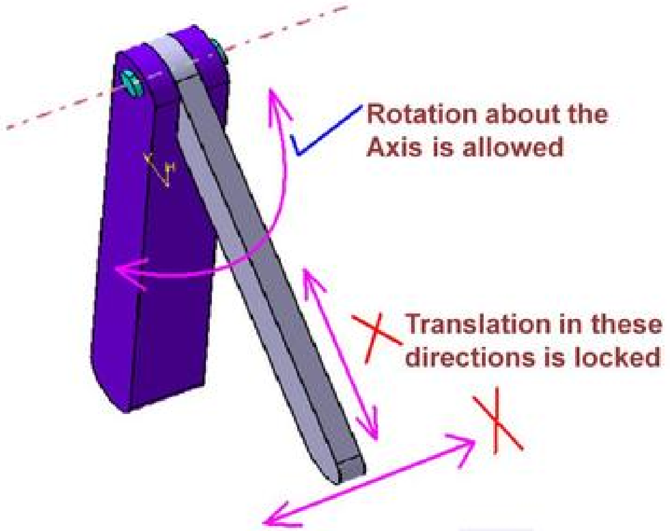


Student Notes:

Student Notes:

About Joints

A Joint between two parts is an assembly of the parts, permitting only certain desired relative motion between them.



The Revolute Joint 

Every joint creation locks certain ‘Degrees of freedom’ of the involving components

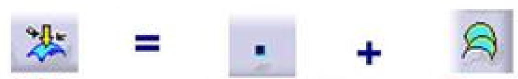
Classification of Joints:

Joints are Classified mainly into 4 types

- **Joints Using Assembly Constraints**

Joint = Constraint + Constraint

- **Joints using Topological or geometric elements**

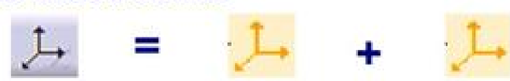


Joint = Geometric + Topological Elements

- **Compound joints which take other joints as input**

Compound Joint = Joint + Joint

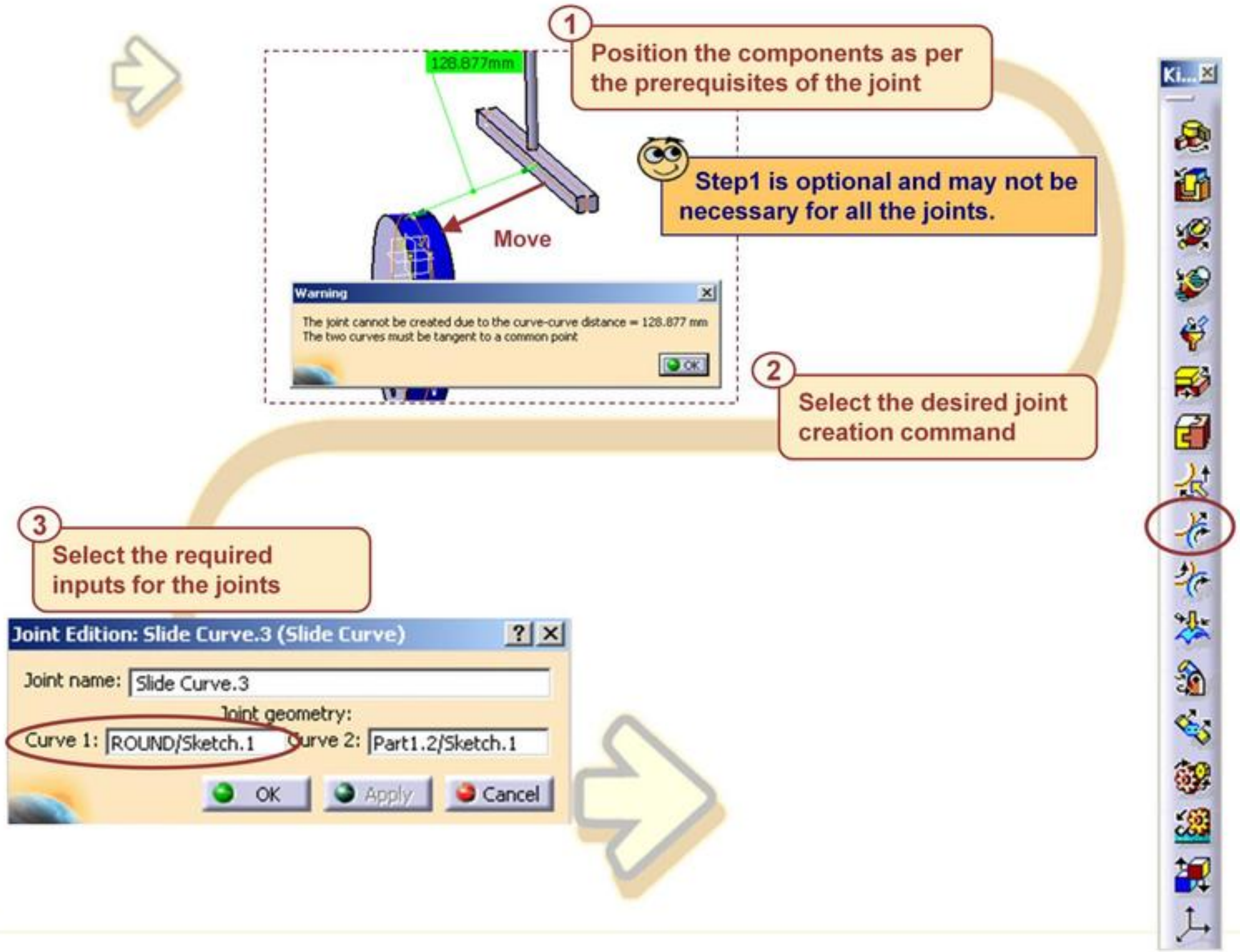
- **Axis based Joints**



Axis Based Joint = Axis System + Axis System

Student Notes:

General Process for Creating Joints



Student Notes:

Joints Using Assembly Constraints

In this lesson you will learn to create joints that use assembly constraints.

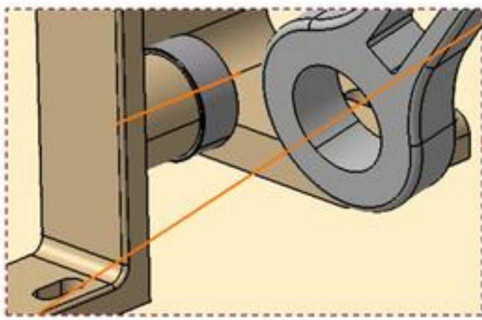
Student Notes:

How to Create a Revolute Joint

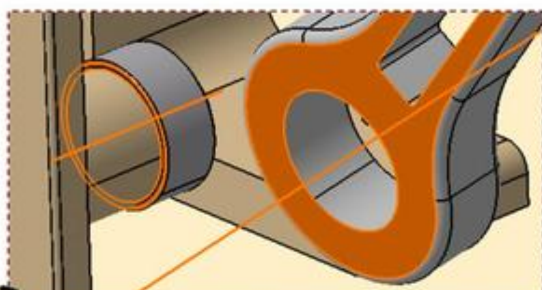
A 'Revolute Joint' requires two lines (or axes) of two different parts and two planes of the two parts. This joint keeps the selected lines (axes) always collinear and the selected planes always mutually coincident during the relative motion of the parts.

1. Click on the 'Revolute Joint' tool.

3a. Select the required lines (Axes).



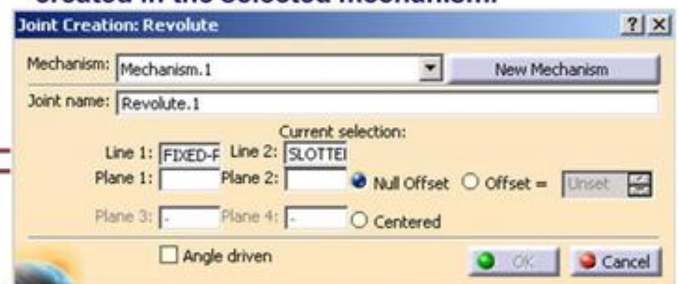
3b. Select the required planes.



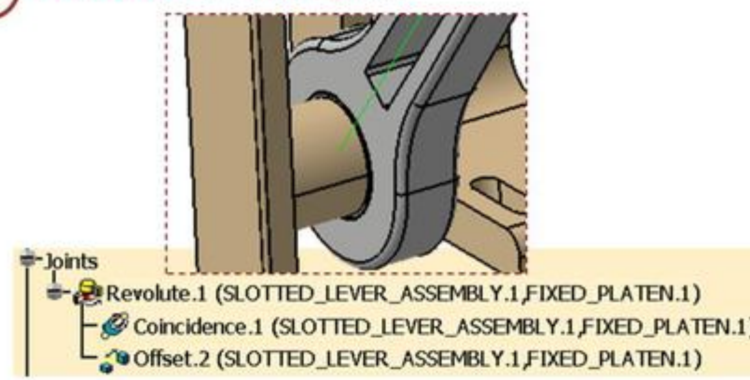
The line and plane you select for the part must be mutually perpendicular.

2. Define a new mechanism by clicking the 'New Mechanism' button in the 'Joint Creation' panel.

You can also select an existing mechanism from the 'Mechanism' drop box in the panel. The joint will be created in the selected mechanism.



4. Click OK to create the joint.



Student Notes:

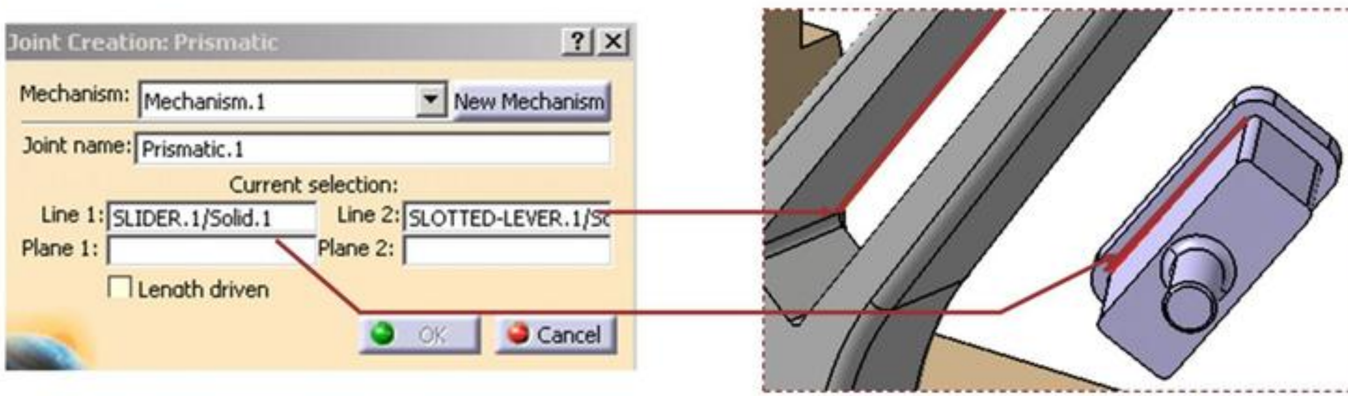
How to Create a Prismatic Joint (1/2)

The 'Prismatic Joint' requires two lines of two different parts and two planes of two parts. This joint keeps the selected lines always collinear, and the selected planes always mutually coincident during the relative motion of the parts.

1 Click the 'Prismatic Joint' tool.

2 Select the required inputs.

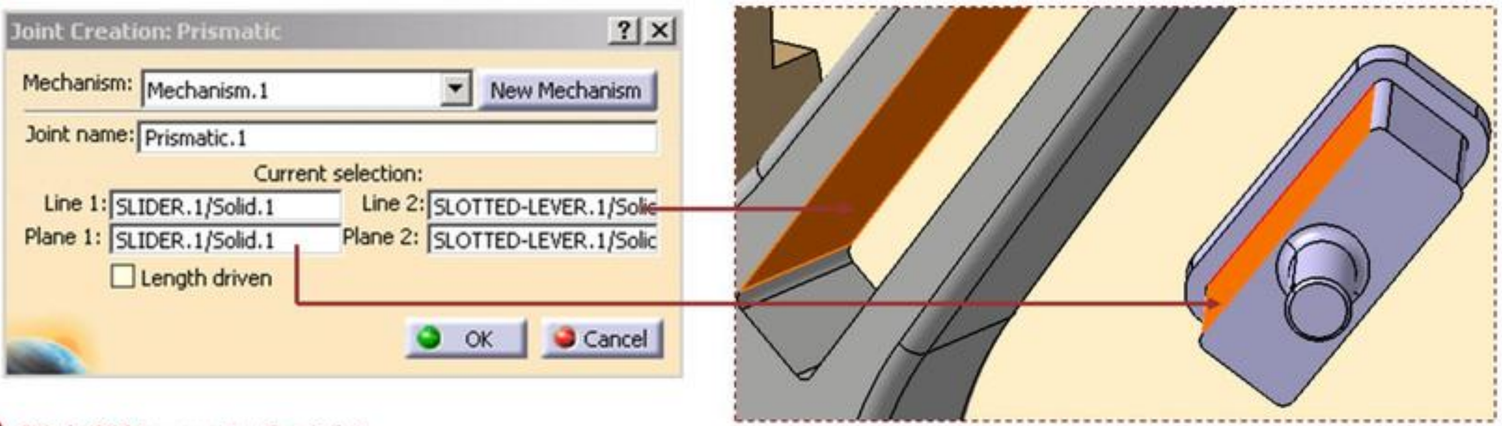
2a Select the required lines.



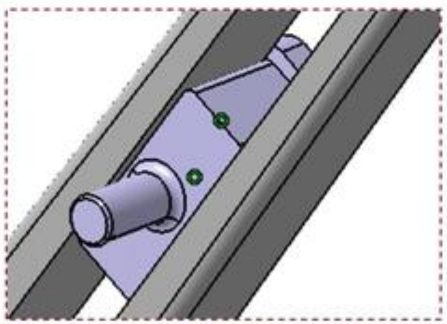
Student Notes:

How to Create a Prismatic Joint (2/2)

2b Select the required planes.



3 Click OK to create the joint.



Prismatic.2 (SLIDER.1,SLOTTED_LEVER_ASSEMBLY.1)
- Coincidence.3 (SLIDER.1,SLOTTED_LEVER_ASSEMBLY.1)
- Coincidence.4 (SLIDER.1,SLOTTED_LEVER_ASSEMBLY.1)

The line that you select for a part must be contained in the corresponding plane of the part.

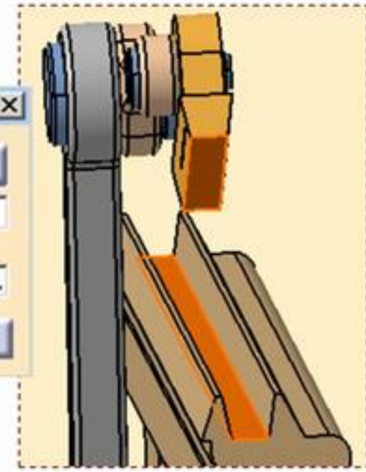
Student Notes:

How to Create a Planar Joint

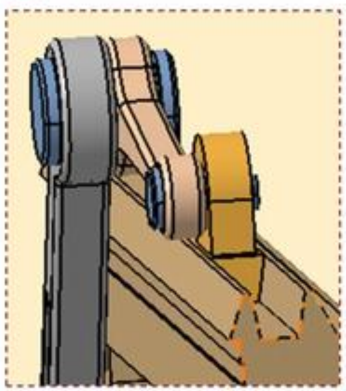
The 'Planar Joint' requires two planes or two planar faces of two different parts. This joint keeps the selected planes mutually coplanar during the relative motion of the two parts.

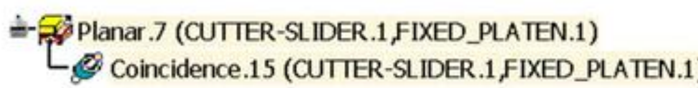
1 Click the 'Planar Joint' tool.

2 Select the required planes.



3 Click OK to create the joint.





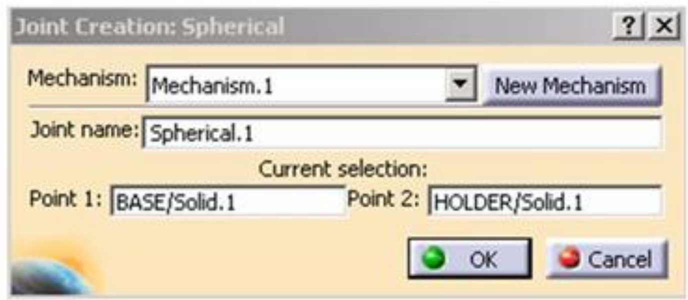
Student Notes:

How to create a Spherical Joint

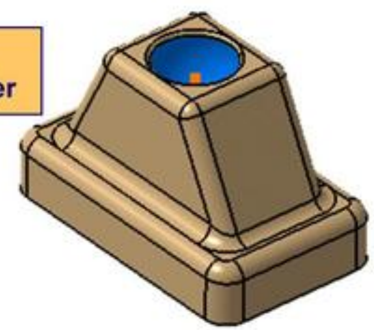
The 'Spherical Joint' requires points from two different parts as an input. This joint will keep the selected points coincident during the relative motion of the two parts.

1 Click the 'Spherical Joint' tool.

2 Select the required points OR select the required spherical faces of the parts.



Selecting the spherical faces ultimately selects the sphere center



3 Click OK to create the Joint.

Spherical.1 (HOLDER,BASE)
Coincidence.1 (HOLDER,BASE)

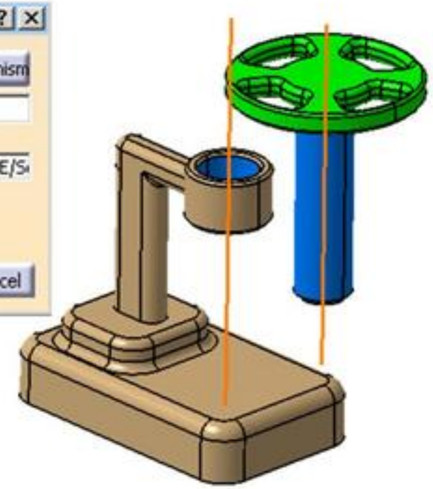
Student Notes:

How to create a Screw Joint

The 'Screw Joint' requires two lines (or axes) of two different parts as input.The screw joint keeps the selected lines (or axes) collinear and maintains a combination of rotating and linear (screw like) motion in between the components.

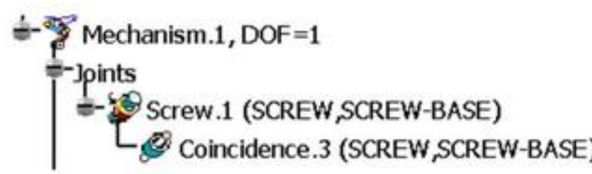
1. Click the 'Screw Joint' tool.

2. Select the required lines (Axes).



Pitch = 10 means the screw part will move 10 mm in the axial direction during its completion of one revolution about the axis

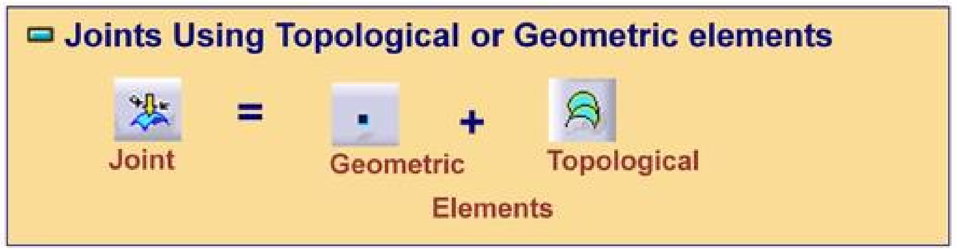
3. Click OK to create the joint.



Student Notes:

Joints Using Geometric Elements

In this lesson you will learn to create the joints using topological or geometric elements.



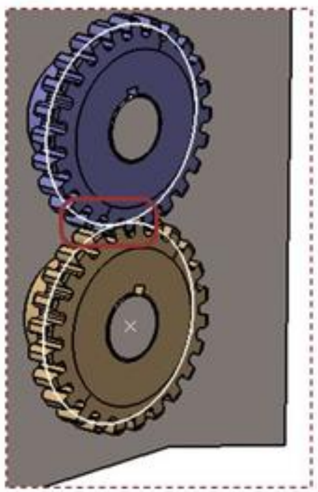
Student Notes:

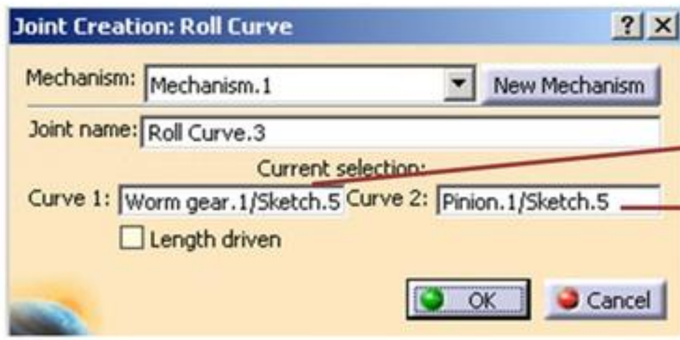
How to Create a Roll Curve Joint

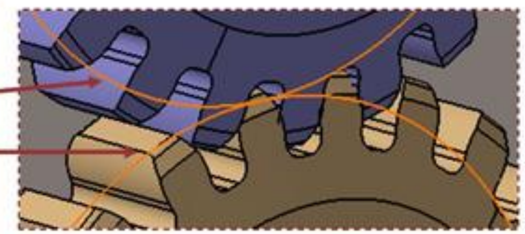
The 'Roll Curve Joint' requires two coplanar curves of two different parts as input. In this joint, the relative motion between the two parts is such that the two curves roll over each other without sliding.

1. Click the 'Roll Curve Joint' tool.

2. Select the required input curves.







3. Click OK to create the joint.

Roll Curve.3 (Worm gear.1,Pinion.1)

The two curves must be coplanar and tangent at a point before the creation of the joint. So before the creation of this joint, the parts may be required to be positioned.

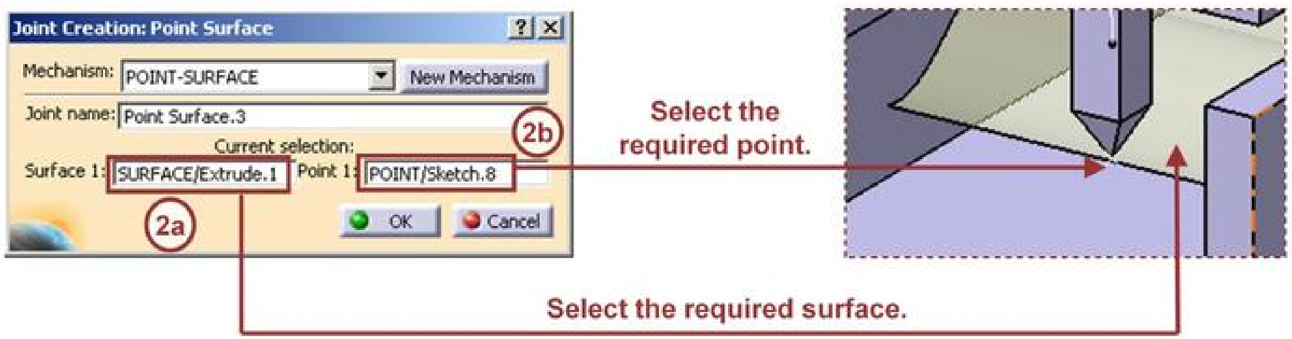
Student Notes:

How to Create a Point Surface Joint

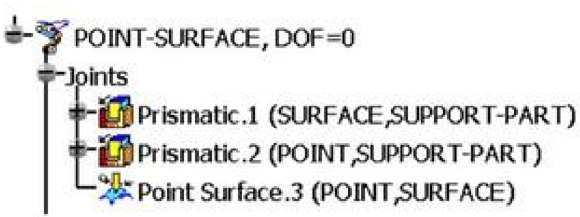
The ‘Point Surface Joint’ needs a point (or vertex) from one part and a surface from the other part. This joint will keep the point on the surface during the relative motion of the parts.

1. Click the ‘Point Surface Joint’ tool.

2. Select the required inputs.



3. Click OK to create the joint.

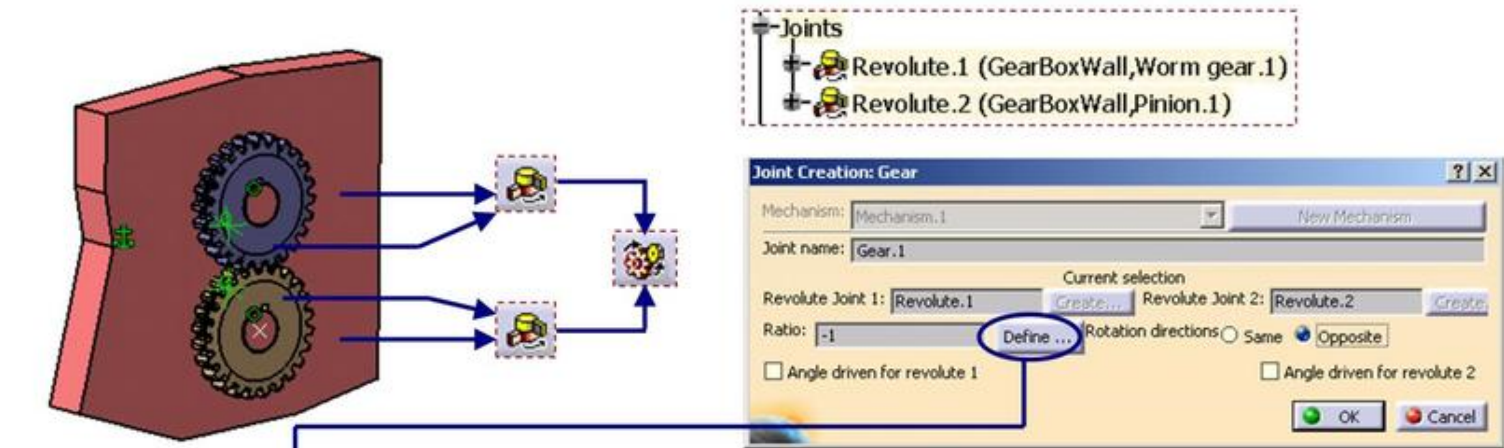


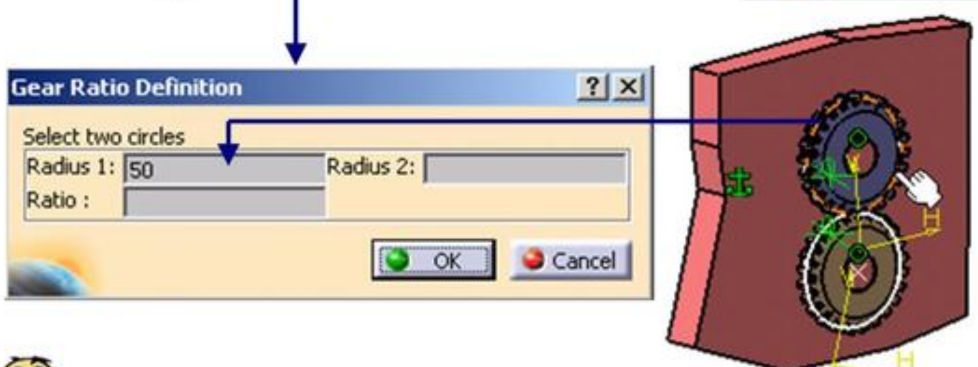
The point and the surface for this joint need to be coincident before the creation of this joint. So they may be required to be positioned before the joint creation.

Student Notes:

Gear Joint

A Gear Joint is a compound joint which involves two Revolute joints.





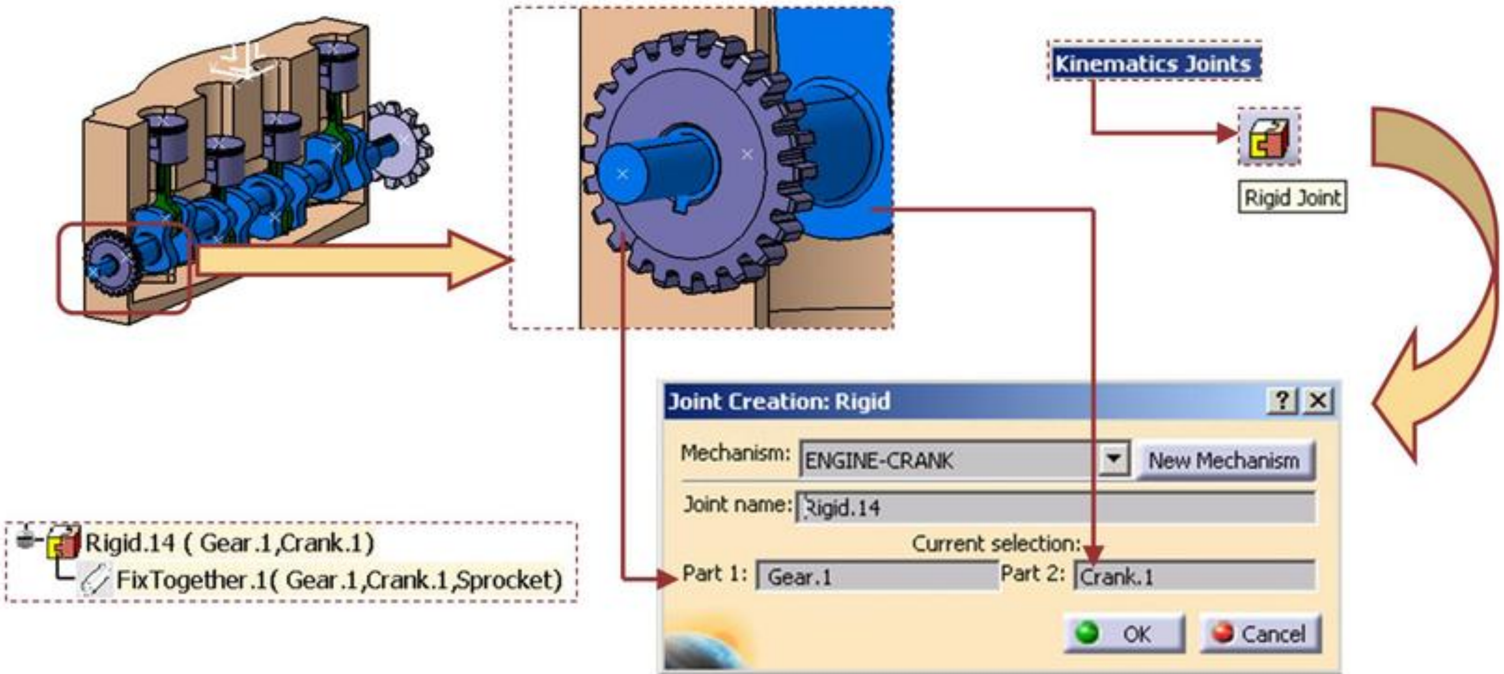
You can also define the Gear Ratio on the fly by selecting the radii of the mating gears.

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

Student Notes:

The ‘Rigid’ Joint

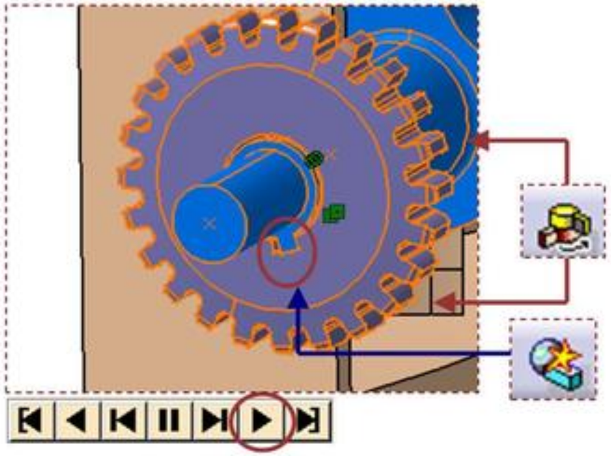
The ‘Rigid’ joint is used to fix two components in the mechanism such that there is no relative motion between them.



In this case, the gear and the crank shaft are supposed to rotate at the same time. However there is no relative motion between these components. Hence a ‘Rigid’ joint should be created for such case.

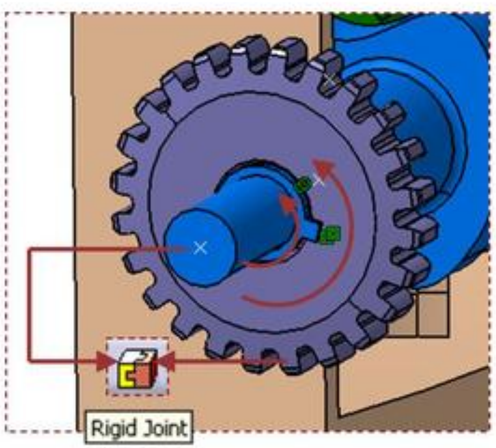
Student Notes:

Why create a 'Rigid' Joint



In this example, a revolute joint exists between the crank shaft (blue colored) and the gear housing. The gear is not involved in any of the joint.

When this mechanism is simulated, the crank shaft moves and causes a clash with the gear which is still.



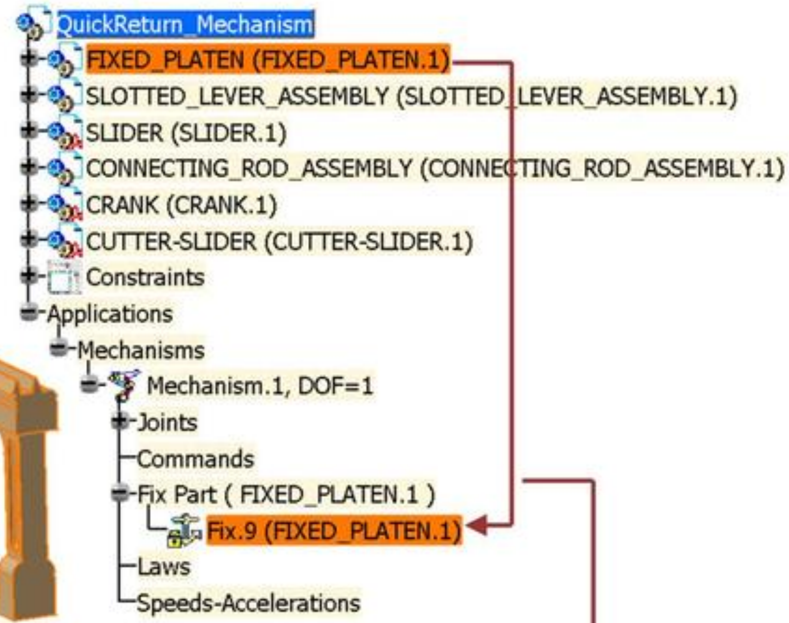
Creating 'Rigid' joint will create a 'Fix Together' constraint between the components. Hence, both the components will move during simulation.

How to Fix a Part in the Mechanism

Motion is relative to a reference. In DMU Kinematics, by defining a fix part, we define a reference for simulating and measuring the motion parameters.

1. Click on the 'Fixed Part' tool.

2. Select the part to be fixed either from specification tree or from graphics area.



The selected component will be considered as the 'Fixed Part' for the mechanism.

Student Notes:

To Sum Up

In this lesson, you have learned to define mechanism by learning:

- Various types of Joints' Creation methods
- Fixing a part in a mechanism

Student Notes:

Simulating Mechanisms

In this lesson you will learn to simulate and sequence mechanisms using commands and laws.

- General Process for Simulating Mechanisms
- How to Define Commands for the Mechanism
- How to Simulate Mechanisms with Commands
- How to Simulate Mechanisms with Laws
- Sequencing Mechanisms
- Resetting a Mechanism
- To Sum Up

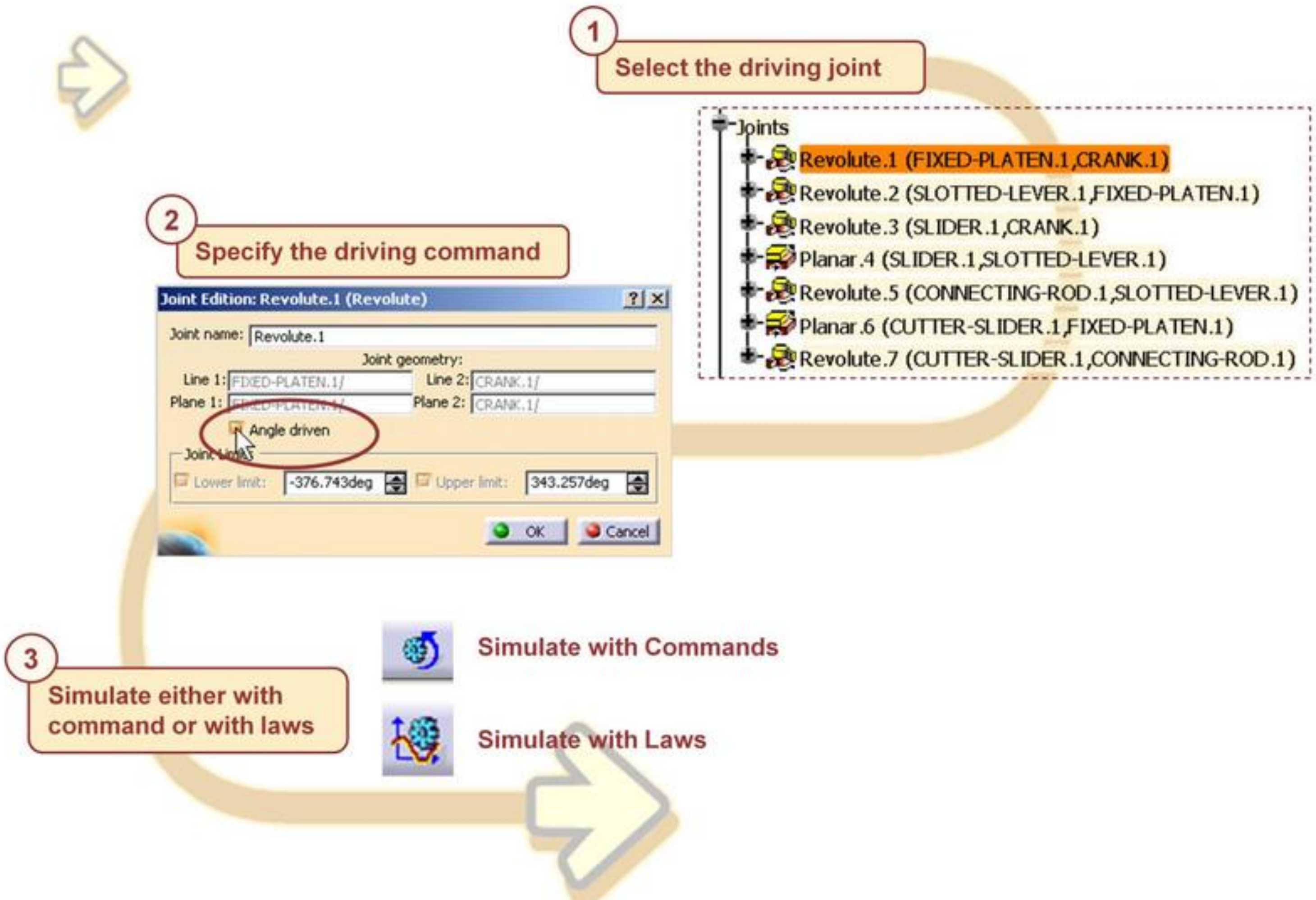
Student Notes:

Student Notes:

Simulating Mechanisms

In this lesson you will learn to simulate and sequence mechanisms using commands and laws.

General Process for Simulating Mechanisms



Student Notes:

Student Notes:

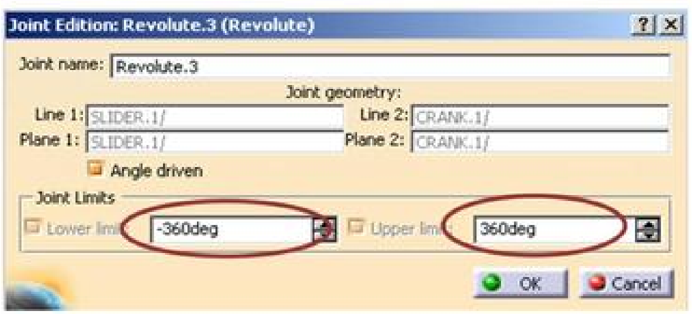
How to Define Commands for the Mechanism

The command is the driving instruction for the mechanism.

1 Double-click the joint which will impart motion to the mechanism.

2 Select the driven parameter.

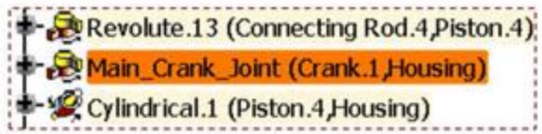
3 Change the joint limits if you want limits other than the default joint limits.



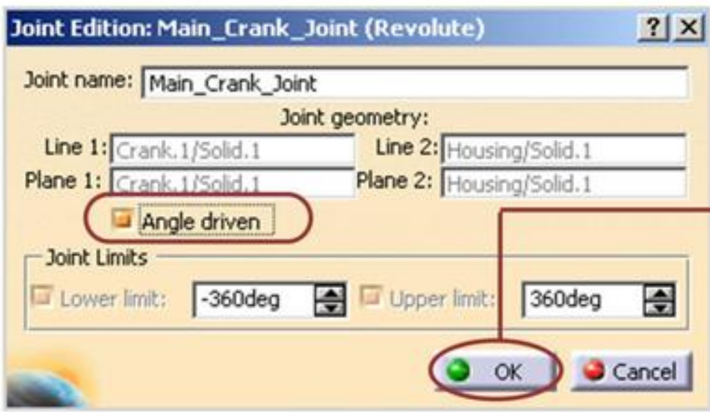
4 Click OK and the 'Command' for joint will be created with the selected driven parameter.

How to Simulate Mechanisms with Commands (1/2)

1. Select the driving joint by double clicking it.

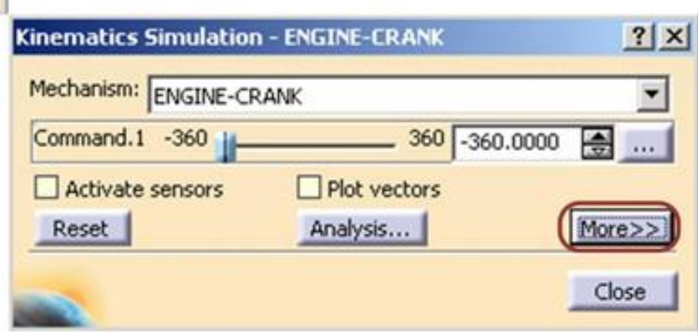


2. Select the driving command and click OK.



A message box informing that the mechanism can be simulated will appear. Click OK to this message box.

3. Select the “Simulation with commands” tool to open the ‘Kinematic Simulation’ panel and click on ‘More>>’ button (to display the full dialog box) if required.



Student Notes:

Student Notes:

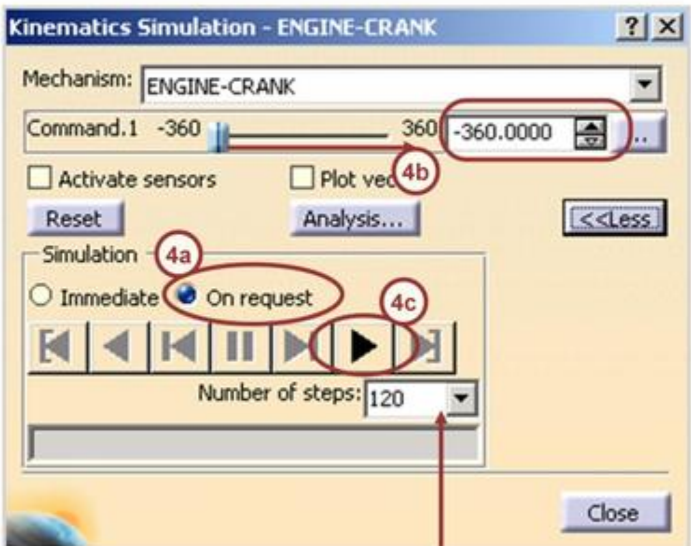
How to Simulate Mechanisms with Commands (2/2)

4 For the Kinematics Simulation Panel,

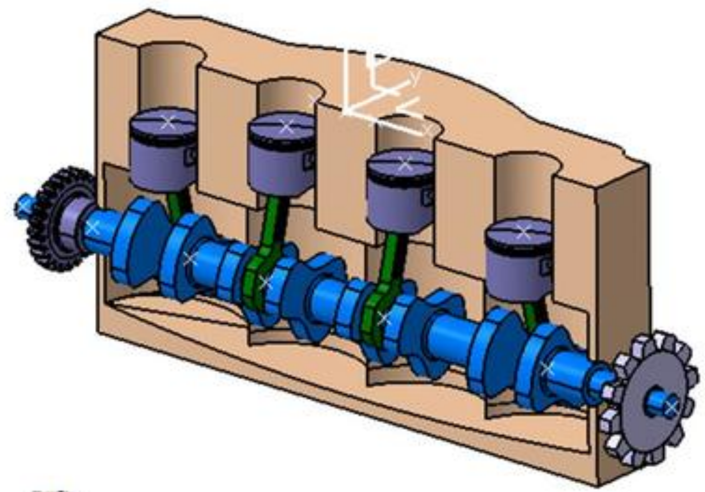
4a Select the 'On request' radio button,

4b Change the value of command by dragging the slider and

4c Click the 'Play Forward' button.



You can increase the 'Number of steps' in the panel to obtain a smoother simulation.

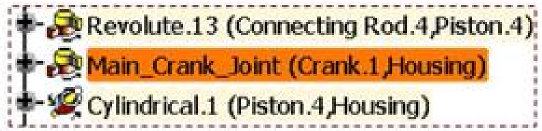


The mechanism will be simulated and the components can be seen moving until the new value of the command is reached.

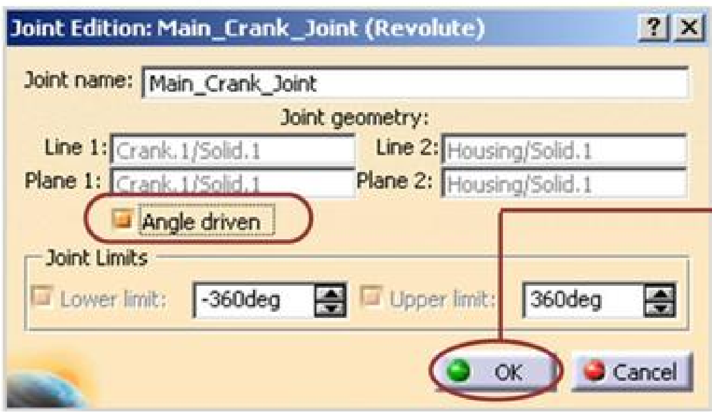
Student Notes:

How to Simulate Mechanisms with Laws (1/4)

1 Select the driving joint by double-clicking it.

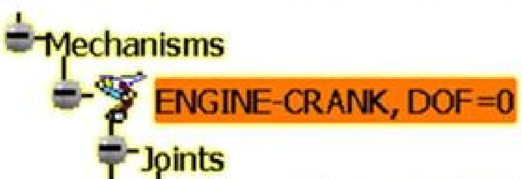


2 Select the driving command and click OK.



A message box informing that the mechanism can be simulated will appear. Click OK in this message box.

3 To define laws, select the desired mechanism and click the f(x) tool to add a relation between the 'Driving Parameter' and time.



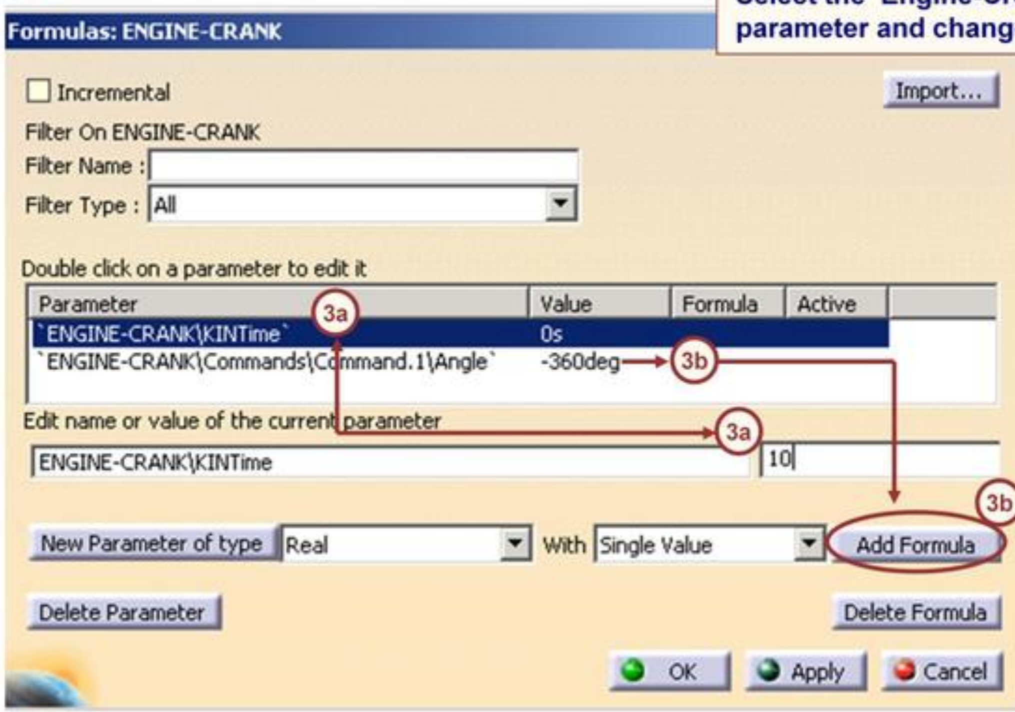
The panel which shows the parameters related to the mechanism is shown in the next slide

Student Notes:

How to Simulate Mechanisms with Laws (2/4)

3a **Select the time parameter and change its value.**

For this Example:
Select the ‘Engine-CRANK\KINTime’ parameter and change its value to ‘10s’



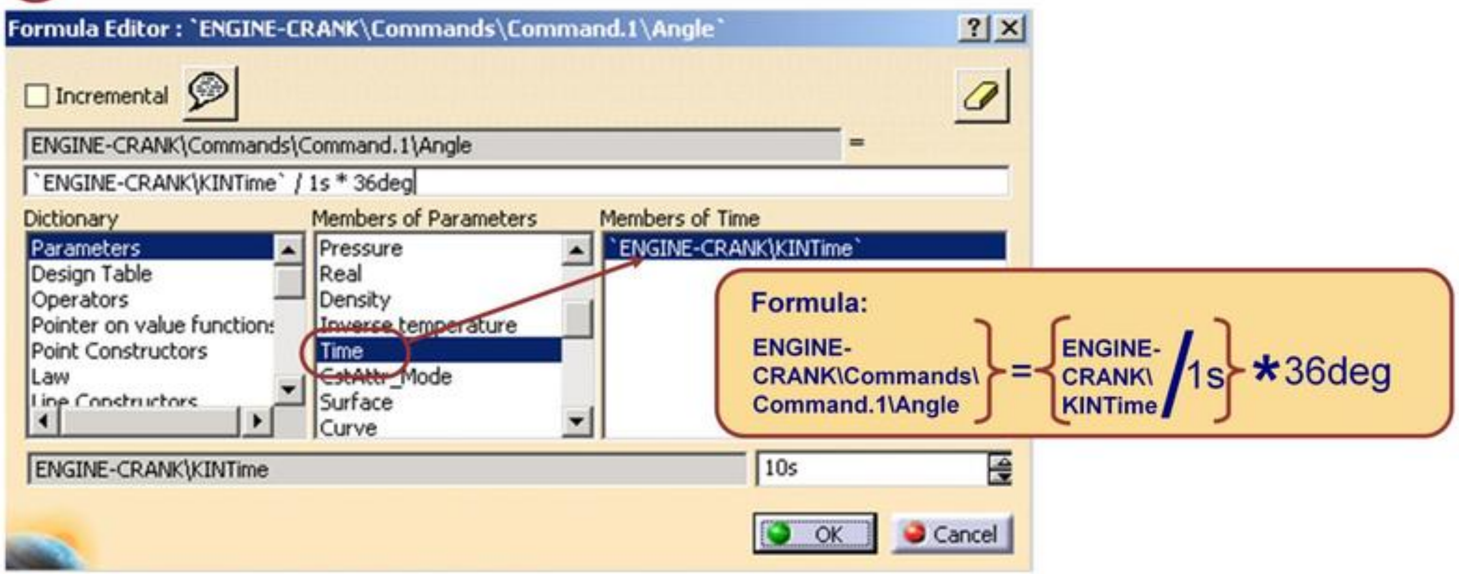
3b **Select the driving command “…Command.1\Angle” and click on the ‘Add Formula’ button.**

For this example:
Select the driving command ‘ENGINE-CRANK\Commands\Command.1\Angle’ and click the ‘Add Formula’ button to open the Formula Editor panel.

Student Notes:

How to Simulate Mechanisms with Laws (3/4)

3c Click on 'Time' in the 'Members of Parameters' list to filter out the 'Time' parameters.



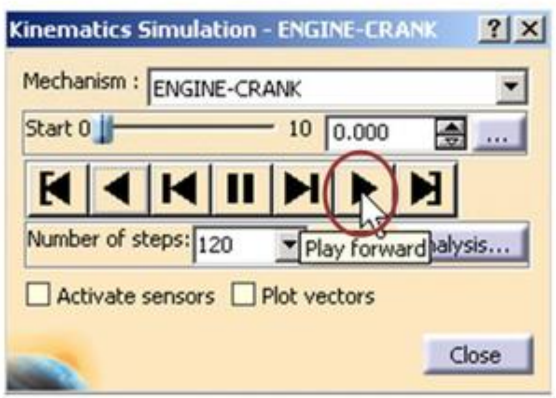
3d Specify the relation between the 'Kinematic time' of mechanism and the driving command

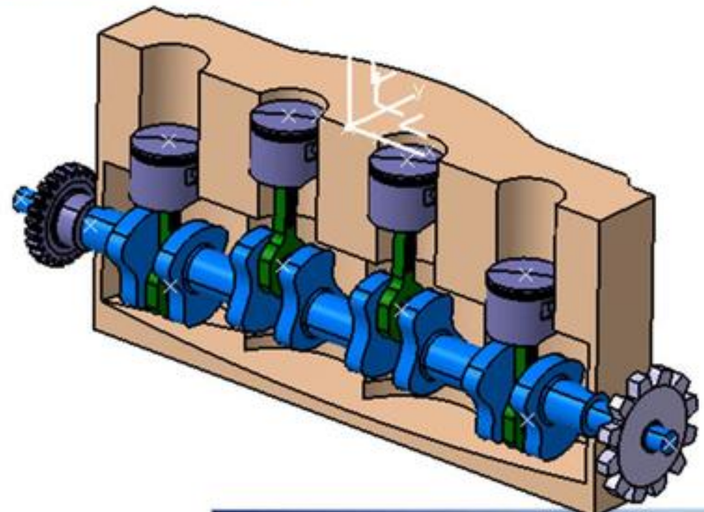
For this example:
Double-click on 'ENGINE-CRANK\KINTime' in the 'Members of Time' list and add " /1s*36deg " after "ENGINE-CRANK\KINTime" in the edit box of the panel, and click OK.

The above formula means that in 1 second the crank angle will change by 36 degrees

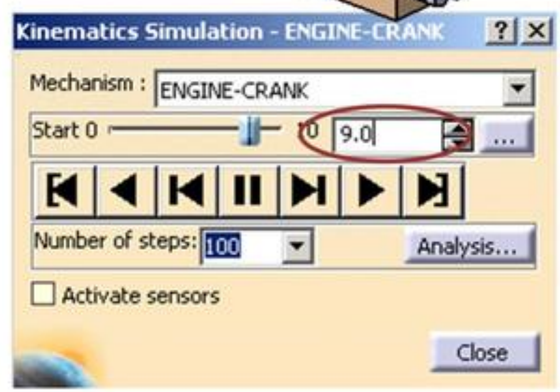
How to Simulate Mechanisms with Laws (4/4)

4 Click on the 'Simulating with Laws' tool and then on the 'Play Back' or 'Play Forward' button to simulate the mechanism.





You can also find any intermediate position of the mechanism at any point of time by entering the value of time in the edit box and pressing the <Enter> key.



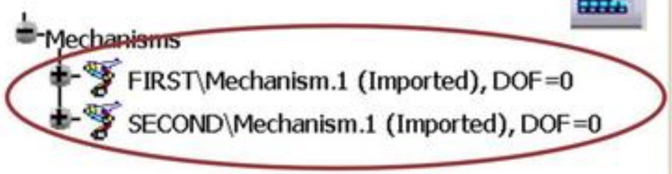
Student Notes:

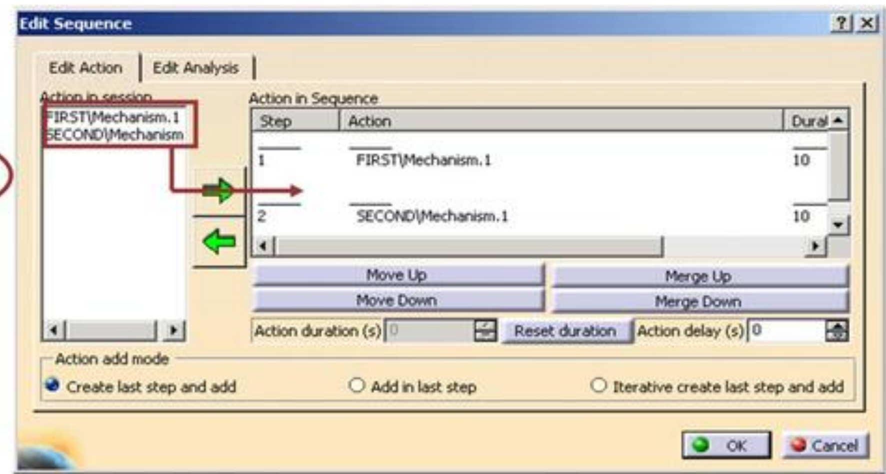
Student Notes:

Sequencing Mechanisms

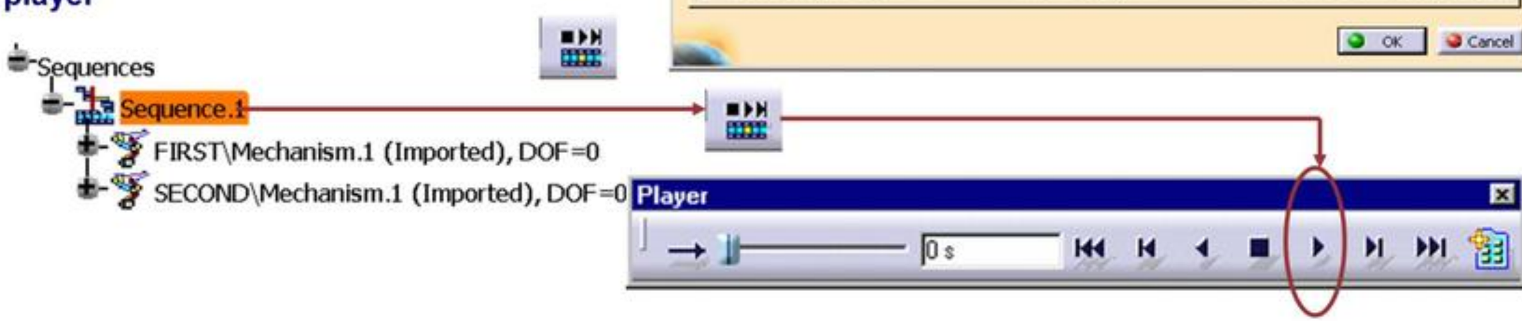
Sequencing mechanism means, playing the simulations of more than one mechanism one after the other, or simultaneously.

You can sequence your mechanisms using the sequencer tool.





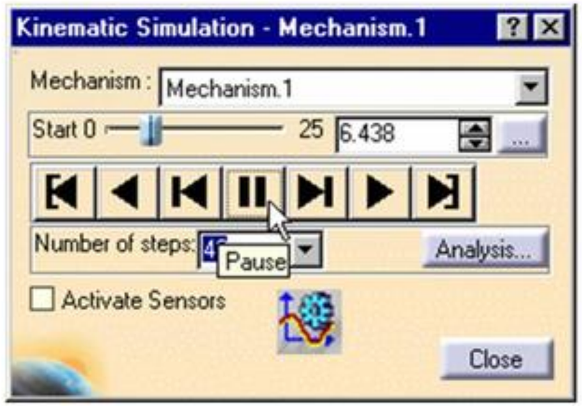
The sequence can be played using the player



Student Notes:

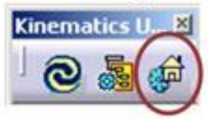
Resetting a V5 Mechanism

At the end of a simulation with laws or with commands, the modified positions or components are retained. You may need to switch the components to their initial position.

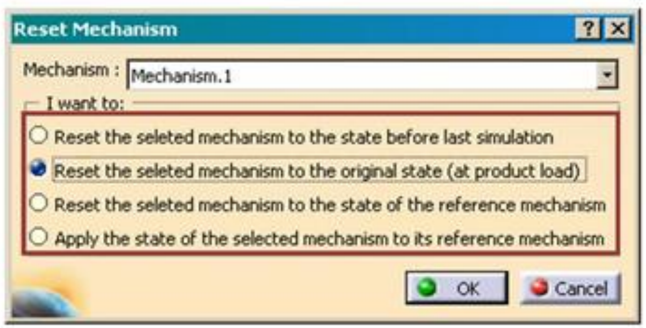


When you exit a simulation, the last position is kept.

Also when you import a sub-mechanism, a 'local copy' of the sub-mechanism is created, and if you simulate it, it becomes desynchronized from its reference.



The 'Reset Position' tool allows you to switch the product to his initial position and to re-synchronize an imported mechanism with its reference.



The 'Reset Position' tool allows you to swap the components' position to the positions they had at different states. You have 4 options:

- **State 1: Before the last simulation**
- **State 2: To the original state**
- **State 3: To the state of the reference mechanism**
- **State 4: The position of the imported mechanism will be applied to its reference mechanism**

To Sum Up

In this lesson, you have seen :

- Simulation of Mechanisms with Commands
- Simulation of Mechanism with Laws
- Sequencing multiple Mechanisms

Student Notes:

Student Notes:

Analyzing Movements

In this lesson you will learn to analyze and trace the movements of parts in the mechanism during the mechanism simulation.

- Simulation Outputs : Sensors
- How to Check Joint Limits
- Plotting Graphs and Exporting Results
- How to measure Speed and Acceleration
- Generating a Trace
- Computing Swept Volume
- How to use Swept Volume tool
- Checking Clashes
- How to Check Clashes
- To Sum Up

Student Notes:

Analyzing Movements

In this lesson you will learn to analyze and trace the movements of parts in the mechanism during the simulation.

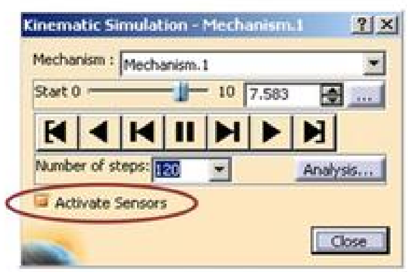
Student Notes:

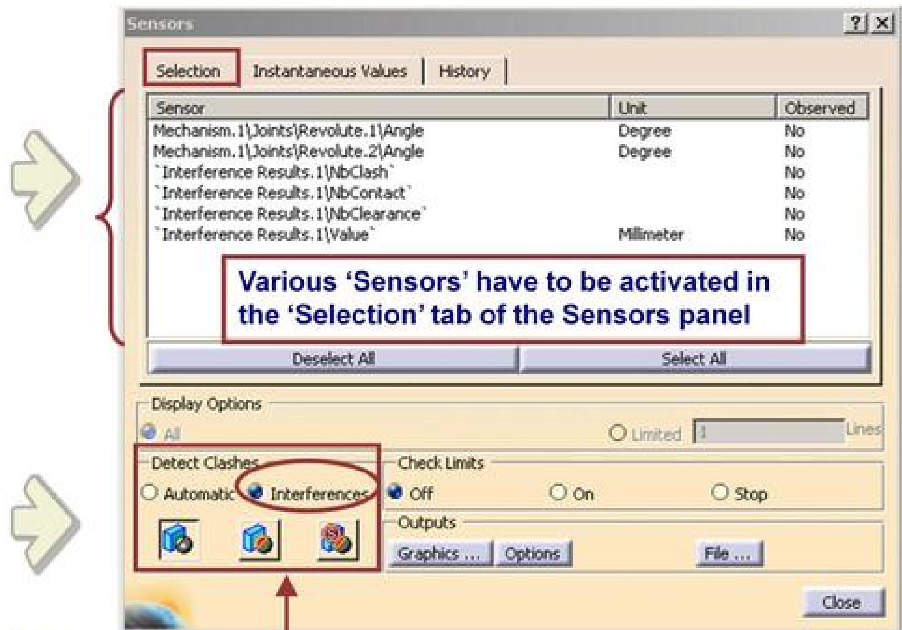
Simulation Outputs : Sensors

While simulating, you can apply Sensors and have the results of the 'Sensors' during the simulation.

While running a simulation you have the facility to apply 'Sensors' which can be :

- Speed and Acceleration
- Measure item: Point-coordinates
- Measure between: Distance, angle
- Joint parameter
- Interference results





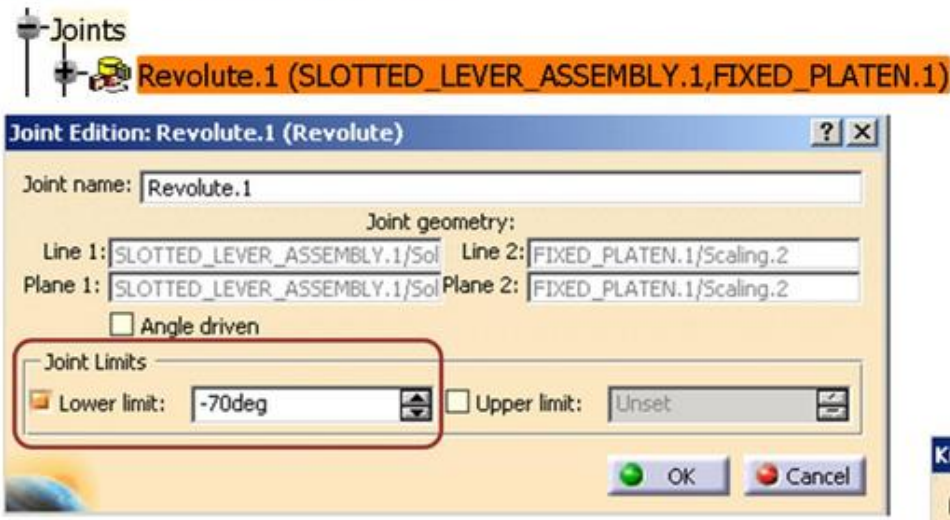
Besides, you can apply interferences detection tools (on V5 product only) which are available within the 'Sensors' panel

Student Notes:

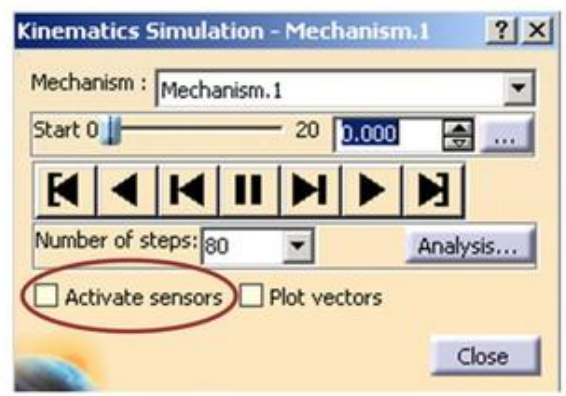
How to Check Joint Limits (1/3)

The 'Check Limits' function is used to determine the range of the joint. Using this tool, you can also find the point of time at which the lower or upper value of limit for a joint is reached.

1. Set the limits for the joint which is to be monitored (by checking the required check box) and enter the required value for the limits.



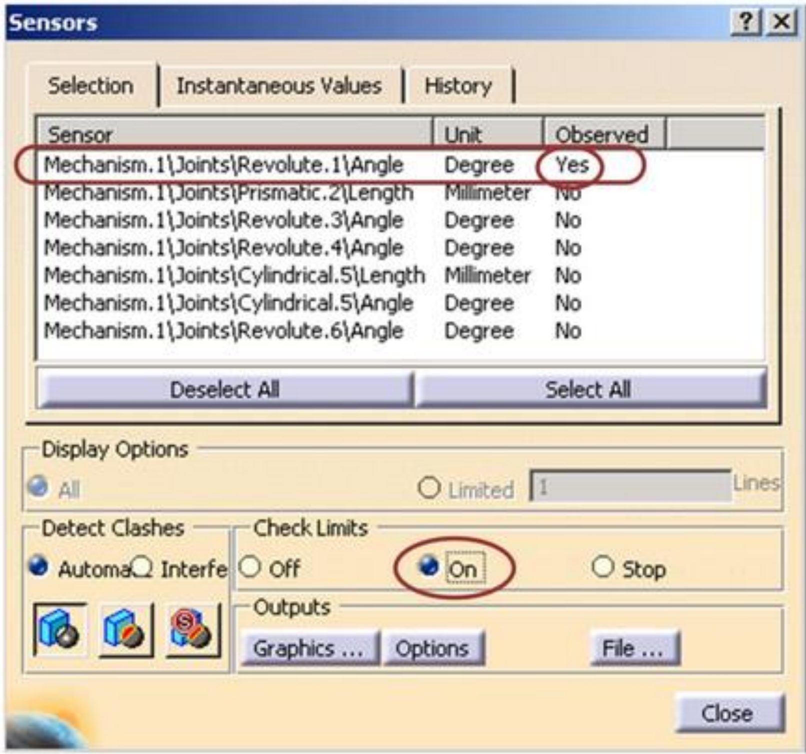
2. Click on the 'Simulation with Laws' tool, and in the 'Kinematics Simulation' panel, click on the 'Activate Sensors' check box to open the 'Sensors' panel.



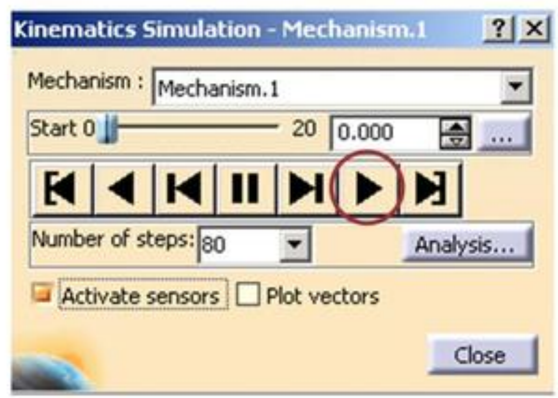
Student Notes:

How to Check Joint Limits (2/3)

3 **Select the parameters that have to be observed and switch 'On' the 'Check Limits' in the 'Sensors' panel.**



A Sensor is any parameter or measure which we want to observe during simulation of the mechanism. It can also be an 'Interference Result'.

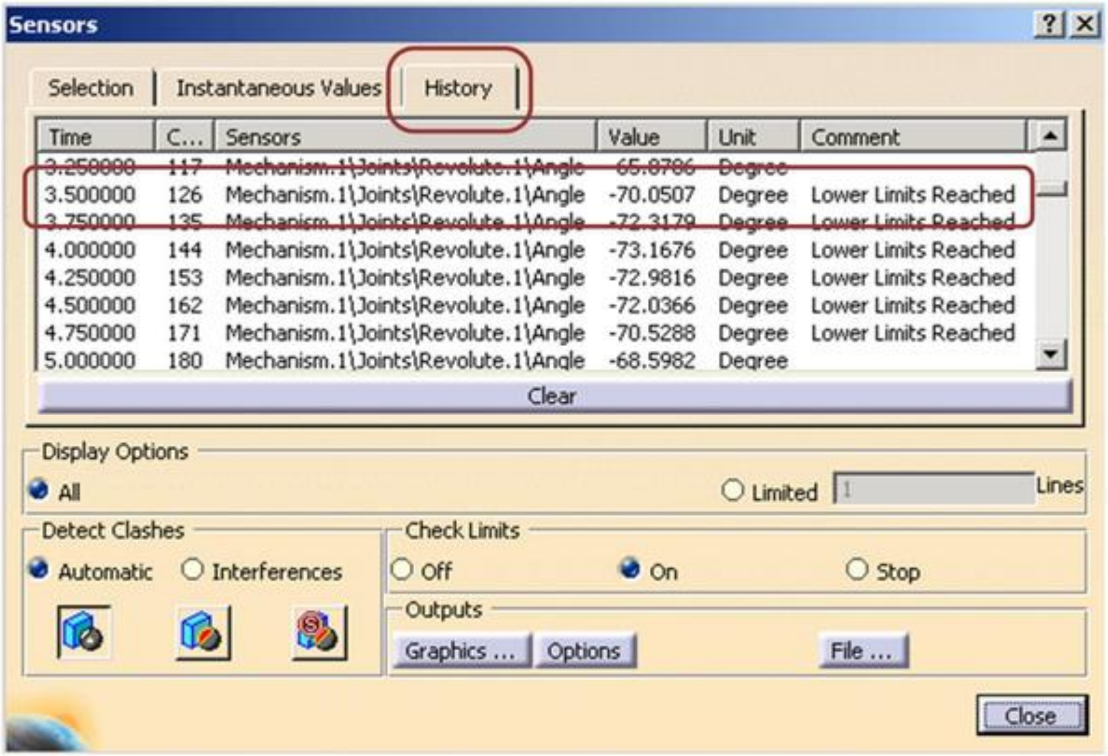


4 **Keeping the 'Sensors' panel open, go to the 'Kinematics Simulation' panel and click on 'Play Forward' to start the simulation.**

Student Notes:

How to Check Joint Limits (3/3)

5 Click on the 'History' tab to note the values of 'Sensors' at different mechanism times. You can also note the point of time when the limit is reached.

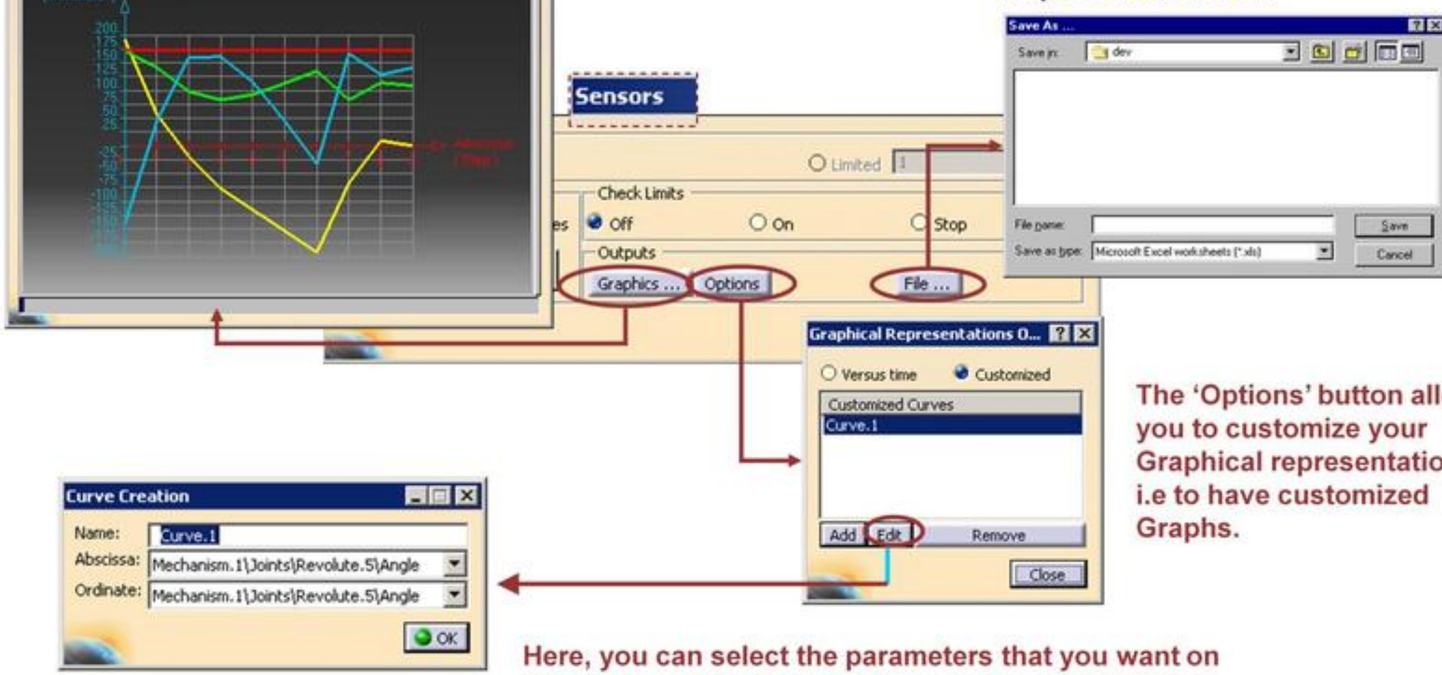


Student Notes:

Plotting Graphs and Exporting Results

In most of the cases, in addition to graphs with respect to time, some customized graphs of parameters are required. In Simulation Outputs we can have graphs of any parameter of our choice against any other parameter.

Export Sensors Data



The 'Options' button allows you to customize your Graphical representations, i.e to have customized Graphs.

Here, you can select the parameters that you want on Abscissa and the Ordinate for your customized graph.

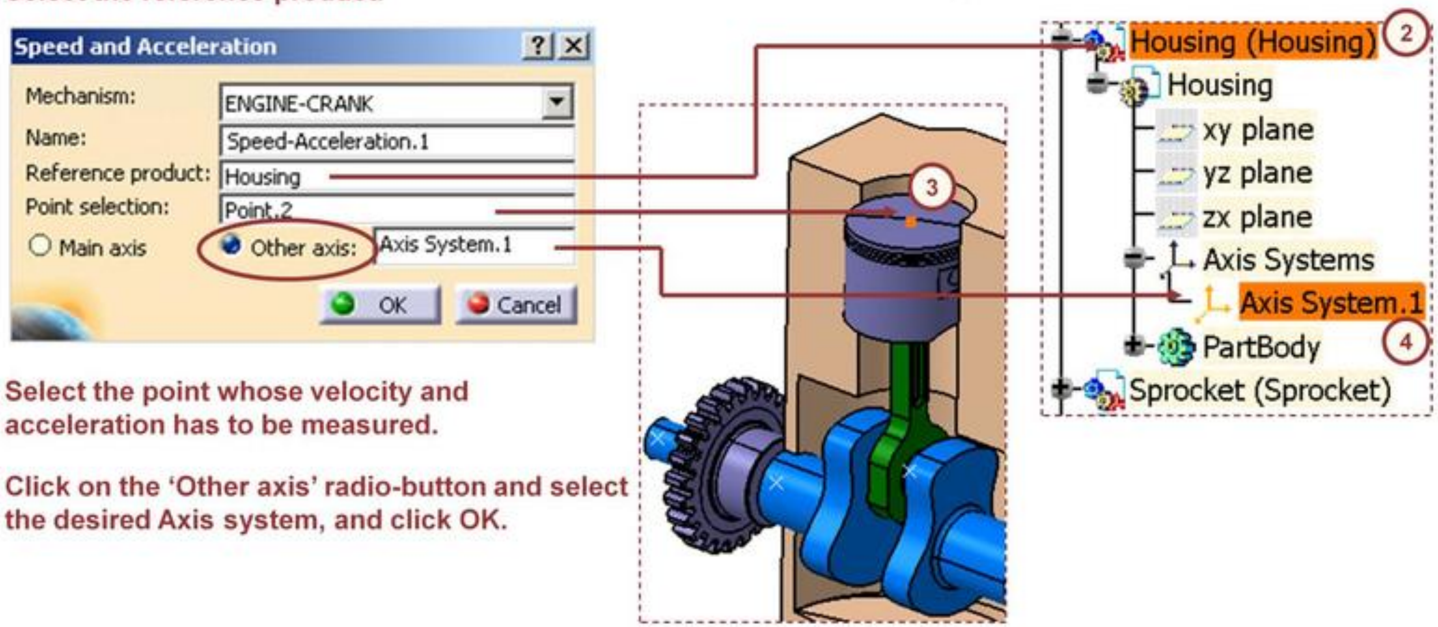
Student Notes:

How to measure Speed and Acceleration (1/3)

In DMU Kinematics, the speed and acceleration are measured with reference to a reference point. The speed and acceleration analysis can be performed only for mechanisms that can be simulated by laws.

1 Click on the ‘Speed and Acceleration’ tool.

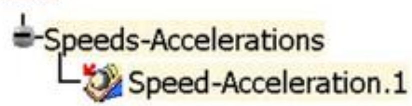
2 Select the reference product.



3 Select the point whose velocity and acceleration has to be measured.

4 Click on the ‘Other axis’ radio-button and select the desired Axis system, and click OK.

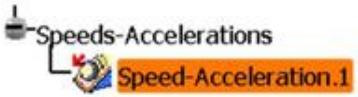
The result ‘Speed-Acceleration’ is listed in the Mechanism’s data.

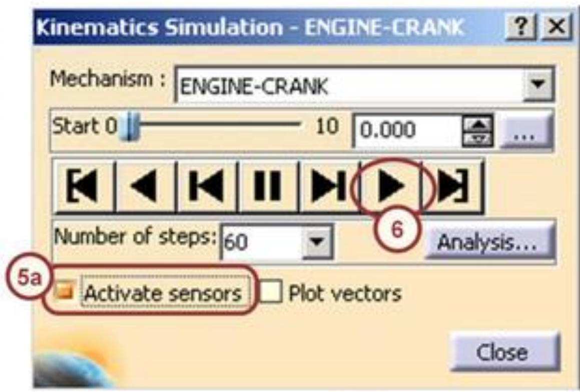


Student Notes:

How to measure Speed and Acceleration (2/3)

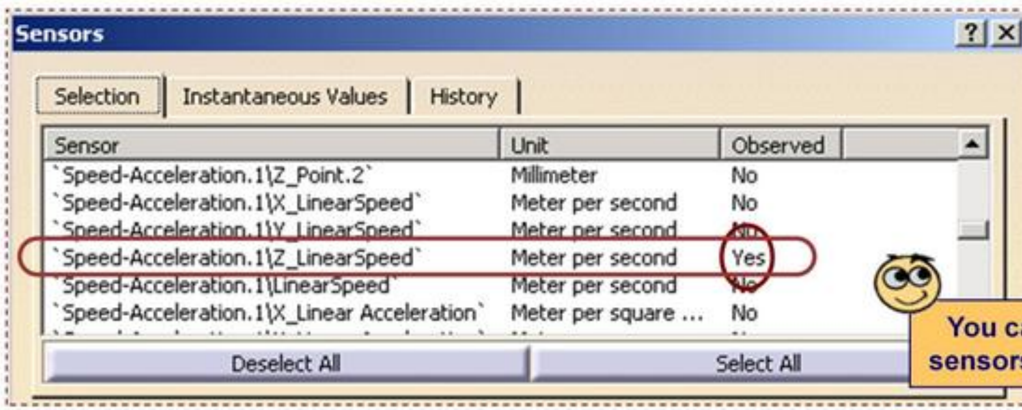
5 Double-click on the 'Speed-Acceleration' in the specification tree and click on 'Activate Sensors' check box to open the 'Sensors' panel.





5a Activate the desired 'Sensors' from the list by clicking them.

6 Keeping the 'Sensors' panel open, run the simulation with laws.

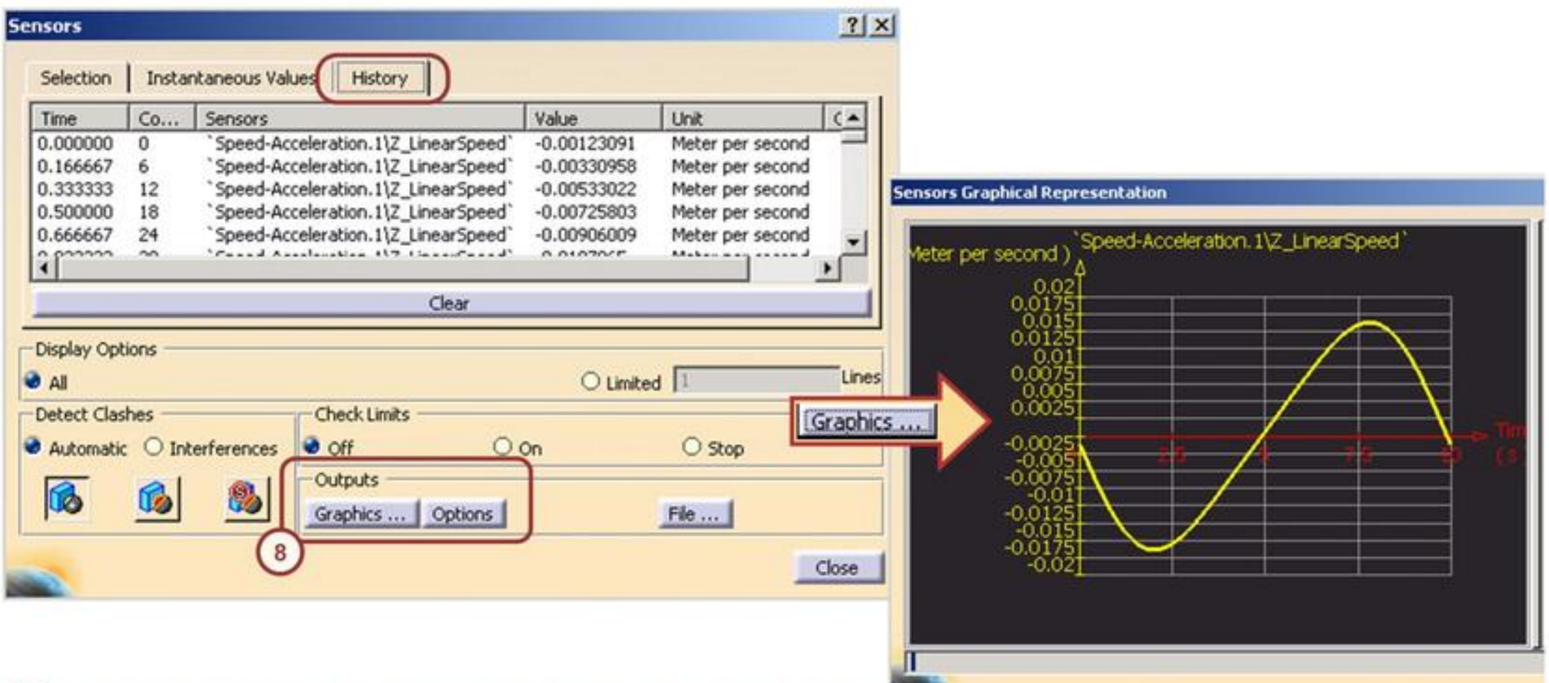


You can also have more than one sensors to be observed at a time.

How to measure Speed and Acceleration (3/3)

Student Notes:

7 The corresponding values for the 'Sensors' are listed in the 'History' tab of the 'Sensors' panel.



8 The graph of the required parameter against time can be obtained by clicking the 'Graphics' button in the 'Sensors' panel.

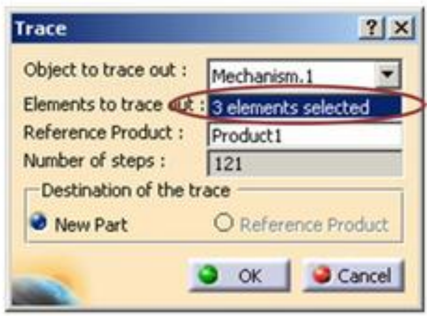
If you have activated more than one sensor at a time, you will have the results for all of them in the History tab and hence more than one curve in the graphics output of the Sensors' Graphical Representation

Student Notes:

Generating a Trace

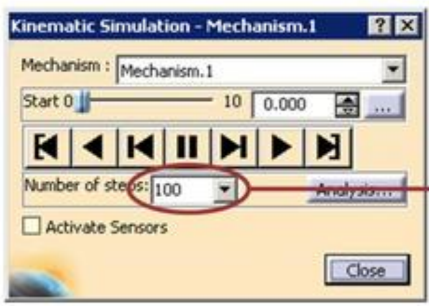
This is very useful command as you can use the resulting trace to design cams.

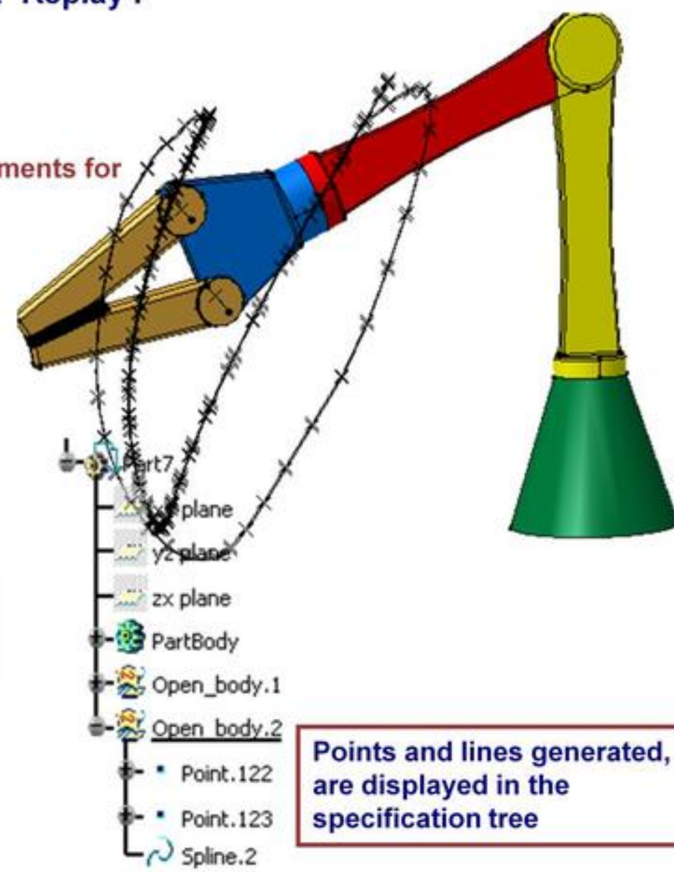
You can use this tool for a Mechanism (with laws) or for a 'Replay'.



You can select several elements for creating the trace.

The resulting points are in coordinate mode.





Points and lines generated, are displayed in the specification tree

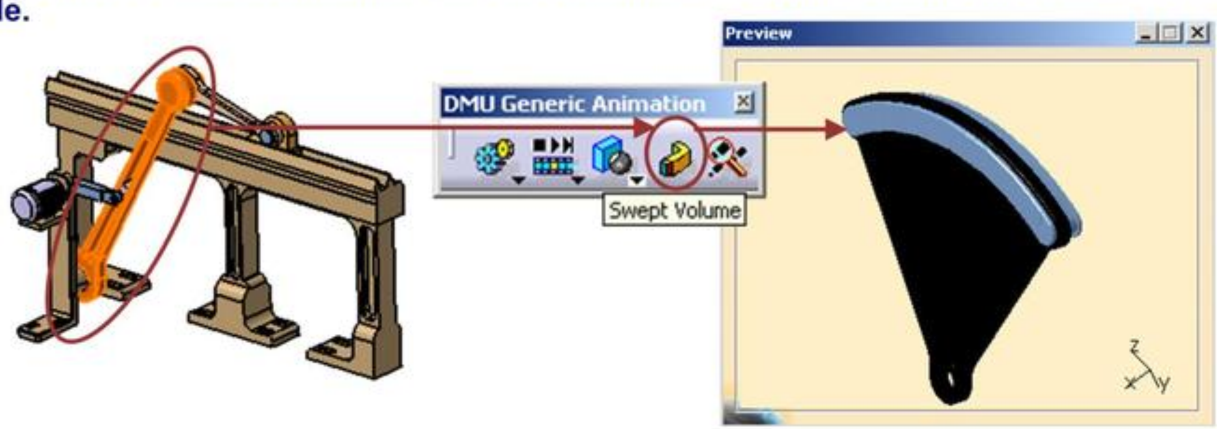
STUDENT GUIDE

Student Notes:

Computing Swept Volume

Swept volume is the volume that a part occupies in course of its motion.

Using the swept volume command you can compute the swept volume and save it as a CGR file.



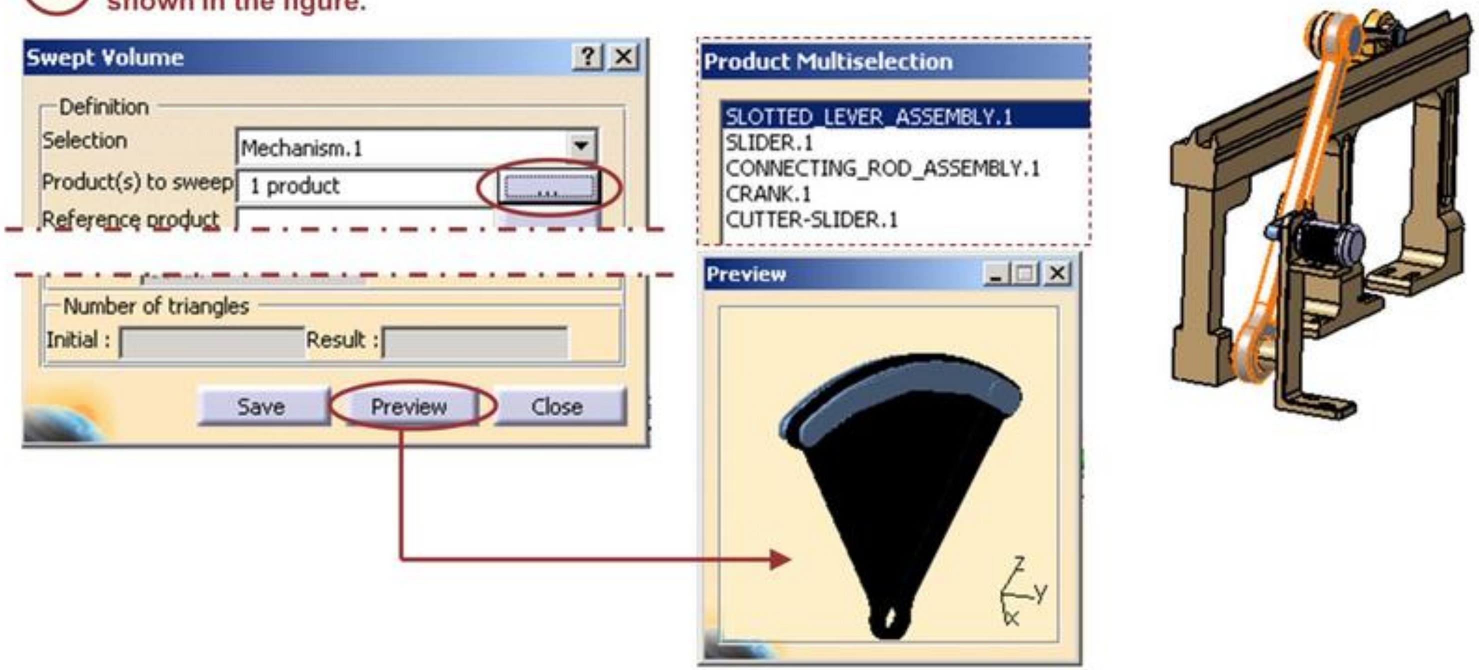
You can insert the swept volume in the same mechanism and perform distance and band analysis with respect to this computed CGR file.

How to use Swept Volume tool (1/3)

Student Notes:

1. Click the 'Swept Volume' tool.

2. In the 'Swept Volume' panel, click the button next to 'Products to sweep' and select the products as shown in the figure.

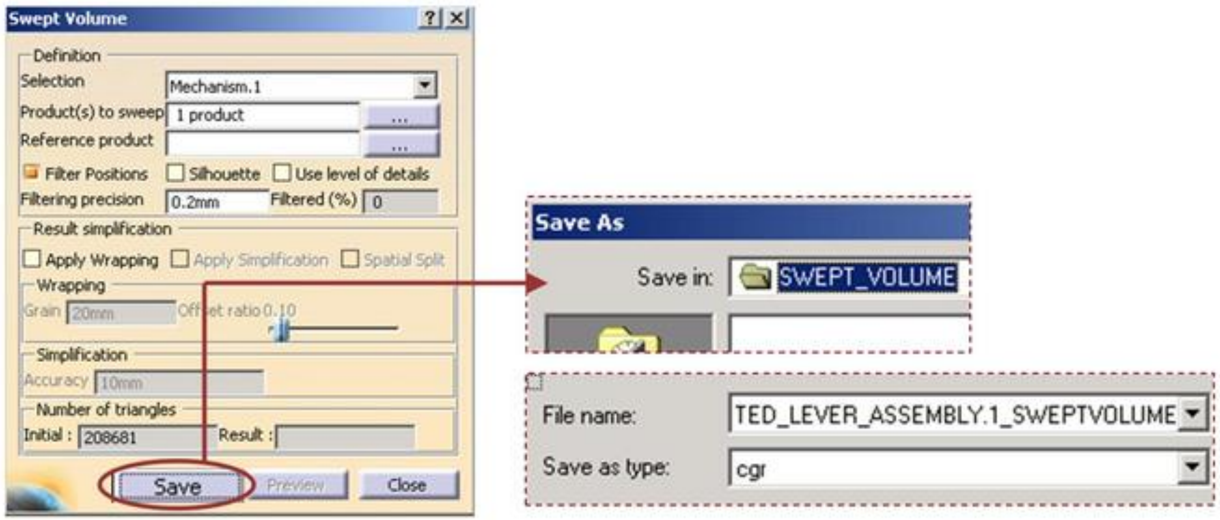


3. Click OK in the 'Product Multiselection' panel and click the 'Preview' button of the 'Swept Volume' panel to preview the swept volume.

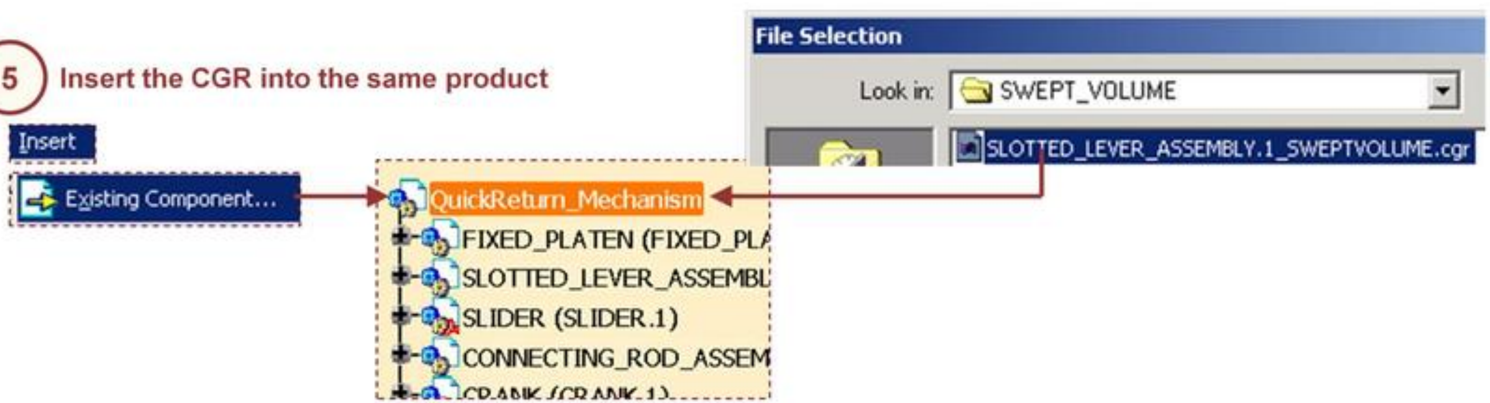
Student Notes:

How to use Swept Volume tool (2/3)

4 Click the 'Save' button in the 'Swept Volume' panel and save the CGR file.

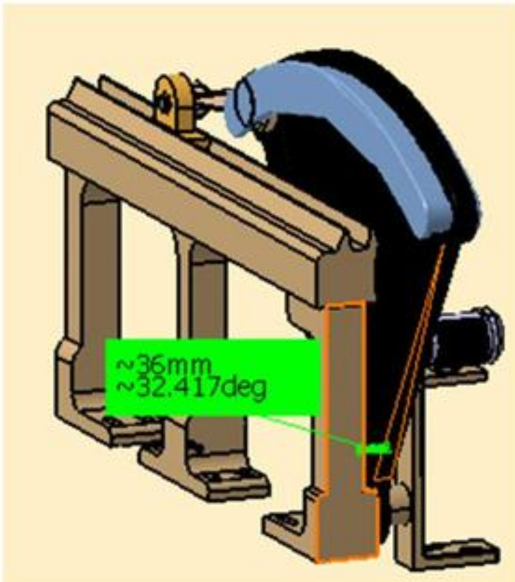


5 Insert the CGR into the same product



How to use Swept Volume tool (3/3)

6 Perform the ‘Measures’ with reference to this CGR file and other components.



Student Notes:

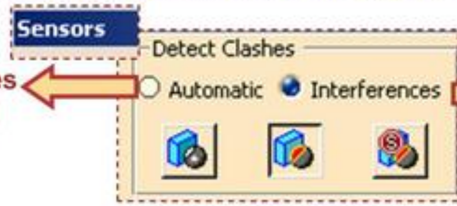
Student Notes:

Checking Clashes

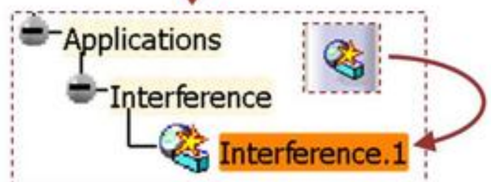
Just as in Assembly design, here you can detect clashes, or stop on intersections while simulating the mechanism.

Here we have two types of clash detection modes, the 'Automatic' and the 'Interferences'

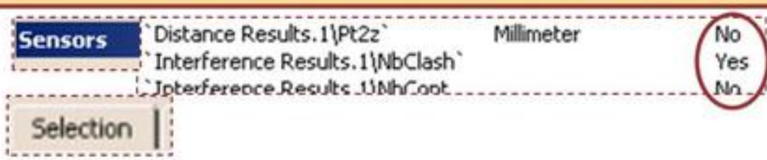
The automatic mode detects clashes between all components against all components of the product.



With the interference mode, the clash detection will run on Specific parts of our choice which we had selected under 'Clash detection', previously run under space analysis.



For the interference mode, you have to activate the sensor of the 'Interference Result' in the 'Selection' tab of the 'Sensors' panel to observe it throughout the mechanism simulation.



Checking Clashes

Student Notes:

Further we have three options in detecting Clashes

Clash detection OFF: No Clashes will be detected.

Clash detection ON: Clashes will be detected but mechanism will continue to run and not stop on clashes.

Clashes will be detected and mechanism will stop on clash.

Student Notes:

How to Check Clashes (1/4)

For checking clashes in DMU Kinematics, you have to specify the 'Interference Result' as an input to the Simulation and then run the Simulation.

You need to have 'Space Analysis' license to run this Job Aid.

1 Click on the 'Clash' tool.

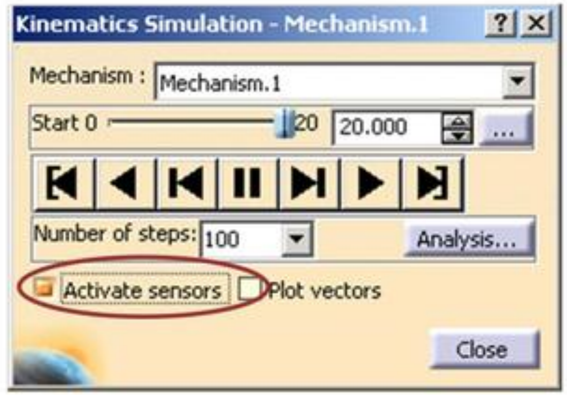
2 Select the following inputs for the 'Check Clash' panel and click 'Apply' and then OK.

An 'Interference Result' will be created in the Specification tree.

Student Notes:

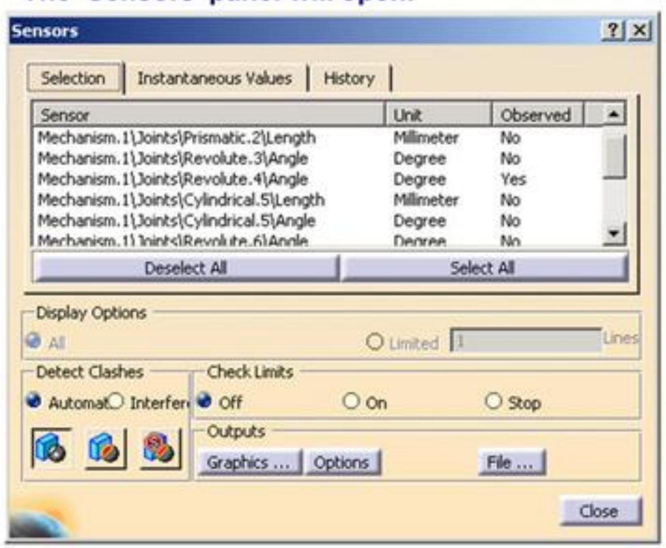
How to Check Clashes (2/4)

3 Double-click on 'Mechanism.1' in the specification tree.



4 In the 'Kinematics Simulation' panel, check the 'Activate sensors' option.

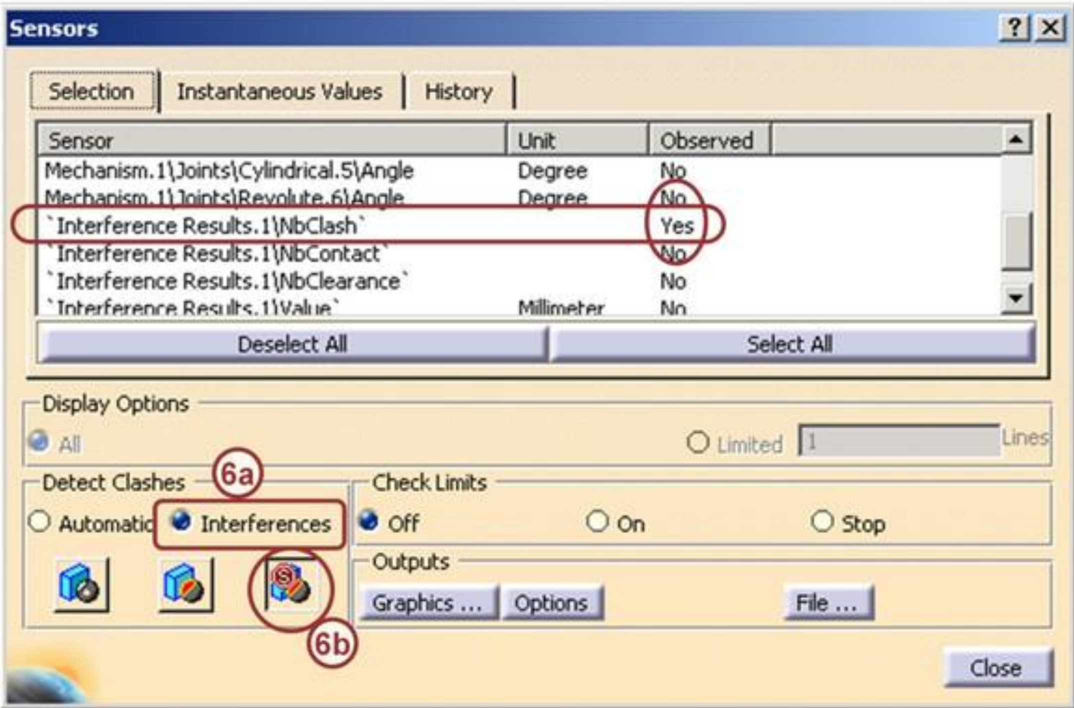
The 'Sensors' panel will open.



Student Notes:

How to Check Clashes (3/4)

5 In the Sensors panel, click the 'Interference Results.1\NbClash' from the list to activate it for observing.



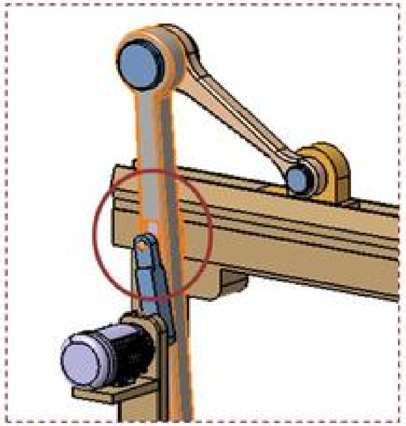
6 In the 'Detect Clashes' box, select 'Interferences' and 'Stop' option.

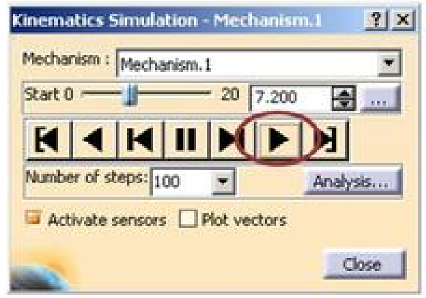
The 'Automatic' mode detects clashes between all components against all components of the product.

Student Notes:

How to Check Clashes (4/4)

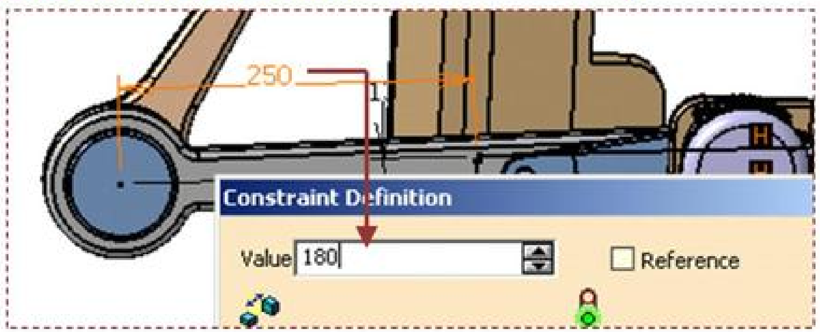
7 Keeping the 'Sensors' panel open, play the simulation.





The simulation will stop because a 'Clash' will be detected.

To avoid the above 'Clash', you can edit 'Sketch.4' of the 'SLOTTED-LEVER' and replay the above clash scenario to verify that there is no clash.



Student Notes:

To Sum Up

In this lesson, you have learned the tools for Mechanism analysis.

- **Checking Joint Limits.**
- **Applying Sensors**
- **Generating Graphical output of the Mechanism analysis**
- **Generating Swept Volume during Mechanism Simulation**
- **Generating Trace of a point in any part of the Mechanism during Simulation**
- **Checking Clashes during Simulation**

Recording and Playing Simulations

You will learn to record and play simulations involving more than one mechanism.

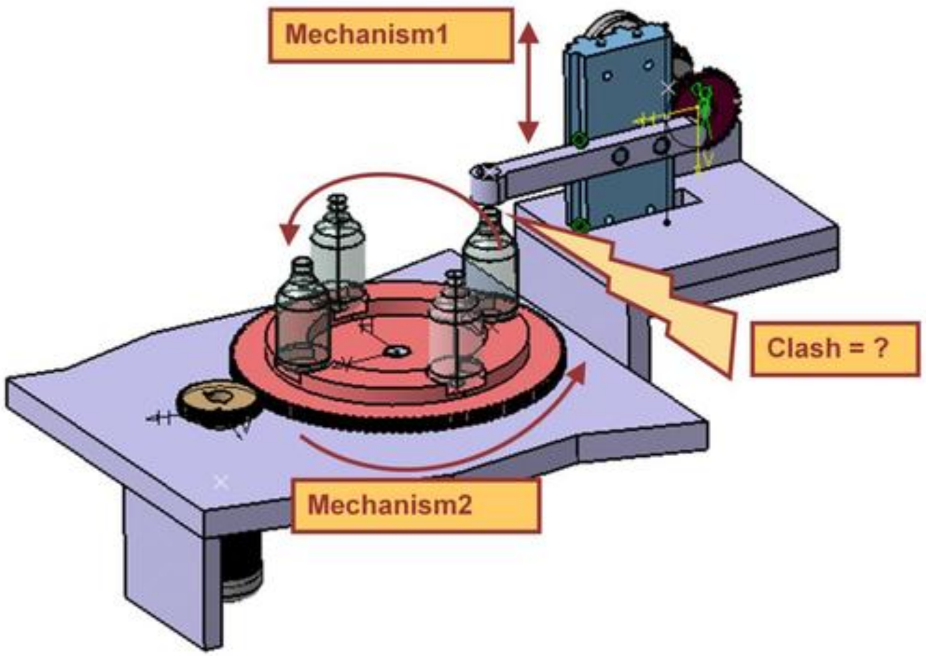
- Why do You Need to Record 'Simulations'
- How to record a Simulation
- How to Play a Simulation
- To Sum Up

Student Notes:

Student Notes:

Recording and Playing Simulations

You will learn to record and play Simulations involving more than one mechanism.

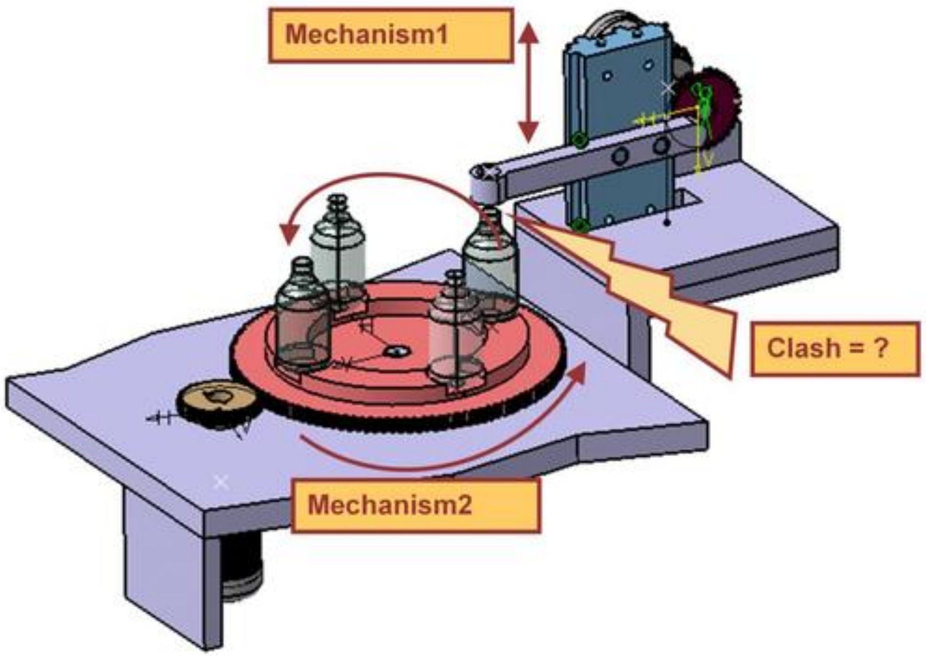


Student Notes:

Why do You Need to Record 'Simulations'

Many times it is necessary to compute the clash between components of two different mechanisms. So we need to run both the mechanisms simultaneously in order to test the behavior.

With the facility of 'Recording Simulations', we can make two mechanisms work concurrently and analyze the results, as we can do for an individual mechanism.

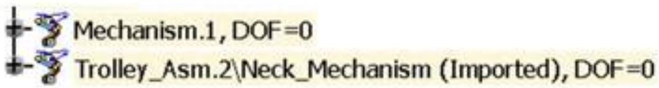


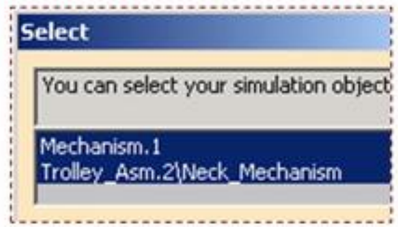
Student Notes:

How to record a Simulation (1/2)

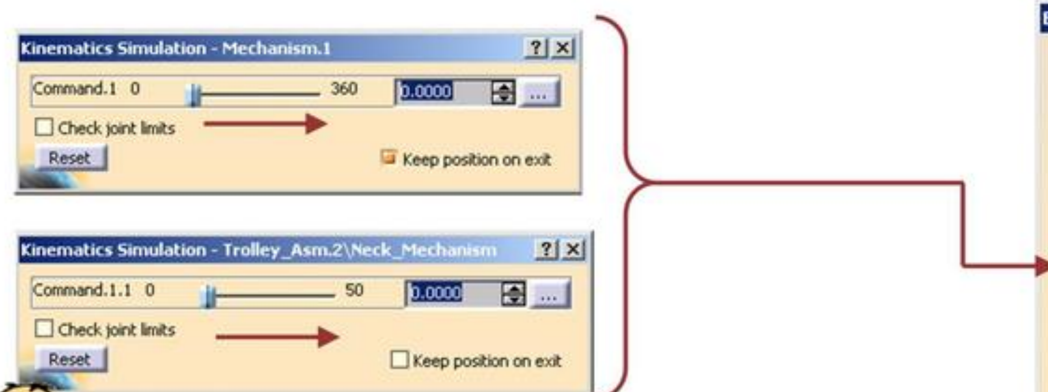
1. Click on the ‘Simulation’ tool.

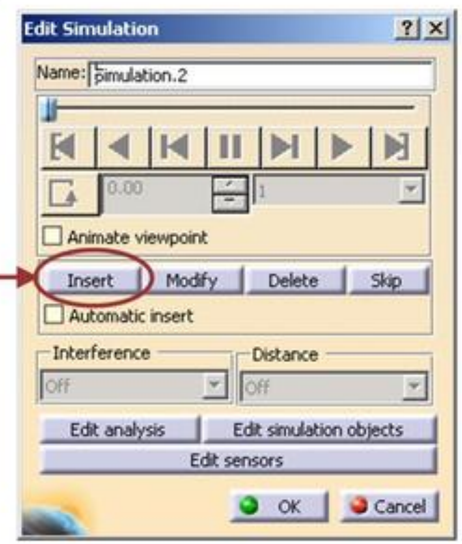
2. Using the ‘Ctrl’ key select the mechanisms (in the ‘Select’ panel) which you want to include for the simulation and click OK.





3. In the panels for the mechanisms, drag the sliders of the each command one after the other and click on the ‘Insert’ button of the ‘Edit Simulation’ panel.





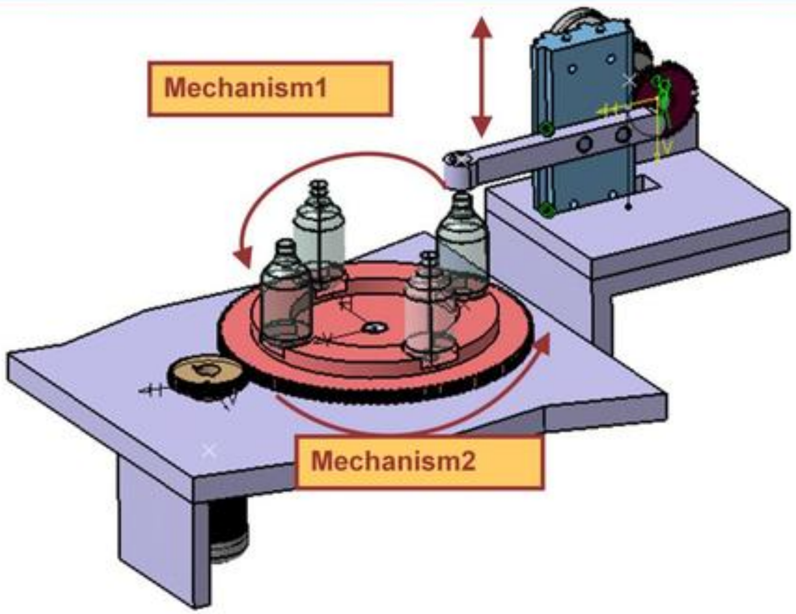
Simulation of both the mechanisms will take place simultaneously when you run the Simulation.

STUDENT GUIDE

Student Notes:

How to record a Simulation (2/2)

If you want the 'Mechanism2' to start after 'Mechanism1', you have to drag the slider for 'Mechanism1', click 'Insert' then drag the slider for 'Mechanism2' and again click the 'Insert' button.

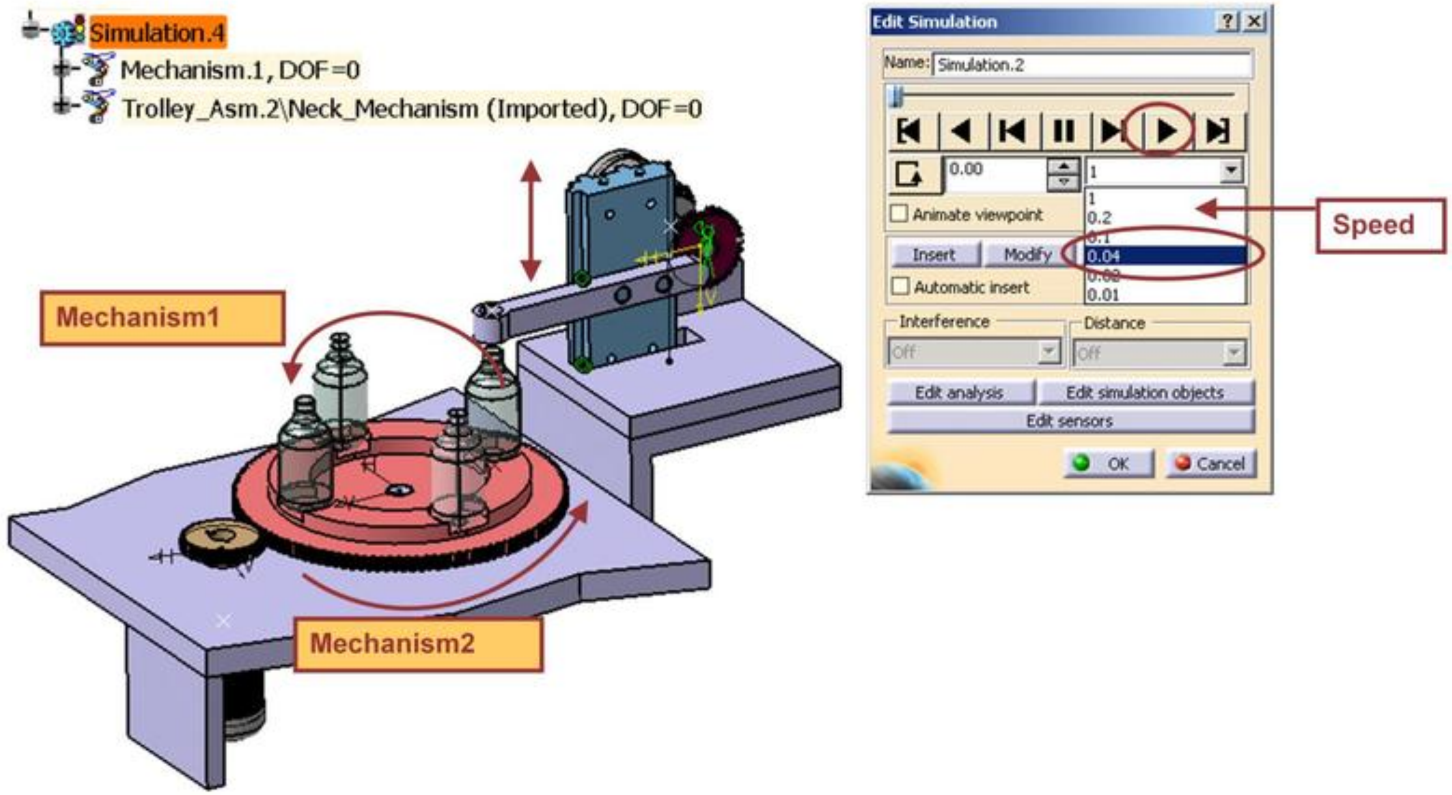


Student Notes:

How to Play a Simulation

1. Double-click on the ‘Simulation’ in the specification tree.

2. Select appropriate speed and click on the ‘Play’ button in the ‘Edit Simulation’ panel to run the simulation.



Student Notes:

To Sum Up

In this lesson of 'Recording and Playing Simulation', you have learned:

- The need to Record Simulation
- The methods of Recording and Running Simulations
- The additional tasks that you can perform while running the Simulations

Student Notes:

Miscellaneous Functionalities

You will learn some miscellaneous functionalities of the DMU Kinematics workbench.

- Converting Assembly Constraints into Joints
- How to Convert Assembly Constraints into Joints
- Importing Mechanisms in Sub-Products
- Mechanism Dress Up
- Importing a Mechanism Dress-up
- Converting V4 Kinematics
- To Sum Up

Student Notes:

Miscellaneous Functionalities

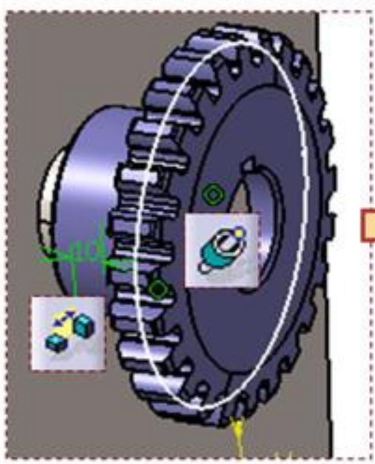
You will learn some miscellaneous functionalities of the DMU Kinematics workbench.

Student Notes:

Converting Assembly Constraints into Joints

You can convert the existing assembly constraints into joints using the ‘Assembly Constraints Conversion’ tool.

This tool compares the set of constraints between the components, with the constraints involved in a joint, and suggests a joint that could be created between the components.



This tool saves a lot of joint creation time and facilitates the reuse of the already created constraints.

Student Notes:

How to Convert Assembly Constraints into Joints (1/3)

1. Click on the tool.

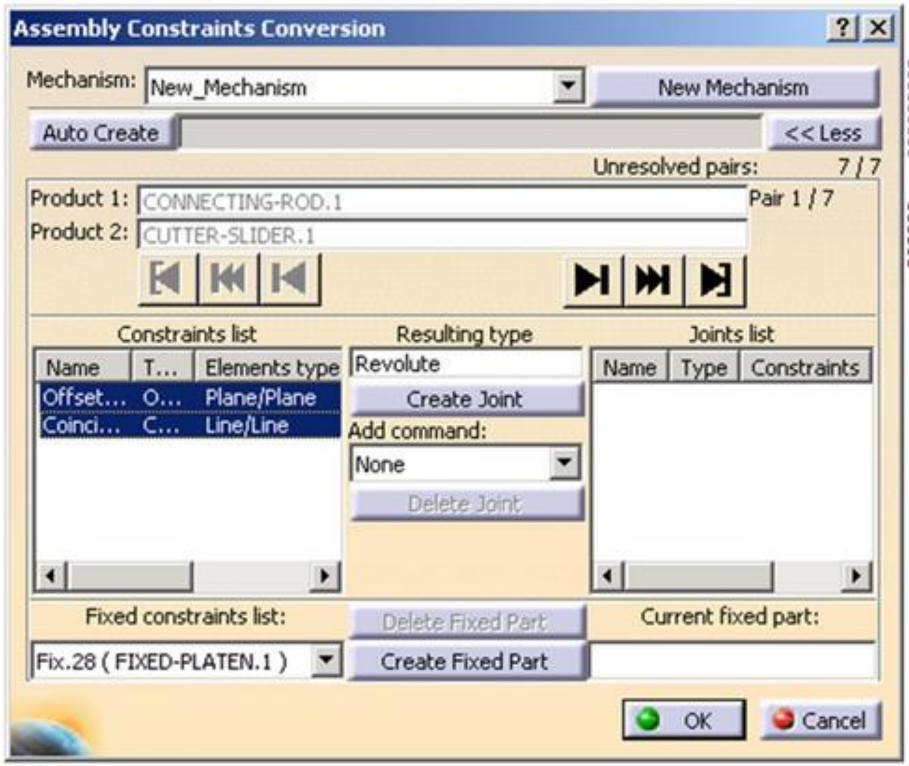
2. Click on 'New Mechanism' button to define a new mechanism or select an existing mechanism for which you want the joints to be created.

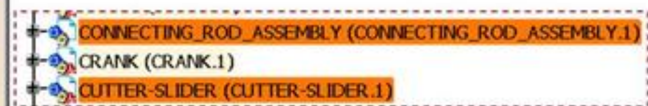
3. Click on the 'More>>' button to display the full 'Assembly Constraints Conversion' panel.

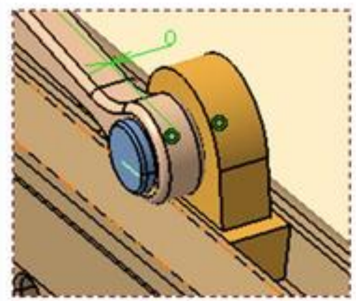
How to Convert Assembly Constraints into Joints (2/3)

Student Notes:

4 Select constraint/s from the 'Constraints list' and click on 'Create Joint' button after confirming the desired joint in the 'Resulting type' box.







The resulting type of joint depends upon the number and type of constraints you select in the 'Constraints list'.

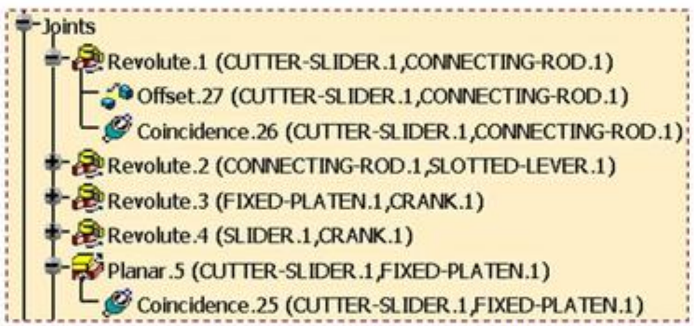
Student Notes:

How to Convert Assembly Constraints into Joints (3/3)

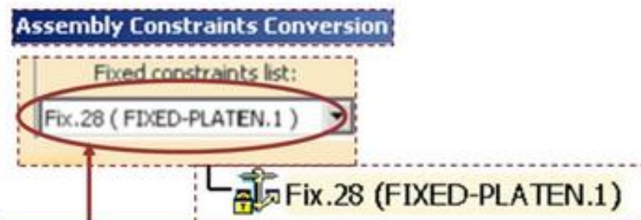
5 Click on the 'Step Forward' button to proceed and create joint for another set of constraint/s.

6 Repeat steps 4 and 5 if required.

7 Click OK to the 'Assembly Constraints Conversion' panel.

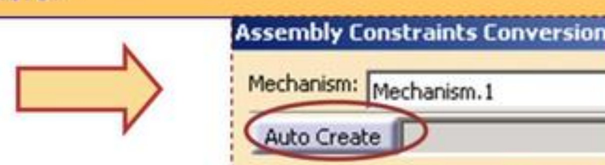


You can check the joints in the specification tree



You can also specify the 'Fix Part' for your mechanism by selecting it from the 'Fixed constraints list'. The components that are fixed in the assembly design, are listed here.

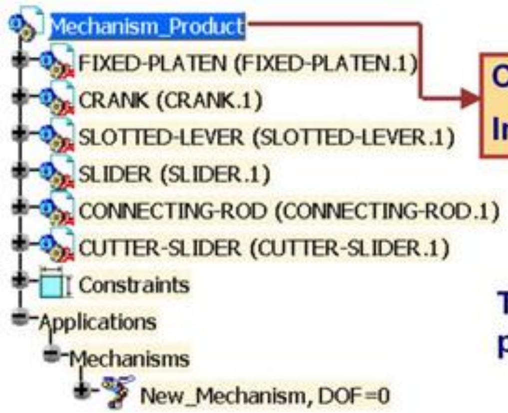
The 'Auto Create' tool automatically creates joints from the constraints. But the joints created using this function may not always be the joints that the user intents.



Student Notes:

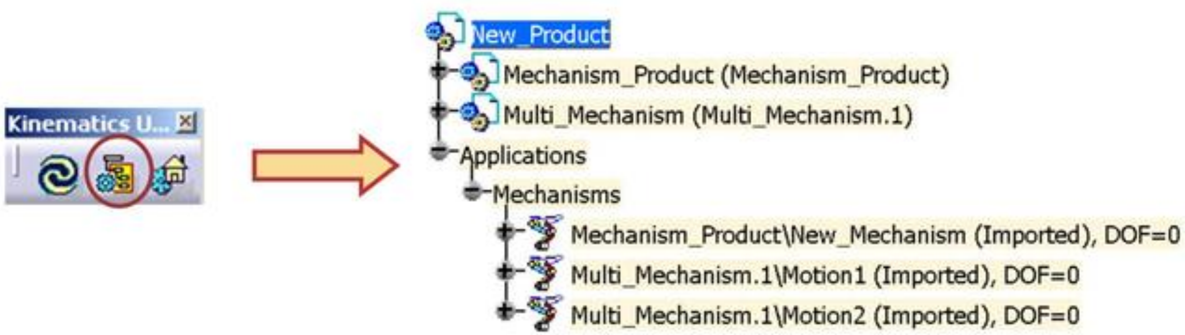
Importing Mechanisms in Sub-Products

When you copy-paste a product containing a mechanism into a new product, the mechanism and all its data does not get imported in the 'Applications' node of the new product.



This is because you may not want the mechanisms' data to be present in the root-product every time.

However you can always import the mechanism's data for a sub-product by clicking the 'Import Sub-Mechanisms' tool. The mechanism for the sub-products will be imported and the mechanisms' data can be seen in the 'Application' node.

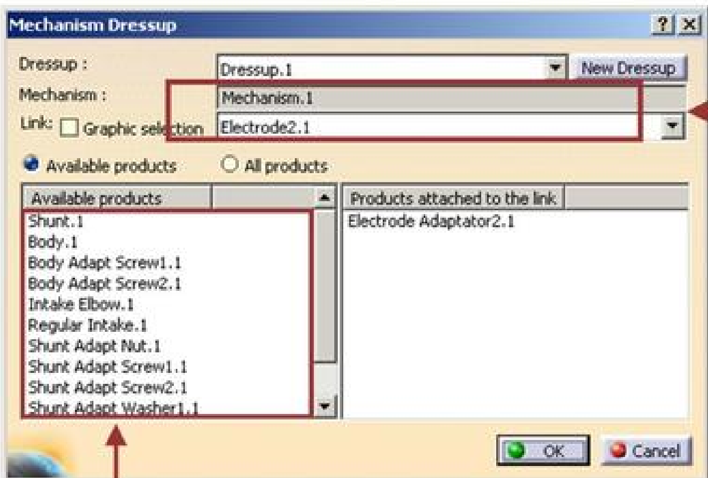


Student Notes:

Mechanism Dress Up

The "Dressup" tool allows you to associate a mechanism part movement to a flying part

The "Dressup" tool need a mechanism to be defined



Here are displayed the Mechanisms and their involved parts

Here are all the products associated to the part selected above

Here is the list of the available products which can be attached to the selected part of the mechanism

- In the perspective of Kinematics integration in ENOVIA VPM the dressup is directly accessible from the specification tree

- You can simulate it and it can be saved in ENOVIA VPM

- You can create a « Dressup » of a sub-product's mechanism

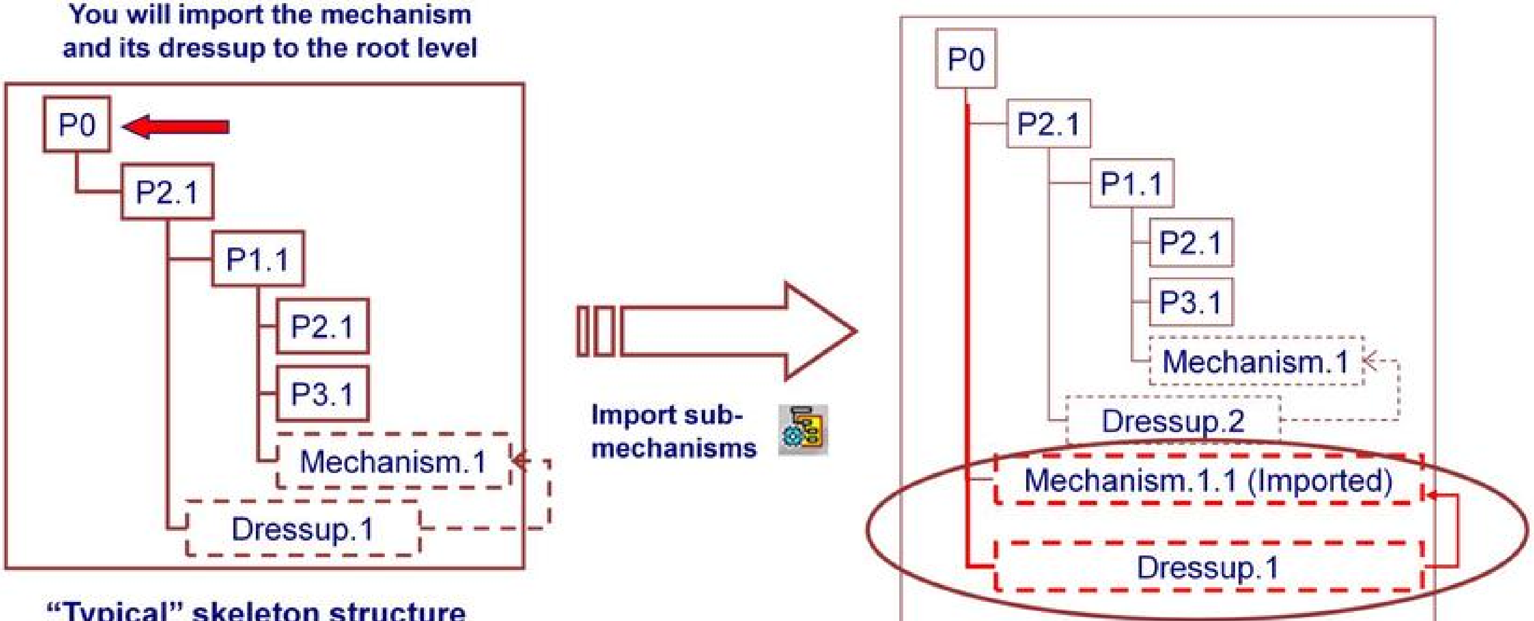
Importing a Mechanism Dress-up

When importing a Sub Mechanism, it is also possible to import the associated dressup

You may have a skeleton structure with a mechanism (with a dressup) applied on a child product.

You can decide to import this mechanism to the root level or even copy/past the product and its mechanism under a new product. The dressup will be imported too as below:

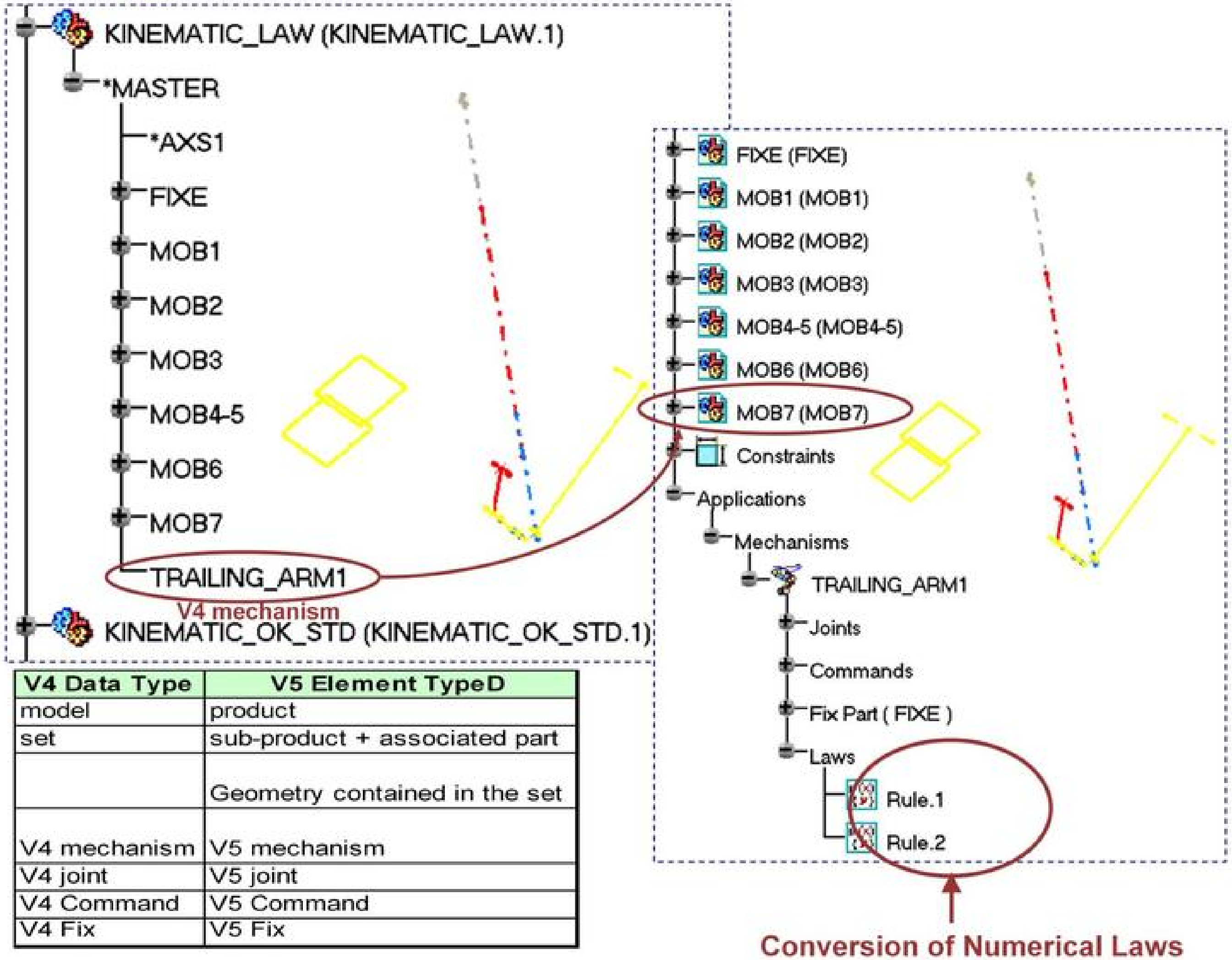
You will import the mechanism and its dressup to the root level



"Typical" skeleton structure

Student Notes:

Converting V4 Kinematics



V4 Data Type	V5 Element TypeD
model	product
set	sub-product + associated part
	Geometry contained in the set
V4 mechanism	V5 mechanism
V4 joint	V5 joint
V4 Command	V5 Command
V4 Fix	V5 Fix

Student Notes:

Student Notes:

To Sum Up

In this lesson you have learned Miscellaneous Functionalities for

- Converting assembly constraints into Joints
- Importing Mechanisms in Sub-Products

To Sum Up

In this Course, you have seen:

- **Defining a Mechanism**
- **Simulating Mechanisms**
- **Analyzing Movements**
- **Recording and Playing Simulations**

Student Notes: