

# DMU Fitting Simulator



## **Preface**

- Using This Guide
- More Information
- Conventions

## **What's New?**

## **Getting Started**

- Using Tracks
  - Starting a Session
  - Recording a Track
  - Using Automatic Path Finder
  - Using the Smooth Command
  - Updating a Track
  - Replaying a Track
  - Defining a Swept Volume
- Using Shuttles
  - Starting a Session
  - Exploding a Product
  - Defining a Shuttle
  - Recording a Simulation
  - Replaying the Simulation

## **Basic Tasks**

- Setting Up Your DMU Fitting Simulator Session
  - Entering The Workbench
  - Defining a Shuttle
- Shuttle
  - About Shuttles
  - More About ManipulationToolbar
  - Editing a Shuttle
  - Snapping Objects
- Mono-Shuttle Fitting Simulation
  - Recording a Simulation
  - Recording a Simulation Automatically
  - Using the Smooth Command
  - Animating a Viewpoint
  - Generate a Replay
  - Replaying
- Target and Snap Capabilities
  - Using Target Capability
  - Using Smart Target

- Using Smart Snap
- Moving the Shuttle

## Tracks

- About Track Capabilities
- Track Editor and Recorder
- Player
- About Track Operators
- Analyzing in Track Context
- Exporting/Importing Tracks in Neutral Format
- Editing Time Line in Tracks
- Copying and Pasting Tracks

## Sequences

- About Sequence Capabilities
- Sequence Editor
- Displaying Sequence Gantt Chart
- Defining a Sequence

## Generating Animations

- Generating a Video Using Standard Tools

## Exploding

- Exploding a Product
- Exploding a Constrained Assembly
- Exploding a Shuttle
- Generating a Sequence from an Exploded View

## Advanced Tasks

### Using Multiple Shuttles

- Defining Several Shuttles
- Defining a Shuttle Made of Shuttles
- Recording a Multi-Shuttle Simulation
- Defining a Shuttle Motion using another Shuttle as Reference
- Defining a Shuttle Motion Using a Product as Reference
- Adding a Shuttle in a Simulation

### Using Track and Sequence Capabilities

- Dismounting the Chainsaw Handle
- Illustrating the Dismounting Operation
- Finalizing the Dismounting Operation
- Generating a Film

### Converting a Simulation into a Sequence

### Path Finder

- Using Path Finder in Interactive Mode
- Writing a Path Finder in Macro
- Publishing Path Finder Results
- More About Path Finder
- About Path Finder and Smooth Operations

### Validating a Motion

- Validating Positions Automatically
- Measuring Distances
- Detecting Clashes
- Performing an Experiment and Generating a Report

### Swept Volume

- Defining a Swept Volume
- Filtering Swept Volume Positions
- Defining a Swept Volume from any Moving Reference
- More About Swept Volume

## **Interoperability**

- Interoperability With Catia V4
  - Writing Axis Systems From a V5 Track to a V4 Model
  - Reading a V4 Route and Sequence in V5
- How to do a Safe Save in ENOVIA LCA from CATIA V5

## **Workbench Description**

- Menu Bar
- Simulation Toolbar
- DMU Check Toolbar
- Manipulation Toolbar
- Automatic Clash Detection Toolbar
- Specification Tree
- Recorder
- Player

## **Customizing for DMU Fitting Simulator**

- DMU Fitting Settings
- DMU Manipulation Settings
- Customizing DMU Navigator Settings

## **Glossary**

## **Index**

# Preface

DMU Fitting Simulator is a software dedicated to simulating part motions for assembly and maintainability issues. 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 Fitting Simulator is a dedicated DMU Navigator Version 5 workbench and is available on both UNIX and Windows NT environments.

DMU Navigator Version 5 comprises the following main applications:

- Kinematics Simulator
- Fitting Simulator
- Space Analysis
- DMU Optimizer

The above applications are delivered as totally interoperable workbenches. From a user interface standpoint, switching from one to another is completely transparent and done in a context-sensitive fashion. In addition, to these workbenches, DMU Navigator is an open solution which offers:

- Support of native CATIA Version 4 and Version 5 data
- Interface with the VRML industry standard for data exchange
- Native OLE (Object Linking and Embedding) compliance. This facilitates the system integration within the office environment and across the digital enterprise.

## DMU Kinematics Simulator

- Offers motion simulation capabilities. Kinematics Simulator can be cooperatively used with other current or future companion products of the DMU Navigator next generation such as DMU Fitting Simulator and DMU Space Analysis.

## DMU Fitting Simulator

- Allows the user to define and simulate assembly and disassembly procedures thereby validating product assembly and maintenance at the design stage. Fitting Simulator can be cooperatively used with other current or future companion products of the DMU Navigator next generation such as Kinematics Simulator and Space Analysis.

## DMU Space Analysis

- Offers advanced interference analysis, sectioning and measurement capabilities. Space Analysis can be cooperatively used with other current or future companion products of the DMU Navigator

next generation such as DMU Kinematics Simulator and Fitting Simulator.

## **DMU Optimizer**

- Improves user's productivity by computing an optimized representation of data for mockup verification in the context of the immersive and collaborative design review environment of the full digital mockup.

DMU Optimizer is a dedicated DMU Navigator workbench and is available on both UNIX and Windows NT environments.

[Using This Guide](#)  
[More Information](#)  
[Conventions](#)

# Using This Guide

This guide is intended for the user who needs to quickly become familiar with DMU Fitting Simulator. The user should be familiar with DMU Navigator Version 5 concepts such as document windows, standard and view toolbars.

To get the most out of DMU Fitting Simulator, use the following user guide wizard. It will help you better locate information relevant to you as well as to the way you work.

## User Guide Wizard

### Go to:

#### **I am a first time user**

The [Getting Started](#) tutorial. Once you have finished, you should move on to the user task section of this guide. This steps you through basic procedures.

#### **I have used DMU Fitting Simulator before**

Your DMU Fitting Simulator Version 5 session and start reviewing your own documents. If you need some help in understanding tools and commands, use the on-line help. You can also take a look at the [Basic Task Section](#) and [Advanced Task Section](#) of this guide to locate information with which you are not already familiar.

#### **Inserting Sample Documents**

You will use the samples contained in [C:/Program Files/Dassault Systemes/B010doc/online/fitug/samples](#) folder or in [C:/Program Files/Dassault Systemes/B010doc/online/cfyug/samples](#)

Sample documents (installed along with the online help library) are provided in many (but not all) cases, to support the topic scenario explaining how a specific command works.

The sample documents are installed in user guide-specific sample folders. In the online documentation filetree, there is one samples folder for each users guide. For more information on where sample documents are installed by default, see [Accessing Sample Documents](#) in the *Infrastructure User's Guide*.

# Where to Find More Information

Prior to reading this book, we recommend that you read the *DMU Infrastructure User's Guide*.

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

- *DMU Navigator User's Guide*
- *DMU Kinematics Simulator User's Guide*
- *DMU Space Analysis User's Guide*
- *DMU Optimizer User's Guide*
- [\*Conventions\*](#)

# Conventions

Certain conventions are used in CATIA, ENOVIA & DELMIA documentation to help you recognize and understand important concepts and specifications. The following text conventions may be used:

- ◆ The titles of CATIA documents *appear in this manner* throughout the text.
- ◆ **File** -> **New** identifies the commands to be used.

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

*Use this mouse button, whenever you read*



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



Drag  
Move



Right-click (to select contextual menu)

Graphic conventions are denoted as follows:



indicates the estimated time to accomplish a task.



indicates a target of a task.



indicates the prerequisites.



indicates the scenario of a task.



indicates tips



indicates a warning.



indicates information.



indicates basic concepts.





indicates methodological information.



indicates reference information.



indicates information regarding settings, customization, etc.



indicates the end of a task.



indicates functionalities that are new or enhanced with this Release. Enhancements can also be identified by a blue-colored background in the left-hand margin or on the text itself.



indicates functionalities that are P1-specific.



indicates functionalities that are P2-specific.



indicates functionalities that are P3-specific.



allows you to switch back the full-window viewing mode.

These icons in the table of contents correspond to the entries or mode.



"Site Map".



"Split View" mode.



"What's New".



"Preface".



"Getting Started".



"Basic Tasks".



"User Tasks" or the "Advanced Tasks".



"Workbench Description".



"Customizing".



"Reference".



"Methodology".



"Glossary".



"Index".

# What's New?

## New Functionalities

### Reuse a shot

While creating or editing a track, users may now select a portion (shot) from an existing track and use that shot within the new or modified track.

### How to do a Safe Save into ENOVIA LCA from CATIA V5

The objective of Safe Save is to prevent the user from building / editing data in CATIA V5 that cannot be saved in ENOVIA LCA. Therefore, in interoperability mode, some CATIA V5 commands are grayed out / hidden in the DMU Fitting workbench.

## Customizing Settings

### Fast Clash Detection settings have moved

They can now be accessed at: **Tools->Options->Digital Mockup->DMU Navigator** tab.  
The previous settings were on the [Digital Manipulation](#) tab.

# Getting Started



Before getting into the detailed instructions for using DMU Fitting Simulator, the following tutorial aims at giving you a feel of what you can do with the product. It provides a step-by-step scenario showing you how to use key functions.

The main tasks described in this section are:

[Using Tracks](#)

[Using Shuttles](#)



These tasks should take about 15 minutes to complete.

# Using Tracks



Before getting into the detailed instructions for using DMU Fitting Simulator, the following tutorial aims at giving you a feel of what you can do with the product. It provides a step-by-step scenario showing you how to use key functions.

The main tasks described in this section are:

Starting a Session

Recording a Track

Using Automatic Path Finder

Using the Smooth Command

Updating a Track

Replaying a Track

Defining a Swept Volume



These tasks should take about 15 minutes to complete.

# Starting a Fitting Simulator Session



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 Fitting Simulator workbench and select your products.

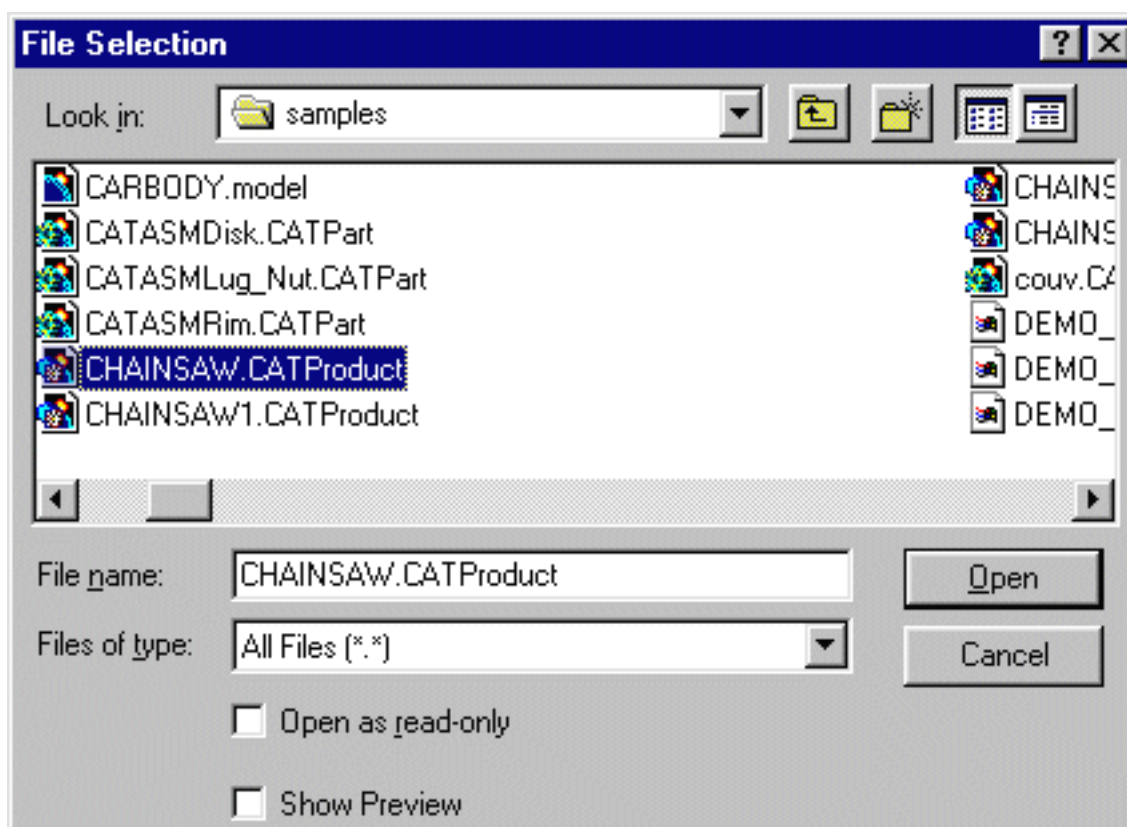


1. Select **Digital Mockup** -> **DMU Fitting** from the **Start** menu.

The Fitting Simulator workbench is displayed.

2. Select **File**->**Open**.

The File Selection dialog box displays:



3. Select the **CHAINSAW.CATProduct** from the samples folder.

The samples are contained in **C:/Program Files/Dassault Systemes/B08doc/online/fitug/samples** folder.

4. Click **Open** to open the selected files.

The specification tree is displayed showing all the selected products. In our example, the CHAINSAW.CATProduct has been divided in four main groups of objects.

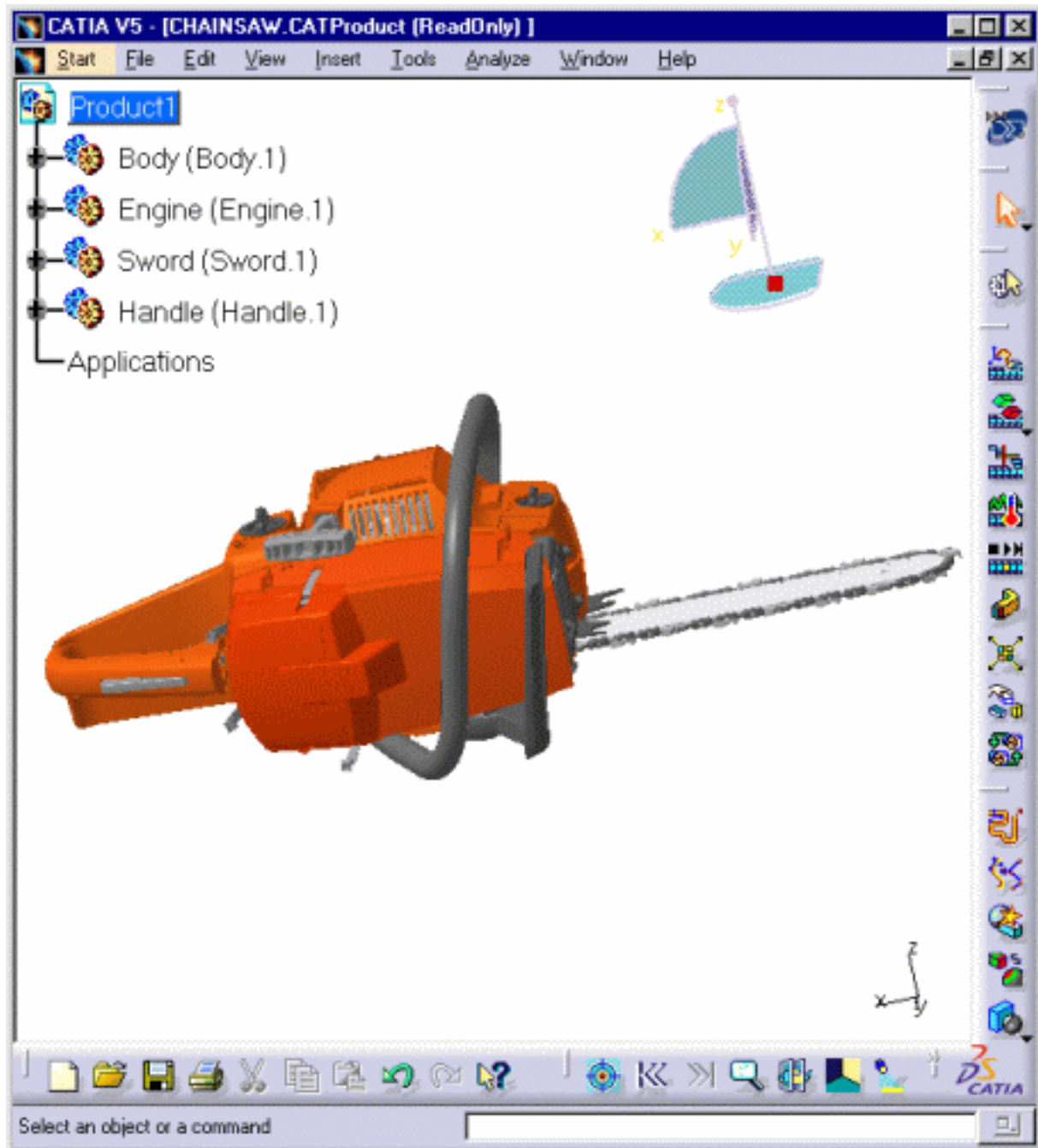


Note that you can also select model files using **insert**-> **existing component**.

In this case, select the desired model files by clicking the first one then shift-clicking the last one you want.



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



Clicking off **View -> Specifications** visible in the menu bar removes the specification tree and lets you use the entire screen for the product. You can also use the F3 key to toggle more quickly.



# Recording a Track



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*.



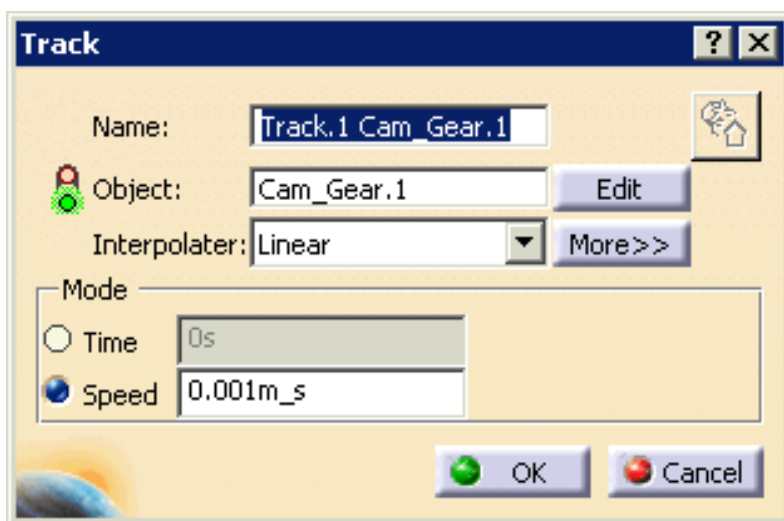
This task will show you how to record a track using products

The [CHAINSAW.CATProduct](#) document is already open



1. Click the Track icon  from the DMU Simulation toolbar

The Track dialog box along with the DMU recorder and player are displayed:



You can alter the name of the track by entering one in the Name field, or you can accept the default name provided. You can also set a speed for the movement or input a time by which the movement should be complete.



2.

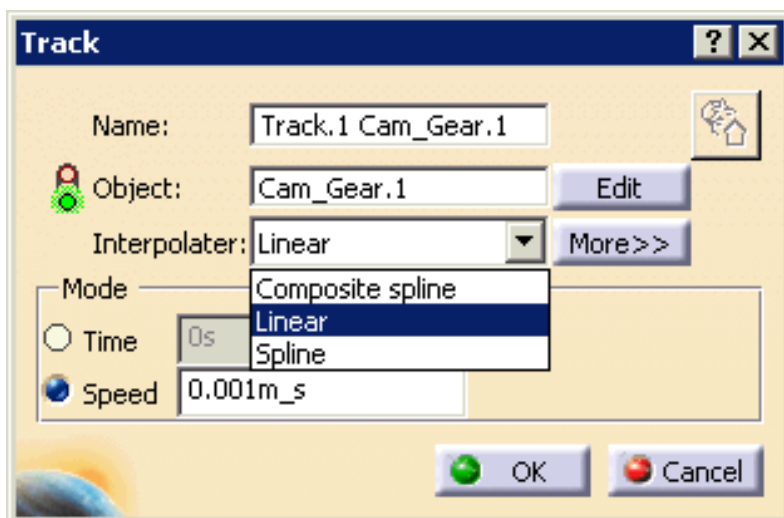
You are ready to record your track: select the product to move  
In our example, select DEMO\_CGE\_CHAINSAW\_BODY\_FILTERCOVER.1 either in the specification or in the geometry.

The player becomes active and the Manipulation toolbar appears








3. Select the required interpolator. In our example keep the default one (Linear)  
**Note:** you can change the interpolator type at any time during your track recording



4. Move the 3D compass and click Record at each time position of your choice.  
 If you double-click Record, the automatic insert mode is activated.

**Note:** You can also use the Insert Key instead of clicking Record.

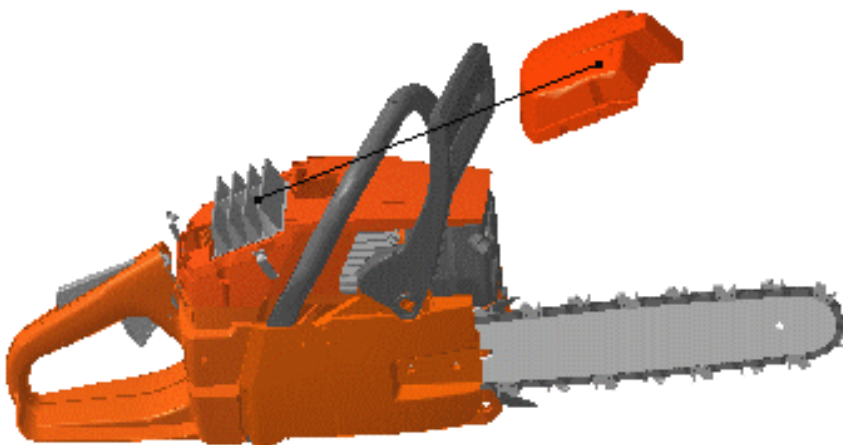
You can modify or delete a position (referred to a shot) at any time, for this select the required shot in the geometry (use the Step Backward shot  or Step Forward  buttons in the Player to jump to a shot, make sure the Player unit is shot and the sampling step in the Player Parameters dialog box is set to 1), then perform the deletion (use the Delete key or the Delete icon ) or modification. The track is automatically updated.

Alternatively, you can use the Reorder Shots command,  which calls up the Reorder Shots dialog box.

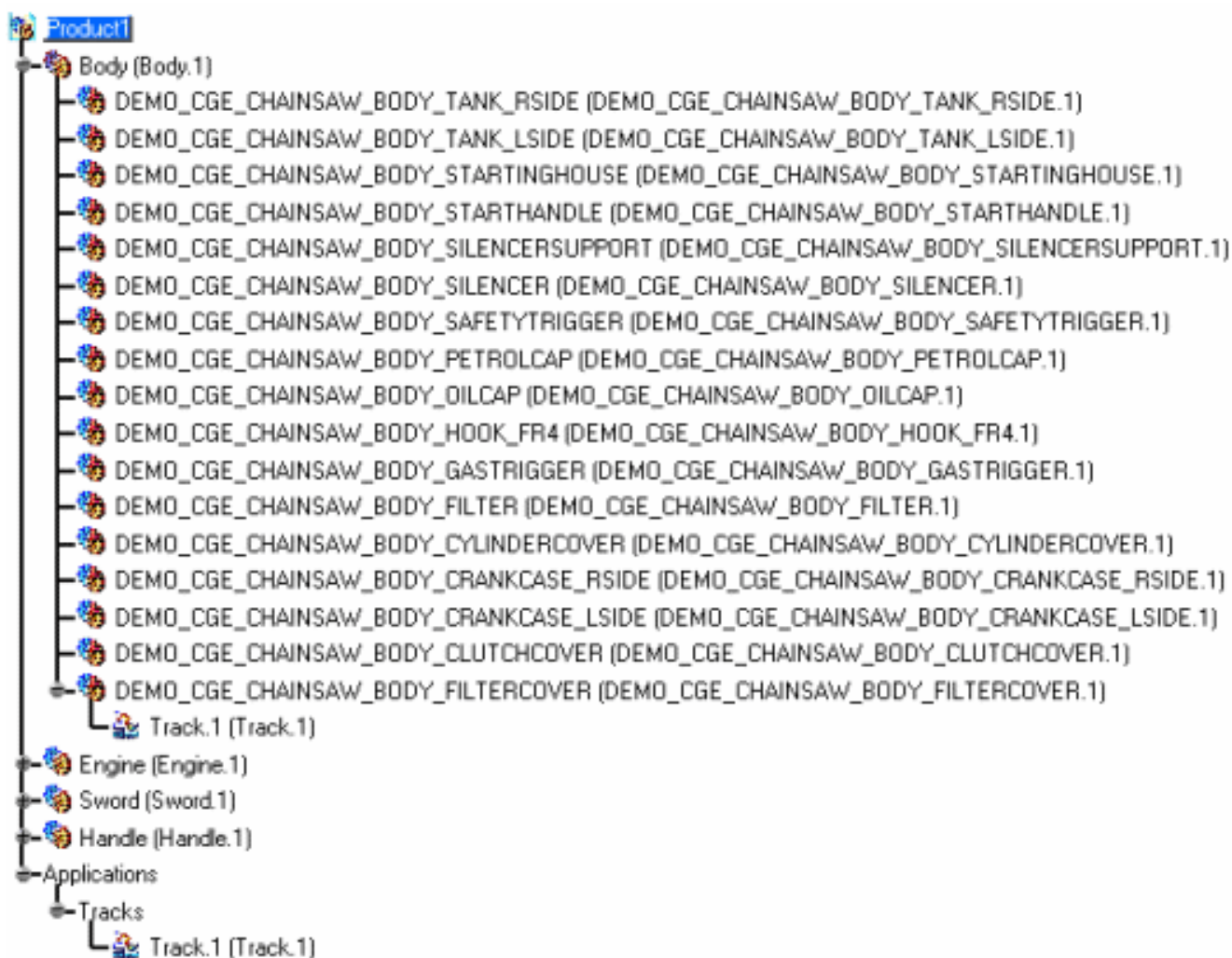
- To move a shot up one place in the list, select the shot, and press the Up button.
- 
- To move a shot down one place in the list, select the shot, and press the Down button.
- 
- To insert a selected shot after next clicked shot, select the shot, click on the shot you wish it to follow and push the After button.

**Note:** when editing a track, you can multi-select shots to be deleted (Select + CTRL key or Select + SHIFT key)

For more detailed information, please read [Track Editor and Recorder](#) and [Player](#) sections  
 This is an example of what you can get:



The track is created and identified in the specification tree (under Tracks item and within the product too)



4. Click OK to exit Track command. The modified position is kept

5.

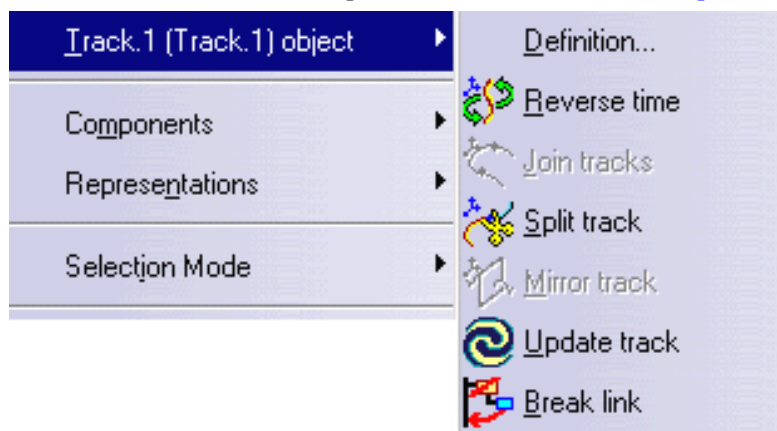


Click Reset to go back to the initial position.

**Note:** you can at any time delete the track if you are not satisfied.

6. Right-click Track.1 and select track.1 item to access the track contextual menu and various operators.

For detailed information, please read [About Track Operators](#).



For more detailed information about this menu items (referred to as track operators), see

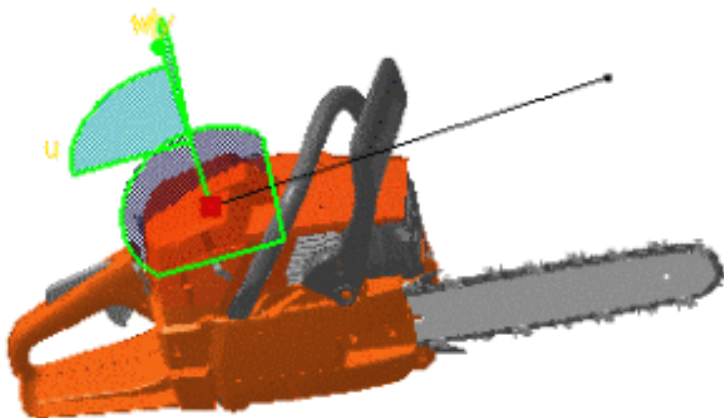
[About Track Capabilities](#) and [About Track Operators](#).

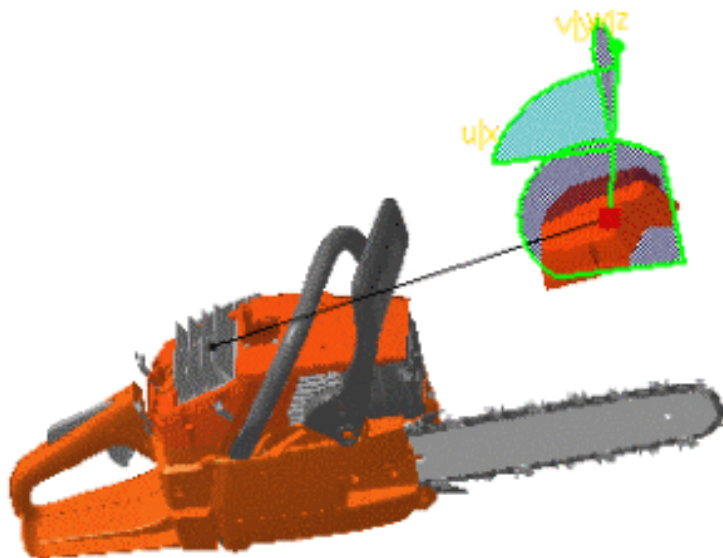
7. Double-click Track.3 if you need to:

- replay your track simulation (use the dedicated Player)
- edit the track (modify a position, delete a position...)

For instance play your track using the Play forward button:

The time is specified in seconds in the Player (corresponds to the current time)





- 8.** Check the speed and duration of your track simulation and modify it if necessary.  
All you need to do is right-clicking Track.1 in the specification tree and selecting **Properties** item from the contextual menu displayed.  
Then, enter a new speed value, the duration is recalculated.  
You can also modify the track name as desired.

Click **Ok** when done.

**Note:** the default speed is 0.001m\_s (see: [Customizing DMU Fitting](#))

When you modify the speed and/or duration, the default value is automatically overwritten with the new speed value.

#### **About modifying the speed:**

For instance, you might need to reduce the track speed because your object is too heavy (keep in mind a track comprises of the track itself + the object) , the duration is automatically recalculated (longer duration) and the speed is updated in **Tools->Options-> DMU Fitting->Default speed**.

Properties

Current selection : Track.1 (Track.1)

Track | Graphic

Naming

Name: Track.1

Instance name: Track.1

Time management

Speed: 0.001m\_s Duration: 466.711s

Log management

N°	Operator
----	----------

More...

OK Apply Close



### About customizing speed:

If you modify the speed in the DMU Fitting tab, note that this modification will be taken into account for the new tracks to be created. For already created track use the Track Properties.

For more detailed information, please refer to [Customizing DMU Fitting](#)

At any time, you can:

- modify the loop mode,
- stop at a specific step or go back to the previous step (use step backwards and step forward and stop buttons)
- skip to the starting position or skip to the end (to the very last recorded position)

Please also see: [Player](#)



# Using Automatic Path Finder



Before starting this scenario, you should be familiar with the path finder command, please refer to [Path finder chapter](#).



This task will show you how to use the path finder capability on a track simulation.

The [CHAINSAW.CATProduct](#) document is already open



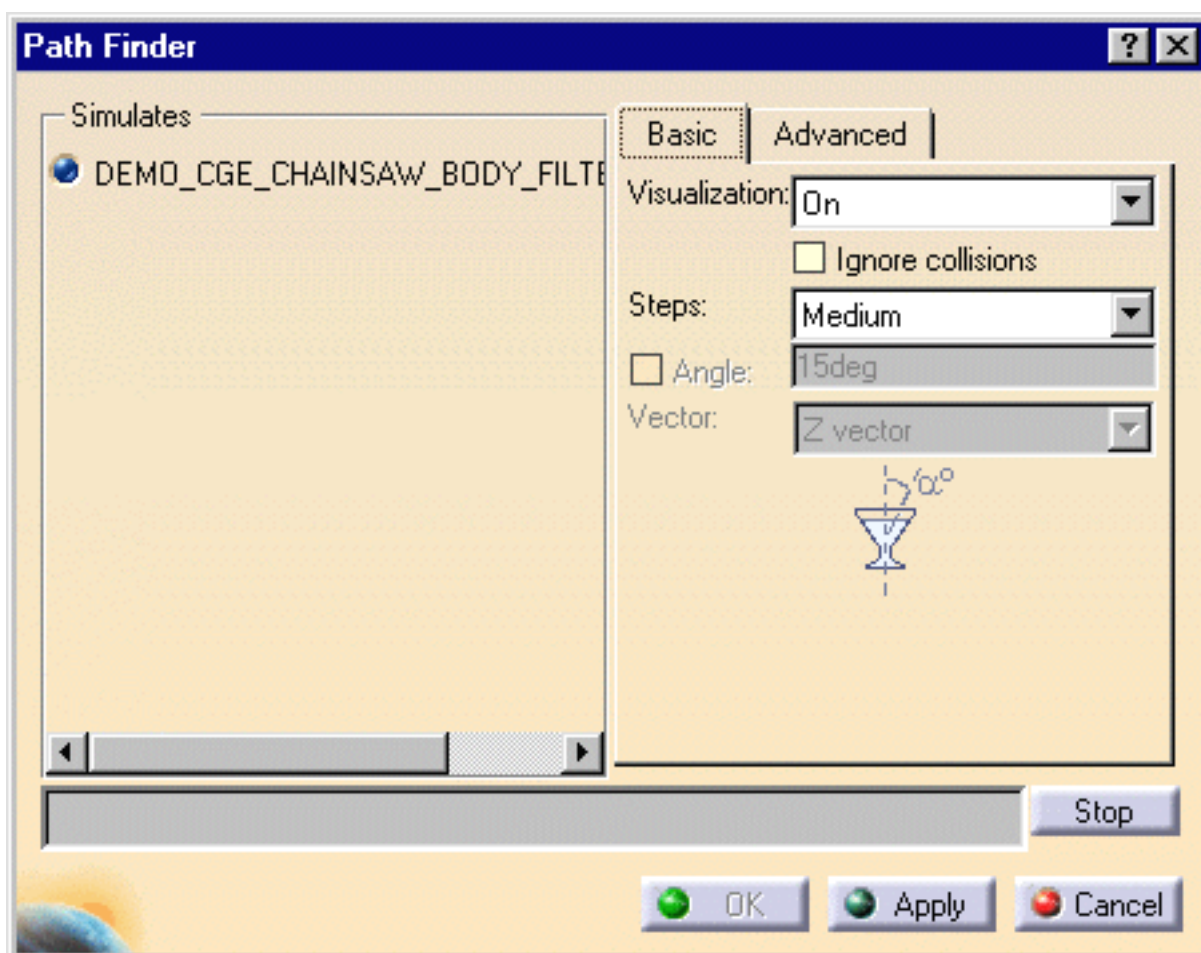
1. Select the track created in the previous task (Track.1) either in the geometry area or in the specification tree.

**Note:** you can also click the Path finder icon first and the object afterwards, in this case, the select dialog box lets you choose the objects

2.



Click the Path finder icon from the DMU Check Toolbar.  
The Path Finder dialog box is displayed...



... and lets you select options for: (The Basic mode is set by default)



### Visualization

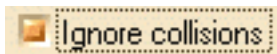
- On (shows the object move and its various positions during calculation. You can stop the process at any time)
- Off (You are not allowed to stop the process and you do not see the progression)
- Strombo (shows positions based on a specific parameter (every 20 steps))

### Steps

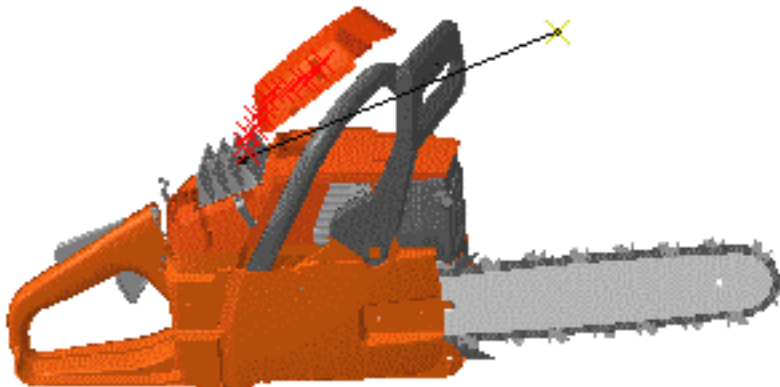
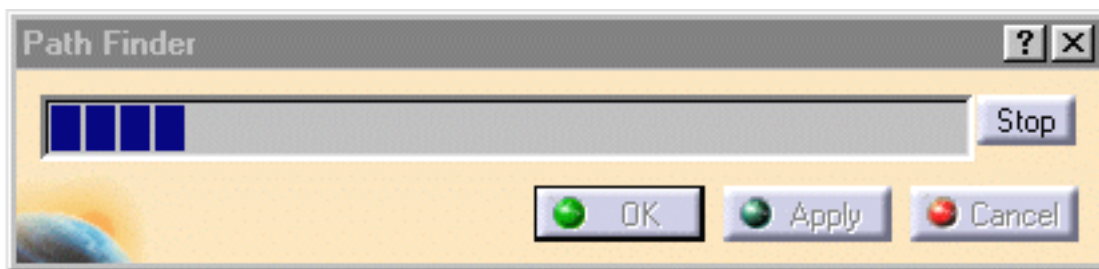
- Small
- Medium
- Large

3. Keep the default settings:  
Visualization: on  
Steps: Medium

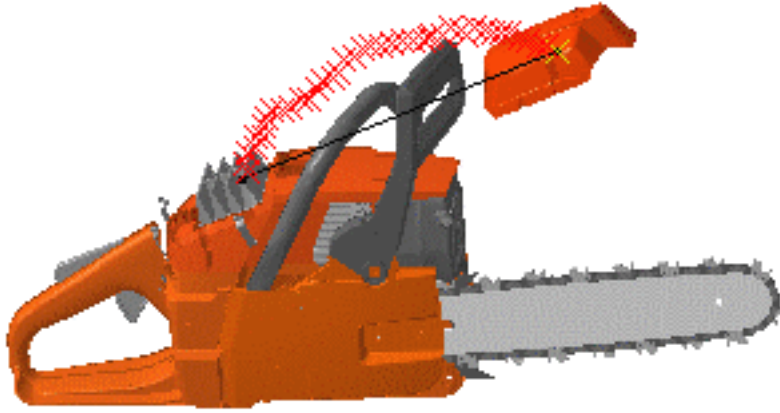
4. Check the Ignore collisions checkbox to get rid of the irrelevant interferences.



5. Click **Apply** to confirm your operation.  
The progress bar is displayed letting you monitor and, if necessary, interrupt (Stop option) the calculation.

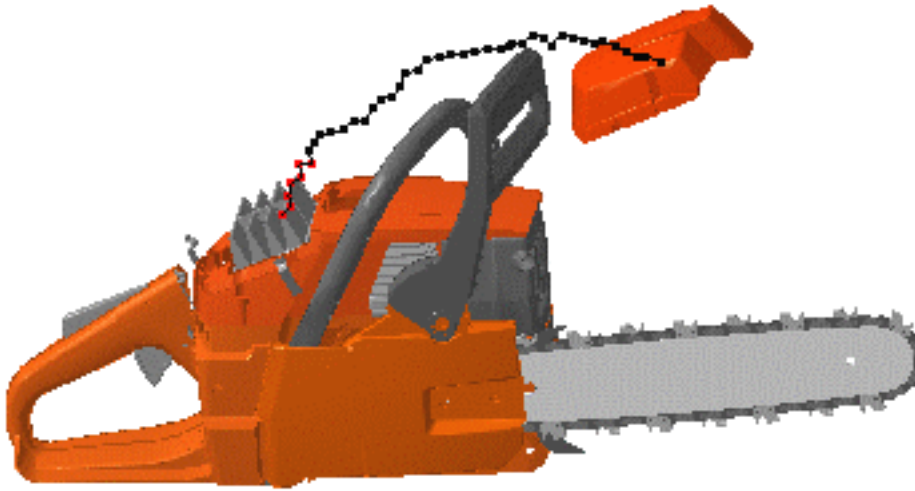






5. Click **Ok**.

This is what you obtain:

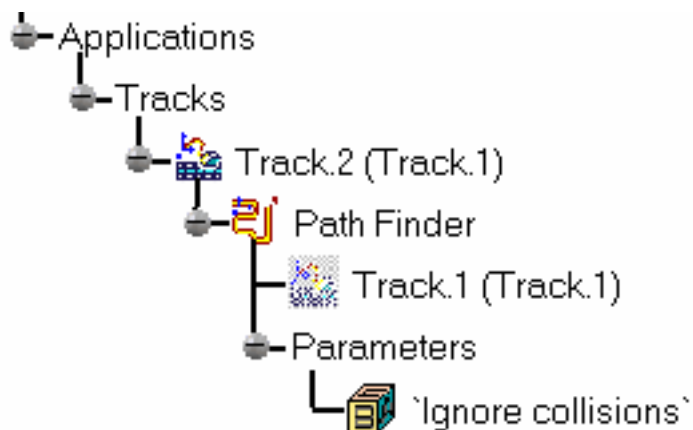


The path finder is a track operator.

It is identified in the specification tree under a new track (Track.2), Track.1 is

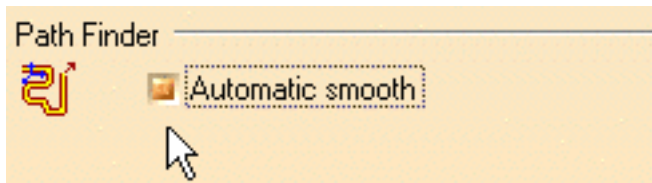
The option "ignore collisions" is identified under Parameters (knowledge parameters)

Please see: Managing Operators' Options As Knowledge Parameters





By default, the OK button does not perform the smooth, please select **Tools->Options->Digital Mockup->DMU fitting**. Check the Automatic Smooth option.



You are now ready to get rid of the unnecessary positions detected during path finder operation.



# Using The Smooth Command



This task will show you how to smooth the automatic path finder defined in the previous task.



1.

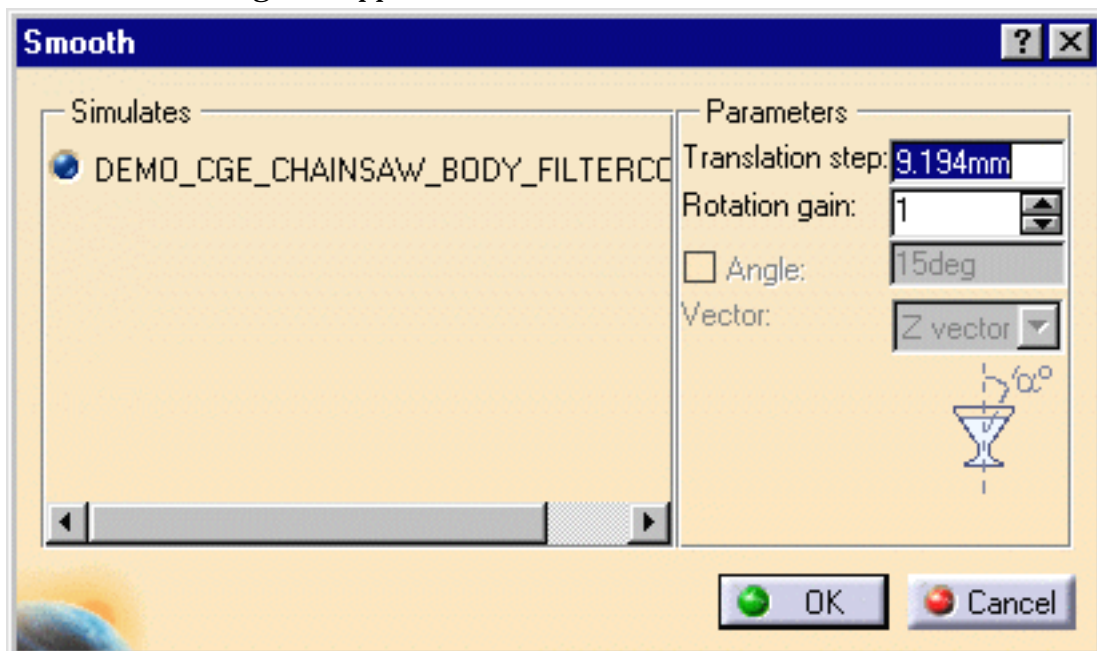
Click the Smooth icon  from the DMU Check toolbar

2.

Select the required track in the Select dialog box displayed.  
i.e: Track.4



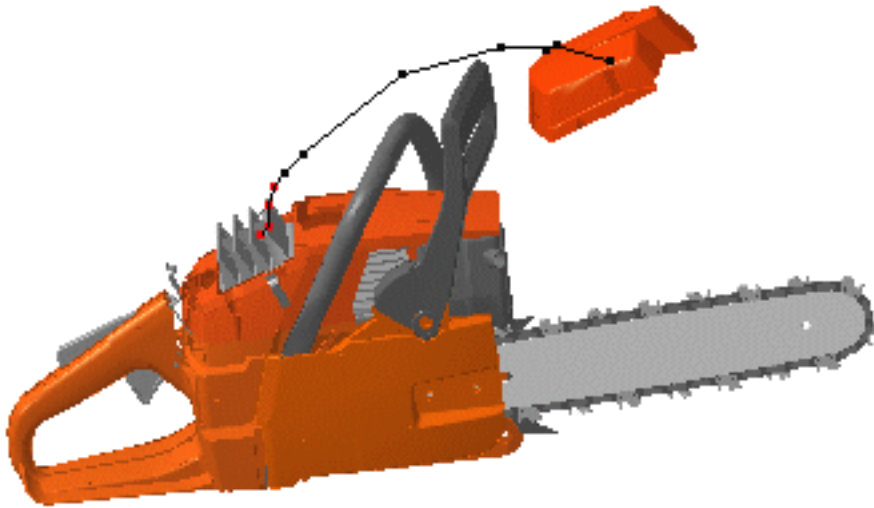
The Smooth dialog box appears



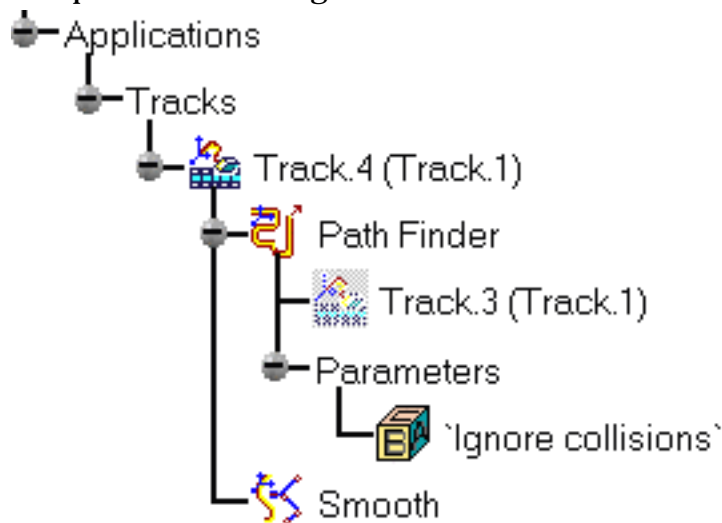
3.

Click **Ok**

This is what you obtain:

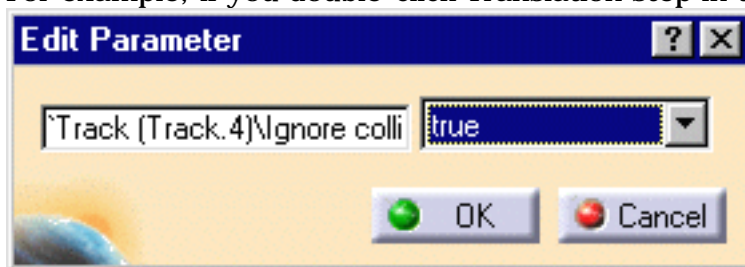


The smooth result ( track operator) is identified in the specification tree under Track.2  
The parameters changed are identified too.



**Note:** you can edit the parameters double-clicking them to display the Edit parameter dialog box.

For example, if you double-click Translation step in the specification tree:



**Note:** You can also display the track properties using the contextual menu.



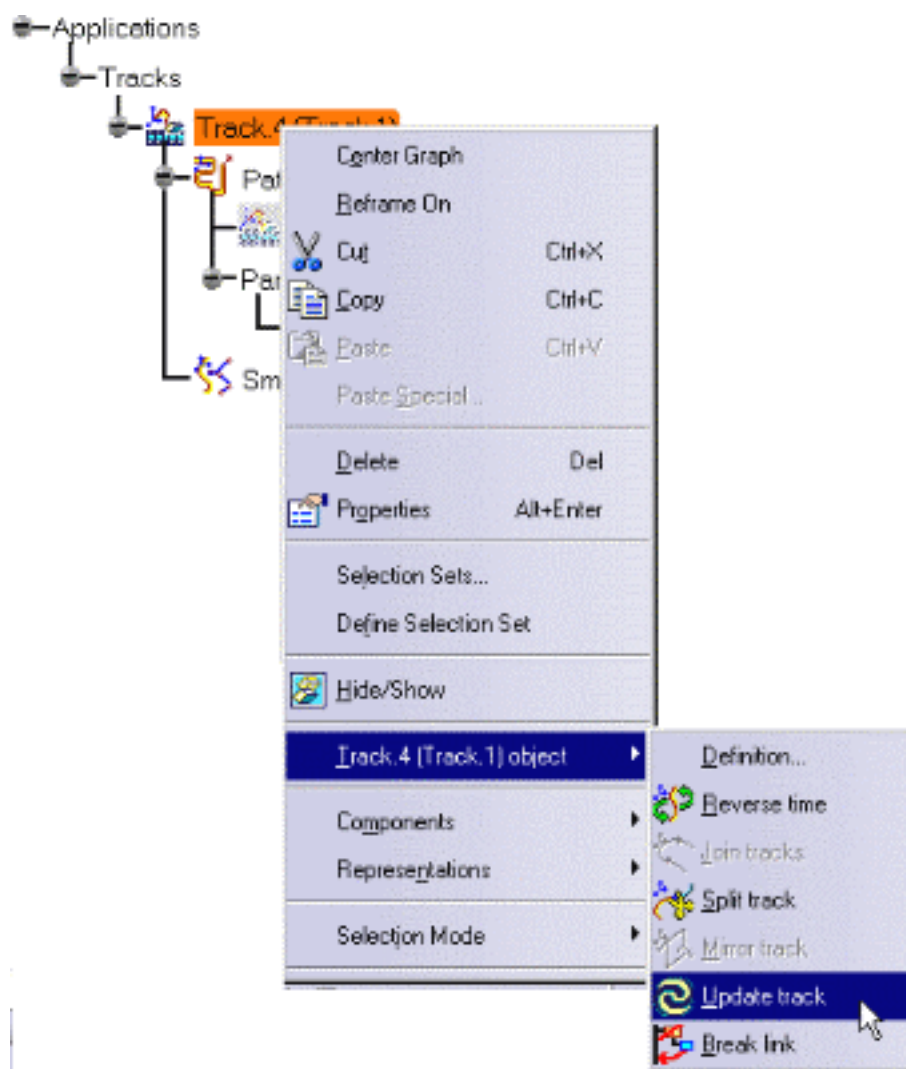
# Updating a Track



This task will show you how to update a track. We applied consecutively the following track operators: **path finder** and **smooth**, we need now to update the resulting track.

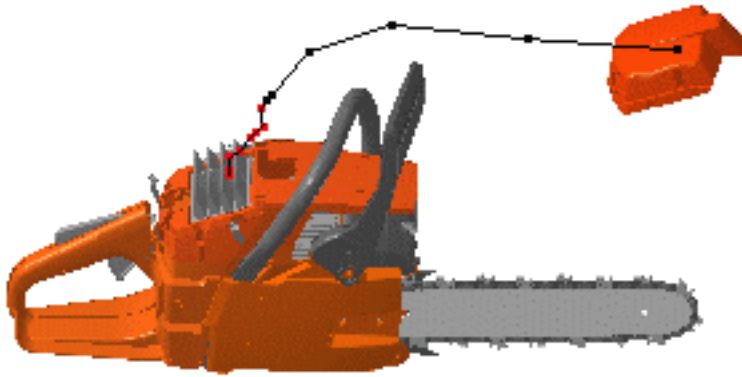


1. Right-click the track to be updated in the specification tree, i.e Track.4
2. Select the **Track.4 object**->**Update track** items from the contextual menu displayed



The path finder and the smooth computations are automatically launched

The track is updated accordingly



# Replaying a Track



This task will show you how to replay a track using the dedicated Player



1. Double-click the track created (i.e. Track.1) in the specification tree or click the Track icon



in the DMU Simulation toolbar

- 2.

The player is displayed:

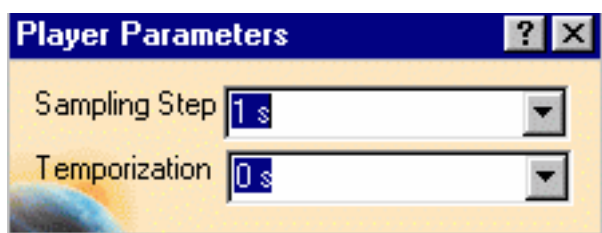


- 3.



Click the Parameters icon

4. The Play Parameters dialog box appears:




- 5.

Modify the time step and the temporization values if necessary.



- 6.

Use the VCR buttons or the slider to simulate your track.

-  skip to beginning
-  step backwards
-  play backwards
-  stop
-  play forwards
-  step forwards

-  skip to end



For more information, please refer to [Player](#)





# Defining a Swept Volume




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*.

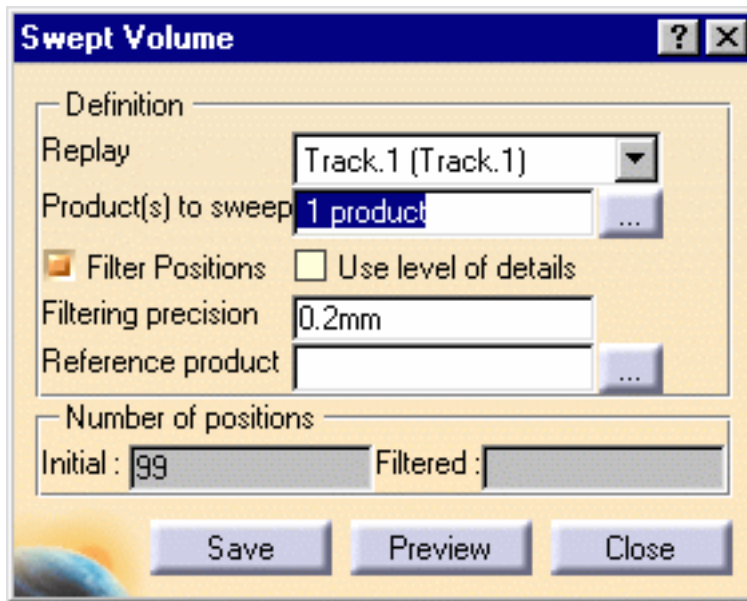


This first task will show you how to enter the Fitting Simulator workbench and select your products.



1.

Select Track.1 in the specification tree and click the Swept volume icon . The Swept volume dialog box appears:

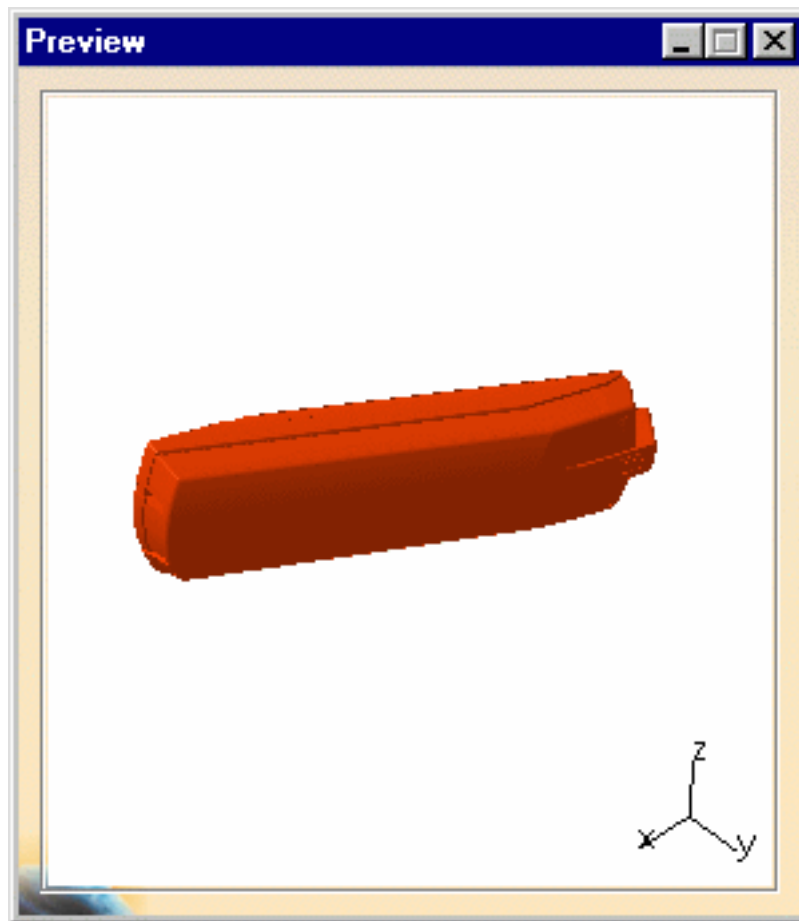


2.

Click **Preview**.

The computation is in progress

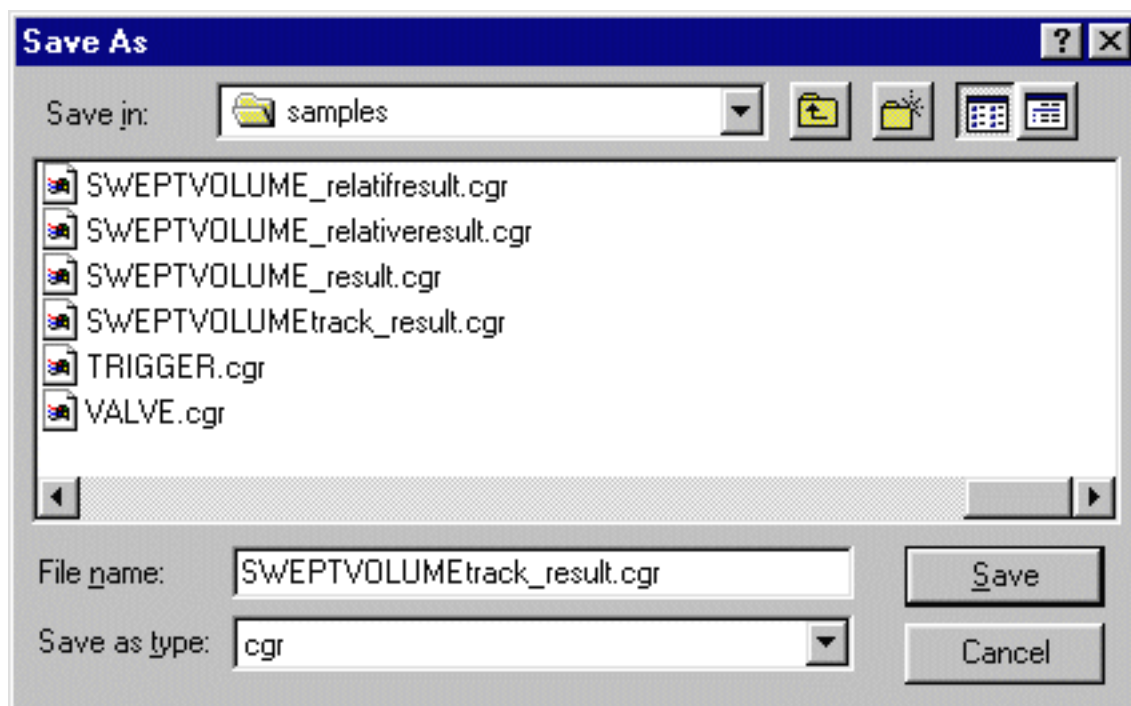
The preview window appears:



3. Click **Save** in the Swept volume dialog box.

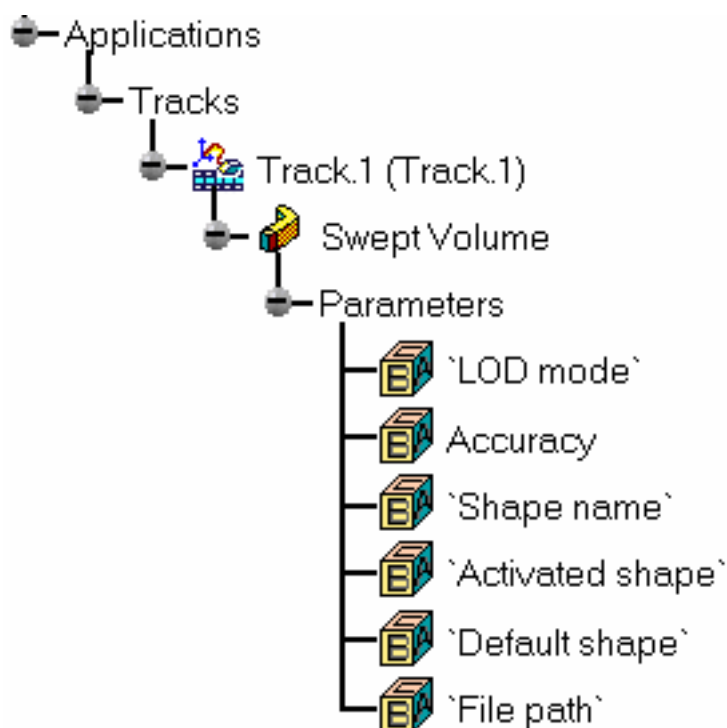
The Save As dialog box is displayed

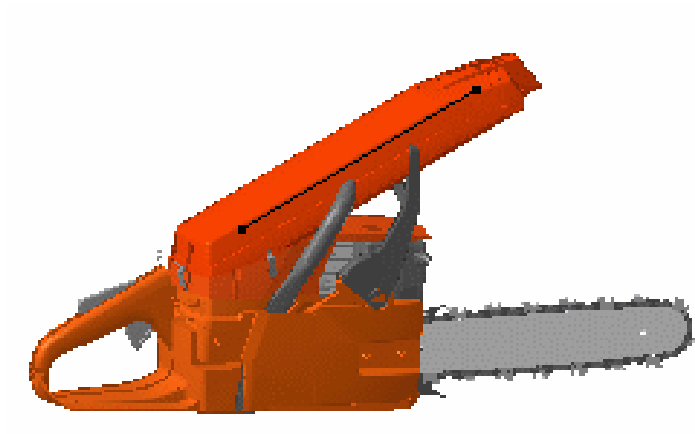
4. Select cgr file from the drop-down list



5. Enter a meaningful name and click **Save**.
6. Insert the SWEPTVOLUME\_track.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.





# Using Shuttles



Before getting into the detailed instructions for using DMU Fitting Simulator, the following tutorial aims at giving you a feel of what you can do with the product. It provides a step-by-step scenario showing you how to use key functions.

The main tasks described in this section are:

Starting a Session

Exploding a Product

Defining a Shuttle

Recording a Simulation

Replaying the Simulation



These tasks should take about 15 minutes to complete.

# Starting a Fitting Simulator Session



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 Fitting Simulator workbench and select your products.

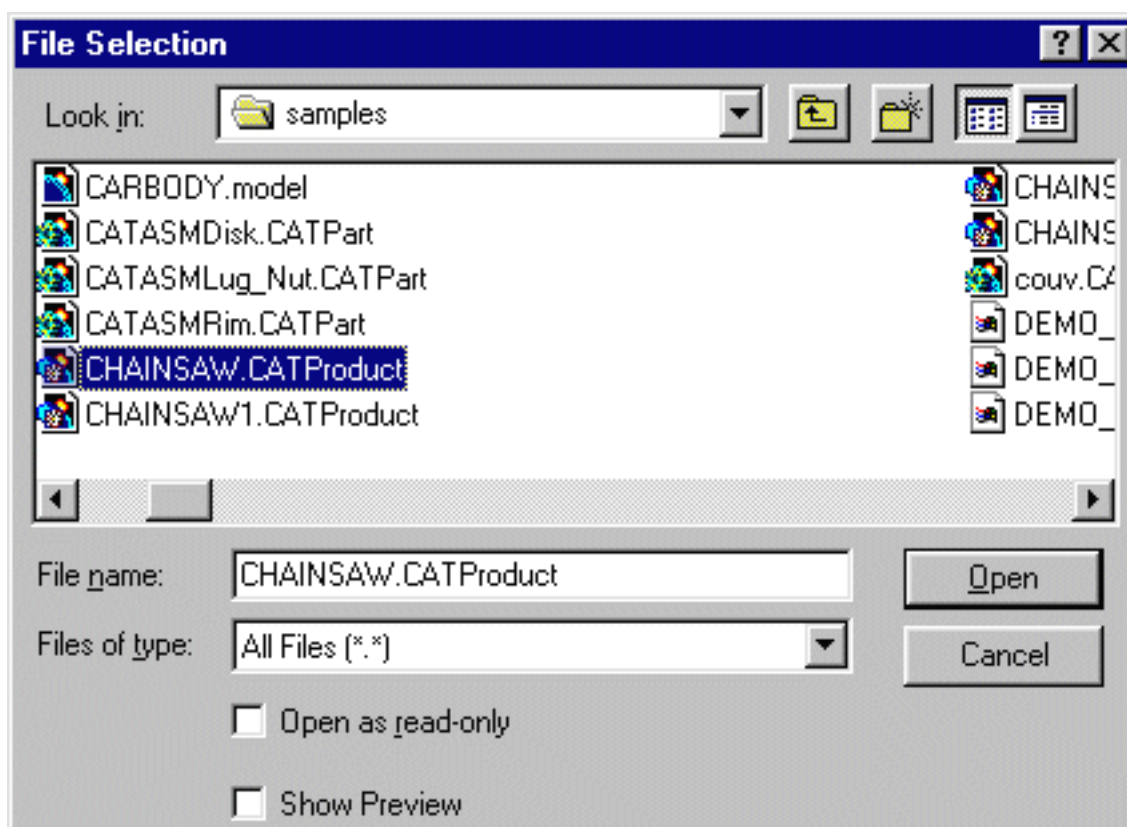


1. Select **Digital Mockup** -> **DMU Fitting** from the **Start** menu.

The Fitting Simulator workbench is displayed.

2. Select **File**->**Open**.

The File Selection dialog box displays:



3. Select the **CHAINSAW.CATProduct** from the samples folder.

4. Click **Open** to open the selected files.

The specification tree is displayed showing all the selected products. In our example, the CHAINSAW.CATProduct has been divided in four main groups of objects.

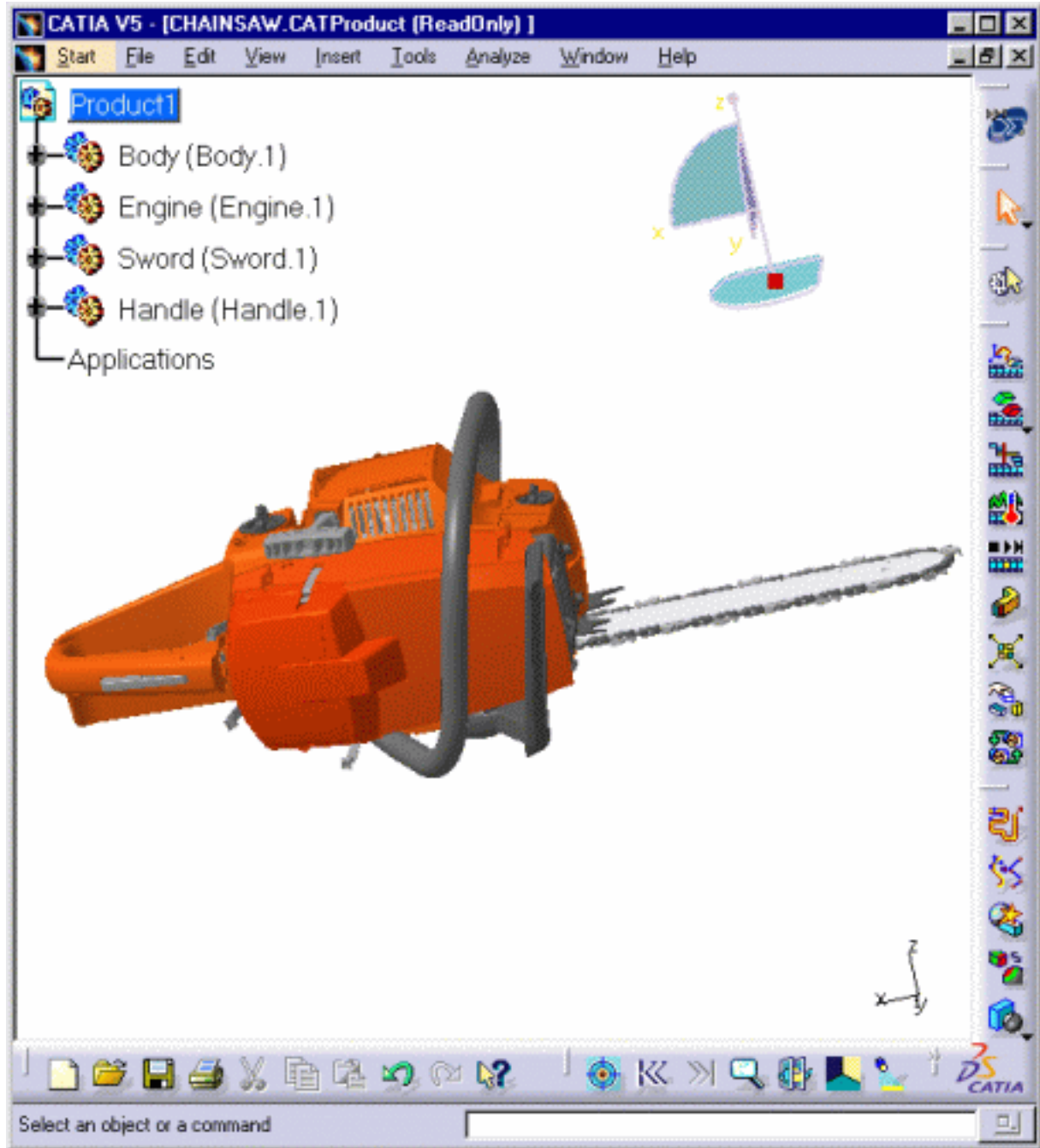


Note that you can also select model files using **insert**-> **existing component**.

In this case, select the desired model files by clicking the first one then shift-clicking the last one you want.



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



Clicking off **View -> Specifications** visible in the menu bar removes the specification tree and lets you use the entire screen for the product. You can also use the F3 key to toggle more quickly.



# Exploding a Product



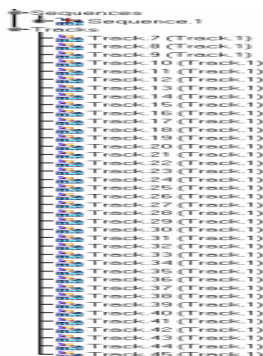
This task illustrates the use of the Explode capability. Exploding the view of an assembly means separating the components of this assembly to see their relationships. At any time though, you can check the product structure via the specification tree. The explode capability allows you to fully understand the product structure in a 3D context.



CHAINSAW.CATProduct document is opened



1. Select **Product1** in the specification tree.

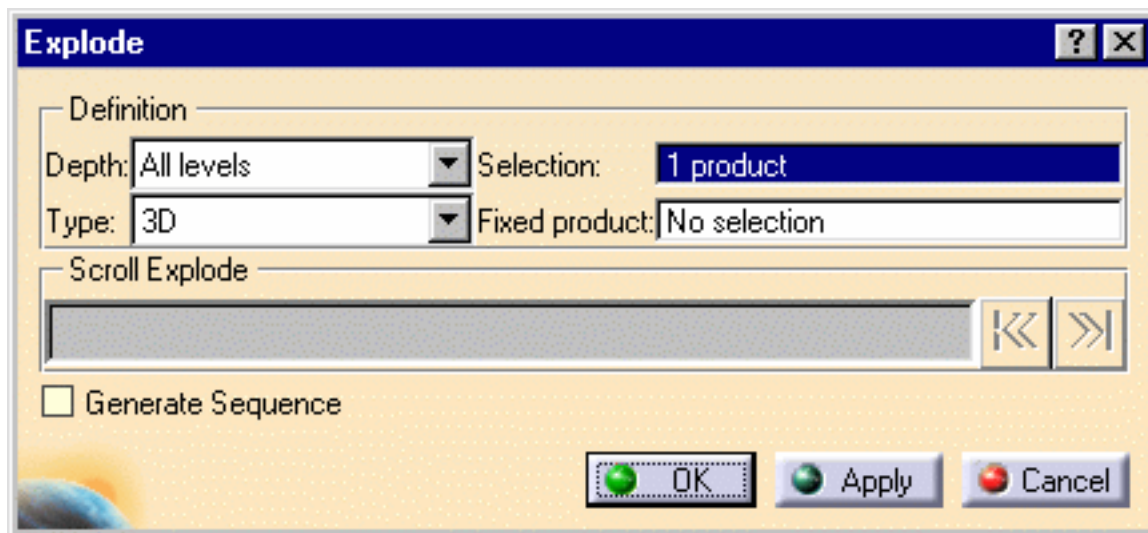


- 2.



Click the Explode icon.

The Explode dialog box and the Manipulation toolbar are displayed



Product 1 is the assembly to be exploded. The **Depth** parameter lets you choose between a total (**All levels**) or partial (**First level**) exploded view.

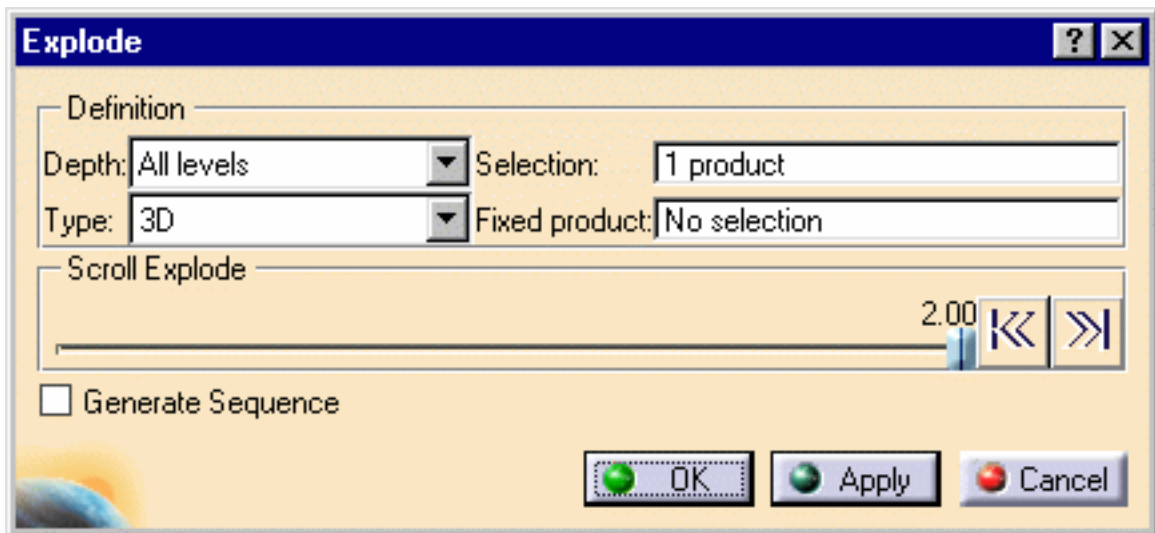
3. Set **All levels** if not already set.
4. Set **3D** as explode type.

You can now generate a sequence from the exploded view, all you need to do is click the Generate Sequence checkbox before launching the explode operation.

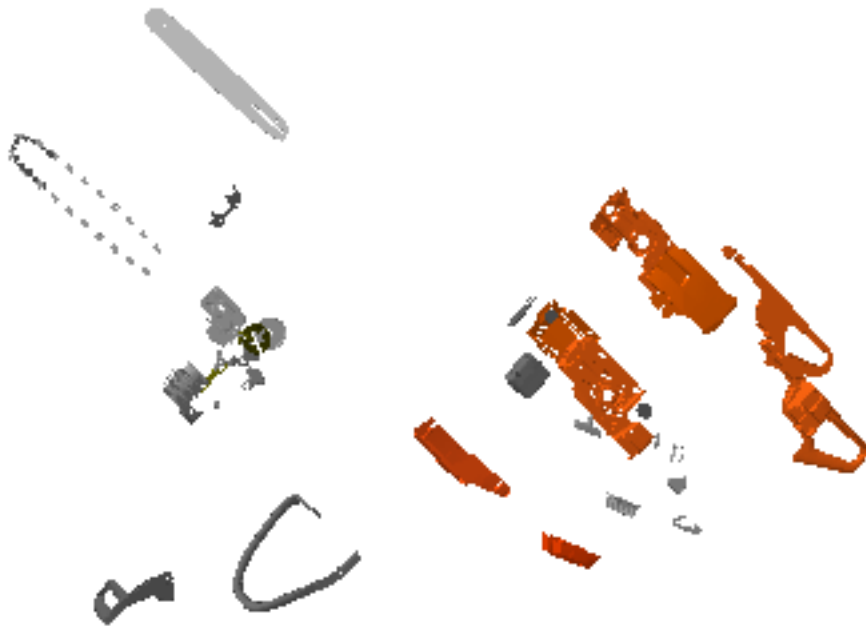
For more detailed information refer to [Generating a Sequence from an Exploded View](#)



5. Click **Apply** to perform the operation.



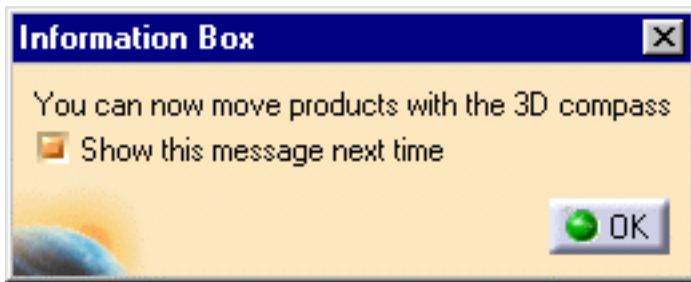
The **Scroll Explode** field gradually displays the progression of the operation. The application assigns directions and distances. Once complete, the assembly looks like this:



The interest of this operation lies in the ability of viewing all components separately.



You can easily move products within the exploded view using the 3D compass.



6. Click **Apply** to confirm the operation or click **Cancel** to restore the initial view.



# Defining the Shuttle



Before starting this scenario, you should have loaded your CHAINSAW.CATProduct as described in the previous step.



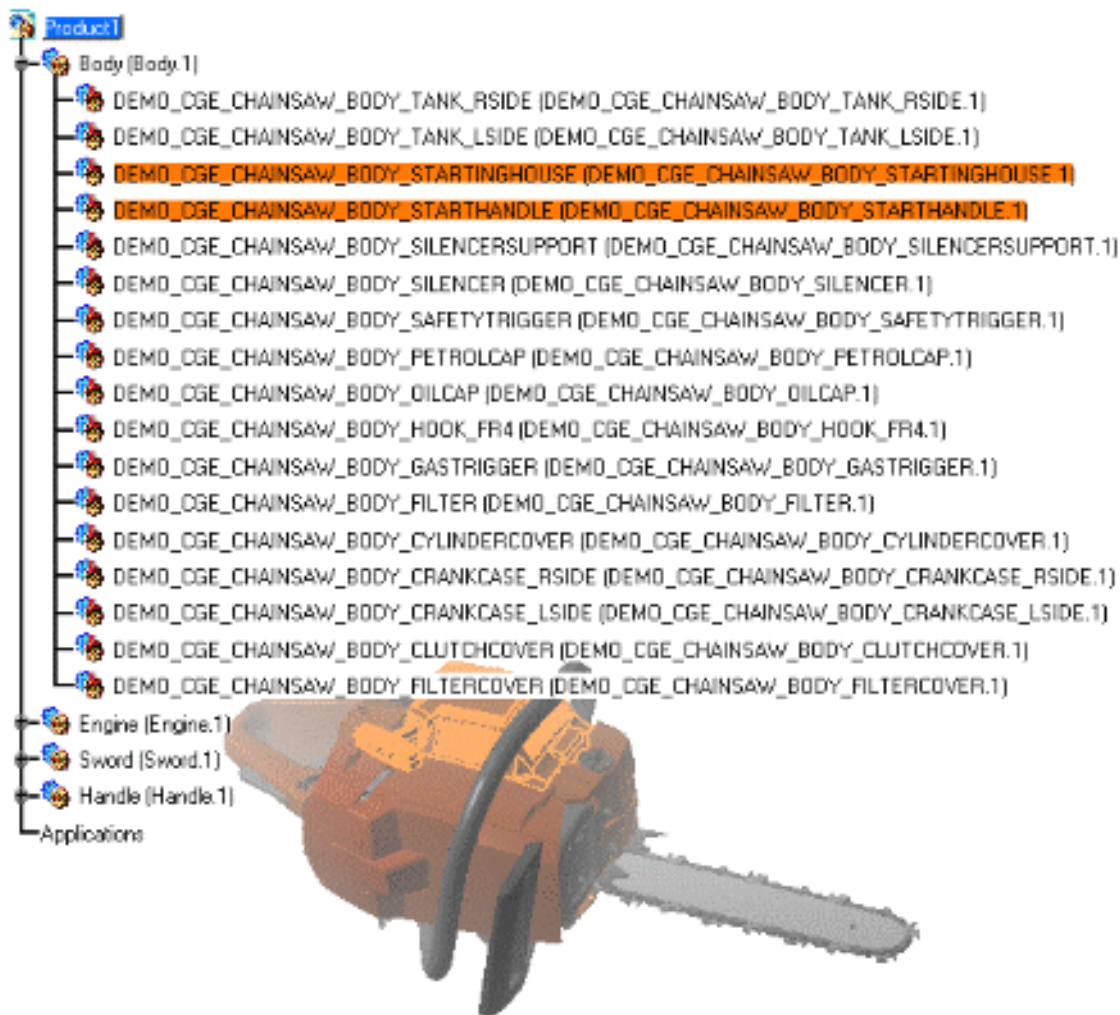
A shuttle is a set of products defined explicitly by selecting products individually. Shuttles are persistent and can be stored in the document.




This task explains to you how to define the list of products you want to move together.



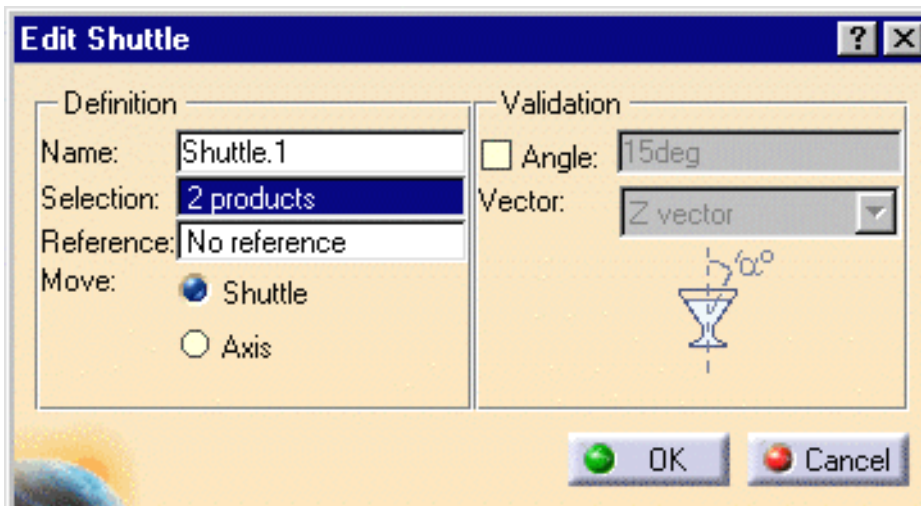
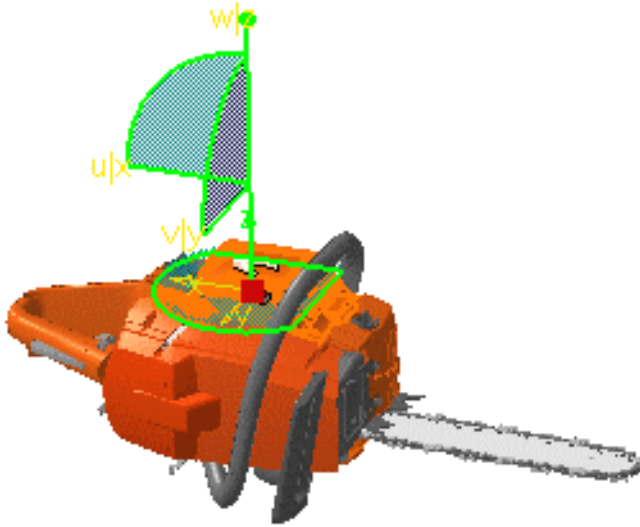
1. Select a product in the geometry area or in the specification tree.
2. Ctrl-click other products to add them to the initial selection



3.

Select **Insert->Shuttle...** from the menu bar or click the **Shuttle**  to create a shuttle:  
The Edit Shuttle dialog box and the Preview window appear.

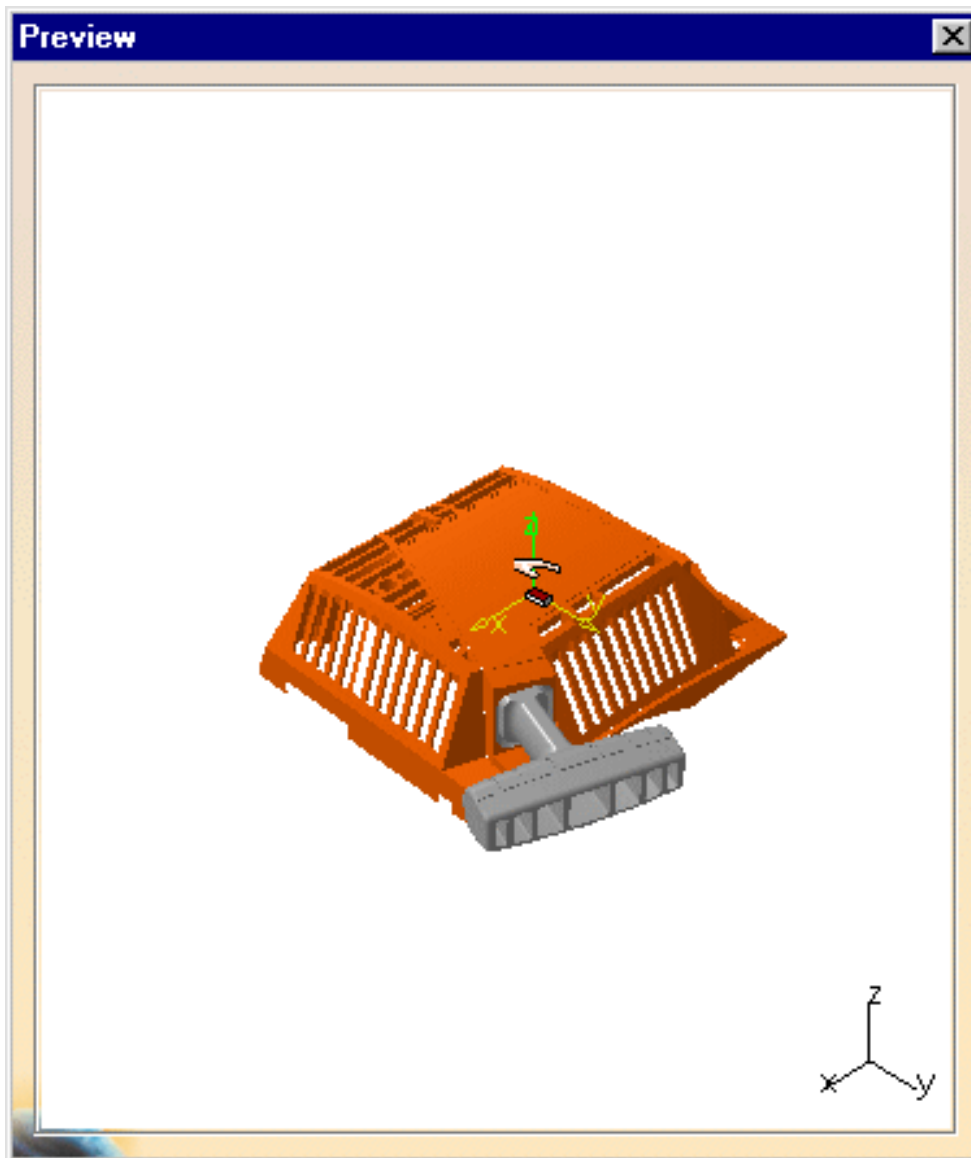
The 3D compass automatically positions according to the to-be-created shuttle axis. See below:



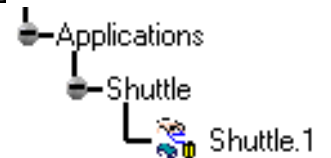
You can define a maximum angle of rotation for the shuttle around the absolute axis you select. It means the shuttle motion is defined and validated with respect to the angle value defined. The Preview window shows selected products.

**4. (Optional)**

Enter a meaningful name for the shuttle you want to create.



5. Click **OK** to create the shuttle.  
The shuttle is identified in the specification tree.



Shuttles created in this manner are persistent and can be stored in the document. They are listed as separate entity in the specification tree and can be selected at any time and modified.



# Recording a Simulation



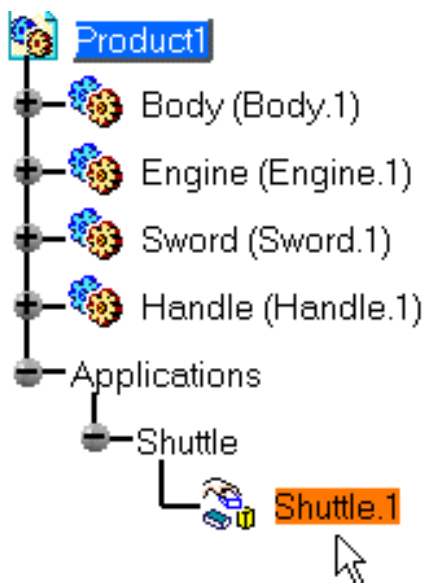
Before starting this scenario, you should have created a "Shuttle" as described in the previous steps.



This task explains how to quickly define the motion of a "Shuttle" and to record it in a simulation object.



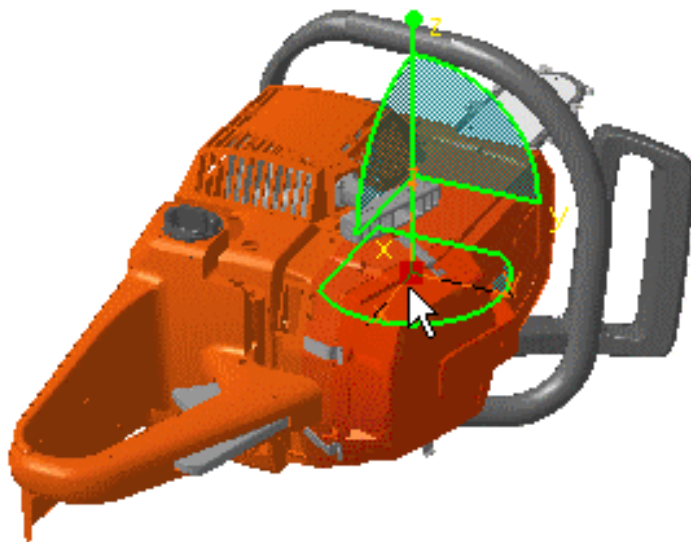
1. Select Shuttle.1 in the specification tree.



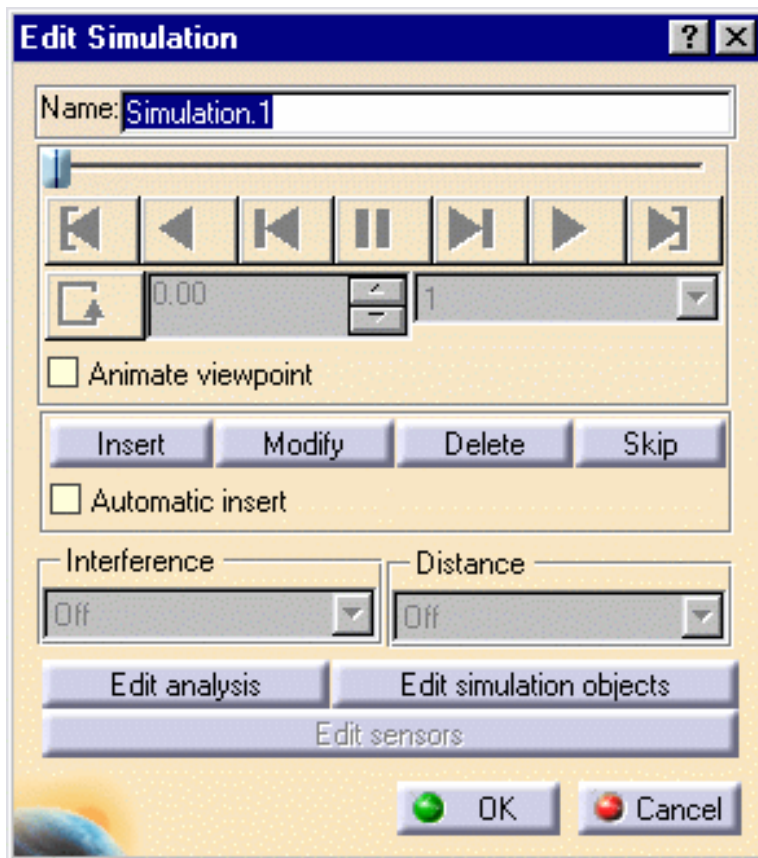
- 2.

Select **Insert->Simulation** ...

The 3D compass automatically positions according to the shuttle axis.

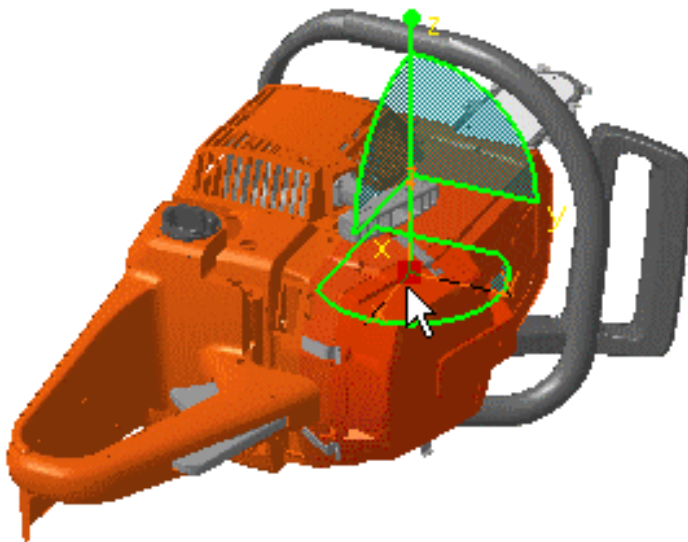


The Edit Simulation dialog box and the Preview window appear.  
You are ready to record a simulation.



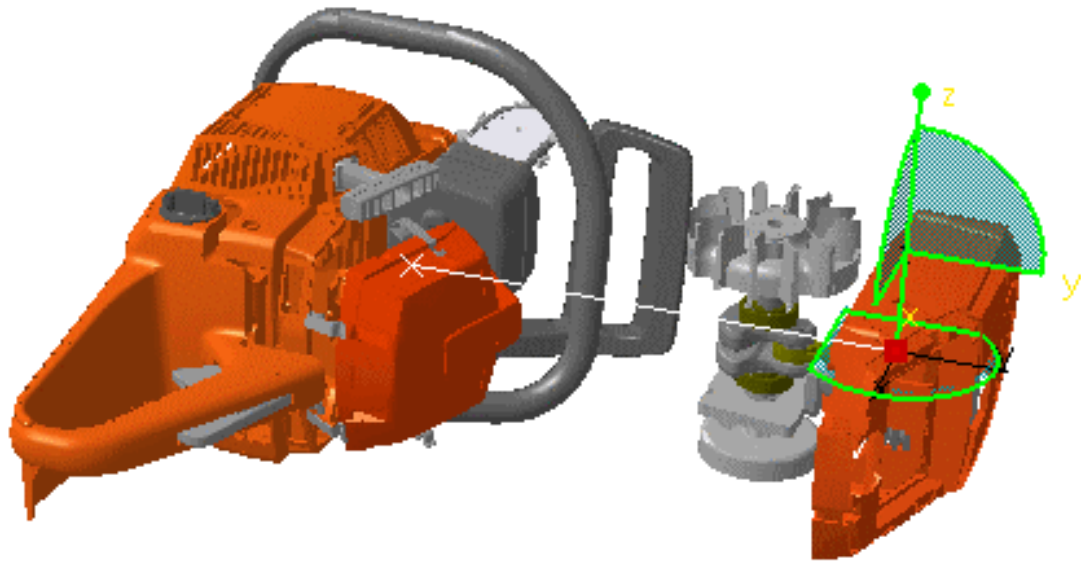
You can now record a simulation with an animated viewpoint. For this, activate the Animate viewpoint option. For more details, please refer to [Animating a Viewpoint](#) .

The first shot: shuttle in the origin position is already recorded in the simulation.

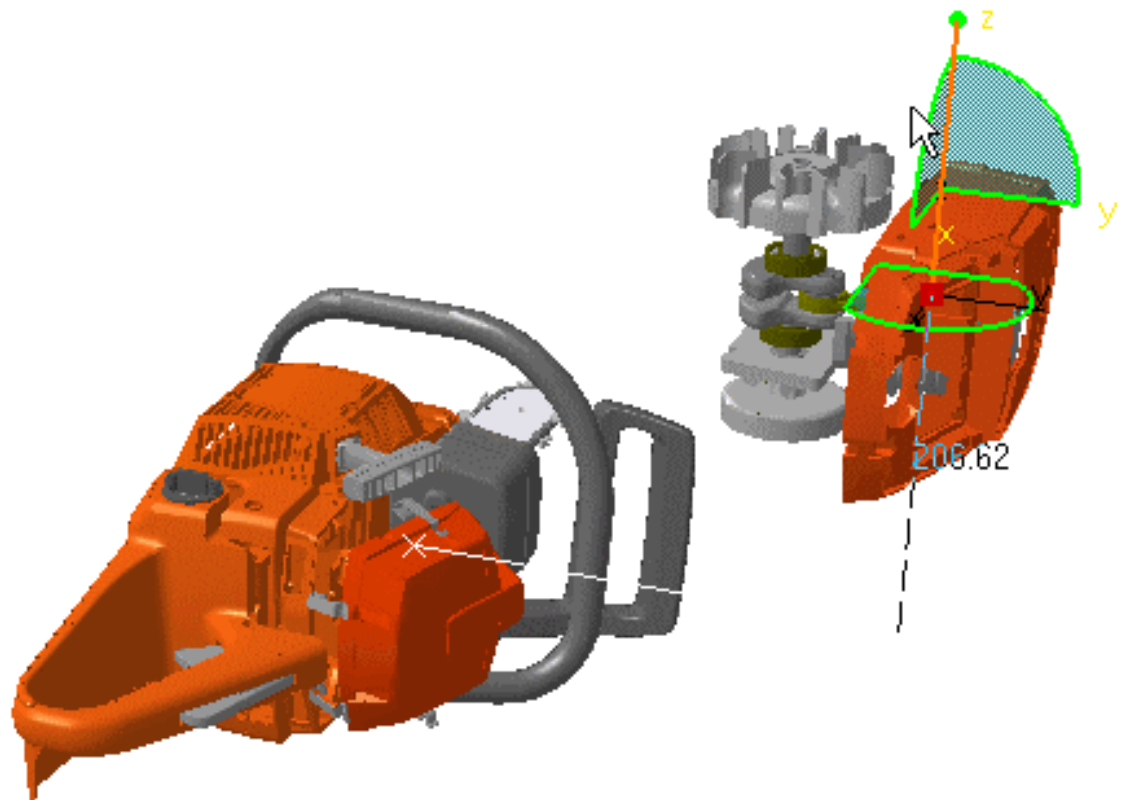


- 3.** Select the manipulator to move the shuttle as desired.
- 4.** Drag the cursor to the desired shuttle location.





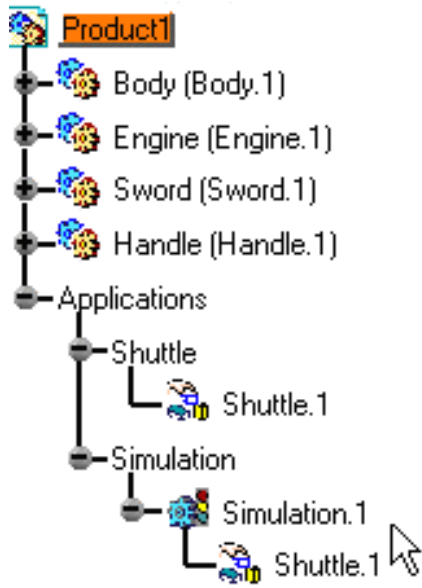
5. Click **Insert** to save and insert in the "Simulation" the shuttle in the new position.
6. Insert another location in the same manner:



You can select and drag all the components of the manipulator (axis, plane, arc, point) to move in different ways.

7. You can do several manipulations before inserting a new position in the "Simulation".
8. Click **OK** to end the Simulation creation.  
The simulation is identified in the specification tree.





# Simulating a Replay



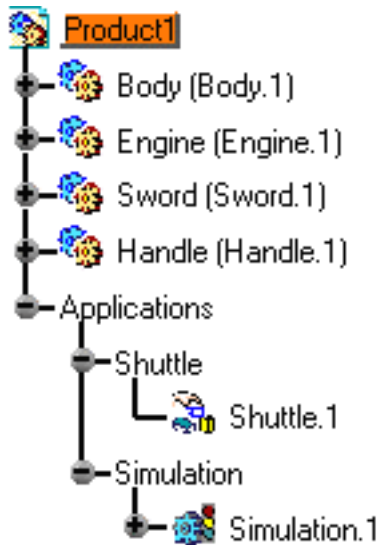
Before starting this scenario, you should have created a "Simulation" with a shuttle at different locations as shown in the previous step. The CHAINSAW.CATProduct document is already opened.



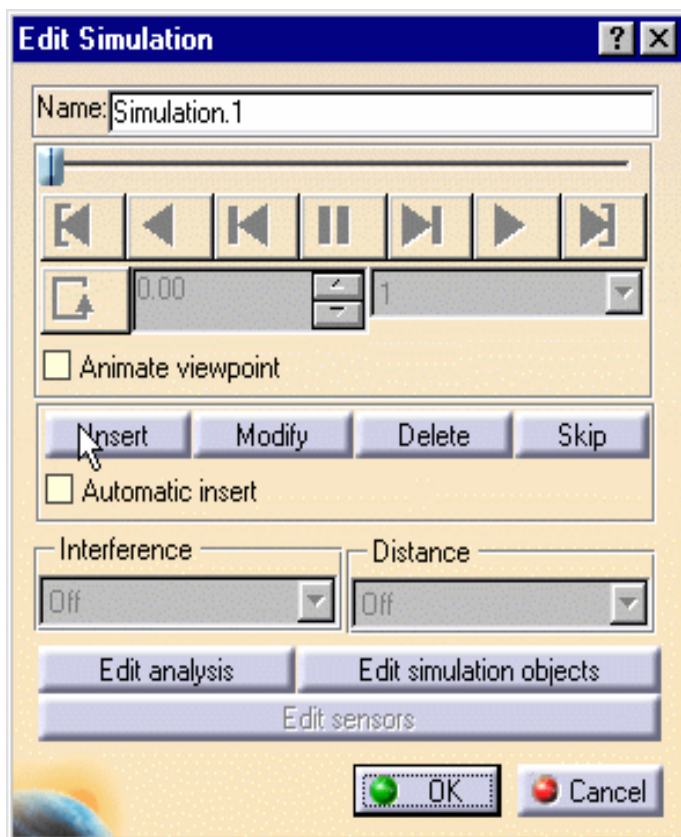
This task explains how to run a shuttle motion recorded in a "Simulation" using the Simulation command.



1. Double-click your "Simulation" in the specification tree (**Simulation.1**)



The Edit Simulation dialog box and the Preview window appear.

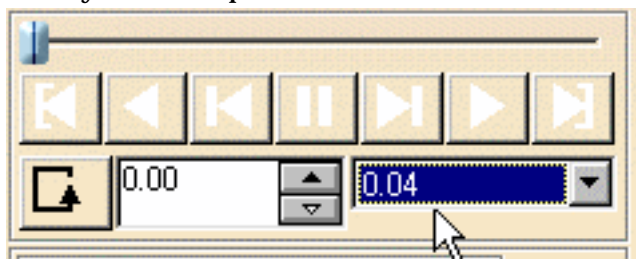


If you checked the Animate viewpoint option in the simulation recording, you can now animate viewpoints while recording a simulation. They are taken into account. For more details, please refer to [Animating a Viewpoint](#).



You also can select a "Simulation" graphically by clicking on the tracks.

2. Modify the interpolation value as desired. For example: 0.04



3. Click the Play Forward button.  
The recorded motion is replayed.





At any time, you can go back to the starting shot. Use the VCR buttons as desired:

- Play Forward (step by step)
- Go to Maximum Time position
- Pause
- Go to Zero Time position
- Backward (step by step)
- Play Back.



# Basic Tasks

The table below lists the information you will find in the Basic User Tasks section.

Setting Up Your DMU Fitting Simulator Session

Shuttle

Mono-Shuttle Fitting Simulation

Target and Snap Capabilities

Tracks

Sequences

Generating Animations

Exploding

# Setting Up Your Fitting Simulator Session



**Enter the Workbench:** select **Digital Mockup** -> **DMU Fitting** from the **Start** menu. An empty document opens.



**Define a Shuttle:** 3 methods are available to create shuttles:

- Clicking the **Shuttle** icon first and select the objects afterwards.
- Selecting the objects first and activating the shuttle command after.
- Right-clicking Shuttle in the specification tree. The contextual menu is displayed.

# Entering the Fitting Simulator Workbench

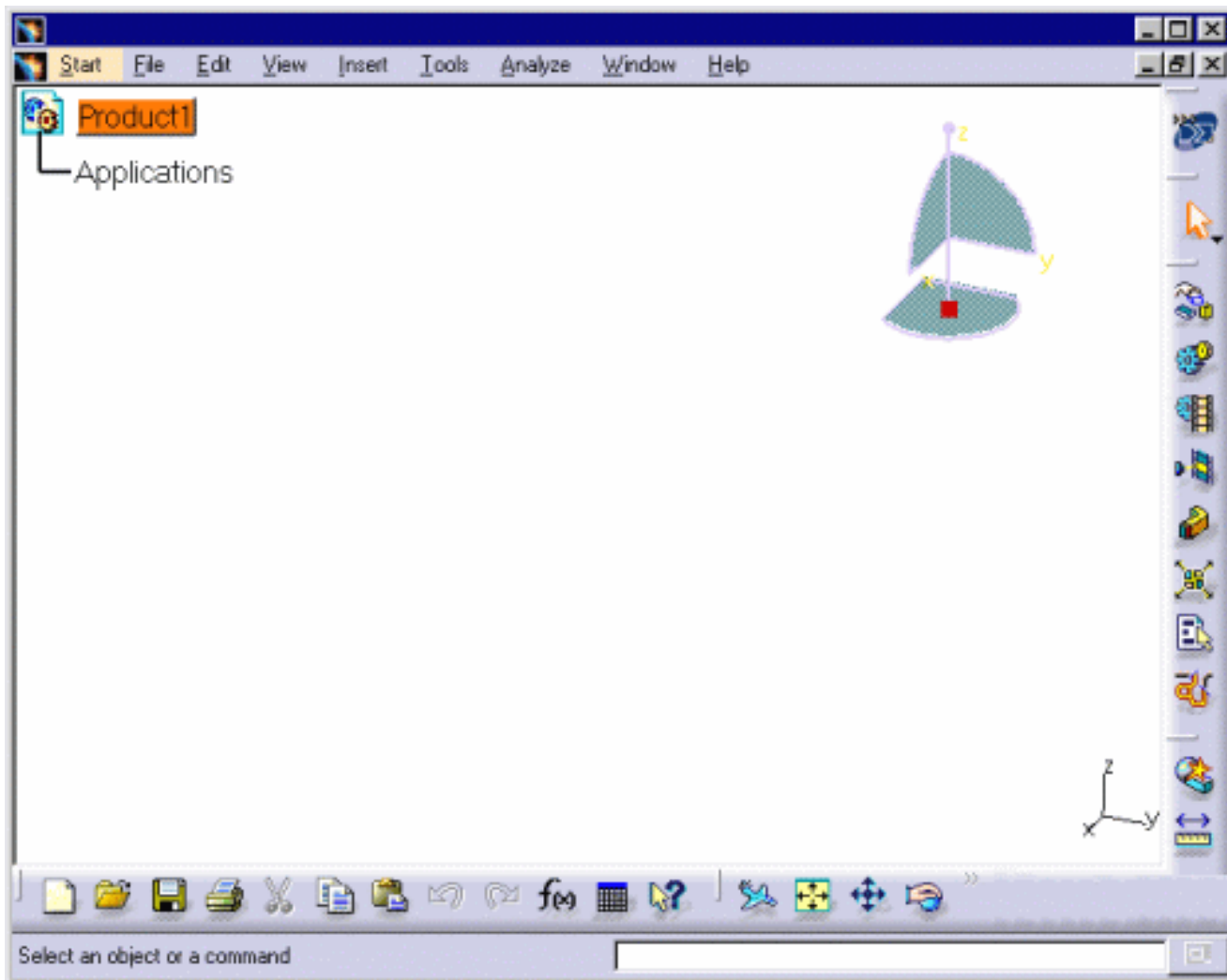


This task shows you how to enter the DMU Fitting workbench and open a new document.



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

The Fitting Simulator workbench is loaded and an empty document opens:



The Fitting Simulator workbench comprises:

- A Specification tree and a geometry area
- Specific Toolbars
- A number of contextual commands available both in the specification tree and the geometry area.

2. You are now ready to open models and products either using the **Insert** -> **Existing Component...** command or the **Open**->**File...** command.



Clicking off **View** -> **Specifications Visible** in the menu bar removes the specification tree and lets you use the entire screen for the geometry. You can also use the F3 key.



Note that more toolbars may appear next to the Standard toolbar when you create a document.





# Defining a Shuttle



A shuttle is a set of products defined explicitly by selecting products individually. Shuttles are persistent and can be stored in the document



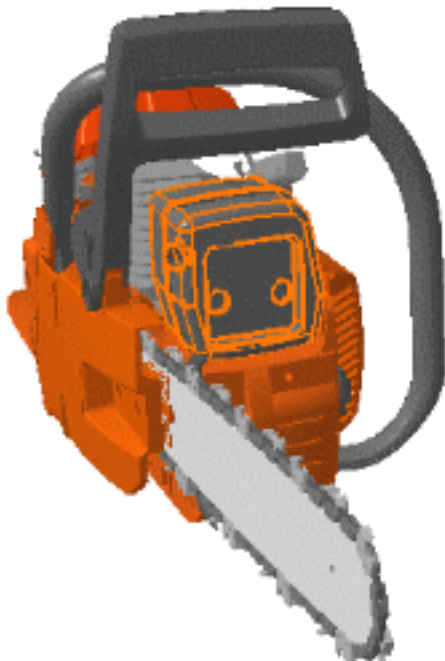
Open the [TANK\\_WITHOUT\\_SHUTTLE.CATProduct](#) document.



This task explains how to define a shuttle. You will move this group of objects afterwards using a manipulator. For more details, please refer to [Moving the Shuttle](#).

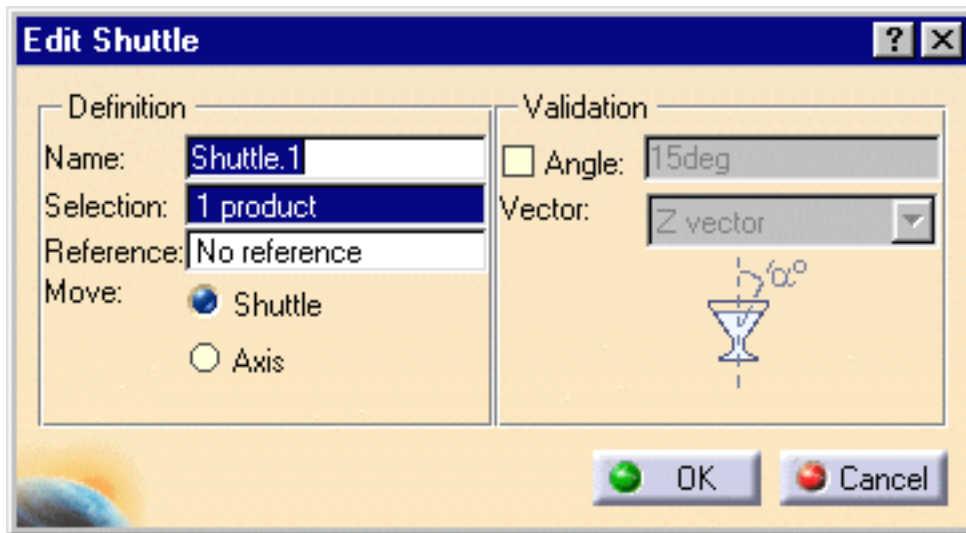


1. Select the required product in the geometry area as shown below:



2. Click the **Shuttle** icon  .

The Edit Shuttle dialog box and the Preview window appear:



The manipulation toolbar is also displayed:

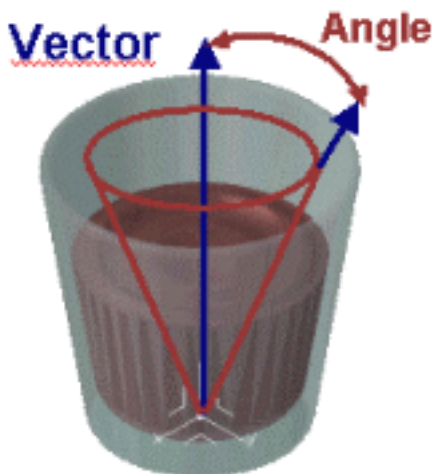


The shuttle graphic representation is displayed. It corresponds to the to-be-created shuttle axis.



You can specify a maximum rotation angle around the absolute axis for the shuttle. It can be very useful to avoid liquids to spill out from specific assemblies such as a gas tank for instance.

It means the shuttle motion is defined and validated with respect to the angle value defined.



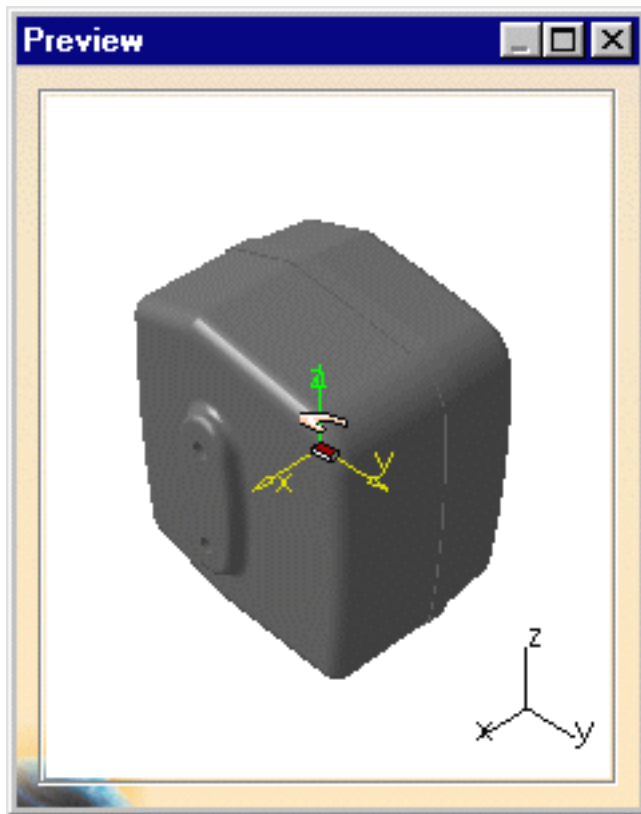
- ### 3. Check the Angle value option
- The value fields are no longer grayed out

4.

Enter 45deg as angle value and select Y vector

**Note:** Z vector is the horizontal axis therefore in our example we need to select Y vector which is the vertical axis to avoid spilling liquids

The Preview window shows selected products.



5.

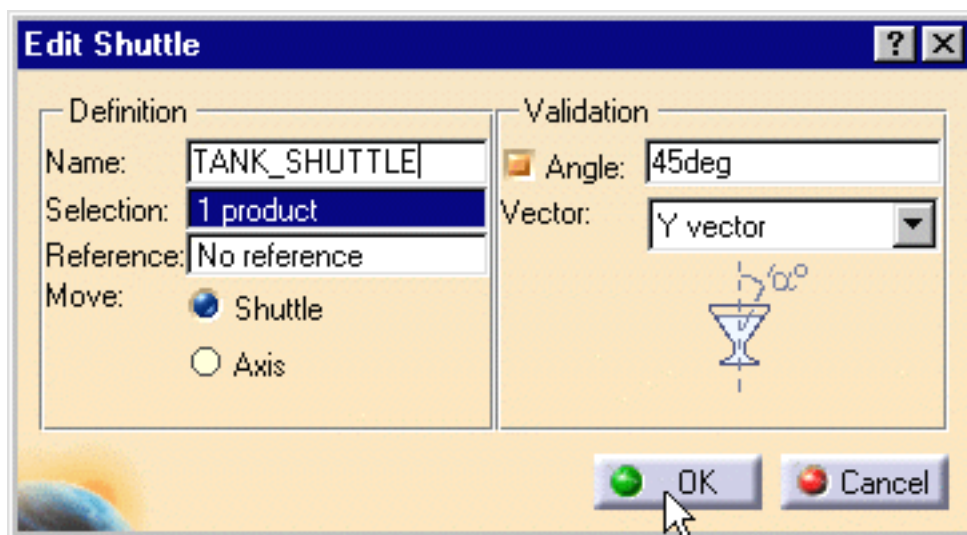
(optional)

Select products in the specification tree or in the geometry area to remove them from the shuttle.

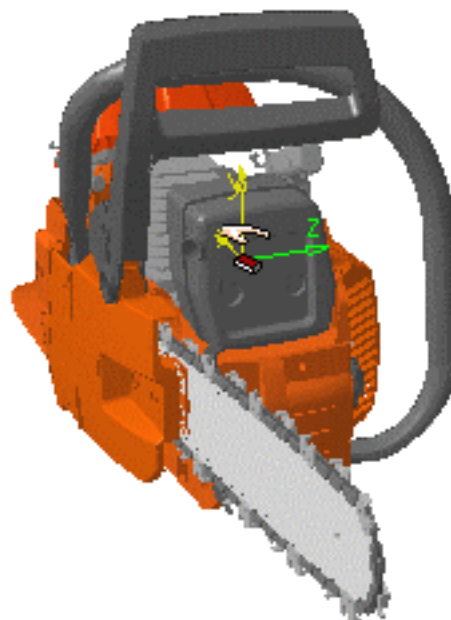
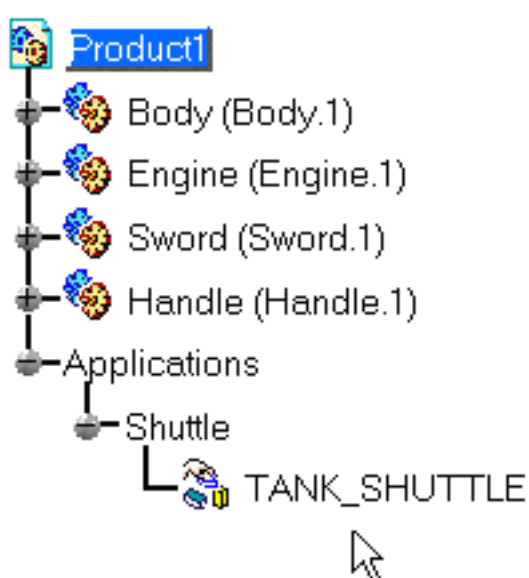
6.

(Optional)

Enter a meaningful name for the shuttle you want to create.



7. Click **OK** to end the shuttle creation.  
The shuttle is identified in the specification tree and in the geometry area.



Shuttles created in this manner are persistent and can be stored in the document. They are listed as a separate entity in the specification tree and can be selected at any time and modified. See [Editing a Shuttle](#).



If you need to deactivate the automatic display setting of the Preview window, uncheck the **Manipulating objects** option in the DMU Navigator tab via the **Tools->Options** command... For more details, please refer to [Customizing DMU Navigator Settings](#) in the DMU Navigator User's Guide.



# Shuttle



**About Shuttles:** gives background information about shuttles

**Edit the Shuttle:** in the specification tree, double-click the shuttle you defined to modify it or, right-click the shuttle and select **Shuttle.1 object->Definition** from the contextual menu. Edit the fields required in the Edit shuttle dialog box and click Ok.

**More About Manipulation Toolbar:** gives background information about manipulation tools




**Snap objects:** Create a shuttle to be snapped onto an object and click the Snap icon. Customize Snapping settings if necessary (**Tools->Options->DMU Fitting->DMU Manipulation**). Record your simulation with automatic Insert option.

# About Shuttles



A shuttle is a set of products defined explicitly by selecting products individually. Shuttles are persistent and can be stored in your document.

Shuttles are identified by name in the specification tree and by a symbol  in the geometry area.



## Shuttle creation:

4 methods are now available to create shuttles:

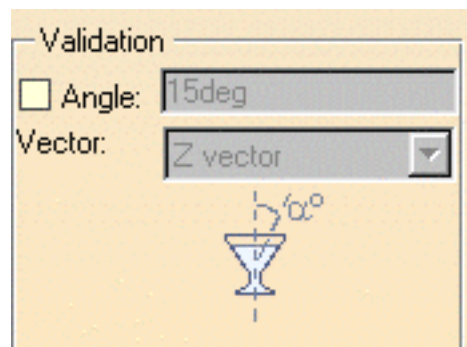
- Selecting **Insert->Shuttle...**
- Clicking the **Shuttle** icon first and select the objects afterwards.
- Selecting the objects first and activating the command after.
- Right-clicking Shuttle in the specification tree. The contextual menu displays the New Shuttle creation option. This is valid only if there is at least one shuttle defined

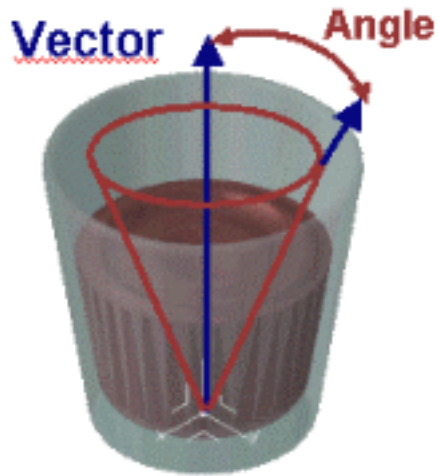
Shuttles created in this manner are persistent and can be stored in the document. They are listed as separate entities in the specification tree and can be selected at any time and modified.

## About defining angular validation for shuttles:

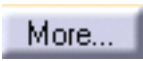
You can specify a maximum rotation angle around the absolute axis for the shuttle. It can be very useful to avoid liquids to spill from specific assemblies such as a gas tank for instance.

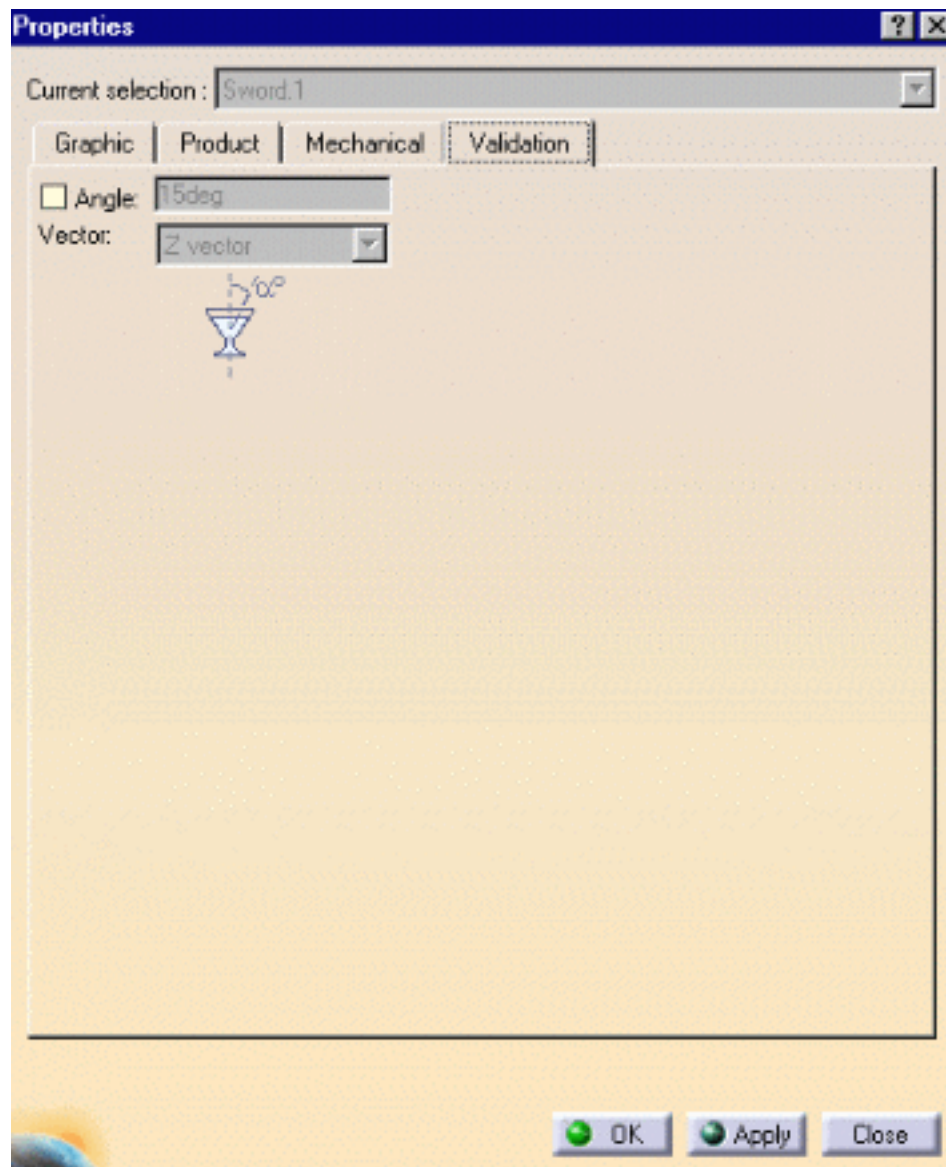
It means the shuttle motion is defined and validated with respect to the angle value defined.





You can define angular validation on products see [definition above](#)  
For this:

- Right-click the product in the specification tree
- Select **Properties** item from the contextual menu displayed
- Click  More...
- Select the **Validation** tab
- Check the Angle option. Now, enter the appropriate value and select the vector of your choice
- Click **Ok** to confirm your operation



### What are you moving?

By default, the Shuttle option is set, which means that when you move a shuttle the axis and geometry move together. If you choose to set the Axis option instead, only the shuttle axis moves. Please refer to [Moving the Shuttle](#)



## Shuttle axis behavior:

You created a shuttle clicking the icon first ( action->object)

- the shuttle is empty, the shuttle axis is at the absolute origin (document).
- you add an object to this shuttle, by default the shuttle axis **moves** to the bounding box center of this object.

You created a shuttle selecting objects first (either in the geometry area or in the specification tree) (object->action)

- the shuttle comprises one or more objects, the shuttle axis is positioned at the bounding box center of the objects selected.
- you add an object to this shuttle, the shuttle axis **does not move**.

## Snapping Shuttle on Geometry:

Use the **Ctrl key** and the **3D compass manipulation handle** to drag the shuttle onto the required object. Then, release the mouse button to drop both **the shuttle and compass** onto the object.



You can use at any time the Manipulation toolbar. For more details, please refer to [Manipulation Toolbar](#). and More about Manipulation toolbar.



# More About Manipulation Toolbar



## Shuttle-based manipulation toolbar



## Track-based manipulation toolbar

DMU Fitting Simulator lets you validate the dismounting of assemblies by moving objects and generating simulations. Therefore, this capacity to manipulate 3D objects with a 2D mouse comes to the fore.

Thus, DMU Fitting Simulator provides powerful tools available from the manipulation toolbar displayed every time you are working in edit shuttle mode, Explode, Track and Simulation commands (if based on one or various shuttle(s).)

Let's detail it more carefully



**Preview:** click this icon to view the selected and manipulated objects in a dedicated window



**Reframe:** Fits manipulated objects into the space available in the preview window



### Target:

In some cases, you need to reposition the shuttle on the geometry (namely on the symmetry axis, on rotation centers...) . When you do not work in design mode, you do not have the original geometry. various options are available

- use Target in one shot
- use Target in two shots
- use CTRL key + 3D Compass

Please refer to [Moving the Shuttle](#), [Using Target Capability](#), and [About Shuttles](#)



**Editor:** lets you edit 3D Compass position (displays the Compass manipulation dialog box)  
See [Moving the Shuttle](#)



**Invert:** inverts the 3D compass orientation and the shuttle attached



**Reset:** swaps to the initial shuttle position



**Snap:** lets you define the snapping axis, to snap the shuttle onto an object in approach phase (in simulation)


Please refer to [Snapping objects](#) and [Customizing DMU Manipulators](#)





**Attach/Detach:** lets you move the 3D compass independently from the shuttle.




# Editing a Shuttle

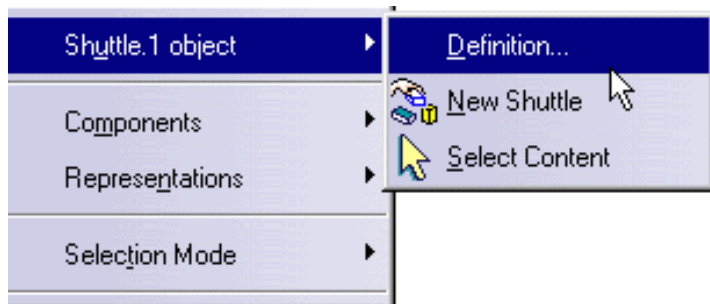
 A shuttle is a set of products defined explicitly by selecting products individually. Shuttles are persistent and can be stored in the document.

 This task shows how to edit a shuttle. This procedure can be performed either before the shuttle creation or after.

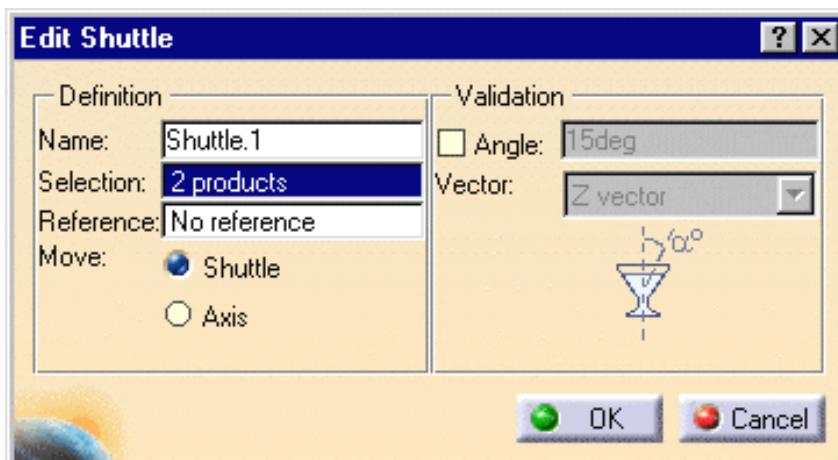
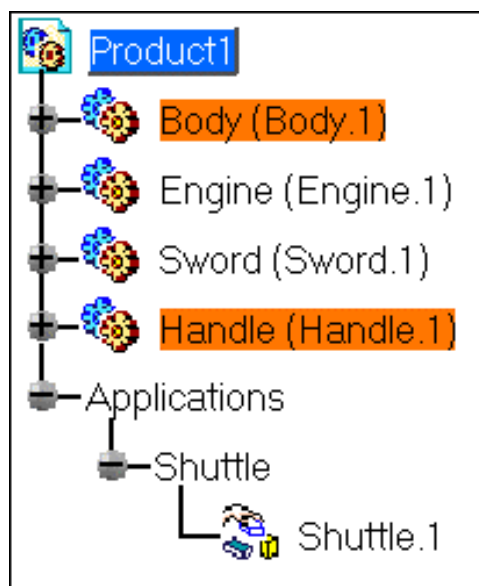
 Open the [CHAINSAW.CATProduct](#) document, then select **Digital Mockup -> DMU Fitting** from the **Start** menu.

You defined at least one shuttle. For more details please refer to [Defining a Shuttle](#).

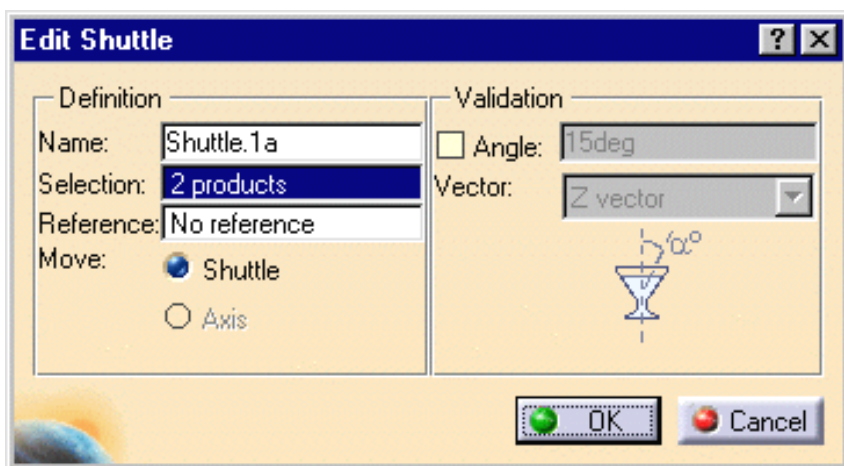
-  1. In the specification tree, double-click the shuttle you defined to modify it or, right-click the shuttle and select **Shuttle.1 object->Definition** from the contextual menu.




The Edit Shuttle dialog box appears and displays the contents of the shuttle you selected. Objects included in the shuttle are highlighted in the specification tree and in the geometry area.




2. Modify shuttle contents as desired. For this:
3. (optional)  
Select objects in the specification tree or in the geometry area to remove them or select new ones to be added.
4. (optional)  
Rename the shuttle. For instance, enter Shuttle.1a in the Name field.



 Your operation is automatically taken into account. The selection field from the Edit dialog box is automatically updated as well as the Preview window. The new selection is highlighted in the specification tree and in the geometry area.

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

 By default, the Shuttle option is set, which means that when you move a shuttle the axis and geometry move together. If you choose to set the Axis option instead, only the shuttle axis moves. Please refer to [Moving the Shuttle](#)

 You can use at any time the Manipulation toolbar. For more details, please refer to [Manipulation Toolbar](#).



# Snapping Objects in Approach Phase



This scenario aims at showing you how a snapping position can be useful in a mounting operation.

We are going to focus here on the nuts.

The nut initial position is interesting and useful this is why we are going to create a shuttle on this object



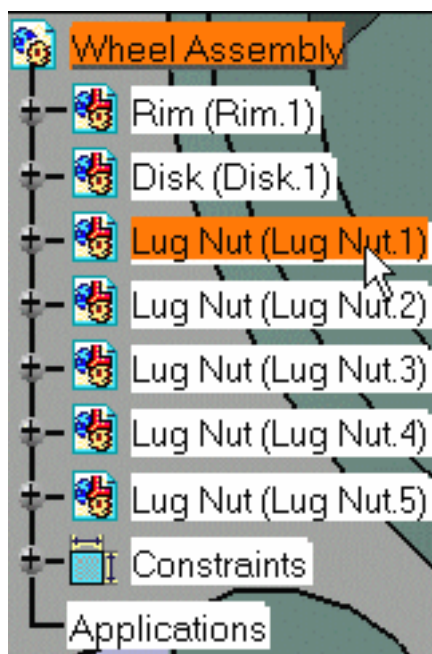
This task shows how to reposition with precision objects on the geometry using the snap capability.




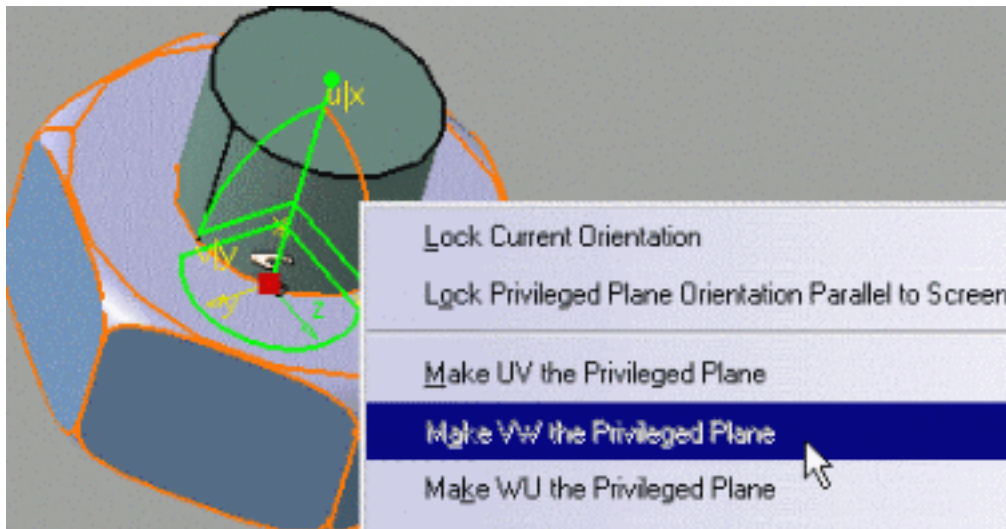
Open the [SNAPPING.CATProduct](#) document.



1. Use the **View->Zoom In** command and drag (left mouse button) to zoom in progressively until you are satisfied.
2. Select Nut 1 either in the geometry area or in the specification tree.



3. Click the Shuttle icon  from the DMU Simulation toolbar.
4. Make sure you work with the privileged plane VW.  
Point to the compass and right-click to display the contextual menu. Select the command **Make VW the Privileged Plane**.

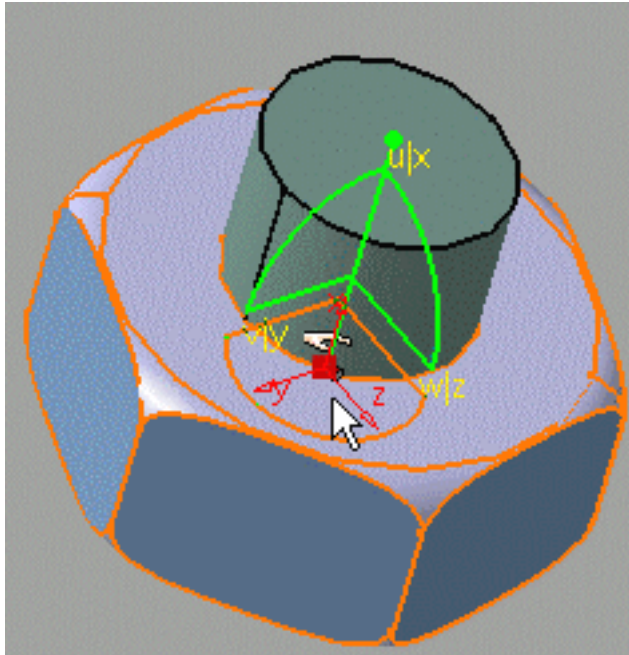


5.



Click the Snap icon from the Manipulation toolbar.

A red axis system appears on the shuttle. The current shuttle axis position is recorded.



6. Click Ok to end the shuttle creation.

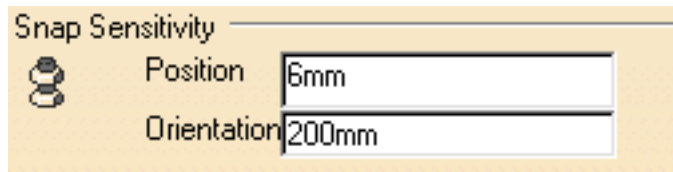
*Now you might need to customize the snapping settings*

7. Use **Tools->Options->DMU Fitting->DMU Manipulation** to customize snapping settings

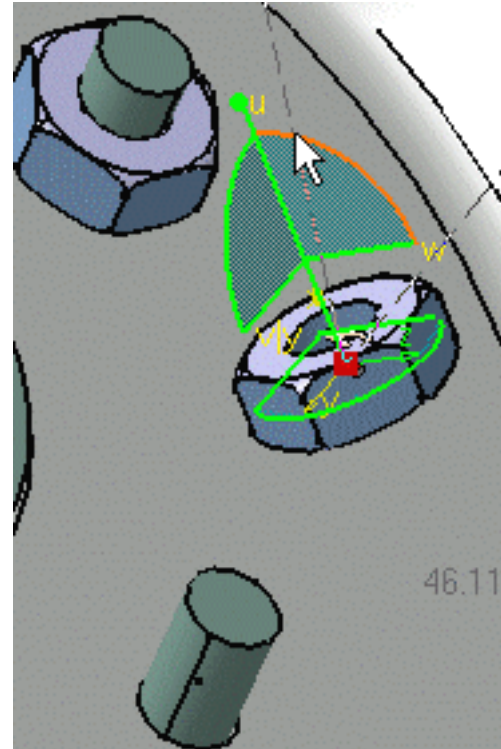
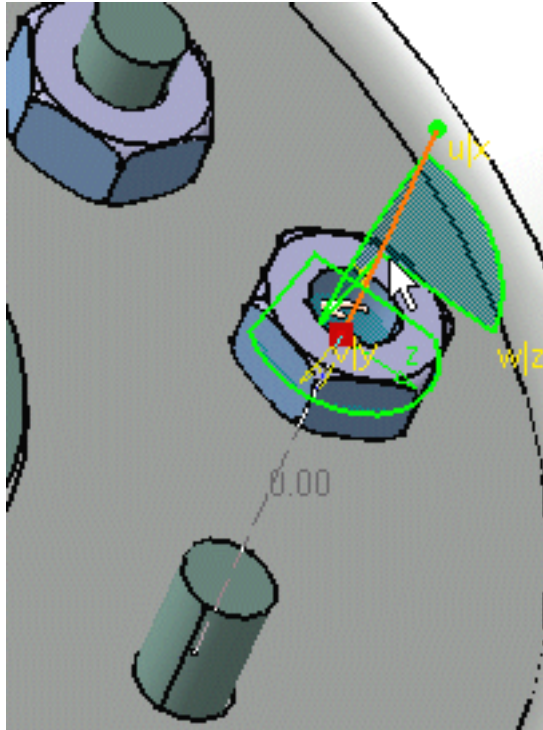
Set values for the:

- Position: 6mm (keep the default value)
- Orientation: 200mm (20mm is the default value)

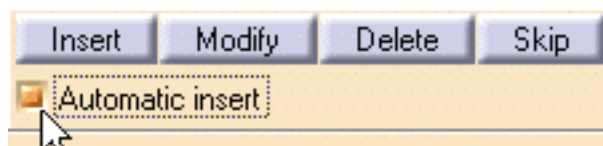




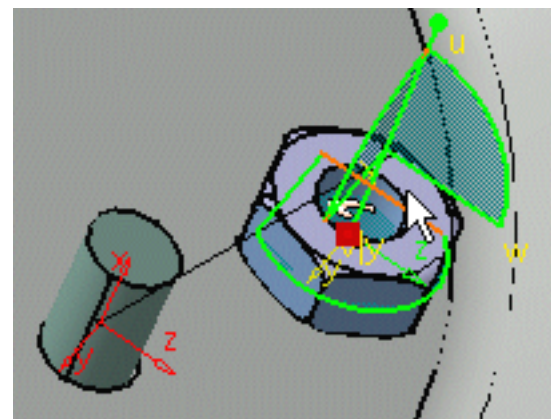
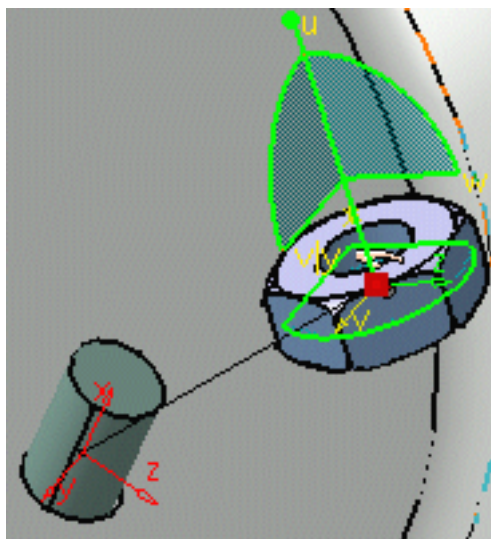
8. Click Ok to take your modifications into account
9. Drag and drop the Simulation icon on shuttle.1 in the specification tree, please refer to [Dragging and Dropping Icons Onto Objects](#) in the *Infrastructure User's Guide*
10. Move the shuttle as shown below:



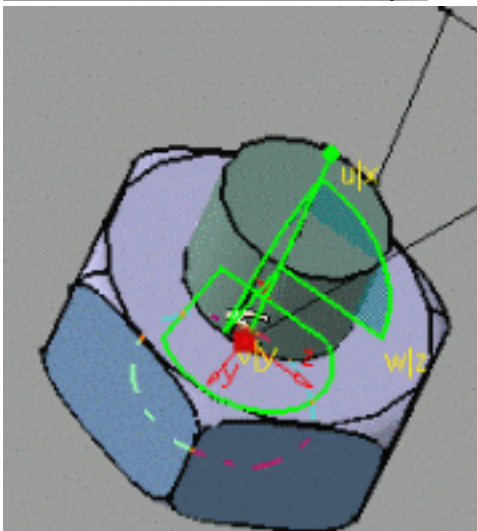
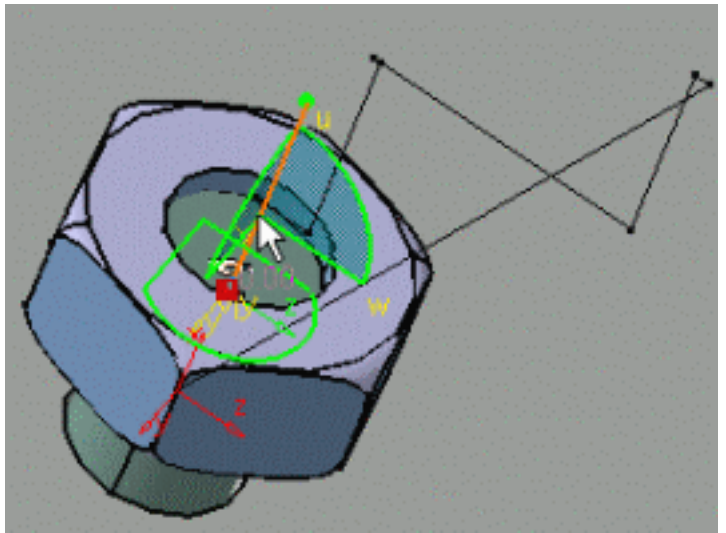
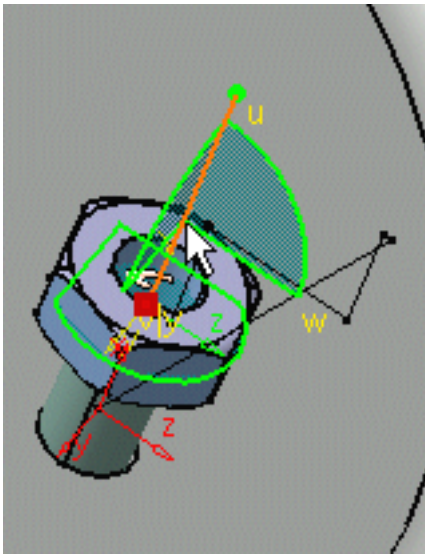
11. Set the Automatic insert option in the Edit Simulation dialog box.



12. Start recording your simulation moving the shuttle towards the disk.








The nut is snapped as shown below



Use the smooth command  to get rid of the unnecessary positions.



# Mono-Shuttle Fitting Simulation



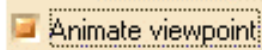
**Record a Simulation:** click the Simulation icon or select **Insert->Simulation...** from the menu bar. Move the shuttle as often as necessary, clicking Insert to record shots then press **OK** to confirm your operation.



**Record a Simulation Automatically:** click the Simulation icon or select **Insert->Simulation...** from the menu bar then set the auto insert option in the Edit Simulation dialog box. Click the Auto Insert icon from the Manipulation toolbar and choose the required settings. When done click **OK** and start recording your simulation.



**Using the Smooth Command:** select the simulation in the specification tree and click the smooth icon. Click **Ok**.



**Animate a Viewpoint:** replay your simulation now with the Animate viewpoint option set. The simulation is replayed with the viewpoint changes recorded in the simulation.



**Generate a Replay:** select **Tools->Simulation->Generate Replay** Select the track or the simulation you want to generate a replay from. Click **OK** to create a replay object.

**Generate an Animation File:** provides information on how to generate an AVI file with the standard tools



**Replay a Simulation:** Select **Tools->Simulation->Replay** or double-click the Replay in the specification tree. Replay the recorded simulation using the VCR buttons.

# Recording a Simulation



This task shows how to record a Simulation, in other words a scenario on mono-shuttle fitting simulation.

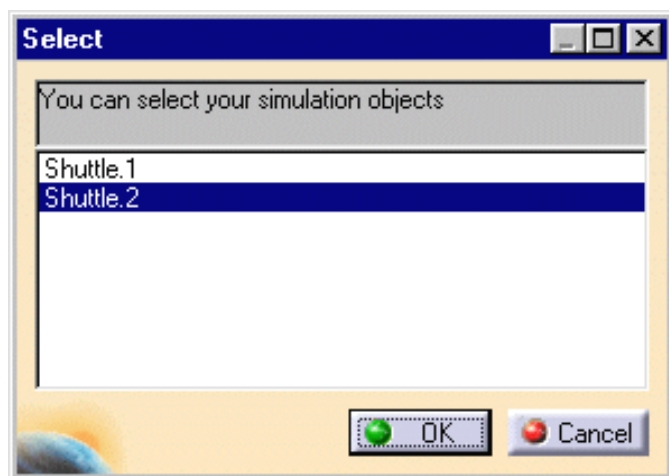


Open the [RECORD\\_SIMULATION.CATProduct](#) document.



1. Select **Insert->Simulation...** from the menu bar.

The Select dialog box is displayed and lets you select the shuttle you want to use in your Simulation. The Edit Simulation dialog box and Preview window appear.



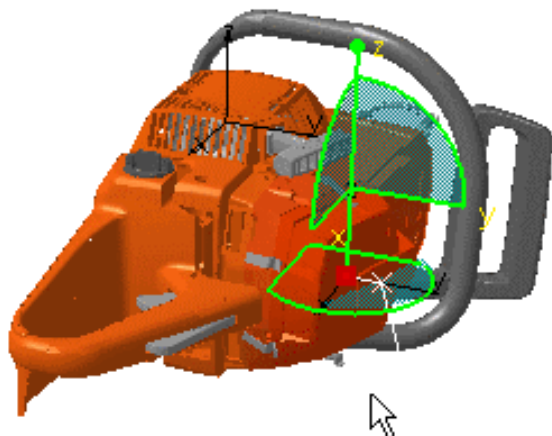
Note that you can also choose to select the shuttle first, and the simulation icon afterwards. In this case, the select dialog box is not displayed.

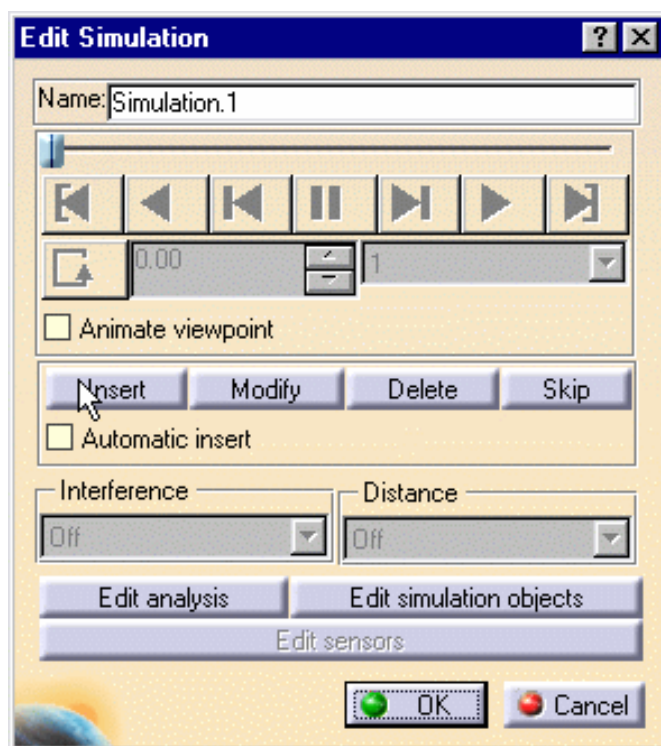
2. Click the **Insert** switch and record the starting shot.



Note that you can change the automatic display setting of the Preview window via the **Tools->Options** command...

- **Insert** means that you record and insert shots inside the scenario.  
The initial location of the shuttle is automatically recorded as a starting shot.  
If you need to reposition the shuttle, please delete the first position or modify it.



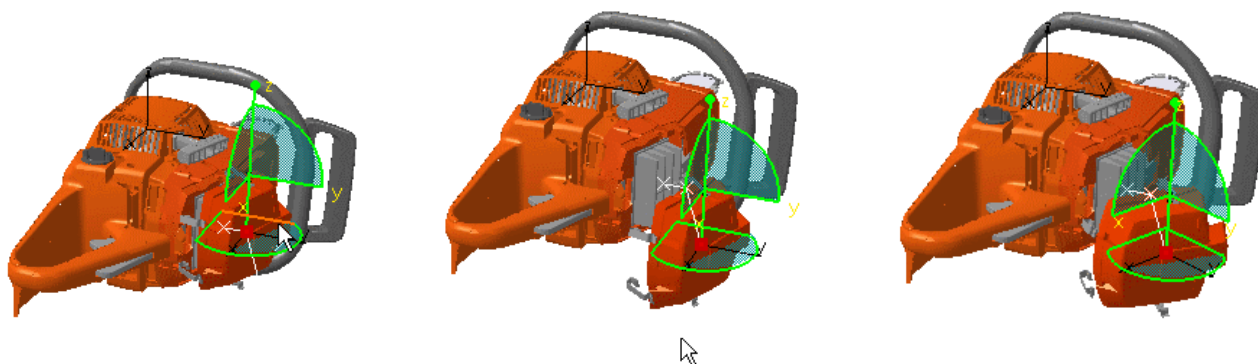


*i* For more detailed information about the sensors, please refer to *DMU Kinematics Simulator User's Guide, Using Sensors*

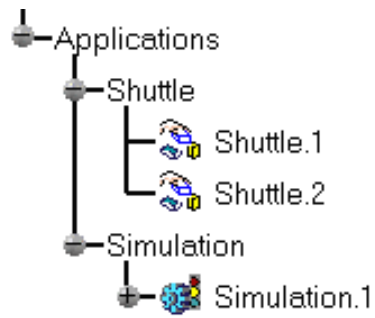
[Edit sensors](#)

3. Move the shuttle to the desired location via the manipulator.
4. Click the **Insert** switch and record the desired shot.
5. Move the shuttle as often as necessary, clicking Insert to record shots.

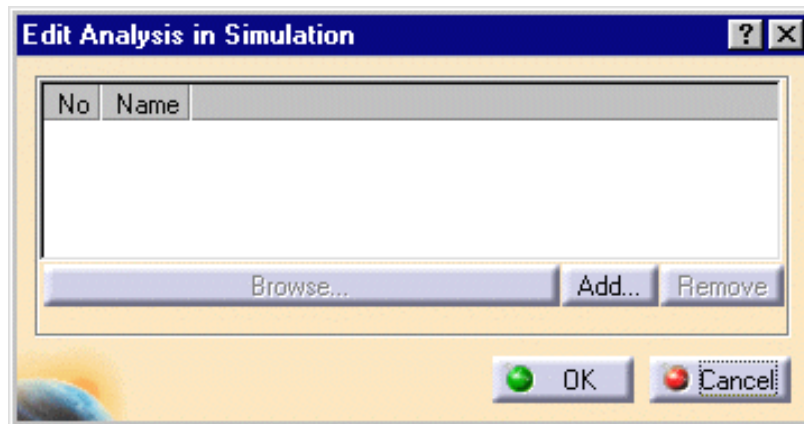
*i* You can activate the automatic insert at any time. For more detailed information, please refer to [Recording a Simulation Automatically](#)



6. Select the appropriate interpolation, for instance 0.04. You can modify or choose the automatic insert at any time.
7. Click **OK** to confirm your operation.  
The Simulation is identified in the specification tree.



At any time, you can press the Edit Analysis button from the Edit Simulation dialog box: It is empty because you have not defined interferences.



The Edit Simulation objects button lets you add simulation objects such as cameras...



You recorded a simulation based on shuttles. Here, you selected the shuttle.2. Keep in mind that editing shuttles impacts the simulation.



# Recording a Simulation Automatically



This task shows you how to record a Simulation automatically using the **automatic insert** functionality, the **automatic clash detection** in Mouse moving mode ( possibility to record distance and angle steps)

You can manipulate the shuttle with the mouse or the space ball/space mouse



Open the [TANK\\_SHUTTLE.CATProduct](#) document.

Make sure the settings are correct:

For more information, please see [Customizing DMU Manipulators](#)

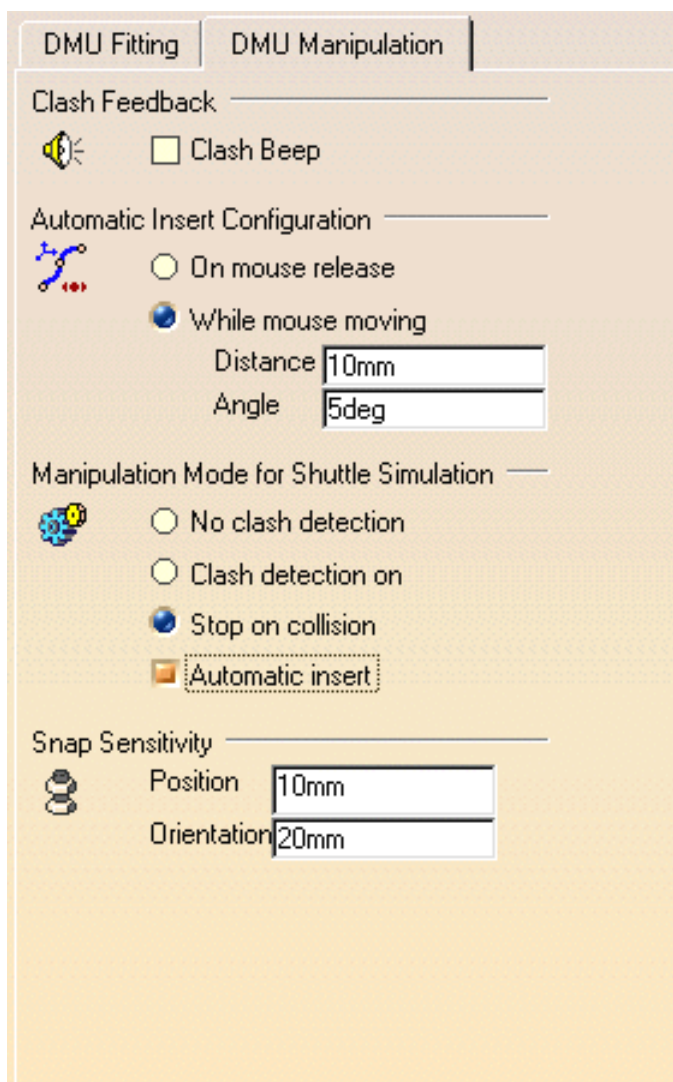


1. Select **Tools->Options ...** from the menu bar

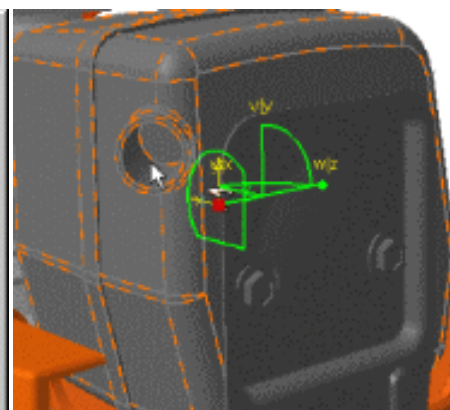
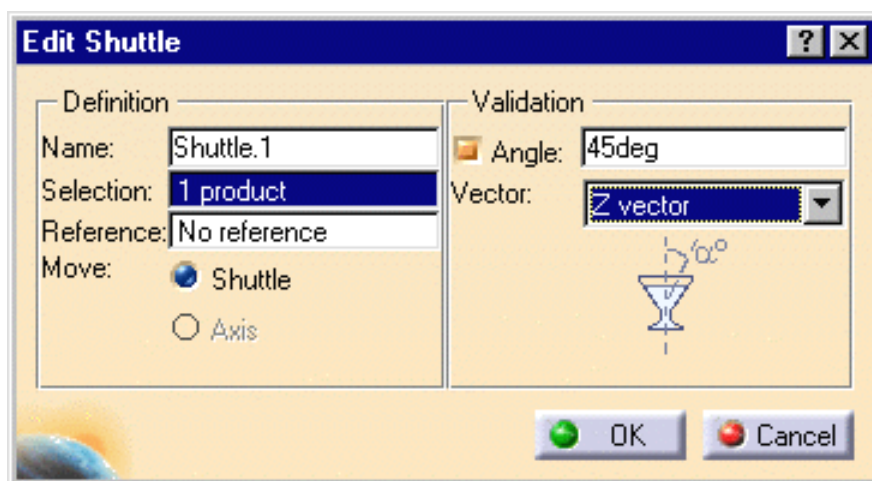
The Options dialog box appears.

- Click the DMU Fitting item
- Click the DMU Manipulation tab

2. In the Automatic Insert Configuration field, check the "While mouse moving" option
3. In the Manipulation Mode for Shuttle simulation, check the "Stop on collision" mode as well as the "Automatic insert" option



4. Click Ok to confirm your operation. Your new settings are taken into account.
5. Double-click the Shuttle.1 (defined on the tank) in the specification tree:
  - the angle option is set to 45 deg
  - close the edit shuttle dialog box
 The shuttle motion is defined with respect to the rotation angle defined



6. Select Shuttle.1 in the specification tree.


7.

Select **Insert->Simulation**

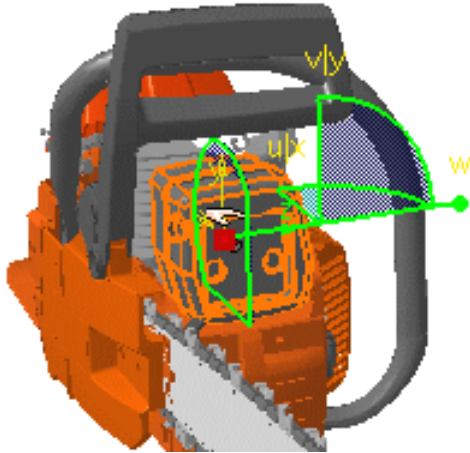




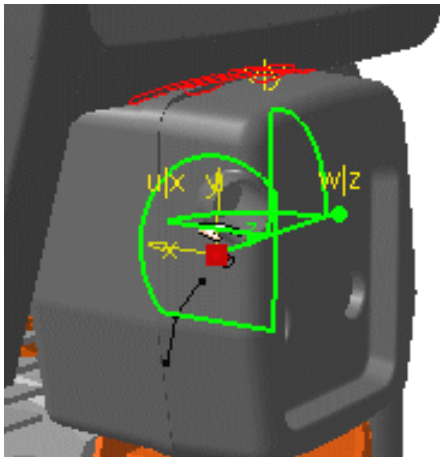
The Automatic insert option is checked in the simulation dialog box and the Stop on collision mode is

activated ()

8. Start recording your simulation using the mouse left-button.

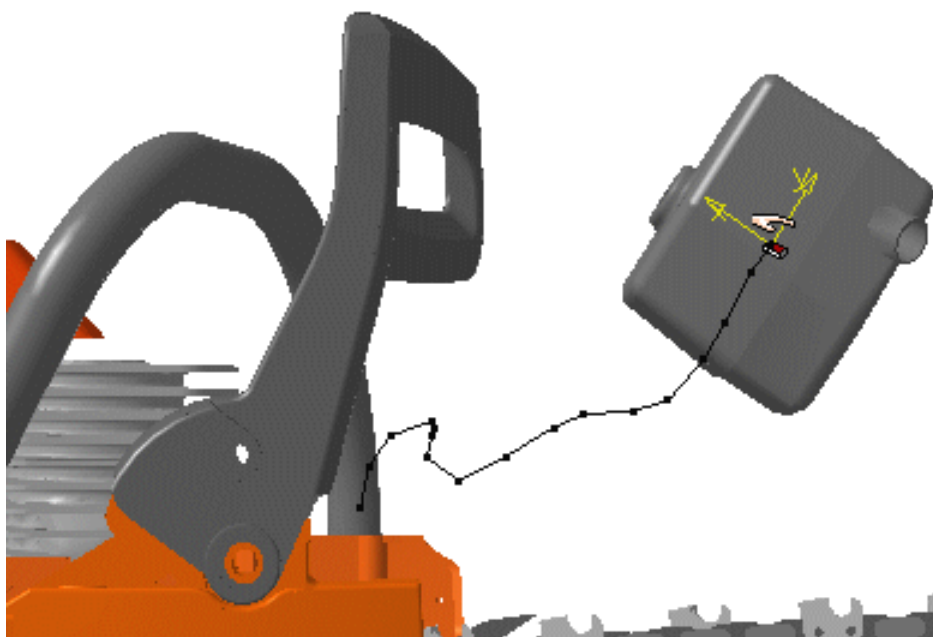


9. Move the shuttle as often as necessary, the shots are recorded automatically
10. The simulation stops when the shuttle is in collision, the product is highlighted. This position is not recorded, the stop on collision lets you validate the simulation.

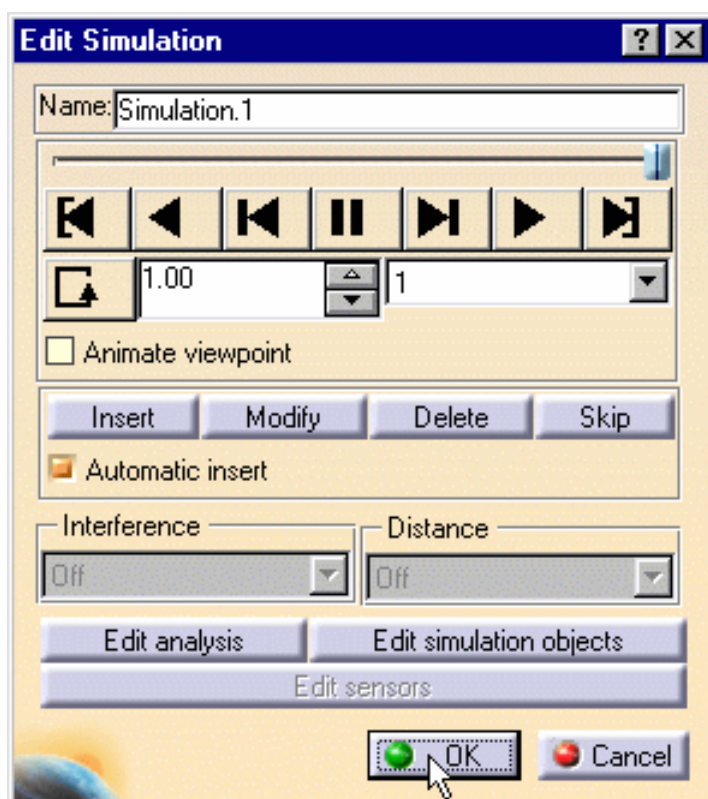


This is what you obtain:





11. Click Ok in the Edit Simulation dialog box.



The recording precision depends on the manipulation speed. It means the precision is approximate as the recording limit is based on the upper distance selected.



# Using the Smooth Command



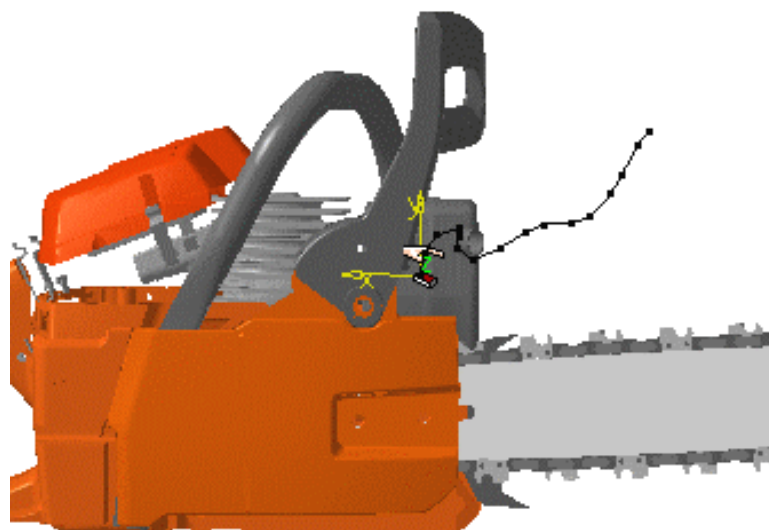
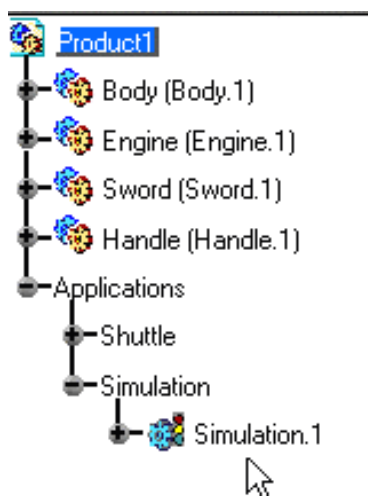
This task shows how to smooth positions in a simulation. You recorded a simulation using the space ball and automatic insert capability. You used the automatic clash detection. The smooth command lets you obtain a path without collisions as it removes the useless positions.



Open the [SMOOTH.CATProduct](#) document.



1. In the specification tree, select **Simulation.1**.

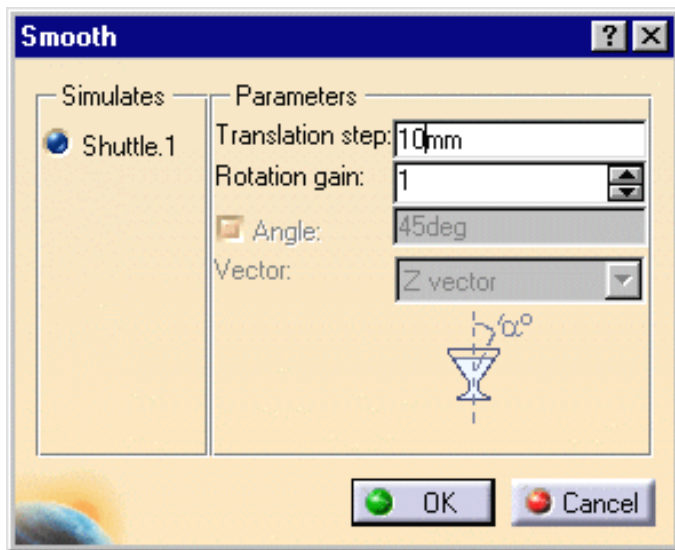


- 2.



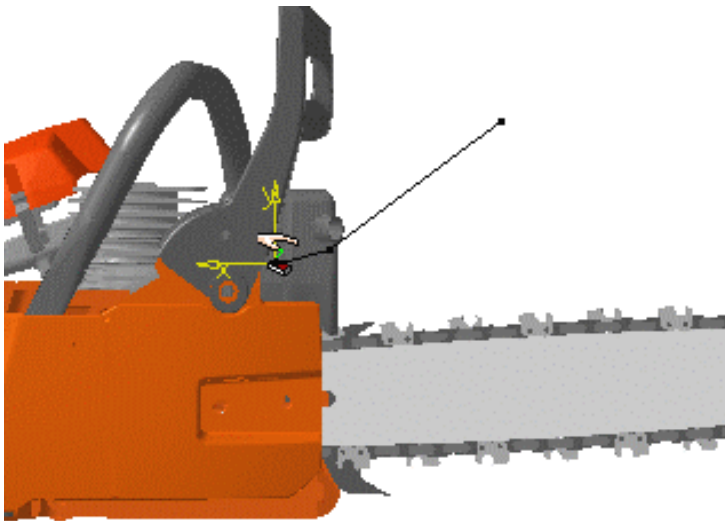
Click the Smooth icon

The Smooth dialog box is displayed  
Enter 10 mm in the Translation step field.

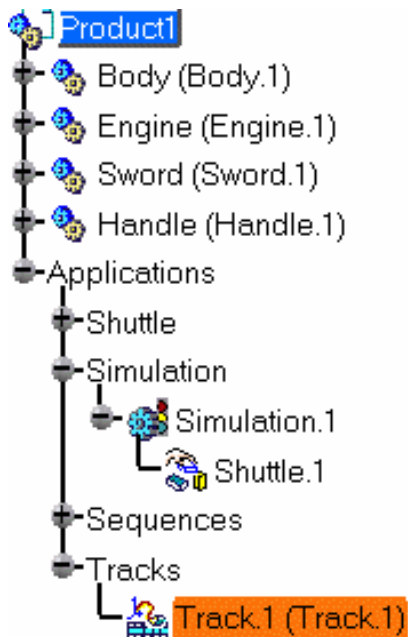


3. Click OK.

This is what you obtain:



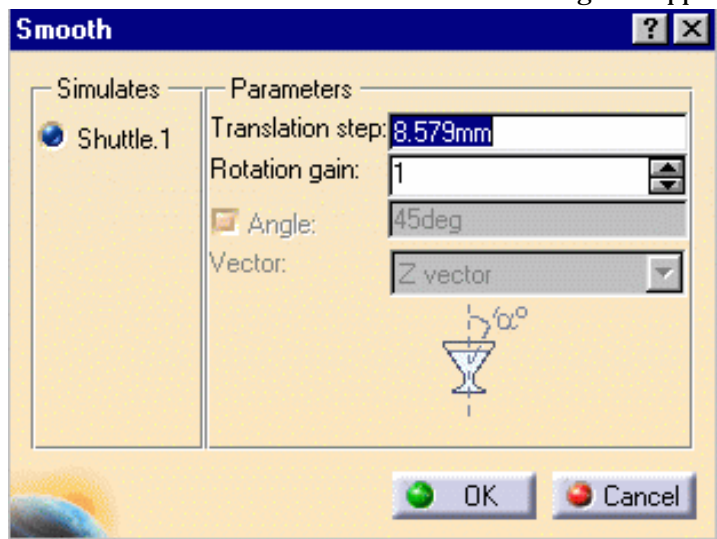
4. Now close your [SMOOTH.CATProduct](#) without saving the modifications.  
Open the [SMOOTH\\_TRACK.CATProduct](#) document (a track is defined)
5. Select Track.1 either in the specification tree or in the geometry area



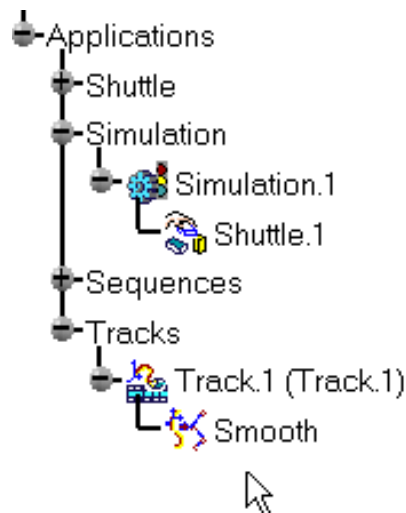
6.



Click the Smooth icon. The Smooth dialog box appears. Keep the parameters values and click Ok



The track is smoothed



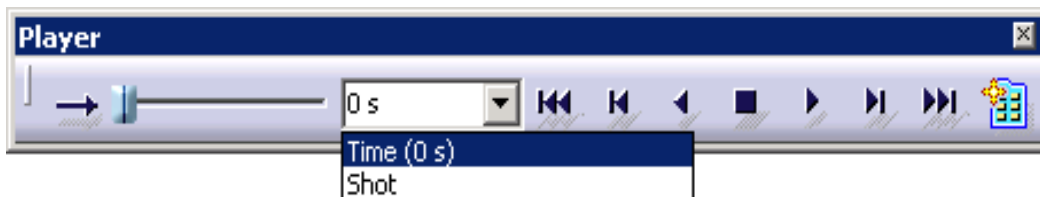
Now you are going to play your smoothed track:

7.

Click the Play a Simulation icon  and select Track.1 in the specification tree

The Player is active. New parameters are available to replay your track with respect to:

- time (in seconds),
- shot parameter or,
- path finder or smooth specifications if previously defined (this is the case in our example)




To avoid interpolation consistency problems, (fixed by the auto controlled step option in the previous releases), you can now replay your smoothed track using the pathfinder parameter in the player drop-down list:

- selecting pathfinder parameter in the drop-down list swaps to the right interpolation (spatial-based interpolation, smooth specification)



Of course, you can choose to keep the default parameter (time in seconds), but in this case the consistency between the smooth specification and the way it is played are not guaranteed

8.

Click  to play your smoothed track.



# Animating a Viewpoint in a Simulation



This task shows how to animate viewpoints in a simulation.



Open the [RECORD\\_SIMULATION.CATProduct](#) document.



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

2.



Click the **Simulation** icon .

The Edit Simulation dialog box and Preview window appear.

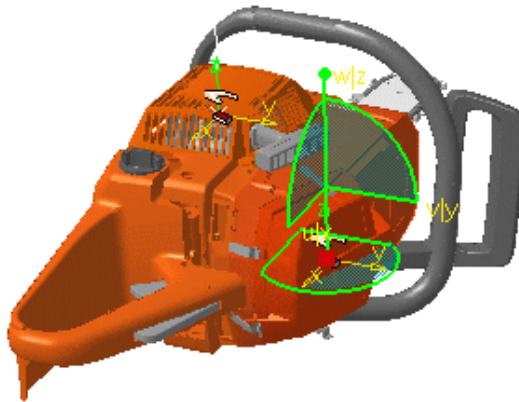
3. Click the **Insert** switch and record the shots.

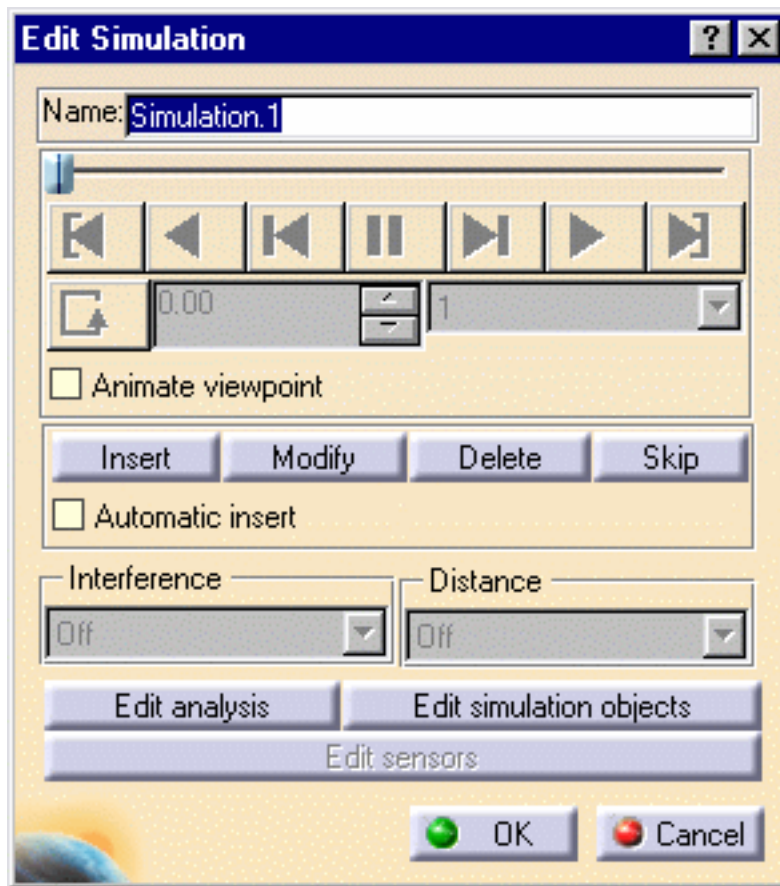


Remember the initial position is automatically recorded. If you need to reposition the shuttle, please delete the first position or modify it.

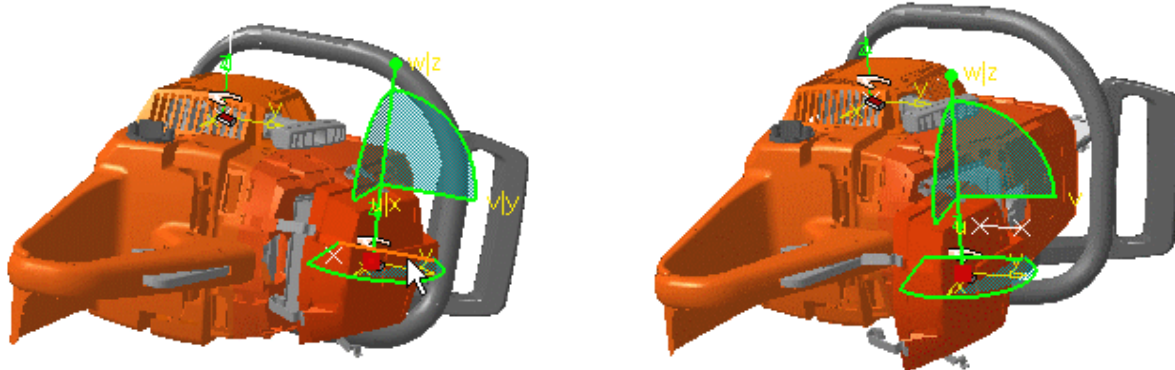


- **Insert** means that you record and insert positions with the current viewpoints inside the scenario.

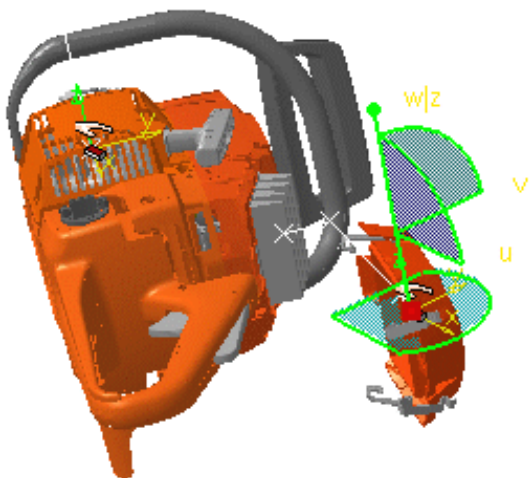




4. Move the shuttle to the desired location via the manipulator.
5. Click the **Insert** switch and record the desired position.
6. Move the shuttle as often as necessary, clicking Insert switch to record shots. Change the viewpoint as shown below:







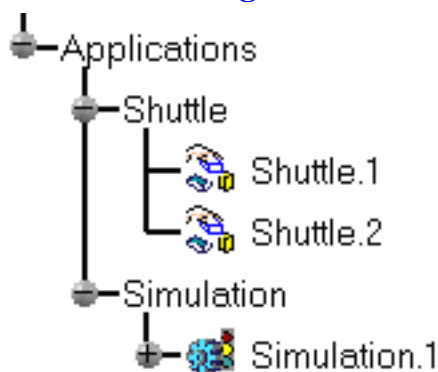
7. Select the appropriate interpolation. You can modify or choose the automatic insert at any time.

The Simulation is identified in the specification tree

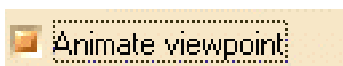


**Automatic Insert:**

You can activate the automatic insert at any time. For more detailed information, please refer to [Recording a Simulation Automatically](#)



8. Replay your simulation now with the Animate viewpoint option set. The simulation is replayed with the viewpoint changes.



Note that you can activate the Animate viewpoint option at anytime either before recording the simulation or after.

9. Click OK to save your simulation. You are now ready to create a replay or an animation. This is done by compiling your simulation.



You recorded a simulation based on shuttles. Here, you selected the shuttle.2. Keep in mind that editing shuttles impacts the simulation.



# Generating a Replay



This task shows how to generate a replay object a recorded Simulation to generate a replay or an animation



You recorded a Simulation. Please refer to [Recording a Simulation](#)  
Open the [COMPILE\\_SIMULATION.CATProduct](#) document.



1. Select Simulation.1 in the specification tree.

2.

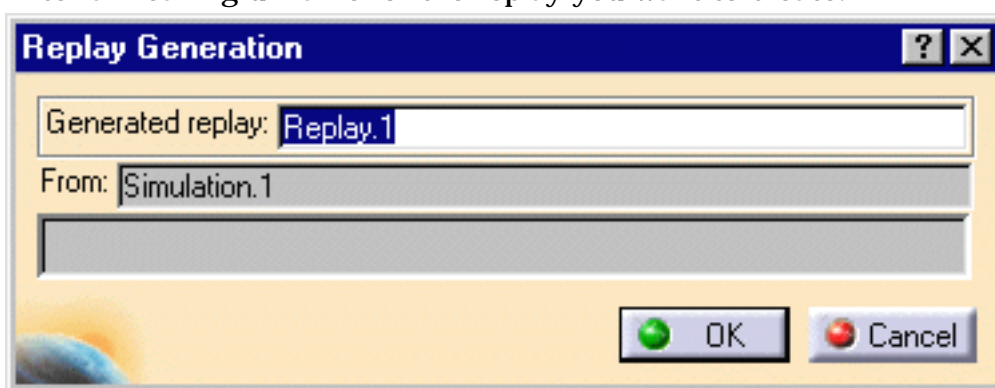


Select **Tools->Simulation->Generate Replay**.

The Replay Generation dialog box is displayed.

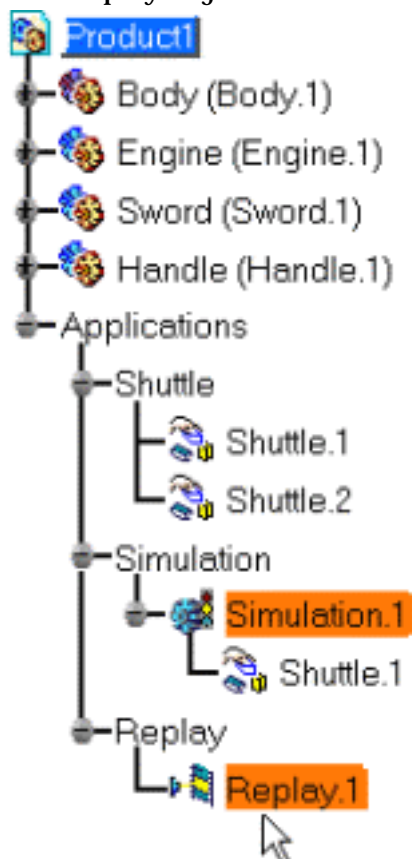
(Optional)

Enter a meaningful name for the replay you want to create.



3. Click Ok to confirm your operation

The Replay object is identified in the specification tree





# Replaying



This task shows how to rerun the Replay object resulting from the Compile Simulation.



Open the [REPLAY\\_SIMULATION.CATProduct](#) document.



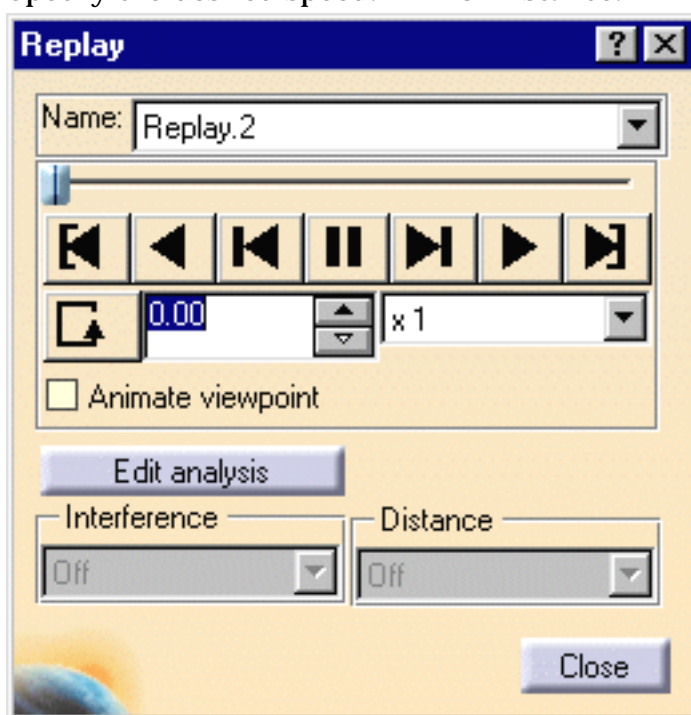
1.

Select **Tools->Simulation->Replay**  or double-click the Replay.2 in the specification tree.

The Replay dialog box is displayed:

2.

Specify the desired speed: **x 1** for instance.



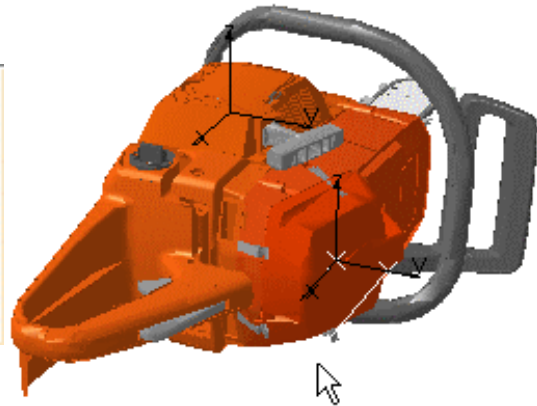
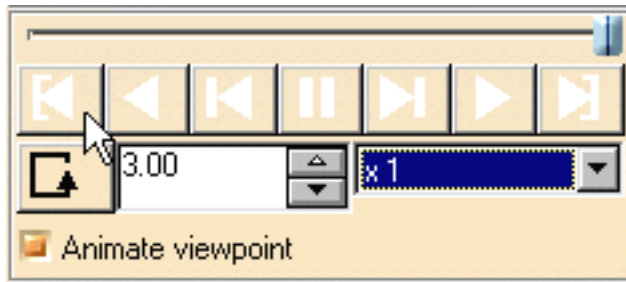
Check the Animate viewpoint option if you want to take into account the viewpoints recorded during the simulation.



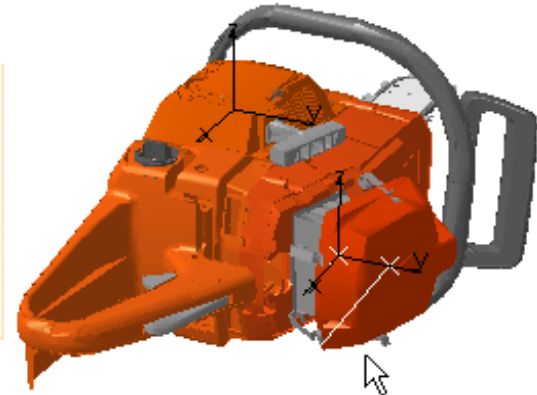
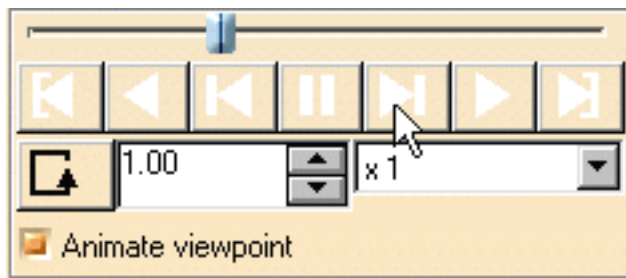
Of course assigning a high number of interpolations amounts to replaying the scenario at a very low speed.

3.

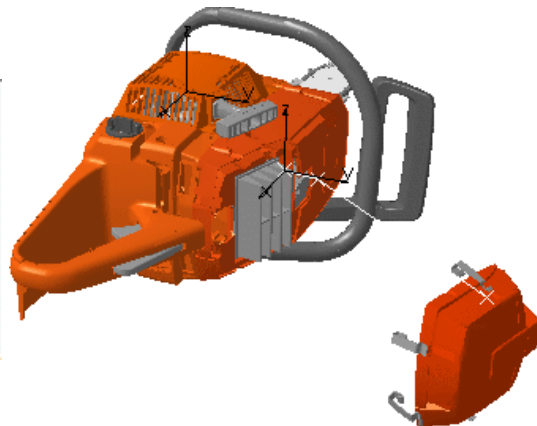
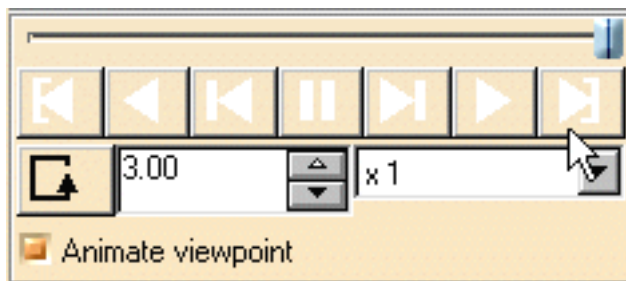
Go back to the starting shot.  
For instance, use the VCR buttons.



4. Step forward.



5. Modify the speed at any time.  
6. Step forward again



7. Replay the scenario back to the starting shot and at a 1 interpolation number between recorded shots.



You can choose one of the loop modes to re-run the animation in a continuous way (either in one direction only or in one direction then the other).



# Target and Snap Capabilities



**About the Target Capability:** Use the Ctrl key and 3D compass to drag the shuttle onto the required object.


**Use Smart Target:** Double-click the target icon, select the first object to be moved, select the geometry and release the mouse button, repeat the operation with a second object a constraint is created. Select as many objects as necessary

**Use Smart Snap:** Double-click the snap icon, select the geometries you need to detect, click Ok, record your track. The geometries are snapped interactively

**Move the Shuttle:** double-click the shuttle in the specification tree. The 3D compass lets you move the shuttle as desired.

# About Target Capability

## About Snapping objects (shuttle, products in track) on Geometry using the Smart target


While creating or editing a track, a simulation or a shuttle, it is welcome to be able to catch an entity very precisely. The smart target command  lets you do this.

Beside, it can be useful to cumulate several targets. Indeed, using a historical target can help to put some elements in a specific position.

In some cases, you need to reposition objects on the geometry (namely on the symmetry axis..)


For example you need to reposition the screw in the hole.

If you do not work in design mode, which means you do not have the original geometry, three options are available to reposition the object (shuttle or product in track)

1. Use the Target icon  from the manipulation toolbar in one shot.

- Double-click the shuttle or the track in the specification tree or in the geometry area. The manipulation toolbar is displayed.
- Click the target icon.
- The object to be moved (shuttle, product, camera included in a shuttle or a track ) is selected
- Point the cursor on the geometry (belonging to the to- be- moved object)
- Click the mouse button
- Select a second geometric element (belonging to the receiving object)
- Click the mouse button to validate your selection

A coincidence constraint is created, the object is snapped to the desired location.

2. Use the Target icon  from the manipulation toolbar in several shots

- Double-click the shuttle or the track in the specification tree or in the geometry area. The manipulation toolbar is displayed.
- Click the target icon to activate the **smart target**
- The first object to be moved is selected (object attached to the shuttle, product or camera included in a shuttle or a track )
- Select a first geometric element (on the to-be-moved element)

- Click the mouse button
- Select a second geometric element (on the receiving element)
- Click the mouse button
- Select a third geometric element (on the to-be-moved element)
- Click the mouse button
- Select a fourth geometric element (on the receiving element)

Two coincidence constraints are created.

### 3. Use CTRL key and the 3D Compass

Now when you work in design mode (the original geometry is available)  
The Target capability recognizes the geometric features (edges and faces)  
**The easiest way to snap the shuttle on geometry is to use**

- CTRL key and the 3D Compass

For more information about Target , please refer to [Using Smart Target](#) .



This task shows how to reposition with precision the shuttle on the geometry using CTRL key and the 3D Compass in design mode.



You defined a [shuttle](#). Please refer to [Defining a Shuttle](#)  
Open the [plane.CATProduct](#) document.

1. Select **Edit->Representations->Design Mode...** to snap the shuttle on geometry (axis, points, intersection points)

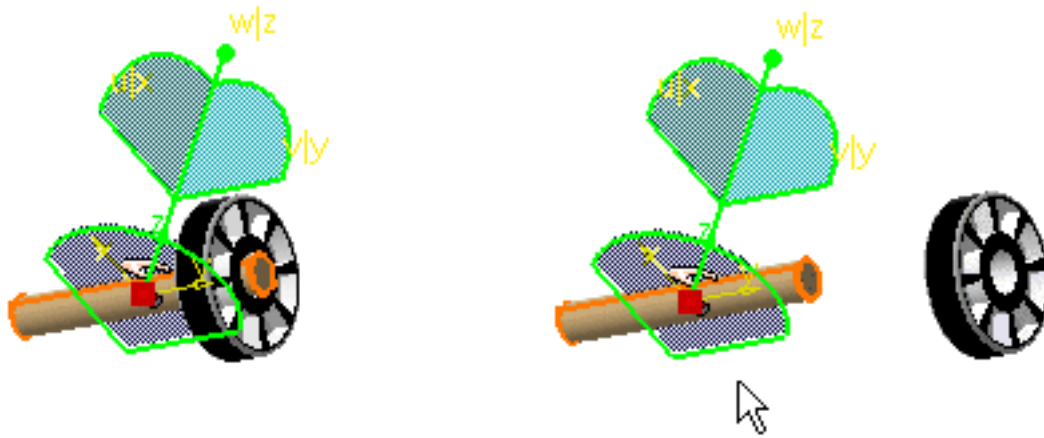


2. Double-click the shuttle in the specification tree (**shuttle.1**).

The Edit Shuttle dialog box, the Preview window and the Manipulation Toolbar appear.

3. Move the shuttle (**shuttle.1**) as shown below

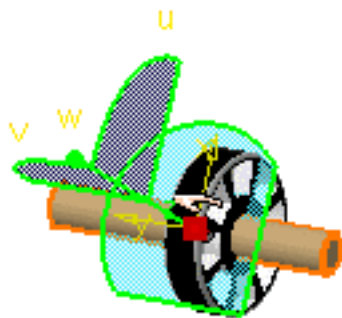




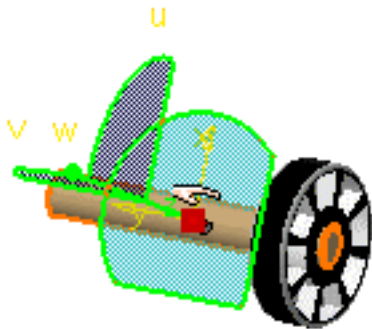
*Now you need to snap the shuttle on geometry (in this case to the wheel axis) you are going to reposition the shuttle in its initial position.*

4. Use the **Ctrl** key and **3D compass manipulation handle** to drag the shuttle onto the required object.
5. Release the mouse button to drop both **the shuttle and compass** onto the object.

The shuttle is snapped on geometry.



6. Reposition the shuttle if needed.




# Using Smart Target



## About Smart Target:



The Smart target  command consists in creating a series of constraints on-the-fly, thereby reducing the degrees of freedom of the components

While creating or editing a track, a simulation or a shuttle, it is welcome to be able to catch an entity very precisely. It can be useful to cumulate several targets. Using a historical target (keeping track of cumulative constraints) can help to put some elements in a specific position.

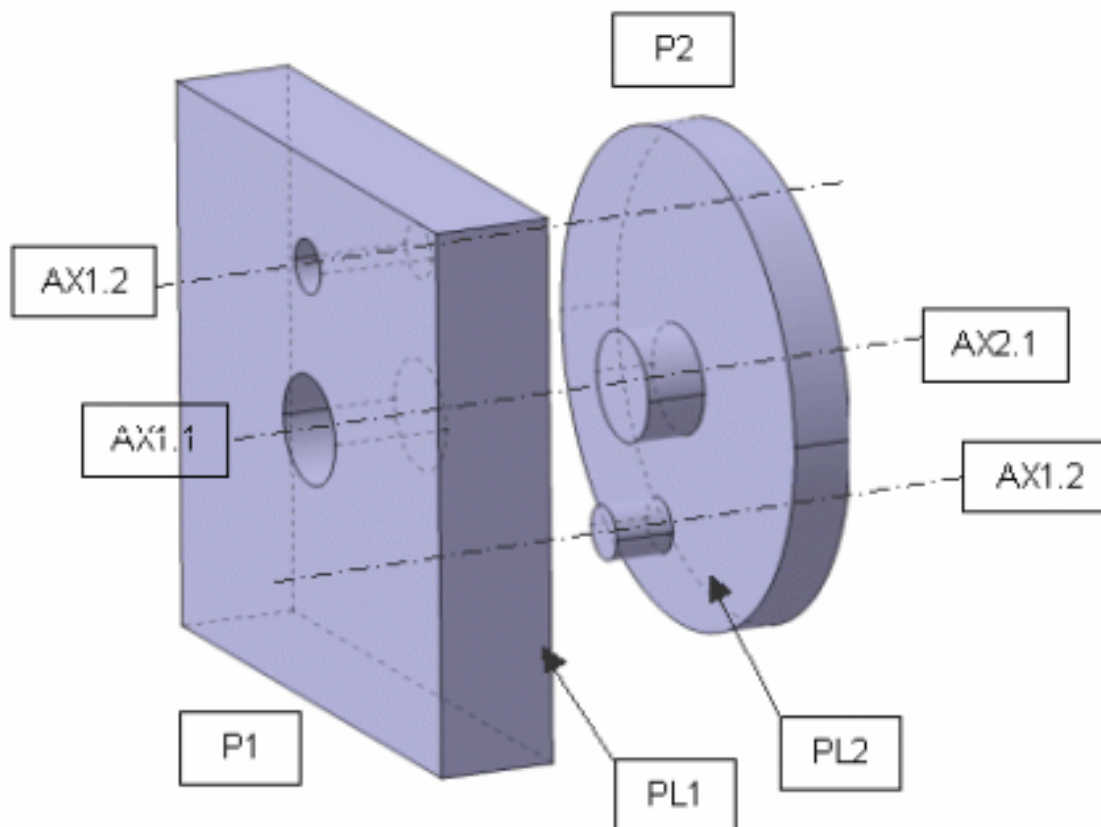
For instance, the scenario below aims at showing you the interest of this new capability:

- you want to move P2 in P1, using the smart target, we can create a first coincidence constraint between the two planes referred to as PL2 and PL1
- then, we add a second coincidence constraint between the two axis AX2.1 and AX1.1
- finally we add a third coincidence constraint between the two axis AX2.2 and AX1.2

### **What is the advantage?**

Using the previous target functionality would have resulted in:

- only P2 translation by the third coincidence constraint
- the coincidence constraint 2 being lost



This task show you to use the Smart Target in a track recording scenario.



Open the [SMART\\_TARGET.CATProduct](#) document.



1. Select Disk.1 either in the specification tree or in the geometry area

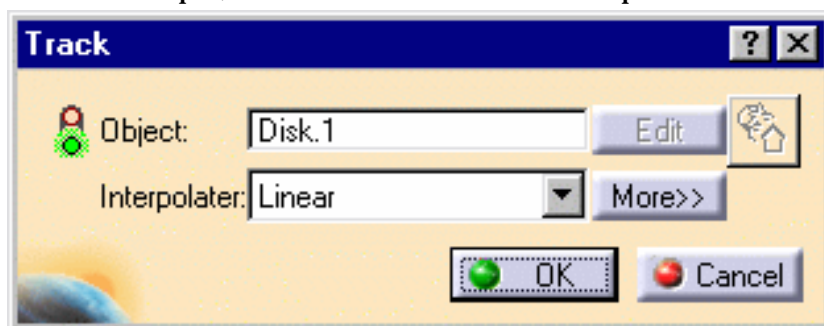
2.



Click the Track icon from the DMU Simulation toolbar.

You are ready to record your track: select the product to move

In our example, select Disk.1 either in the specification or in the geometry.



The player becomes active and the Manipulation toolbar appears

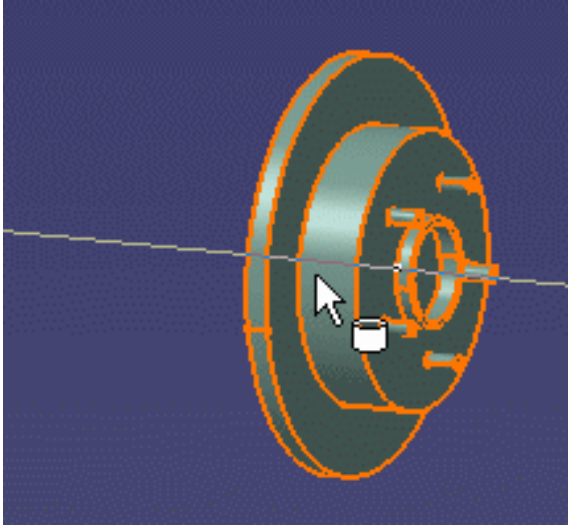
3.



Click the Target icon from the Manipulation toolbar to activate the smart target

The Disk is selected and thus highlighted in the geometry area

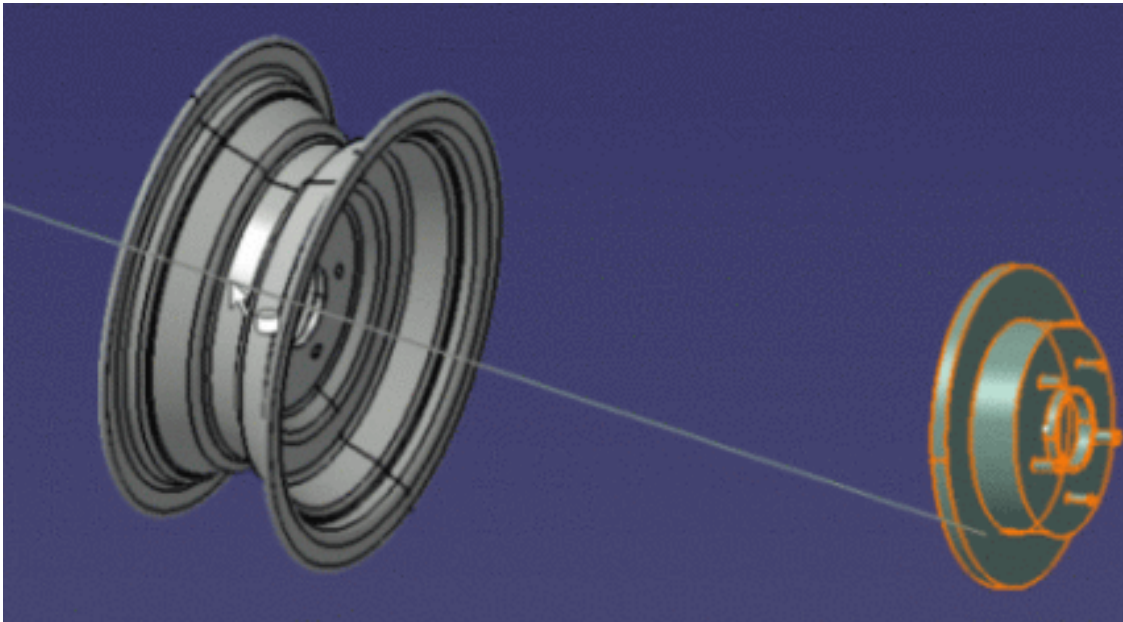
4. Select a geometric element (a point, line, plane or axis system) on the same to-be-moved component, e.g. a circle of which the center is the center of the entire Disk cylinder in the sample product.



5. Double-click Insert  in the Recorder to activate the automatic insert mode.

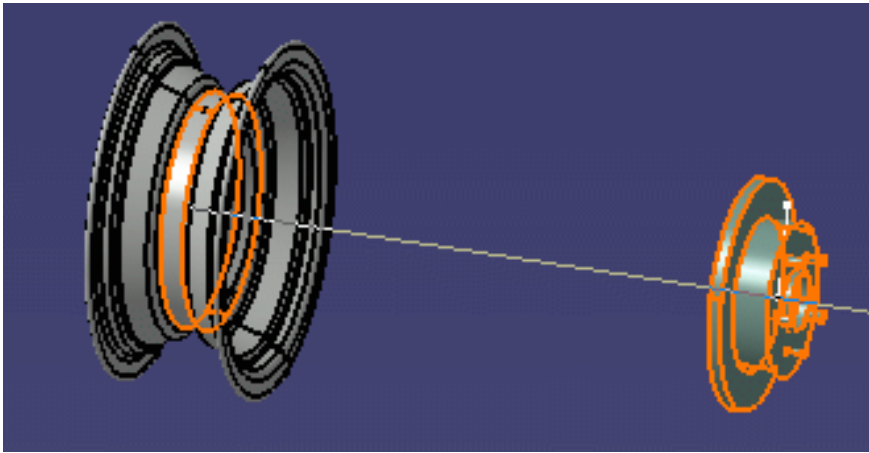
Please refer to [Track Editor and Recorder](#) and [Recording a Track](#)

6. Select the Rim cylinder axis as shown below



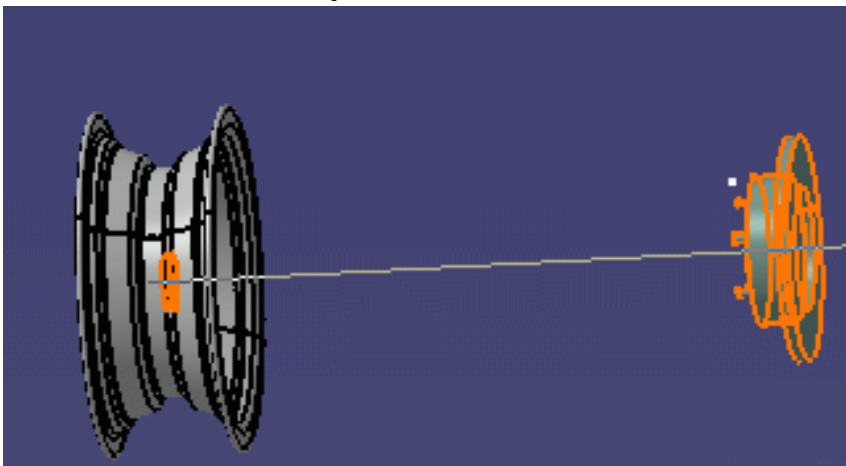
The first coincidence constraint is created. The Disk cylinder is displaced in accordance with the first constraint.

You can continue directly to the creation of the second constraint.

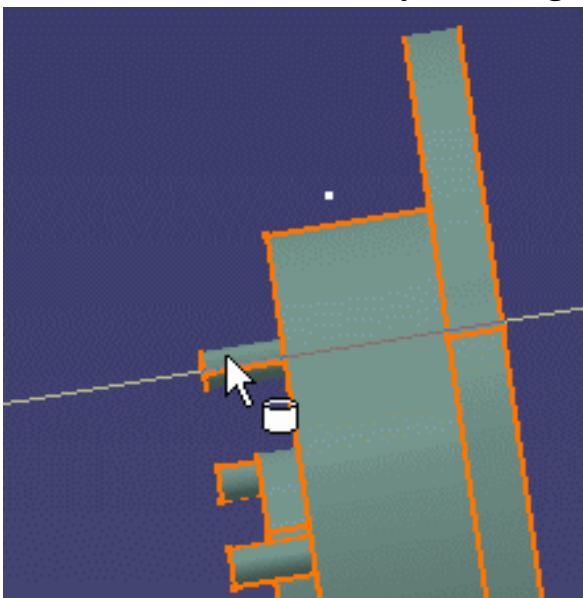


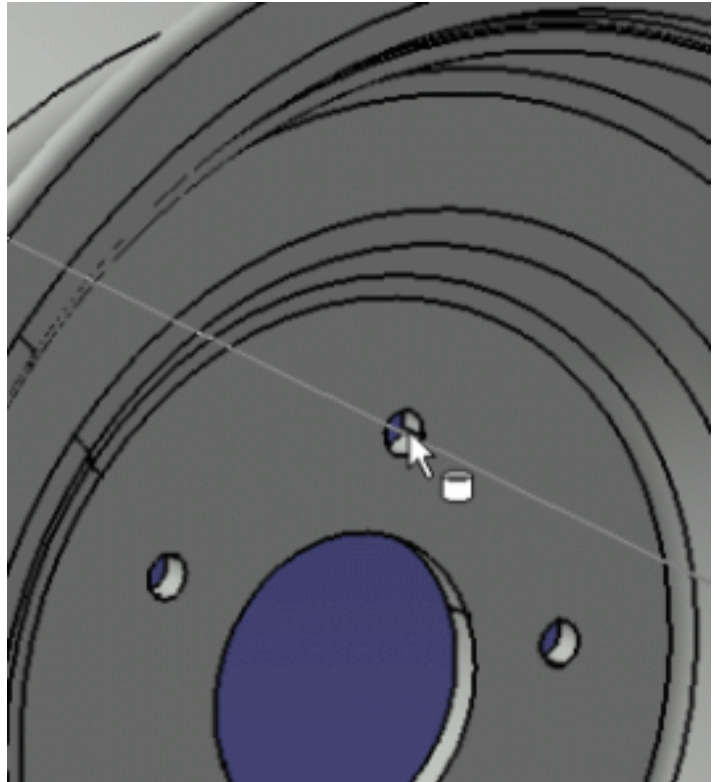
7. Now click the Invert button to position the Disk:

The disk is automatically inverted



8. Select a geometric element on the to-be-moved component, e.g. the axe passing through one of the nuts of the Disk cylinder. (left picture)
9. Select a geometric element on the receiving component, e.g. the axe passing through one of the nut holes of the Rim cylinder. (right picture)





The second constraint is created. The Disk cylinder is displaced in accordance with the second constraint.

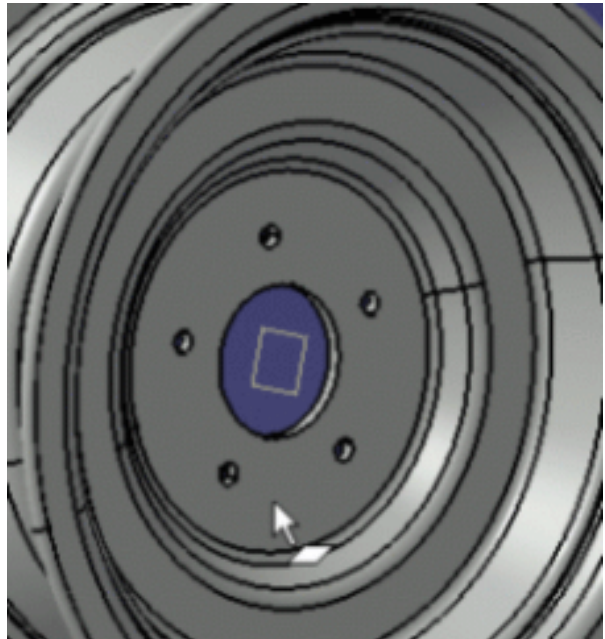
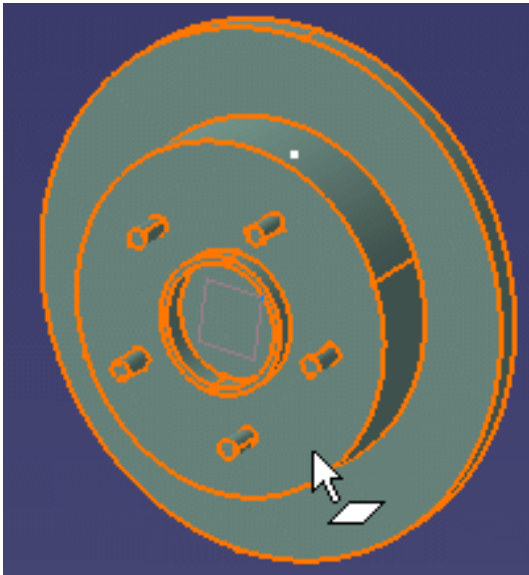
You can continue directly to the creation of the third constraint.

**10.** Select the plane of the Disk cylinder containing the nuts.

**11.** Select the plane of the Rim cylinder containing the nut holes.

The third constraint is created.

The two components have been placed together as a function of the three constraints.

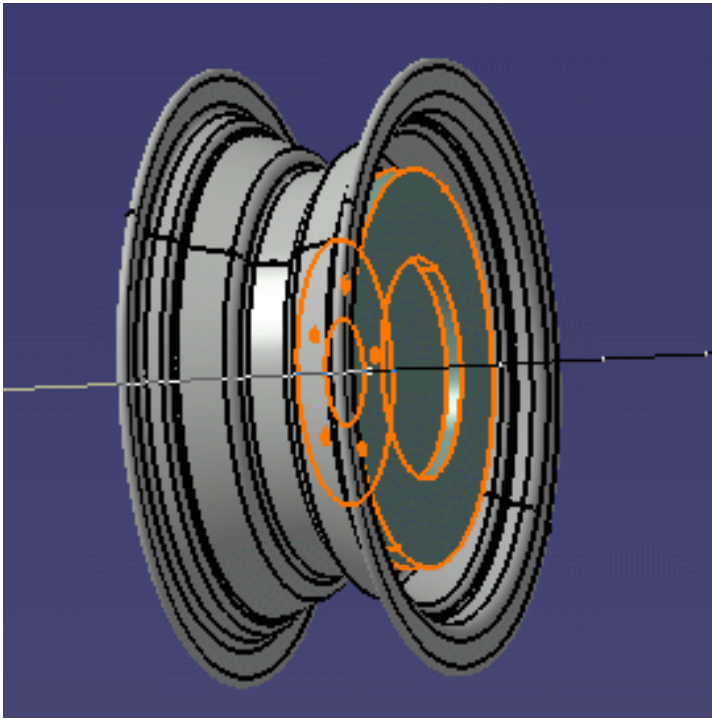


This is what you obtain

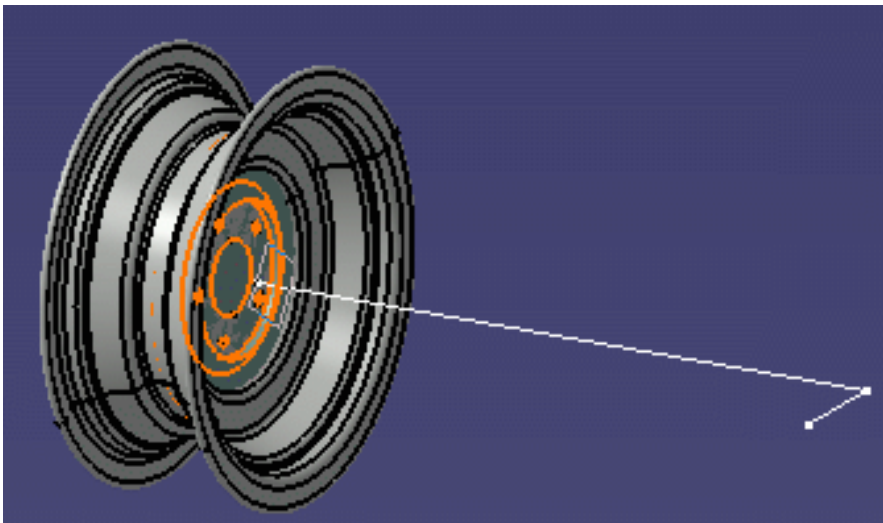
The third constraint is created.

The two components have been placed together as a function of the three constraints.






The track is created, you can replay it



Click the Target icon  to exit the command



## What about the geometric features recognized when using the Smart Target capability?


Geometric Features		Design mode	Visualization mode
cylinder axis		Y	Y
surfaces 		Y	Y
		Y	Y
edge :	extremities	Y	Y
	Any point (CTRL key)		
camera (eye)		Y	Y
V4 axis system		Y	Y
construction lines construction axis		Y	Y



The table below describes the smart target behavior and its advantages with respect to the previous one

Selectable elements		Previous Target	Smart Target
Any point (using the Mouse + CTRL key)		Yes	Yes
Line	  	Yes	Yes
V4 Axis System		Yes	Yes



Plane		Yes	Yes
Shuttle Axis (using the Mouse + SHIFT key)		Yes	Yes
<b>Selection order</b>			
<b>1.</b> Object to be moved is already selected	shuttle or product in track		shuttle or product in track
<b>2.</b> first geometrical element	could belong to the active element, or to the global environment in order to snap the compass base plane		must belong to the object to be moved (the active element)
<b>3.</b> second geometrical element	on the same or on a different component		on a different object
Design/visualization modes	Yes	except for V4 axis systems, construction axis and construction lines	Yes
Repetitive Mode	No		Yes



Two constraints can be opposite and thus become a limitation for the use of smart target.



# Using Smart Snap




## About Smart Snap:



The Smart snap command consists in creating a series of constraints on-the-fly, thereby reducing the degrees of freedom of the components

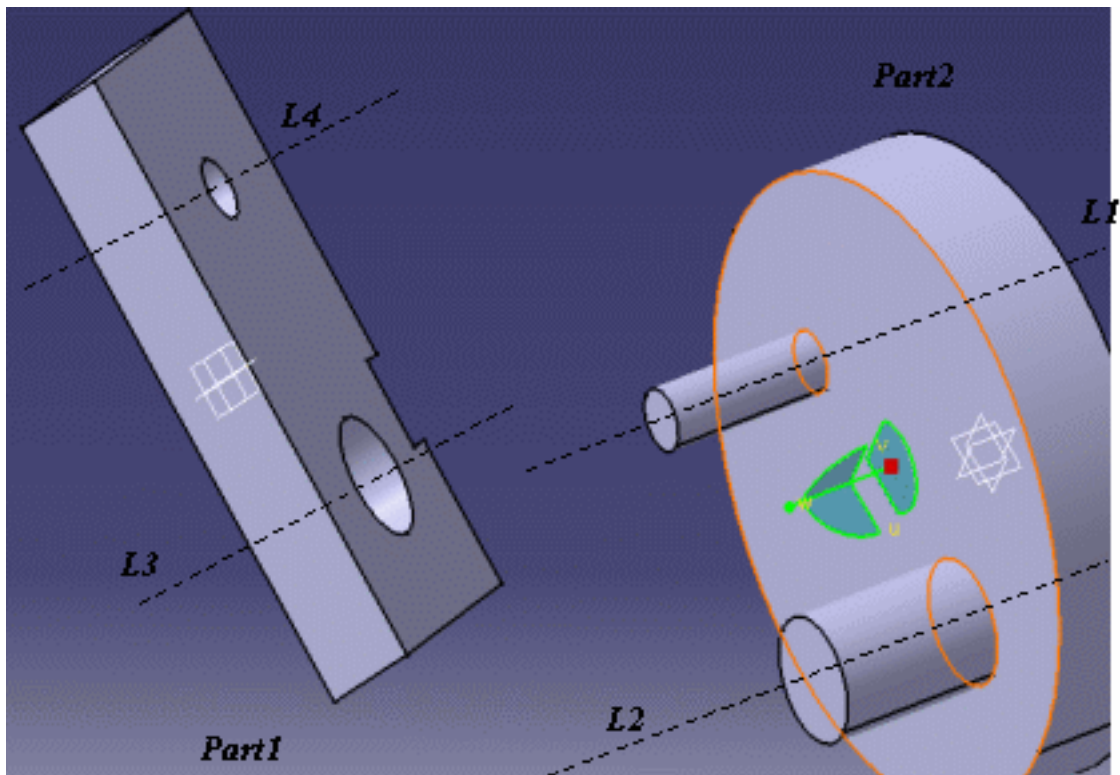
While creating or editing a track, a simulation or a shuttle, it is welcome to be able to position an entity very precisely. It can be useful to cumulate several targets. Using a historical target (keeping track of cumulative constraints) can help to put some elements in a specific position.

For instance, the scenario below aims at showing you the interest of this new capability:

- You are creating a track attached to Part2, you need this part to be plugged into Part1
- Click the smart snap icon  in the manipulation toolbar
- Select the geometries you want to focus on: L1, L2, L3 and L4 (the selection order does not matter)
- Click Ok in the dialog box displayed
- You can manipulate easily Part2 and creating the track, each time a geometry or several geometries of Part2 approaches another geometry (or several other geometries) of the environment, the magnet effect acts, snapping the geometries (in our example L1 with L3, or L2 with L3 or L1 with L4 and L2 with L3...)

## What is the advantage?

The smart snap can be used interactively



This task show you to use the Smart snap in a track recording scenario.

The objective is to snap the CHAINSAW\_ENGINE\_FLYWHEEL to the CHAINSAW\_ENGINE\_CRANKSHAFT\_LSIDE.1



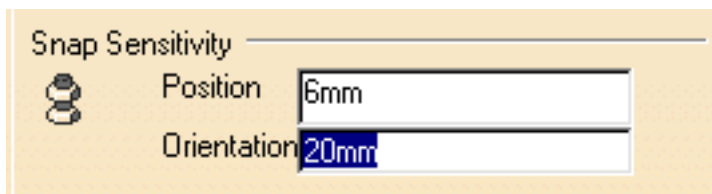
Open the [SMART\\_SNAP.CATProduct](#) document.

1.

Select **Tools->Options->DMU Fitting->DMU Manipulation** to customize snapping settings

Set values for the:

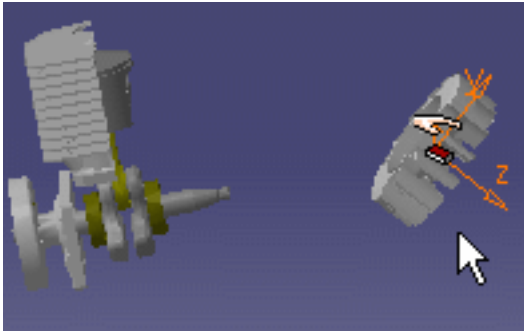
- Position: 6mm (keep the default value)
- Orientation: 20mm (keep the default value)



2.

Select the object to move either in the specification tree or in the geometry area

In our example: select shuttle.1



**3.**



Click the Track icon from the DMU Simulation toolbar.

You are ready to record your track

The player becomes active and the Manipulation toolbar appears

**4.**



Click the Target icon in the Manipulation toolbar to activate the smart snap

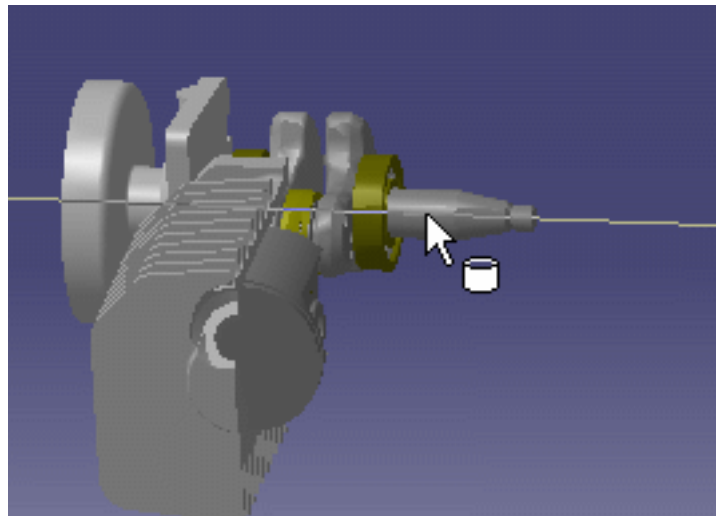
**5.**

Select the geometries you need to detect :

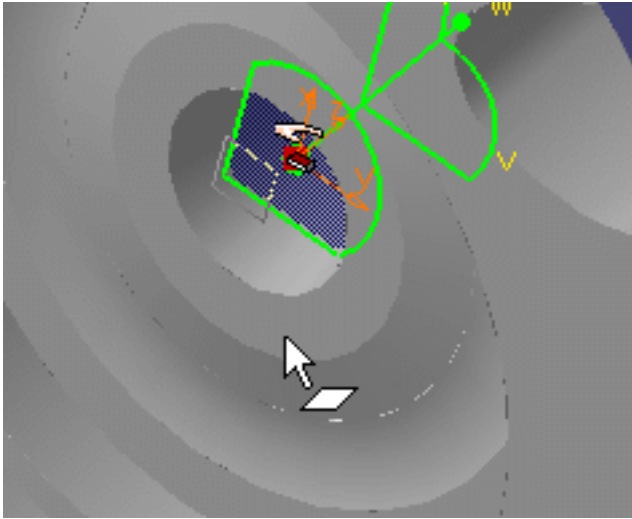
You want to snap the CHAINSAW\_ENGINE\_FLYWHEEL1 product to the CHAINSAW\_ENGINE\_CRANKSHAFT\_LSIDE product

**6.** Select the appropriate geometries (the selection order is not relevant)

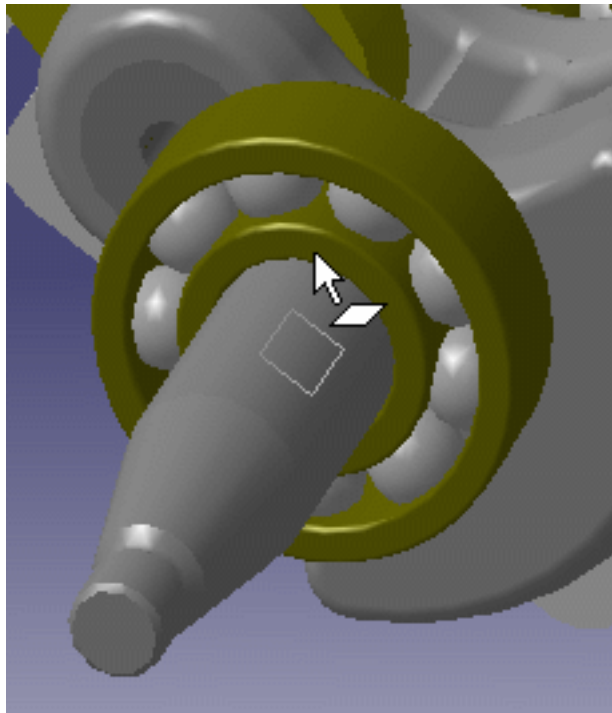
- For instance, first select the axis (cylinder) on the CHAINSAW\_ENGINE\_CRANKSHAFT\_LSIDE product and click to confirm the operation



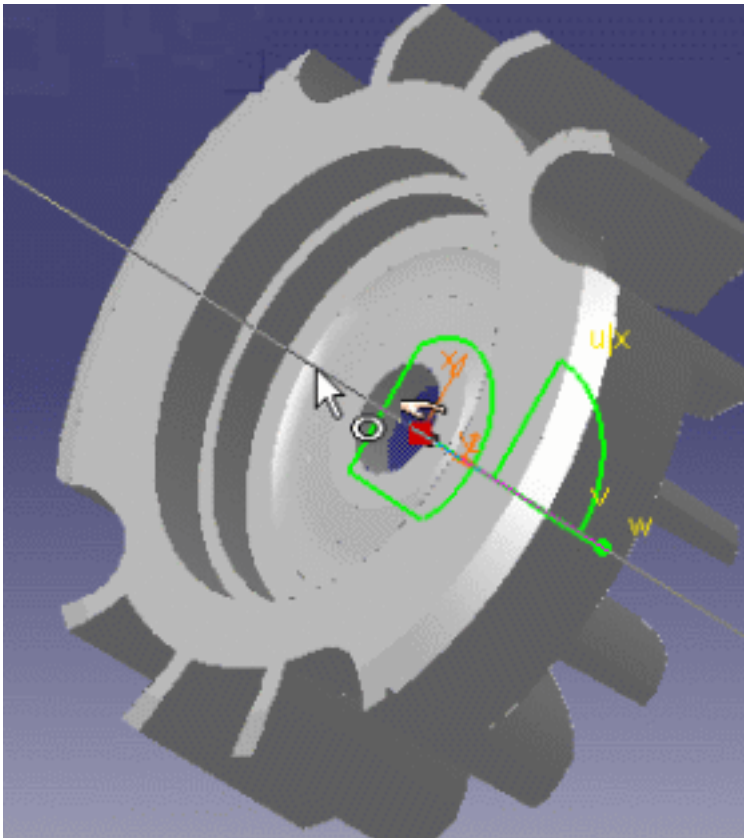
- Then, select a plane (inner one) on the CHAINSAW\_ENGINE\_FLYWHEEL1 product and click



- Back to the CHAINSAW\_ENGINE\_CRANKSHAFT\_LSIDE product, select the inner plane as shown below and click



- To finish, you need to select another axis on the CHAINSAW\_ENGINE\_FLYWHEEL1

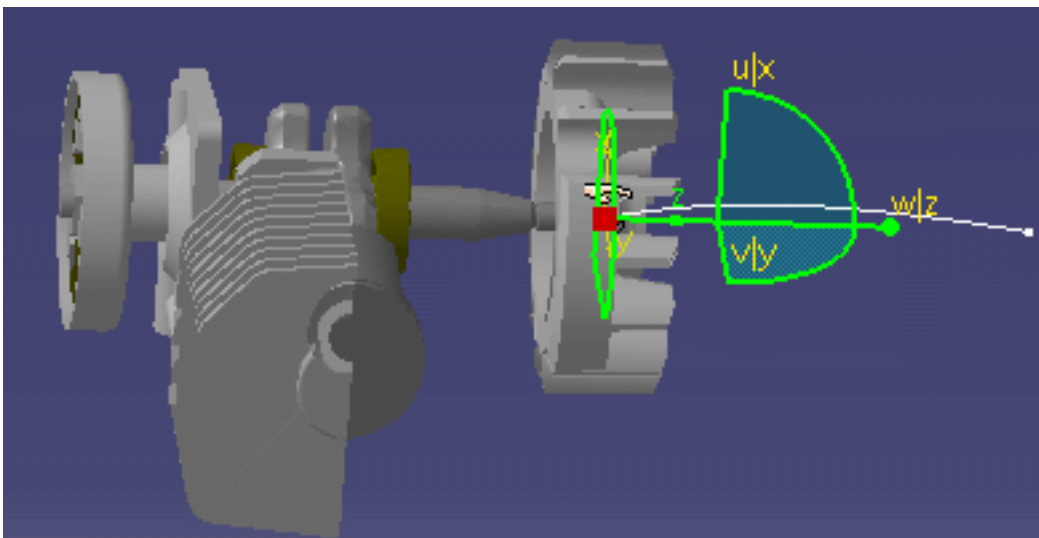


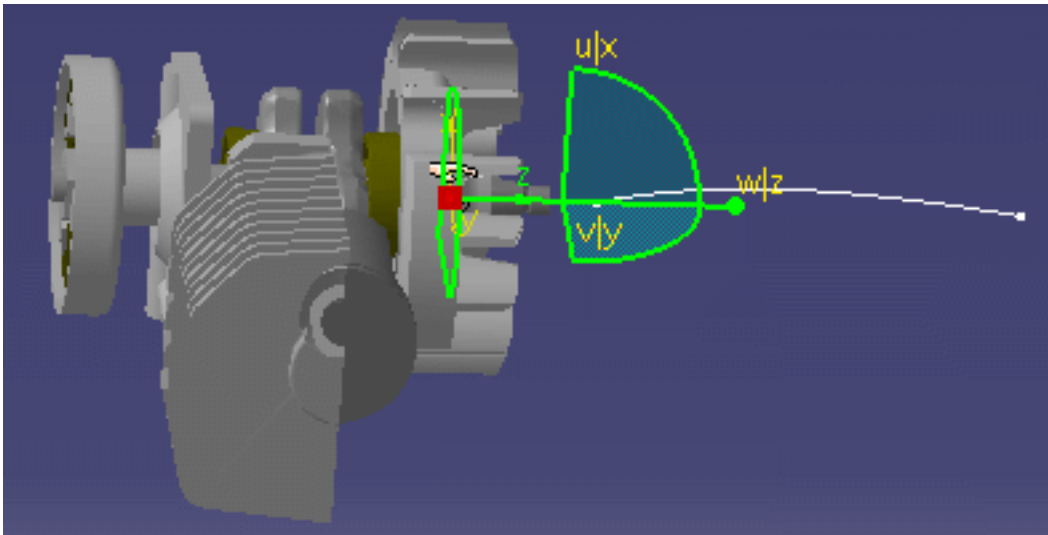
7. Click Ok in the dialog box displayed

8. Double-click Record  in the Recorder to activate the automatic insert mode.

Please refer to [Track Editor and Recorder](#) and [Recording a Track](#)

Move the shuttle so as to approach it towards the object









The planes are snapped together as well as the axes.  
The track is created, you can replay it using the player

9. Click the Target icon  to exit the command
10. Click Ok in the Track dialog box



What about the geometric features recognized when using the Smart Target capability?

Selectable elements		Design Mode	Visualization mode
Point		Yes	Yes
Line		Yes	Yes
V4 Axis System		Yes	Yes
Plane		Yes	Yes




# Moving the Shuttle



This task shows how to move the shuttle you just defined to the desired location. For this, you will use the graphic manipulator referred to as the **3D compass**.



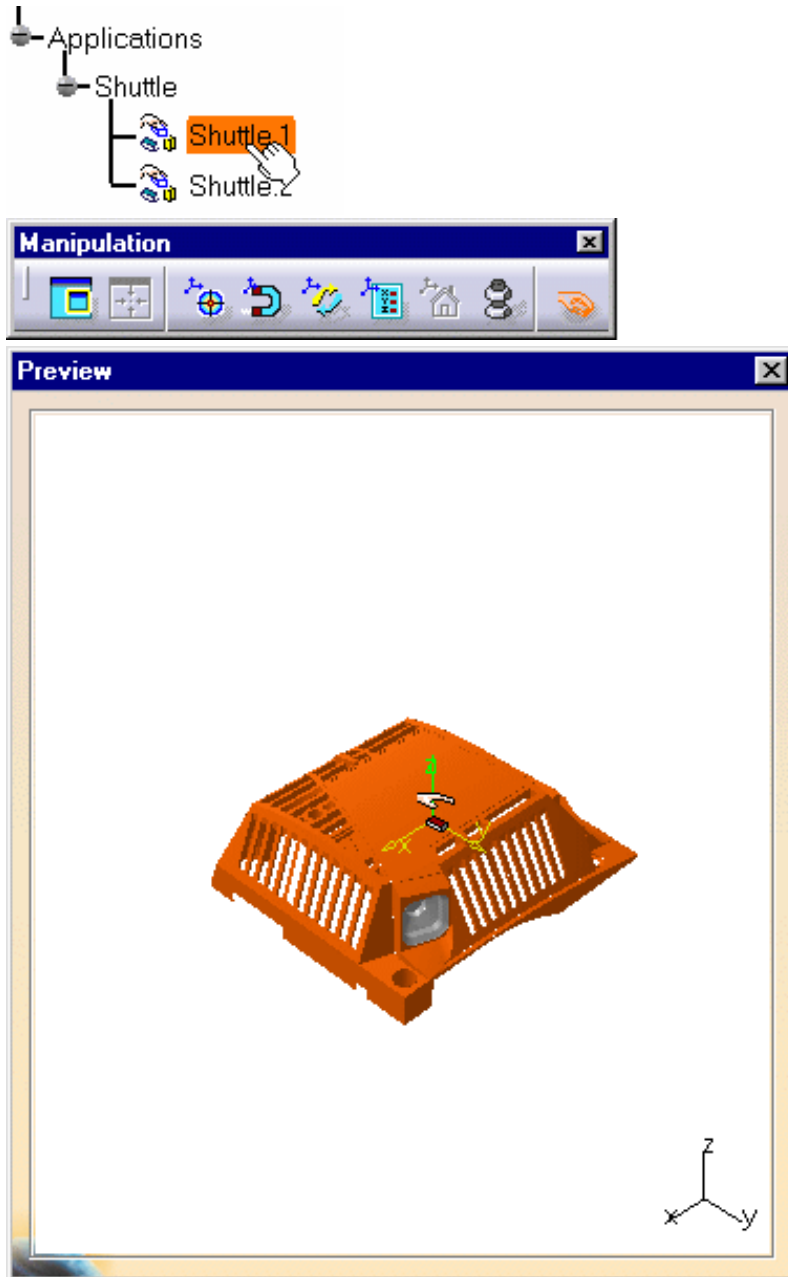
Note that, by default, the graphic manipulator is attached to the shuttle. See the Manipulation toolbar the Attach icon  is activated.



You defined a [shuttle](#). Open the [MOVE\\_SHUTTLE.CATProduct](#) document.

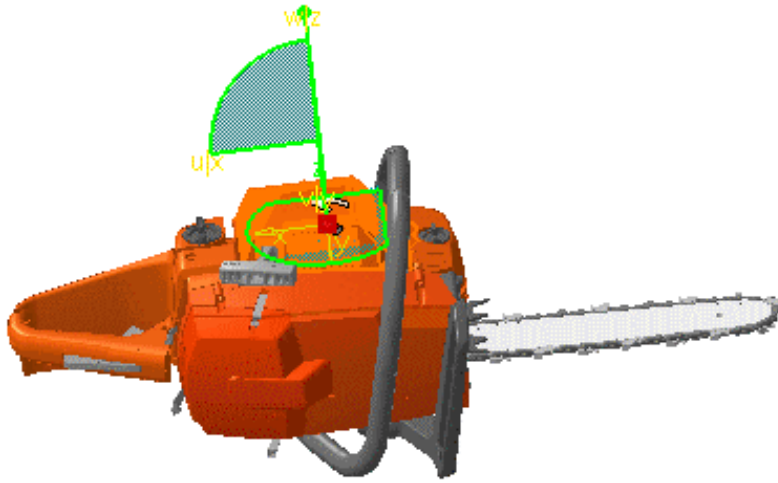


1. Double-click the shuttle in the specification tree (**shuttle.1**)  
The Edit Shuttle dialog box, the Preview window and the Manipulation Toolbar appear.



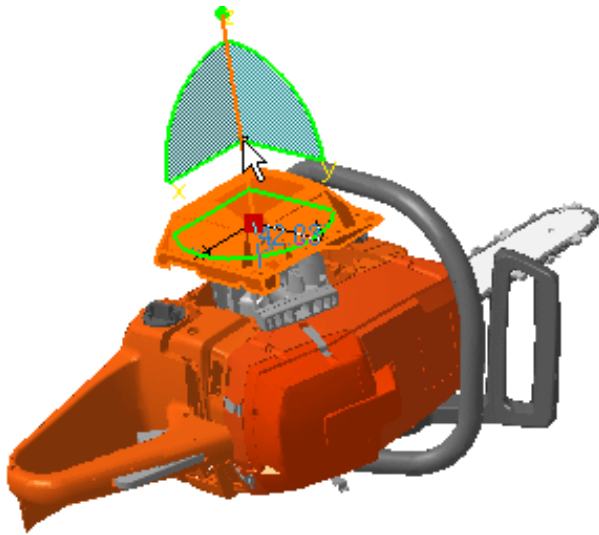
2. The 3D compass snaps to the shuttle axis.



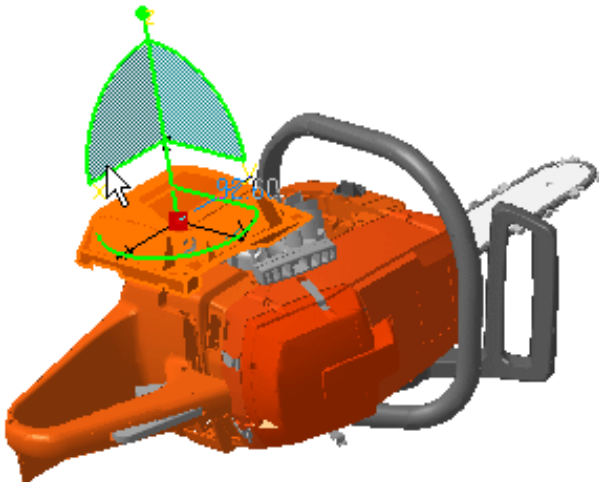


The Move shuttle option is activated by default which means that both the shuttle axis and geometry move together.

3. Click the z axis from the manipulator.
4. Drag the cursor to the shuttle desired location.

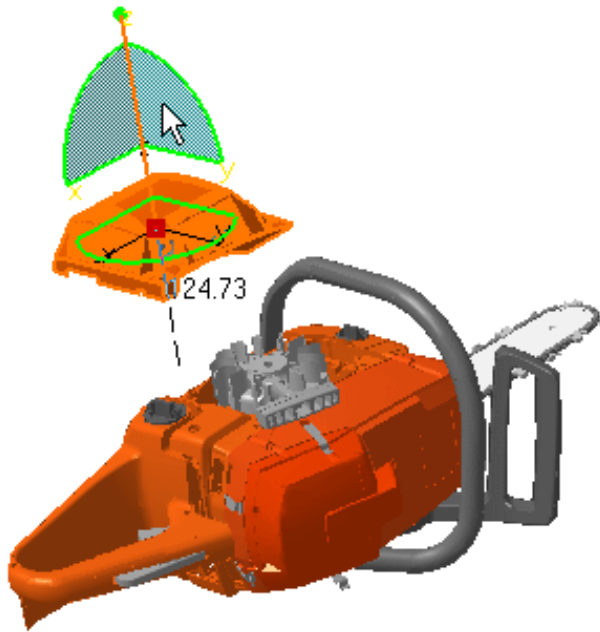


5. Click the x plane from the 3D compass.
6. Drag the cursor to the left.
7. Use the Reset command when needed.



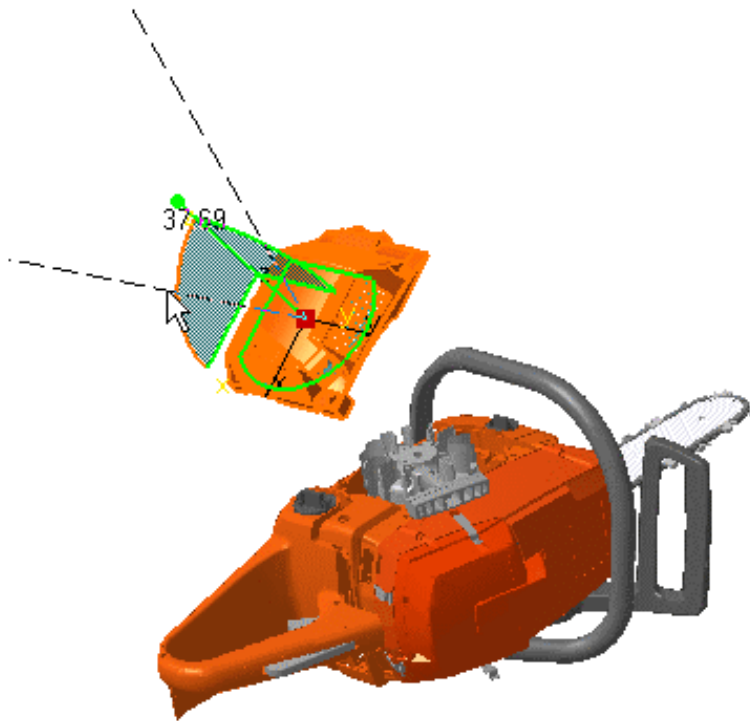
8. Select the z axis and drag the 3D compass.

9. Drag the cursor to the shuttle desired location.



10. Now rotate the shuttle. For this:

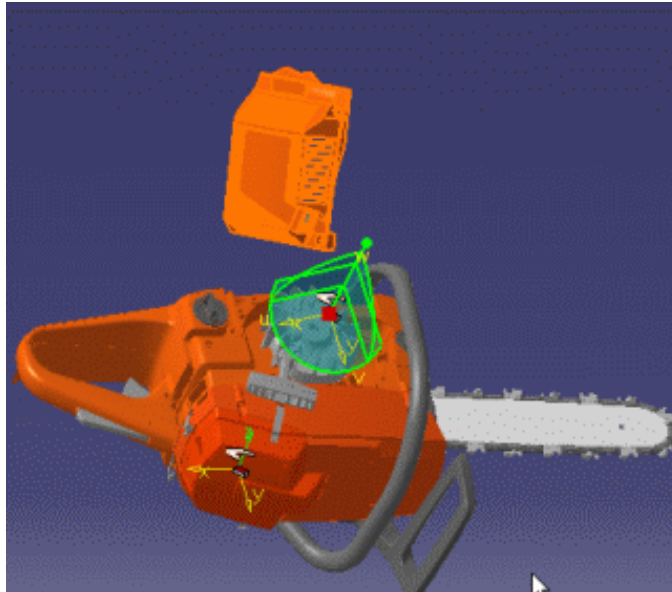
drag an arc of the compass, for example dragging the arc x z to the left rotates the shuttle in the plane subtended by the arc x z like this:



11. Click the Reset icon .

*Now, you are going to use the target icon*

12. Move the shuttle.1 as shown below

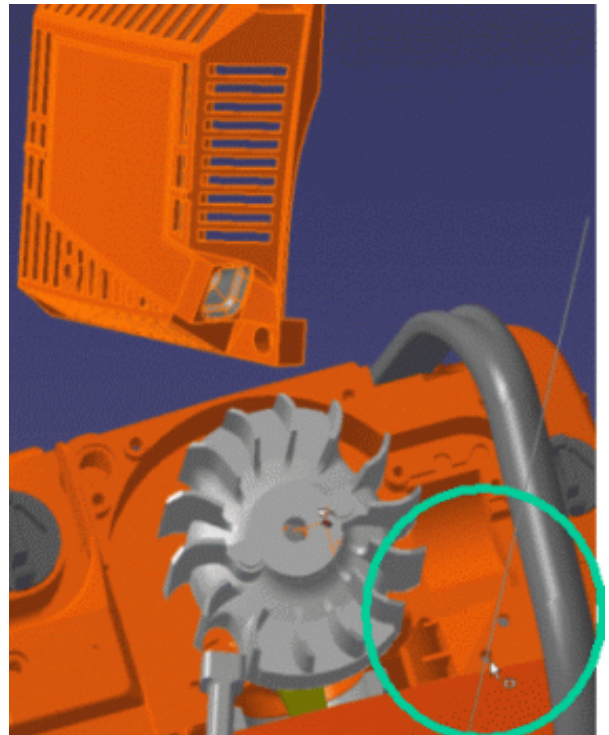
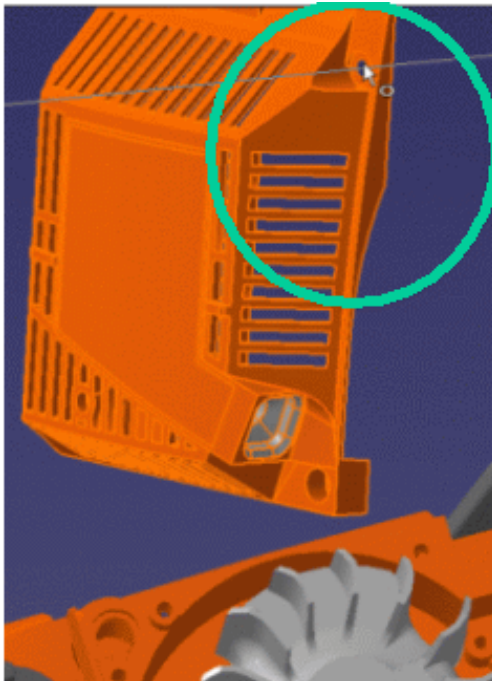


*Now you want to reposition the shuttle (startinghouse) as it was/*

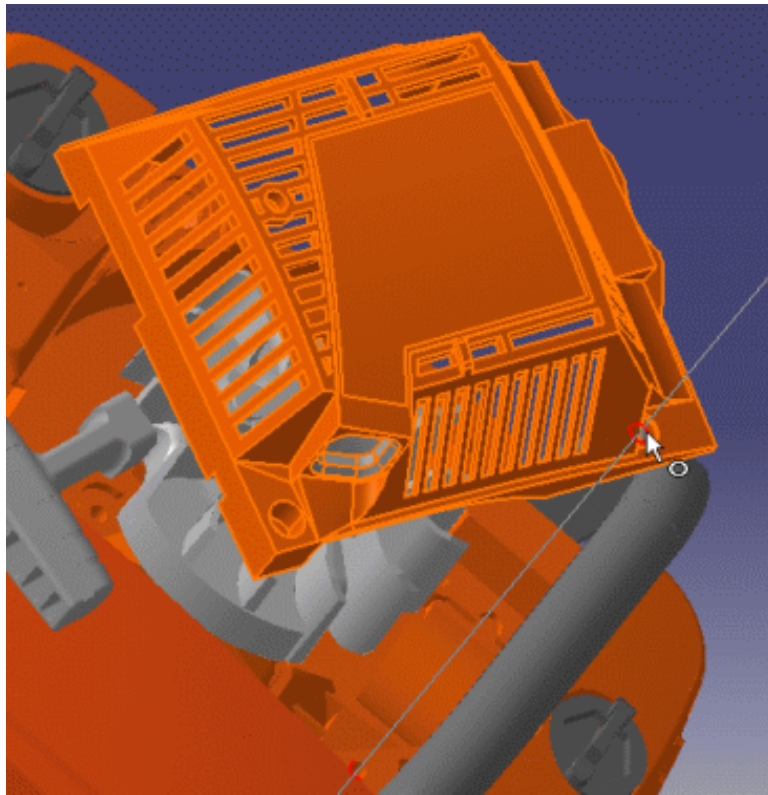
**13.**

Click the Target icon .

- The shuttle is selected and highlighted in the geometry area
- Point the cursor on the first geometric element (a cylinder axis)
- Click the mouse button.
- Select geometric element on the receiving component

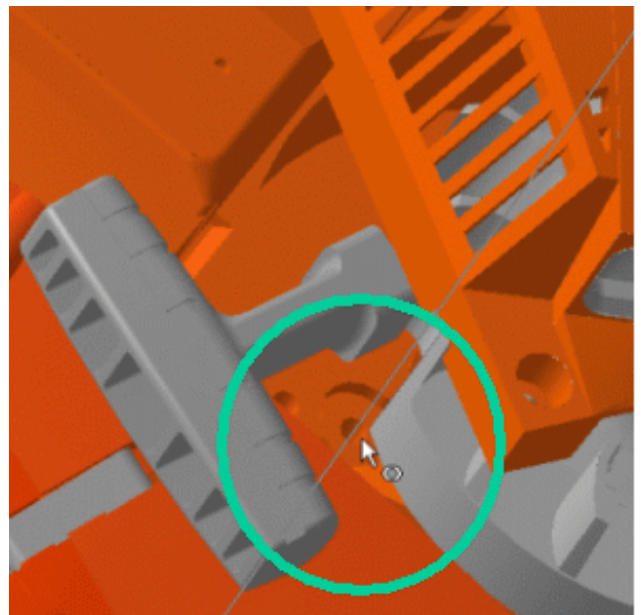
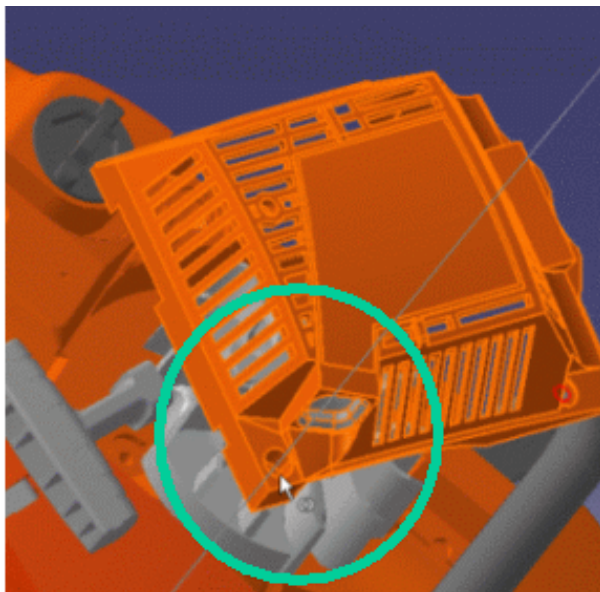


The first coincidence constraint is created. The shuttle is displaced in accordance with the first constraint. You can continue directly to the creation of the second constraint.



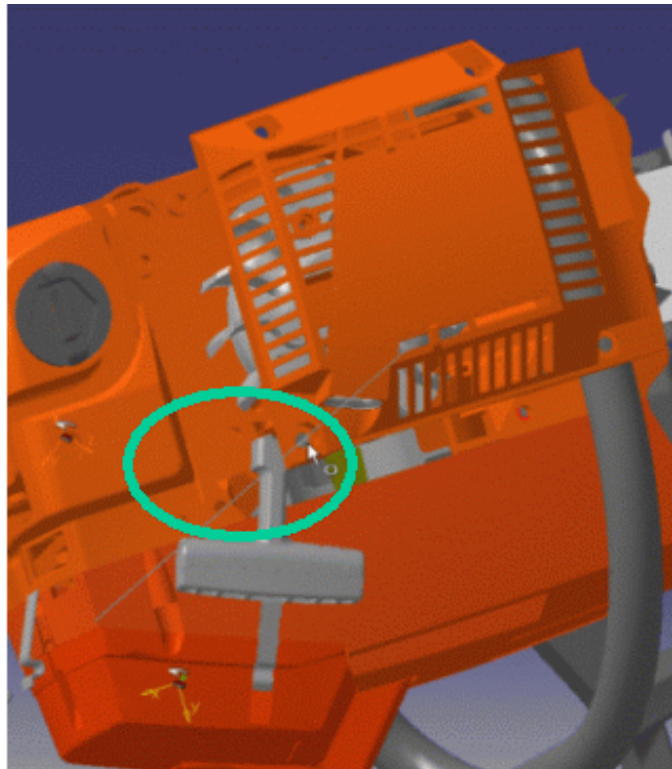
14.

- Select a geometric element on the shuttle (point the cursor) e.g. the left hole axis as shown below
- Click the mouse button.
- Select geometric element on the receiving component, a hole axis (right picture)

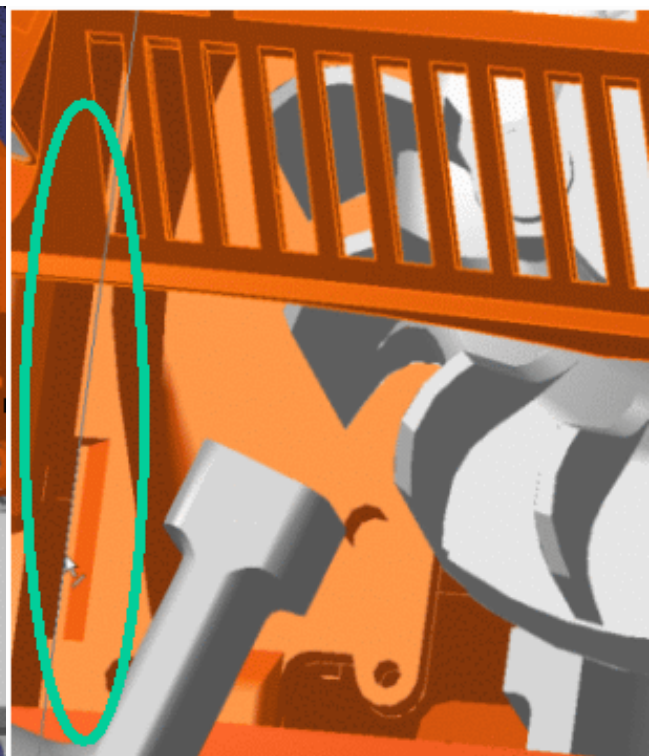
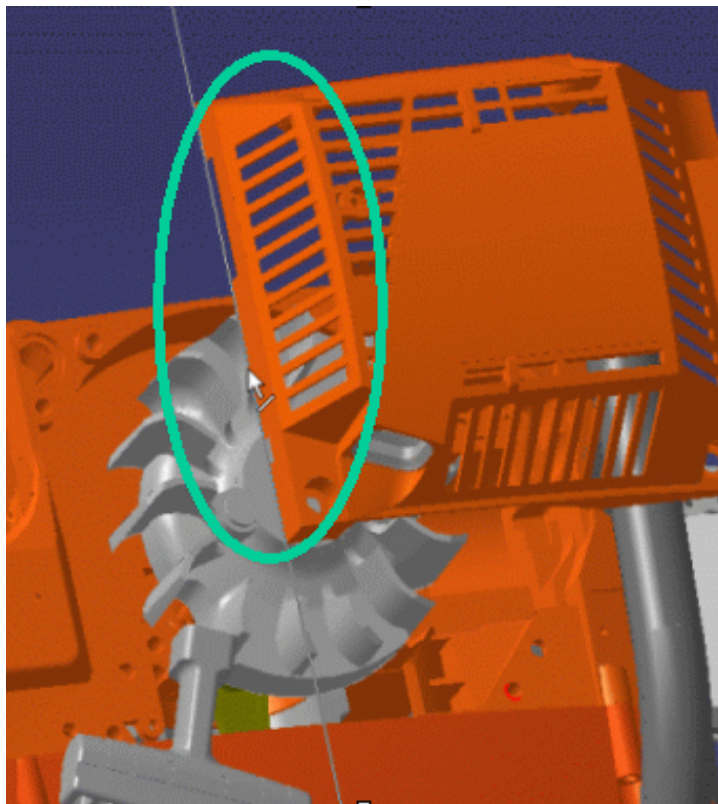


The second coincidence constraint is created. The shuttle is displaced in accordance with the second constraint. You can continue directly to the creation of the third constraint.

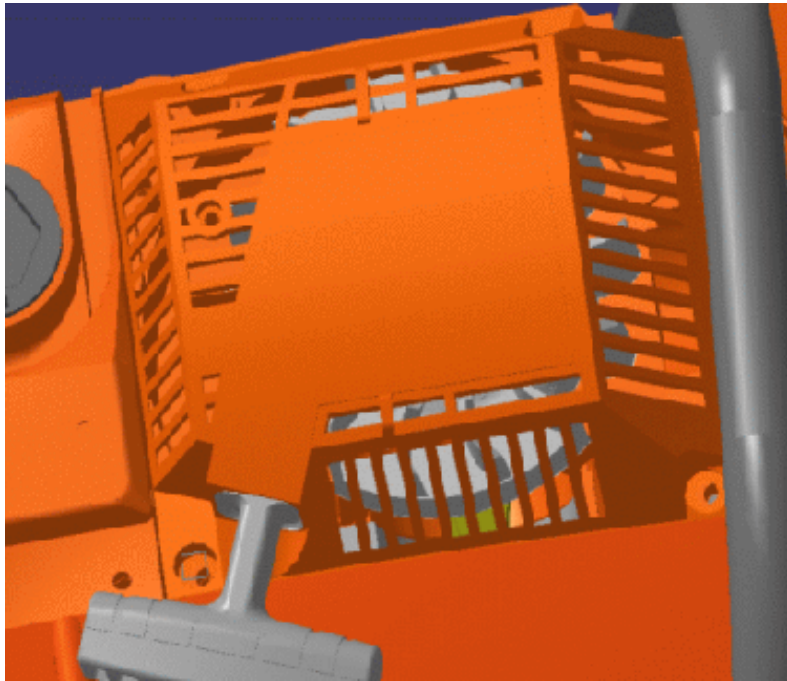




- 15.
- Select a geometric element on the shuttle e.g. a line on the shuttle as shown below
  - Click the mouse button.
  - Select geometric element on the receiving component, a line (right picture)



The third coincidence constraint is created. The shuttle is displaced in accordance with the third constraint.




16.



Click the Target icon to exit the Smart target command.

You may also enter more precise coordinates using the Compass Manipulation dialog box. For this you can either:

- right-click the 3D compass and select edit from the displayed contextual menu
- or click the Editor icon  from the Manipulation toolbar.



**Parameters for Compass Manipulation** [?] [X]

Coordinates

Apply Position Angle

Along X -10.999mm [▲] [▼] 0deg [▲] [▼]

Along Y 1mm [▲] [▼] 0deg [▲] [▼]

Along Z 79.5mm [▲] [▼] 0deg [▲] [▼]

Increments

Translation increment Rotation increment

Along U 0mm [▲] [▼] [↕] [↕] 0deg [▲] [▼] [↻] [↻]

Along V 0mm [▲] [▼] [↕] [↕] 0deg [▲] [▼] [↻] [↻]

Along W 0mm [▲] [▼] [↕] [↕] 0deg [▲] [▼] [↻] [↻]

Measures

Distance 0mm [↵] [↵] Angle 0deg [↻] [↻]

Close

17. Click OK in the Edit Shuttle dialog box.



# Tracks



**About track capabilities:** provides detailed information about track capabilities (track entity is considered in its overall)

**Track editor and recorder:** provides information about the DMU Fitting dedicated tools.

**DMU player:** provides detailed information about the player.

**Copy and paste tracks:** right-click the track to be copied. Select **Copy** from the contextual menu, then right-click tracks item in the specification tree and select **Paste** from the contextual menu. Double-click the pasted track to change the object selection.

**About track operators:** provides detailed information on how to use operators

**Analyze in track context:** select a track, click add clash on track button in the Track dialog box, then select analyses in the Analysis list and click Bound

**Export and import tracks in neutral format:** select the track and then **Tools->Simulation->Track File Export** or **Tools->Simulation->Track File Import**. Enter a meaningful name and select .xml format from the Save as Type drop-down list:

**Edit Time Line in Tracks:** select the track to be edited, click more and either drag and drop the segment or enter a precise value in the shot time field.

Please also read [About Target Capability](#)



# About Track Capabilities



A track is a route of a moving object

Objects can be:

- products
- shuttles
- section planes
- lights

- cameras:

It is possible to have an active camera automatically updated from the current view, exactly as if you'd selected Update from View each time you change the view using any combination of translates, rotations and zooms. (A camera is considered "active" when it is selected or edited in a track.)

Please refer to [Using Camera Capabilities and Recording a Camera Track](#)

- inverse Kinematics (IK) points from Human Builder manikin
- constrained .CATPart attached to a manikin (the part is moved as well as the manikin with respect to his IK)

**Note:** Those capabilities are available if you have a Human Builder license.



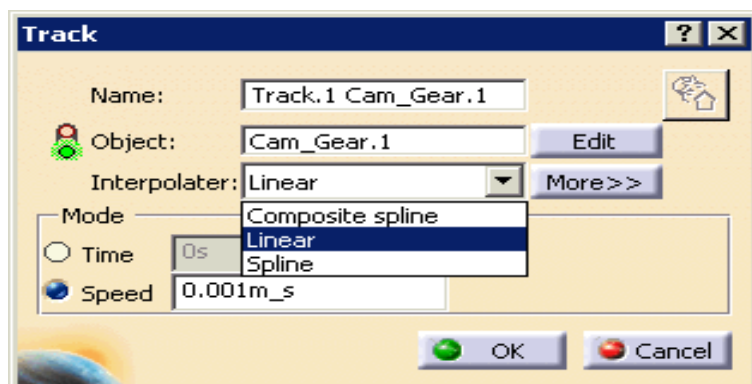
Tracks are persistent and can be stored in your document. Tracks comprise defined positions associated with time parameter. The current time is materialized with a green bullet.

### About track properties (speed/duration)

You can insert, modify, delete a position with a dedicated tool (see: [Track Editor and Recorder](#) and [Recording a Track](#))

This is a time-based trajectory. This trajectory can be interpolated with different interpolation types:

- linear ( default type for product, shuttles, section planes and lights)
- spline (default for cameras and lights)
- composite spline (enables to minimize the impact of position modifications on the entire trajectory)



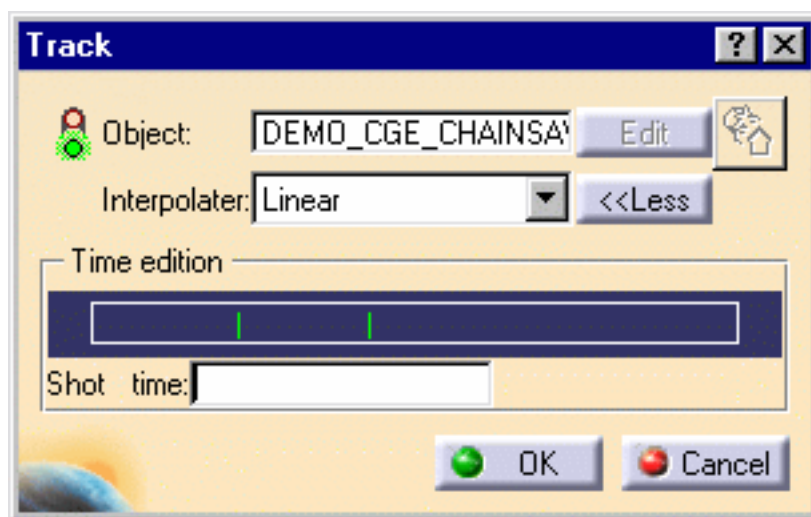
The **More>>** lets you access and edit the duration for each segment (between two positions) you can :


- edit segment duration within the Track dialog box using the More button
- modify quickly the segment duration using drag and drop capability
- enter a precise value to modify this duration.

### Useful keyboard shortcut:

Select + CTRL key: lets you drag each and every segment of the time line representation without changing the global duration.

Please read: [Editing Time Line in Tracks](#)



The  button lets you access the edit object dialog box (if the object is a shuttle, a section plane, a light or a camera (when the moving object is a camera, a light or a section plane: clicking the Edit button display the Properties dialog box)

**Note:** the Edit button is grayed out if the moving object is a product.

Positions in the track are defined with respect to the moving object coordinates.

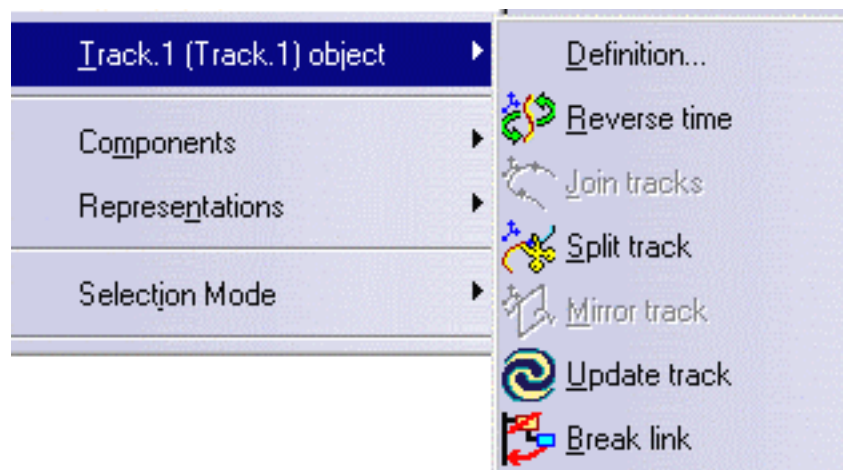
(i.e. let's say a track is defined for a light bulb, if the light bulb position is modified in the product definition, the track is updated accordingly and therefore remains consistent)

A track can be modified through a various number of operations referred to as **track operators**

- Reverse time
- Join tracks
- Split track
- Mirror track
- Transform
- Path finder
- Smooth
- Swept volume

The following can be accessed through the track contextual menu:

- Reverse time
- Join tracks
- Split track
- Mirror track



Others can be accessed using standard commands:

- Transform (rotation/translation using 3D compass)
- Path finder (DMU Check toolbar)
- Smooth (DMU Check toolbar)
- Swept volume (DMU Simulation toolbar)

The various track operators are logged using a history displayed in the specification tree. The initial track is considered as a specification, track operators are applied to this initial track. The new track defined is the final result (this is displayed in the specification tree and in the geometry area).

The original tracks are hidden, in other words visible in no show space but displayed in the specification tree.

This "history" can be deleted at any time but note that keeping this history gives you the possibility to update the resulting track as you wish when the original specifications change.

## Simulation

You can simulate your track using the dedicated player (see [DMU Player](#))

You can therefore generate an animation file ( AVI format) with DMU standard tools. (select **Tools->Image->Video...**)

You can compile your track to generate a replay object (using **Tools->Simulation->Generate Replay...**)

You can validate your track using the default clash available from the Automatic Clash detection toolbar as well as check interferences and calculate distances specifications. (See: [Validating a Motion](#) and [Analyzing in Track Context](#))

## Leaving product in modified position

When you exit the track command, the product remains in its modified position. it can be useful to:

- use it as starting position for a new simulation (i.e. you need to open the front hood before dismounting the spark plugs)
- save this position as a new product configuration

If you need to go back to the initial product position, either play the simulation from the starting position (but it is not very handy when you created several tracks) or use the **Reset**

command  from the DMU Simulation toolbar.

## Changing the moving object

You can change the moving object at any time using the track editor (simply click the object selection field)

The track can be relocated on this new object or not.

Please refer to [Track Editor and Recorder](#)



### Example .1

(keep current track position option)

A track is defined to dismount various objects through a bottleneck. The track needs to remain at the same location with respect to the bottleneck whatever the object is, in this case, you should keep current track position.

### Example .2

(relocate track on new object option)

A track is defined to unscrew a spark plug. You want to make sure this track can be applied to another spark plug. In this case, changing the moving object along the track is valid only if you can unscrew the second spark plug from its current location, use the automatic relocate track (Do not keep positioning option).

## Copy/Paste capability

You can copy and paste tracks to create instances of reference tracks. If you modify the "shot positions" of a track, the reference track is therefore modified and all the instances will be modified (either instance or reference tracks).

Then, you can apply track operators (see [About Track operators](#)) on instances, for instance to relocate them keeping the links existing between the references and instances)

i.e. You defined a track to remove a spark plug. You create instances for the other spark plugs, you can modify the moving object along the track to move the other spark plugs with respect to the current position of the spark plug instance. All the spark plug instances will be moved with the same motion.

See: [Copying and Pasting Tracks](#)

## Break link

This capability lets you break the link existing between the reference track and its instances. i.e. in the above example, one of the spark plug cannot be dismounted in the same manner, you can use break link to modify this particular instance track without impacting the others.


## Clash reporting

You can through the publish capability, obtain a concise clash .html report on a single track simulation (automatic clash detection + regular clash analysis)

## Scenario

1. Interferences specifications are defined and linked to tracks in your document.
2. Activate the publish functionality (select **Tools->Publish->Start publish...**) please read Publishing in the *DMU Navigator User's Guide*, and also read [Publishing](#)



3. Click the Player icon
4. Activate the automatic clash detection
5. Click Play forwards button  in the Player toolbar

The clash detection is launched

6. Click Stop publish icon or select **Tools -> Publish -> Stop Publish**.
7. Read your published clash report

For more detailed information, please refer to [Analyzing in Track Context](#)

## About Journaling/automation

Tracks are journalized. You can generate a macro using **Tools->Macro->Record...** (see the Infrastructure user's Guide)



### Track creation:

2 methods are now available to create tracks:


- Clicking the Track icon first and select the objects afterwards.
- Selecting the objects first and activating the command after.

Tracks created in this manner are persistent and can be stored in the document. They are listed as separate entities in the specification tree and can be selected at any time and modified.





# Track Editor and Recorder















Information follows about the track capability display, including the [player](#) toolbar, the DMU [recorder](#) toolbar, and the [track editor](#) function.

## About the Player Toolbar

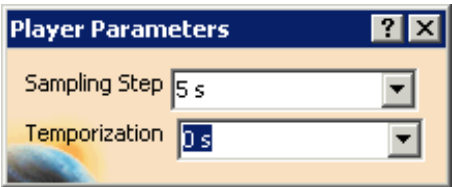
More detailed information is provided (about Player shortcuts for instance) in the [Player](#) task.



Use the VCR buttons or the slider to simulate the movement of parts along your track.







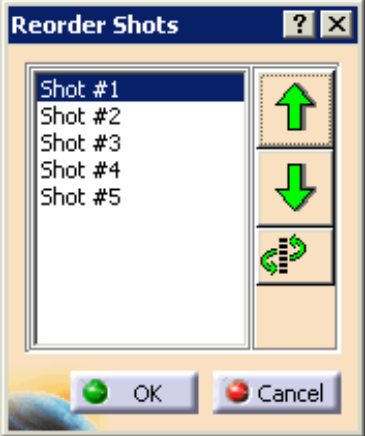




	Skip to beginning		Slider: As you move the slider, you make the simulation move forwards or backwards.
	Step backwards	<div><div>s</div><div>Time (0 s)</div><div>Shot</div></div>	Selecting Time makes the simulation proceed according to the simulation time. The time units (by default, seconds) are set in <b>Tools-&gt;Options-&gt;Parameters and Measure-&gt;Units</b> .  Selecting Shot makes the simulation jump from one shot to another, in sequence.
	Play backwards		Player parameters button. More information appears <a href="#">below</a> .
	Stop	<b>Loop Mode</b>	
	Play forwards		Single loop (shows simulation once, from beginning to end).  Note: This is the default value. To see other loop options, click on this button. The option that is visible is the one that will be operating.
	Step forwards		Continuous loop, from beginning to end, then jumps back to beginning.
	Skip to end		Continuous loop, from beginning to end, then end to beginning.

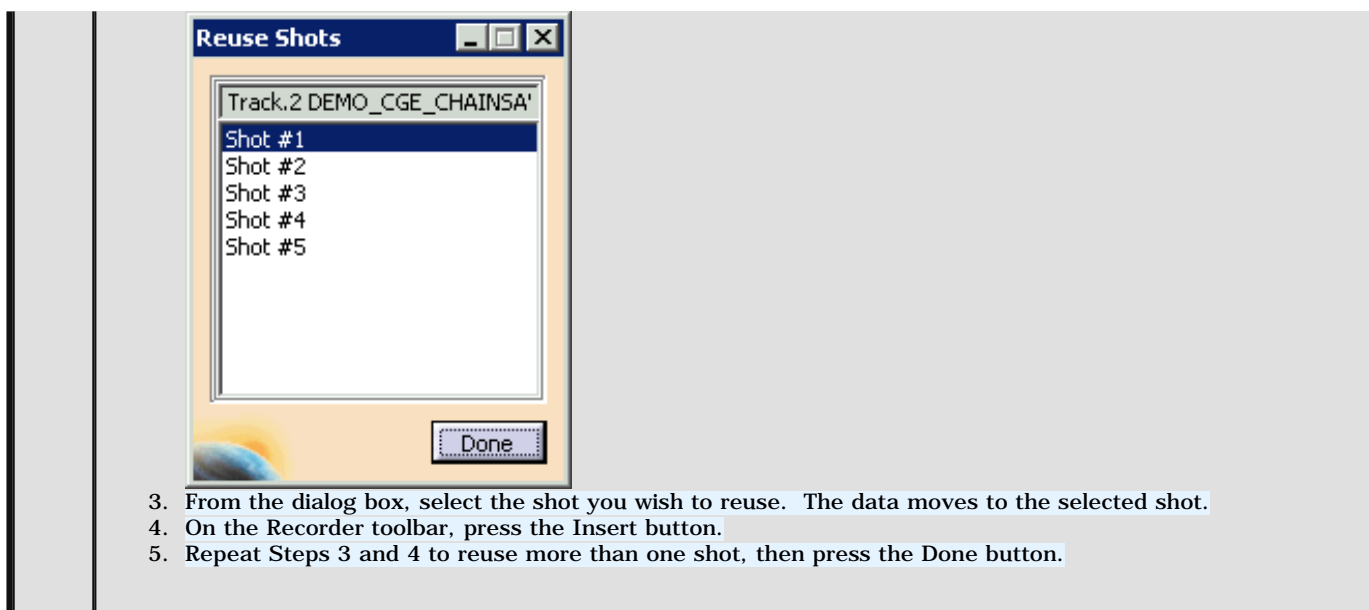
Selecting the player parameter button brings up a dialog box that enables you to set a sampling size so that you see the simulation at steps of every 1, 2, or 5 seconds. You can also select an temporization option, which enables you to determine how quickly you see the simulation. Without selecting the parameters, you see one second of the simulation in one second of real time. If you want to view the simulation more quickly, you can set the temporization size smaller (e.g., to .25 s) and the sample size larger (e.g., 5 s).





## About the Recorder Toolbar



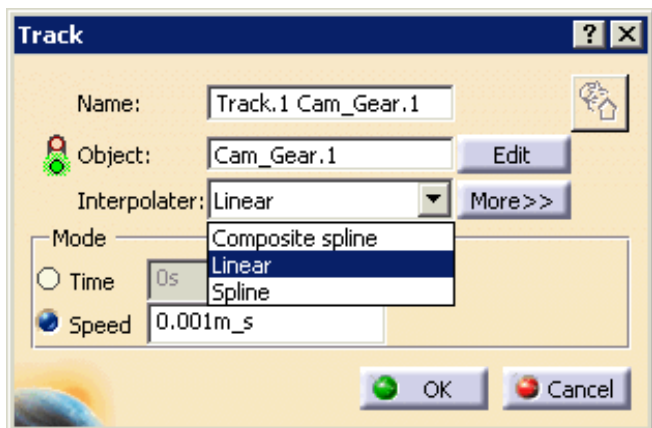
	<ul style="list-style-type: none"><li>• <b>One click:</b> Records shots one after another.</li><li>• <b>Double-click:</b> Activates the auto insert mode (see Automatic Insert configuration using <b>Tools-&gt;Options-&gt;Digital Mockup-&gt;DMU Fitting-&gt;DMU Manipulation</b>).<ul style="list-style-type: none"><li>◦ To deactivate this mode: Click once.</li></ul></li></ul>
	<ul style="list-style-type: none"><li>• <b>One click:</b> Records modification(s) on one shot at a time.</li><li>• <b>Double-click:</b> Records modification(s) in continuous mode. You need to be positioned on the required shot.<ul style="list-style-type: none"><li>◦ On the Player toolbar, use the Steps Backwards  and/or Steps Forwards  icons to position the manipulator on the required shot.</li><li>◦ To deactivate this mode: Click once.</li></ul></li></ul>
	<p>Deletes one shot after another. The cursor must be position on the shot to be deleted.</p> <p><b>Note:</b> In track edition context, if you need to delete various shots at a time, multi-select them (Select +SHIFT key or Select +CTRL key) and use the Delete Key or the Delete item from the contextual menu.</p>
	<p>Reorders shots. To use:</p> <ol style="list-style-type: none"><li>1. Select a track.</li><li>2. Click on the Reorder button. The Reorder dialog box appears:<div data-bbox="349 772 711 1205"></div><ul style="list-style-type: none"><li>◦ To move a shot up one place in the list, select the shot, and press the Up button. </li><li>◦ To move a shot down one place in the list, select the shot, and press the Down button. </li><li>◦ To insert a selected shot after next clicked shot, select the shot, click on the shot you wish it to follow and push the After button. </li></ul></li><li>3. Once the reorder is complete, select the OK button. (Selecting the Cancel button returns the shots to their original order).</li></ol>
	<p>Reuse enables you to use shots from another track while creating or editing a track. To use:</p> <ol style="list-style-type: none"><li>1. Click on the Reuse button.</li><li>2. Select the track containing the shots you want to reuse. The Reuse dialog box appears:</li></ol>



When using the Recorder toolbar, you can use the Player toolbar to select shots. To do so, set the step unit to **Shot** (i.e., not Time), and that the value of the Player Parameters sampling step is set to 1. If this is done, the Player toolbar allows:

-  Steps backwards (lets you to jump to the previous shot -- e.g., to be modified).
- : Steps forwards (lets you jump to the next shot).

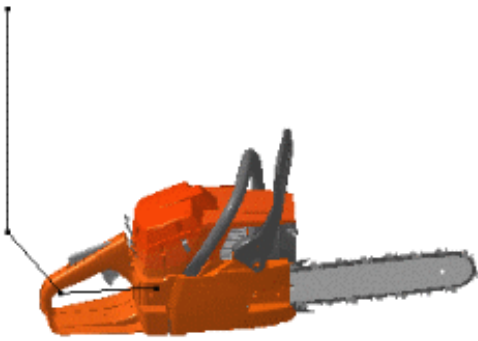
## About the Track Editor



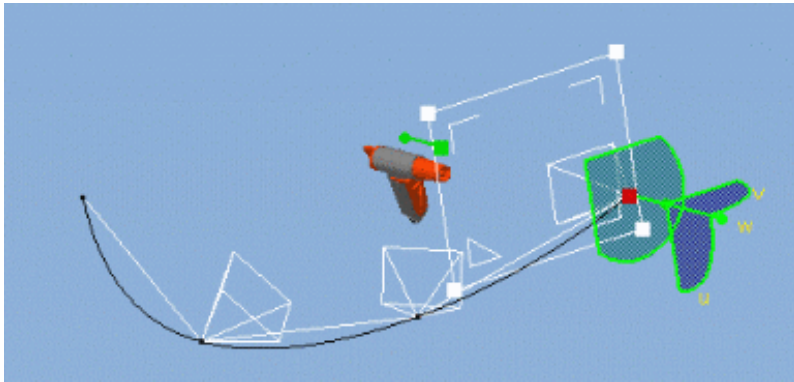
A track is a time-based trajectory which can be interpolated with different interpolation types:

- **linear** (default type for product, shuttles, and section planes)
- **spline** (default for cameras and lights)
- **composite spline** (enables users to minimize the impact of positions modifications on the entire trajectory)

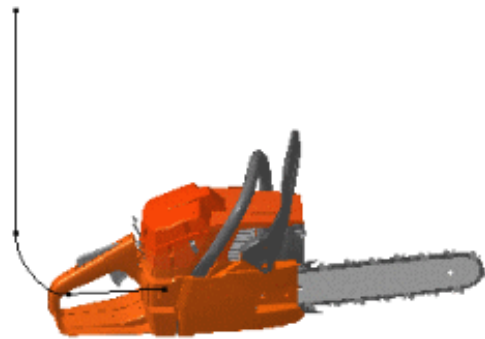
Product track: **Linear**



Camera track : **Spline**



Product track: **Composite spline**

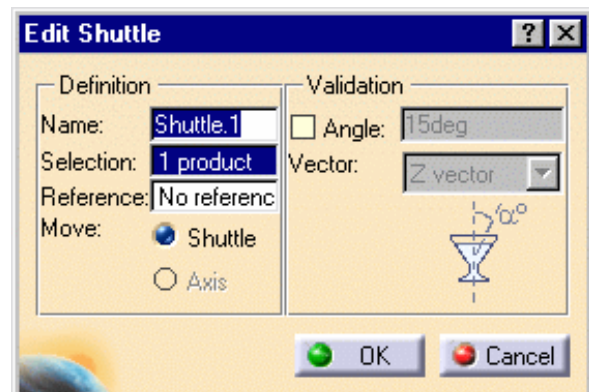
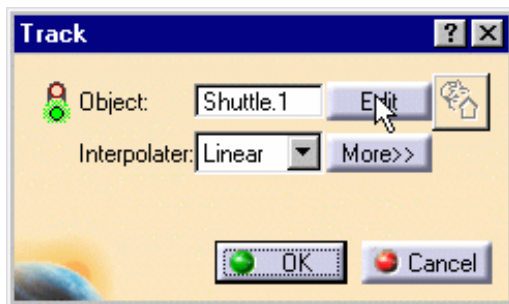


You can easily edit the track object clicking the Edit button

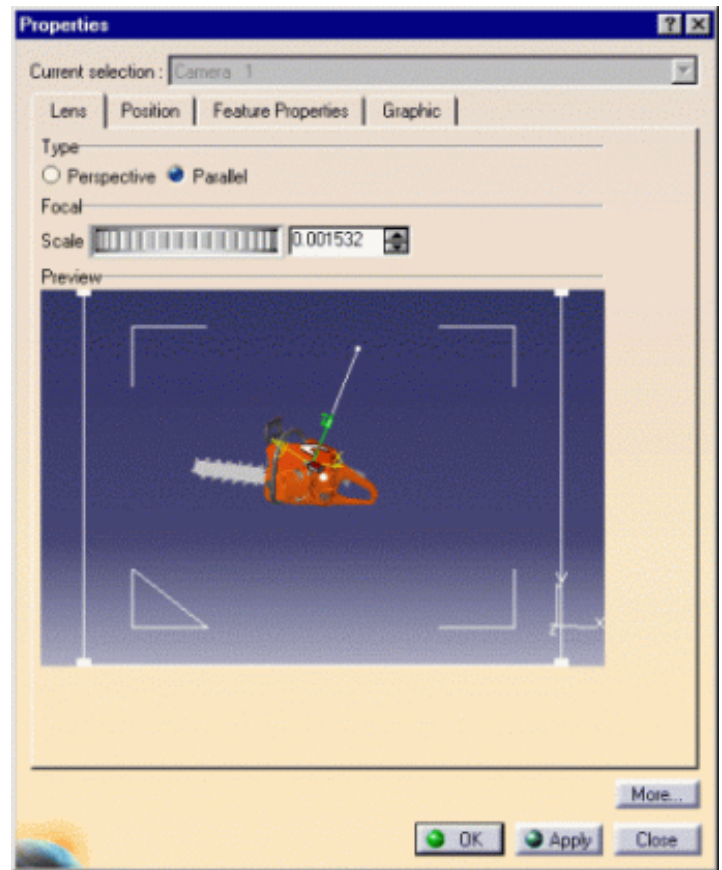
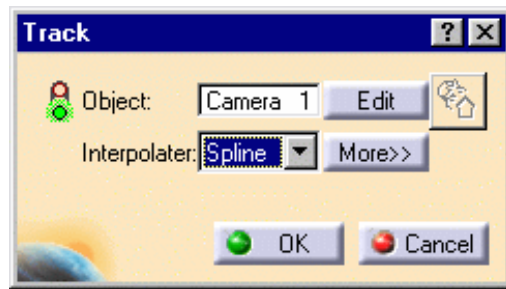
Edit

## Objects and the Associated Dialog Boxes

If the object is a Shuttle, the Edit Shuttle dialog box is displayed.



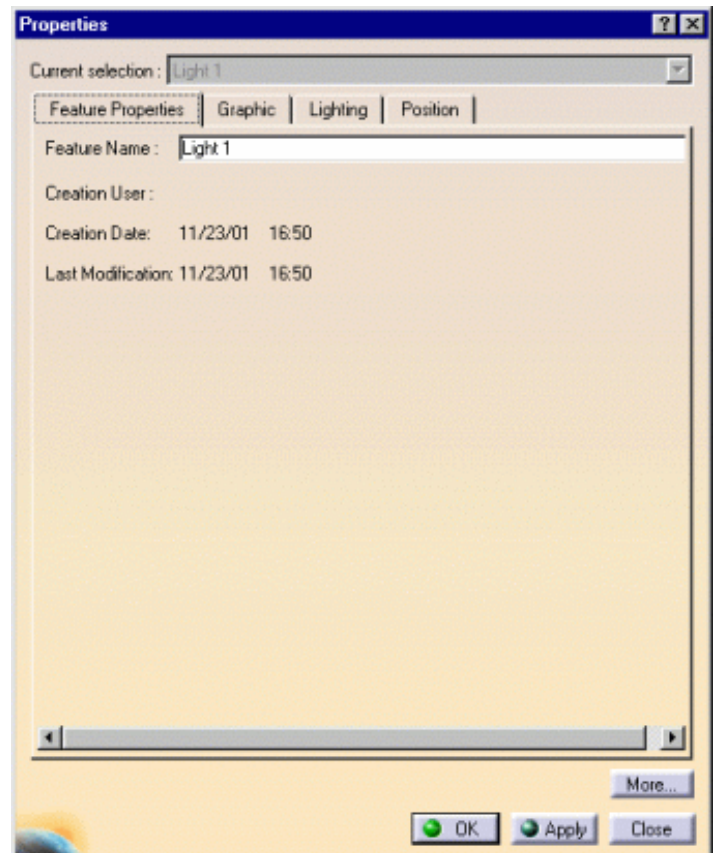
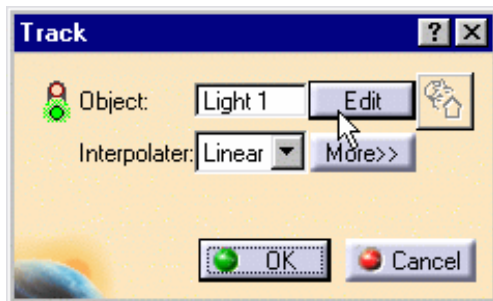
If the object is a camera, the Properties dialog box is displayed.



If the object is a light, the Properties dialog box is displayed.

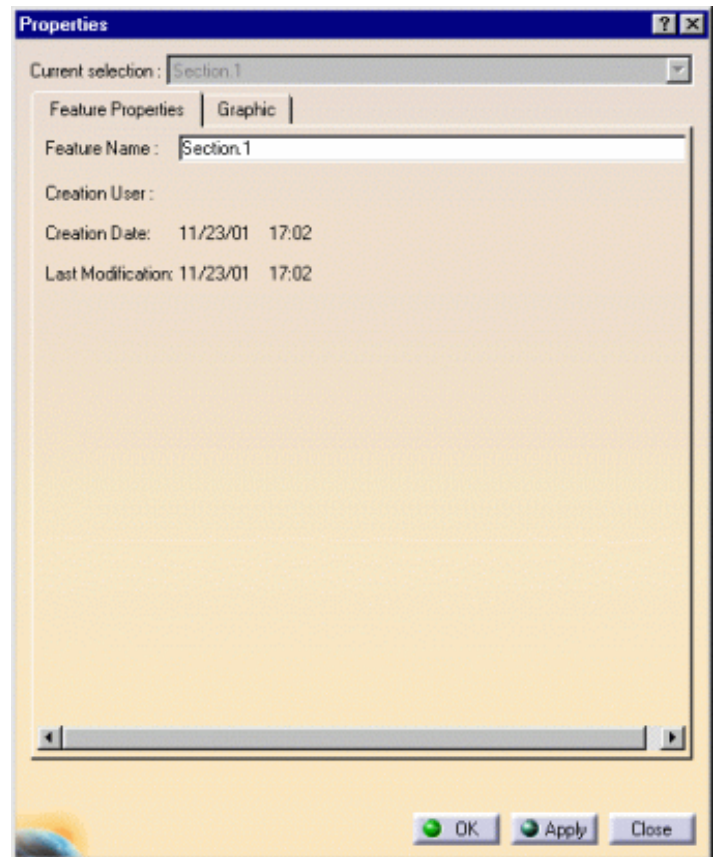
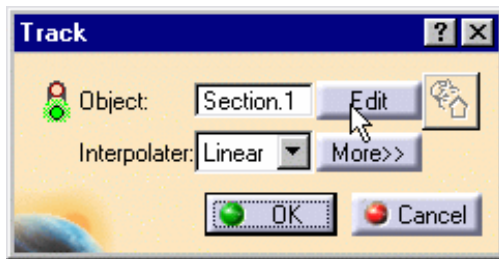


**Note:** Make sure you have the *Real Time Rendering* license to create lights.




If the object is a section plane, the Properties dialog box is displayed.





**Note:** the Edit button is grayed out if the moving object is a product.

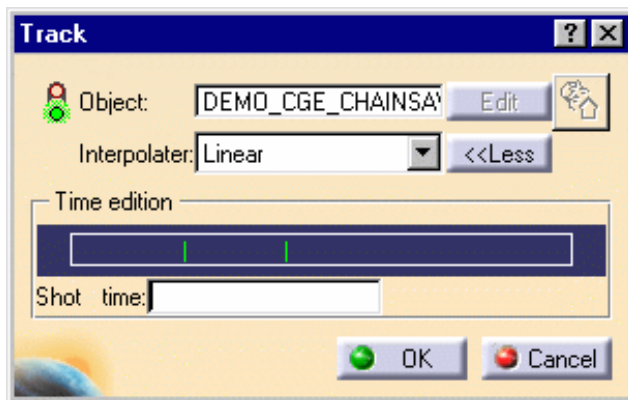
The More button  lets you access and edit the duration of each segment (i.e., the amount of time it takes to travel between two positions). To alter the duration of each segment, you can:

- Edit segment duration within the Track dialog box.
- Modify quickly the segment duration using drag and drop capability (see below).
- Enter a precise value to modify this duration.

### Useful keyboard shortcut:

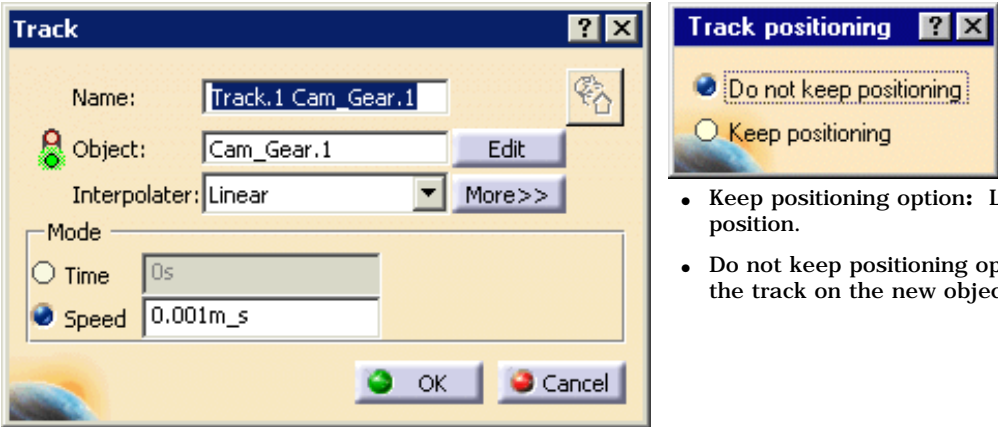
Select + CTRL key: lets you drag each and every segment of the time line representation without changing the global duration.

Please read [Editing Time Line in Tracks](#)



You can change the object selection during the track creation or while editing it:

- Click on the object field, and the track positioning dialog box appears:



- Select a new object either in the geometry or in the specification tree.

About Selecting in Editing Mode

**Note:** In edition mode, you can select objects to be applied only one track at a time.

Keyboard Shortcuts

Use this keyboard key (or combination)	To
Insert	Record
Shift + m key	Modify
Delete Key	Delete
Page Up	To go to the previous step
Page Down	To go to the next step

Another useful shortcut


Clicking an icon lets you run the command associated with that icon only once. However, double-clicking an icon lets you use the associated command as many times as you want without having to click on the icon several times.



# Player




The Player is available every time you create a track, a sequence or when you simulate your track. You can undock the Player toolbar at any time

You can access it at any time (to [generate a replay](#) or publish a clash report for instance), clicking the Play a Simulation icon  from the DMU Simulation toolbar or selecting **Tools->Simulation->Player...**





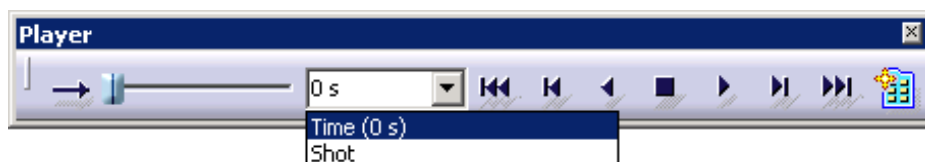
Let's look at it more carefully

Three loop modes are available:

-  : Single loop (shows simulation once, from beginning to end).

Note: This is the default value. To see other loop options, click on this button. The option that is visible is the one that will be operating.

-  : Continuous loop, from beginning to end, then end to beginning.
-  : Continuous loop, from beginning to end, then jumps back to beginning.



The Parameter field lets you enter a precise value according the parameter unit selected in the drop-down list:

For a track, this parameter can be either :

- time in seconds (default parameter)
- shots (key frame)

Use the Player buttons or the slider to simulate your track directly.

-  skip to beginning
-  step backwards
-  play backwards
-  stop
-  play forward
-  step forward
-  skip to end



**Note:** to access a recorded shot for modification purposes, use step backward

 and step forward  buttons from the Player toolbar.



Keyboard Shortcuts




When working in full screen, the Player toolbar is not accessible. Use the following keyboard shortcuts to access the Player capabilities:

**Note:** These shortcuts are also available in other screen modes

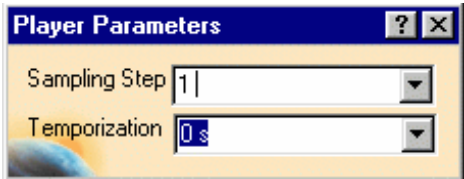


## Keyboard Shortcuts

Use this keyboard key (or combination)... To...	
Right arrow	Play Forward
Left arrow	Play Backward
Up arrow	Step Forward
Down arrow	Step Backward
l (l for loop in lower case)	Set the Loop Mode
p (for parameters in lower case)	access Speed and Pause settings

2. Click the Parameters icon 

The Player Parameters dialog box is displayed:



The track is replayed at a constant speed.

### About Sampling Step

The sampling step corresponds to the sampling step value in seconds (the total duration is divided into intervals calculated in seconds)

By default, four "sampling" time steps are available, but feel free to edit these values whenever you need to.

### About Temporization

Lets you introduce a short pause between sampling steps.



# About Track Operators

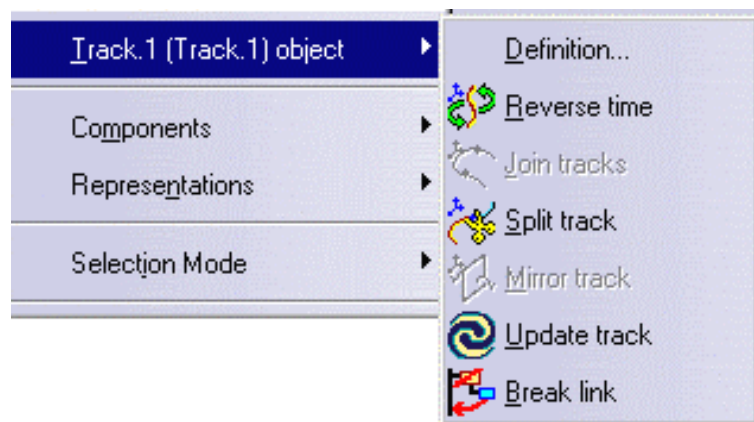


A track can be modified through a various number of operations.

- [Reverse time](#)
- [Join tracks](#)
- [Split track](#)
- [Mirror track](#)
- [Transform](#)
- [Path finder](#)
- [Smooth](#)
- [Swept volume](#)

The following can be accessed through the track contextual menu:

- Reverse time
- Join tracks
- Split track
- Mirror track



**Note:** for detailed information about "Break link" functionality, please refer to [Copying and Pasting Tracks](#).

Others can be accessed using standard commands:

- Transform (rotation/translation using 3D compass)
- Path finder (DMU Check toolbar)
- Smooth (DMU Check toolbar)
- Swept volume (DMU Simulation toolbar)



These operations are referred to as **track operators**

These operations modify the original trajectory in different ways.

The initial track is considered as an input specification and the operation output is a new track considered as a result.

If the original specification is modified (because of design or packaging changes), you can automatically update the resulting track (using the "

Update track" capability   accessible from the track contextual menu)

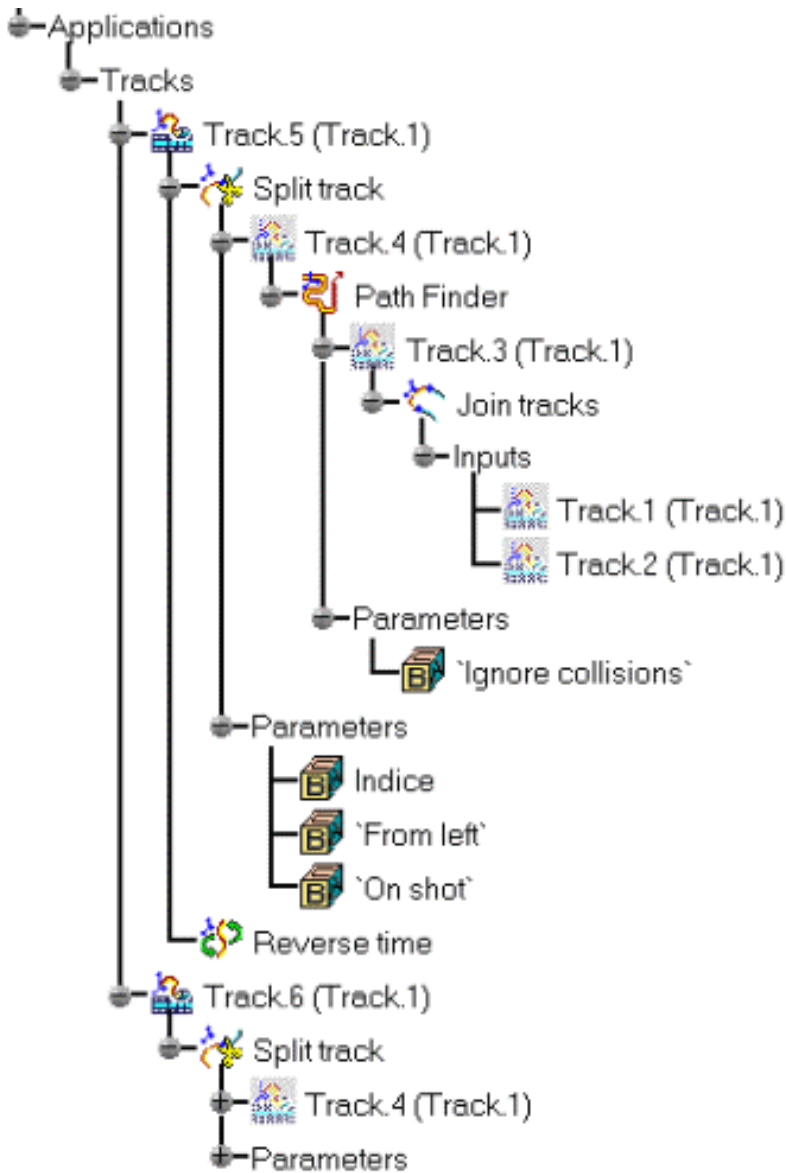
The various track operators are logged using a history displayed in the specification tree. The original track is considered as a specification, track operators are applied to this initial track.

The new track defined is the final result (this is displayed in the specification tree and in the geometry area).

The original tracks are hidden, in other words visible in no show space but displayed in the specification tree.

This "history" can be deleted at any time but note that keeping this history gives you the possibility to update the resulting track as you wish when the original specifications change.

This is an example of a track operator history:





### About reversing time:

Lets you modify the trajectory direction (by default, the direction is defined with respect to the creation order of the track positions)

Use this functionality to simulate your track the other way round (i.e. a dismounting track becomes a mounting track)

### What you need to do:

Right-click the track, select the track object item and select the Reverse time item from the contextual menu displayed.



### About joining tracks:

Lets you merge several tracks. You can select more than two tracks at a time.

### What you need to do:

Multi-select tracks, select the Join tracks item from the contextual menu: a new track is created. The initial tracks are swapped in no show space

Be careful because the result depends on the tracks selection order.



See: example below

**Two tracks to be merged**



**Result 1**



**Result 2**





## About splitting a track:

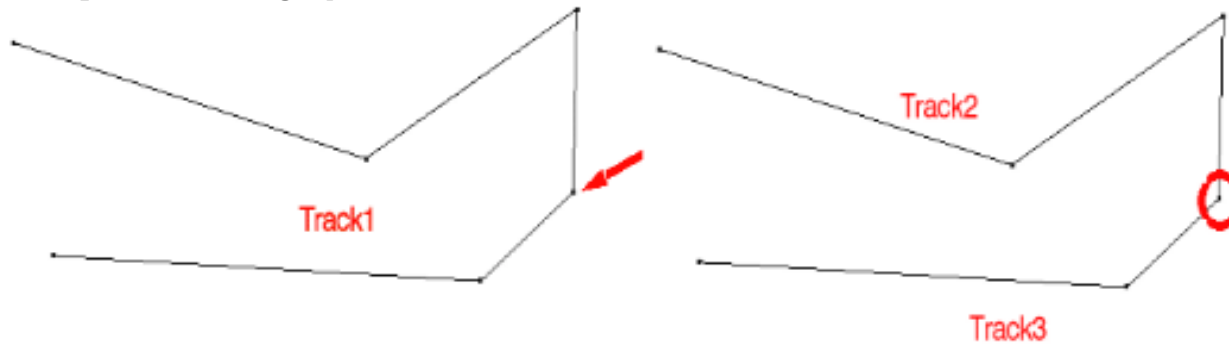
Lets you split a track into two new tracks. Two selections are available:

- if you select a point on the original track, the result will be two continuous tracks.
- if you select segments, the result will be two discontinuous tracks.

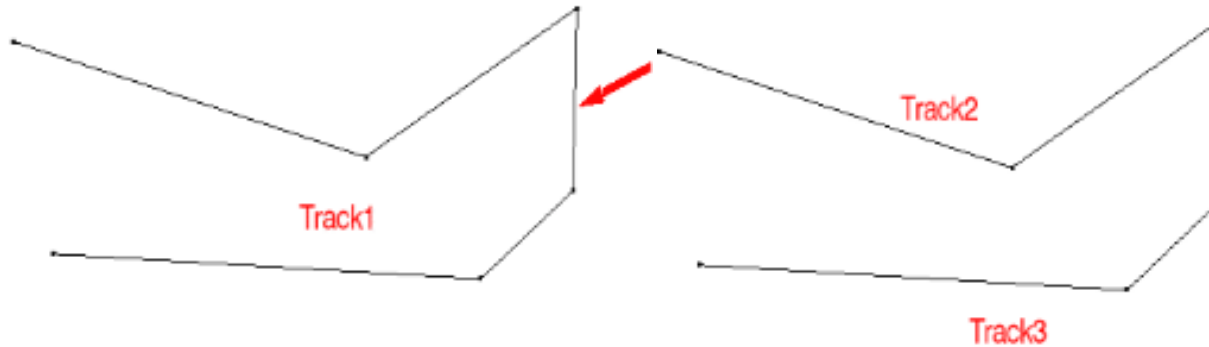
## What you need to do:

Right-click the track, select the track object item and select the Split track item from the contextual menu. Now, select a point or a segment: two new tracks are created. The initial track is swapped in no show space.

**Example1:** if selecting a point



**Example2:** if selecting a segment





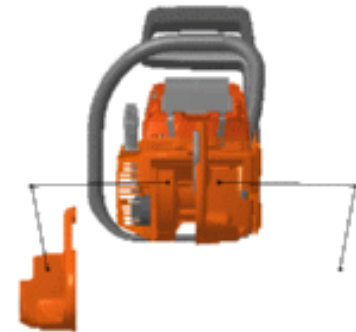
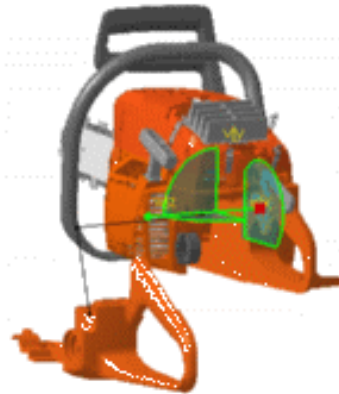
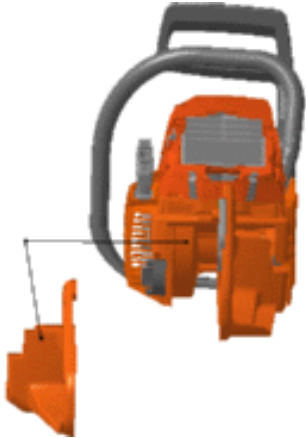
### About mirroring a track:

Lets you apply a symmetry on a track using a plane of your choice. Define the required plane using the 3D compass. See example below.

Position the 3D compass wherever you want, this position defines the symmetry plane with respect to the uOv plane of the 3D compass.

### What you need to do:

Right-click the track, select the track object item and select the Mirror track item from the contextual menu: a new symmetrical track is created and the initial track remains displayed in the geometry area

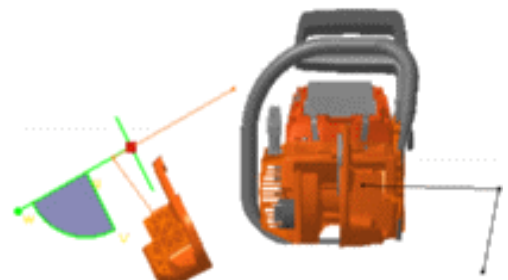
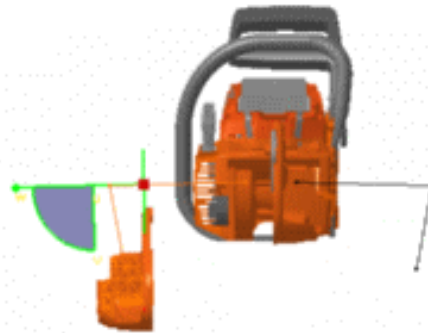
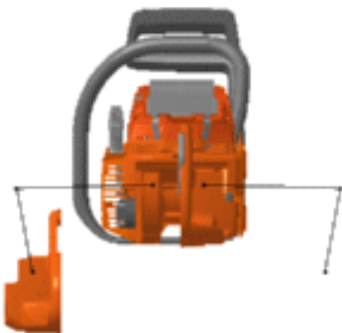


### About transforming a track:

Lets you reposition the entire tracks (translate and/or rotate) as any other object, using the 3D compass.

Drag and drop the 3D compass handle onto a track, then move the compass as you wish.

**Note:** you can select a track or multi-select tracks and their related objects



## About launching a path finder on a track:

Lets the system find automatically a dismounting path without any collisions.

This capability uses an input specification (that is the initial track) and results in a new track. The path finder uses the original positions on the track to optimize the path.

## What you need to do:

Select a track, click the Path finder icon in the DMU Check toolbar and click Apply: a new track is created, the initial track is swapped in no show space.

**Note:** The Update track capability is very useful when performing a path finder. Any modification of the initial track launches a new path finder computation, the resulting track is updated accordingly.

For more information about path finder, please read [Path Finder](#) section


## About smoothing:

Lets the system automatically get rid of the unnecessary positions without introducing any collisions.

This capability uses an input specification (that is the initial track) and results in a new track.

## What you need to do:

Select a track, click the Smooth icon in the DMU Check toolbar and click Apply: a new track is created, the initial track is swapped in no show space.

**Note:** The "Update track" capability  is very useful when smoothing. Any modification of the initial track launches a new smooth computation, the resulting track is updated accordingly.

For more information about smooth, please read [Using the Smooth Command](#)




## About launching a swept volume operation:

Lets the system compute automatically the swept volume of a moving part along a track.

This capability uses an input specification (that is the initial track) and results in a new swept volume which can be saved in different format types.

Select a track, click the Swept volume icon in the DMU Simulation toolbar and click Apply: a new swept volume is created, the initial track is remains displayed.

**Note:** The "Update track" capability  is very useful when performing a swept volume. Any modification of the initial track launches a new swept volume computation, the resulting swept volume is updated accordingly.

For more information about swept volume, please read [Swept Volume](#) section



# Analyzing in Track Context



This task depicts how analyses are bound to tracks and how are the analyses used.

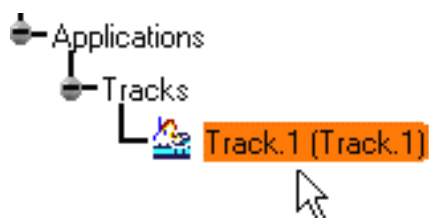
Open the [ANALYZE\\_TRACKS.CATProduct](#) document.



A track is already defined on the DEMO\_CGE\_CHAINSAW\_BODY\_TANK\_RSIDE.1 object and interferences specifications are also defined in your document.

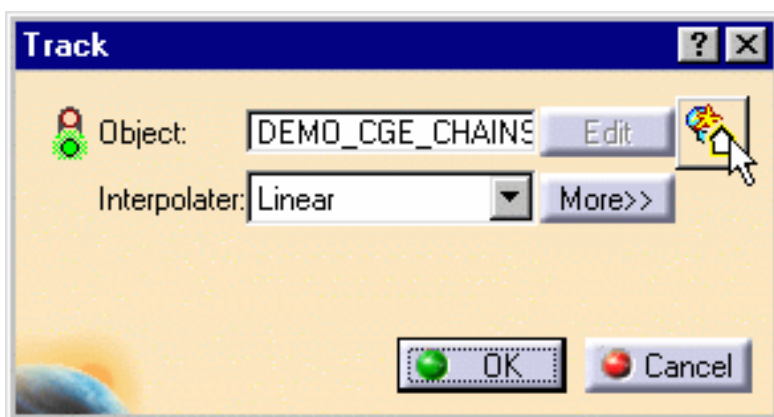


1. In the specification tree or in the geometry area, double-click Track.1

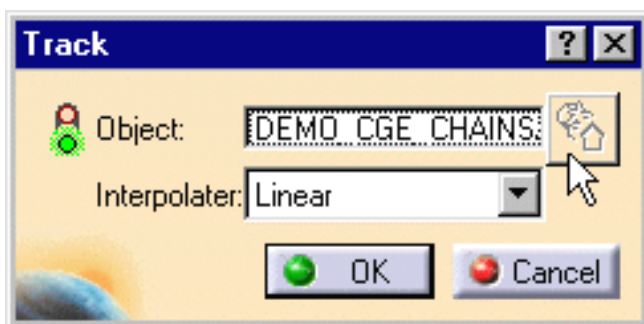


- 2.

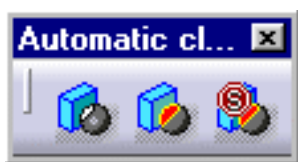
In the Track dialog box, click the "clash on track" button  to display the analysis list.



This button is grayed out if there aren't any analyses (interferences and distances) defined in your document

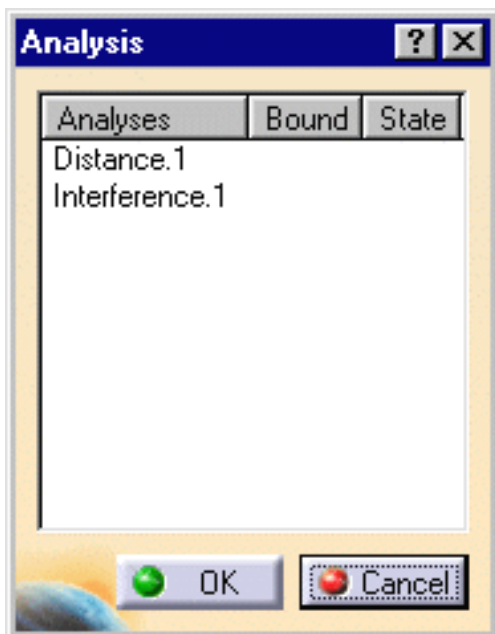


But you can still perform automatic clash detection (without defining a clash specification): select your track and click the arrow within the Clash Detection icon from the DMU Check toolbar. Undock the toolbar if necessary. Activate Clash detection (on or stop modes), then simulate your track.



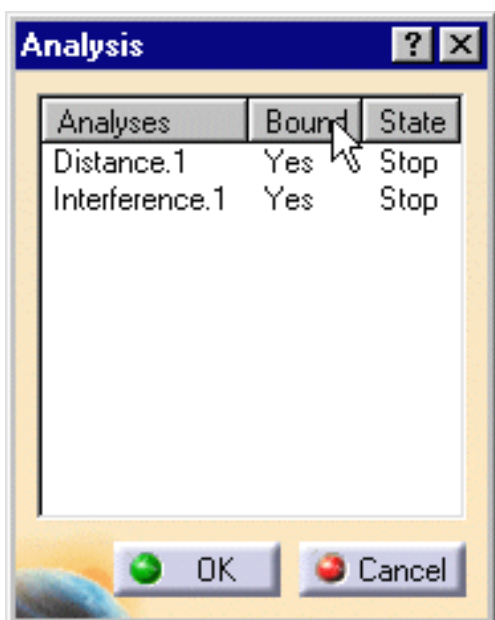
Please refer to [Validating Positions Automatically](#)

The Analysis dialog is displayed:

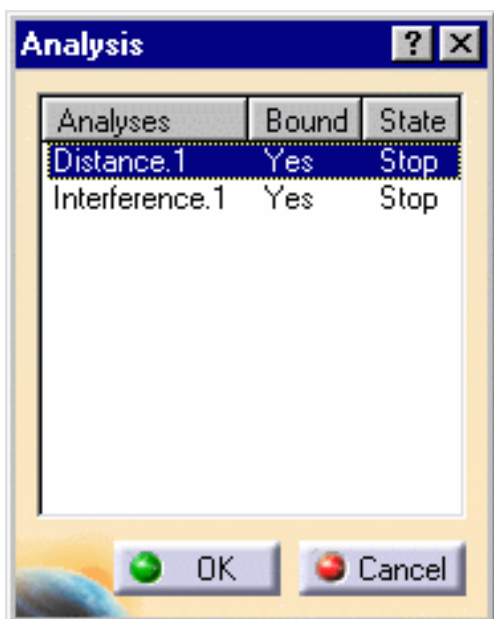


3. Click the Bound item to link Distance.1 and Interference.1 to Track.1. (Click the Bound item again to cancel the operation)

**Note:** you can also select the analyses one after the other or multi-select them to bound them

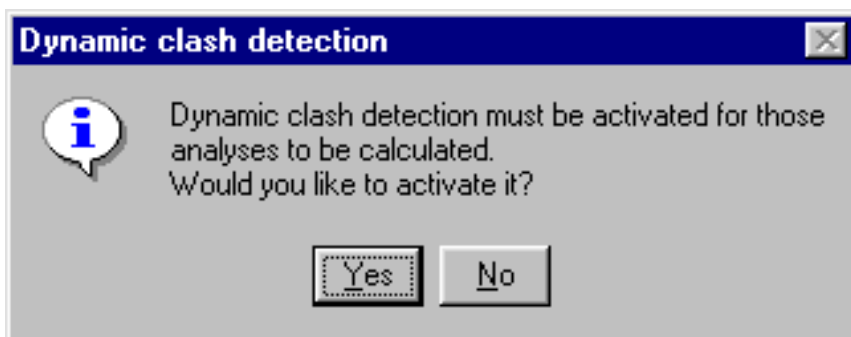


4. Click the State item if you want to change the detection mode (stop, on, off). The default mode is "Stop". Read [More about clash detection in track context](#) to better understand the combination of the analyses states and the clash detection modes



5. When satisfied (i.e. keep the default state Stop) click OK in the Analysis dialog box.

A Dynamic clash detection information message is displayed letting you activate automatic clash detection




6. Click Yes to activate the Clash Detection

The Stop mode is automatically activated

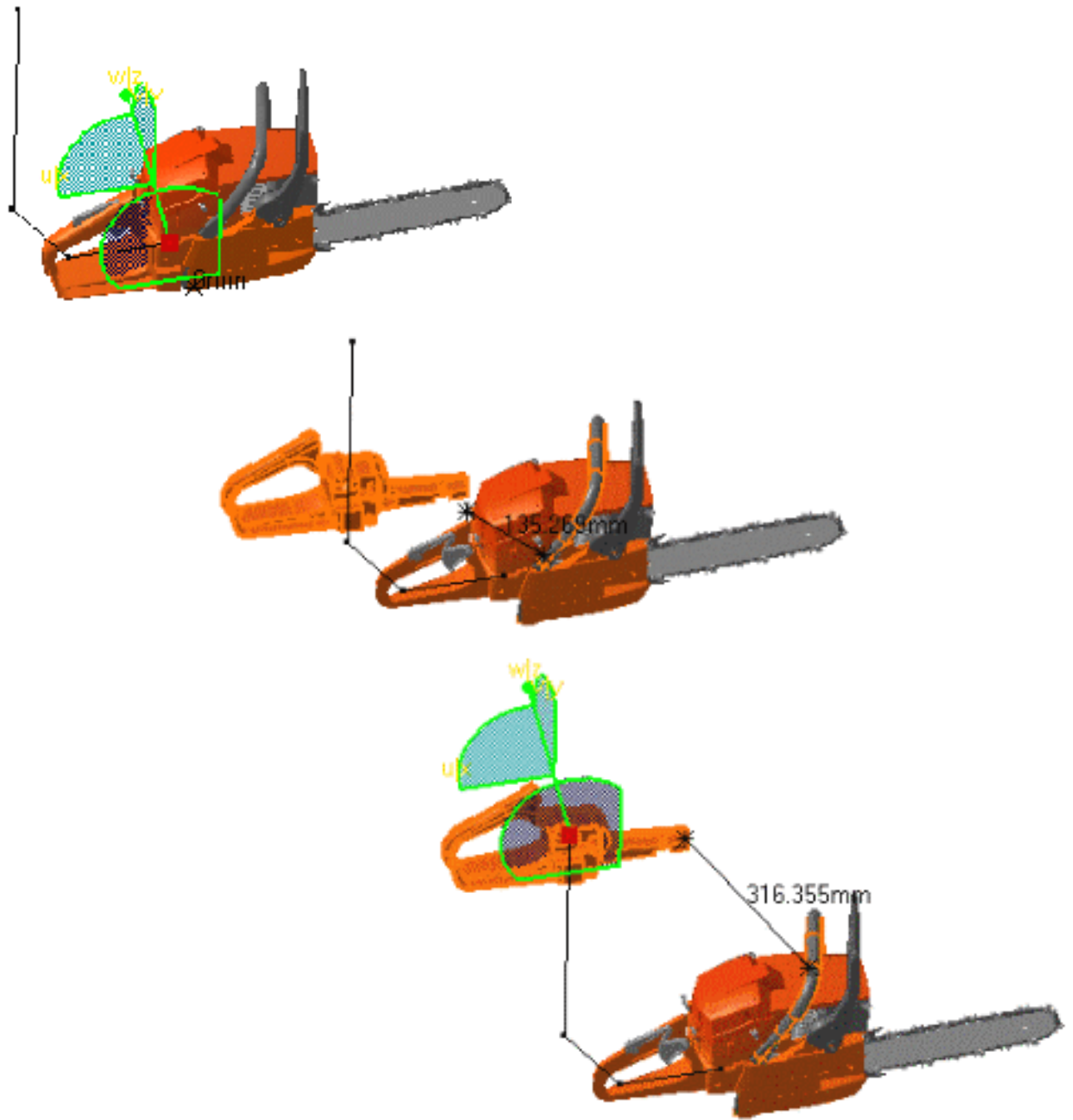


: the simulation stops at the first clash, the products are highlighted




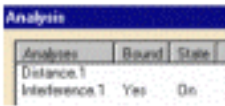





7. Click the Play forwards button  in the Player

The results are shown below:

The product in clash is highlighted and the distance is calculated at each position.



**More about clash detection in track context:**

<div> <div>1</div> <div>2</div> </div>	ON 	STOP 	OFF 
			
ON 	Calculation only	Calculation only	No calculation
STOP 	Calculation only	Calculation + Stop (if collisions detected)	No calculation
OFF 	No calculation	No calculation	No calculation

1

specifies the states in the Analysis dialog box (stop is the default mode)

2

specifies the clash detection modes ( clash detection toolbar)

**Note:** if the detection mode is set to off (by default) and analyses are bound and set to on, there is no calculation.

**Example:** you have a track containing analyses (see [bound tab-Step .3](#)) (let's say two interferences)

Interference 1 is set to stop

Interference 2 is set to on



Clash detection mode selected: (Stop mode)

You simulate your track, the track is stopped when interference 1 is detected, interference 2 is calculated and displayed but does not stop the track simulation.



# Exporting and Importing Tracks in Neutral Format



## About XML:

XML (Extensible Markup Language) is a standard format that can be used as a simple way to exchange data. The advantage of using this format to import and export tracks is two-fold:

- Results and how the results are presented are two separate entities, so you can customize your own style sheet to present the results the way you want without impacting the results themselves.
- XML makes documents smarter and more portable. You can for example export and publish the same results to a web browser as well as to your company database.

**Note:** XML processors letting you generate HTML pages corresponding to the XML file and style sheet are available on the market.



This task show you to import and/or export track definitions in neutral format and how to use them.

Two neutral formats may be exported:

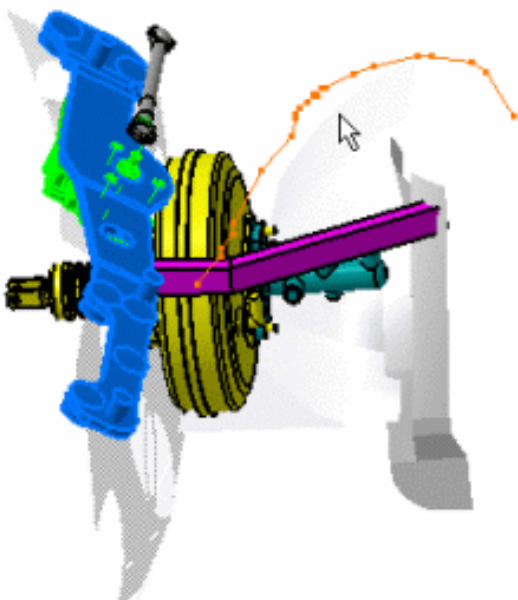
- .xml and
- a text format, which has a .fitting extension. The text file provides information on what object the track is associated with and a description of each recorded shot. If the track- associated object is a product or a shuttle, additional information also includes: file path or VPM identifiers to the associated parts, and relative position of parts to the shuttle axis.



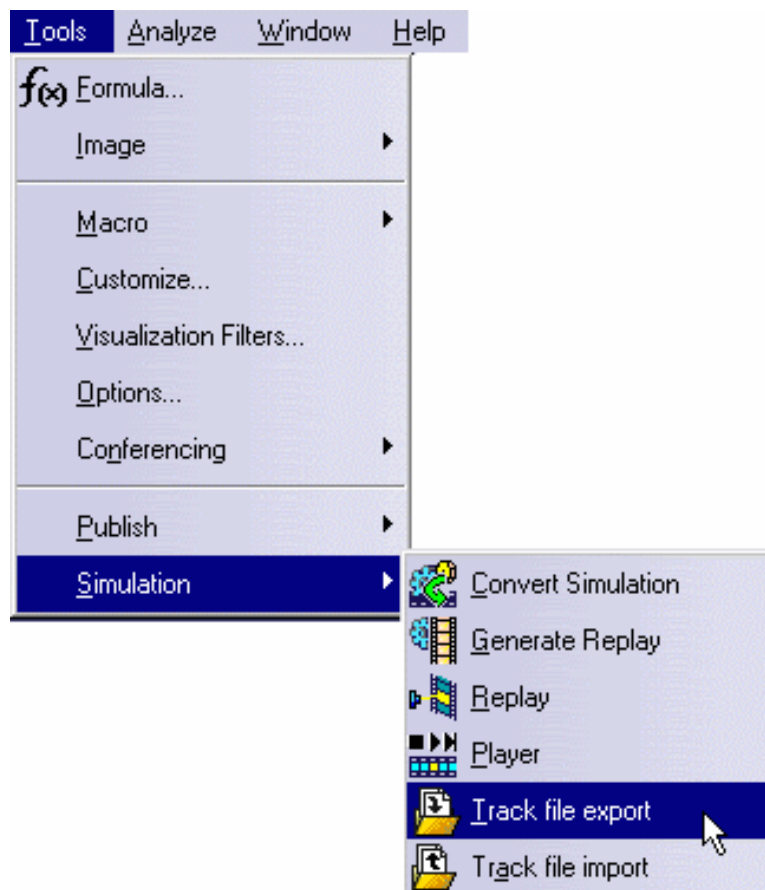
Open the [EXPORT\\_V5\\_TRACK.CATProduct](#) document.



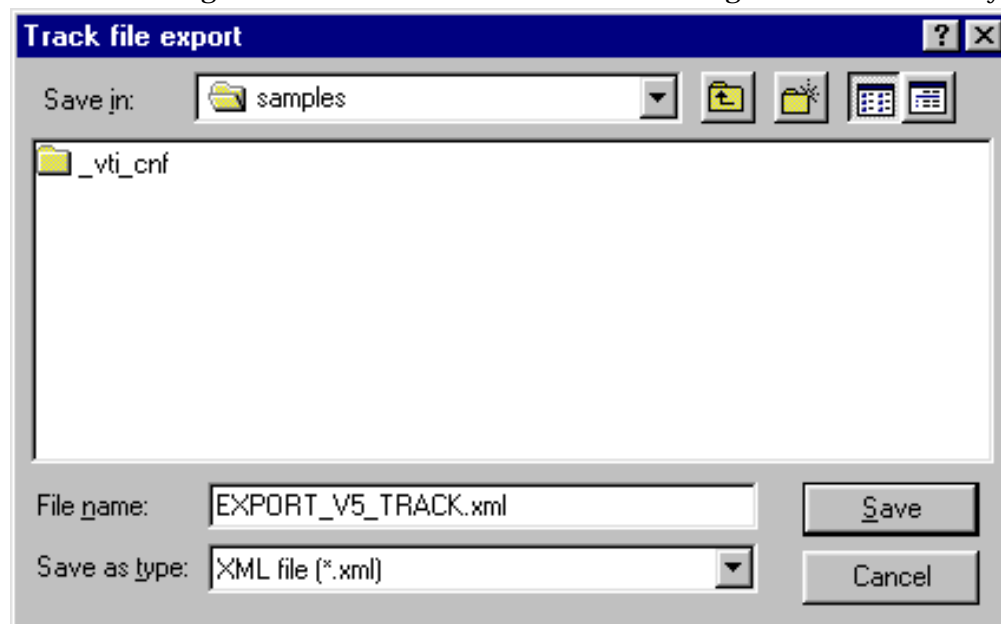
1. Select Track.2 either in the specification tree or in the geometry area





**2. Select Tools->Simulation->Track File Export**

The Track File Export Track dialog appears:

**3. Enter a meaningful name and select .xml format or .fitting from the Save as Type drop-down list:****4. Open the [EXPORT\\_V5\\_TRACK.xml](#) document in an internet browser you have just created**

This is what you obtain:

```

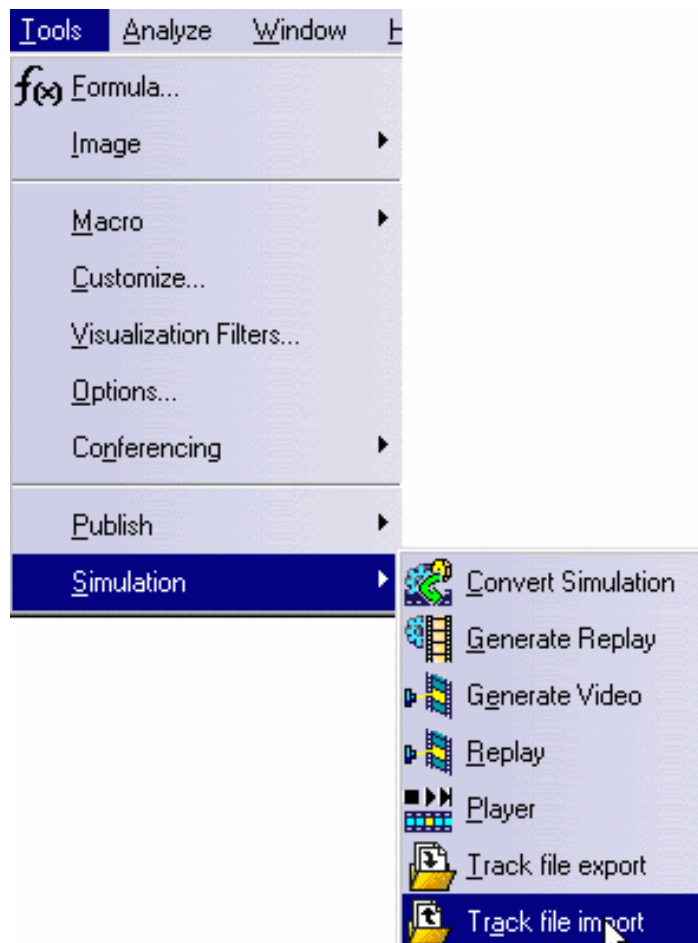
<?xml version="1.0" encoding="ISO-8859-1" ?>
- <Tracks>
- <Shots id="Track.2:479ec6e3_1b3_3bdeb0e1_14ba" name="Track.2" version="1.0">
- <!--
    Positions are expressed relatively to the track coordinate system.
    assuming that :
        pLocal is a local position in the track coordinate system (a recorded shot)
        pGlobal is a position in the track associated object father coordinate system
    You have : pGlobal = inverse(guaranteed) * pLocal * frame
-->
- <Shot name="Shot.332" time="0">
- <object_position>
  <V1 value="1.000000000000037" />
  <V2 value="0" />
  <V3 value="0" />
  <V4 value="0" />
  <V5 value="1.000000000000037" />
  <V6 value="0" />
  <V7 value="0" />
  <V8 value="0" />
  <V9 value="1.000000000000042" />
  <T1 value="0" />
  <T2 value="0" />
  <T3 value="0" />
</object_position>
</Shot>
- <Shot name="Shot.333" time="41.8185854824074">
- <object_position>
  <V1 value="0.944316688739503" />
  <V2 value="0.0213635842798416" />
  <V3 value="-0.328343705034147" />
  <V4 value="-0.0210991802136647" />
  <V5 value="0.999767844156653" />
  <V6 value="0.0043683389766942" />
  <V7 value="0.328360801502038" />
  <V8 value="0.00280268760678754" />
  <V9 value="0.944548214216689" />
  <T1 value="-27.976818307448" />
  <T2 value="-2.58958666569833" />
  <T3 value="30.9739530903398" />
</object_position>
</Shot>
- <Shot name="Shot.334" time="50.3567512823416">
- <object_position>
  <V1 value="0.911683928008618" />
  <V2 value="0.0433417028664075" />

```

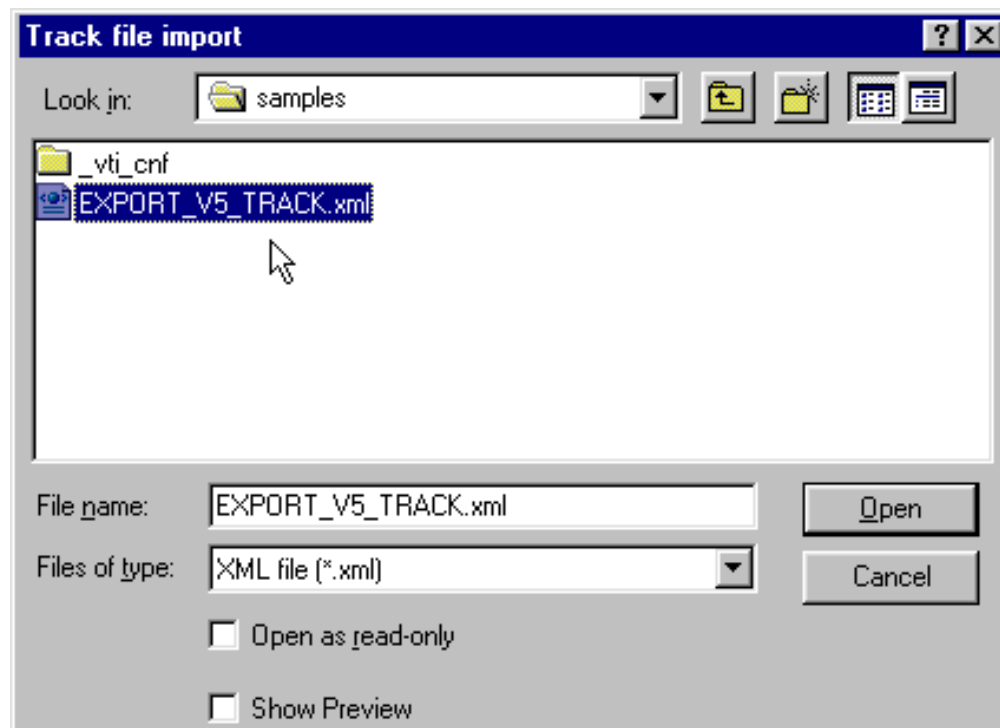


1. Now open the [IMPORT\\_V5\\_TRACK.CATProduct](#) document.

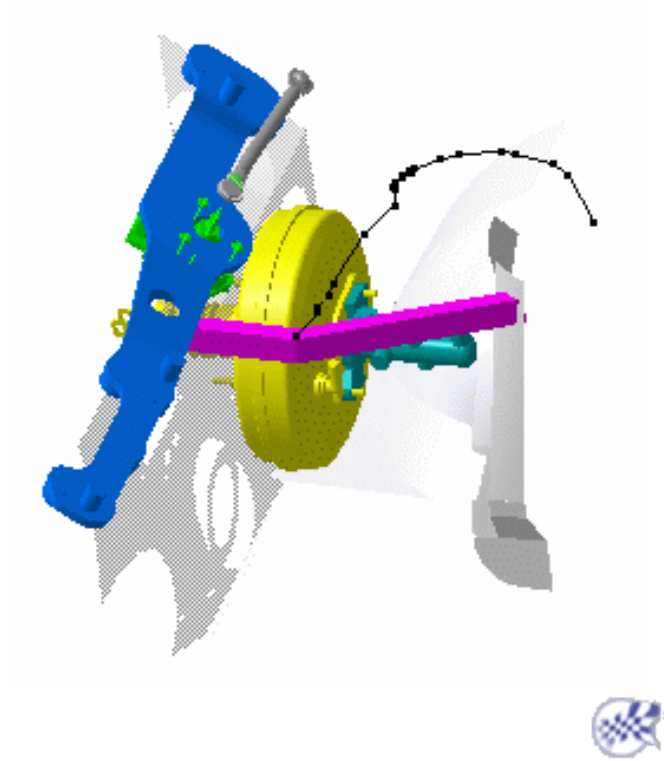
Select **Tools->Simulation->Track File Import**



2. Select [EXPORT\\_V5\\_TRACK.xml](#) document in the samples folder



This is what you obtain:



# Editing Time Line in Tracks



When creating a track, the duration of each segment is, by default, calculated to keep a uniform speed over the whole track.

It is now possible to change the duration of each segment, you can

- edit segment duration within the Track dialog box using the More button
- modify quickly the segment duration using drag and drop capability
- enter a precise value to modify this duration.



This task show you to edit time line in track command for demo purposes. It can be useful to differentiate simulation segments editing each and every segment duration (i.e. speeding up or slowing down depending on the part you are being dismounting



Open the [CHAINSAWAT.CATProduct](#) document.



1. Double-click Safety\_Handle\_Track (Track.1) either in the geometry area or in the specification tree.

2.

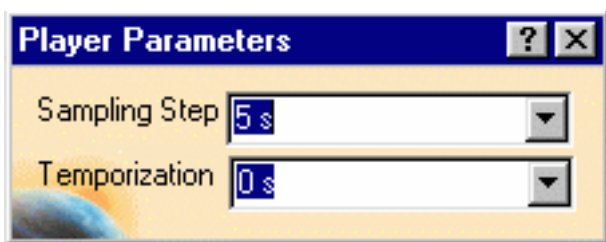


Click the Parameters icon in the Player

The Player Parameters dialog box is displayed:

3.


Enter 5 s in the sampling step field

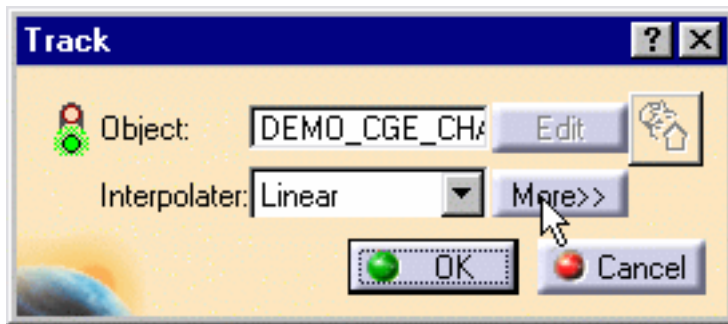


4.

Play your track simulation using the play forward button  from the player

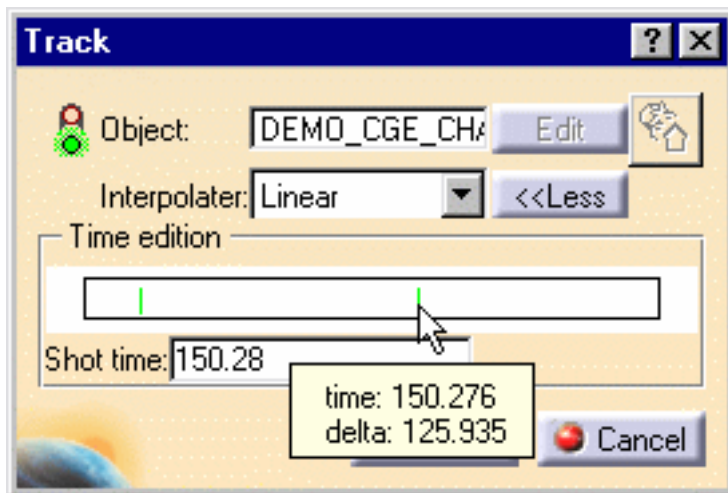
5.

Click the more button 

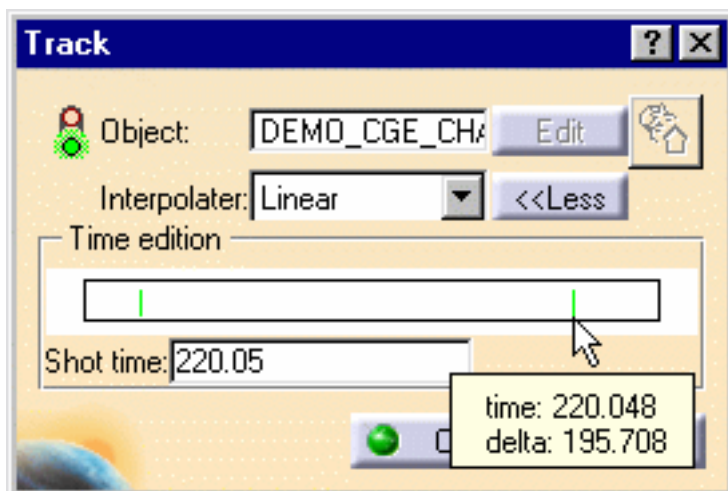


*You are going to reduce the last segment duration*

**3. Select the last segment**

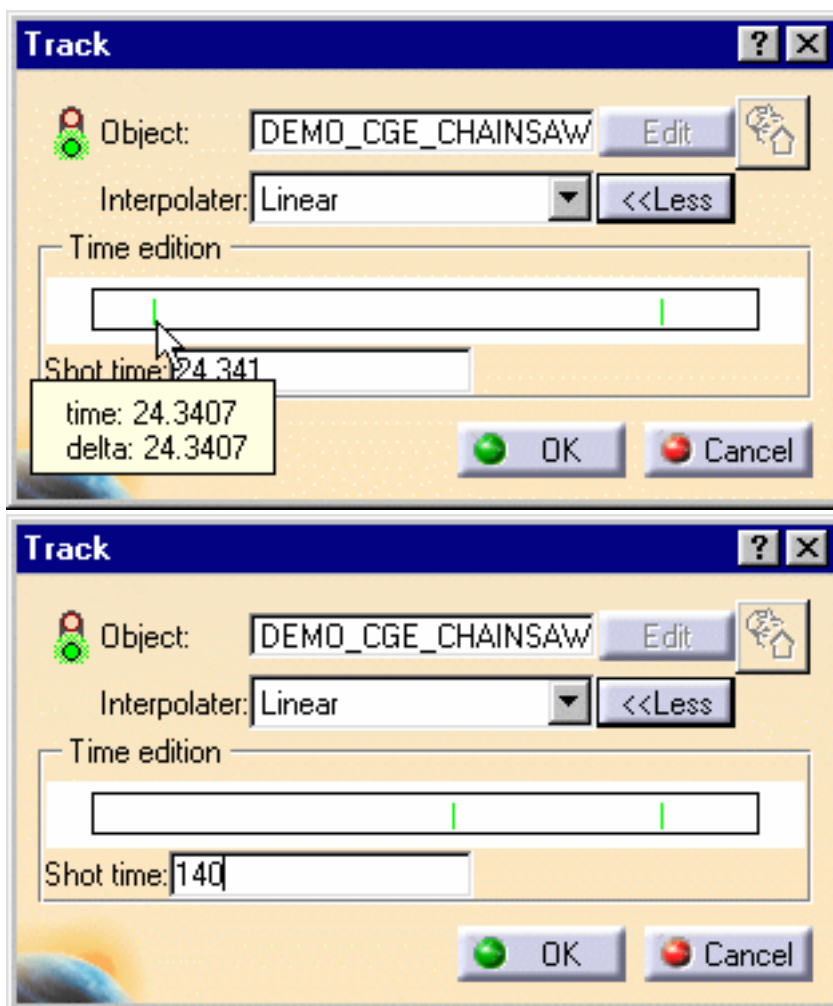



**4. Drag the last shot which duration is 150.276 s to approximately 220**



**5. Drag the first shot from 24.34. to approximately 140**

or if you want a finer accuracy: enter the value in the shot time field and press ENTER key.



6. Play your track simulation again using the play forward button  from the player.

If you are not satisfied with the result, modify the duration again.

The track simulation is slow at the beginning and speeds up in the end.



## Keyboard Shortcut

Select + CTRL key: lets you drag each and every segment of the time line representation without changing the global duration



# Copying and Pasting Tracks



This task illustrates the use of the copy/paste capability. It can be very useful to apply the same track to another object which has the same dismounting path.



Open the [COPY\\_PASTE\\_TRACKS.CATProduct](#) document.

A track is already defined on the DEMO\_CGE\_CHAINSAW\_BODY\_TANK\_RSIDE.1 object

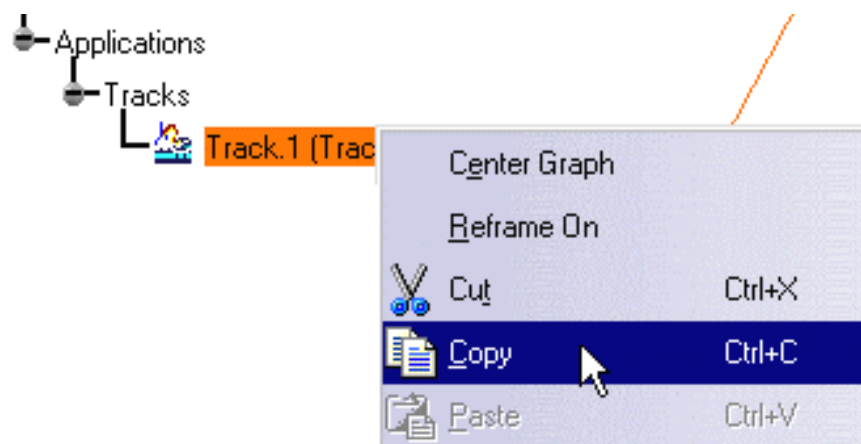


1. In the specification tree or in the geometry area, select the track you wish to copy. In our example, select Track.1



2.

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.



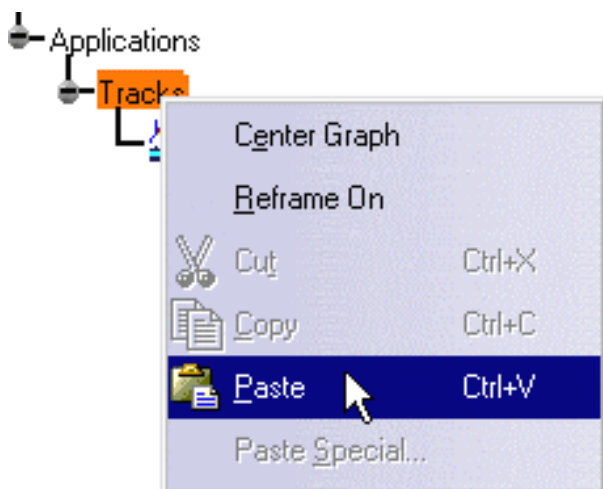
3. Select Tracks item under Applications in the specification tree.

4.

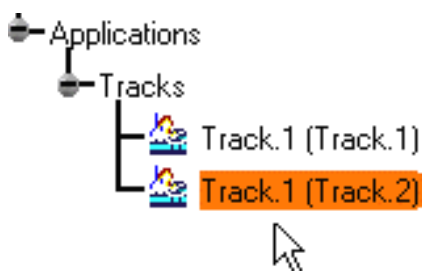
Now either click the Paste icon, select the **Edit->Paste** command or select the **Paste** command in the contextual menu.

This operation recovers the data previously put in the clipboard.





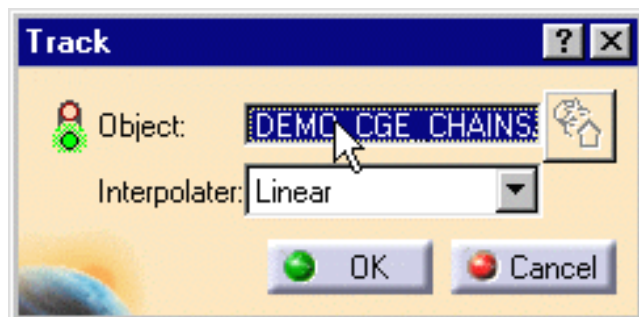
The track is pasted:



5. Double-click the pasted track, in our example: Track.1 (Track.2) to modify the object selection

The Track dialog box along with the DMU recorder and player are displayed:

6. In the Track dialog box, click the Object selection field



The Track positioning dialog box appears:  
In our example, we will keep the default option set

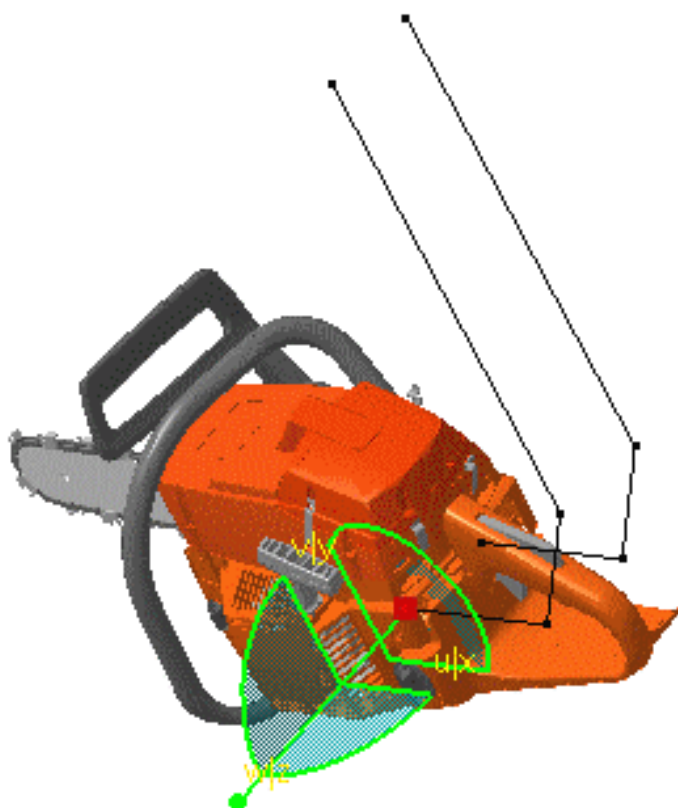


For more detailed information, please refer to [Track Editor and Recorder](#)



7. Select DEMO\_CGE\_CHAINSAW\_BODY\_TANK\_LSIDE.1 object either in the specification tree or in the geometry area



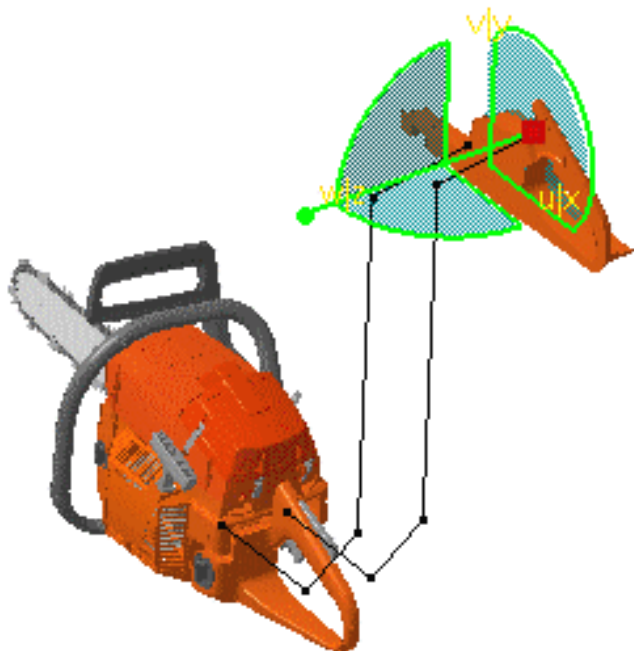
This is what you obtain:



The two tracks are linked together.  
For example if you insert a new shot in the first track:

8. Double-click Track.1(Track.1) in the specification tree.
9. Click the Skip to end button  in the Player and insert a shot (moving the 3D compass).  
When done click .


The shot is automatically added in the second track. The two tracks are synchronized.



10. Click Ok to confirm your operation



The **modified position is kept** when exiting the track command (dismounted).

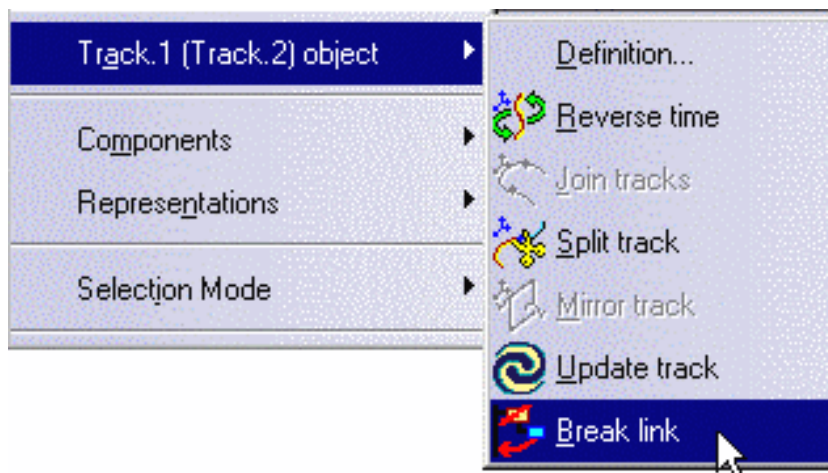
If you need to restore the initial position click the Reset icon  from the DMU simulation toolbar.



Remember you can break the link between the two tracks using the Break link functionality



You can access it via the track contextual menu





Be careful because the result depends on the tracks selection order.



# Sequences

Sequences have been enhanced: we recommend you to read the Sequence section in its overall to discover more about:

- scheduling actions: "iterative create last step and add" mode
  - interface change for action duration
  - Delay option
  - Color and transparency actions
- To have a look, please click on the appropriate keyword in the [What's New](#) section (items under Sequences)



[About sequence capabilities](#): provides background information about sequence capabilities.

[Sequence editor](#): provides information about the sequence editor

[Display Gantt chart](#): provides background information about the Sequence Gantt Chart.

[Define a sequence](#): Click the Edit Sequence icon, add actions, sequence them, modify the actions duration if necessary. Click the Edit Analysis tab and add interferences and distances. When satisfied click OK.

# About Sequence Capabilities



A sequence is a way to put together and schedule actions to perform simulations.

Sequences are persistent and can be stored in your document.


## What is an action?

Actions are entities of different nature organized within the sequence.

They can be objects from the following list:

- tracks (camera tracks, product tracks, shuttle tracks, section plane tracks, light tracks) please refer to [About Tracks Capabilities](#)

- color and transparency actions (  )

- visibility actions i.e. Show/Hide (  )

- simulations (R6 simulations)

- sequences (  )

- *FEA Analysis*

Please read "*DMU Engineering Analysis Review*"- Animating Images

- mechanisms which can be simulated with laws

Please read "Running Mechanisms within a Sequence" in the *DMU Kinematics Simulator User's Guide*

## About actions duration:

Actions are characterized by a duration:

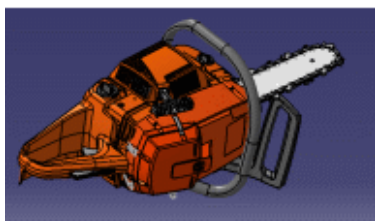
i.e. Track duration is linked to the trajectory length

## About Visibility and color actions

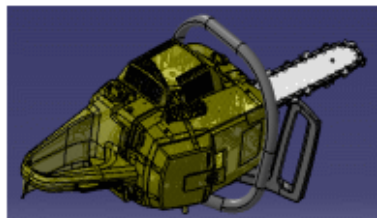
Using a a color/transparency action, you could only define an initial and a final state. This highlight aims at offering more capabilities for Color/Transparency actions based on the track model:

- recording of multiple states for the same action using the standard graphic properties toolbar.
- possibility to change the object on which is applied the action
- recording capabilities using the same Recorder tool than the track
- time edition of the action (possibility to change the duration of each segment).
- VB exposition
- The visibility or color actions are created in sequence context and their effects are seen only when you use the Player (in the sequence for instance)
- color and transparency actions have now a duration
- Visibility actions are instantaneous (duration=0)
- For more information, please refer to [Defining a Sequence](#)

Initially the product looks like this:

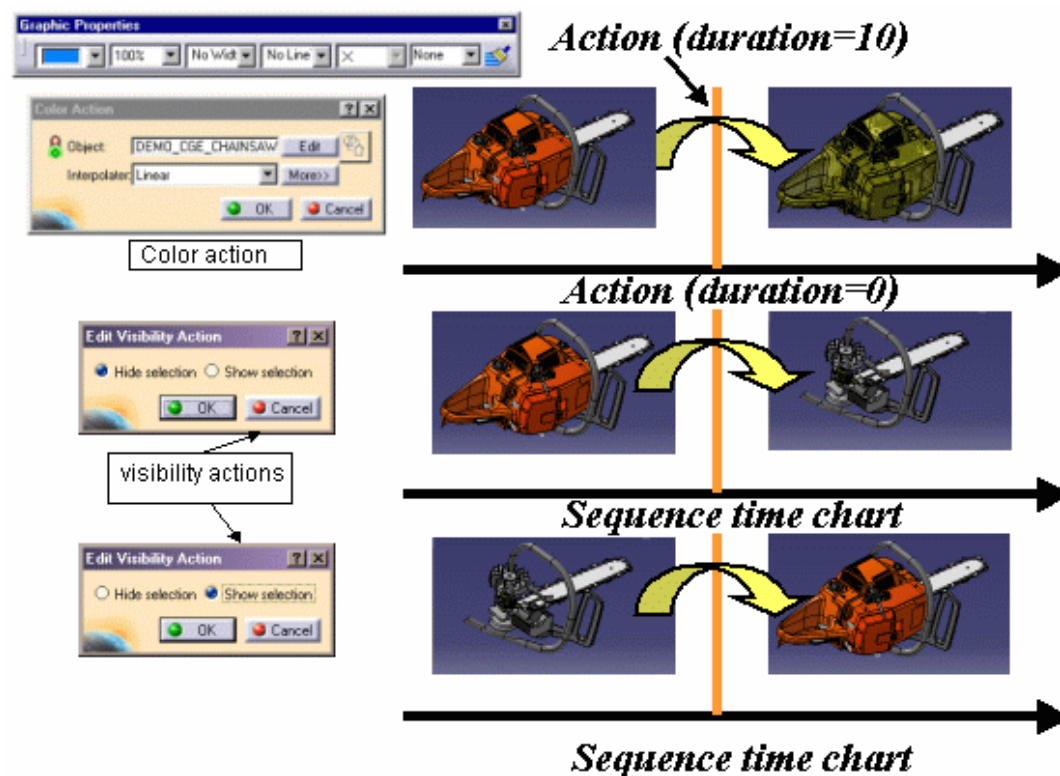


when creating a color action in the sequence, it becomes:



Make sure though, you schedule this action properly using the Action Delay parameter (i.e. depending on the effect you want to obtain)

The picture below gives you the various results after action creation according to the option set:



This duration is recovered in the sequence but you can also apply a specific duration in sequence context.

#### Example:

Duration in the player of a sequence comprising two tracks:

The two actions are scheduled to start one after the other (see: [sequencing modes](#))

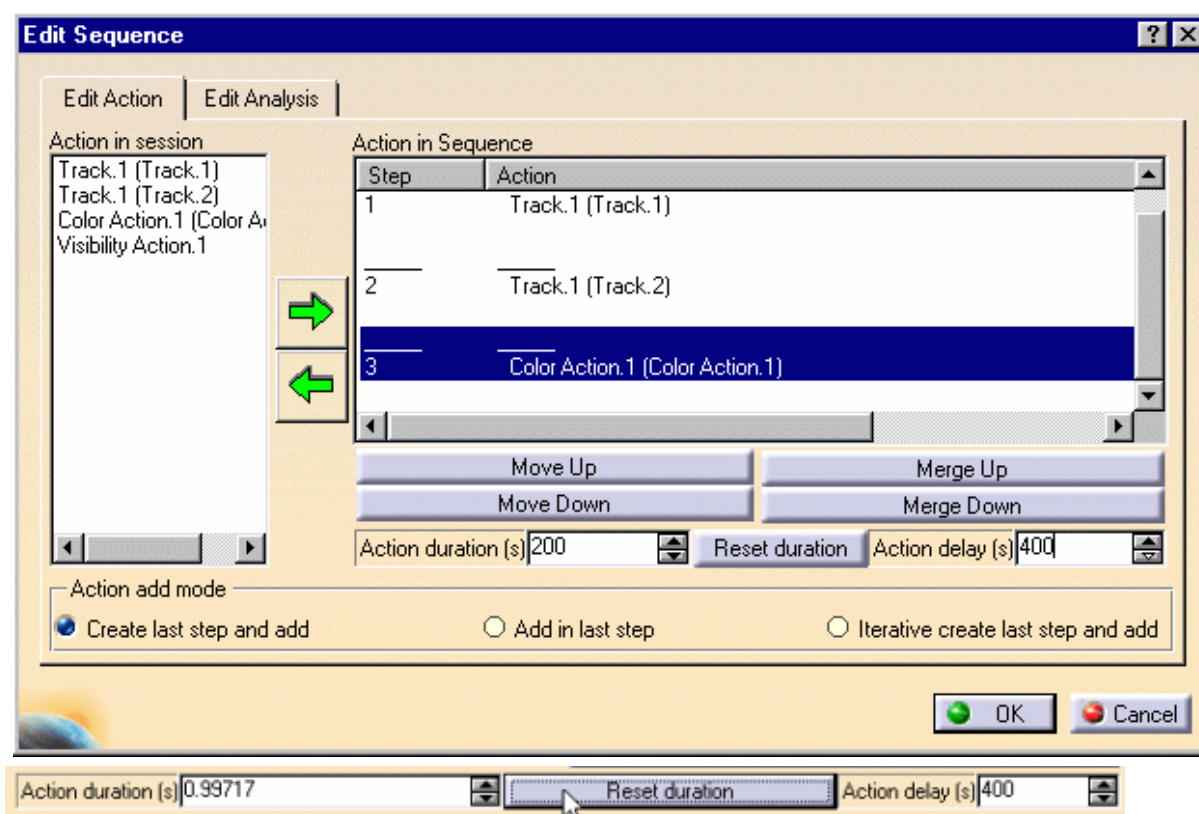


#### About Action Modification

You can modify the action duration, all you need to do is:

- select the action in the Actions in Sequence list
- Enter the new value in the action duration (i.e. 200)
- If you need a delay, enter a value in the Action delay field (i.e. 400)
- You can use the Reset button to swap to the default action value (intrinsic duration)





### About sequencing

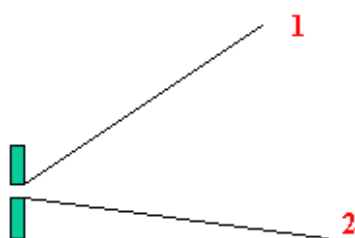
Sequencing aims at defining a time frame within which the actions are scheduled.

**Two sequencing modes are available:**

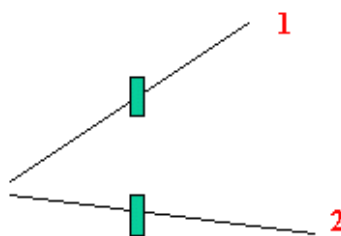
- actions start together (**simultaneous mode**)
- actions start right one after the other (**consecutive mode**)

#### Simultaneous mode

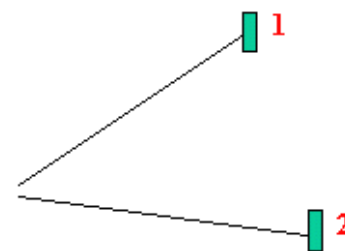
**Beginning**



**Middle**



**End**



#### Consecutive mode

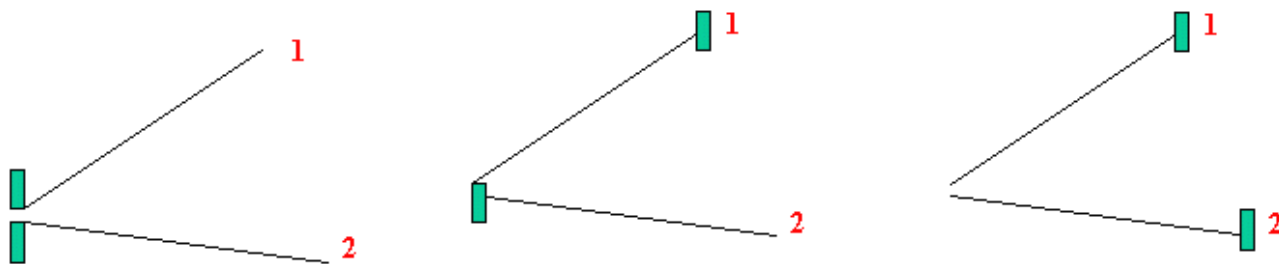
**Beginning**

**Middle**

**End**





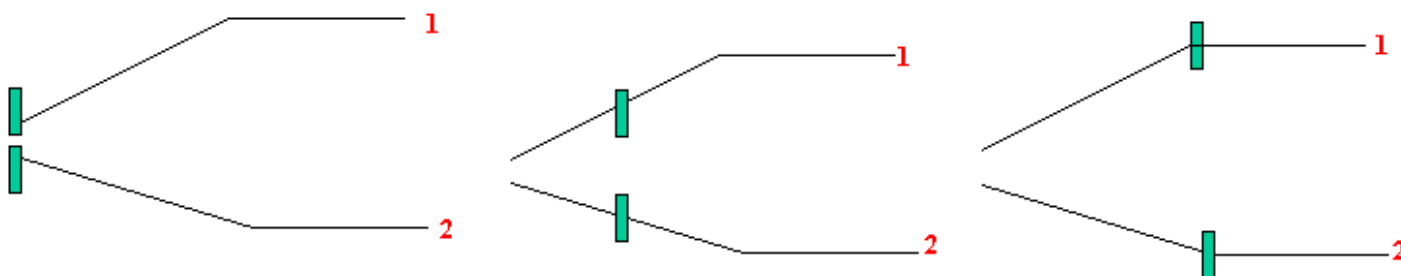


You can combine the two modes and modify the scheduling at any time.

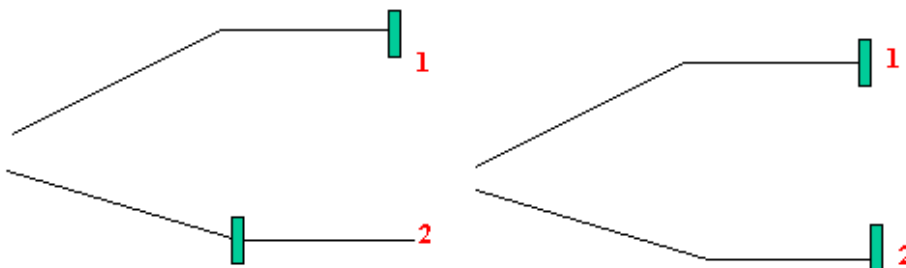
Sequences lets you put together existing actions, anyway, you can easily create a new action on the fly and add it in the current sequence. This capability lets you edit actions in context and synchronize meeting points in different actions.

The example below illustrates the two modes combination:

### Simultaneous mode



### Consecutive mode



### About Journaling/automation

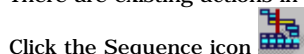
Sequences are journalized. You can generate a macro using **Tools->Macro->Record...** (see the Infrastructure user's Guide)



### Sequence creation:

3 methods are available to create sequences:

1. There are existing actions in your document (actions in session list),



Click the Sequence icon, and add them using the arrow into the Action in sequence list and schedule them, using the sequencing tools (Refer to: [Sequence editor](#))

2. Open an empty sequence and create actions on the fly (the sequence editor remains opened)
3. Combine the two methods (1. and 2.)

Sequences created in this manner are persistent and can be stored in the document. They are listed as separate entities in the specification tree and can be selected at any time and modified.



# Sequence Editor



The sequence editor lets you manage and simulate **actions** from the following:

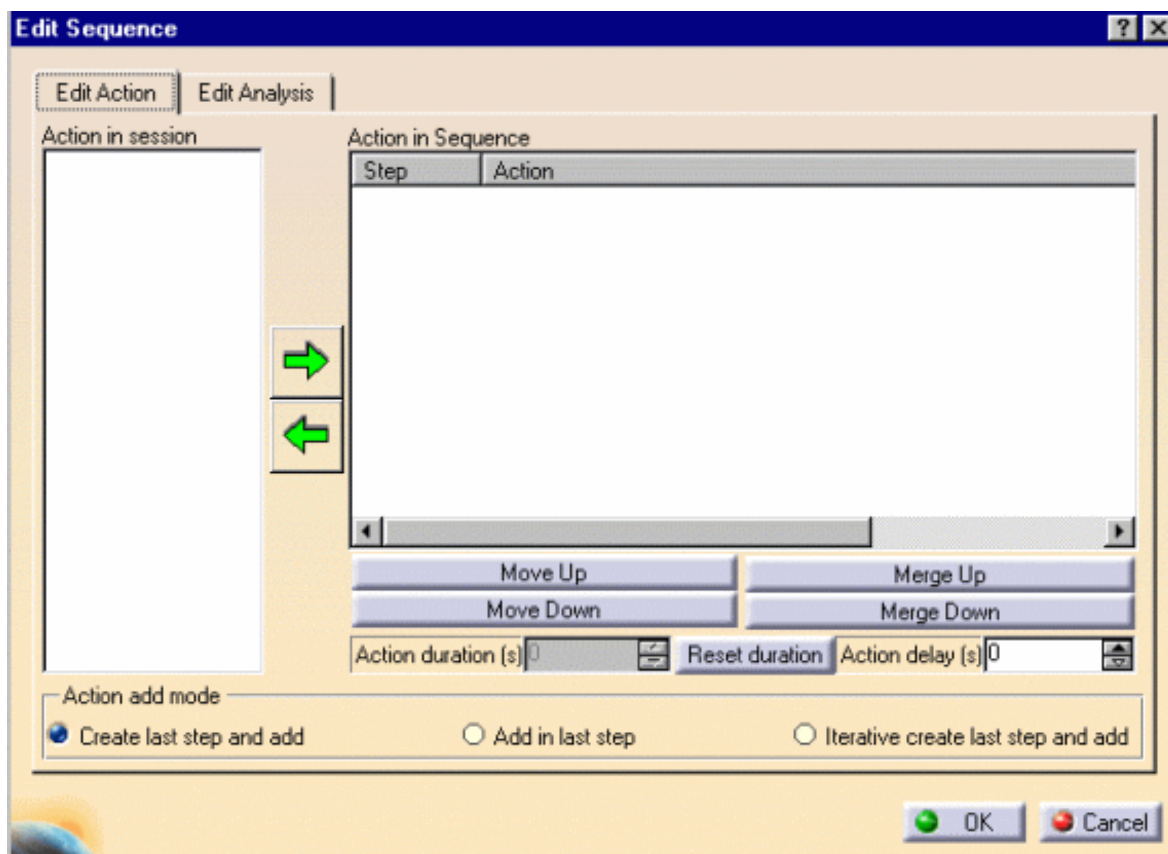
- moving objects (part, camera, ...)
- graphic attributes ( show/hide, colors, transparency)

You can also manage time with Gantt chart

Let's look at it more carefully:

The Edit Sequence dialog box comprises of two tabs:


- [Edit Action tab](#)
- [Edit Analysis tab](#)

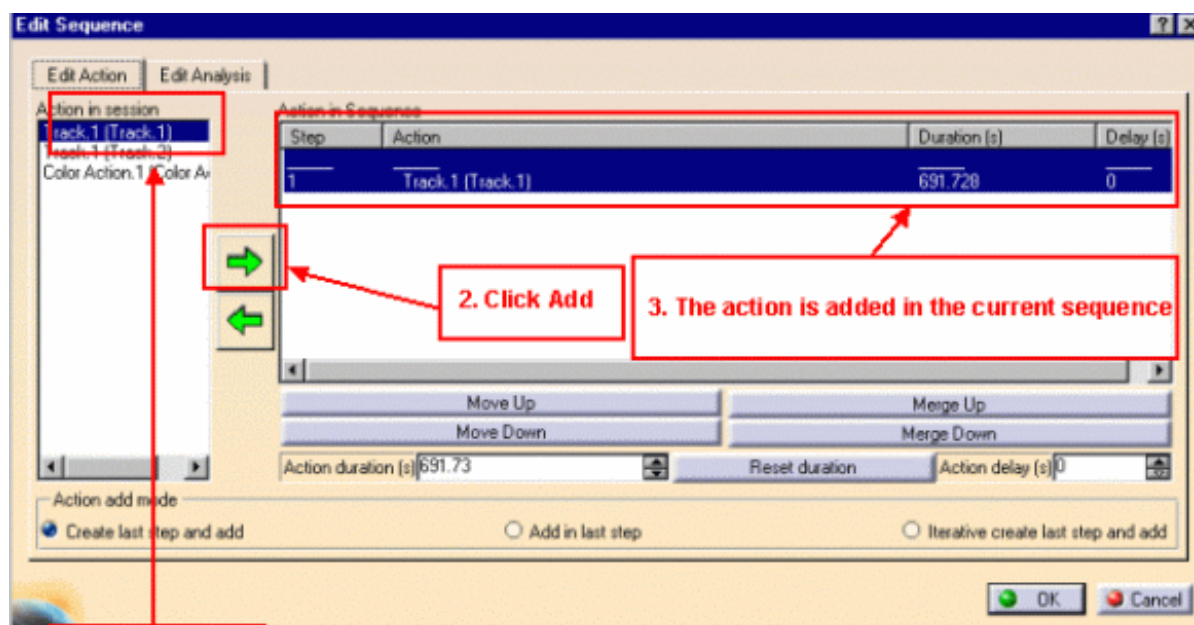


The Edit Action tab lets you perform the following operations:

- Add/remove actions using the green arrows



**1.** Select an action in the "Action in session" list and click . The action is added in the sequence list. See example below:




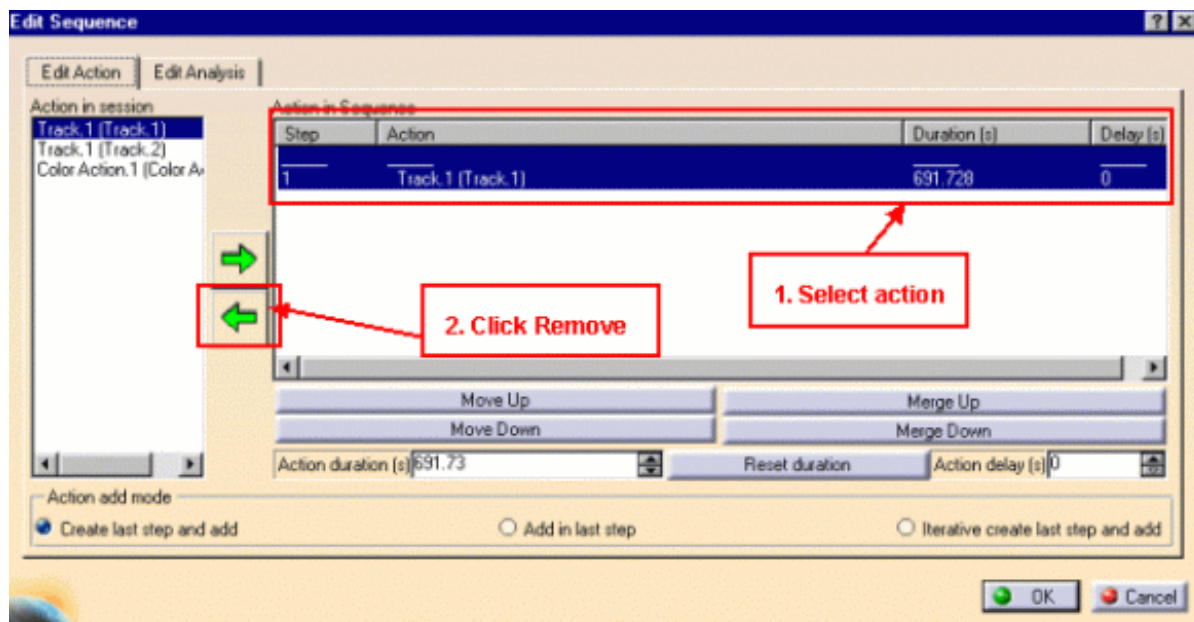
1. Select action

2. Click Add

3. The action is added in the current sequence



2. Select an action and click . The action is removed from the "Action in Sequence" list



1. Select action

2. Click Remove

**Note:** If you multi-select actions in the session list and click the Add button, the actions are added in simultaneous mode.

- Customize the add mode settings:
  - (1) **Create last step and add** option: creates a last step and add the selected action into it. (default mode) (in **consecutive mode**)
  - (2) **Add in last step** option: lets you add an action in last step (in **simultaneous mode**)
- A new add mode appears
  - (3) **Iterative create last step and add** option: lets you add the actions in consecutive steps (1-2-3...)

(1) Track.1 (Track.1)  
Track.1 (Track.2)  
Color Action.1 (Color Action.1)

Step	Action
1	Track.1 (Track.1)
2	Track.1 (Track.2)

(2) Track.1 (Track.1)  
Track.1 (Track.2)  
Color Action.1 (Color Action.1)

Step	Action
1	Track.1 (Track.1)
1	Track.1 (Track.2)

(3) Track.1 (Track.1)  
Track.1 (Track.2)  
Color Action.1 (Color Action.1)

Step	Action
1	Track.1 (Track.1)
2	Track.1 (Track.2)
3	Color Action.1 (Color Action.1)

- Sequence actions using:

if working in **consecutive mode**

- Move up:** moves up a selected action
- Move down:** moves down a selected action

if working in **simultaneous mode**

- Merge up:** merges the selected action up
- Merge down:** merges the selected action down



**Note:** remember you can combine the two modes within the same sequence.  
Please read [About Sequence Capabilities](#)

- Customize the action duration
  - action duration:** the numerical field lets you enter a specific duration for an action (this capability enables to simulate the same action with a different time scaling)
  - reset duration** lets you reset the selected action to its intrinsic duration

Action duration (s) 200    Reset duration    Action delay (s) 400

Action duration (s) 691.73    Reset duration

Reset default duration

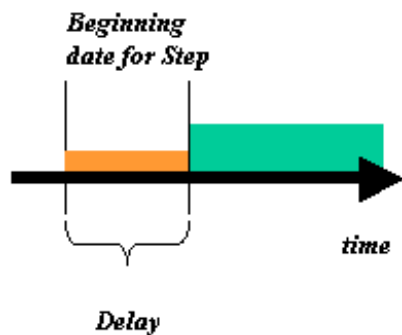
**Note:** you can enter a specific duration for all action types except visibility actions which are instantaneous (duration=0)

Action duration (s) 0

- A new option is available :
  - **action delay**: lets you delay the starting time of an action (i.e. it is now possible to overlap two actions)

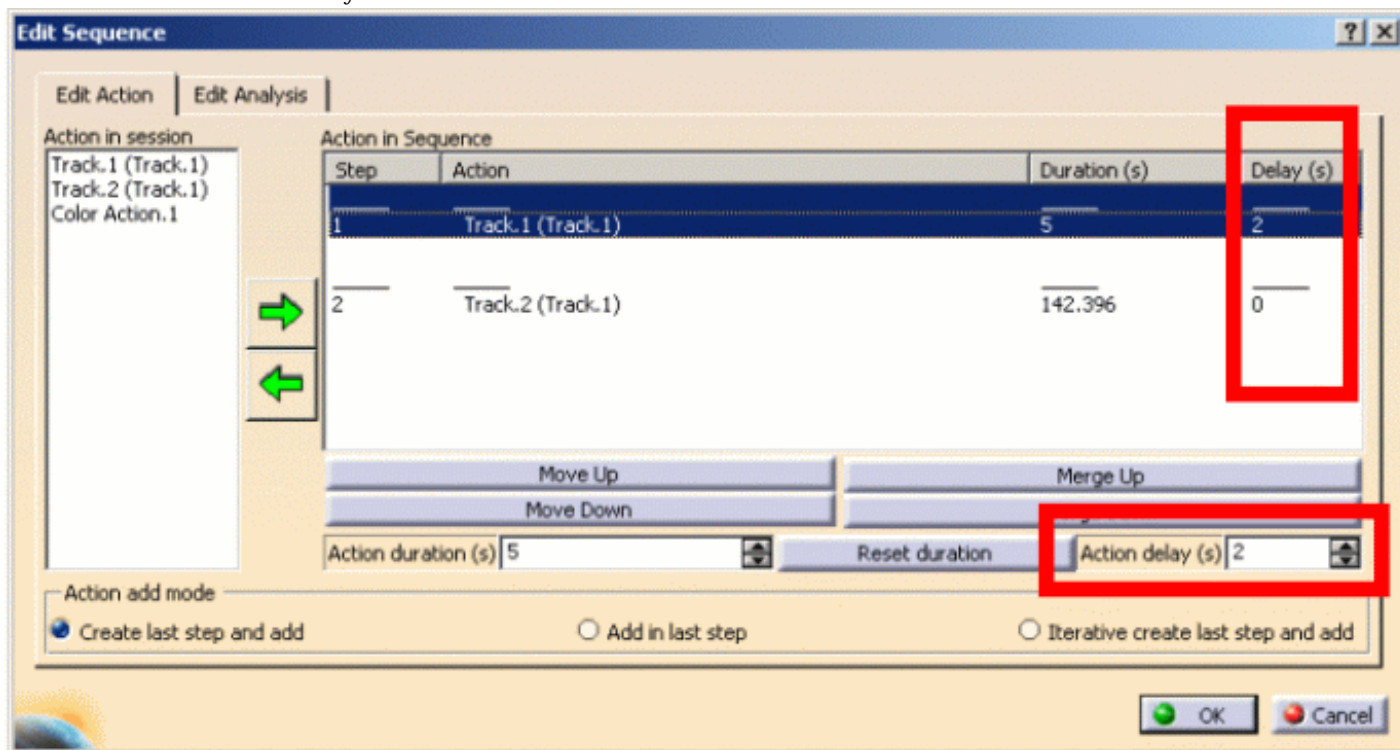
For instance, two tracks within the same sequence step can be synchronized, in order to achieve passing by specific waypoints simultaneously.

For all actions contained in the sequence, the delay is a time attribute, just like their duration. It means the action will start with respect to the specified delay with the theoretical beginning of the step, which the action belongs to. Valid delay values are zero or positive.



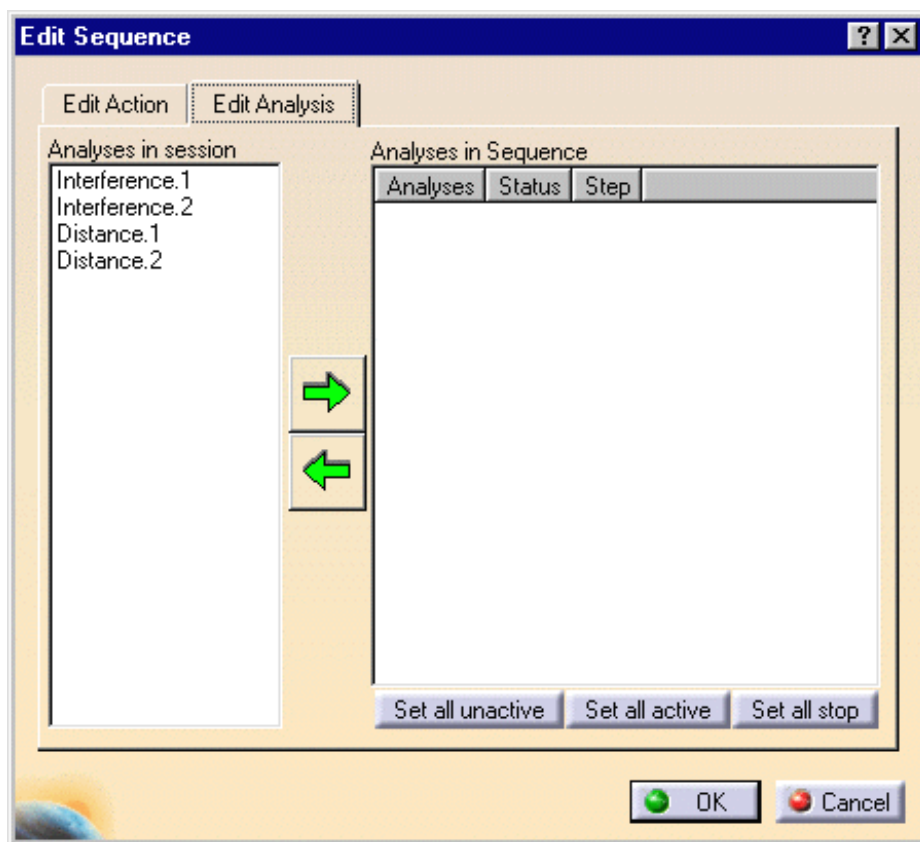
The actions appear in the "Action in sequence" list, they are scheduled in steps and their duration and delays are displayed.

**Note:** To apply a delay to an action or modify it, all you need to do is select the required action in the Action in session list and enter a value in the Action delay field:



Also read : [About actions duration](#)

Now click the *Edit Analysis* tab:



The Edit Analysis tab lets you perform the following operations:

- Add/remove interferences or distances using the green arrows
  - 1. Select an analysis in the "Analyses in session" list and click add. The action is added in the sequence list.
  - 2. Select an analysis in the "Analyses in Sequence" list and click remove

**Note:** you can add existing interferences or distances or create them on the fly ( in this case they are automatically displayed in the "Analyses in session "list

- Set the clash detection mode
  - **Set all inactive** option: (default mode) as you simulate your sequence, the detection is set to off, the interferences and/or distances defined in your sequence are not taken into account
  - **Set all active** option: as you simulate your sequence the detection is set to on, the interferences and/or distances defined in your sequence are taken into account
  - **Set all stop** option: as you simulate your sequence, the detection is set to stop (on collision), the simulation stops when an interference defined in your sequence is detected. The distances defined remain active.



### About Editing an action and analysis

Double-click actions, interferences, distances to display the dedicated editor. Perform the required modifications, the modifications are automatically taken into account in the Edit Sequence dialog box.





# Displaying Gantt Chart



This task shows you how to display the Gantt Chart viewer.

A Gantt chart allows users to do a basic, overall cycle-time analysis for a set of actions. The Gantt chart visualization is based on the cycle time parameter defined in each action.

## About Gantt Chart:

The Gantt chart is another way to visualize your sequence. Note that you cannot modify the action duration using the Gantt window.

A Gantt chart is bar graph of a sequence. It shows start and stop times as well as dependencies. The Gantt chart is a 2D view of the sequence process.

Open the [CHAINSAWAT.CATProduct](#) document

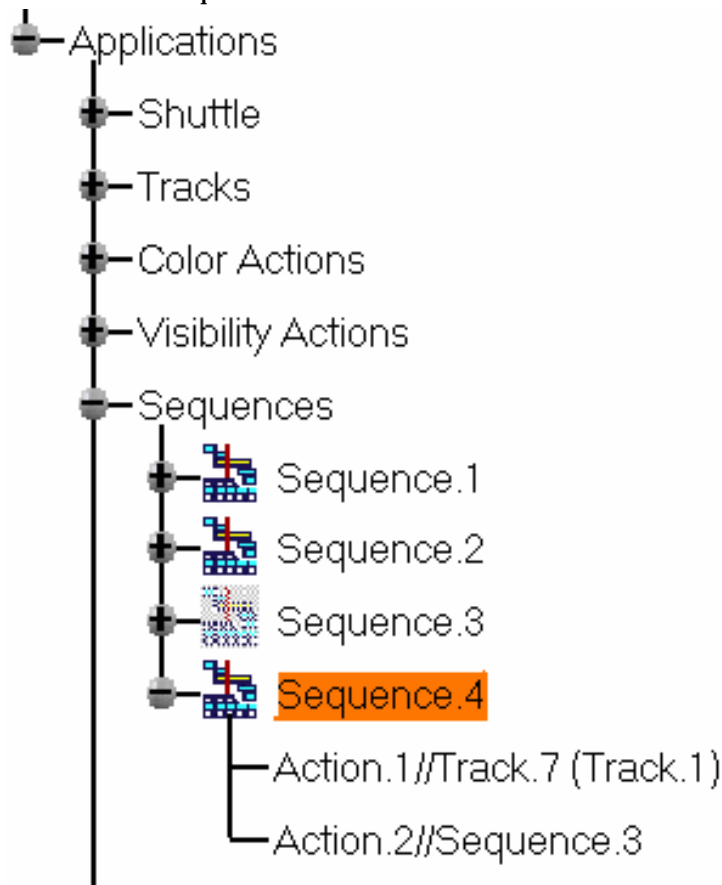


A sequence is defined



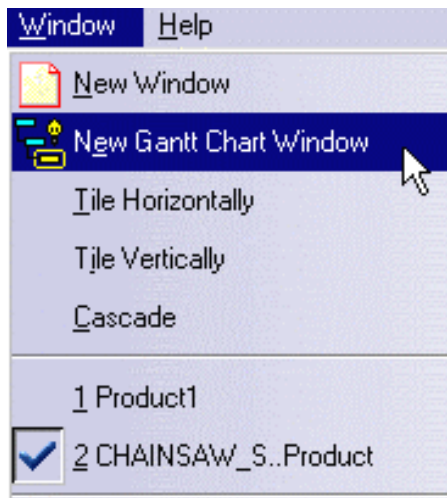
1. Select a sequence in the specification tree:

for instance Sequence.4

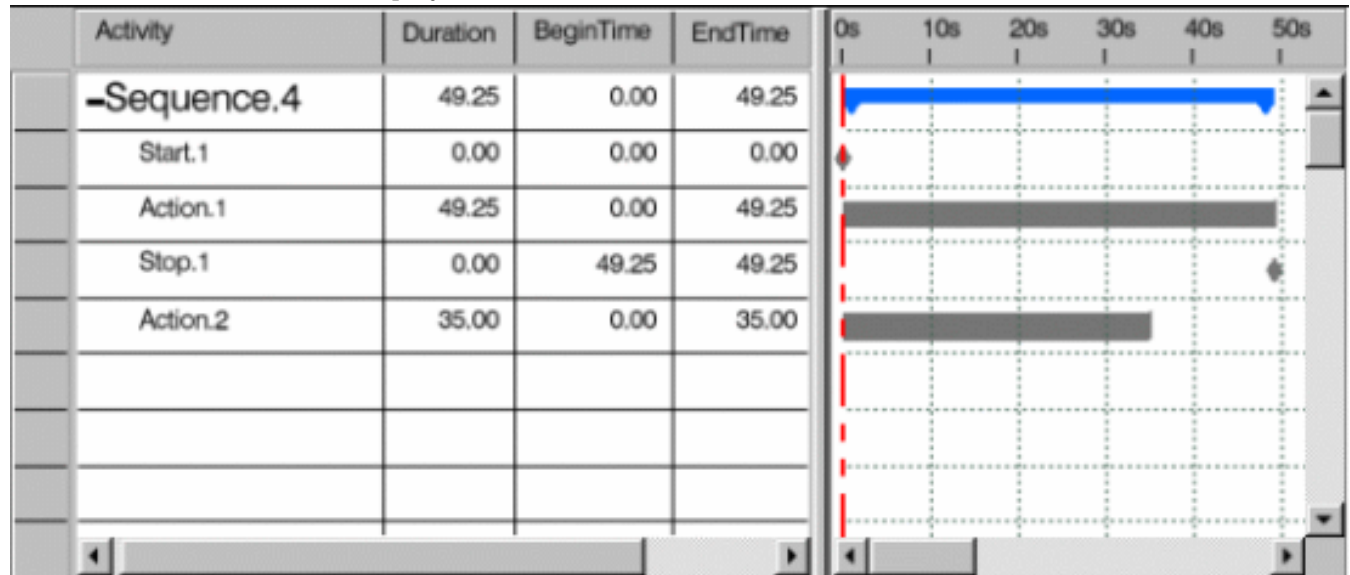


Gantt Chart for multiple sequences in a single Gantt Chart are not supported

2. Select **Window->New Gantt Chart Window**



The Gantt chart window is displayed:



**Let's describe it more carefully:**

- The left frame of the Gantt chart lists each of the individual actions and /or analyses that exist in your document, displaying the duration, start time and end time for each.
- The right frame provides a graphical representation of each action or analysis (along the line of time) which also indicates the start, duration and end of each.
- A dashed vertical line in the right window (called the Time Line), provides a visual indication of the current time during the execution.

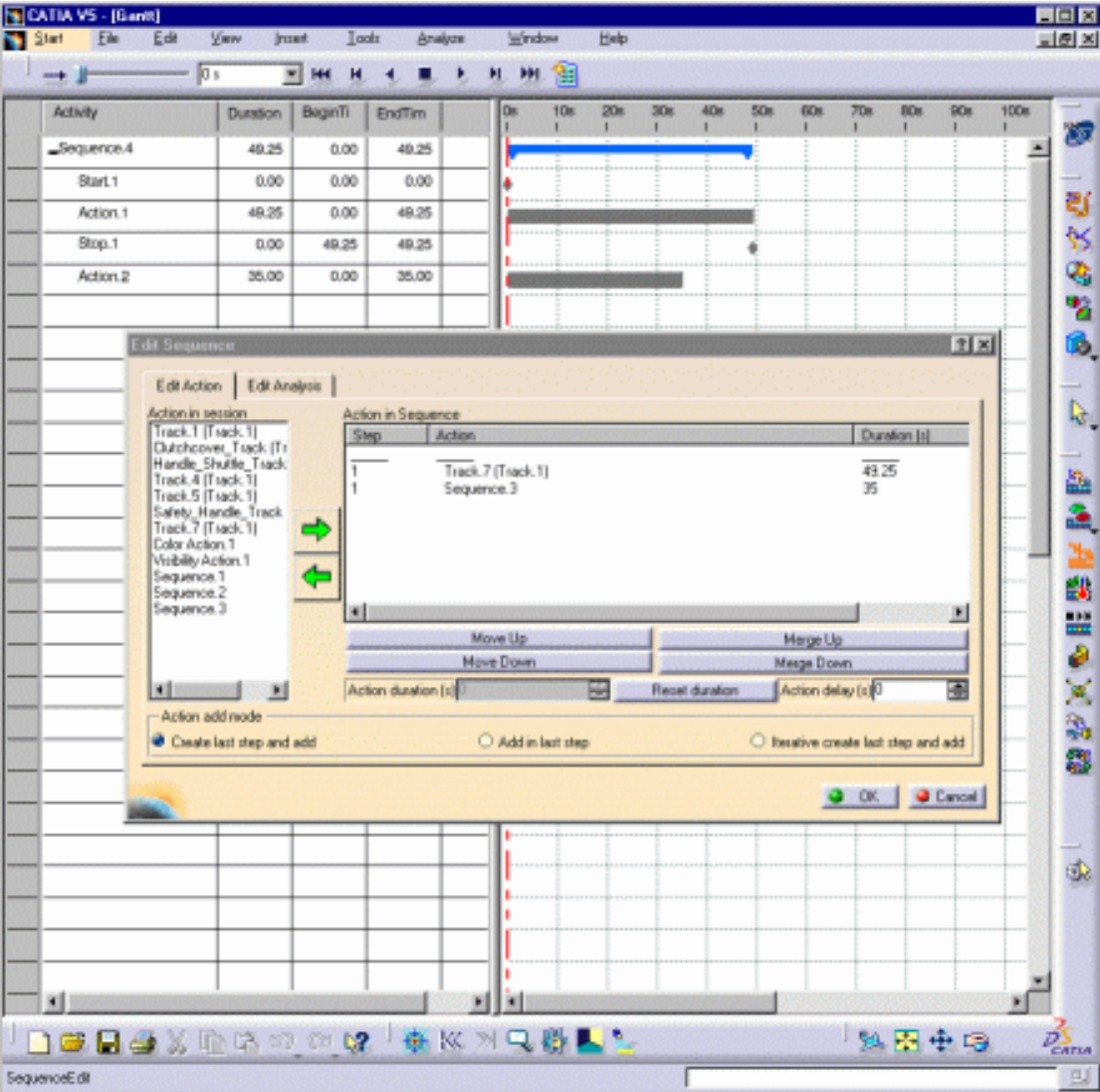


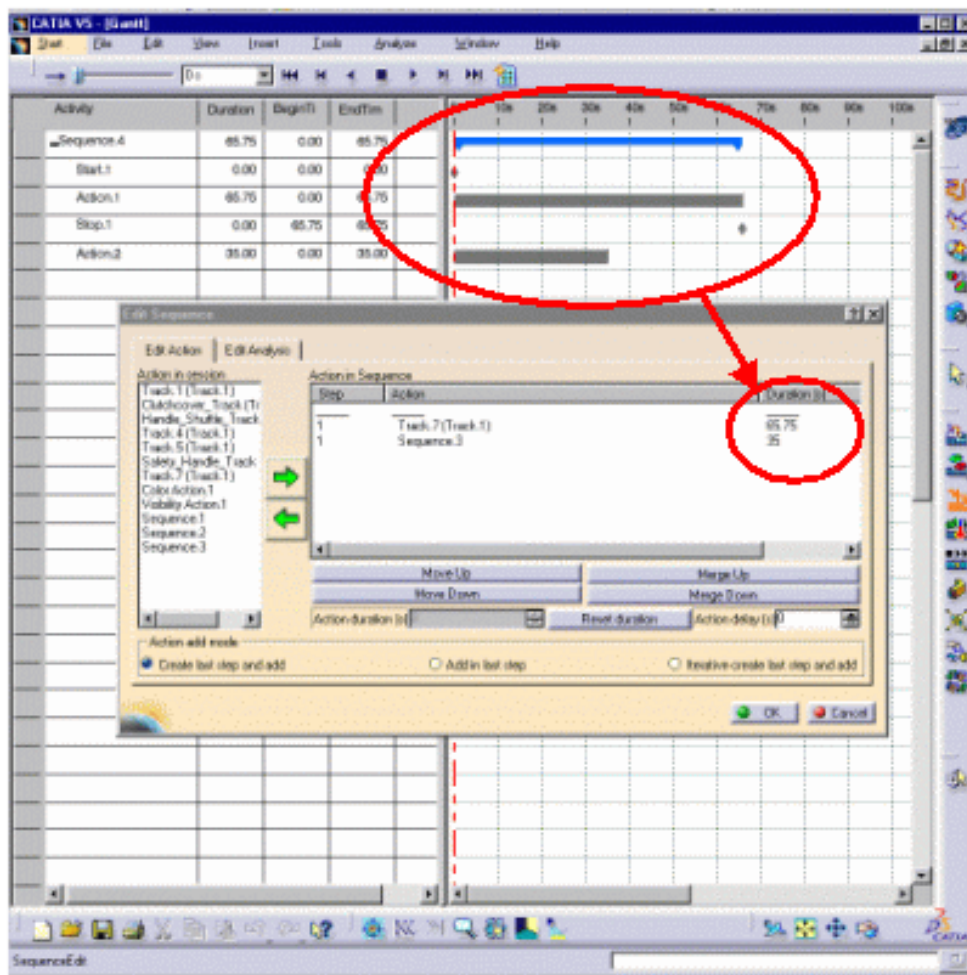
You can easily modify the action duration within the Gantt Chart, stretching the action graphical representation, the Gantt Chart as well as the Edit sequence dialog box are automatically updated.

For instance, double-click a sequence in the specification tree. You are in the sequence command.

Then select **Window->New Gantt Chart Window:**







It does work, the other way round: modifying the action duration within the Edit Sequence updates the Gantt Chart window automatically



# Defining a Sequence



This task shows you how to define a sequence





Open the [DEFINE\\_SEQUENCE.CATProduct](#) document.  
Tracks are already defined

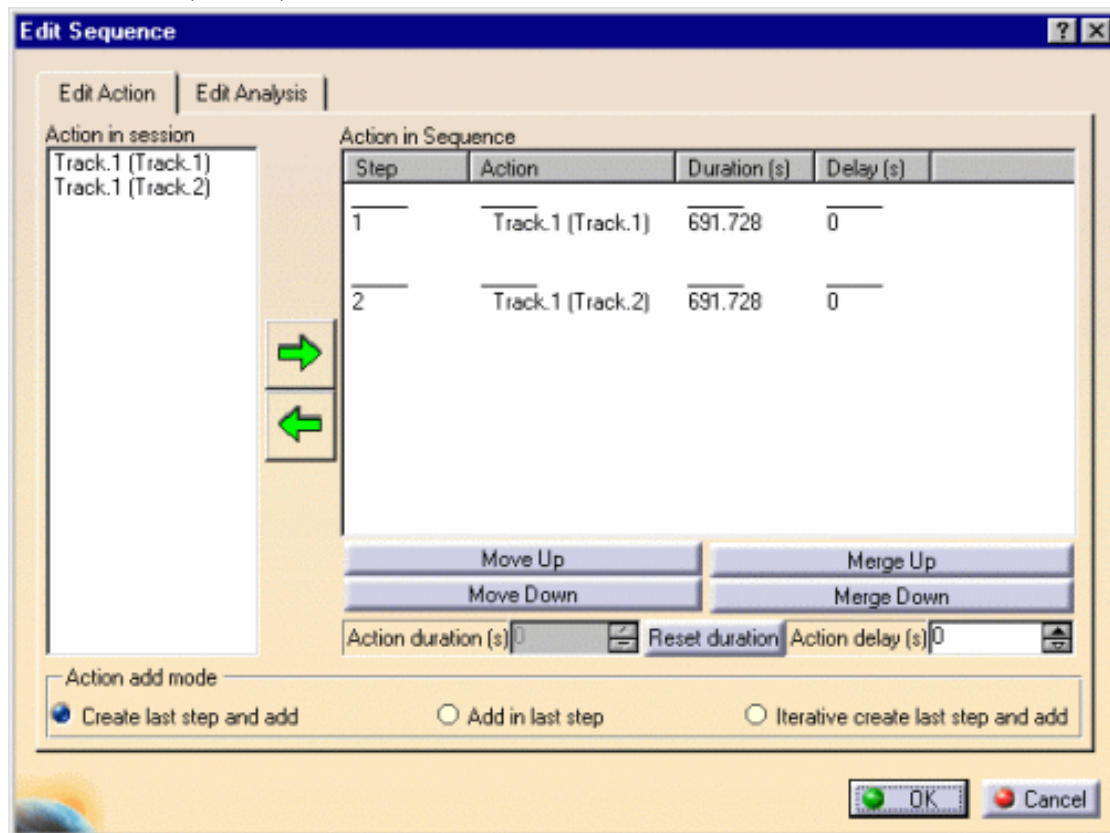


1. Click the Edit Sequence icon  in the DMU Simulation toolbar

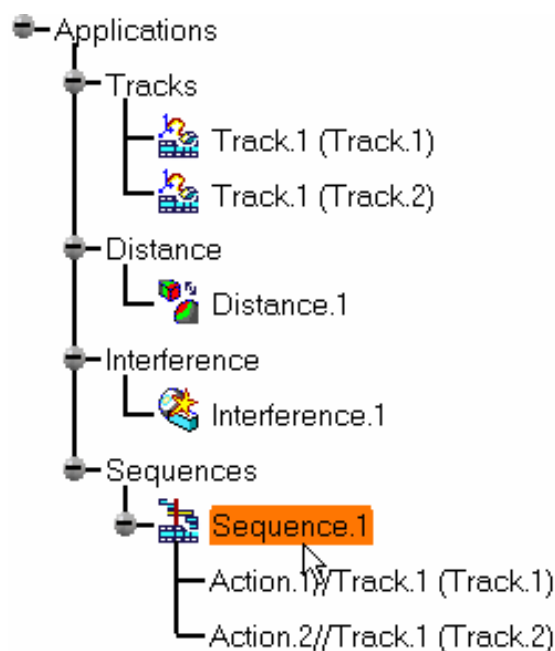
The Edit Sequence dialog box is displayed

2. Select Track.1 (Track.1) in the action in session list and click .

3. Select Track.1 (Track.2) in the action in session list and click .

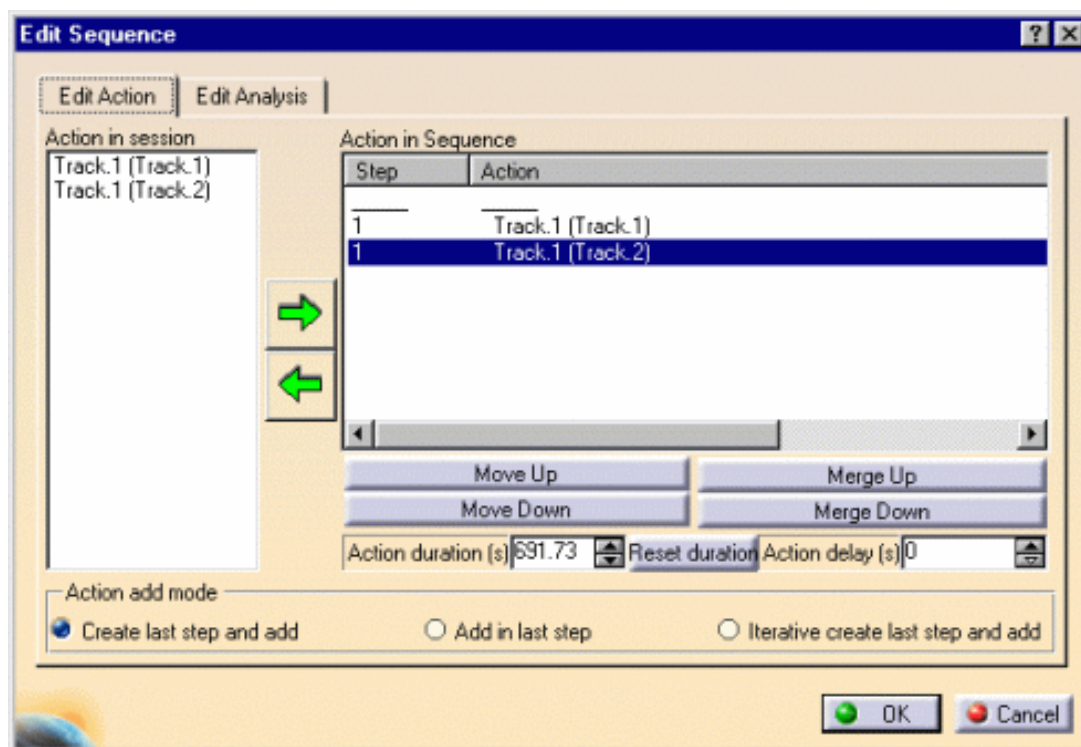


The Sequence.1 is identified in the specification tree



*you want them to start together*

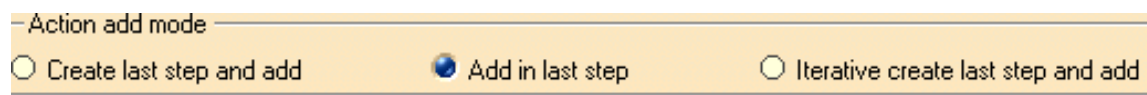
4. Click Merge Up button



*The two actions will start together*

**Note:** you could have selected both (simultaneous mode) in the action in session list.

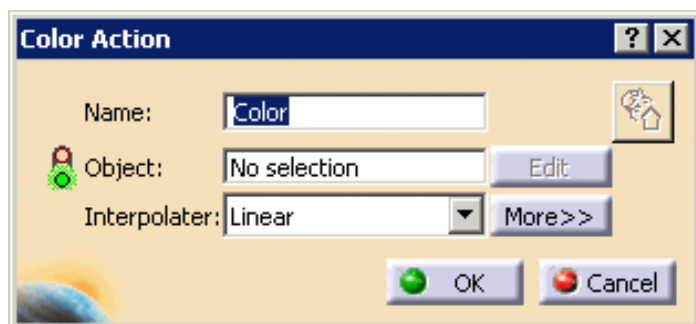
5. Check the "Add in last step" option in the Edit Sequence dialog box  
(Create last step and add option is set by default)




6. Add a color action (you are going to [create it on the fly](#)), for this

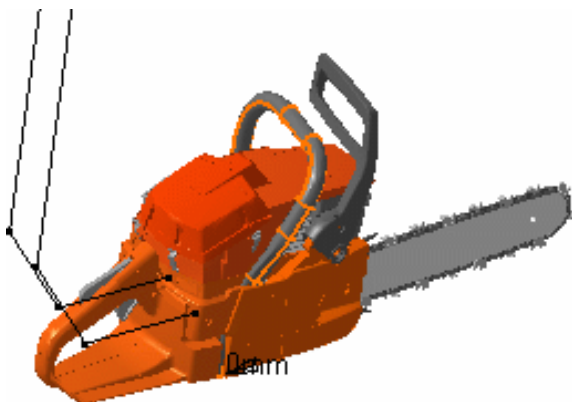
Click the Color action icon  in the DMU Simulation toolbar

The Color Action edition dialog box appears



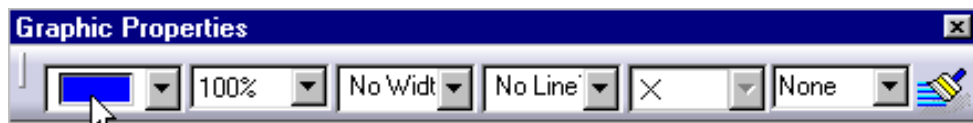
 You can alter the name of the color action by entering one in the Name field, or you can accept the default name provided.


7. Select Handle.1 either in the specification tree or in the geometry area




The Graphic Properties toolbar appears

8. Select a color of your choice using the arrow and combo list. For instance blue



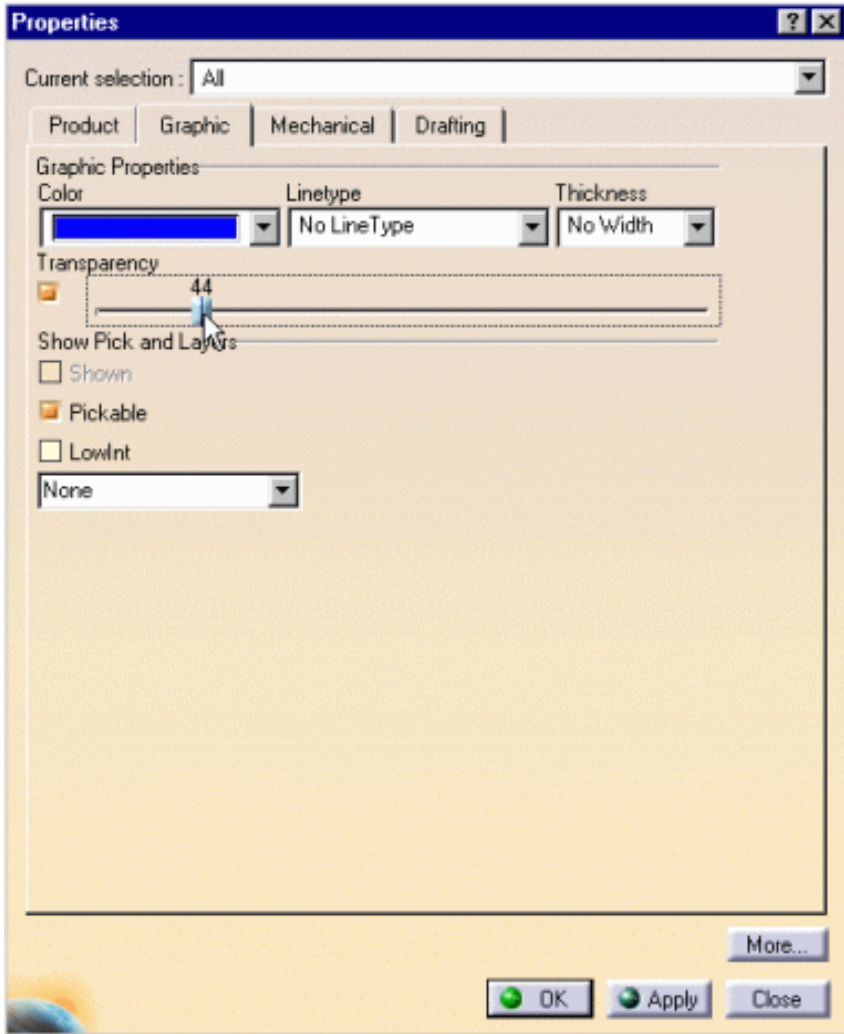
9. Click Record 


10. Set the transparency, for this:

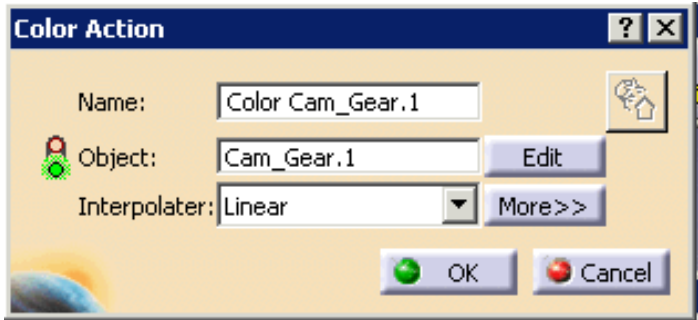
- click the Edit button  in the color action dialog box
- select the Graphic tab in the Properties dialog box displayed
- check the transparency option if needed, and move the slider as desired

**Note:** you can access the Properties dialog box at any time to change color, transparency. The Graphic Properties toolbar a quicker way to modify graphic properties.



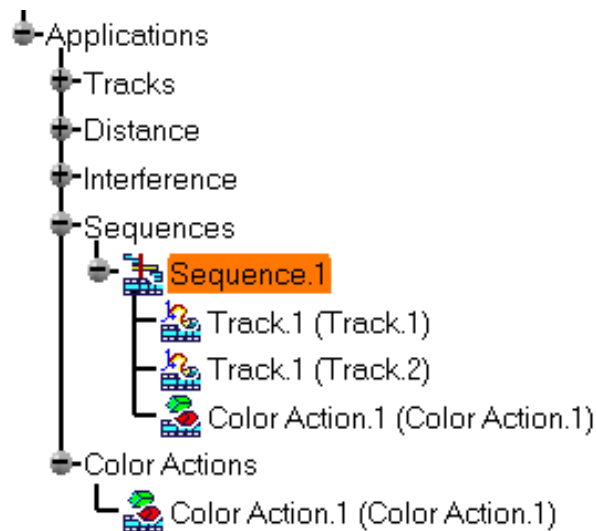


11. Click Apply, when done, click close to exit the Properties dialog box
12.  Click Record
13. Click Ok in the Edit Color Action dialog box when you are satisfied.



The color action is automatically added in the action in the sequence and identified in the specification tree

Step	Action	Duration (s)	Delay (s)
1	Track.1 (Track.1)	691.728	0
1	Track.1 (Track.2)	691.728	0
1	Color Action.1 (Color Action.1)	0.99717	0



**14.** Modify the action duration if necessary

For more detailed information, please read: [About Action Modification](#)

- Enter 200 in the Action duration field

**15.** Select the Color Action.1 in the action list and modify its delay:

- enter 400 in the Action delay field

Step	Action	Duration (s)	Delay (s)
1	Track.1 (Track.1)	691.728	0
1	Track.1 (Track.2)	691.728	0
1	Color Action.1 (Color Action.1)	200	400

Action duration (s) 200  Action delay (s) 400

**16** Play your sequence if needed using the Player

*Now add analyses in your sequence for validation purposes*

**17.** Click the Edit Analysis tab

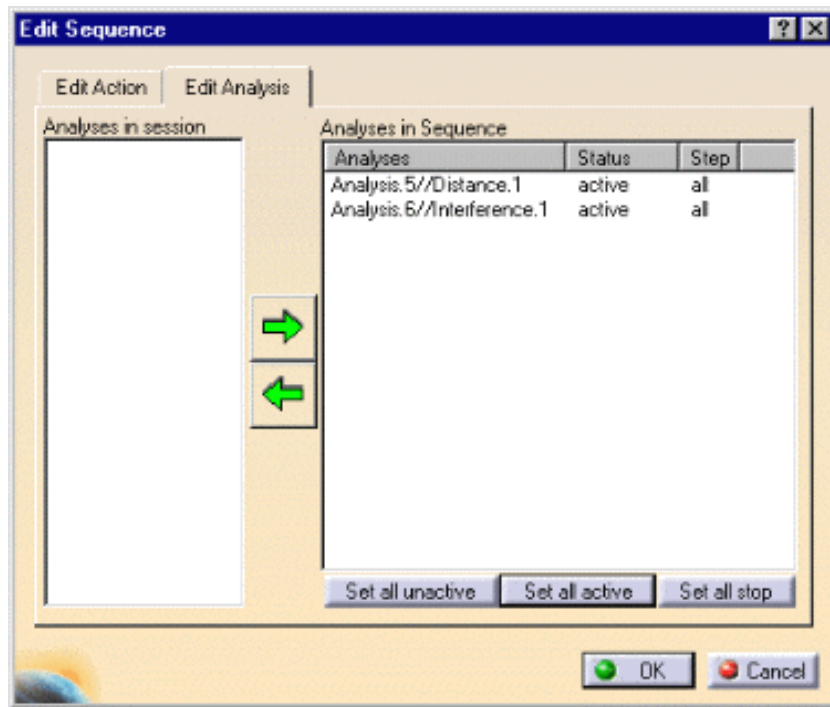
**18.**



Multi-select the existing analyses (i.e. Distance.1 and Interference.1) and click


**Note:** you can create and add analysis specifications on the fly. You can also edit existing analysis specifications, double-clicking them in the "Analyses in sequence" list

**19.** Click Set all active button




**20.** Click Ok in the Edit Sequence dialog box when satisfied.

**21.**

Select your sequence in the specification tree and click the player icon .

**22.** Simulate your sequence using the Player buttons.

**23.**

If you need to restore the initial positions, click the Reset icon .

**24.** Open the [DEFINE\\_SEQUENCE\\_RESULT.CATProduct](#) to check your result





# Generating Animations

[Generate a video using DMU dedicated tool](#) (using **Tools->Simulation->Generate Video**),

[Generate a video using standard tools](#) (using **Tools->Image->Video**),

Please also read [Generate an Animation](#)

# Generating a Video Using Standard Tools



This task shows you how to generate a video file selecting **Tools->Image->Video...**



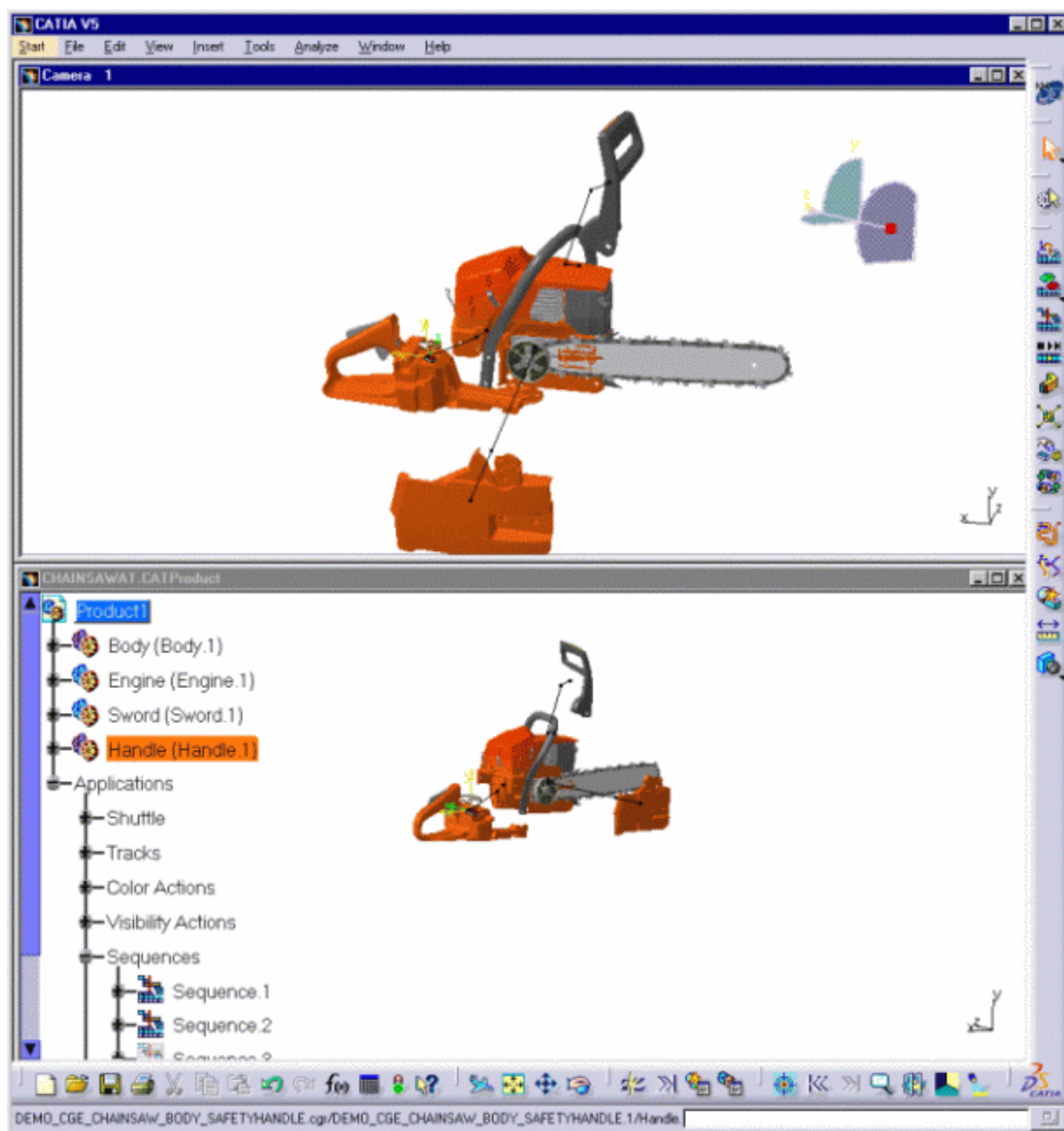
Open [CHAINSAWAT.CATProduct](#) document.




1. Select Sequence.4 in the specification tree

2. Open a camera window, select **Window->Camera window->Camera.1...**

3. Select **Window->Tile Horizontally...**






3. Multi-select the tracks in the specification tree and click the Hide/Show  icon, or select the **View->Hide/Show->Swap Hide/Show** command.

The track objects are no longer displayed: they have been transferred into the No Show space.  
*You are now ready to generate a film in AVI format*

4. Select **Tools->Image->Video** to display the Video Recorder dialog box

- Set Up the Capture Session
- Click the Recording button  to access the Video Properties dialog box which lets you set up the video capture parameters prior to recording the video
- Choose the Video File Format and Location
- in our example select Microsoft AVI (Windows)
- Specify which part of the screen to record
- Setting Movie Replay Parameters
- Click OK in the Video Properties dialog box to start the recording.



Please read Recording Interactions in Video Format in the *Infrastructure User's Guide*

5. Run your sequence simulation with the Player
6. Click Stop



# Explode

You can use the explode functionality either to understand better the structure of your assembly and analyze the dismount capability or to obtain a printed document of your exploded product for further analysis.



**Explode a product:** Select a product, then click the explode icon. Select the required explode type and depth setting. Click Apply.

**Explode a constrained assembly:** Select a product, then click the explode icon. Set the explode type to constrained, when done click Apply.

**Explode a shuttle:** Select a shuttle, then click the explode icon. Select the required explode type and depth setting. Click Apply. The interest of this operation lies in the ability of viewing all components separately within the same plane. Now, you can print the exploded view.

**Generate a Sequence from an exploded view:** Select a product, then click the explode icon. Select the required explode type and depth setting. Check the Generate a sequence option. Click Apply.

# Exploding a Product



This task illustrates the use of the Explode capability. Exploding the view of an assembly means separating the components of this assembly to see their relationships.

At any time though, you can check the product structure via the specification tree. The explode capability allows you to fully understand the product structure in a 3D context.




Open the [CHAINSAW.CATProduct](#) document.

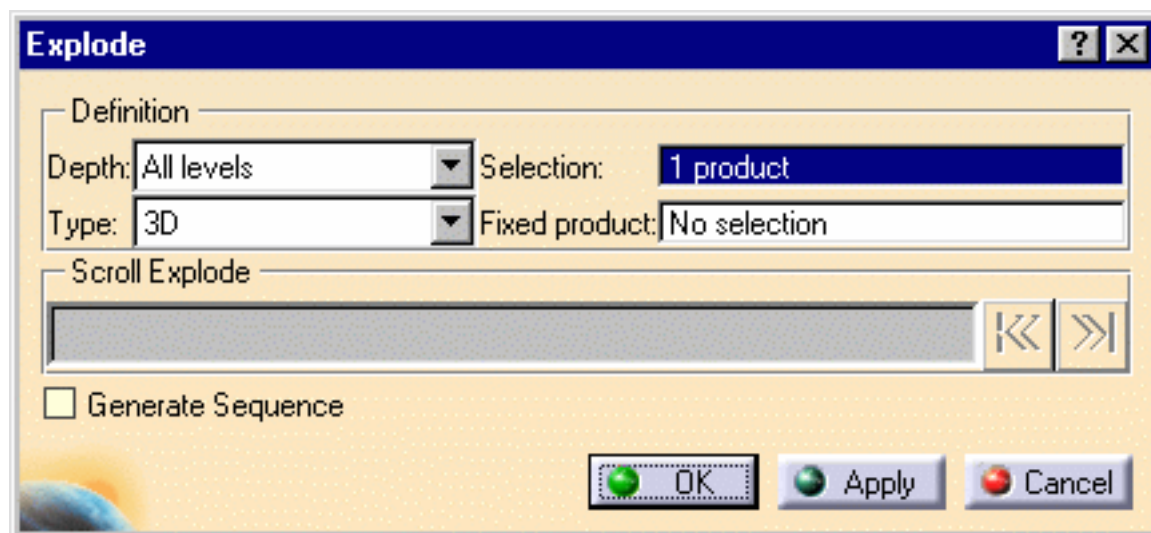


1. Select **Product1** in the specification tree.

2.



Click the Explode icon .  
The Explode dialog box is displayed.



Product 1 is the assembly to be exploded. The **Depth** parameter lets you choose between a total (**All levels**) or partial (**First level**) exploded view.

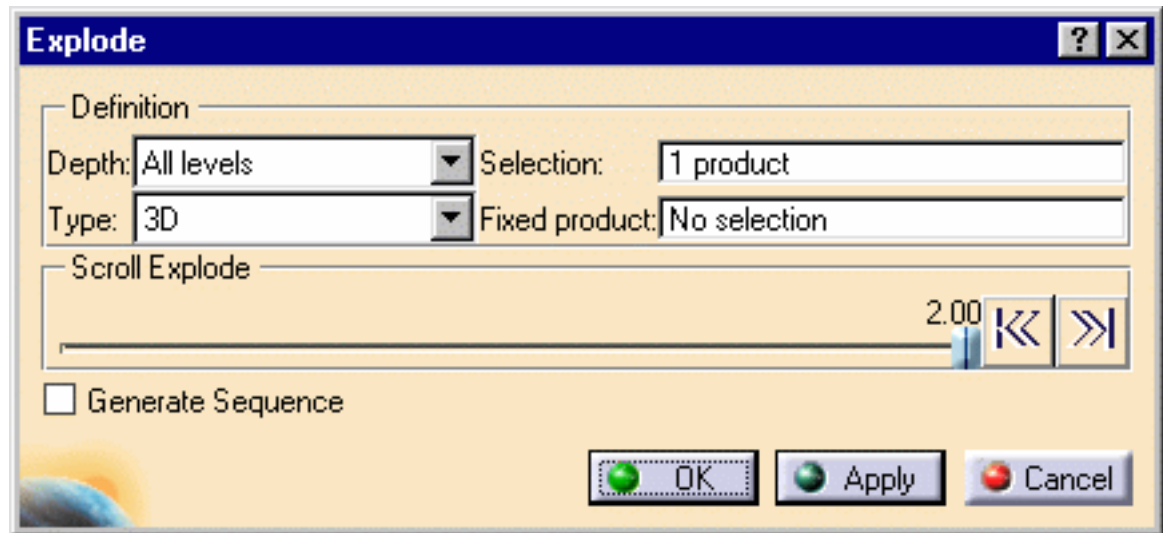
3. Set **All levels** if not already set.

4. Set **3D** as explode type.

You can now generate a sequence from the exploded view, all you need to do is click the Generate Sequence checkbox before launching the explode operation.

For more detailed information refer to [Generating a Sequence from an Exploded View](#)

5. Click **Apply** to perform the operation.



The **Scroll Explode** field gradually displays the progression of the operation. The application assigns directions and distances. Once complete, the assembly looks like this:



The interest of this operation lies in the ability of viewing all components separately.

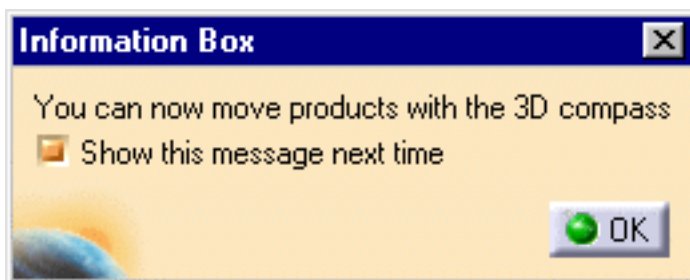
6.




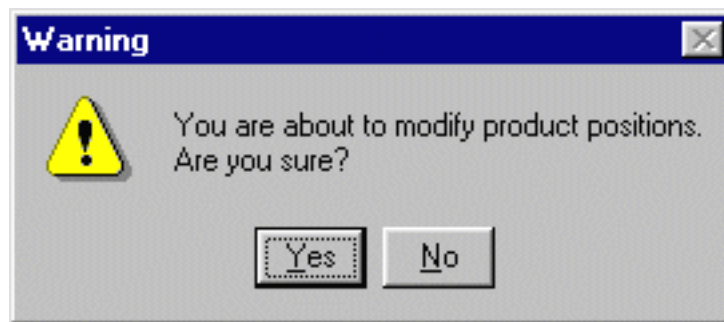
If you click , you go back to the product structure level which corresponds in this case to the sub-entities: engine, handle, sword as shown below:



You can easily move products within the exploded view using the 3D compass.



7. Click OK to confirm the operation or click Cancel to restore the initial view.  
**Note:** If you click OK, the following warning message is displayed as the exploded view is kept when exiting the command. In this case, if you need to restore the initial view click the Reset icon .







# Exploding a Constrained Assembly



This task shows how to explode an assembly taking into account the assembly constraints. This Explode type is applicable only to specific cases: when the assembly is assigned coincidence constraints:

- axis/axis
- plane/plane



Open the [EXPLODE\\_CONSTRAINED\\_ASSEMBLY.CATProduct](#) document.



1.



Click the Explode icon .  
The Explode dialog box is displayed.

2. Wheel Assembly is selected by default, keep the selection as it is.



You can also use the drag and drop capability (drag the explode icon and drop it onto the required product in the specification tree).

The **Depth** parameter lets you choose between a total (**All levels**) or partial (**First level**) exploded view.

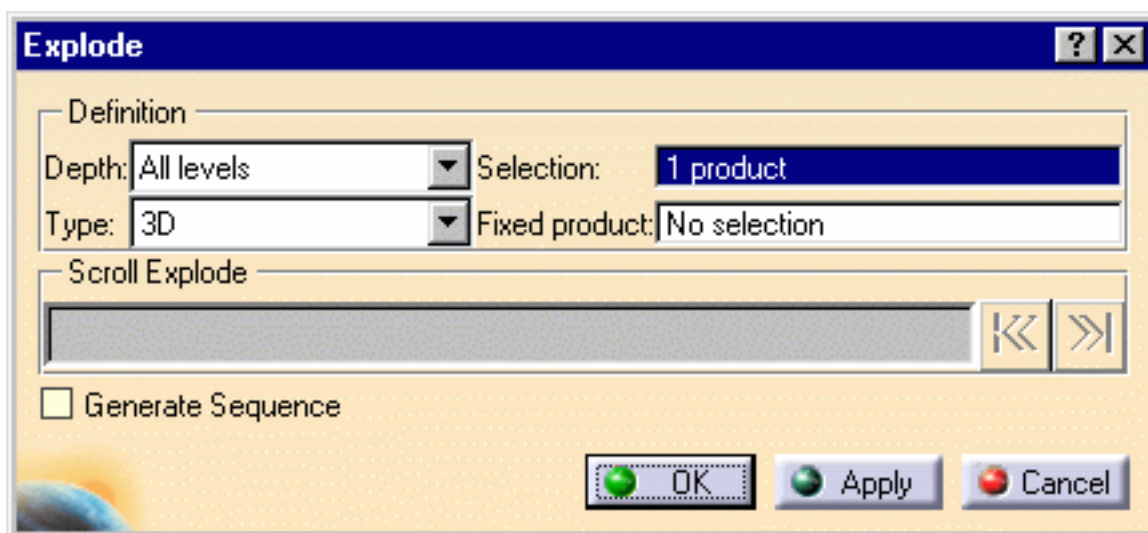
3. Keep **All levels** set by default.

4. Set the explode type. **3D** is the default type. Keep it.

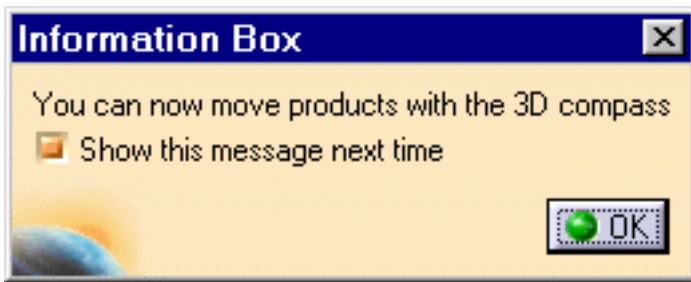
You can now generate a sequence from the exploded view, all you need to do is click the Generate Sequence checkbox before launching the explode operation.

For more detailed information refer to [Generating a Sequence from an Exploded View](#)

5. Click **Apply** to perform the operation.

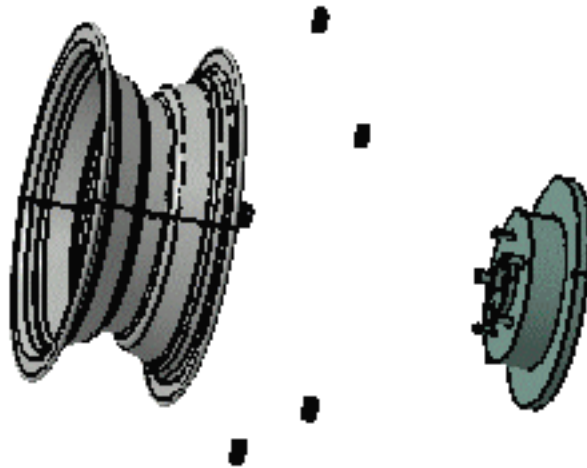


Note that you can move products within the exploded view using the 3D compass.



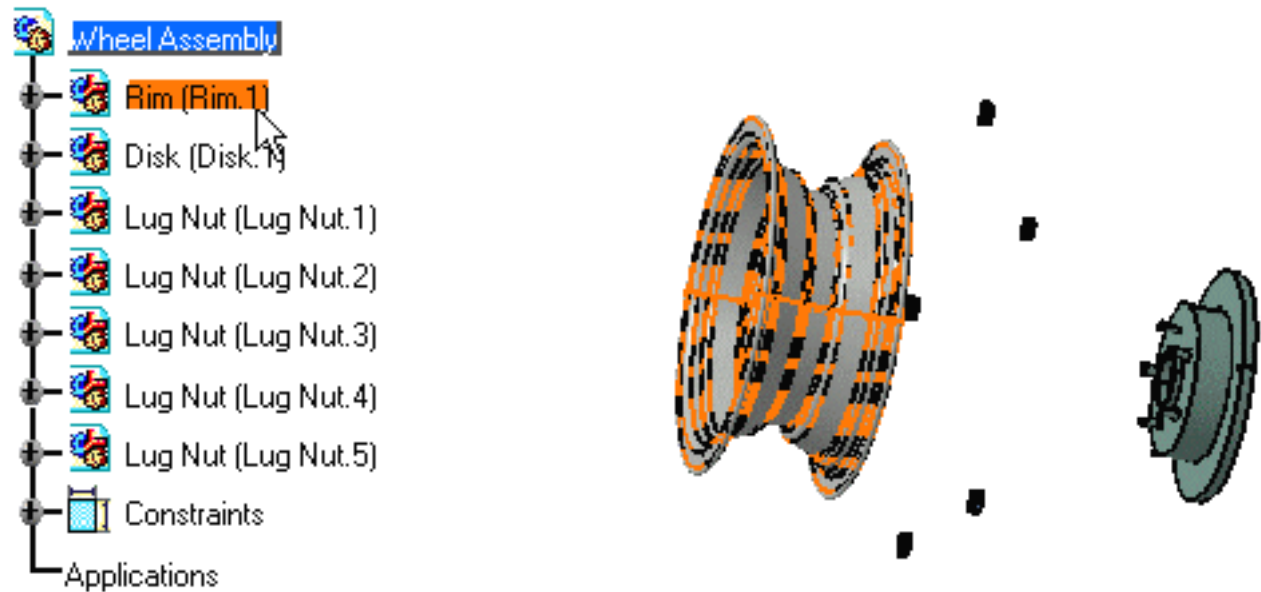
The manipulation toolbar is also available once you move an object with the 3D Compass.

The **Scroll Explode** field gradually displays the progression of the operation. The application assigns directions and distance. Once complete, the resulting exploded view looks like this:

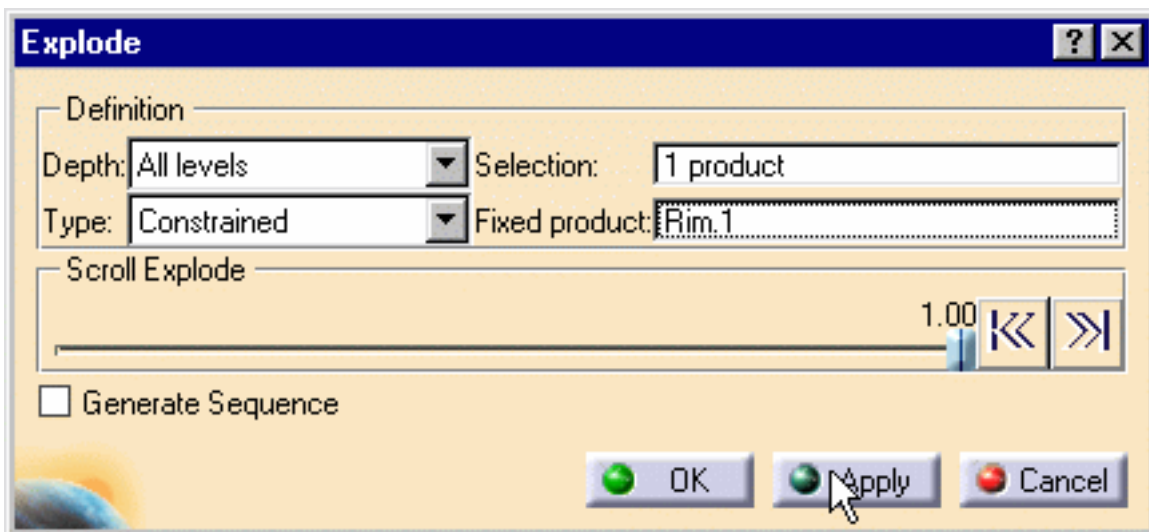


You are not satisfied with this result as the nuts are not correctly positioned. The constraints are not respected. Replay the scenario selecting the constrained type.

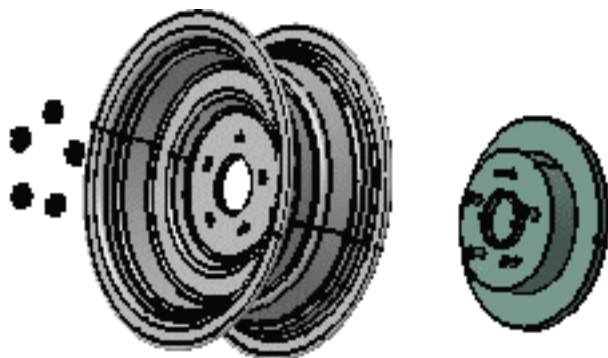
6. Still in the Explode dialog box, set the **constrained type**.
7. Define a fixed product: in our example select the Rim1 either in the specification tree or in the geometry area



8. Click **Apply** to perform the operation.



Once complete, the resulting exploded view looks like this:




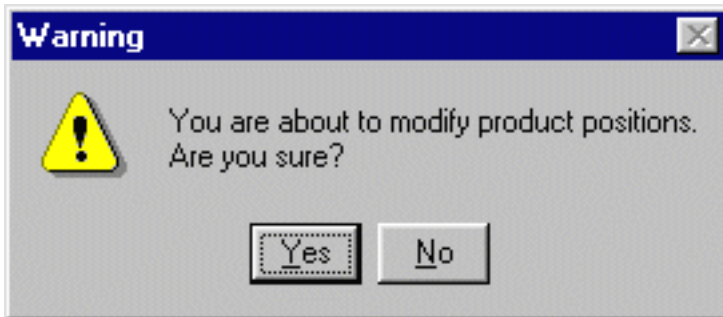
The nuts are correctly positioned, the exploded view corresponds more to the reality and to a technical documentation.

**9.**

Click OK to validate the operation or click Cancel to restore the original view.

**Note:** If you click Ok, the following warning message is displayed as the exploded view is kept when exiting the command. In this case, if you need to restore the initial view click the

Reset icon .



The explode functionality aims at understanding better how the assembly is structured. You can use it for further purposes: creating scenes, print, keep the exploded view as archive document or generate a drawing (please refer to Create Scenes in the *DMU Navigator User's Guide*)



# Exploding a Shuttle



This task shows how to explode a shuttle. This time, you need to print the exploded view to define a bill of material for instance



Open the [EXPLODE\\_SHUTTLE.CATProduct](#) document.

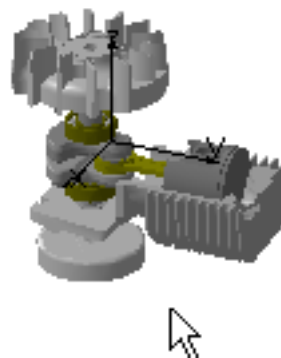


1.



Click the Explode icon

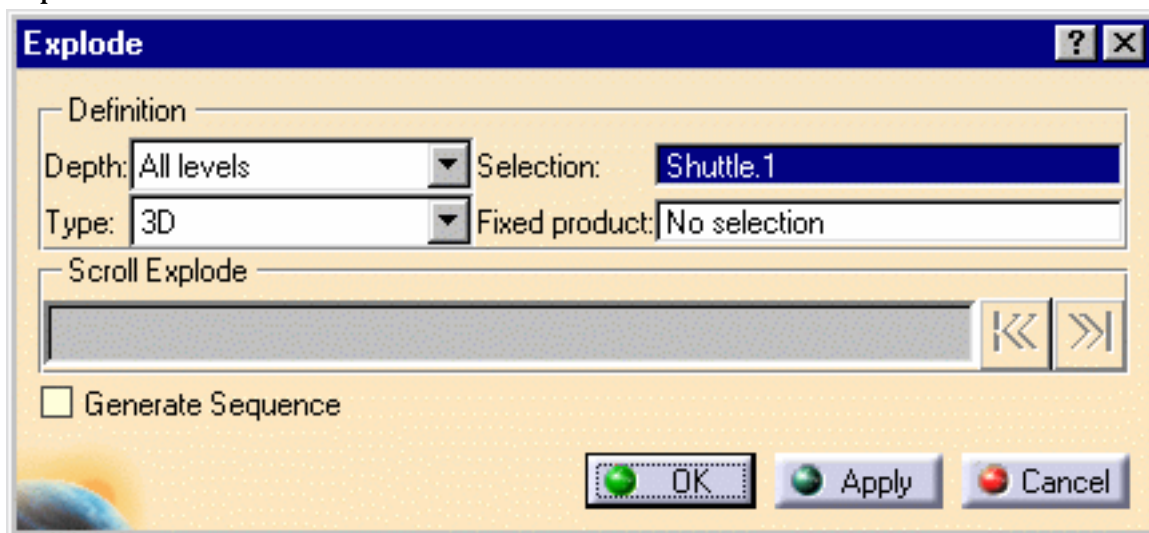
2. Select the shuttle you need to explode in the specification tree or in the geometry area.



You can also use the drag and drop capability (drag the explode icon and drop it onto the shuttle.1 in the specification tree.

The Explode dialog box is displayed :

The **Depth** parameter lets you choose between a total (**All levels**) or partial (**First level**) exploded view.



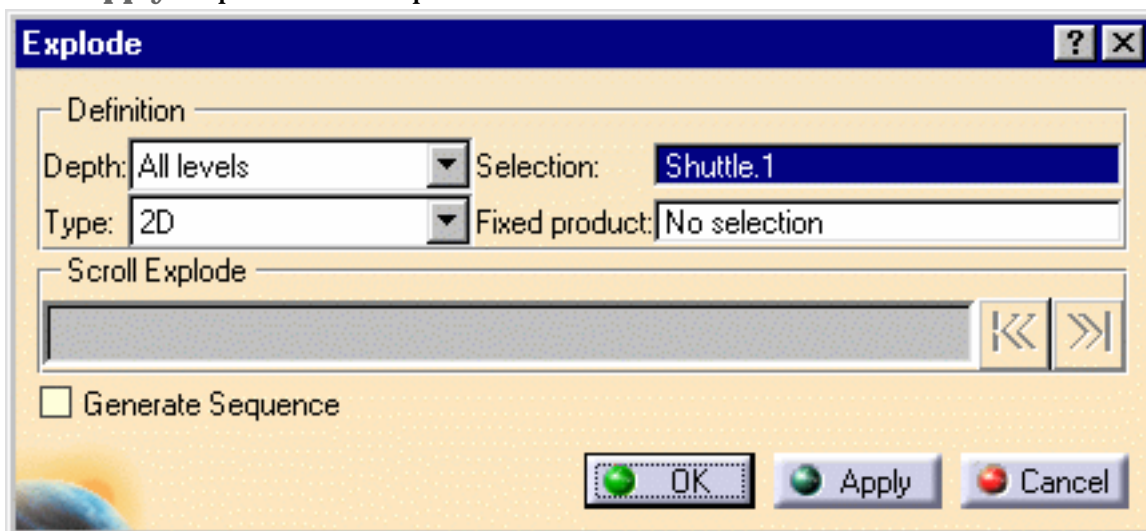
3. Set **first level** if not already set.

4. Set **2D** to define the explode type. The exploded view is set to the screen plane.

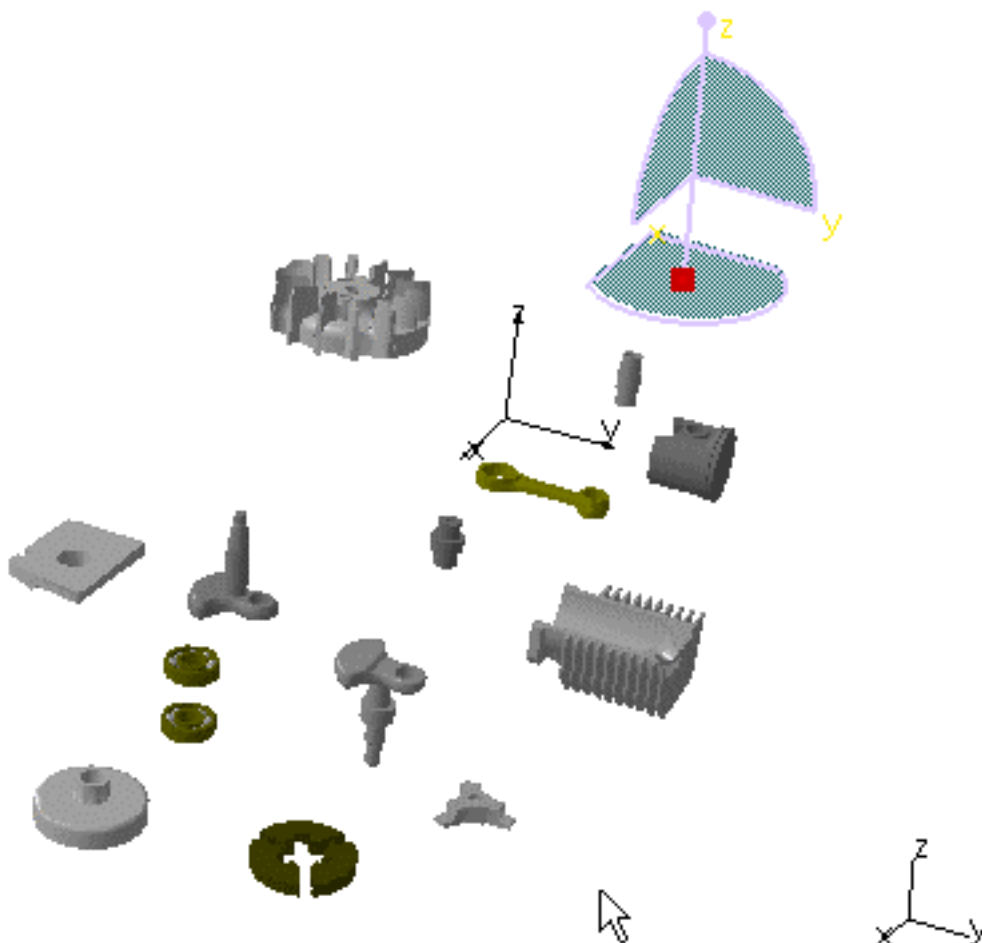
You can now generate a sequence from the exploded view, all you need to do is click the Generate Sequence checkbox before launching the explode operation.

For more detailed information refer to [Generating a Sequence from an Exploded View](#)

5. Click **Apply** to perform the operation.



The **Scroll Explode** field gradually displays the progression of the operation. The application assigns directions and distance. Once complete, the resulting exploded view looks like this:



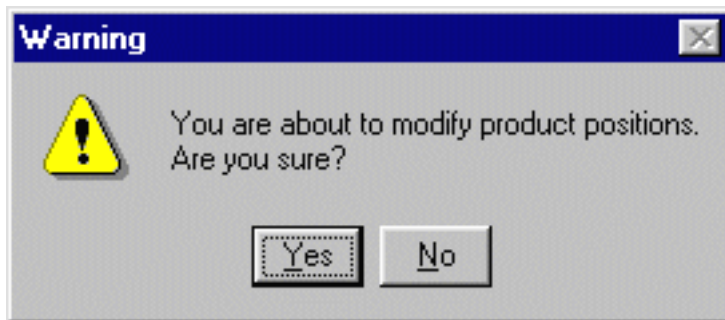
The interest of this operation lies in the ability of viewing all components separately within the same plane.

- Click OK to validate the operation or click Cancel to restore the original view.

**Note:** If you click Ok, the following warning message is displayed as the exploded view is kept when exiting the command. In this case, if you need to restore the initial view click the



Reset icon.



- Now you can print the exploded view. Select **File->Print** command to display the Print dialog box

Once you select the desired settings, press the preview button and click OK







# Generating a Sequence from an Exploded View



This task shows how to generate a sequence from an exploded view



Open the [CHAINSAW.CATProduct](#) document.



1.



Click the Explode icon.

The Explode dialog box and the manipulation toolbar are displayed.

2. Product.1 is already selected, keep the selection as it is.



You can also use the drag and drop capability (drag the explode icon and drop it onto the required product in the specification tree).

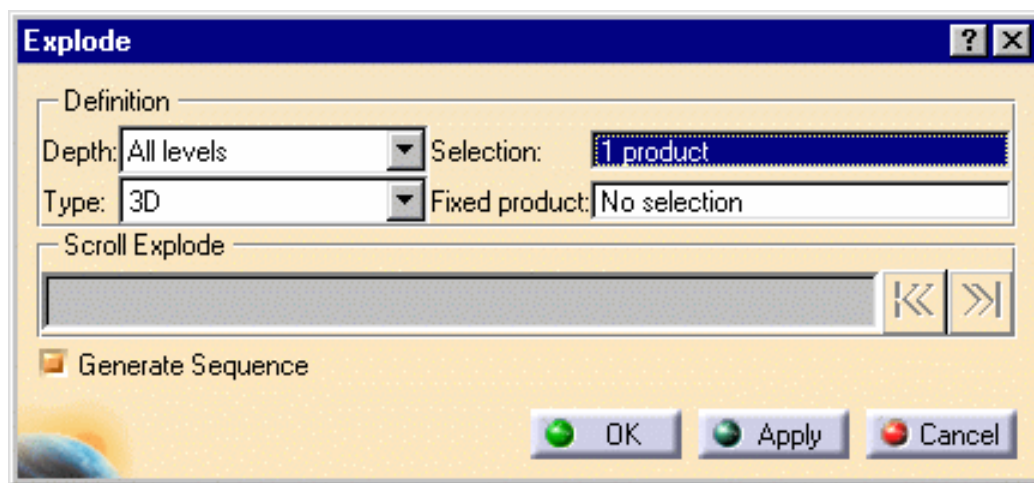
The **Depth** parameter lets you choose between a total (**All levels**) or partial (**First level**) exploded view.

3. Keep **All levels** set by default.

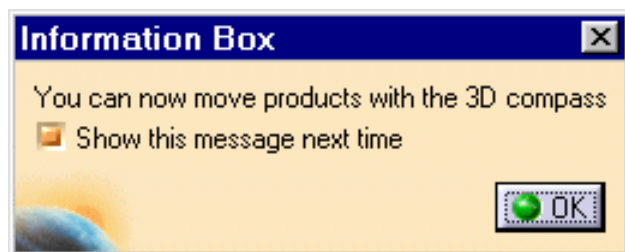
4. Set the explode type. **3D** is the default type. Keep it.

5. Click the Generate Sequence checkbox

6. Click **Apply** to perform the operation.



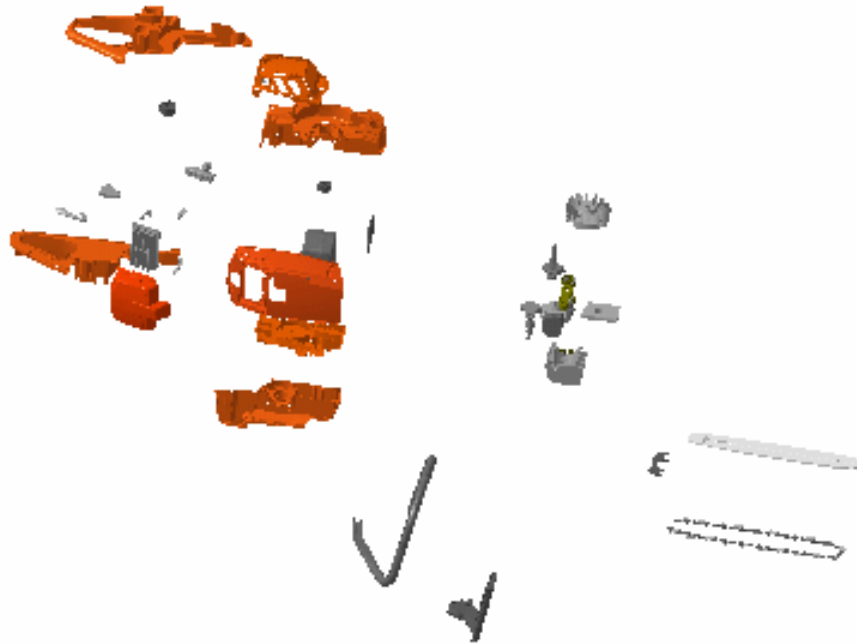
Note that you can move products within the exploded view using the 3D compass.



The manipulation toolbar is also available once you move an object with the 3D Compass.

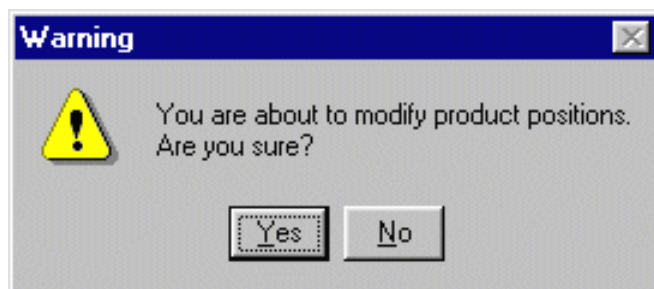
7. Click **Ok** in the Information dialog box

The **Scroll Explode** field gradually displays the progression of the operation. The application assigns directions and distance. Once complete, the resulting exploded view looks like this:

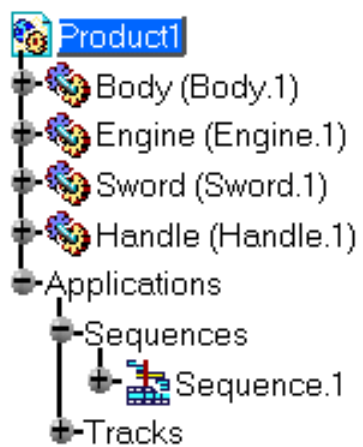
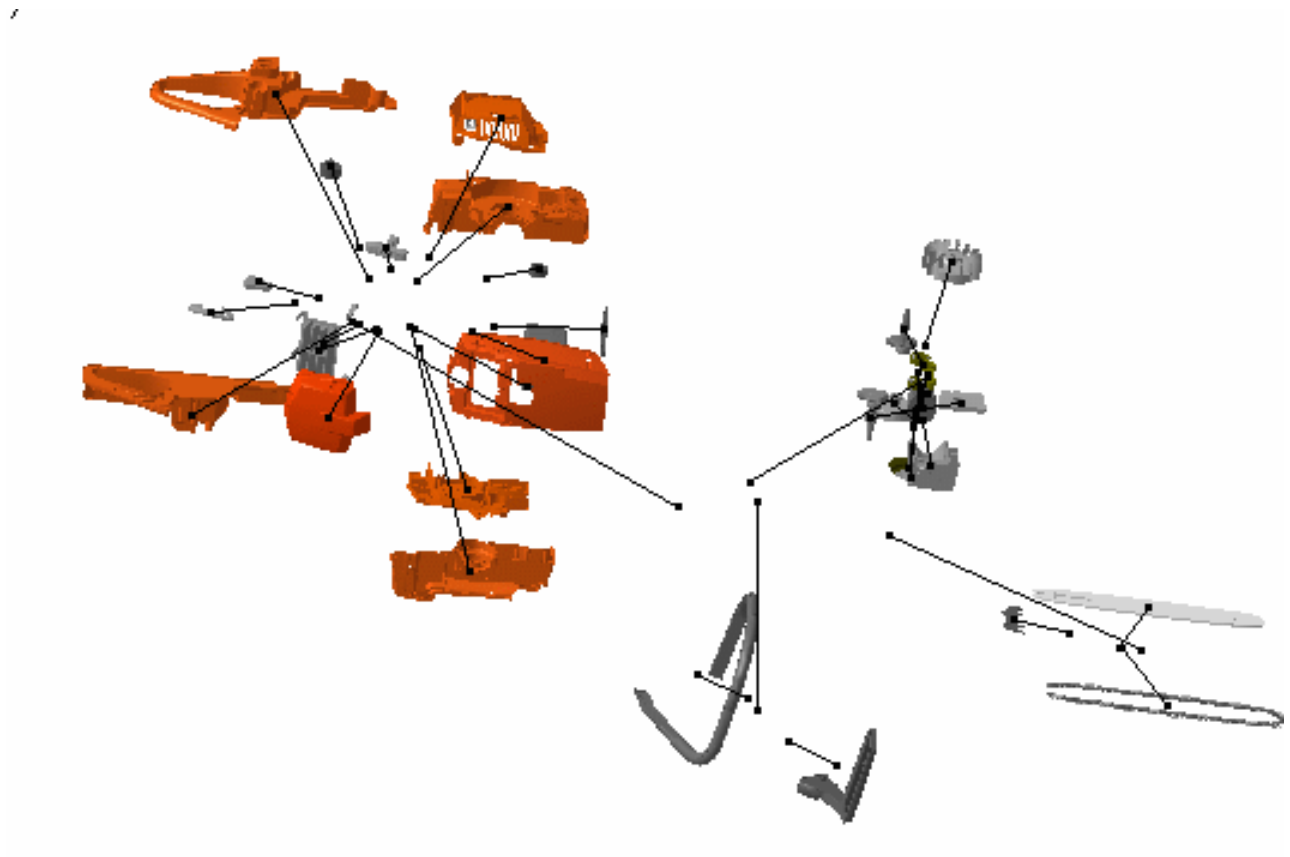


8. Still in the Explode dialog box, click **Ok**

The following warning message is displayed, click Yes



This is what you obtain: the tracks are created and displayed in the geometry area . A sequence is automatically created and displayed in the specification tree



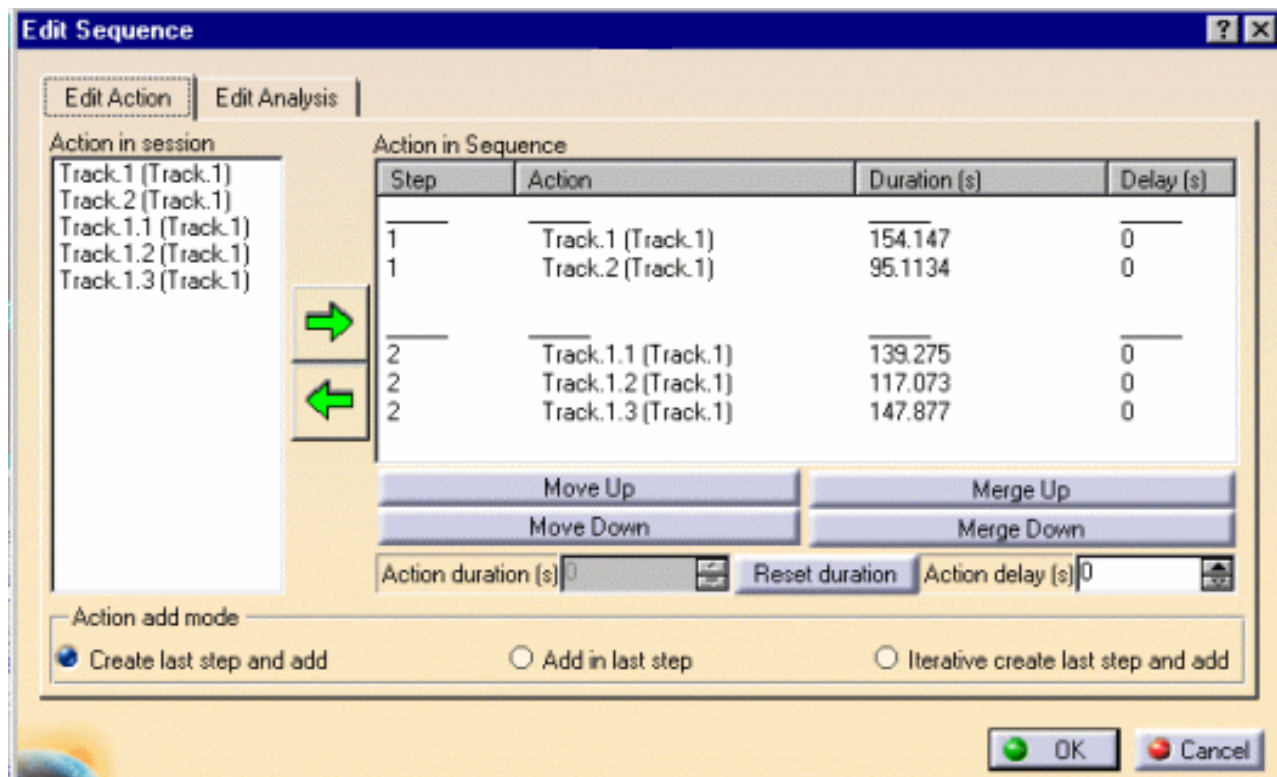
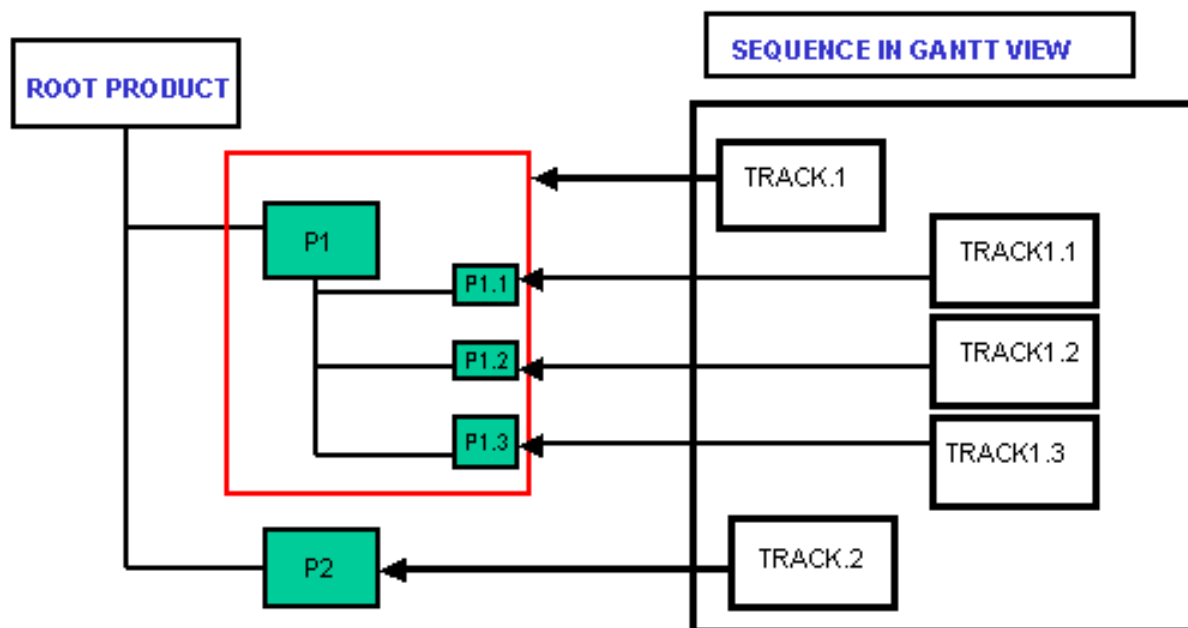


**9.** Double-click Sequence.1 in the specification tree. Play your sequence as desired.



The created sequence reflects exactly the product structure (simultaneous or consecutive tracks are synchronized with the product structure)

The picture below shows you how the generated sequence is organized in a Gantt chart



# Advanced Tasks

DMU Fitting Simulator provides easy methods to define a multi-shuttle that will then be recorded and replayed. Simultaneously to the replay or after, users can analyze the interferences and distance assigned to the assembly.

- Using Multiple Shuttles
- Using Track and Sequence Capabilities
- Converting a Simulation into a Sequence
- Path Finder
- Validating a Motion
- Swept Volume

# Using Multiple Shuttles

This chapter shows how to use multiple shuttles.

[Defining Several Shuttles](#)

[Defining a Shuttle Made of Shuttles](#)

[Recording a Multi-Shuttle Simulation](#)

[Defining a Shuttle Motion using another Shuttle as Reference](#)

[Defining a Shuttle Motion Using a Product as Reference](#)

[Adding a Shuttle in a Simulation](#)

# Defining Several Shuttles



This task shows how to define/edit two shuttles. The final purpose of defining these shuttles to dismount the engine.

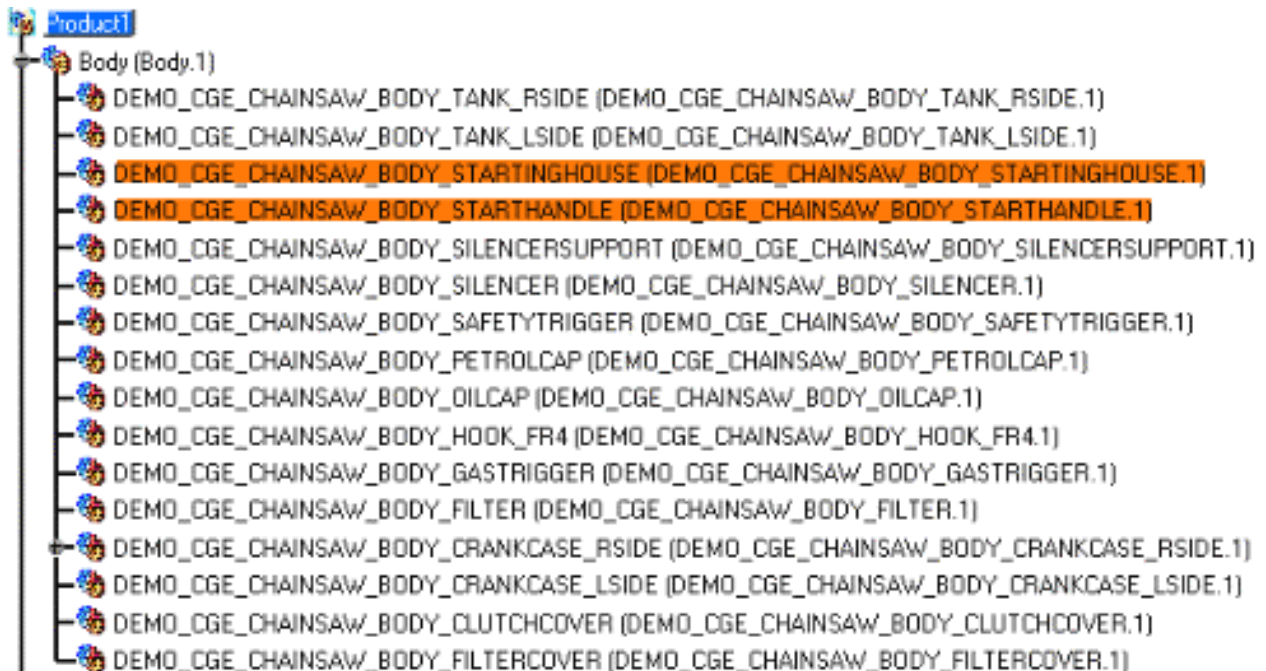


First, perform an explode of product 1 to understand better how to dismount the engine. Open the [SEVERAL\\_SHUTTLES.CATProduct](#) document.



*Let's create a first shuttle.*

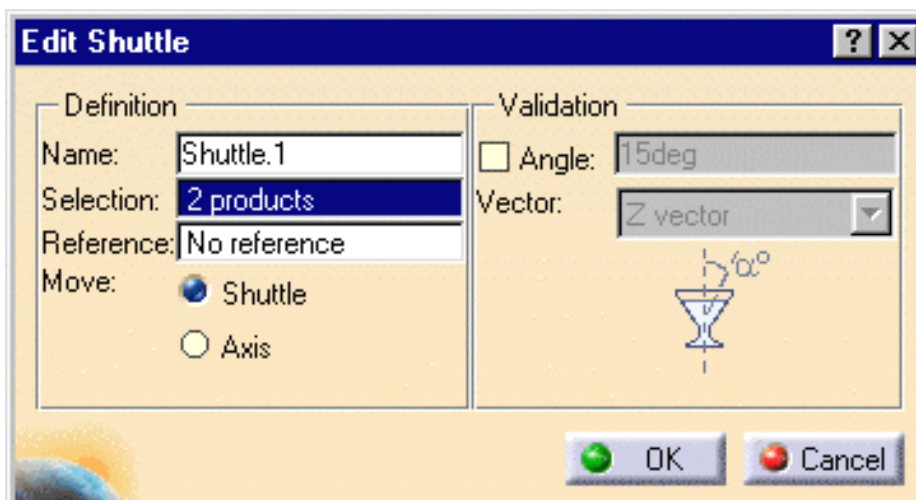
1. Click two objects (for instance, STARTINGHOUSE.1 and STARHANDLE.1) in the specification tree as shown below.



2. Click the **Shuttle** icon .

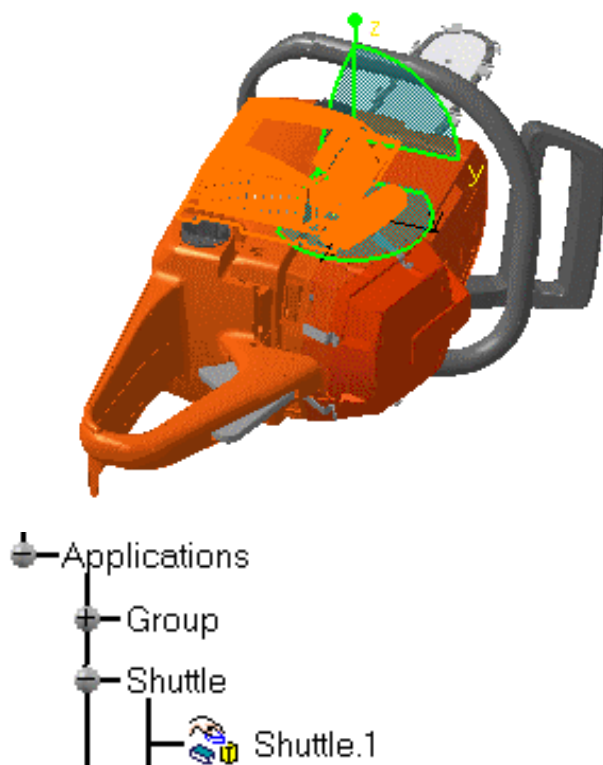
The Edit Shuttle dialog box and the Preview window appear.

3. Press OK to confirm your operation.






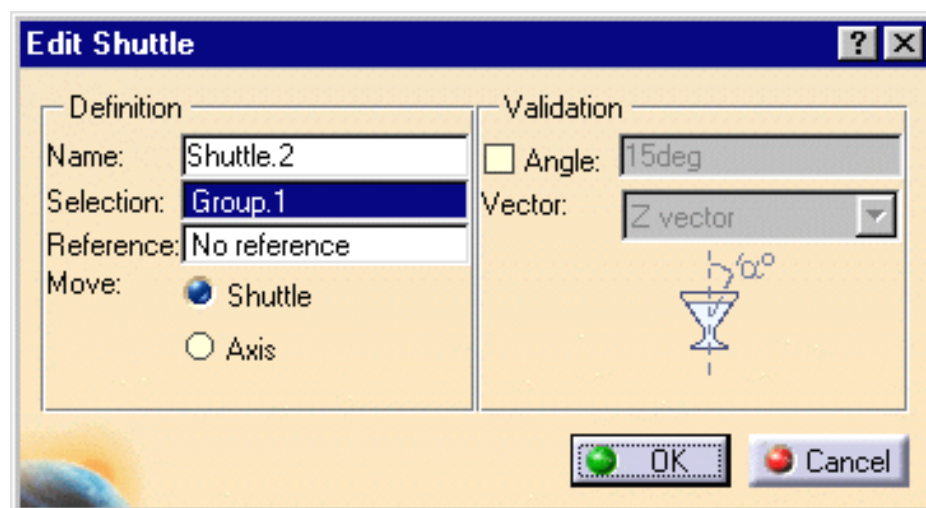
The Shuttle is created and identified in the specification tree.



*Let's create a second shuttle based on an existing group*

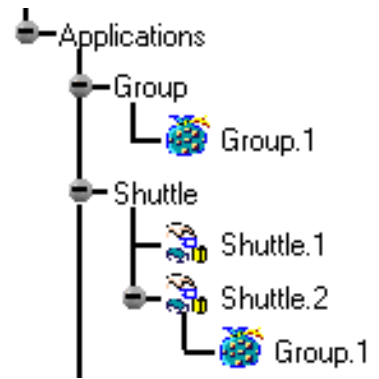
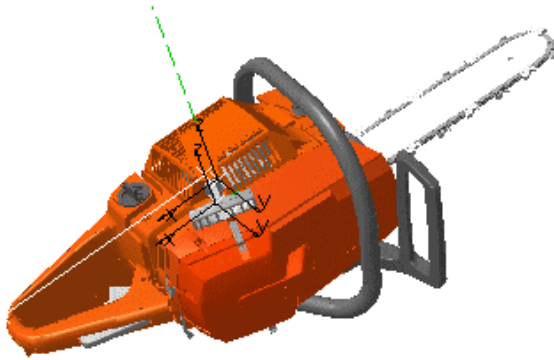
4. You Drag the **Shuttle** icon  and drop it onto the existing group in the specification tree (**Group1**) or click the shuttle icon first and select **Group1** afterwards. The **Edit Shuttle** dialog window appears (Shuttle. 2).

5. Press **OK**.



6. The second shuttle is created and identified in the specification tree.

7. You can edit the shuttles independently. For more information, please refer to [Editing a Shuttle](#).



One manipulator appears on each shuttle. These manipulators will allow you [locating and moving](#) the shuttles to the desired location.



# Defining a Shuttle Made of Shuttles



This task shows how to prepare a dismounting operation defining a shuttle made of shuttles. You decide to dismount the whole clutch system included in the engine group. Three shuttles are defined in our sample:

- Drum-Shuttle
- Hub-Shuttle
- Shoe-Shuttle

In our example, you no longer need to define a fourth shuttle containing the required objects but integrate the Hub-shuttle and Shoe-Shuttle into the Drum-Shuttle.

As you replay the simulation, the three shuttles are single entities but if you need to move the whole clutch system, you will only move the Drum-Shuttle.



Open the [ENGINE\\_WITH\\_SHUTTLES.CATProduct](#) document.

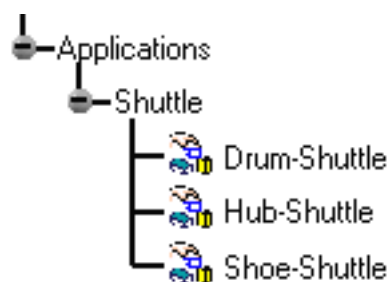
Please refer to [Defining a Shuttle](#) and [Editing a Shuttle](#).

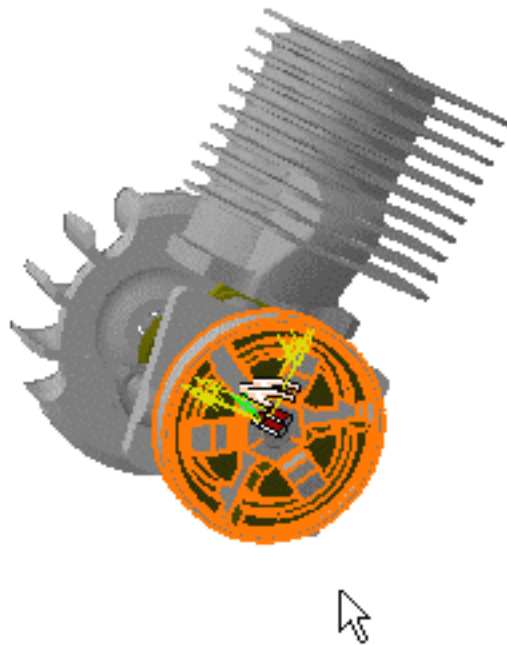


You can use the drag and drop capability create shuttles in existing shuttles. Using drag and drop can help to prepare your dismounting operation more carefully. You can reorganize your assembly into a "shuttle" specification tree which corresponds to a logical dismount operation. The "shuttle" specification tree reflects your product specification tree and the logical dismount procedure.

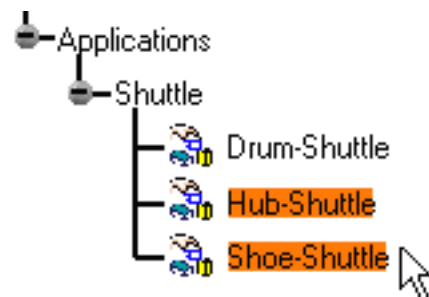


*The shuttles are created and identified in the specification tree*

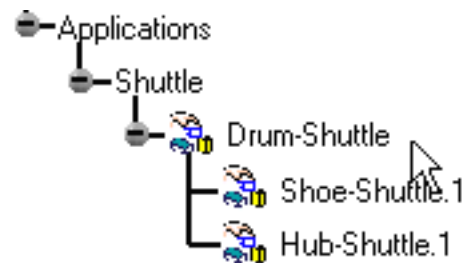




1. Multi-select the Hub-Shuttle and the Shoe-Shuttle

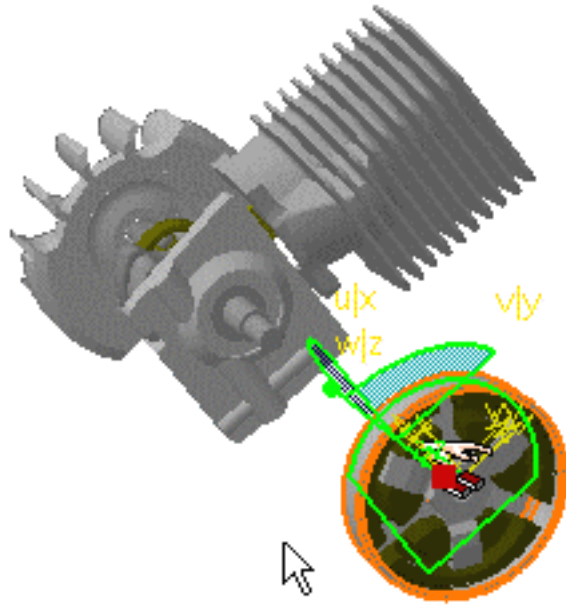


2. Drag your selection onto the Drum-Shuttle object

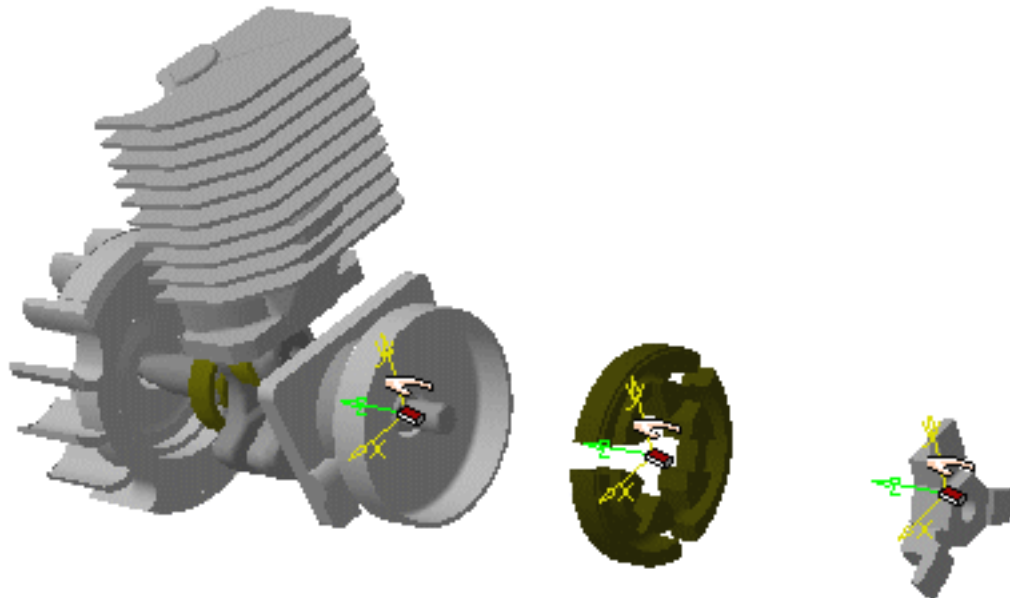


You created a shuttle made of Shoe-Shuttle and Hub-Shuttle which reflects the logical assembly.

3. Double-click the Drum-Shuttle in the specification tree  
The Edit Shuttle dialog box and Preview window are displayed
4. Move the Drum-Shuttle  
The Hub-Shuttle and the Shoe-Shuttle move alongside



5. Click **Ok** Edit Shuttle dialog box to exit.
6. Double-click the Hub-Shuttle.
7. Move the Hub-Shuttle as required
8. Proceed in the same manner for the Shoe-Shuttle.  
This is what you obtain:



*You are now ready to record a simulation.* Please refer to the next task [Recording a Multi-Shuttle Simulation](#).



# Recording a Multi-Shuttle Simulation



This task shows how to simulate a more than one shuttle fitting simulation. You are going to record a simulation with three shuttles.

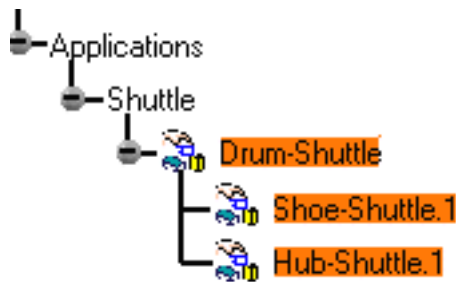


Open the [SHUTTLES\\_MADE\\_OF\\_SHUTTLES.CATProduct](#) document.




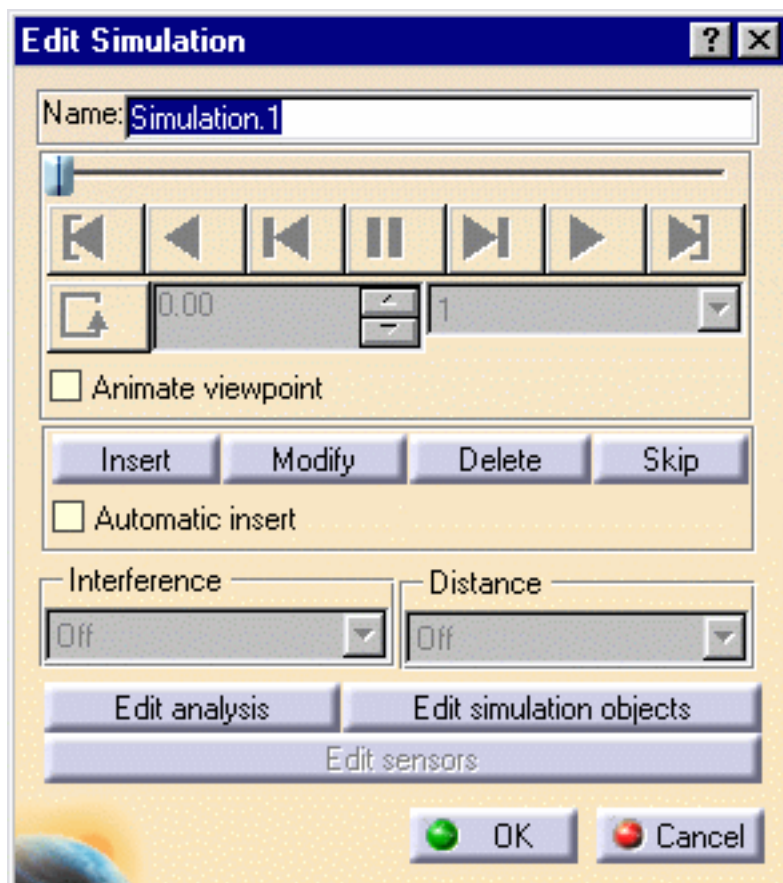
**1.** Multi-select in the specification tree:

- Drum-Shuttle
- Hub-Shuttle
- Shoe-Shuttle



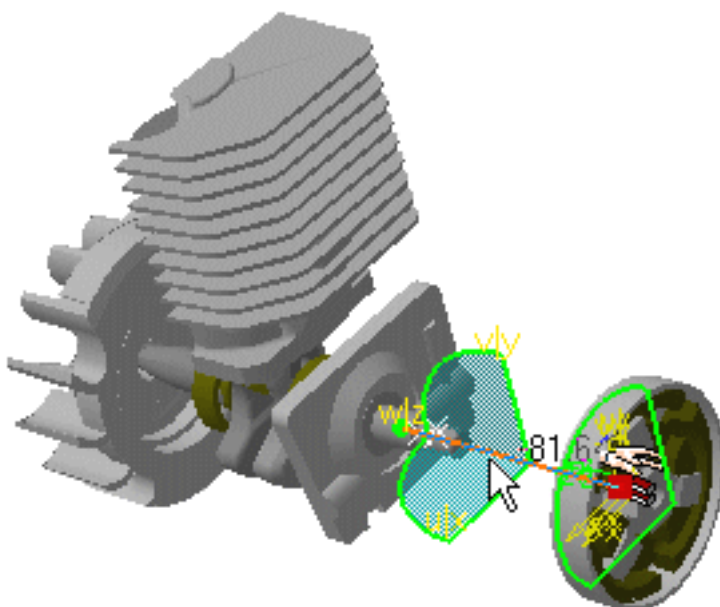
**2.**

Click the Simulation icon  .  
The Edit Simulation dialog box is displayed



*The 3D compass lets you move the shuttle*

3. Move the Drum-Shuttle to the desired location via the manipulator.



4. Click the **Insert** switch to record the shot.



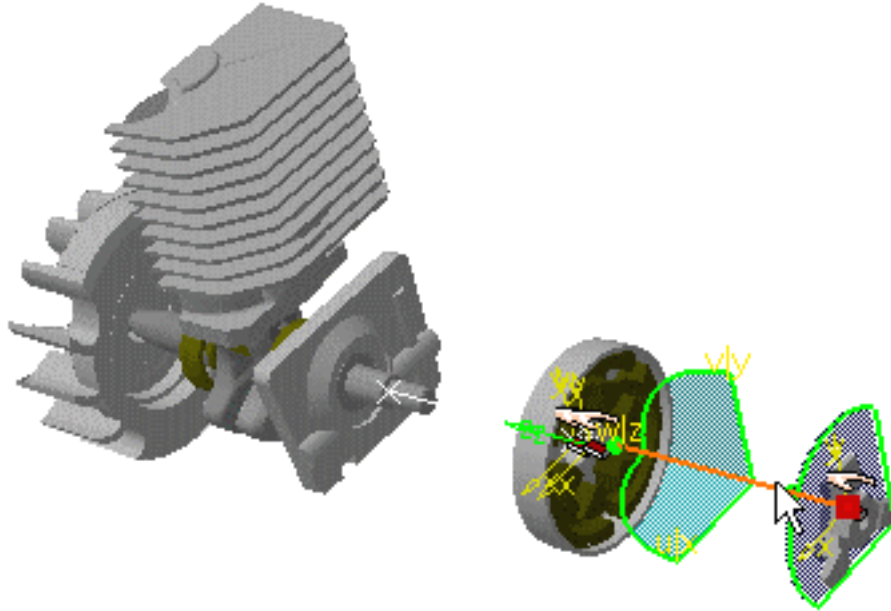
**Insert** means that you record and insert shots inside the scenario. The initial location of the shuttle is automatically recorded as a starting shot. If you need to reposition the shuttle, please delete the first position or modify it



5. Now select the Hub-Shuttle. Three possibilities are available:
  - Select the Hub-Shuttle in the specification tree
  - Select the Hub-Shuttle axis in the geometry area
  - Select the product in the geometry area

The 3D compass snaps onto your selection

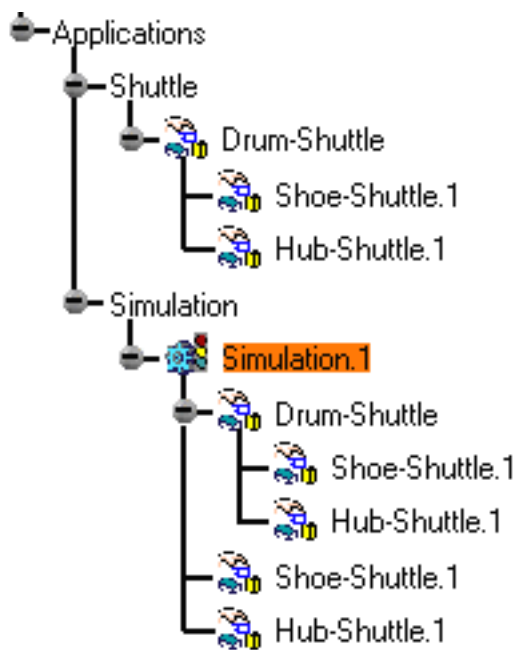
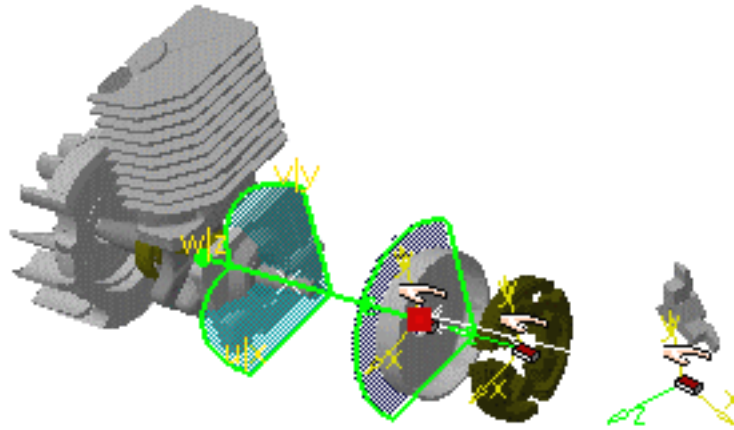
6. Move the Hub-Shuttle to the desired location using the 3D compass.



7. Click the **Insert** switch and record the desired shot.
8. Select the Shoe-Shuttle axis.
9. Drag the Shoe-Shuttle to the required location.
10. Record this motion clicking the **Insert** switch.
11. Press OK in the Edit Simulation dialog box

You recorded the motion of the three shuttles

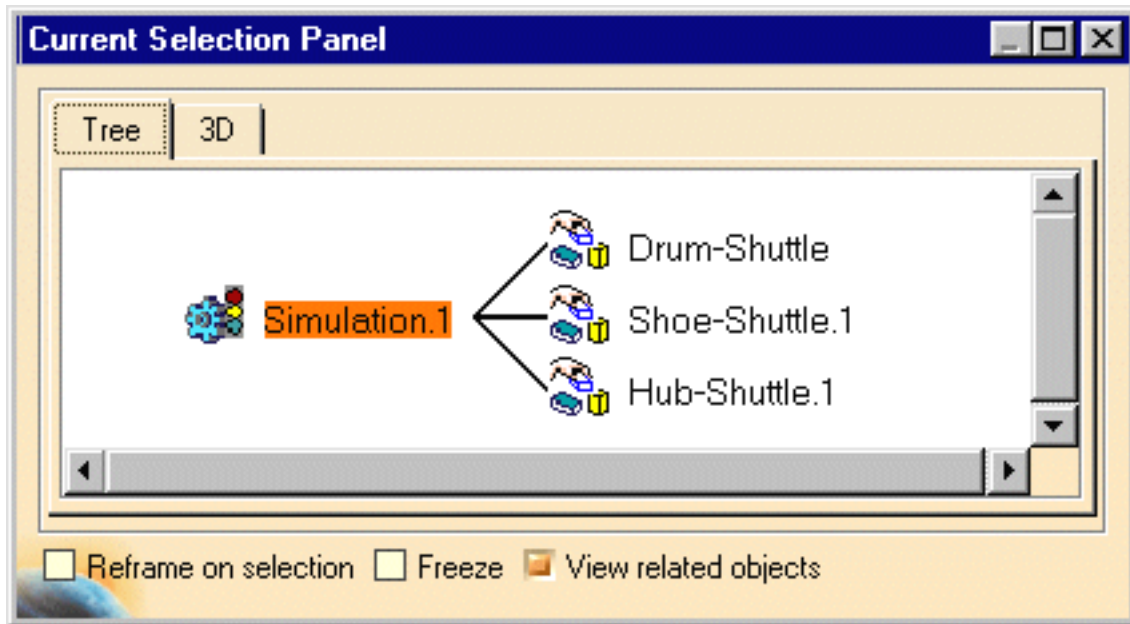
The simulation object is created and identified in the specification tree



- 12.** You can now create a replay. For more information, please refer to [Generating a Replay](#).



To analyze an object and its related objects under the form of a specification tree or in the 3D, select the current selection icon from the Simulation toolbar or select **Analyze - > Current Selection Panel** from the menu bar. The Current Selection Panel dialog box appears as follows:



# Defining a Shuttle Motion Using Another Shuttle as Reference



This task shows how to define a shuttle motion using another shuttle as reference.



Open the [SHUTTLE\\_REFERENCE.CATProduct](#) document.

You decide to dismount the starthandle. You created two shuttles:

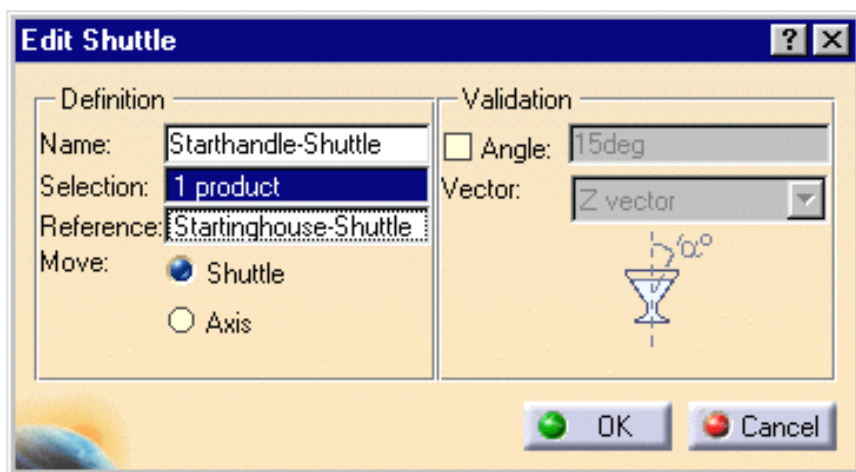
- Starthandle-Shuttle
- Startinghouse-Shuttle

For more details, please refer to [Defining a Shuttle](#).



*You are going to record the motion relative to another shuttle.*

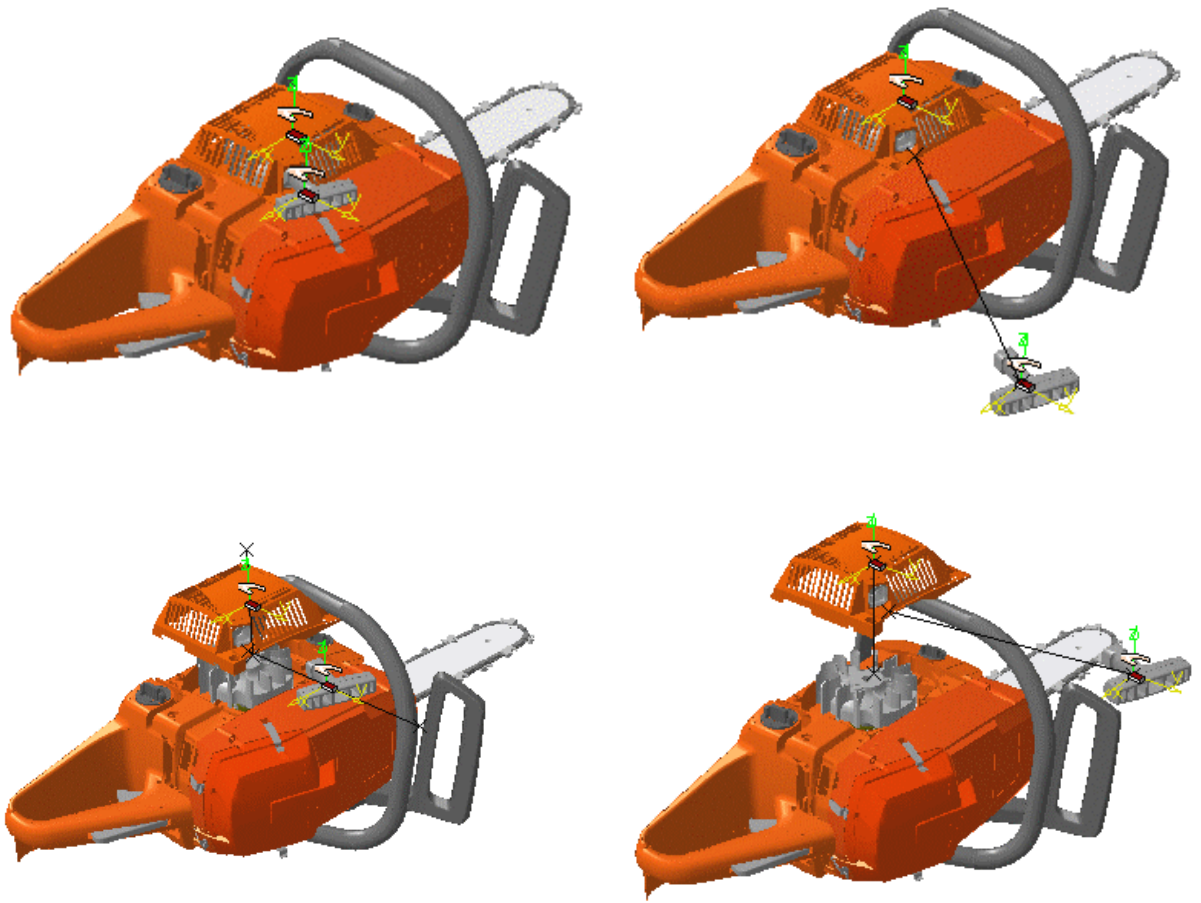
1. Double-click the Startinghandle-Shuttle in the specification tree.
2. Click the Reference field from the Edit Shuttle dialog box, select Startinghouse-Shuttle in the specification tree.
3. Click **Ok** to confirm your operation
4. Multi select Starthandle-Shuttle and Startinghouse-Shuttle either in the specification tree or in the geometry area.



5. Select Insert->Simulation.



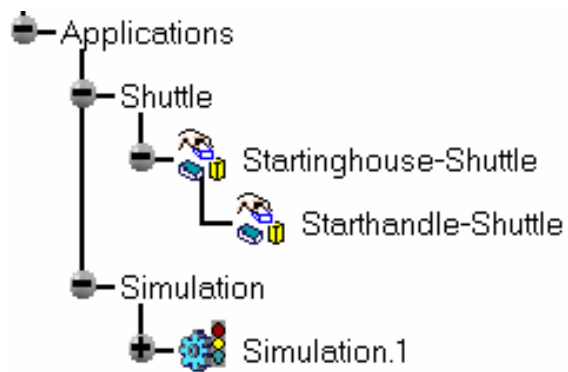
6. Select Starthandle-Shuttle in the specification tree and click Insert.
7. Select Startinghouse-Shuttle and click Ok to end your simulation recording.



The Starhandle-Shuttle and Startinghouse-Shuttle motions are synchronized. The Startinghouse-Shuttle moves relative to the Starhandle-Shuttle.



Note that when you define a reference shuttle in this manner the specification tree is updated as if you had defined a **shuttle made of shuttles**.



# Defining a Shuttle Motion Using a Product as Reference



This task shows how to define a shuttle motion using a product as reference.

In our example, we study the dismounting of the starthandle. The Chainsaw design is still subject to changes and you know the startinghouse is going to be repositioned because the engine position is not defined yet.

You need the starthandle (shuttle) to be associative with the startinghouse (reference product) within the simulation to keep the consistency of the motion.

The capacity to select a product as reference will keep your simulation consistent after design changes (here on the startinghouse)

- You already defined the dismounting path of the starthandle



Open the [CHAINSAW1.CATProduct](#) document.

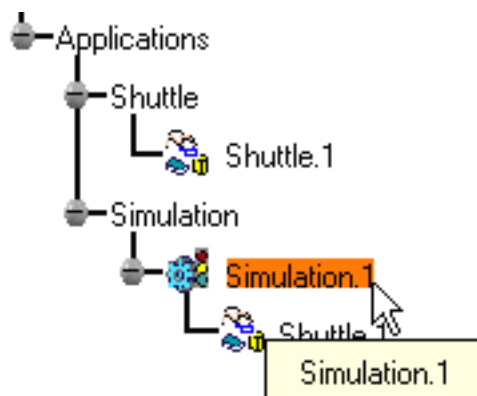
You defined a simulation reflecting the dismounting path of the starthandle. You defined a shuttle on the starthandle

- Shuttle.1

For more details, please refer to [Defining a Shuttle](#).

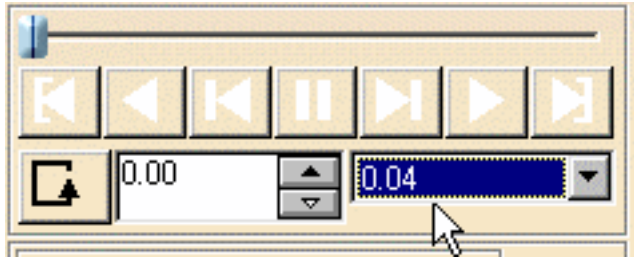


1. Double-click Simulation.1 in the specification tree.

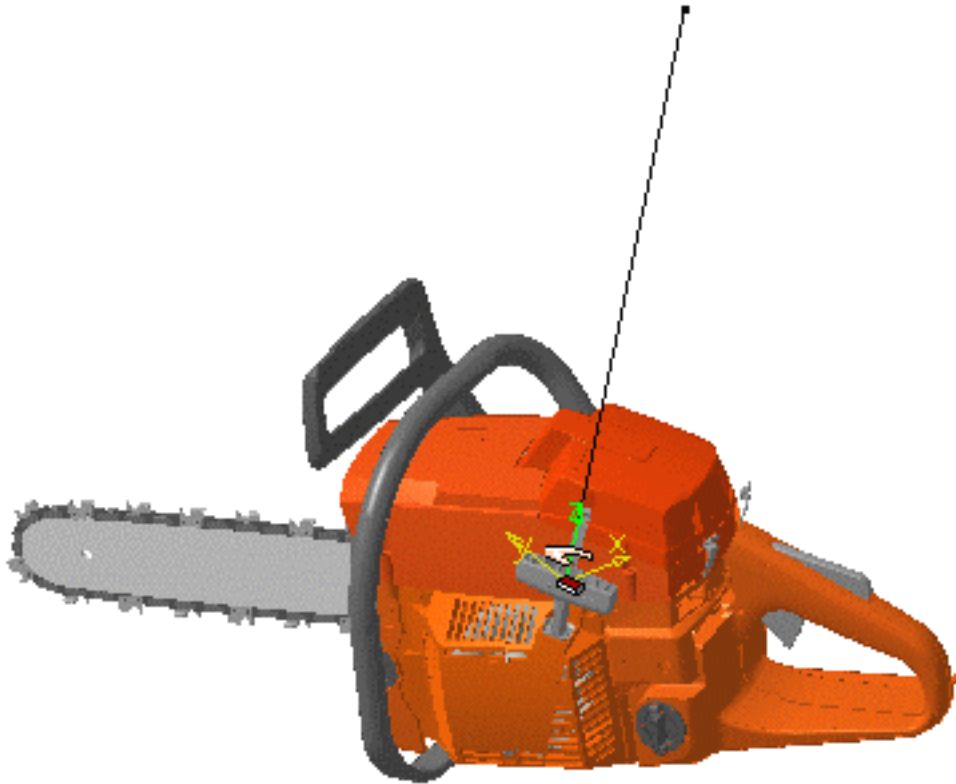


The Edit Simulation dialog box and Preview window appear

2. Modify the interpolation value as desired. For example: 0.04



3. Click the Play Forward button.  
The recorded motion is replayed.

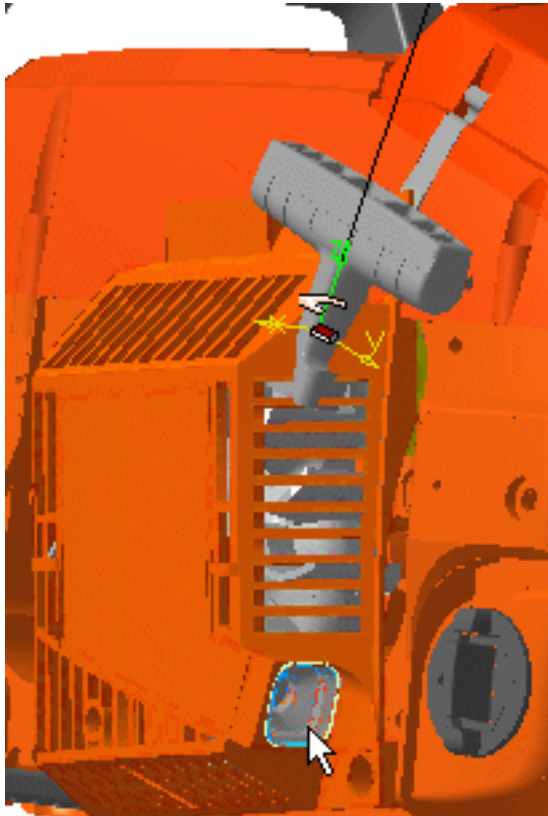


*Now the startinghouse has been repositioned*

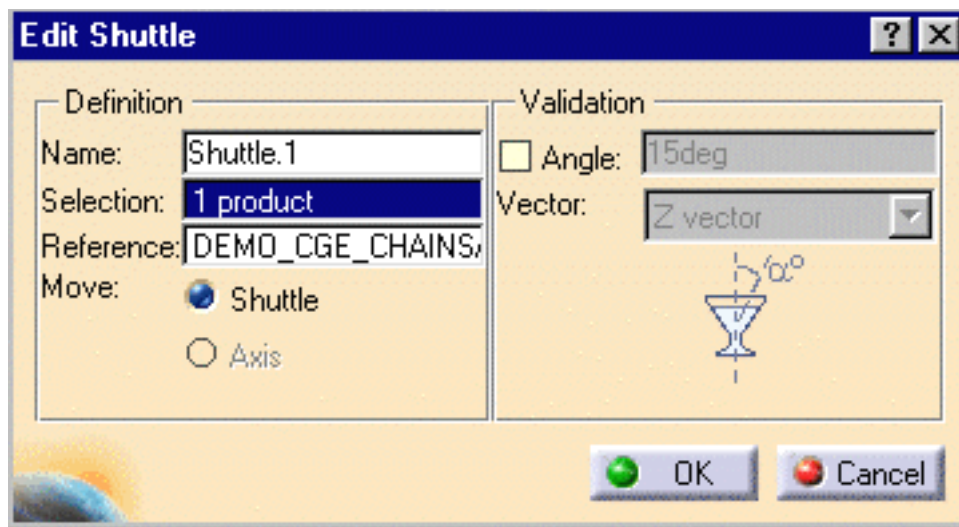
4. Open [CHAINSAW2.CATProduct](#).

*The design has changed and the shuttle motion is no longer valid.*



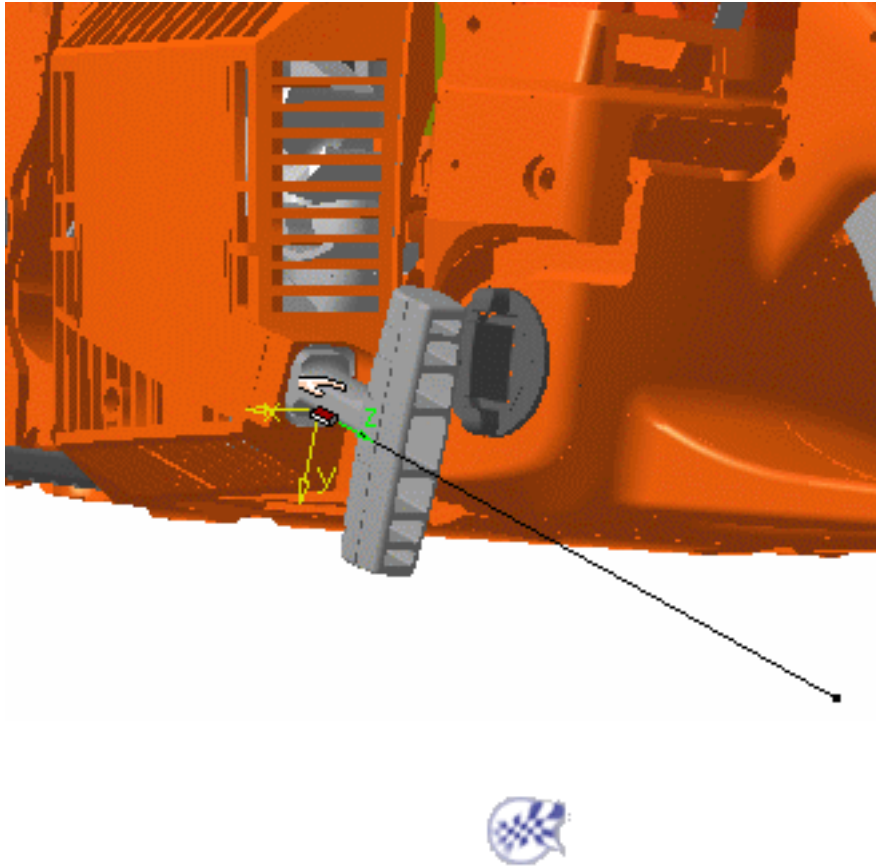


5. Double-click Shuttle.1 in the specification tree
6. In the Reference field from the Edit Shuttle dialog box, the CHAINSAW\_BODY\_STARTINGHOUSE.1 is selected.





7. Click Ok
8. Double-click the Simulation.1 in the specification tree
9. Replay your simulation. The shuttle motion is updated. The design change is taken into account.




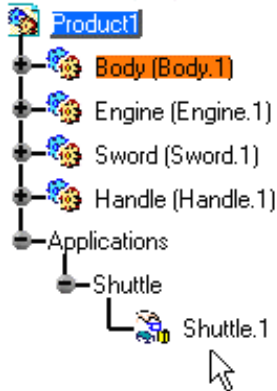



# Adding A Shuttle in a Simulation

 This task shows how to add a shuttle in a simulation. In our example you need to move first the starthandle to dismount the startinghouse.

 Open the [ADDING\\_SHUTTLE.CATProduct](#) document.  
You created a shuttle with the startinghouse. (referred as shuttle.1)

 1. 1. In the specification tree, click **Shuttle.1**.

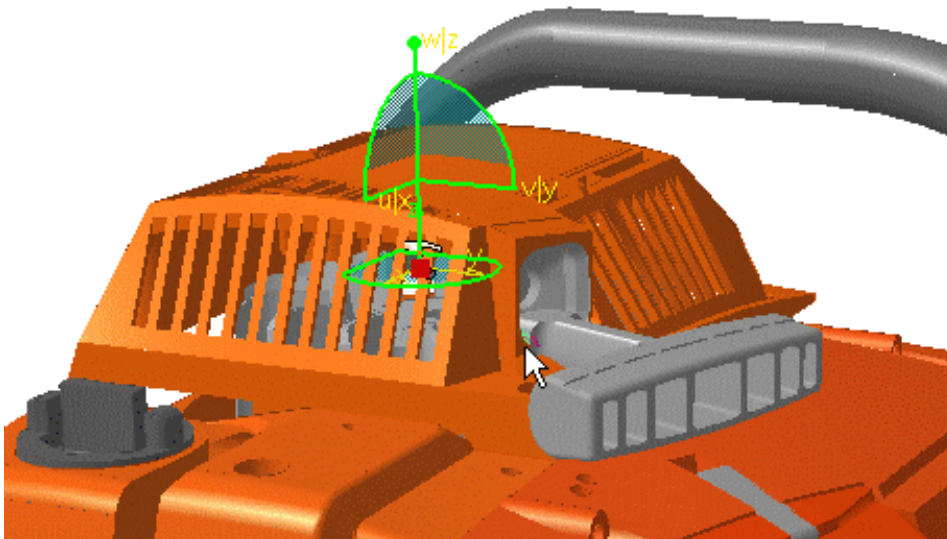


2. Click the Simulation icon .


The Edit Simulation dialog box and Preview window appear.

The starting shot (shuttle initial position) is automatically recorded.

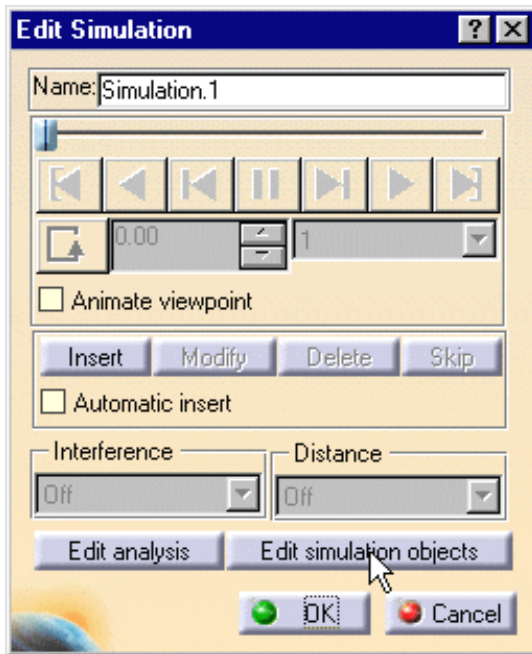
3. Move the shuttle to the desired location with the manipulator.
4. Click the insert switch and record the desired shots



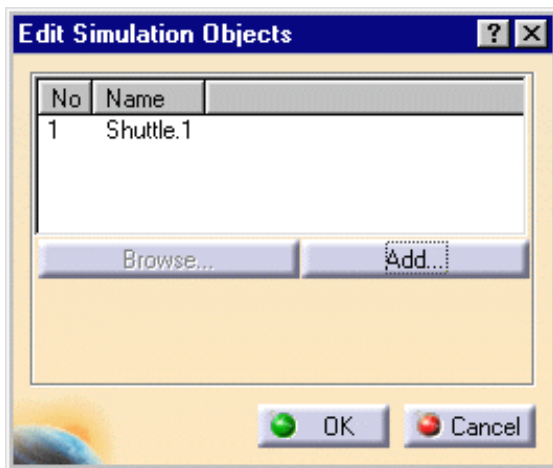
*You need to move the starthandle, for this: create a shuttle without exiting the simulation command*

5. Click the Shuttle icon .

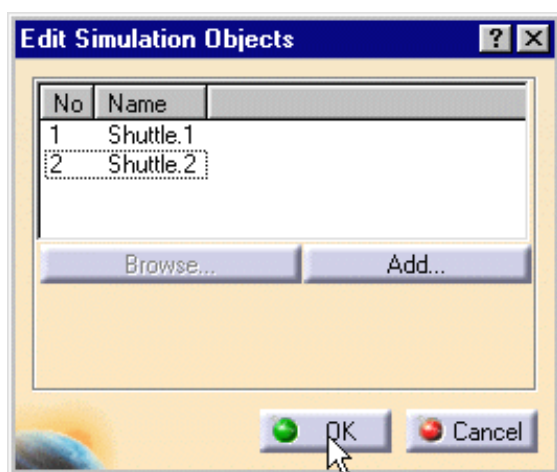
6. Select DEMO\_CGE\_CHAINSAW\_BODY\_STARTHANDLE.1 either in the geometry area or in the specification tree (expand the Body item)
7. Click OK to confirm the shuttle creation.
8. Click Edit simulation objects in the Edit Simulation dialog box as shown below:



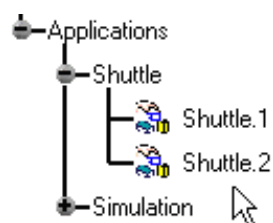
The Edit Simulation Objects dialog box is displayed.



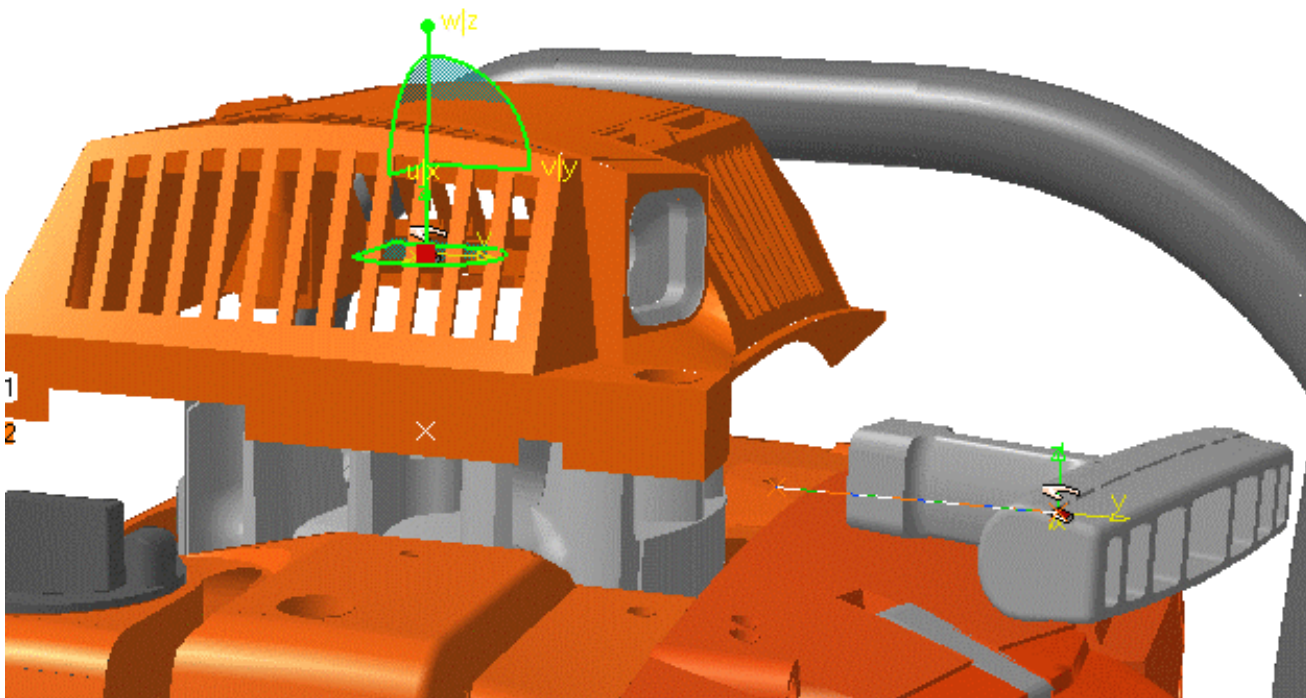
9. Click Add and select shuttle.2 in the Select dialog box and click OK
10. Click OK in the Select dialog box.
11. Click OK in the Edit Simulation Objects dialog box to confirm your operation.



The shuttle.2 is identified in the specification tree



12. Move the shuttle.2 to the desired location with the manipulator
13. Click the **insert** switch and record the desired shots.



*You can dismount the startinghouse.*



# Using Track and Sequence Capabilities

This chapter aims at illustrating the useful combination of tracks and sequences throughout a step-by-step dismounting scenario:

## Road Map

### **Phase 1:**     [Dismount the Chainsaw Handle](#)

- Creating a Chainsaw\_Handle Shuttle
- Performing clash detection (standard clash tool) within the shuttle command.
- Hiding the gray handle object (CHAINSAW\_BODY\_HANDLE.1)
- Resetting products position
- Creating a Chainsaw handle shuttle track

### **Phase 2:**     [Illustrate the dismounting operation](#)

- Displaying the hidden gray handle (CHAINSAW\_BODY\_HANDLE.1)
- Creating a visibility action on the gray handle object
- Defining a sequence containing the handle shuttle track and the created visibility action
- Running the defined sequence

### **Phase 3:**     [Finalize the dismounting operation:](#)

- Editing the sequence
- Creating a track on CHAINSAW\_BODY\_CLUTCHCOVER.1 (Clutchcover\_Track) within the sequence command
- Running the sequence
- Modifying action duration and scheduling within the sequence
- Running the sequence again
- Modifying the DMU player parameters
- Creating a track on CHAINSAW\_BODY\_SAFETYHANDLE.1

**Phase 4:**    **Generate a film**

- Creating a camera (using the **View ->Named Views...**) with 3 different viewpoints focusing on CHAINSAW\_BODY\_SAFETYHANDLE.1, on CHAINSAW\_BODY\_CLUTCHCOVER.1 and on the Chainsaw\_Handle Shuttle
- Playing the created camera
- Defining a camera track
- Defining a new sequence
- Customizing the player parameters
- Editing the camera track
- Defining a sequence and replay it (in a camera window **Window** )
- Multi-selecting and Hiding the current tracks
- Generating a Film (in AVI format) using **Tools->Image->Video..**

# Dismounting the Chainsaw Handle



This scenario is aimed at dismounting the chainsaw handle: each of the following products belongs to the body set.



For this you will need to perform various operations such as:

- Creating a Handle\_Shuttle
- Performing clash detection
- Hiding handle
- Resetting products
- Creating a Handle\_Shuttle track

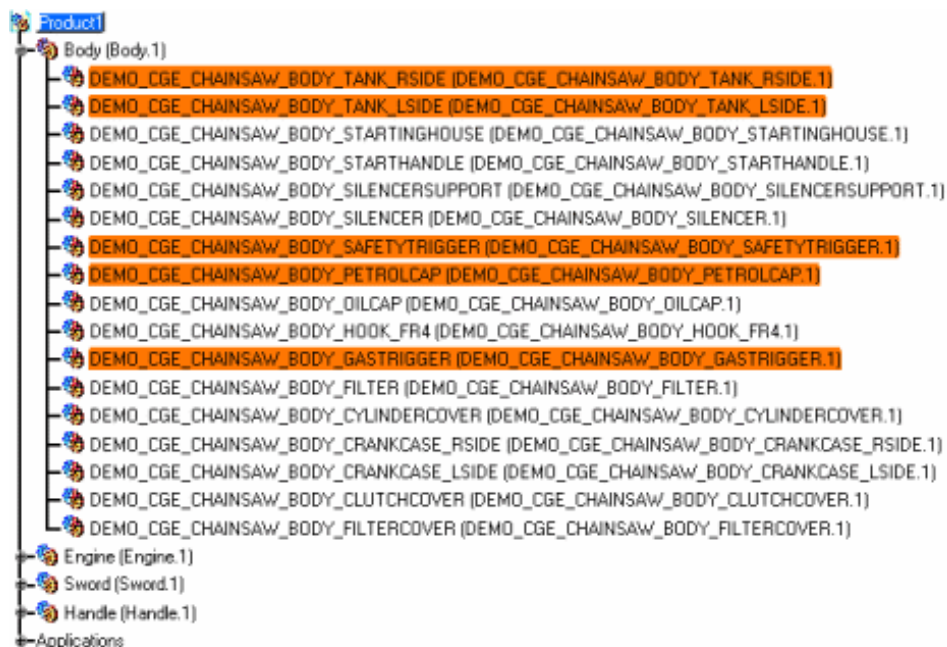
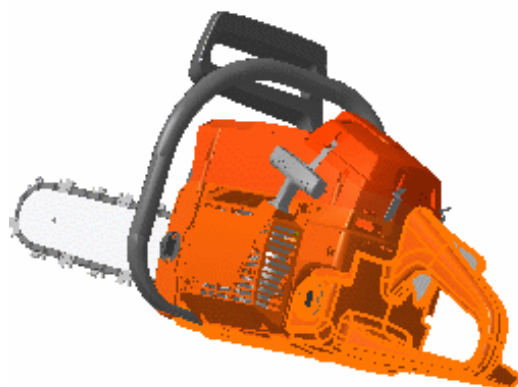


Open the [CHAINSAW.CATProduct](#) document.



1. Expand the Body node, and select in the specification tree or in the geometry area:
  - (CHAINSAW\_BODY\_TANK\_RSIDE.1)
  - (CHAINSAW\_BODY\_TANK\_LSIDE.1)
  - (CHAINSAW\_BODY\_SAFETYTRIGGER.1)
  - (CHAINSAW\_BODY\_PETROLCAP.1)
  - (CHAINSAW\_BODY\_GASTRIGGER.1)





2.



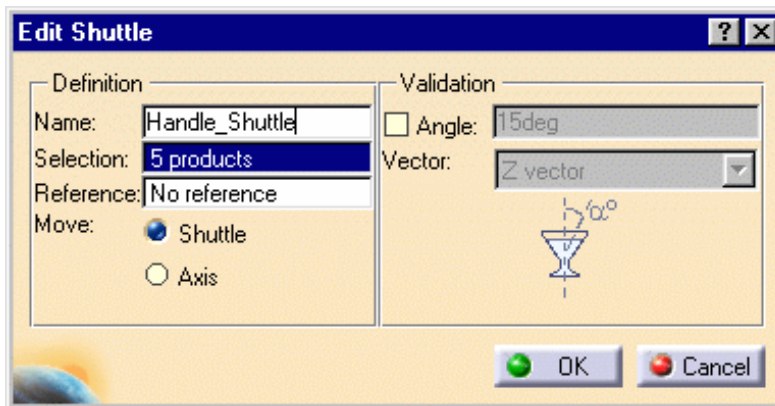
Click the Shuttle icon from the DMU simulation toolbar.

The preview window gives the shuttle content






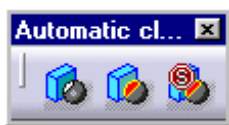
3. Give the shuttle a meaningful name (Handle\_Shuttle for instance)



Now still in the shuttle command you are going to move the shuttle and perform a rough clash detection to prepare your dismounting path.

- 4.

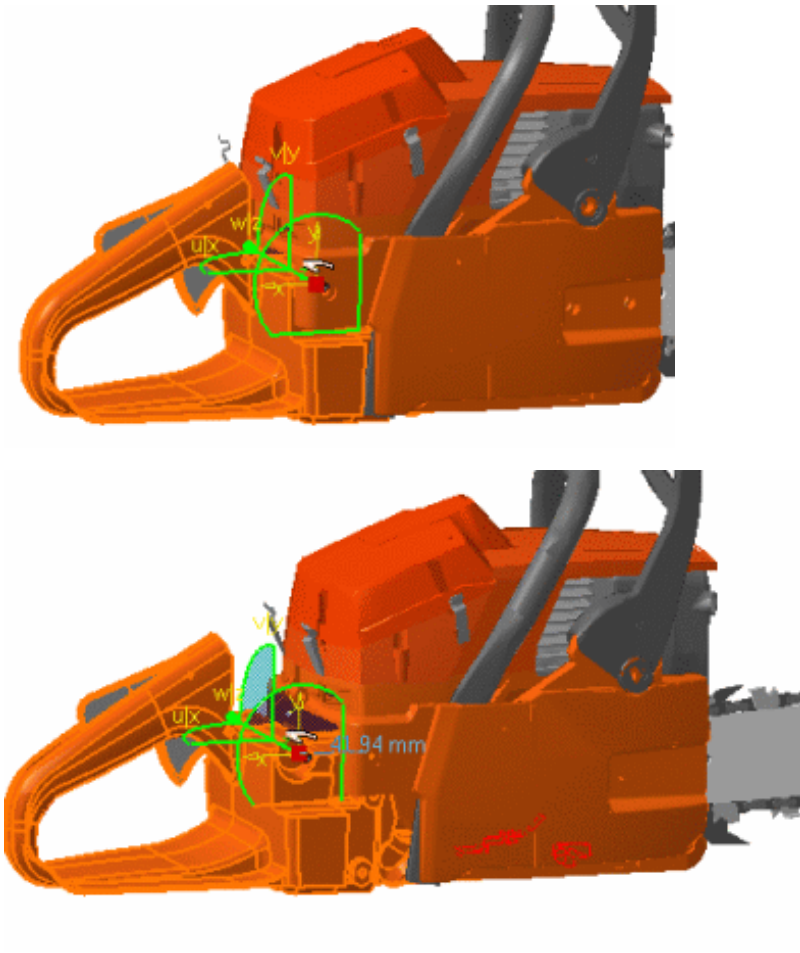
Click the arrow within the Clash Detection icon  from the DMU Check toolbar. Undock the toolbar if necessary.



- 5.

Set the Clash detection to on 

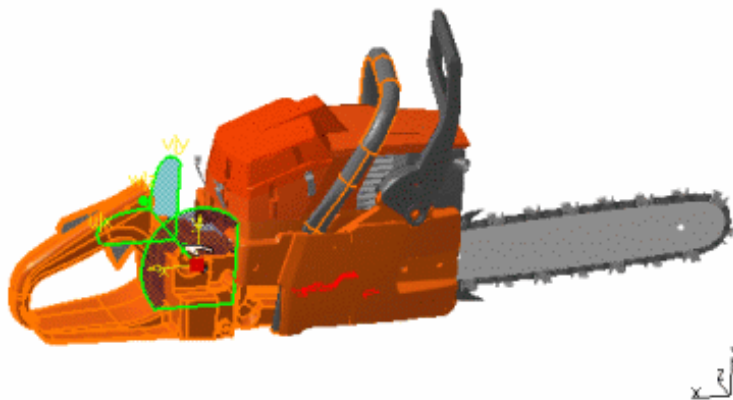
6. Move the shuttle, using the 3D compass. There is a collision between the Handle\_Shuttle and the Clutch cover and the object BODY\_HANDLE makes the dismounting operation impossible



There is a collision between the Handle\_Shuttle and the Clutchcover, besides, the object


(CHAINSAW\_BODY\_HANDLE) makes the dismounting operation impossible. You are going to hide it but now validate the shuttle creation

CHAINSAW\_BODY\_HANDLE.1



7. Click Ok

8.

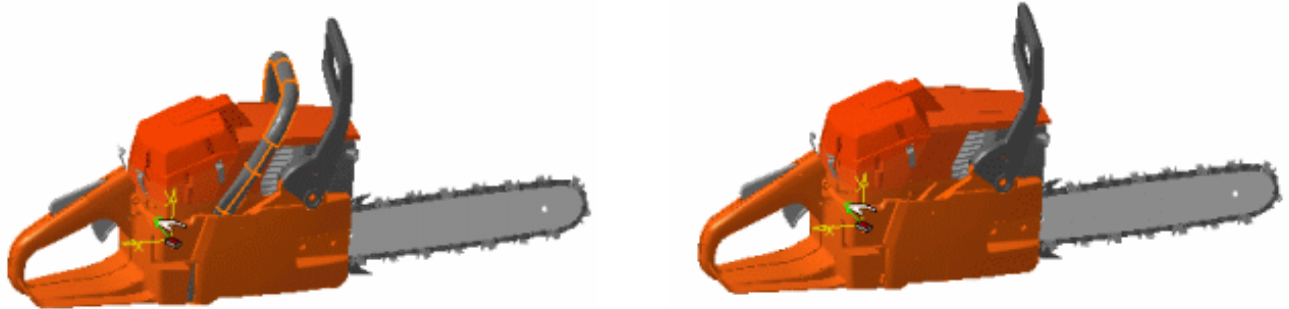
The position is kept when exiting the command. In this case, you need to restore the initial view click the Reset icon .

9. Select the (CHAINSAW\_BODY\_HANDLE.1) either in the specification tree or in the geometry area


Click the Hide/Show  icon, or select the **View->Hide/Show->Swap Hide/Show** command.

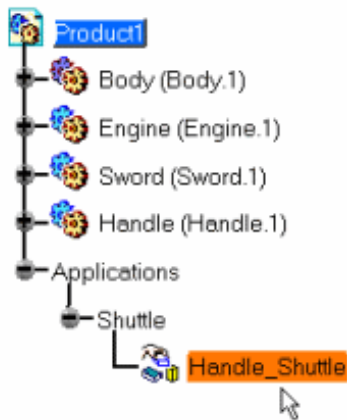
The object is no longer displayed: it has been transferred into the No Show space.

In our example, the CHAINSAW\_BODY\_HANDLE.1 is no longer displayed.

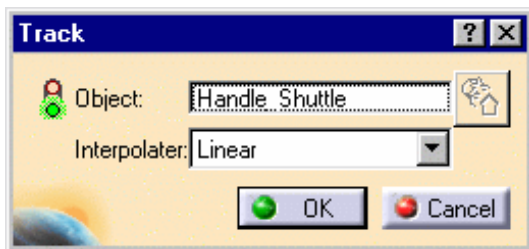


Now, you are ready to define a track for the created *Handle\_Shuttle* object. For this,

10. Select *Handle\_Shuttle* in the specification tree and click the Track icon  from the DMU Simulation toolbar.



The Track dialog box along with the DMU recorder and player are displayed:

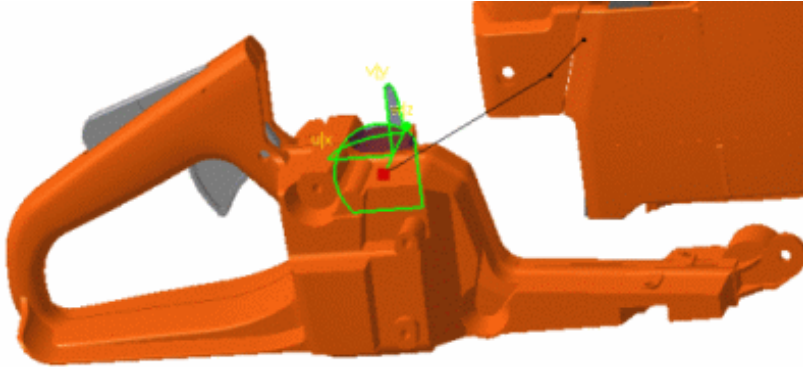


9. Record your track shuttle, Move the 3D compass and click Record at each time position of your choice. If you double-click Record, the automatic insert mode is activated.

**Note:** You can also use the Insert Key instead of clicking Record.

Please refer to [Recording a Track](#)

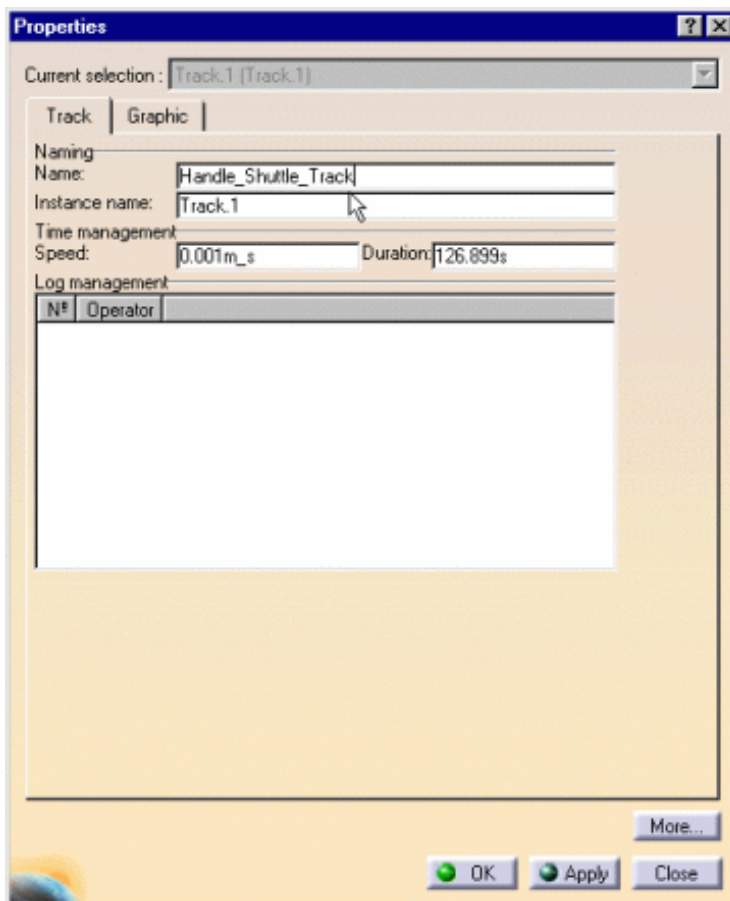
When satisfied, click Ok.



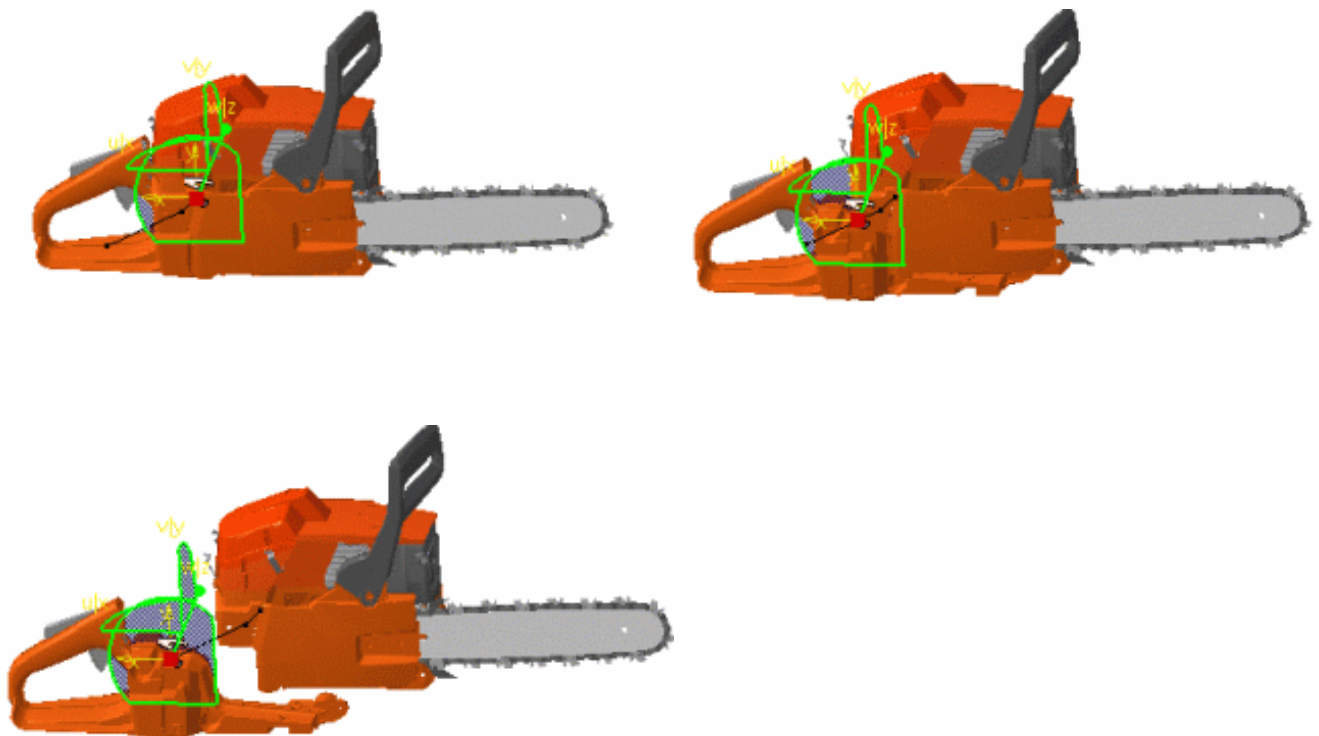
10. Rename your track with a meaningful name: in our example, choose Handle\_Shuttle\_Track. for this:

Right-click Track.1 object in the specification tree and select the properties item from the contextual menu displayed

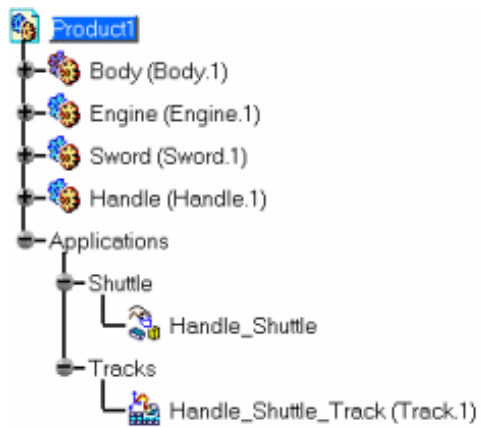
Click the track tab.



11. Replay your Handle\_Shuttle\_Track using the VCR buttons from the dedicated player



You created a shuttle and a track using this shuttle:



Now, you are going to illustrate this dismounting operation.



# Illustrating the Dismounting Operation



This scenario is aimed at illustrating the chainsaw handle dismounting operation

For this you will need to perform various operations such as:

- Displaying the CHAINSAW\_BODY\_HANDLE.1 object
- Creating a visibility action on CHAINSAW\_BODY\_HANDLE.1 object
- Defining a Sequence containing the Handle\_Shuttle\_Track and the visibility action
- Running the sequence simulation using the Player.



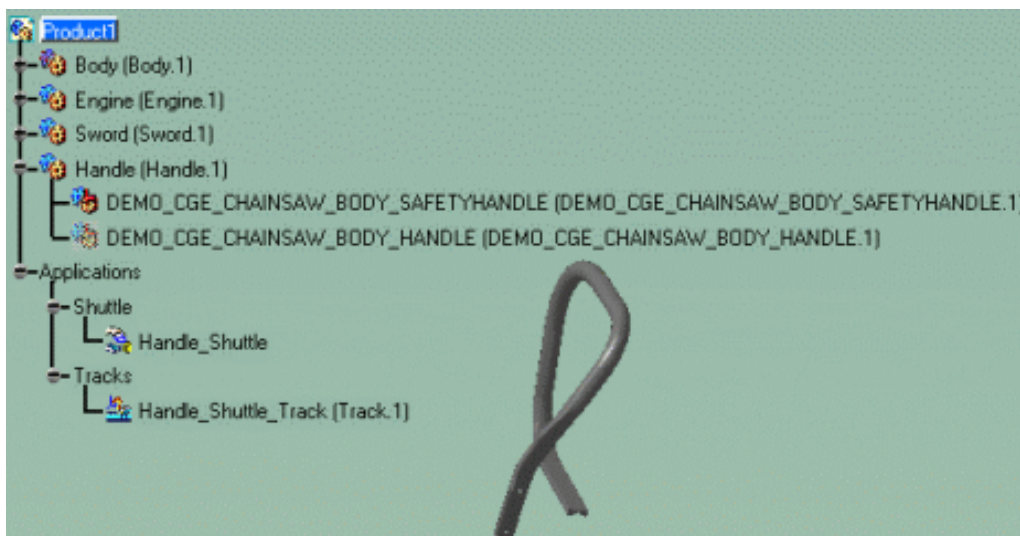
The CHAINSAW.CATProduct document is already open



While, performing the chainsaw handle dismounting operation, you realized other objects needed to be dismounted such as the:

- CHAINSAW\_BODY\_HANDLE.1 object which makes the dismounting impossible (you already hide it)
- CHAINSAW\_BODY\_CLUTCHCOVER.1
- CHAINSAW\_BODY\_SAFETYHANDLE.1

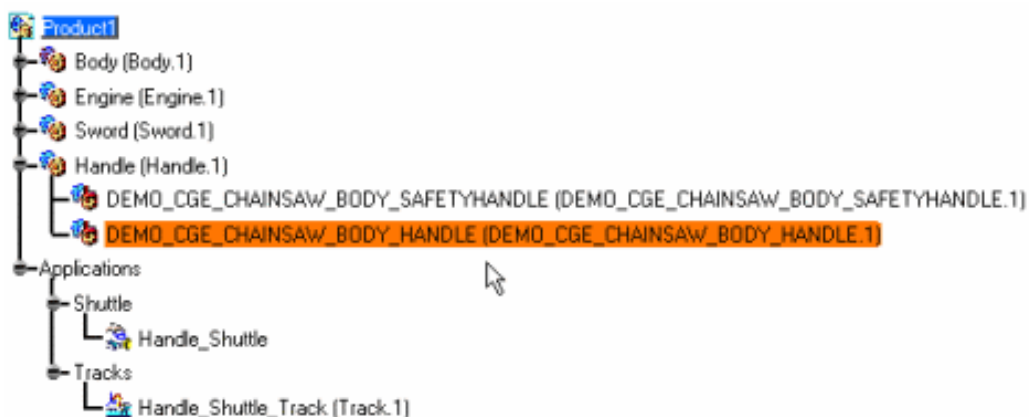



1. The CHAINSAW\_BODY\_HANDLE.1 has been transferred to the no-show area.

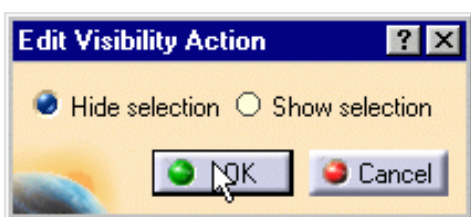


2. Click the No Show  icon to display the contents of the no show area. CHAINSAW\_BODY\_HANDLE.1 object and transfer it back to the Show space using the Hide/Show  icon.
3. Select CHAINSAW\_BODY\_HANDLE.1 object either in the specification tree or in the geometry area





4. Click the arrow within the color action icon to display the entire toolbar. Undock it if necessary, and click the visibility icon .
5. In the Edit Visibility Action dialog box displayed, keep the Hide selection option set as default and click Ok

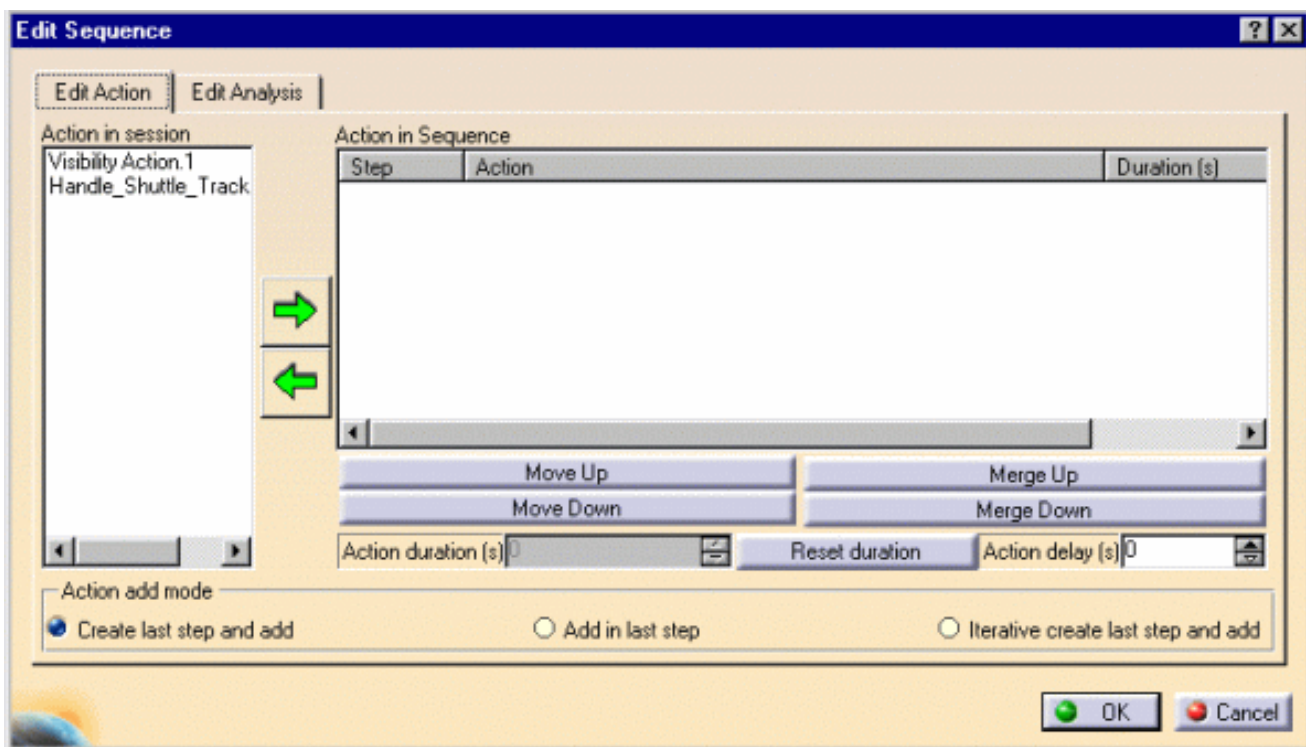


Now you are going to create a sequence


6.

Click the Edit Sequence icon 


The Edit Sequence dialog box is displayed

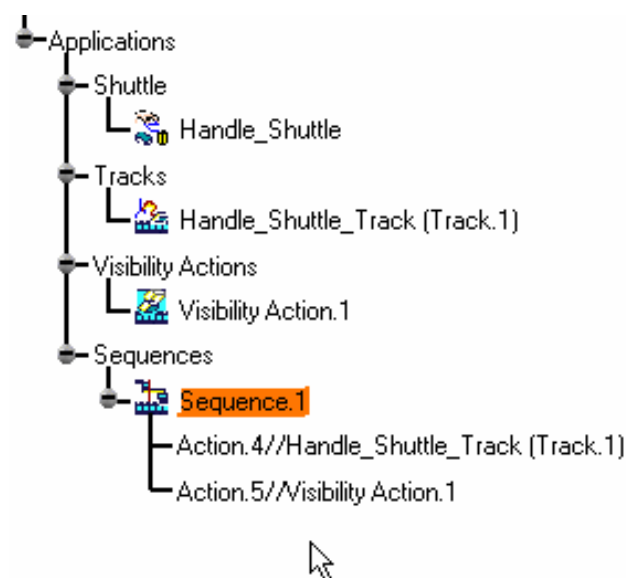
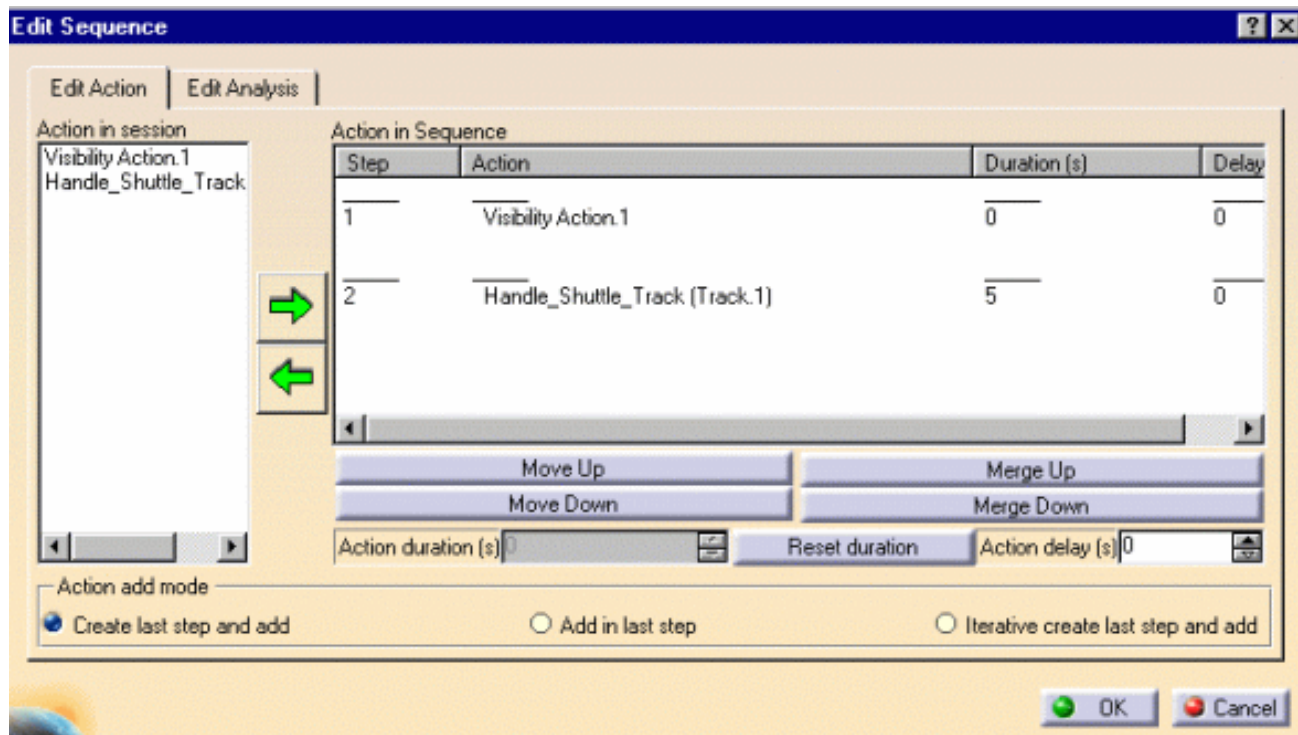


5.

Select Handle\_Shuttle in the "Action in session" list and click . The action is added in the sequence list.

6.

Select Visibility Action.1 in the "Action in session" list and click . The action is added in the sequence list.



7. Use the DMU Player to run your sequence.



The dismounting operation is valid but you are still not satisfied with the result

- We need to dismount the CHAINSAW\_BODY\_CLUTCHCOVER.1 and therefore to create a Clutchcover track.





# Finalizing the Dismounting Operation



This scenario is aimed at illustrating the chainsaw handle dismounting operation

For this you will need to perform various operations such as:

- Editing the sequence
- Creating a track on CHAINSAW\_BODY\_CLUTCHCOVER.1 (Clutchcover\_Track) within the sequence command
- Running the sequence
- Modifying action duration and scheduling within the sequence
- Running the sequence again
- Modifying the DMU player parameters
- Creating a track on CHAINSAW\_BODY\_SAFETYHANDLE.1
- Defining a color action on CHAINSAW\_BODY\_CLUTCHCOVER.1
- Adding and scheduling this color action in the current sequence



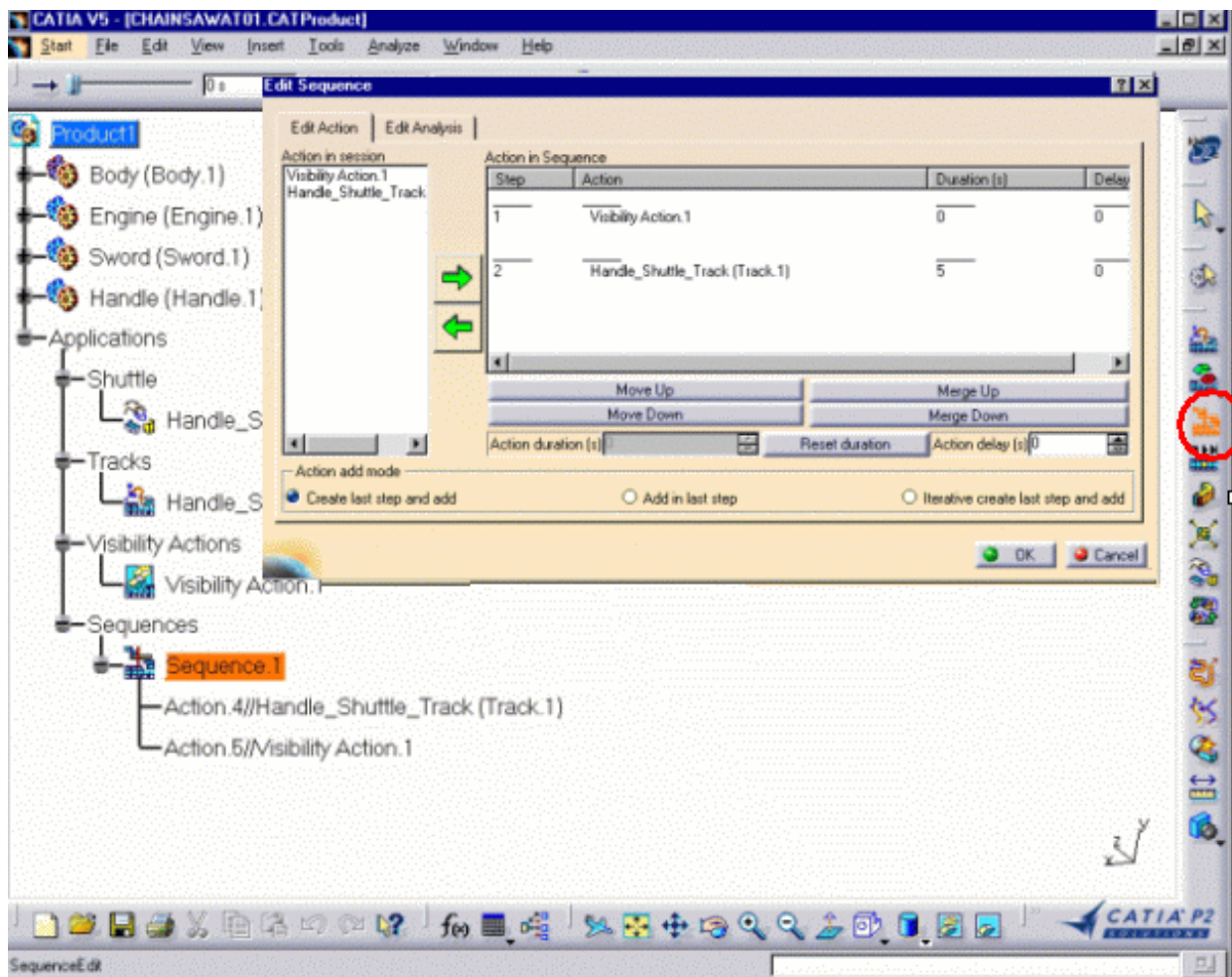
The [CHAINSAW.CATProduct](#) document is already open

While, performing the chainsaw handle dismounting operation, you realized other objects needed to be dismounted such as the:

- CHAINSAW\_BODY\_HANDLE.1 object which makes the dismounting impossible (you already hide it)
- CHAINSAW\_BODY\_CLUTCHCOVER.1
- CHAINSAW\_BODY\_SAFETYHANDLE.1




Still in the sequence command:

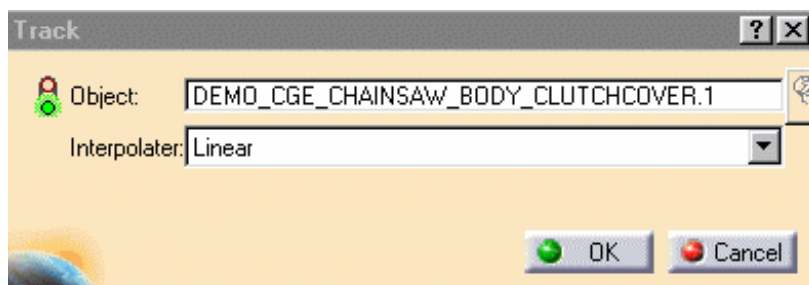


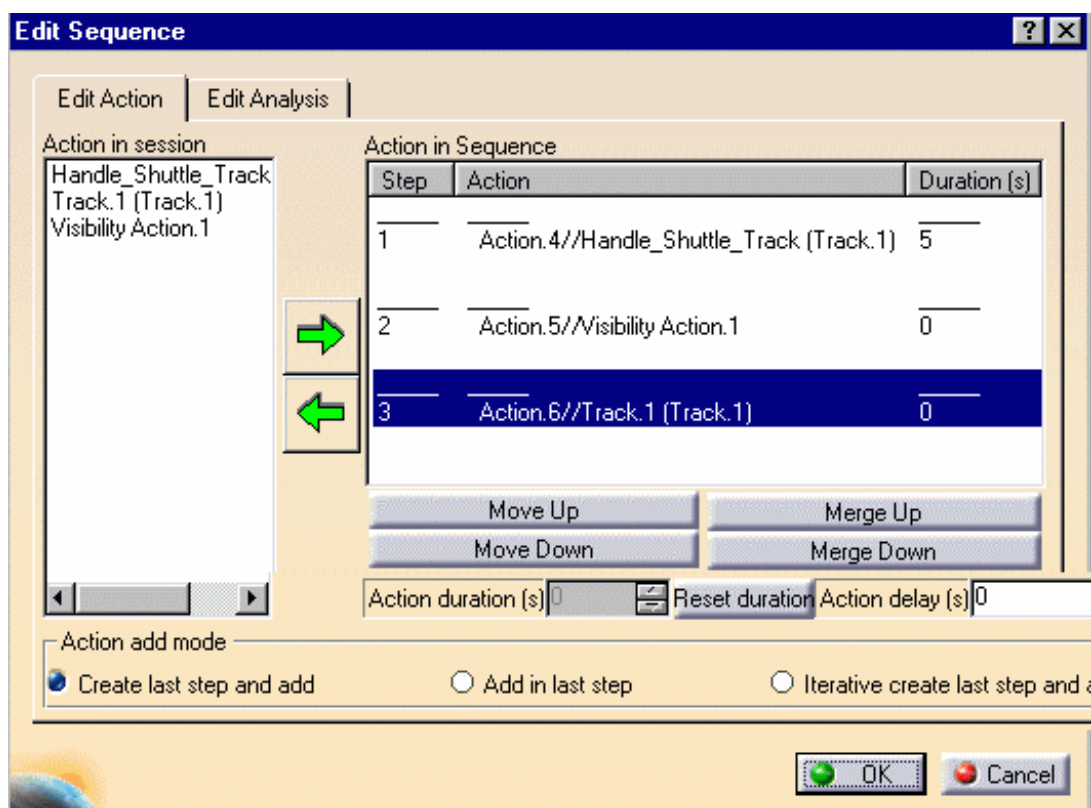
Create a track on CHAINSAW\_BODY\_CLUTCHCOVER.1 in context


1.

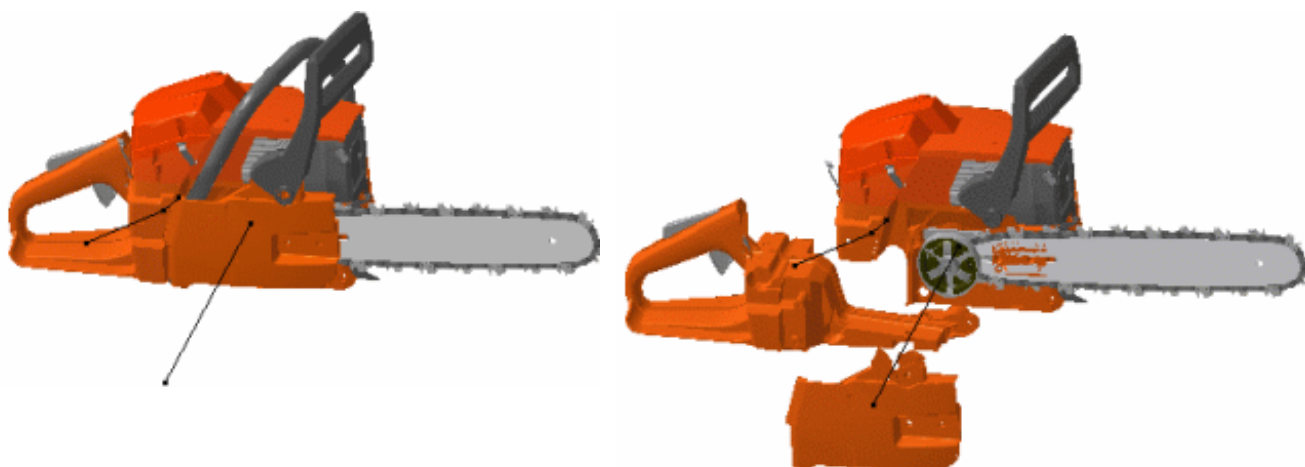
Click the track icon  from the DMU Simulation toolbar and select CHAINSAW\_BODY\_CLUTCHCOVER.1 in the specification tree.

The track dialog box is automatically updated and the new track is automatically created and added in the current sequence last step:





2. Move the 3D compass and double-click Record, the automatic insert mode is activated.
3. Click Ok in the track dialog box to confirm the track creation
4. Play your sequence using the play forward button  from the DMU player

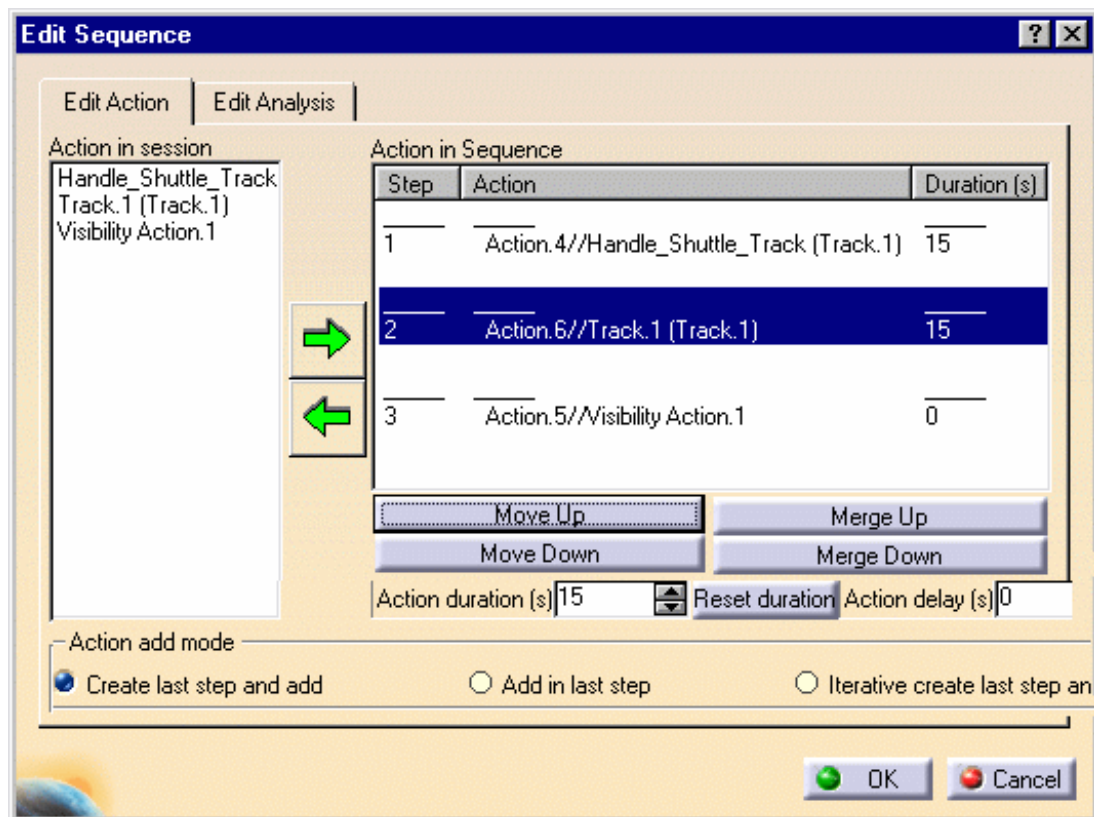



*Result: You are not satisfied with the sequence actions duration (it is too quick) nor with the scheduling*

5. In the Edit Sequence dialog box still opened, select Handle\_Shuttle\_Track
  - in the action duration field, enter 15 .

6. Select Track.1
  - in the action duration field, enter 15
  - click **Move up**

*This what you should have*




7. Play your sequence again using the play forward button  from the DMU player

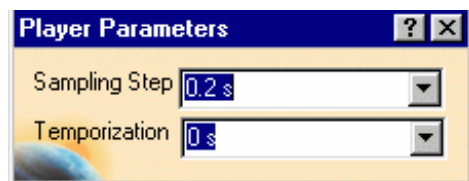



*Result: you are still not satisfied with the actions duration, you want your sequence simulation to run slower.*

*You need to customize*

8. Click the Parameters icon 

The Player Parameters dialog box is displayed:
9. Enter 0.2 s in the Time Step field




10. Play your sequence again using the play forward button  from the DMU player

*The sequence is valid but you are still not satisfied with the result*


- We need to dismount the CHAINSAW\_BODY\_SAFETYHANDLE.1 and therefore to create a Safety\_Handle\_Track.

11.


Click the track icon  from the DMU Simulation toolbar and select CHAINSAW\_BODY\_SAFETYHANDLE.1 in the specification tree .


12.

Swap to Detach mode clicking the Attach/Detach icon  from the Manipulation toolbar

The 3D compass color changes: you are now in the **Detach** mode .  
The icon changes in the Manipulation toolbar

13.

Click the Target icon  from the Manipulation toolbar or drag and drop the icon onto the cylinder.

Click when you are satisfied and swap to Attach mode 

Zoom out if necessary

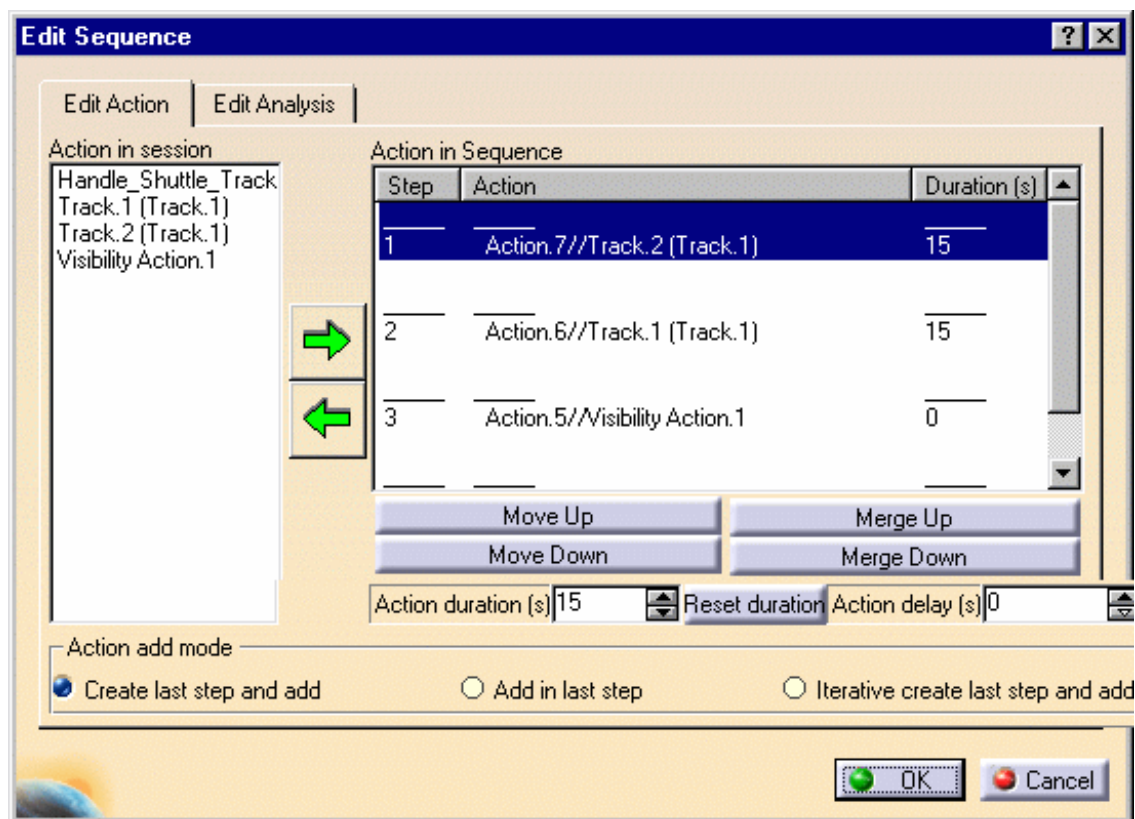


15. Perform a rotation with the 3D compass and click Record ,

16. Click Ok in the Track dialog box to end the the SAFETYHANDLE.1 track creation


17. In the Edit Sequence dialog box, select Track.2

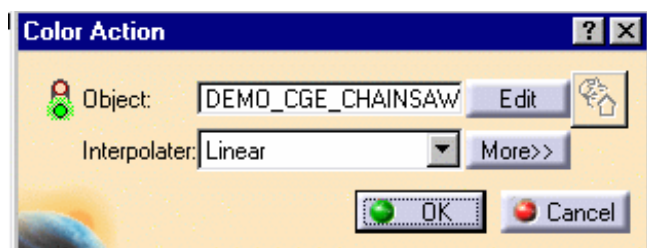
- in the action duration field, enter 15.
- click **Move up** until the action 7 (SAFETYHANDLE.1 track) is in step 1



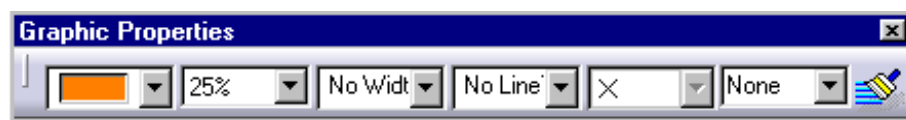


*You want the CHAINSAW\_BODY\_CLUTCHCOVER.1 to become transparent during sequence simulation and before dismounting*

18.  Click the Color action icon in the DMU Simulation toolbar
19. Select CHAINSAW\_BODY\_CLUTCHCOVER.1 either in the specification tree or in the geometry area

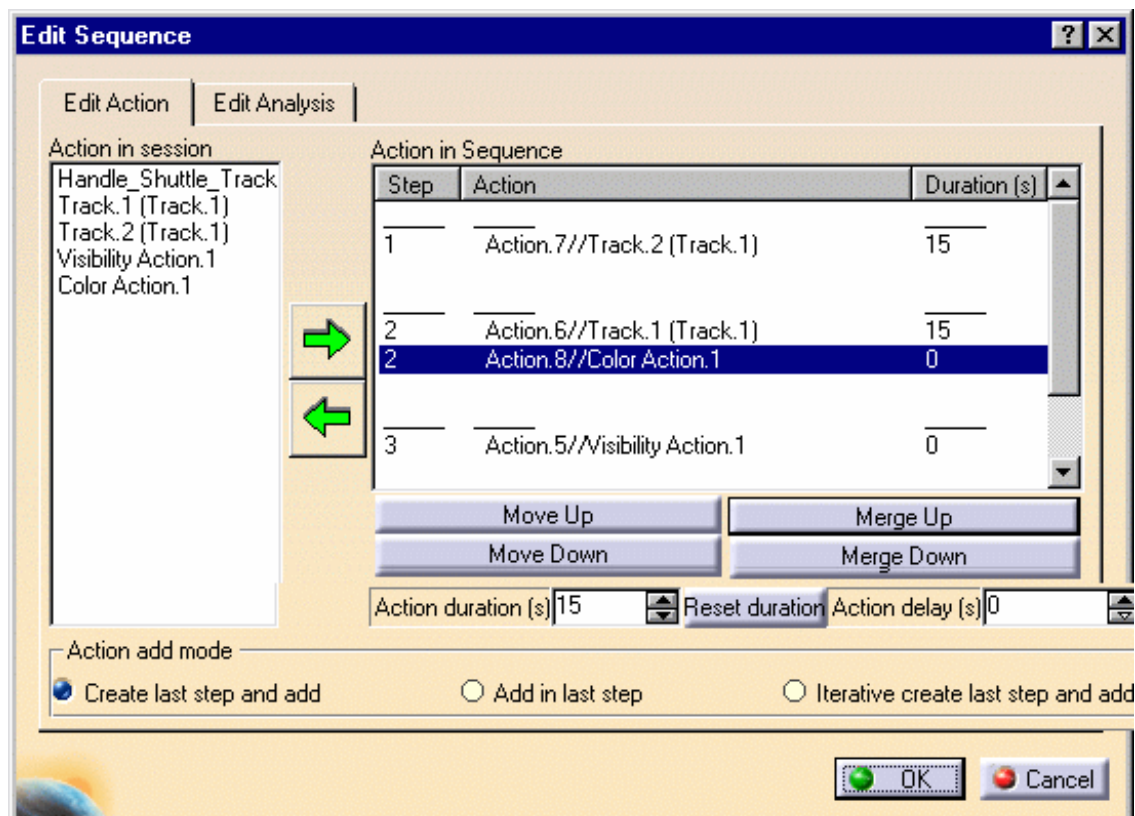


20. In the Graphic Properties toolbar displayed, select a color using the arrow and set the transparency too
  - color: orange
  - transparency: 25%



21. Click Ok in the Color action dialog box
22. In the Edit Sequence dialog box, click **Move up** twice and **Merge up** to schedule this color action in step 2

You make sure, the transparency action will be triggered at the beginning of the CHAINSAW\_BODY\_CLUTCHCOVER.1 dismounting ( Action 7//Track.2).



**23.** Replay your sequence using the DMU Player.

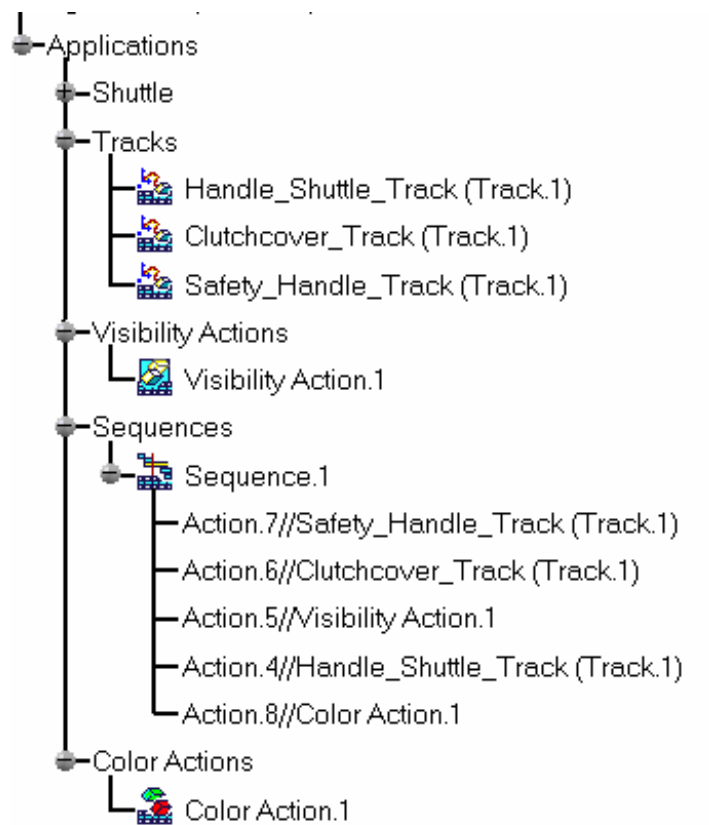
Actions are scheduled in a logical way:

- CHAINSAW\_BODY\_SAFETYHANDLE.1 track (step 1)
- CHAINSAW\_BODY\_CLUTCHCOVER.1 track (step 2)
- Transparency action (step2)
- Visibility action (hiding the CHAINSAW\_BODY\_HANDLE.1) (Step 3)
- Handle\_Shuttle\_Track (Step 4)

**24.** Click Ok in the Edit Sequence dialog box.

Right-click the tracks and select Properties from the contextual menu to rename them with a meaningful name.

This is what you can obtain for instance:



*Now you are ready to generate an AVI document*





# Generating a Film



You defined a sequence containing three tracks and other actions (visibility and color actions),

- CHAINSAW\_BODY\_SAFETYHANDLE.1 track (step 1)
- CHAINSAW\_BODY\_CLUTCHCOVER.1 track (step 2)
- Transparency action (step2)
- Visibility action (hiding the CHAINSAW\_BODY\_HANDLE.1) (Step 3)
- Handle\_Shuttle\_Track (Step 4)

Now you want to generate a film of your dismounting operation:

For this you will need to perform various operations such as:

- Creating a camera (using the **View ->Named Views**) with 3 different viewpoints focusing on CHAINSAW\_BODY\_SAFETYHANDLE.1, on CHAINSAW\_BODY\_CLUTCHCOVER.1 and on the Chainsaw\_Handle Shuttle
- Playing the created camera
- Defining a camera track
- Defining a new sequence
- Customizing the player parameters
- Editing the camera track
- Defining a sequence and replay it (in a camera window **Window** )
- Multi-selecting and Hiding the current tracks
- Generating a Film (in AVI format) using **Tools->Image->Video**

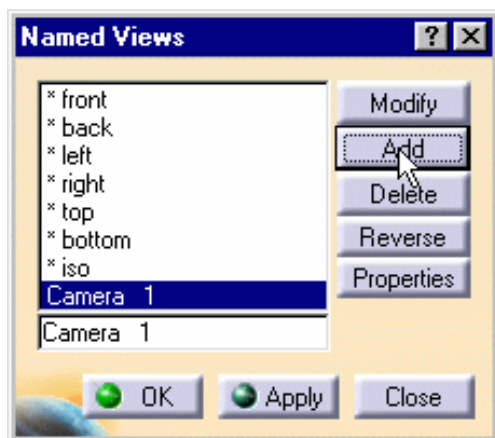


The [CHAINSAW.CATProduct](#) document is already open



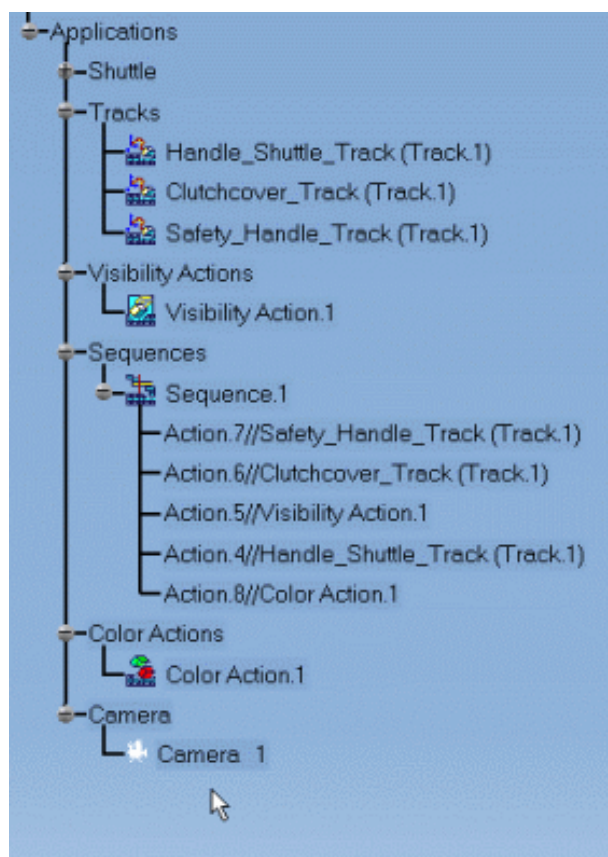
1. Create a Camera for this:

Select **View ->Named Views...**




2. Click Ok.

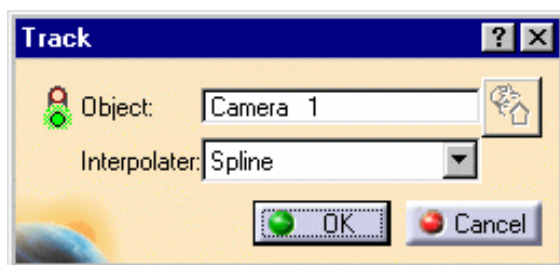
The camera is created and identified in the specification tree



*Create a Camera track*

3.

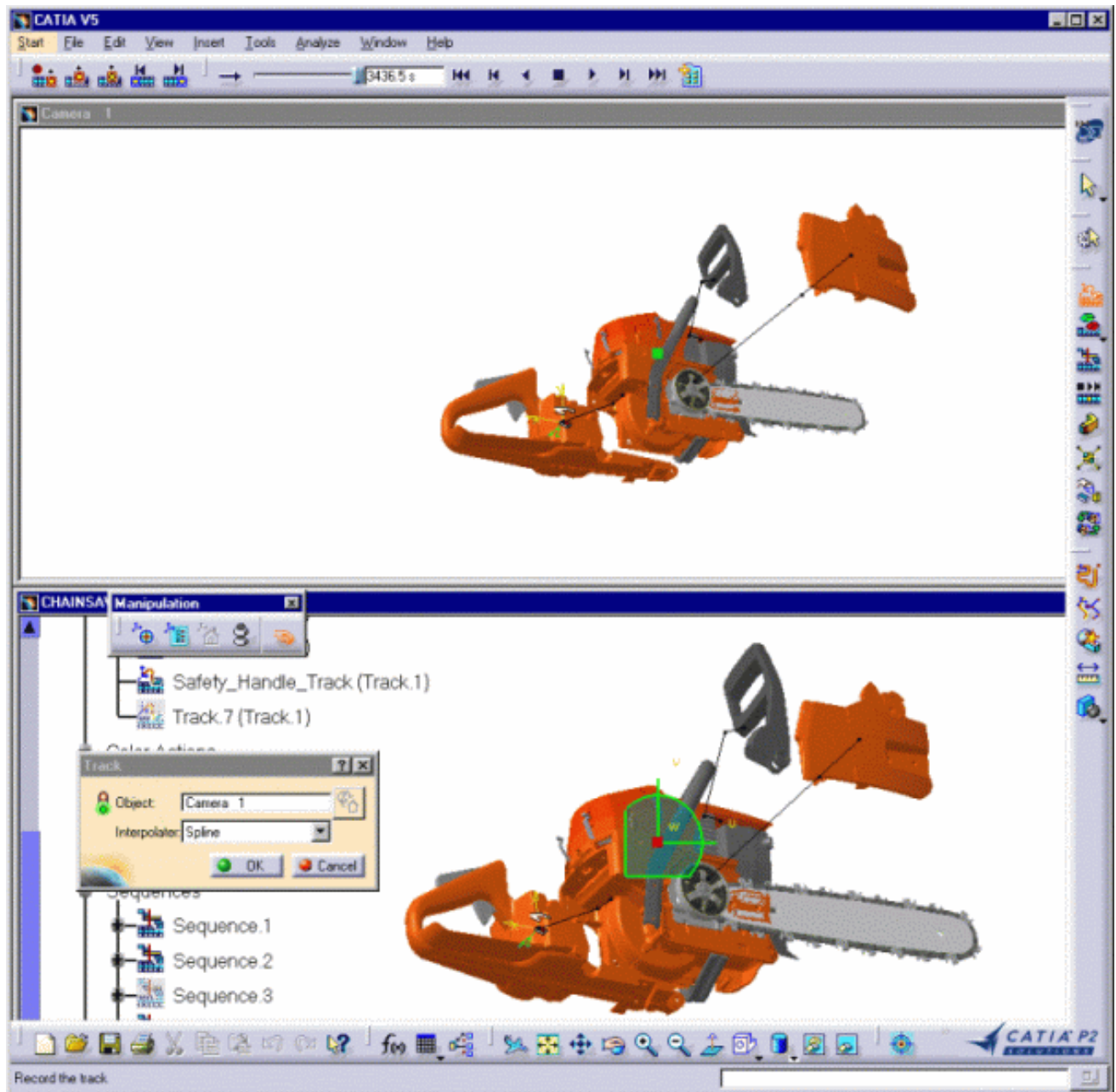
Click the track icon  from the DMU Simulation toolbar and select camera.1 in the specification tree .



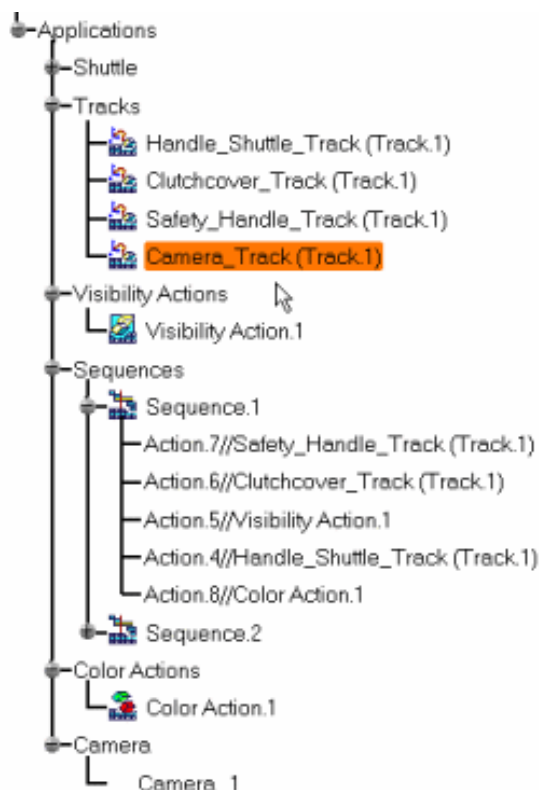
Record 3 different viewpoints focusing on CHAINSAW\_BODY\_SAFETYHANDLE.1 object , on CHAINSAW\_BODY\_CLUTCHCOVER.1object and on the Chainsaw\_Handle Shuttle

4.

Open a camera window (**Window->Camera Window->Camera 1**) and tile your window horizontally **Window->Tile Horizontally**)




5. Click Ok in the track dialog box
6. Rename the created track in Camera\_Track using the track contextual menu




*You need this Camera\_track to be played in simultaneous mode with the first sequence defined*

*Define a new sequence*

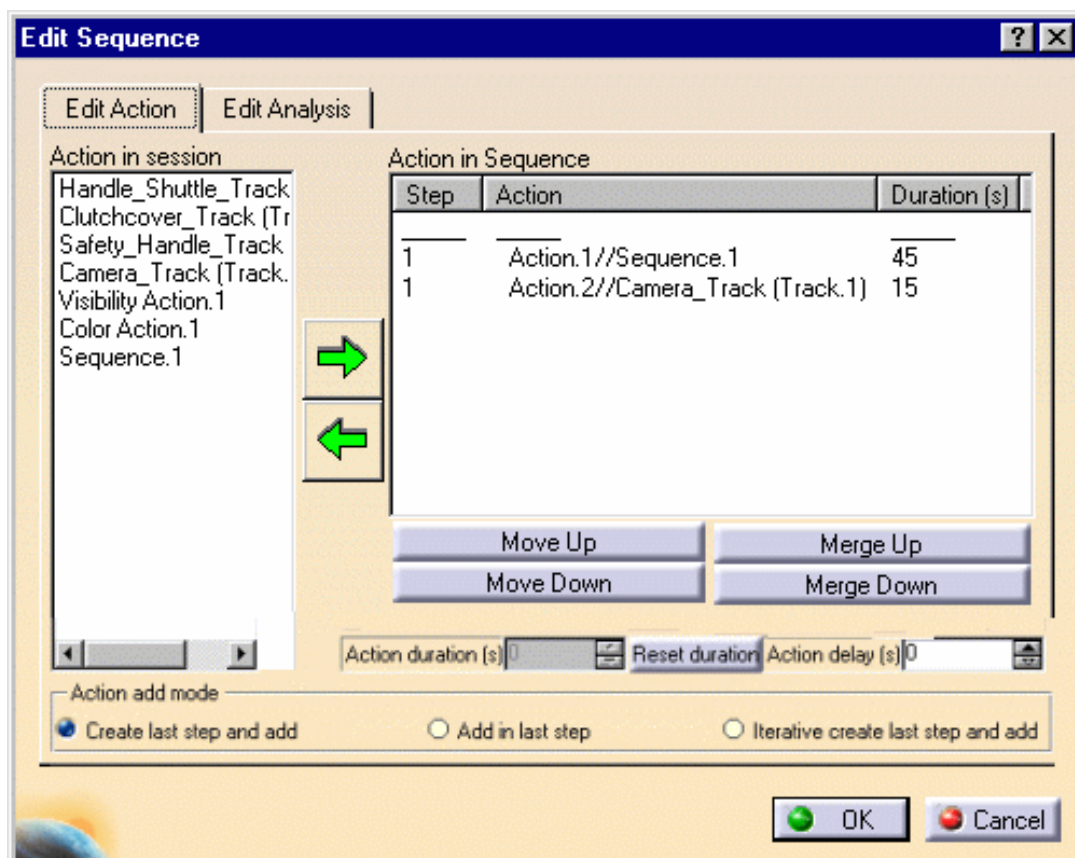
7.  Click the Edit Sequence icon

The Edit Sequence dialog box is displayed

8.  Select Sequence.1 in the "Action in session" list and click . The action is added in the sequence list.

9. Select Action.2//Camera\_Track
- in the action duration field, enter 15 with user defined option set.

10. Click **Merge Up** as you want the actions to be played in simultaneous mode



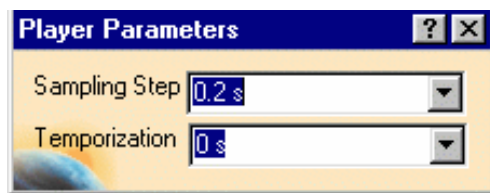
11. Play your sequence using the Player.


*You need to customize the player parameters*

12. Click the Parameters icon 

The Player Parameters dialog box is displayed:

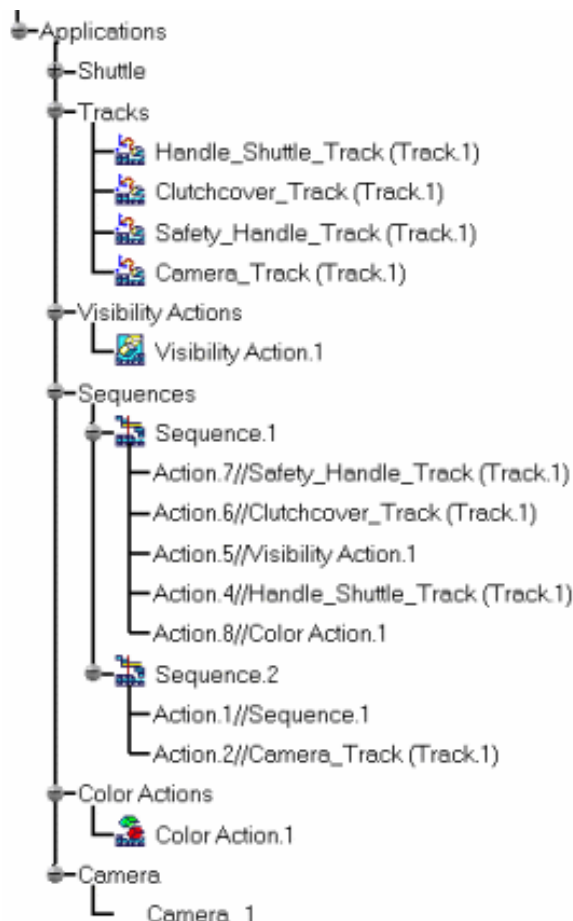
13. Enter 0.2 s in the Time Step field



14. Play your sequence again using the play forward button  from the DMU player


15. Click Ok to create your sequence:

The new sequence is created and identified in the specification tree



*Hide the tracks*


**16.**

Multi-select the tracks in the specification tree and click the Hide/Show  icon, or select the **View->Hide/Show->Swap Hide/Show** command.

The track objects are no longer displayed: they have been transferred into the No Show space

*You are now ready to generate a film in AVI format*

**17.** Select **Tools->Image->Video** to display the Video Recorder dialog box

- Set Up the Capture Session
- Click the Recording button  to access the Video Properties dialog box which lets you set up the video capture parameters prior to recording the video
- Choose the Video File Format and Location
- in our example select Microsoft AVI (Windows)
- Specify which part of the screen to record
- Setting Movie Replay Parameters
- Click OK in the Video Properties dialog box to start the recording.



Please read Recording Interactions in Video Format in the *Infrastructure User's Guide*

**18.** Select Sequence.2 in the specification tree

- 19.** Run your sequence simulation with the DMU Player
- 20.** Click Stop



# Converting a Simulation into a Sequence



This task shows how to convert an existing simulation into a sequence of tracks.

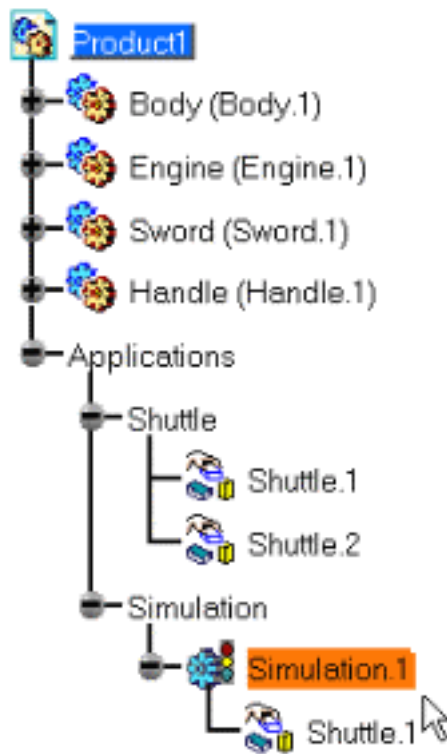
In our example, there is one existing simulation to be converted. The result expected is a sequence one track.



Open the [CONVERT\\_SIMULATION.CATProduct](#) document.

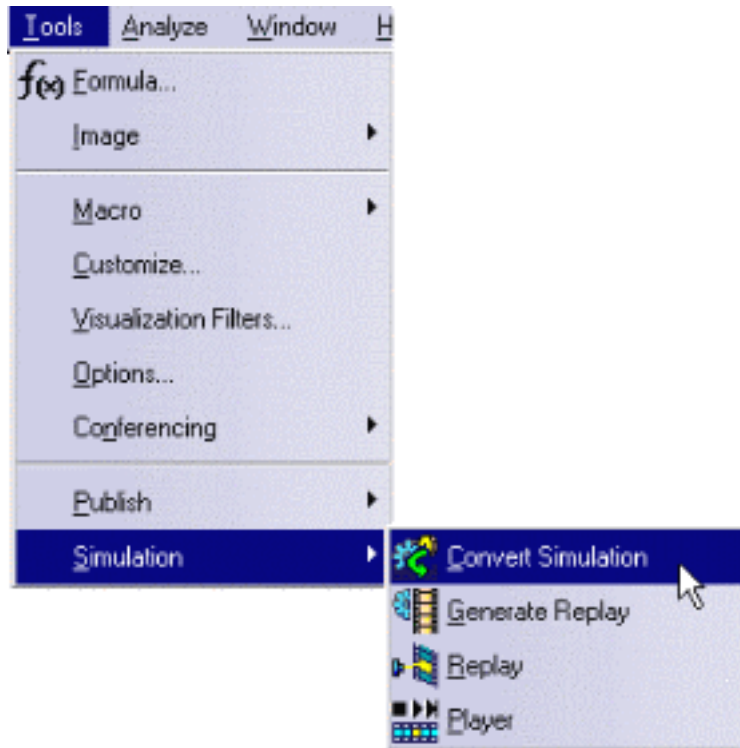


1. Select **Simulation.1** either in the specification tree or in the geometry area



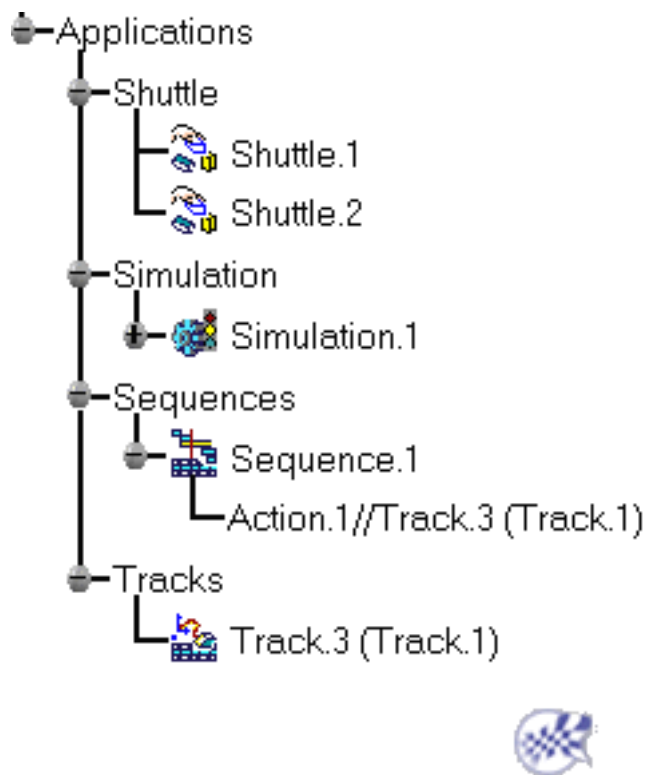
2. Select **Tools->Simulation->Convert Simulation...**





The Convert Simulation progression bar appears

The simulation is automatically converted into a sequence, a shuttle track object is created.



# Path Finder



**Use Path Finder in Interactive Mode:** select a simulation object, then click the Path Finder icon. Select the required settings in the dialog box displayed. Click **Apply**. When done, click **Ok** to obtain a final result.

**Write a Path Finder Macro:** select the **Tools->Macro->Start Recording** command then click Start to begin recording the automatic path finding macro. When this is complete, click Stop in the Stop Recording dialog box, or select the **Tools->Macro->Stop Recording** command. Your macro is now ready for replay.

**Publish Path Finder Reports:** select **Tools -> Publish -> Start Publish** from the menu bar. Identify the path where you want to save the report as well as the report name then click **Save** in the Select Publish File dialog box. use the Path Finder command as required, when done select **Tools -> Publish -> Stop Publish**.

**More About Path Finder:** provides examples and explanations about path finder.

**About Path Finder and Smooth Operations:** provides additional information about how to use path finder to smooth operations.

# Using Path Finder in Interactive Mode

Two pathfinder modes are available:

- Beginner's mode
- [Advanced mode](#) for experienced users.

Before using the automatic path finder, you should have recorded a simulation. For more detailed information, please refer to [Recording a Simulation](#).

## Beginner's Path Finder Mode



This task shows how to use Automatic Path Finder in beginner's mode

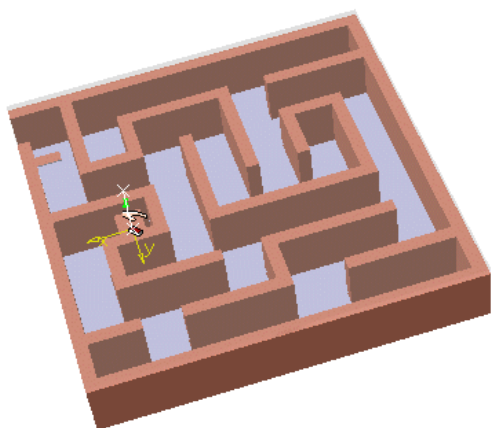
**Note:** Beginner's automatic path finder mode is the default mode.



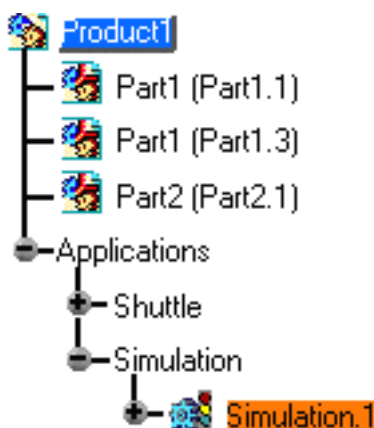
Automatic path finder is a new functionality that lets you easily find the best path to dismount assemblies. You need a simulation to use the Automatic Path Finder.



Open the [LABYRINTH.CATProduct](#) document.



1. Select Simulation.1 in the specification tree





Repositioning the shuttle axis according to a potential extraction direction eases the path finding process.

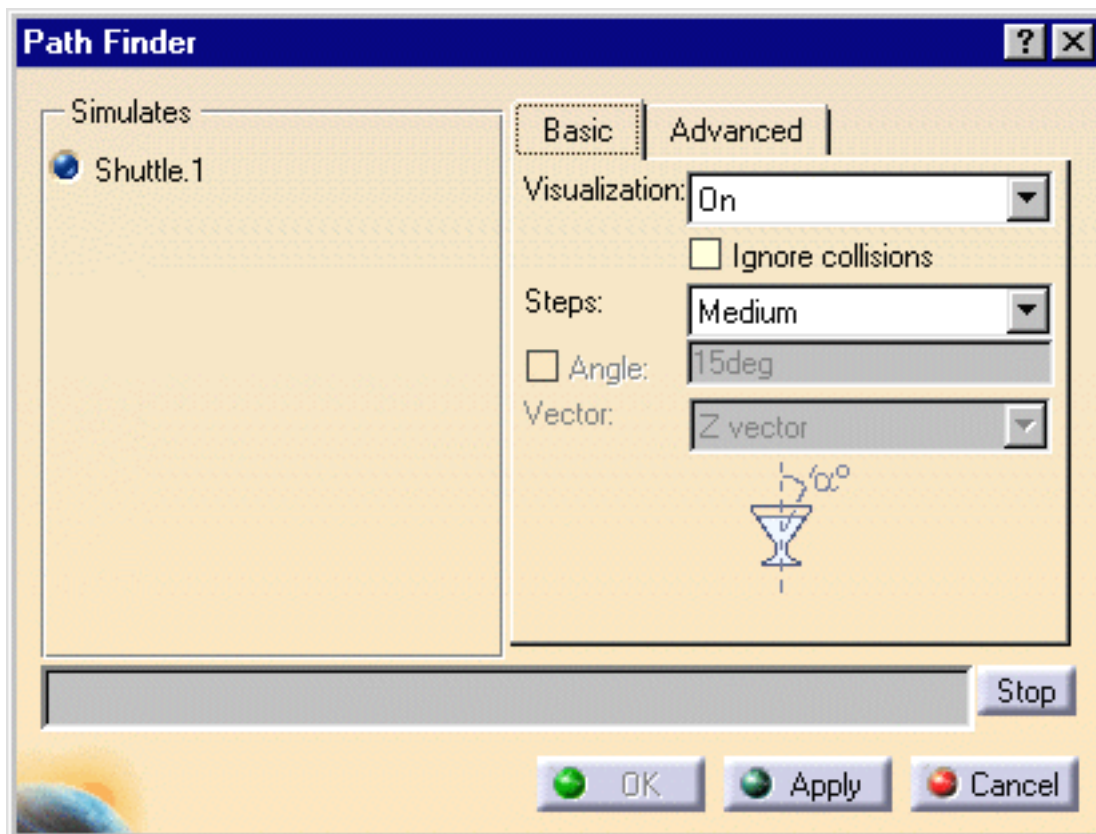
2.



Click the Path finder icon from the DMU Check Toolbar  
The Path Finder dialog box is displayed:



You can also select the Path Finder command first and then select the required simulation afterwards



You can choose to ignore existing collisions



This is very helpful when the various assembly components are already in collision and when you do not want to take them into account when performing path finder operation. The path finder dialog box appears and lets you select options for: (The Basic mode is set by default)

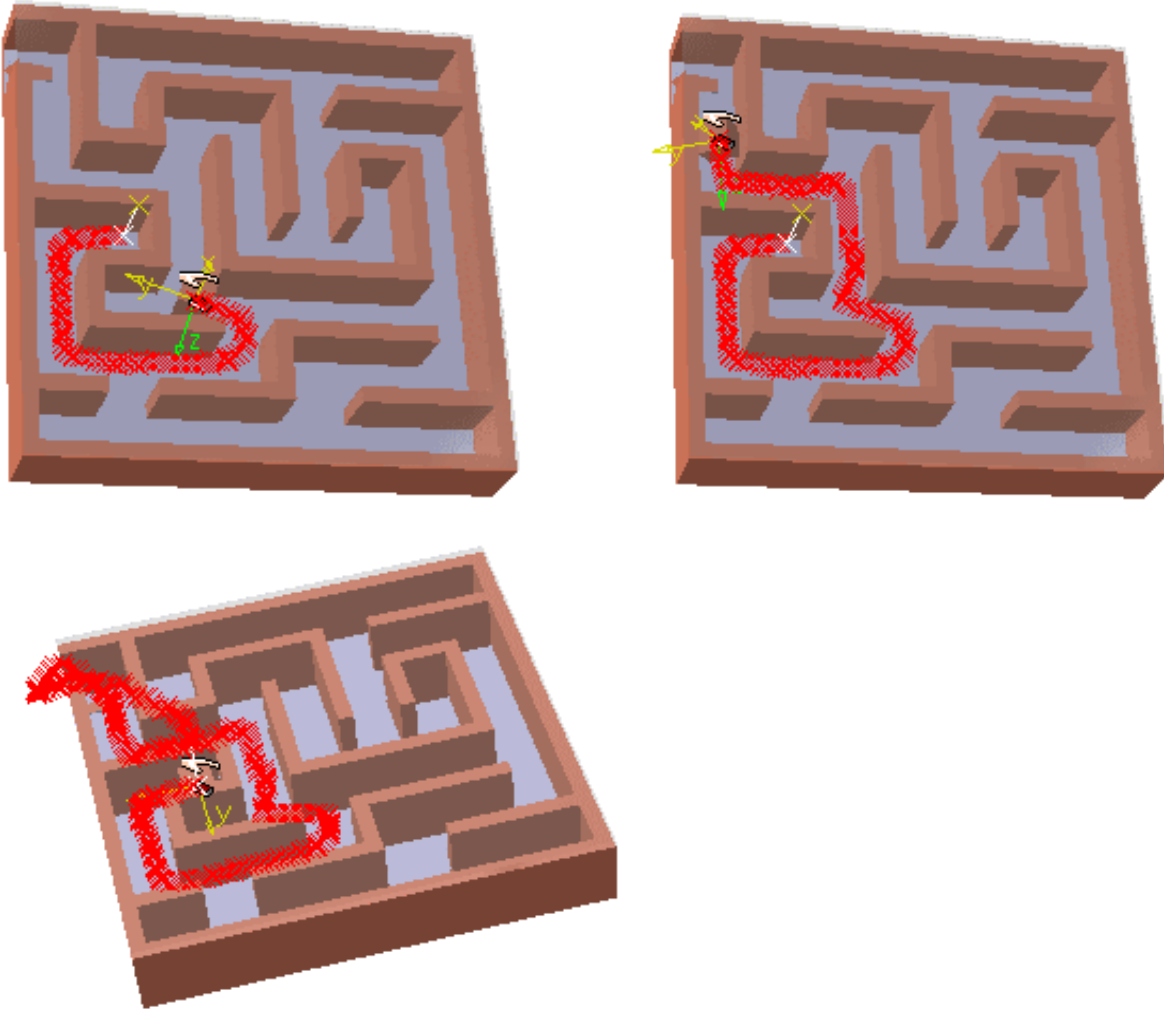
#### Visualization

- On (shows the object move and its various positions during calculation. You can stop the process at any time)
- Off (You are not allowed to stop the process and you do not see the progression)
- Strombo (shows positions based on a specific parameter (every 20 steps))

#### Steps

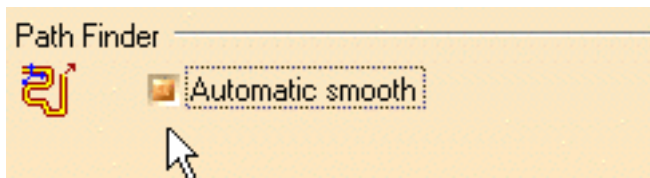
- Small
- Medium
- Large

3. Keep the default settings:  
Visualization : on  
Steps : Medium
4. Click **Apply** to confirm your operation.



5. Click **Ok**.

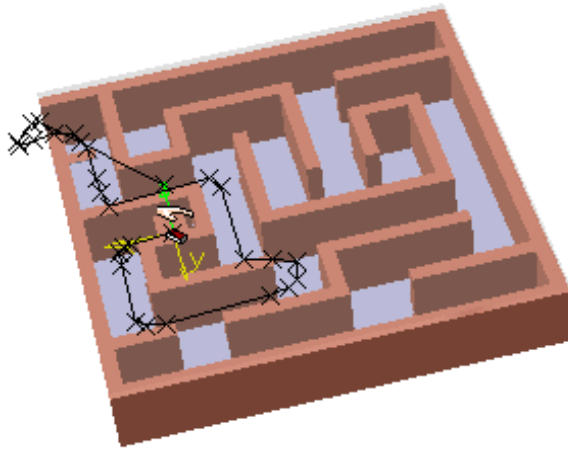
If you need the OK to perform the smooth, please select **Tools->Options->Digital Mockup->DMU fitting**. Check the Automatic Smooth option



In this case the following step is useless.

- 6.

Click the Smooth icon  to smooth the path result.

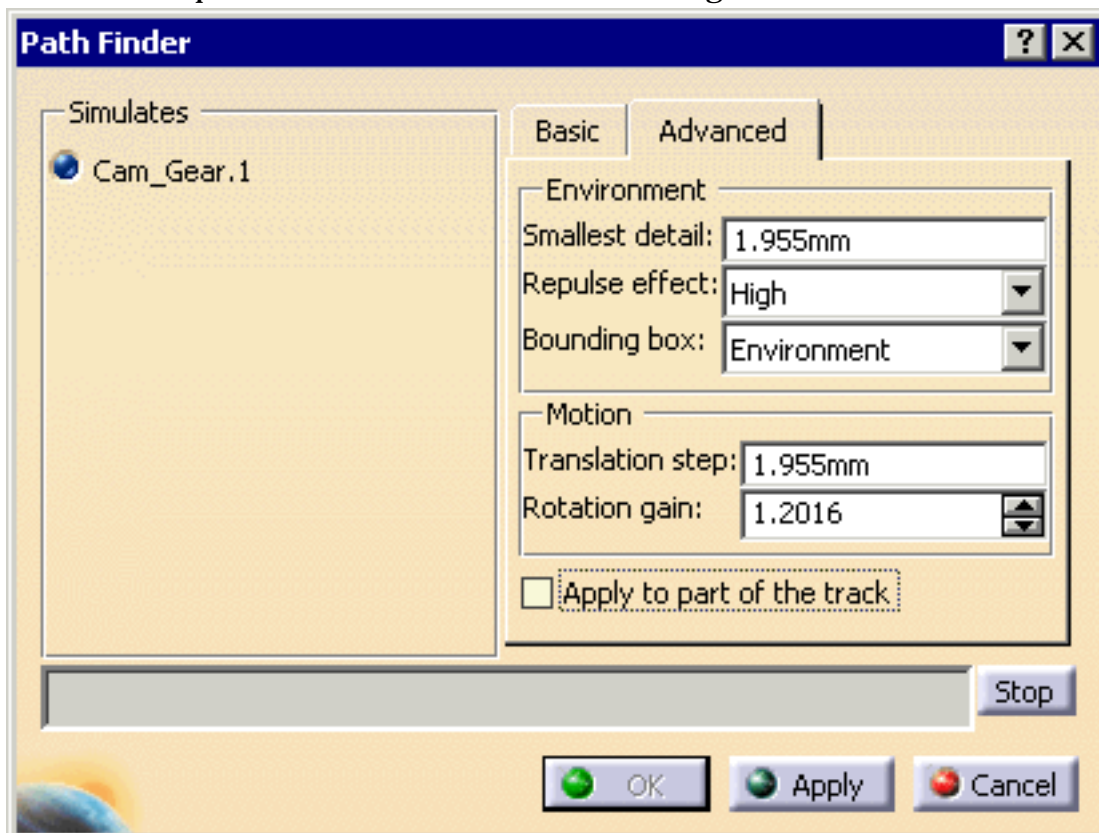


## Advanced Path Finder Mode



Now, repeat the scenario above but select the Advanced tab in the Path finder dialog box.

1. Select the required values in the Path finder dialog box:



The path finder dialog box lets you enter precise values for:

## ENVIRONMENT

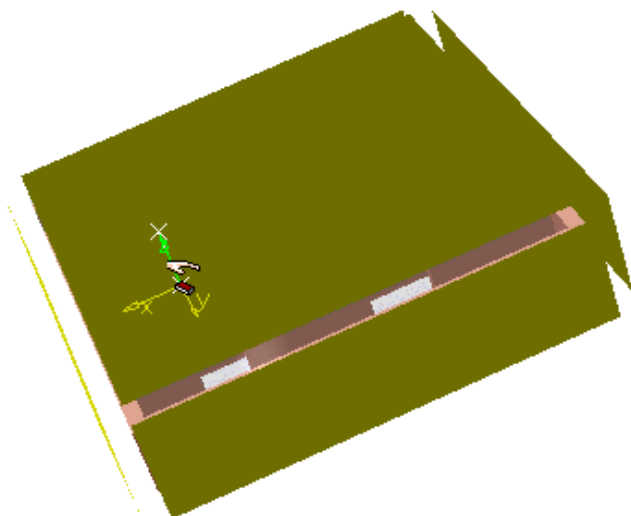
- **Smallest Detail** (accuracy used to calculate the environment description)  
If you enter a small value as smallest detail, the environment description will be very accurate but very costly in terms of memory size.  
If you enter a big value, the description will be coarser (ex. if the diameter of a hole is smaller than the value entered, the hole no longer exists as it is filled with matter). In this case, you gain in memory size.
- **Repulse effect** (low, medium, high). When high, eases the rotation motion and smoothes the path

## MOTION

- **Translation step** (unit used to define the step value in translation mode)
- **Rotation Gain** (linked to the translation step parameter).
- If rotation gain value = 2, the rotation amplitude = 2 x translation step value
- If rotation gain value = 0.5, the rotation amplitude = 0.5 x translation step value
- **Resize bounding box:** (check the Resize bounding box option if needed)

### Apply to part of the track

- Checking this box means that you will be prompted to select a beginning and end segment of the track that will define the area of the track to which you want to apply path finder.
- Leaving the box unchecked (the default) means that pathfinder will apply to the entire path.



2. Click **Apply**. When done, click **Ok**.

3. Now, you are ready to replay your simulation.

Be careful not to choose values which are not compatible with your model.



For instance, here if you enter 30mm as Translation step value, the moving object will jump over the glass material because the path is already considered out of collision, which is not true. The following information message is displayed:





# Writing a Path Finder Macro



A macro is a series of functions, written in a scripting language, that you group in a single command to perform the requested task automatically. You can for example use a macro to automate the generation of an automatic path finding.



This task shows how to use the automatic path finding functionality in batch mode, which means you are going to record an interaction sequence in a macro.



Open the [mastervac.CATProduct](#) document.

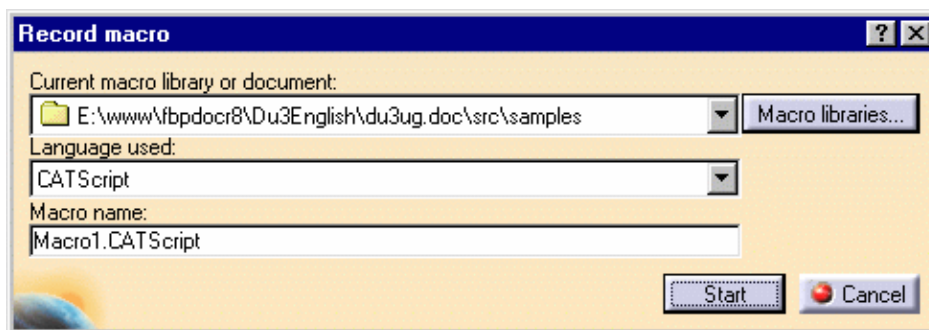



1. Select the **Tools->Macro->Start Recording** command to display the Macro Recording dialog box.
2. In the Macro Record dialog box, specify the current macro library or document.

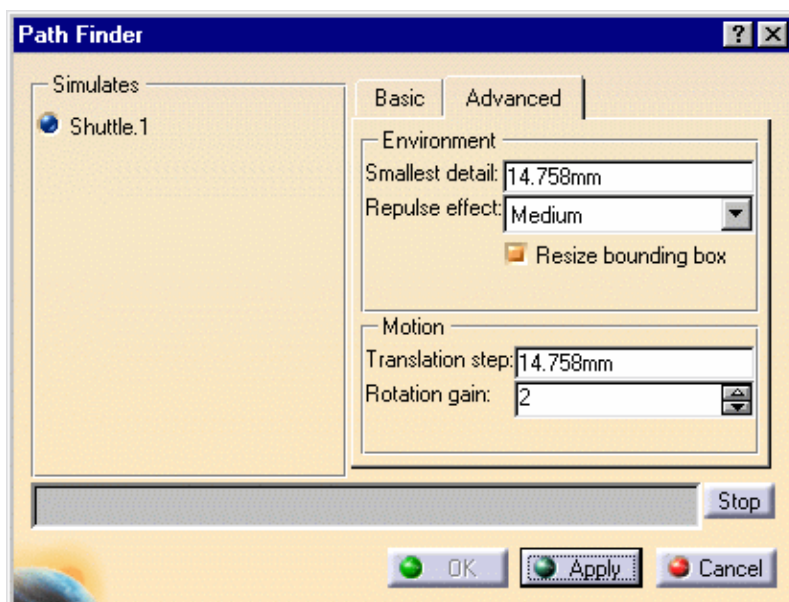
This means deciding where the macro is going to be stored:

- in a **macro library**: the macro will be stored in a directory (or a VBA project if it exists)
- in a **document**: the macro will be stored in the current document.

3. Give a name to the macro: MacroPF.CATScript for example



4. Click Start to begin recording the macro.
5. In the specification tree, select **Simulation.1**.
6. Click the Path Finder icon  from the Simulation toolbar.
7. Set the options in the Path finder dialog box displayed:



8. Click **Apply** to launch the operation.

9. When done, click **OK** to smooth the path result.
10. When this is complete, click Stop in the Stop Recording dialog box, or select the **Tools->Macro->Stop Recording** command. Your macro is now ready for replay.

The macro looks like this:

MACRO	EXPLANATIONS
<pre>Language="VBSCRIPT"  Sub CATMain()  Dim FittingWorkbench0 As Workbench  Set FittingWorkbench0 = CATIA.ActiveDocument.GetWorkbench ( "FittingWorkbench" )  Dim PathFinder1 As PathFinder  Set PathFinder1 = FittingWorkbench0.PathFinders.Add ( "PathFinder" )</pre>	
PathFinder1.set_SimulationByName "Simulation.1"	Name of the simulation selected.
PathFinder1.set_DoubleParameter "Accuracy", 14.758000	Corresponds to the accuracy value chosen in the Path Finder dialog box.  Smallest detail: 14.758mm
PathFinder1.set_DoubleParameter "TranslationStep", 14.758000	Corresponds to the translation step value chosen in the Path Finder dialog box.  Translation step: 14.758mm
PathFinder1.set_DoubleParameter "RotationGain", 2.000000	Corresponds to the rotation gain value chosen in the Path Finder dialog box.  Rotation gain: 2
PathFinder1.set_IntegerParameter "Repulse", 5790	
PathFinder1.set_BoxParameter "Box", -205.110550, - <b>1115.599648, -341.661041, 918.198769,</b> <b>460.187050, 694.402138</b>  PathFinder1.Compute	You resized the bounding box : the new values are displayed.  Resize bounding box
PathFinder1.FillSimulation	Insert the above result into the simulation
FittingWorkbench0.PathFinders.RemovePathFinder PathFinder1  End Sub	



Open the [MacroPF.CATScript](#) document.

You can edit your macro if necessary. For more detailed information, please refer to Recording, Running, and Editing Macros in the *Infrastructure User's Guide*.



PARAMETER NAME	DEFAULT VALUE	METHOD	MACRO ?	EXAMPLE
MaxTime	60 min		can be added to the macro	
Accuracy	—	PathFinder1.set_DoubleParameter "Accuracy",value	can be edited	PathFinder1.set_DoubleParameter "Accuracy", 14.758000
TranslationStep	—	PathFinder1.set_DoubleParameter "TranslationStep",value	can be edited	PathFinder1.set_DoubleParameter "TranslationStep", 14.758000
RotationGain	1.0		can be edited	PathFinder1.set_DoubleParameter "RotationGain", 2.000000
Repulse	High	PathFinder1.set_IntegerParameter "Repulse",value	can be edited	PathFinder1.set_IntegerParameter "Repulse", 5790
Box	—	PathFinder1.set_BoxParameter "Box",values	can be edited	PathFinder1.set_BoxParameter "Box", -205.110550, -1115.599648, -341.661041, 918.198769, 460.187050, 694.402138
VisuMode	do not appear in the macro.			
Steps				

# Publishing Path Finder Results

A number of tools let you prepare html reports that you can publish on the enterprise intranet or the Internet for collaborative communication. You can for example publish a path finder report during your review, add appropriate comments or annotations and even select parts to insert as a VRML link.



This task explains how to publish a path finder report.



Open the [LABYRINTH.CATProduct](#) document.



1. Select **Tools** -> **Publish** -> **Start Publish** from the menu bar.

The Select Publish File dialog box appears.

2. Identify the path where you want to save the report as well as the report name then click **Save**.

**Note:** Reports are published in html format. The date created and the person who created the report are indicated.




The Publishing Tools toolbar appears.

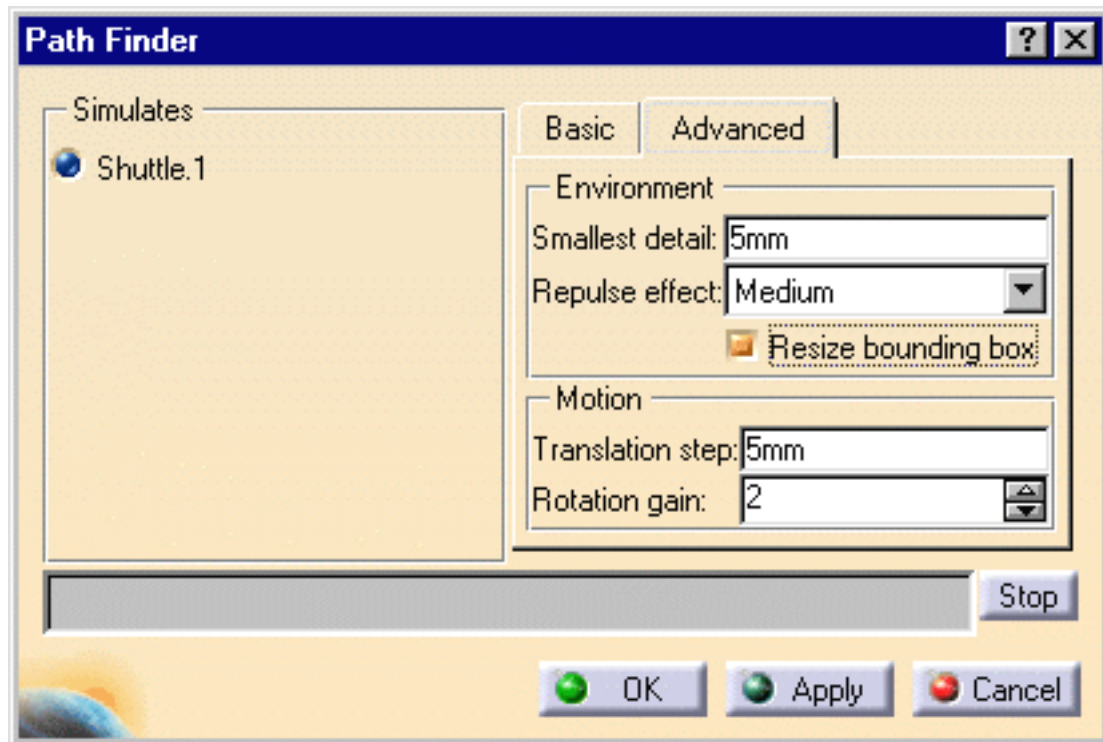
You can now publish a simulation object using the P icon  from the Publishing Tools toolbar.

For more detailed information, please refer to *DMU Navigator User's Guide- Publishing*



Click the Path finder icon  from the Simulation Toolbar  
The Path Finder dialog box is displayed.

3. Select the Advanced tab in the Path finder dialog box.
4. Select the required values in the Path finder dialog box as shown below:



5. Click **Apply**. When done, click **Ok**.
6. Select **Tools -> Publish -> Stop Publish**.
9. Read the published report.



Clicking the images in the html page enlarges them.



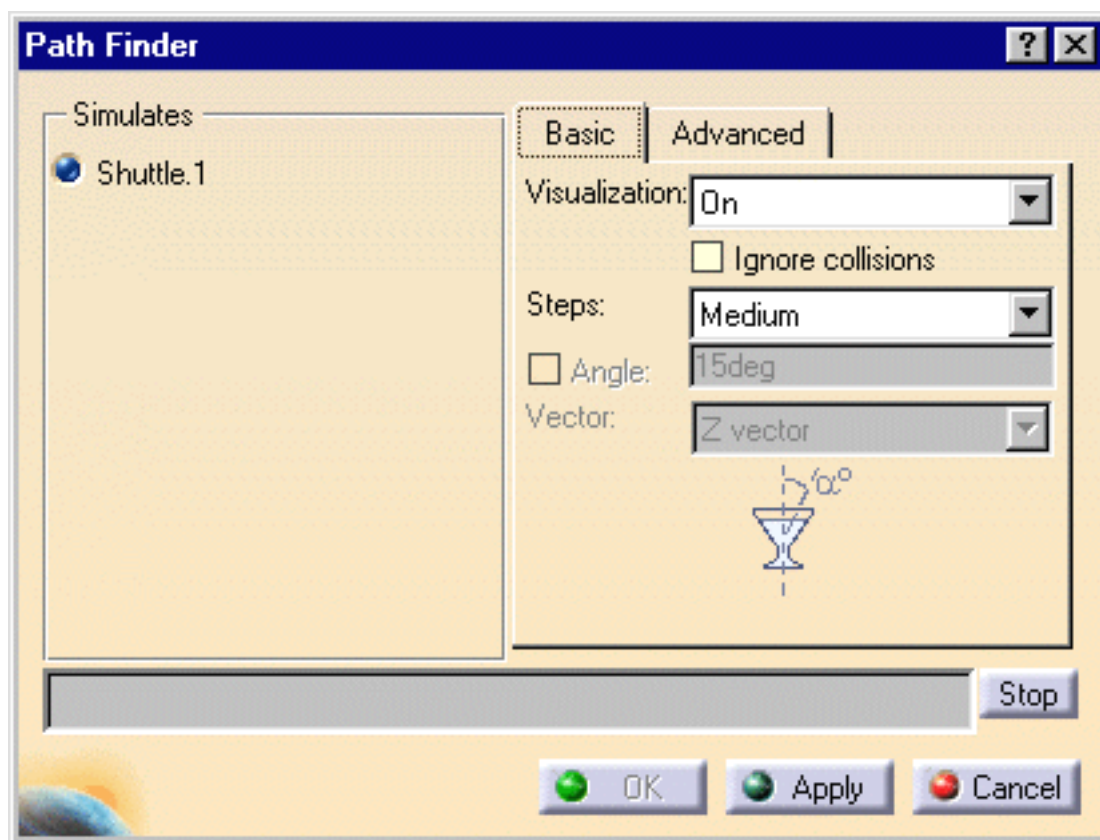
# More About Path Finder

## This Task:

- explains both the behavior and parameters existing in Path Finder
- gives some methodologies and ideas to use Path Finder
- presents remarks about Path Finder

## How to use Path Finder ?

Path Finder can mostly be used to find a first path, to refine a roughly defined path (manually) or when time to display the assembly is too long for manual path definition or to automatically perform checks in batch mode.



The path finder dialog box appears and lets you select options for: (The Basic mode is set by default)

**Visualization:**

- **On:** Each displacement of the part to be moved is displayed
- **Off:** No displacement of the part to be moved is displayed
- **Strombo:** Only one displacement is displayed for twenty computed displacements.

**NOTE :** Performances are better with the **Off** option set than with **Strombo** option set and better with **Strombo** than with **On** option set.

Anyway, Strombo mode can be more user friendly than Off mode as some displacements are displayed



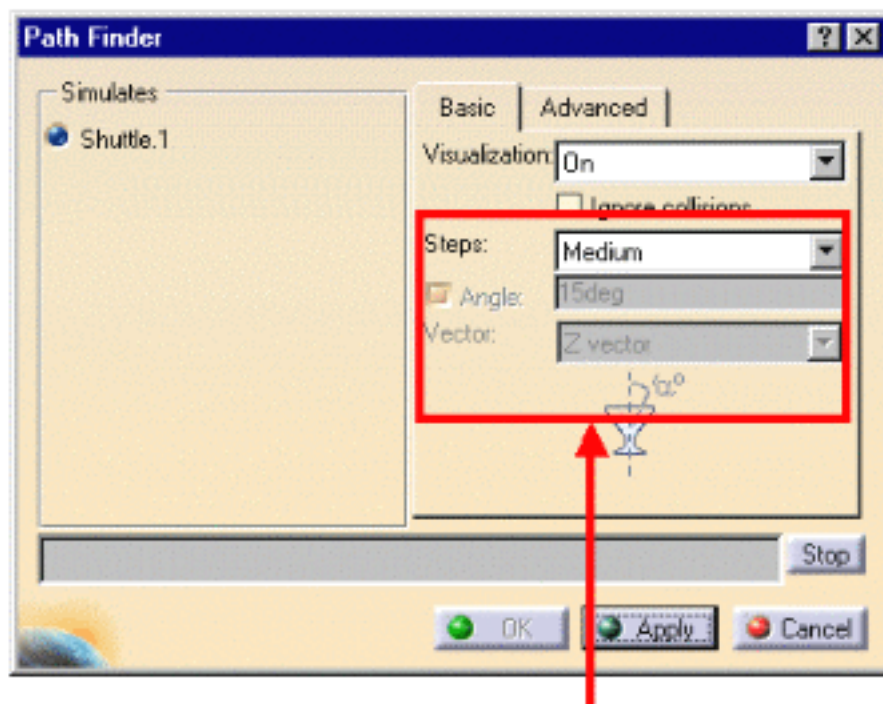
Be careful, if you work with the **Off** option set, you will not be able to **STOP** the computation using the stop button

**Steps:**

High, Medium and Low represent the parameters combination ; these parameters are those existing in the Advanced dialog box.

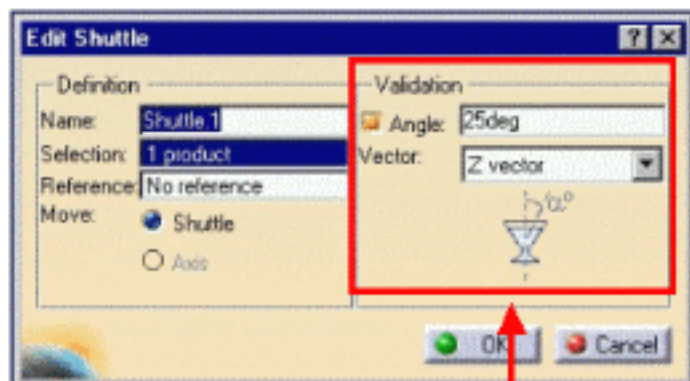
**Angle:**

If a maximal rotation angle has been defined while creating the shuttle, the maximal angle will be considered and taken into account in the path finder computation.



*This maximal rotation angle is considered to find a path*





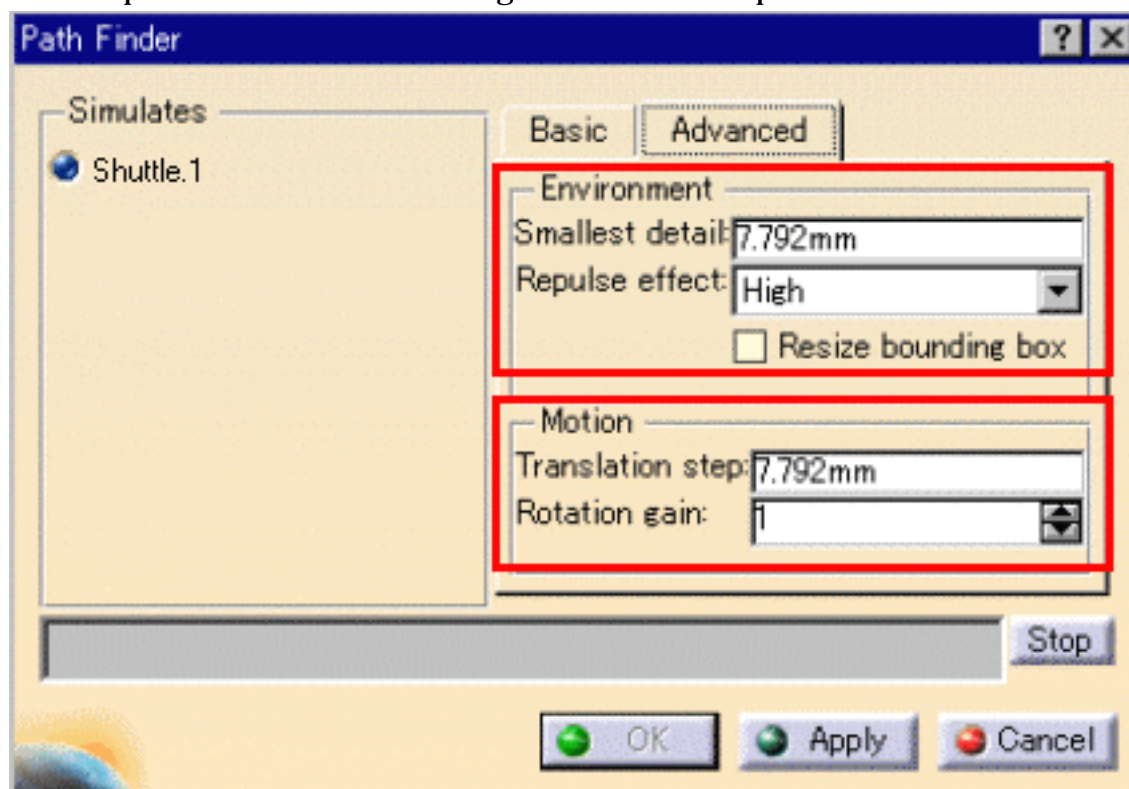
Maximal rotation angle is defined during shuttle creation or edition

### Environment and motion

The path finder dialog box comprises two frames. One for environment, one for motion. They are linked to the algorithm process used for the computation.

First, V5 browses the environment considering parts in session. This builds an internal 3D map describing the obstructions and possible paths. Settings for this first process are set in frame called Environment.

Then, this internal 3D map is used by the path finder algorithm as a global indication to find a way out. Several motions or displacements are successively tried along the path and clashes are computed for each trial. Settings for this second process are set in frame called Motion.





**Environment:**

Smallest detail is the size which will be considered to create a 3D map.

For example, if the smallest detail value equals 7mm:

- if a hole has a size of 6.9 mm, it will not be seen as a hole in the internal 3D map
- if a boss has a size of 1 mm, it will be seen as a boss of 7 mm in the internal 3D map

Smallest detail value is not the accuracy considered for clash computation.

For clash computation, this is the sag value which is taken into account. The clash computation accuracy is twice lower than the sag value.



**NOTE:** As the accuracy of the map is not linked to the clash accuracy, it could be more convenient for the algorithm to comprise a rough map within which it could navigate through, than one containing too many details and possibilities.

For a first usage of path finder, using a big value for smallest detail can be useful to blind useless holes.

- With a High repulse effect, displacement tried to move the part will be performed far away from walls made of other parts.
- With a Low repulse effect, displacement tried to move the part will be performed close to walls made of other parts. Therefore, rotating the part results to be more difficult as more clashes will occur
- Resize bounding box allows to limit the area which will be considered in the internal 3D map

Using the resize bounding box option can significantly reduce computation time necessary to create the 3D map. But resizing too much may create holes in the internal 3D map which will lead to unsuccessful tried displacements afterwards.

**Motion:**

**Translation step** is the distance the part will be moved, therefore, the distance between successively analyzed locations will be equal to the translation step.

Clash analysis is performed for each location but not between these locations. Indeed, it is a discrete analysis but not a continuous one

Only location where no clash occurs are kept in the simulation provided by path finder

**Rotation gain** allows to define how much part will be rotated between two analyzed locations

- If rotation gain equals 1, displacement at the extremity of the part will be equal to the step value

- If rotation gain equals 2, displacement at the extremity of the part will be equal to twice the step value
- If rotation gain equals 0.1, displacement at the extremity of the part will be ten times smaller than the step value.

### **Input for path finder**

To speed up the path finder computation, it is better to use a simulation which already contains intermediate steps. But no clash must occurs for each intermediate step included in simulation used as input

Using such more complete input will make algorithm converge quicker.

Input for a new usage of path finder can be a resulting of a previous usage of path finder, even if this previous usage did not provide a complete simulation without clashes.

It is possible to loop on several path finders, starting with a rough translation step value and refining this value in the next usage.



# About Path Finder and Smooth Operations

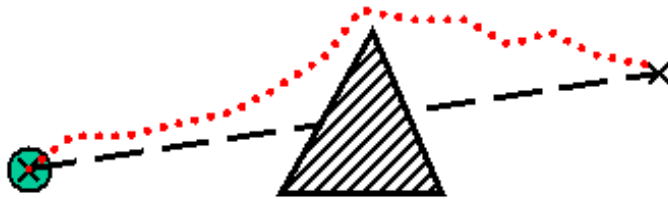


The example below aims at explaining the path finder and smooth operations with respect to a chosen sampling step

**Initial Track:**

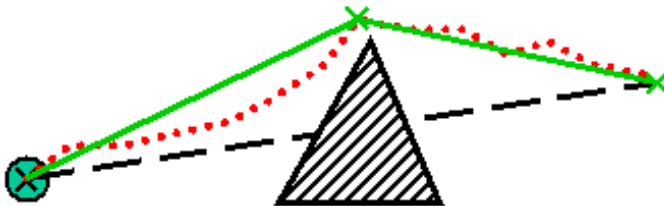
2 positions, 1 clash, sampling step  
value = 5mm

Path Finder operation

**You performed a path finder operation on  
the initial track:**

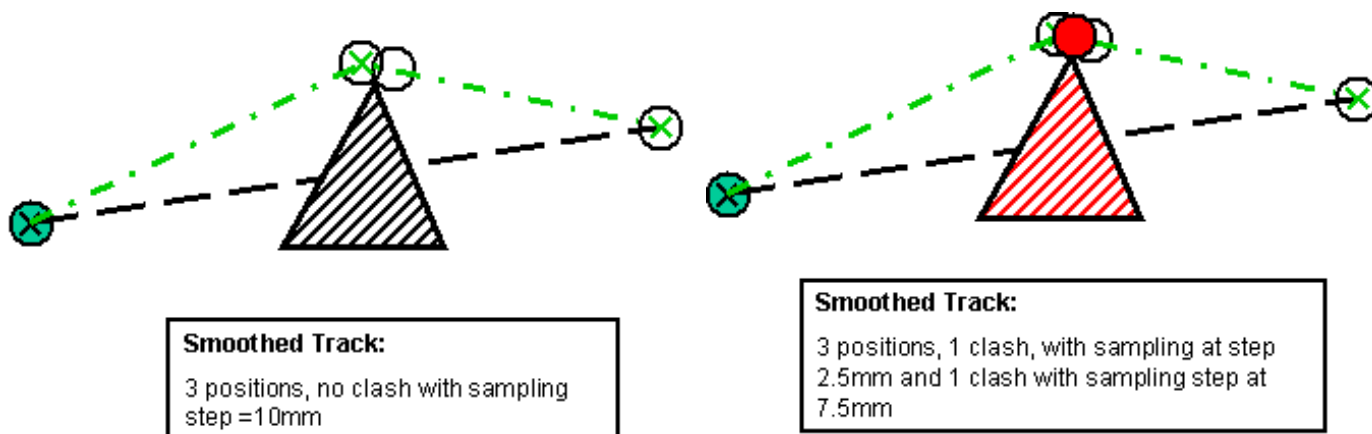
43 positions, no clash, with a sampling step  
value = 5mm

Smooth operations:


**Smoothed Track:**

3 positions, no clash, sampling step  
value = 5mm

We do not guarantee other sampling steps values (translation step). See other examples below:



Please refer to [How to adjust the player...](#)

Now if you want to play your smoothed track use the following icon  from and select the track of interest in the specification tree

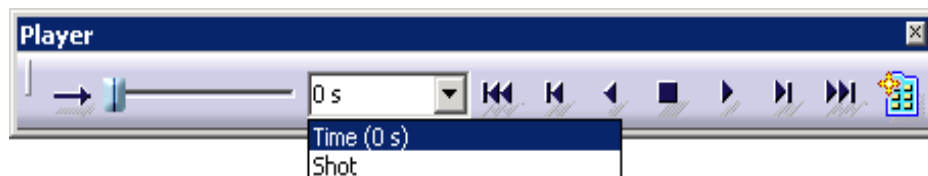
The Player is active. New parameters are available to replay your track with respect to:

- time (in seconds),

**Note:** the default speed is 0.001m\_s (see: [Customizing DMU Fitting](#))

(for instance, if step = 1s the motion corresponds to 1 mm)

- shot parameter or,
- path finder or smooth specifications if previously defined (this is the case in our example)



### How to adjust the player...

To avoid interpolation consistency problems, (fixed by the auto controlled step option in the previous releases), you can now replay your smoothed track using the pathfinder parameter in the player drop-down list:

- selecting pathfinder parameter in the drop-down list swaps to the right interpolation (spatial-based interpolation, smooth specification)



Of course, you can choose to keep the default parameter (time in seconds), but in this case note the consistency between the smooth specification and the way it is played are not guaranteed



# Validating a Motion



**Validate Positions Automatically** (without defining a clash specification): select a simulation and click the arrow within the Clash Detection icon from the DMU Check toolbar. Undock the toolbar if necessary. Activate Clash detection (on or stop modes), then replay your simulation.



**Perform an Experiment and Generate a Report:** Define measures, interferences and distances specifications. Select a sequence (a simulation object), click the Perform Experiment icon from the DMU Simulation.



## Licensing

The following functionalities are available only with a DMU Space Analysis license.



**Detect Clashes:** select two items then click the clash icon. Add the clash specification into an existing simulation. Replay the simulation.



**Measure Distances:** select two items then click the Distance icon. Add the distance specification into an existing simulation. Now as you replay the simulation, distance is measured between the components selected.

# Validating Positions Automatically



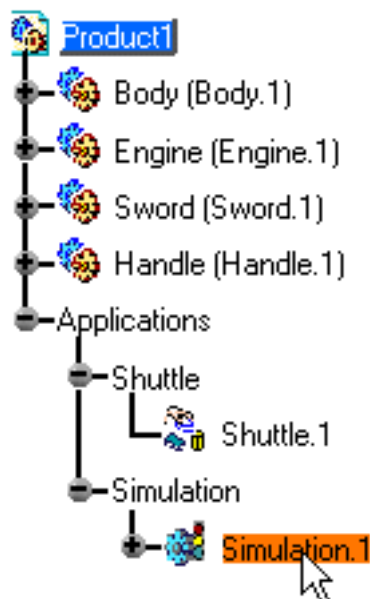
This task shows how to detect clashes automatically within a simulation.



Open the [AUTO\\_CLASH\\_DETECTION.CATProduct](#) document.




1. Double-click **Simulation.1** in the specification tree.






The Edit Simulation dialog box and the Manipulation toolbar appear.

- 2.

Click the arrow within the Clash Detection icon  from the DMU Check toolbar. Undock the toolbar if necessary.



3 modes are available:

- : (Off) default mode, the automatic clash detection is deactivated
- : (On) when activated, the products in collision are highlighted in the geometry area while recording a simulation
- : (Stop mode) the simulation stops at the first clash, the products are highlighted

## Default Clash Detection

When activating the default clash detection, the clash computation can take several minutes before the simulation dialog boxes are displayed.

Sometimes it is more convenient not to compute this clash detection for the first step.

This step can already be validated by a static clash analysis.

- **Activated only while moving:** if checked, no calculation will be performed when the default clash detection is **activated** (ON, STOP). When running tracks, sequences... simulations, clashes are still calculated with respect to the default clash detection state (OFF, ON, STOP)
- By default this option is not checked

3.

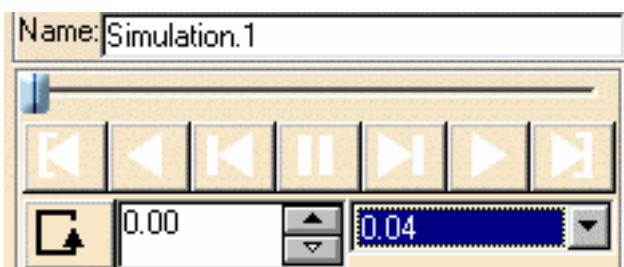


Set the Clash detection to on

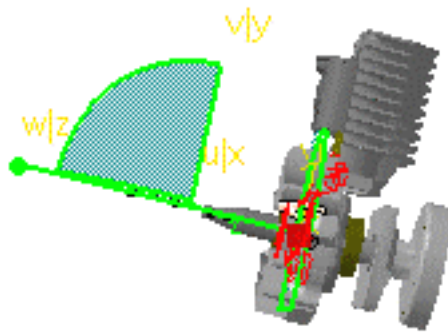
## About Default Clash Detection



On large assemblies, it can take several minutes before the simulation dialog boxes are displayed, you can now deactivate the clash detection on the first step checking the option Activated only while moving available in **Tools->Options->DMU Fitting...**

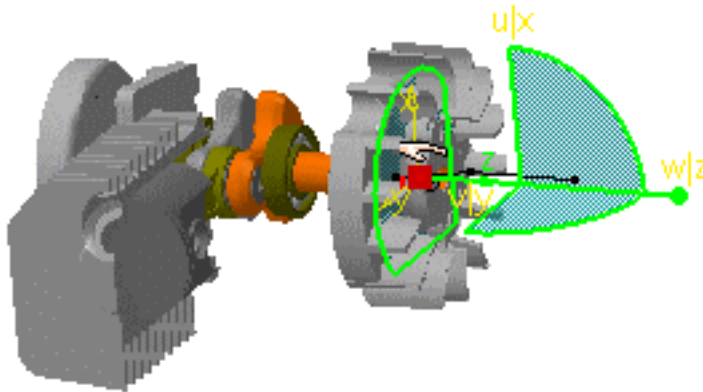
4. Replay your simulation. Select 0.04 via the combo (interpolation steps).



The product in clash with the shuttle is highlighted in the geometry area



5. Set the Clash Feedback option. For this, select **Tools->Options->Digital Mockup->DMU Fitting->DMU Manipulation** and check the Clash Beep option . Click Ok to confirm your operation.  
A beep is emitted when the product is in collision.
6. Set the Clash detection on stop mode .
7. Run your simulation. This time, the simulation stops at the first clash detected.



If you need to perform clash analysis, please refer to [Detecting Clashes](#).



Please refer to the *DMU Space Analysis User's Guide* for more information about detecting and analyzing clashes between products or between groups.





# Measuring Distances



This task shows how to measure distances as or after you record a simulation.



Open the [MEASURING\\_DISTANCES.CATProduct](#) document.

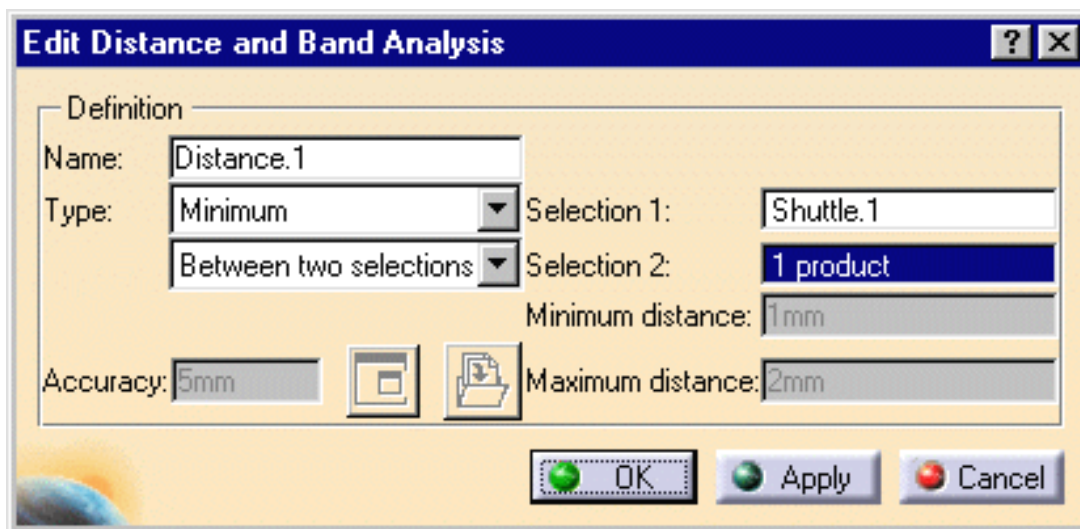


1. In the specification tree, click **shuttle.1**.

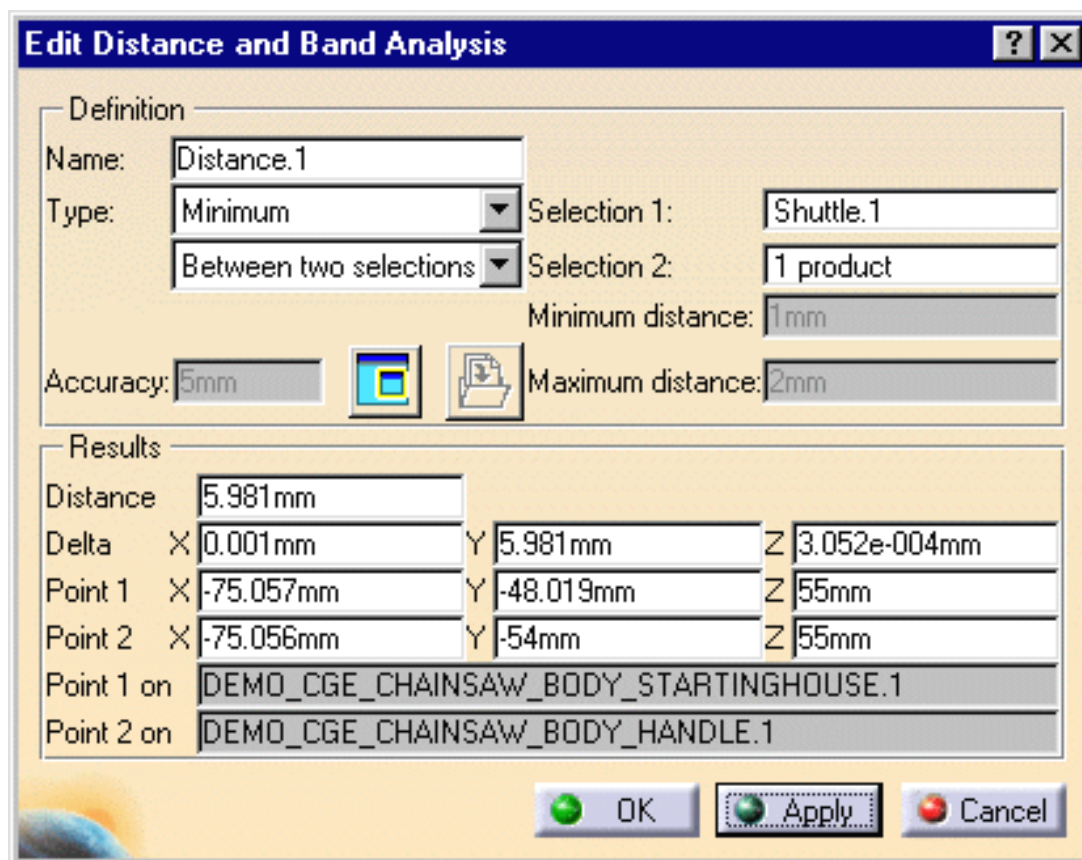
2. Click the **Distance and Band Analysis** icon .


The **Edit Distance** dialog box is displayed. Make sure the distance type is set to **Minimum** and **Between two selections**.

3. You need to define the second selection, for this:  
click in the Selection 2 field and select the handle either in the specification tree or in the geometry area.



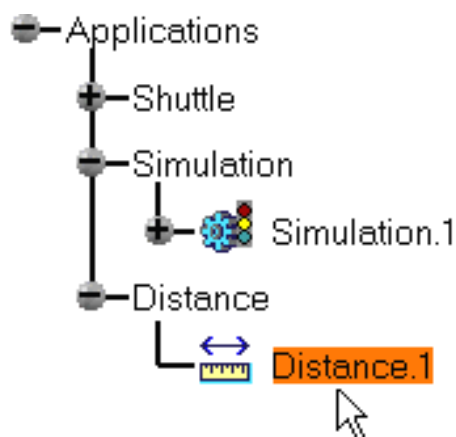
4. Click **Apply**  
The distance is calculated and the Results appear in the Edit Distance and Band Analysis dialog box.



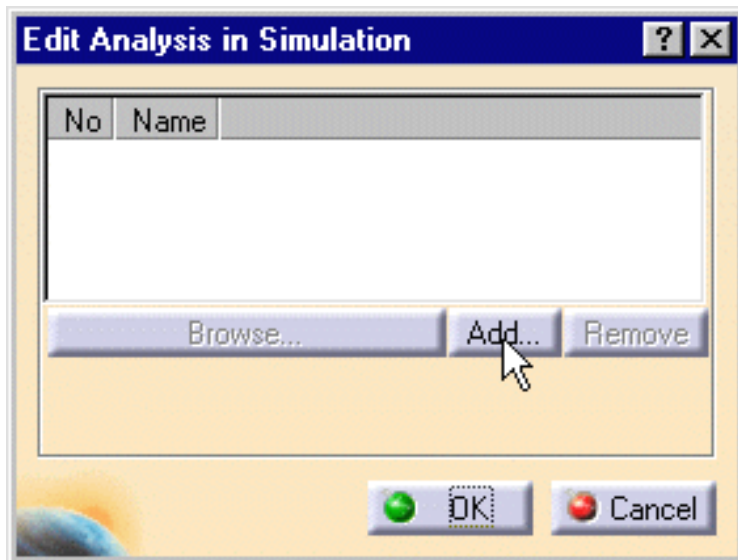
You can also view the results in a separate viewer. To do so, click the Results window  icon which is now in the Edit Distance and Band Analysis dialog box. Object viewing commands and commands in the Window menu are available in this window. For example, using the Windows menu, you can tile the results window and the original document window vertically or horizontally.

**5. Click **Ok****

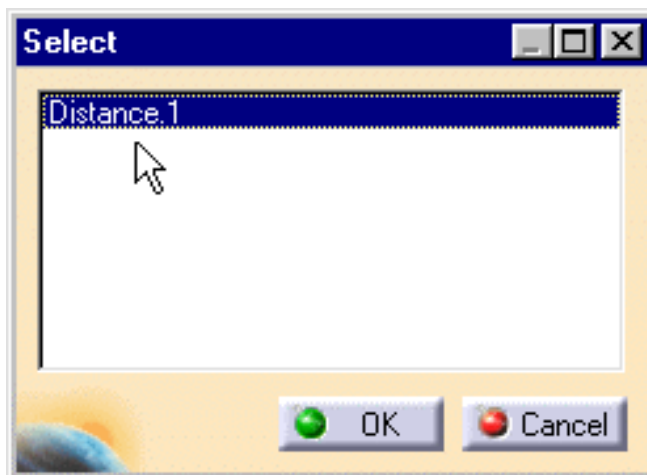
The specification tree is updated and the calculated distance appears in the geometry



- 6. Double-click on Simulation.1 in the specification tree.**  
The Edit Simulation dialog box is displayed.
- 7. Click **Edit Analysis**.**  
The Edit Analysis in Simulation dialog box is displayed.

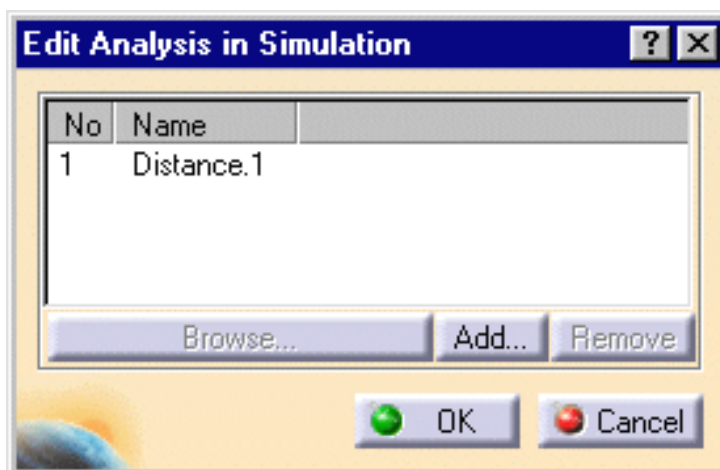


8. Click **Add** then select **Distance1** from the displayed dialog box..

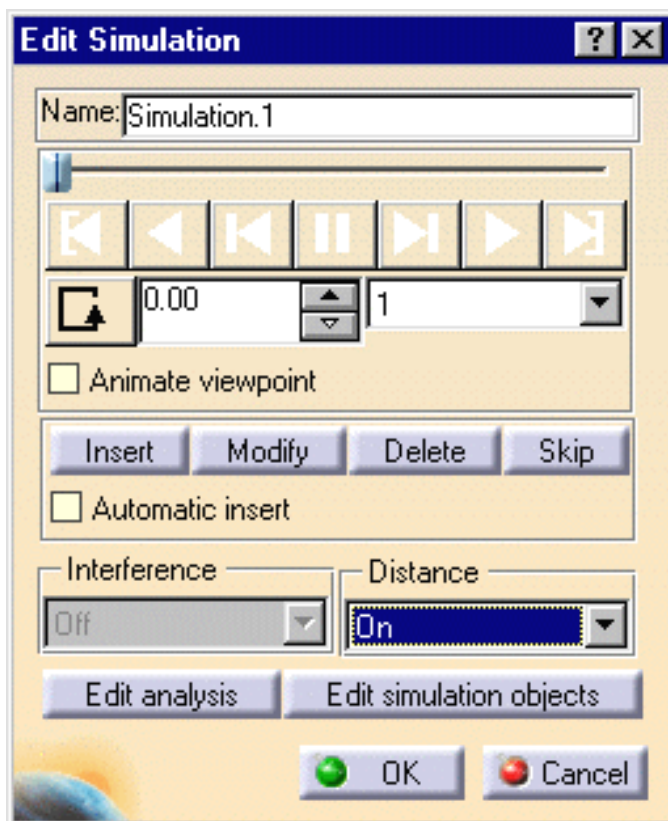


The Edit Analysis Simulation dialog box is updated.

9. Click **Ok**.

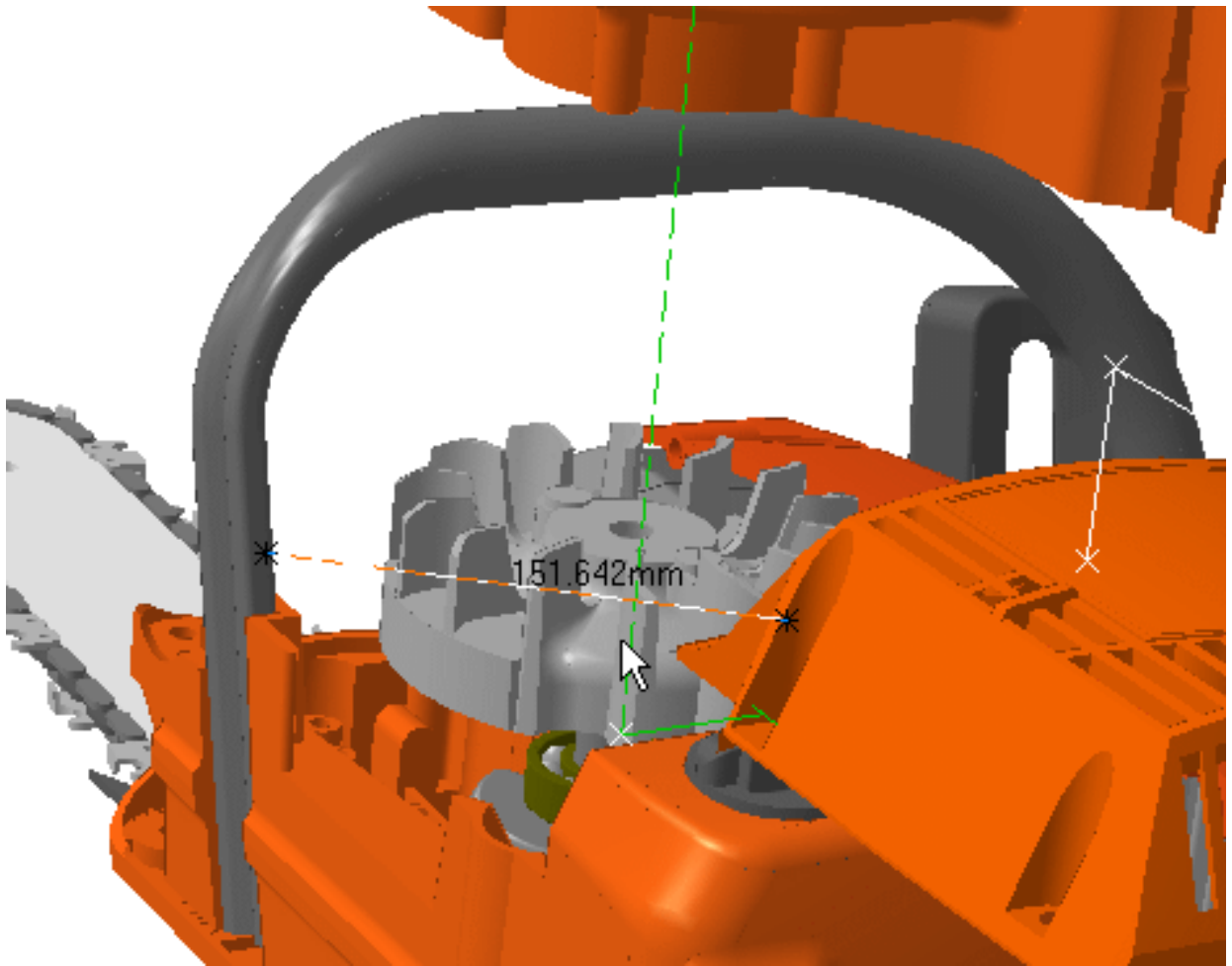


10. Set the **Distance** combo to **On** in the Edit Simulation dialog box, then click OK.

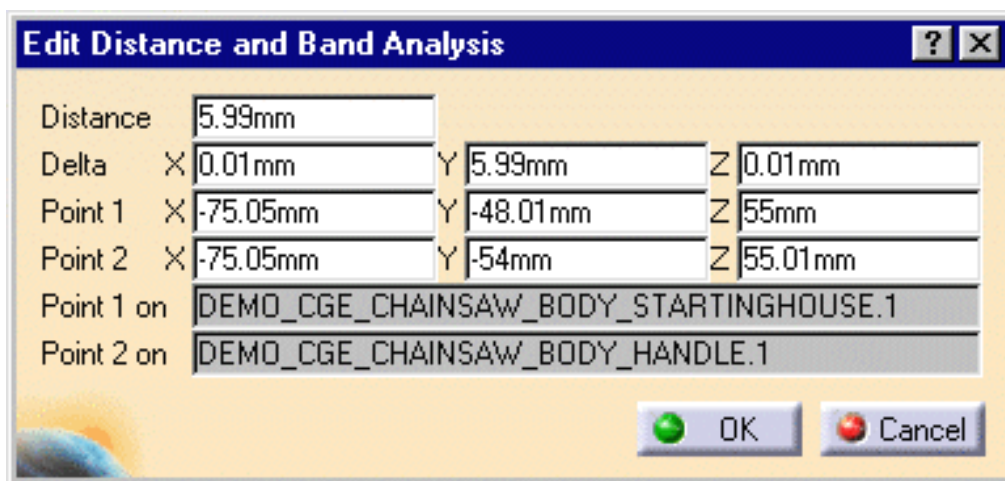


- 11.** Click the required switches for replaying shots.

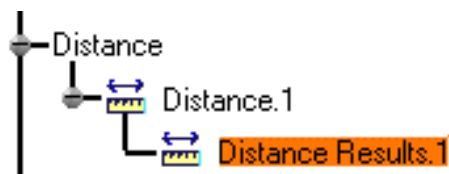
The minimum distance (represented by a line , two crosses and a value) appears in the geometry area as shown below:



- 12.** You can display detailed information about the minimum distance at a given step by clicking on the **Distance Results** object in the specification tree. You can also use the **Browse** switch from the **Analysis** tab.



- 13.** The specification tree is updated.





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




This task shows how to detect clashes while recording a simulation

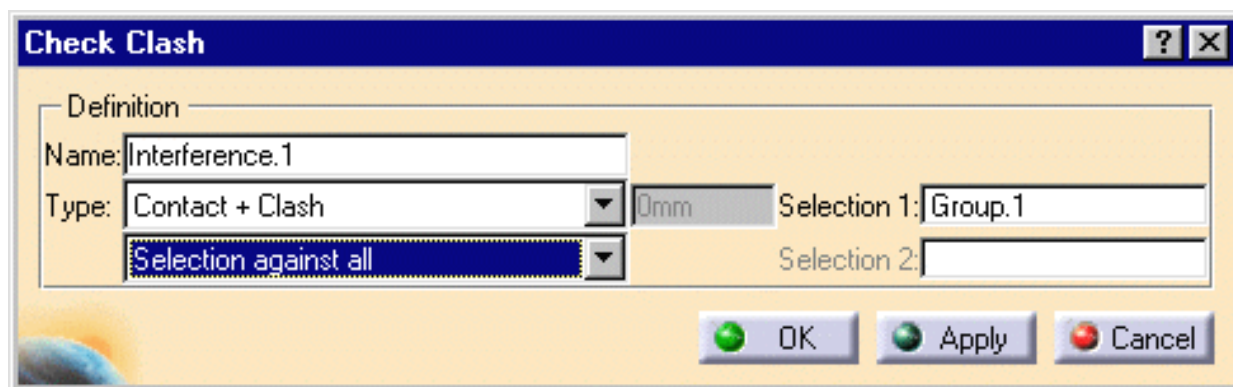


Open the [DETECTING\\_CLASHES.CATProduct](#) document.

**Group:** please refer to [Defining Several Shuttles](#) and for more detailed information refer to *DMU Navigator User's Guide*.



1. In the specification tree, click **Group 1**.  
The product is highlighted in the specification tree.
2.  .  
Click the Clash icon .  
The Check Clash dialog box appears. An entry for the interference appears in the specification tree.
3. Activate the first Type drop-down list box and select Contact + Clash.
4. Click the second Type drop-down list box and select Selection Against All.



5. Click **Apply** to check for interferences  
The clash calculation is launched

Advantage is taken of multi-CPU computers to distribute the clash calculation among processors, thus reducing computation time.

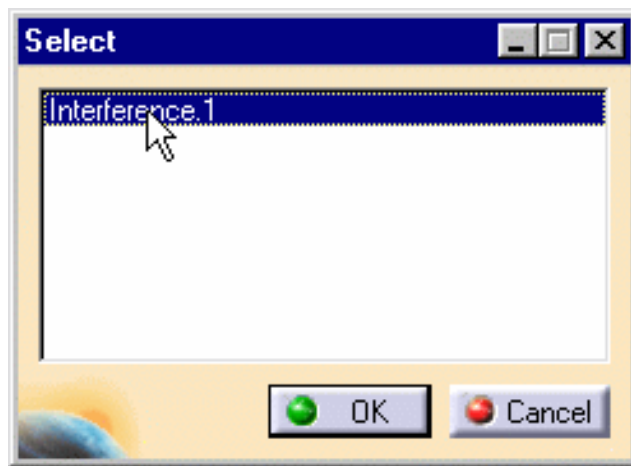
6. Click **OK**.
7. Double-click on **Simulation.1** in the specification tree.  
The **Edit Simulation** dialog box is displayed.

8. Click **Edit Analysis**.

The Edit Analysis in Simulation dialog box appears.

9. Click **Add** then select Interference1 from the displayed dialog box.

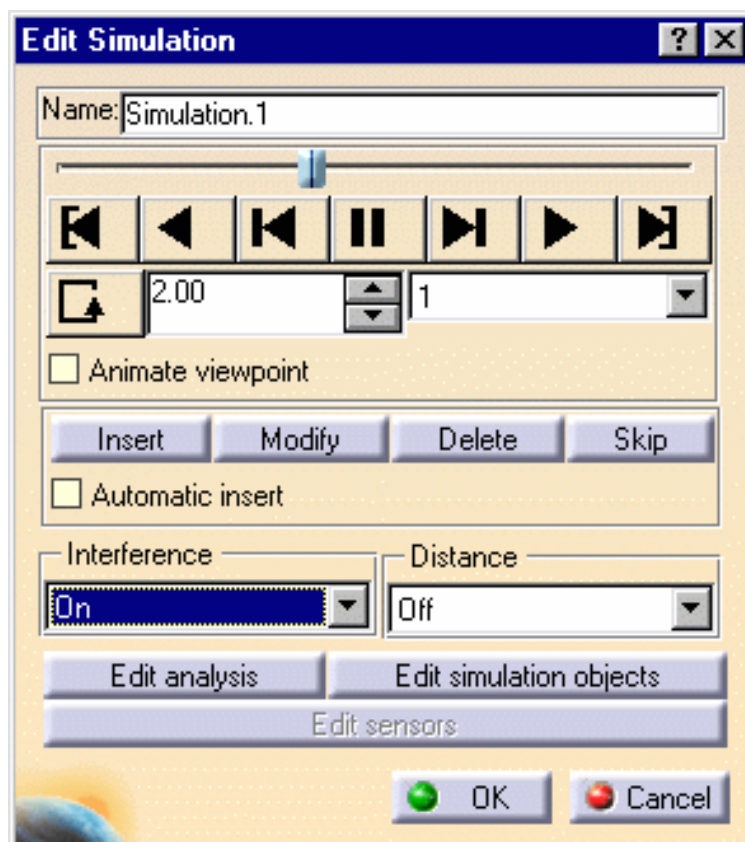




Click **OK**

The **Edit Analysis Simulation** dialog box is updated


10. Click **OK** to confirm your operation.
11. Set the interference combo to **On** in the **Edit Simulation** dialog box, then click **OK**.





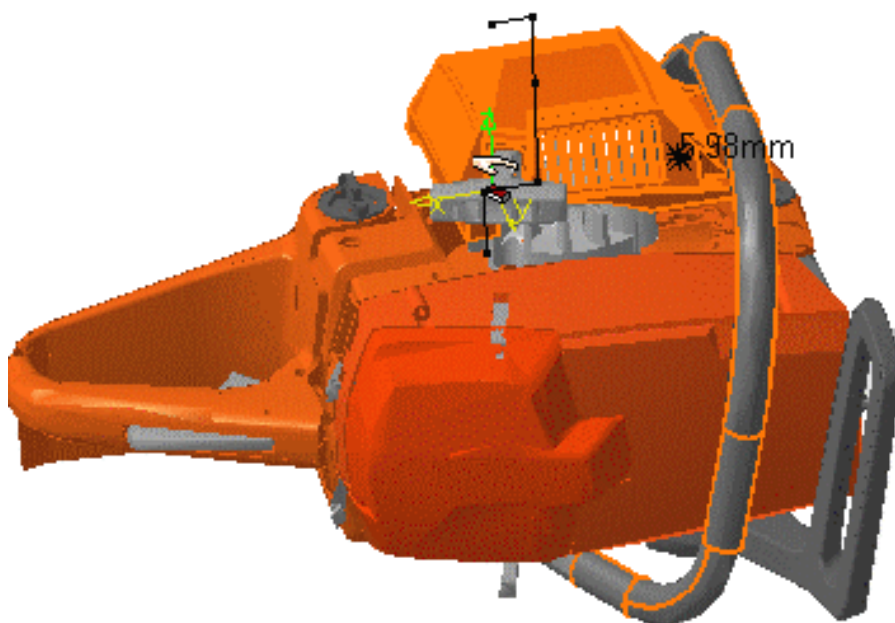
12. Note that you can customize the Clash Beep via **Tools->Options...**

Set the Clash Feedback option. For this, select **Tools->Options ->Digital Mockup->DMU**

**Fitting->DMU Manipulation** and check the Clash Beep option . Click Ok to confirm your operation.

A beep is emitted when the product is in collision.

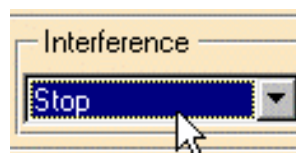
13. Replay the fitting simulation. The product in collision is highlighted and a beep is emitted.



The specification tree is updated.

14. Now set the interference combo to Stop and click **OK**.

The simulation stops at the position where a collision is detected between **Group 1 (Shuttle 1)** and the other products.



The product in collision is highlighted.

The value of the Current time option provides precise time of the clash.



The interference combo is still set to Stop. If you manipulate the shuttle1 directly, the simulation stops just before the collision is detected. Thus, you can record the exact position

15. To locate the interference positions more precisely, select the **Edit Analysis** button and select Interference1 in the the **Edit Analysis in Simulation** dialog box. Then, press the **Browse** switch described in the previous distance step.

Check Clash

Definition

Name:Interference.1

Type:Contact + Clash5mmSelection 1:Group.1Selection against allSelection 2:

Results

Number of interferences: 2 (Clash:0, Contact:2, Clearance:0)

Filter list:All typesNo filter on valueAll statusesApply filters

List by Conflict

List by Product

Matrix

No.	Product 1	Product 2	Type	Value	Status
1	DEMO_CGE_C...	DEMO_CGE_C...	Contact		Not inspect...
2	DEMO_CGE_C...	DEMO_CGE_C...	Contact		Not inspect...

DeselectMore >>

OK

Apply

Cancel



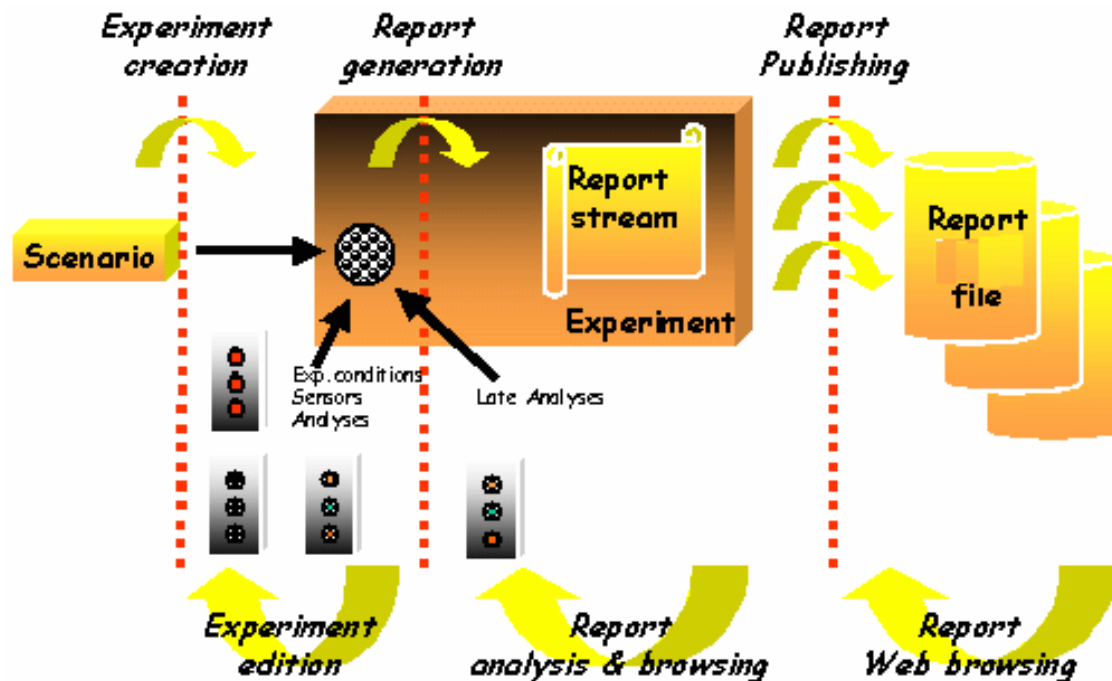
Please refer to the *DMU Space Analysis User's Guide* for more information about detecting and analyzing clashes between products or between groups.



# Performing an Experiment and Generating a Report



## About Experiments and Simulation Reports:



A new object called Experiment is created and displayed in the specification tree when clicking the Perform Experiment icon from the DMU Simulation toolbar.

## How does it work?

- One scenario is associated at creation; scenarios are all objects that can be simulated using the Player (Sequence, Track, Simulation)
- None or several sensors are associated as outputs from the experiment; each of them is then set as observed if their values versus time must be traced in the report of simulation

### Experiment Report Generation:

- Report generation is a one shot operation, where observed sensors are traced versus time, while the simulation is proceeding.
- Report is an object aggregated to the Experiment as a design table (knowledgeware component).



This task shows how to perform an experiment and generate a report.



Open the [PERFORM\\_EXPERIMENT.CATProduct](#) document.

The document already contains analyses (measures, interferences and distances specifications)

The sequence 4 comprises analyses (distances and measures)



When creating measures make sure you perform your selection in the specification tree so that they remain persistent.



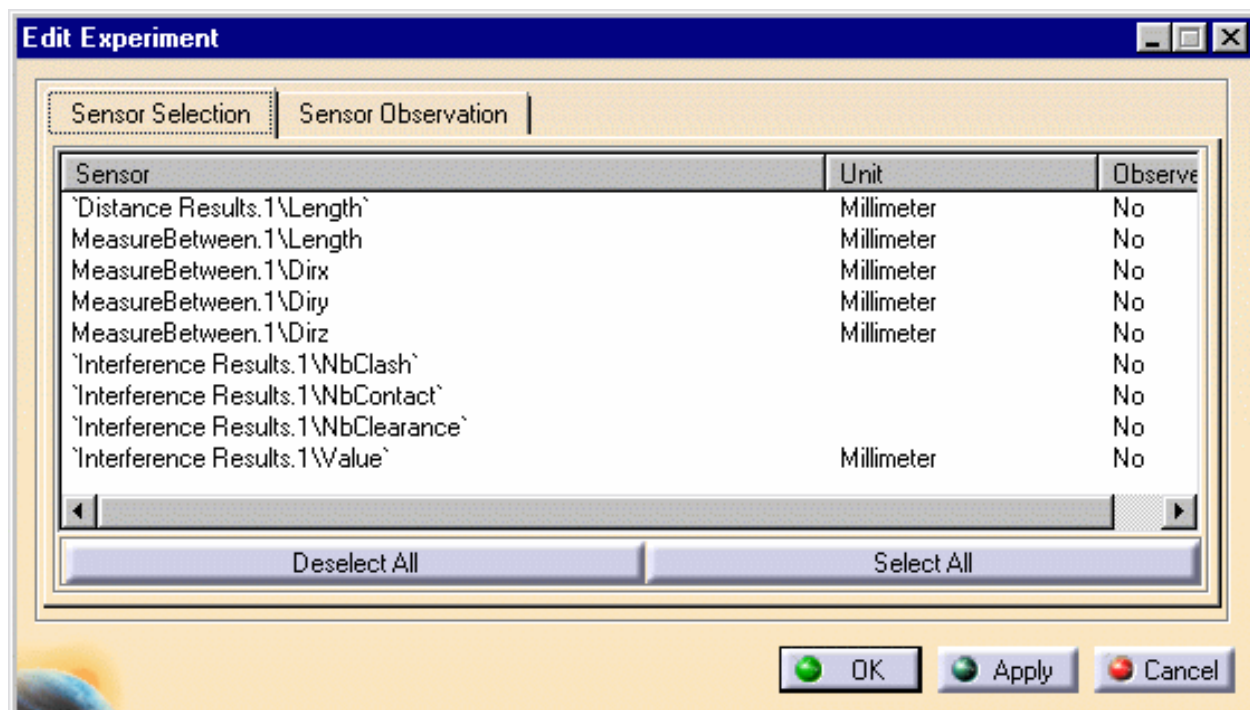
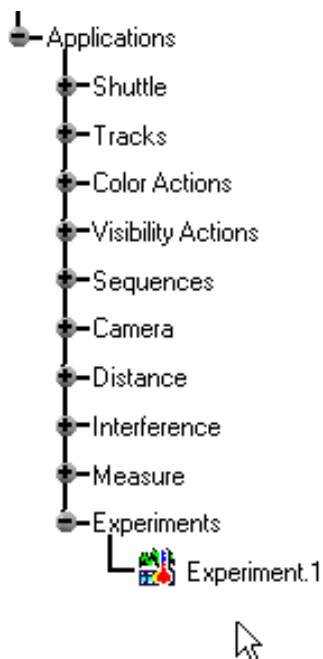
1. In the specification tree, select Sequence.4

2.



Click the Perform Experiment icon from the DMU simulation toolbar

An Experiment object is identified in the specification tree and The Edit Experiment dialog box appears



3. Select the sensors to be observed, in our example click Select All button.



## About interferences

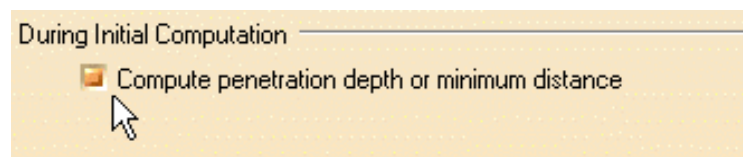
When an interference is associated to an experiment (i.e. as a sensor)  
The sensor "value" :

MeasureBetween.1\Dirz	-2.35188	Millimeter
'Interference Results.1\NbClash'	0	
'Interference Results.1\NbContact'	4	
'Interference Results.1\NbClearance'	16	
'Interference Results.1\Value'	1.83706	Millimeter

represents either:

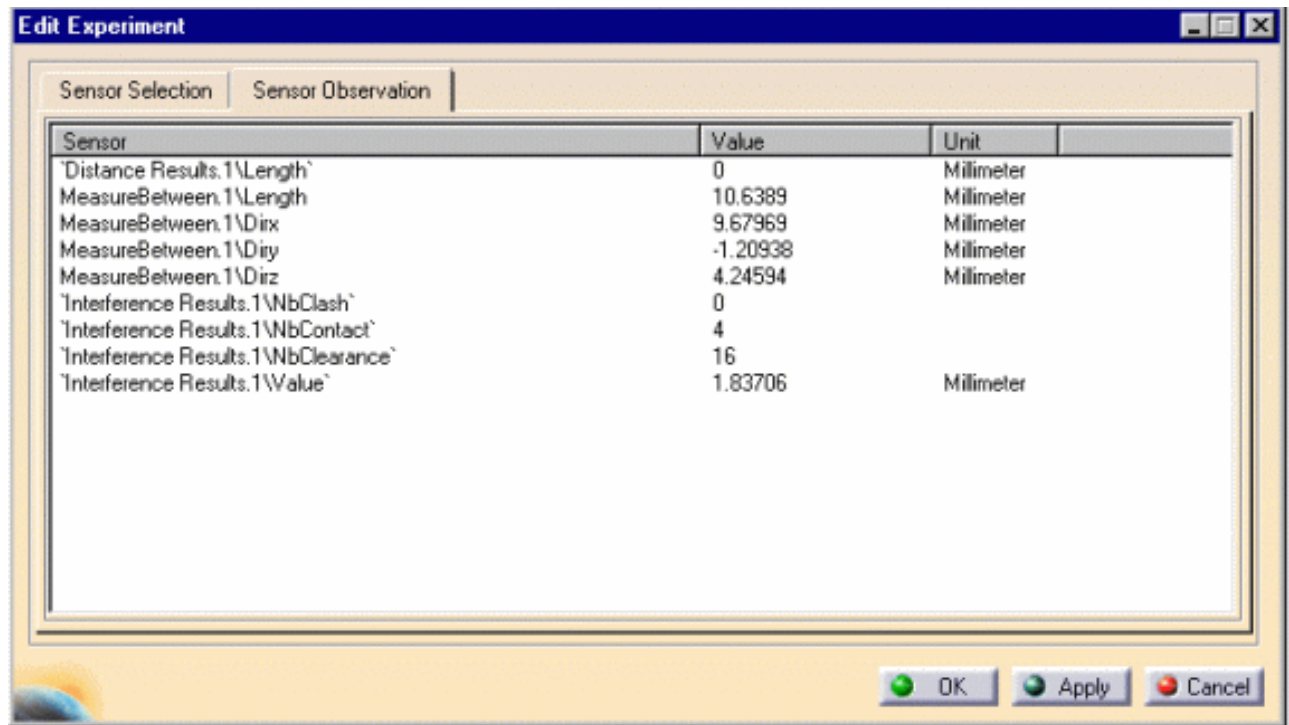
- the penetration depth (if there are clashes in the specification results) or
- the clearance value (if there are only clearances in the specification results)

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



4. Click the Sensor Observation tab and play your sequence using the Player VCR buttons:

The sensors are logged as the sequence is being simulated



5. Move slider to the starting point .

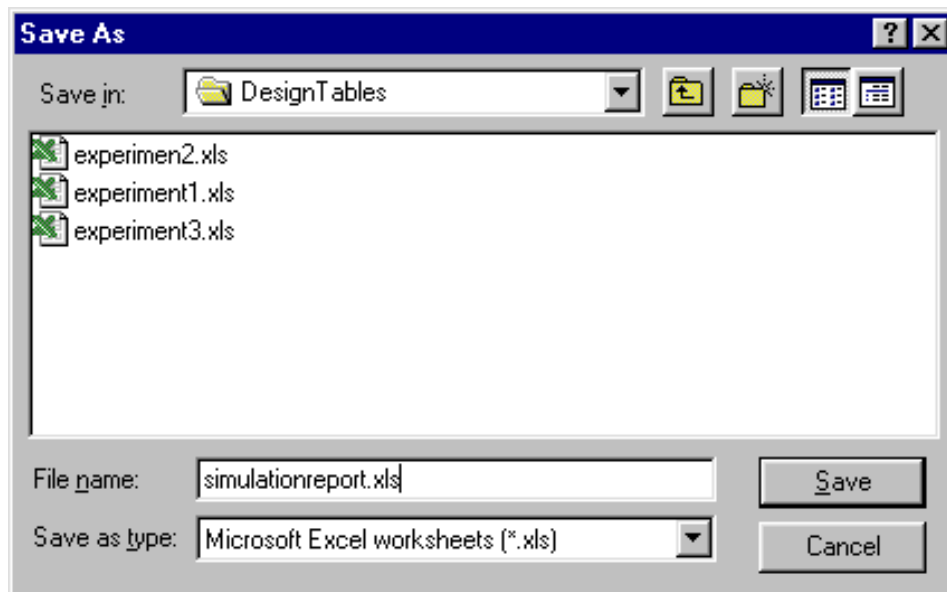


6. Click **Apply**

The Save As dialog box is displayed

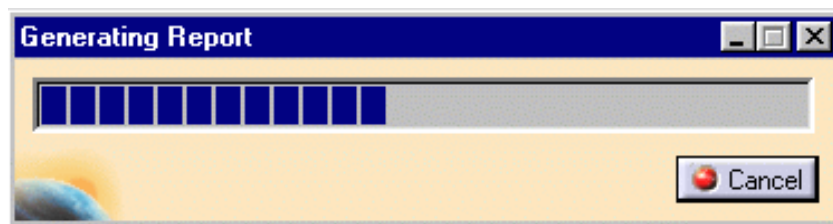
7. Enter a meaningful name, for instance simulation report and select a path.  
You are going to generate a simulation report which is going to be a design table
8. Select the format either txt or xls from the Save As type drop-down list.

In our example select ..xls type



9. Click **Save** in the Save As dialog box

The progress bar is displayed letting you monitor and, if necessary, interrupt (**Cancel** option) the calculation

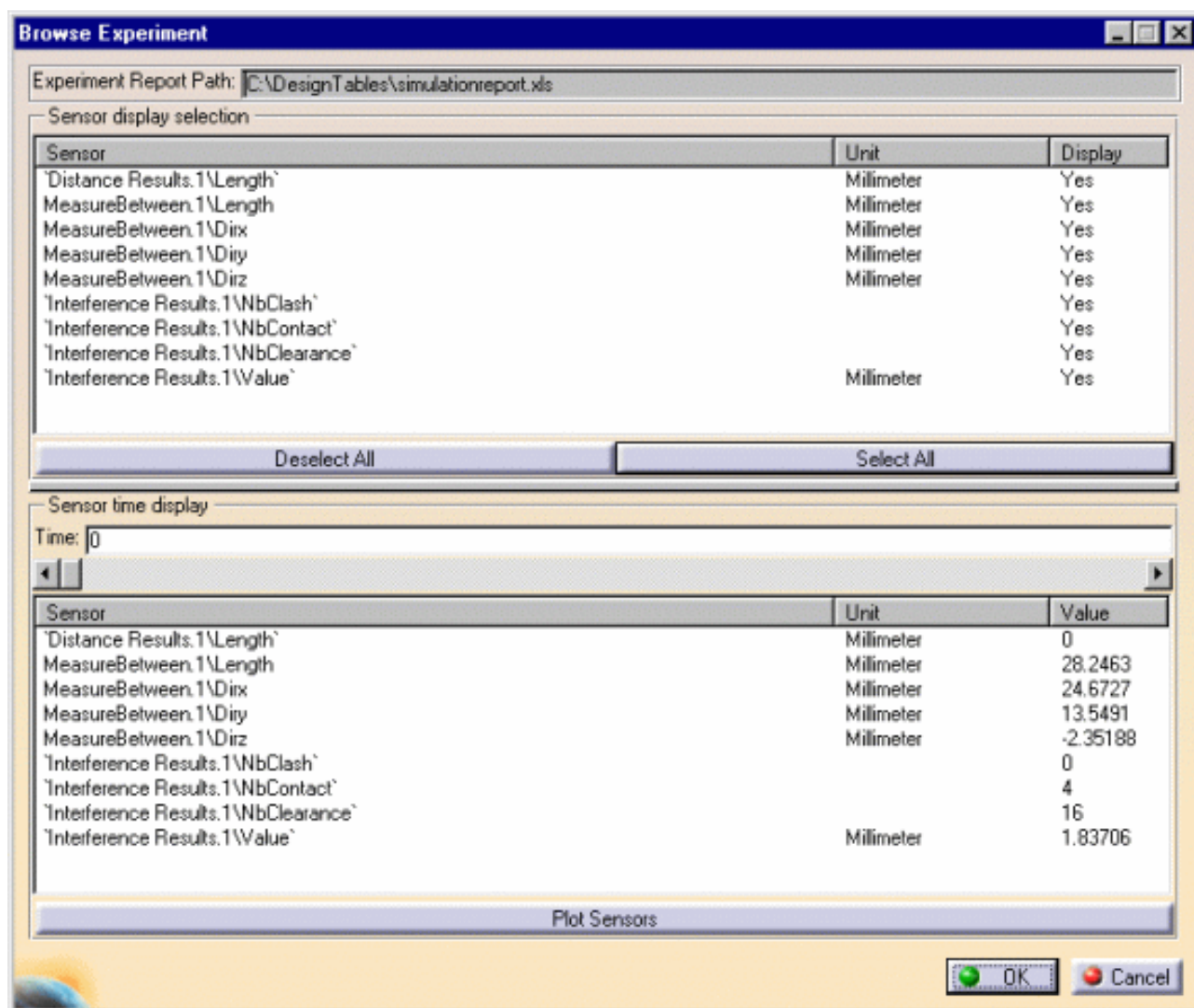


The sequence is simulated

The Browse Experiment dialog box is displayed

10. Select the sensors to be observed:  
Click Select All button

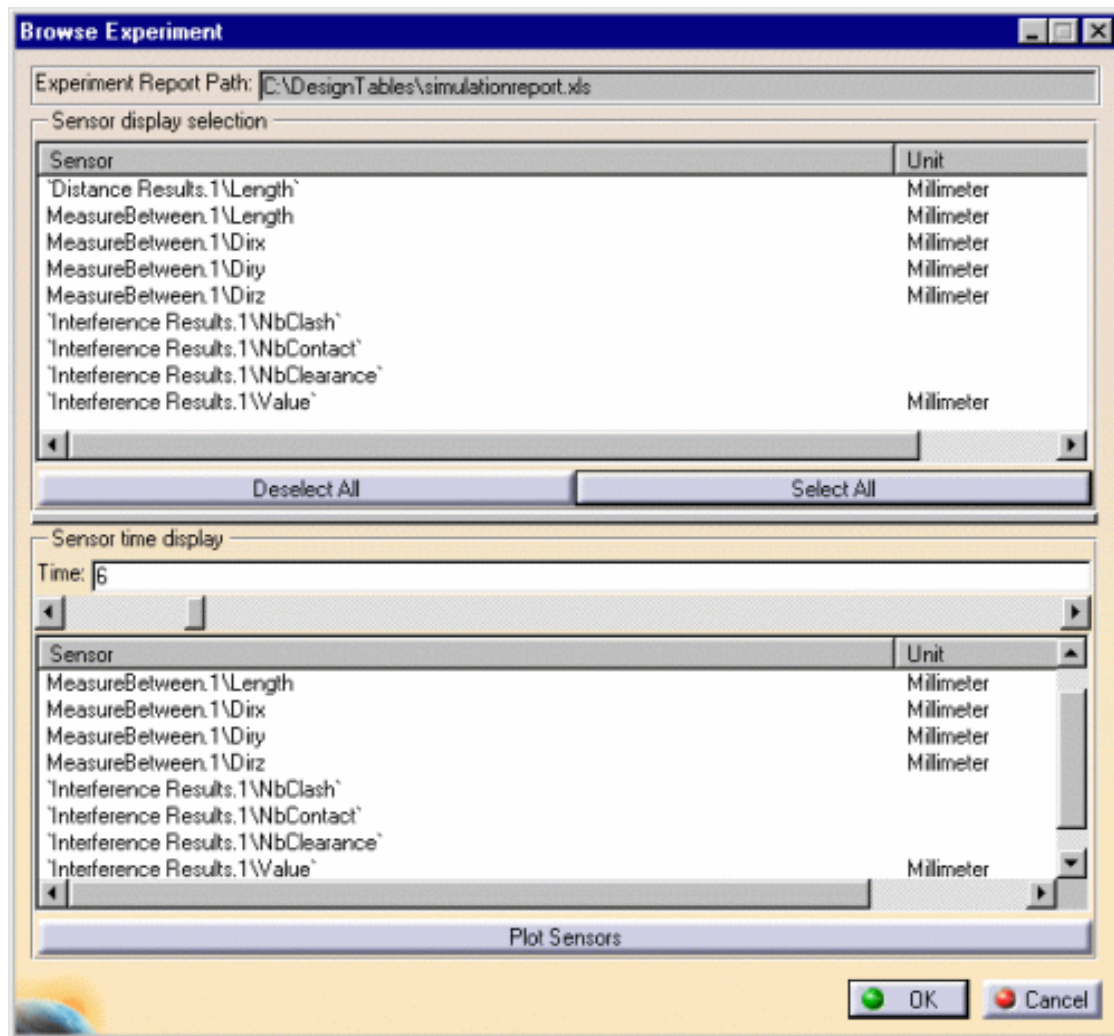




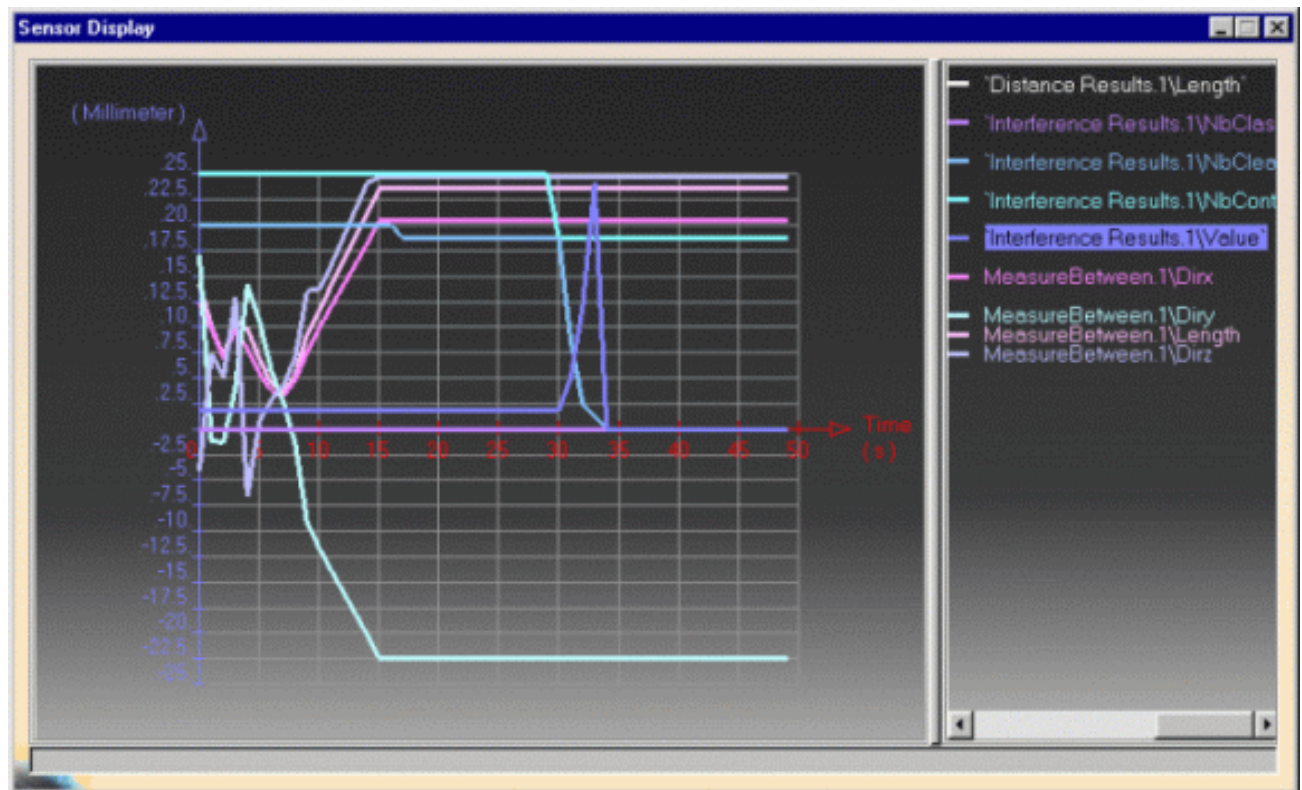
11. The Sensor time display dialog box part lets you refine your analysis
12. Use the slider or enter a precise value in the Time field if you want to check one particular shot

The values are updated accordingly





13. Click Plot Sensors to display a graphical representation of the simulation report generated for experiment.1



14. Open your simulation report

This is what you obtain:

Microsoft Excel - simulationreport.xls

Prompt

File Edit View Insert Format Tools Data Window Help

B1 = 'Distance Results.1\Length' (mm)

	A	B	C	D	
1	Time (s)	'Distance Results.1\Length' (mm)	MeasureBetween.1\Length (mm)	MeasureBetween.1\Dirx (mm)	MeasureBetw
2	0	0	28.2463	24.6727	
3	1	0	20.2157	19.6979	
4	2	0	14.2069	13.8444	
5	3	0	21.5316	19.8274	
6	4	0	19.7268	15.7128	
7	5	0	13.7966	10.8623	
8	6	0	8.88185	7.39387	
9	7	0	7.20755	6.40045	
10	8	0	10.6389	9.67969	
11	9	0	18.73	15.1529	
12	10	0	23.3739	19.7977	
13	11	0	28.1189	23.992	
14	12	0	32.8662	28.1213	
15	13	0	37.6142	32.2507	
16	14	0	42.3629	36.3801	
17	15	0	47.1326	40.9422	
18	16	0	47.1326	40.9422	
19	17	0	47.1326	40.9422	
20	18	0	47.1326	40.9422	
21	19	0	47.1326	40.9422	
22	20	0	47.1326	40.9422	
23	21	0	47.1326	40.9422	
24	22	0	47.1326	40.9422	
25	23	0	47.1326	40.9422	

Sheet1

Ready

NUM



If you delete the design table created (our simulationreport.xls) in our example, the next time you open your document, the experiment will be empty



# Swept Volume



**Define a Swept Volume:** you generated a replay object. click the Swept Volume icon, then select the required settings in the dialog box displayed. Click **Preview**, then **Save** and save as a format type of your choice (cgr, wrl, .model)

**Filter Swept Volume Positions:** click the Swept Volume icon, then select the required settings in the dialog box displayed, check the Filter Positions option and enter a Filtering precision. Click **Preview** and **Save** as a .cgr file.

**Define a Swept Volume from any Moving Reference:** click the Swept Volume icon, then select the required settings in the dialog box displayed, fill in the Reference field. Click **Preview** and **Save** as a .cgr file.

**More About Swept Volume:** provides examples and explanations about swept volume functionality.

# Defining a Swept Volume



Defining a swept volume can be very useful after design changes. For example, you might need to check the assembly dismounting path and thus replay the corresponding simulation. If you define a swept volume you won't need to replay the simulation, you will just have to check the swept volume is not in clash with the environment.



Open the [FIT\\_SWEPT\\_VOL.CATProduct](#) document.

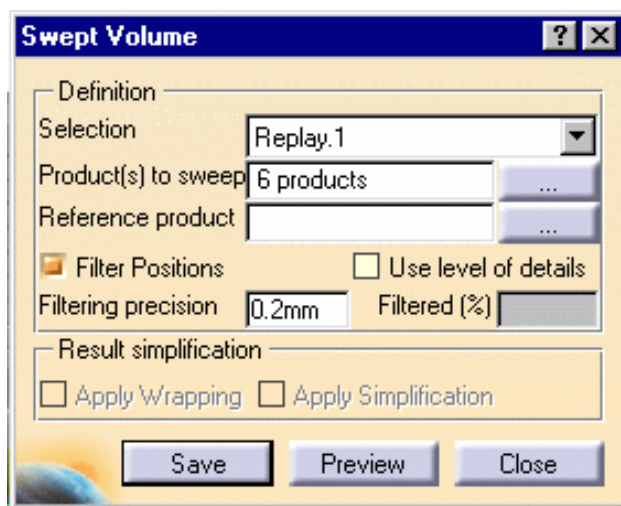
You recorded a Simulation and compiled it. You obtained a Replay object. Please refer to [Generating a Replay](#).



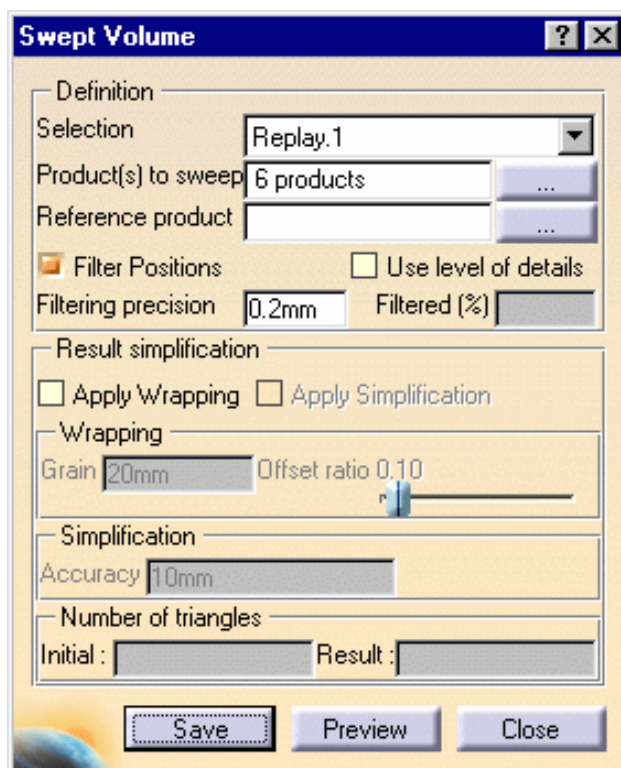
1.



Click the Swept Volume icon.  
The Swept Volume dialog box is displayed.

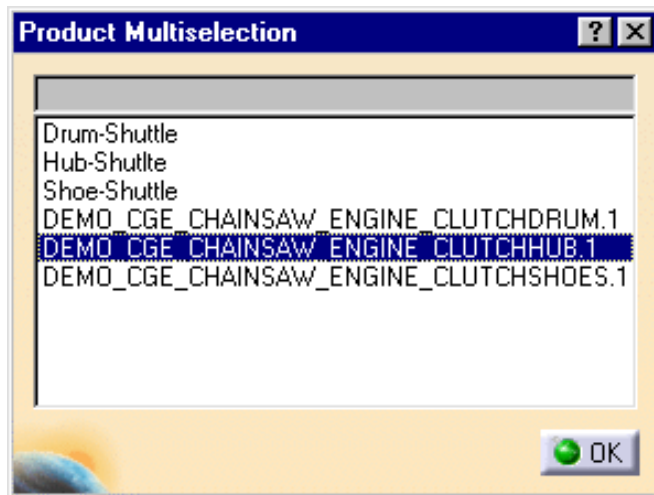


Wrapping and Simplification options are available within the swept volume dialog box if you have DMU Optimizer license:



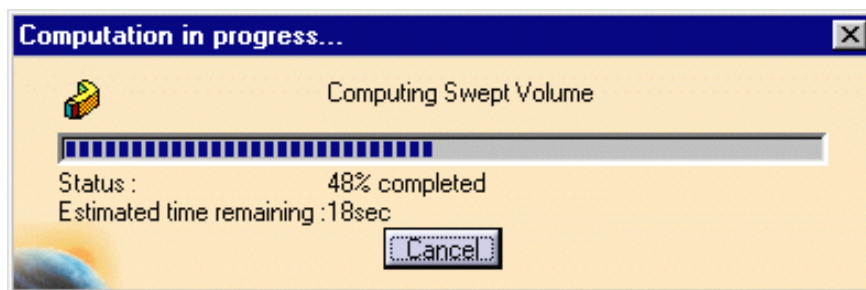
For more information please read Swept Volume section in *DMU Optimizer User's Guide*

2. Click in the spin box, the selection list dialog box lets you select or deselect the products you want to sweep.
3. Select DEMO\_CGE\_CHAINSAW\_ENGINE\_CLUTCHHUB.1
4. Click **Ok**.

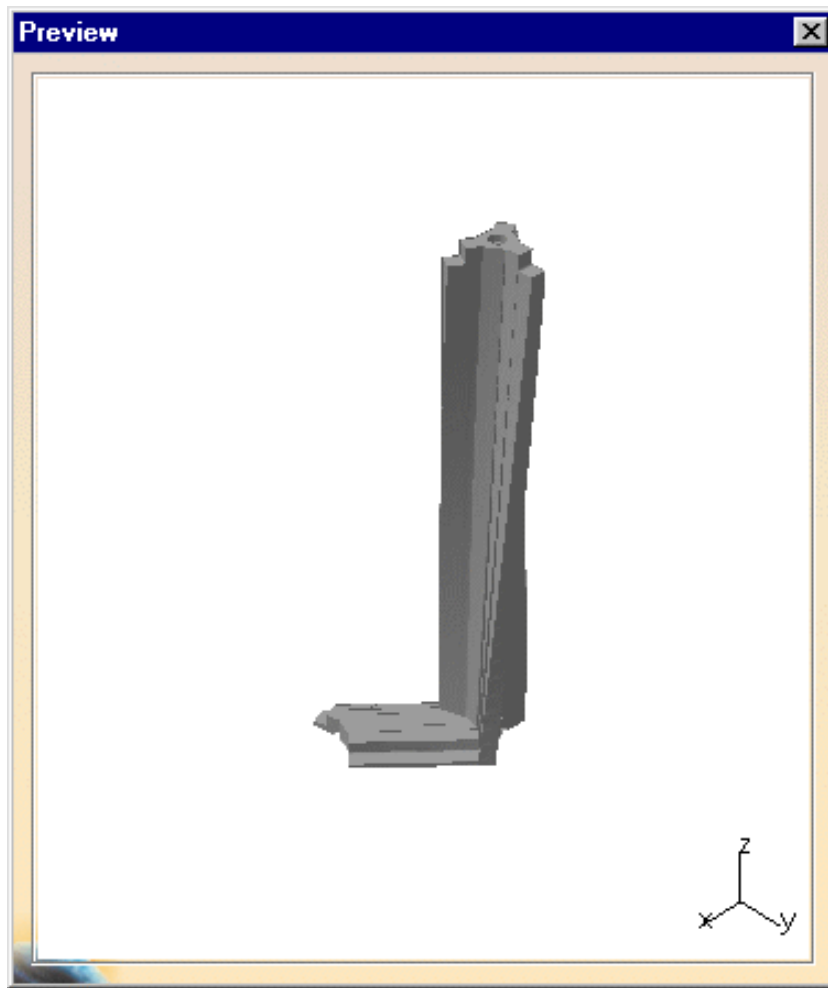


5. Click **Preview** to generate the swept volume

The progress bar is displayed letting you monitor and, if necessary, interrupt (**Cancel** option) the calculation.



This is what you obtain:



6. Click **Save**.

The Save As dialog box appears:

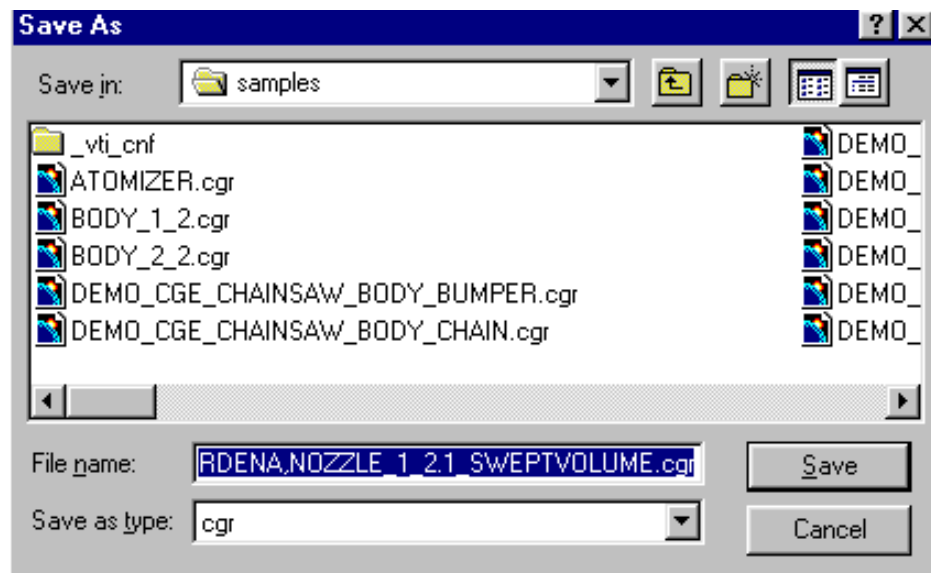
**Note:** you can save your result as a CATIA model file.

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

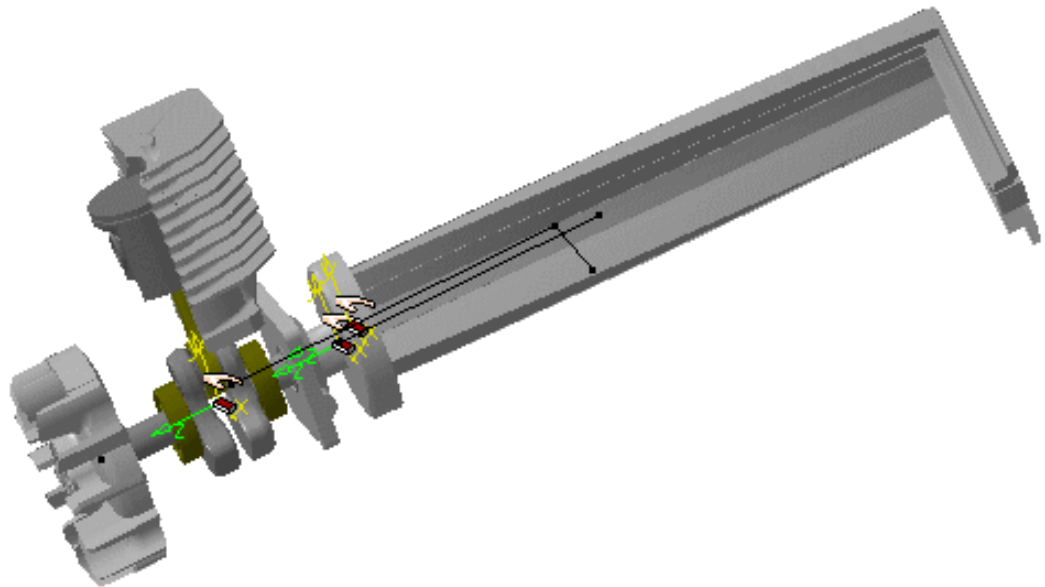
7. Select cgr file from the drop-down list



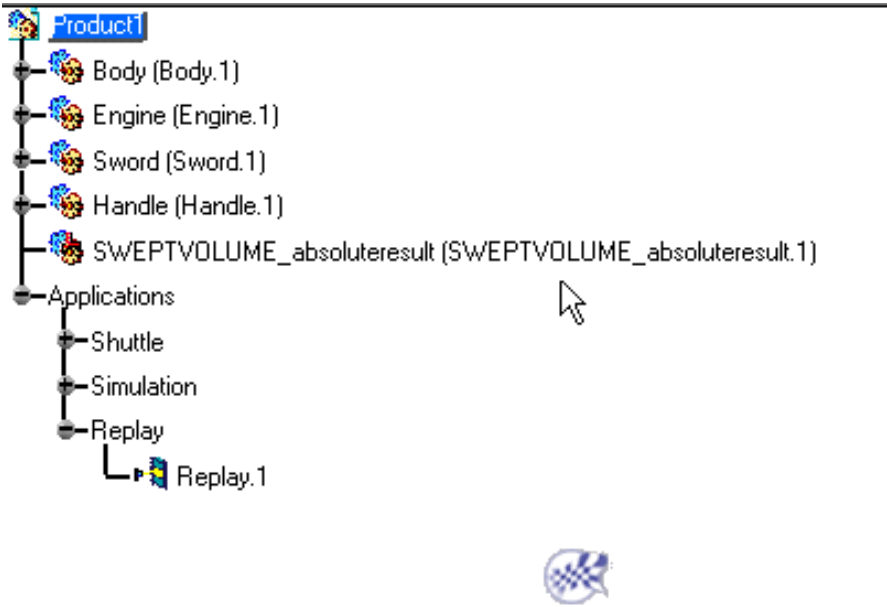


8. Click **Save**.
9. Click **Close**
10. Insert the SWEPTVOLUME\_absoluteresult.cgr into Product1, for this right-click Product1 and select **Components->Existing Component** from the contextual menu displayed.

The Swept volume is identified in the specification tree and in the geometry area.







# Filtering Swept Volume Positions



This task shows how to filter swept volume positions which provides a performance gain.



Open the [Product1duo2.CATProduct](#) document.

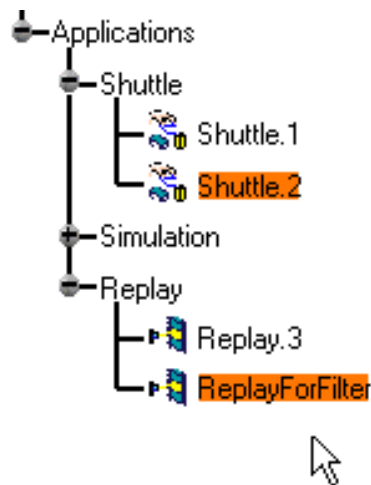
You recorded a multi-shuttle simulation and compiled it. You obtained a Replay object.



You need to perform clash analysis between two items. You generated a simple swept volume but you are not satisfied with the swept volume result. You need a coarser result.




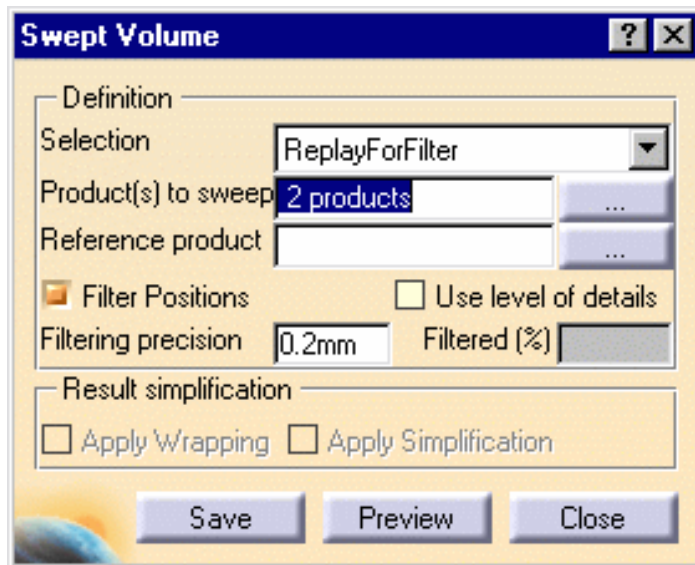
1. Select **ReplayForFilter** in the specification tree.



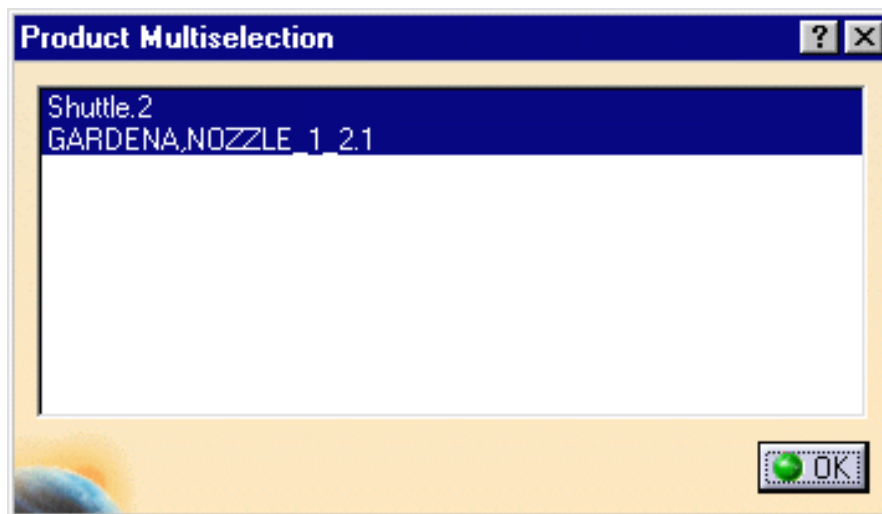
- 2.



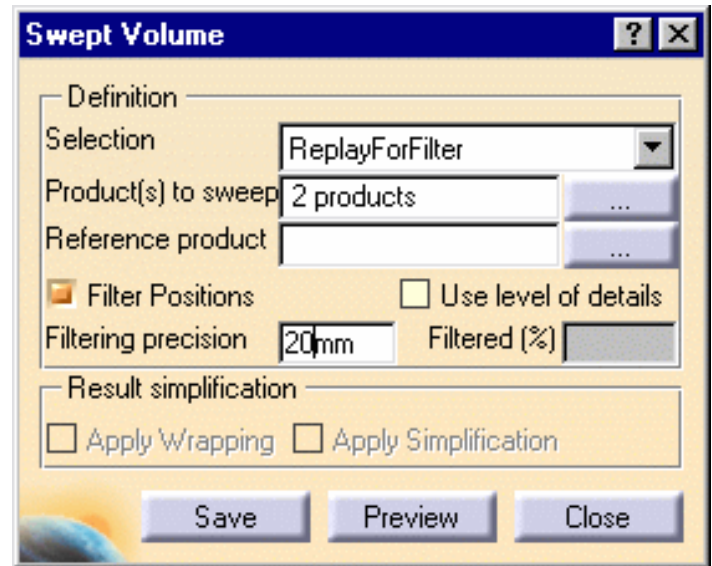
Click the Swept Volume icon .  
The Swept Volume dialog box is displayed.



3. Click in the product(s) to sweep spin box, the Product Multi-selection dialog box lets you select or deselect the products you want to sweep.

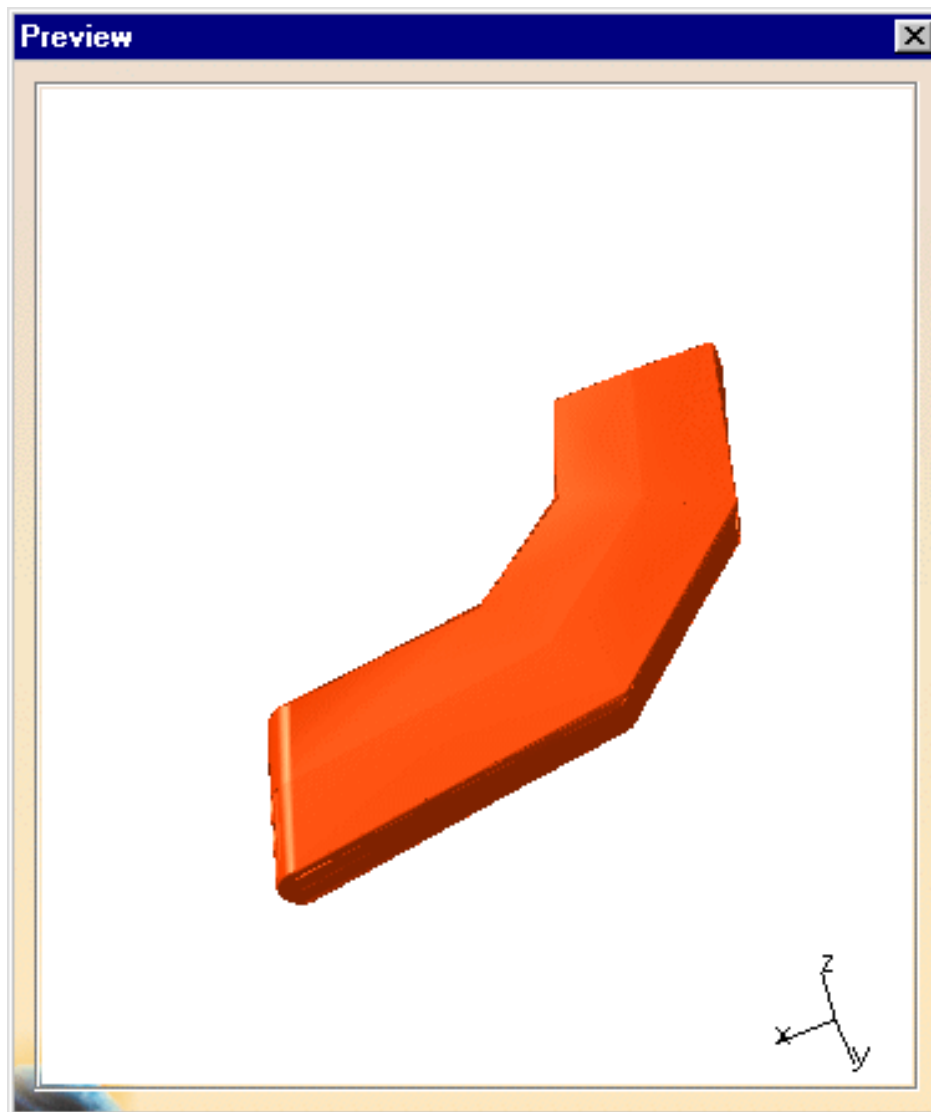


4. Multi-select shuttle 2 and GARDENA\_NOZZLE\_1\_2.1
5. Click **Ok** to confirm your operation
6. Keep the Filter Positions option set.
7. Enter 20mm as filtering precision value.



Click **Preview**

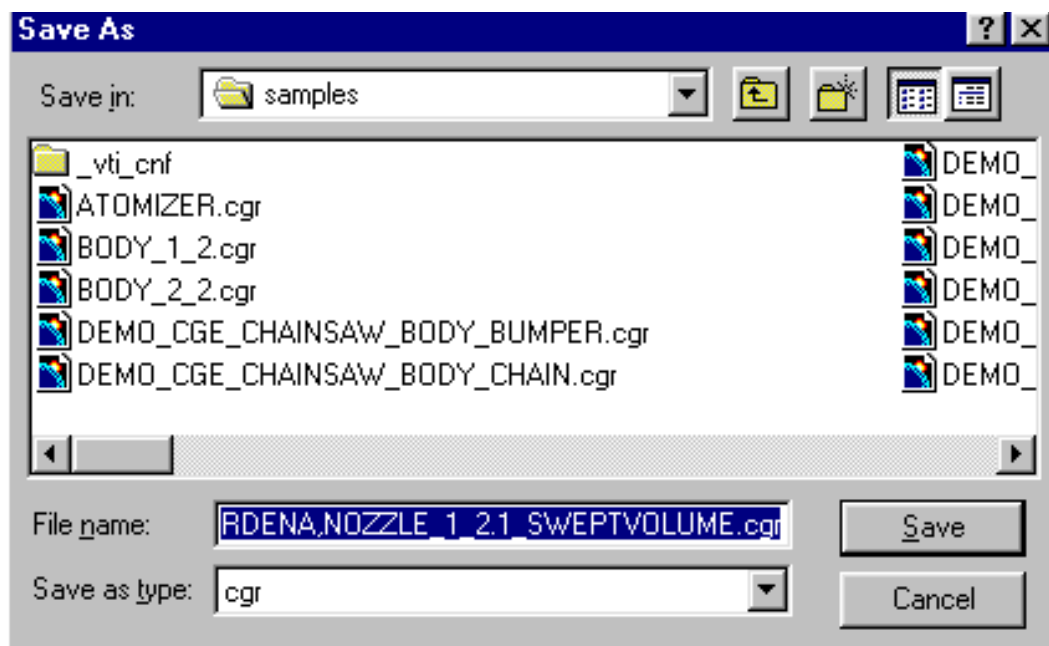
The preview window shows the result:



8. Click **Save** in the Swept Volume dialog box.  
The Save As dialog box automatically appears.

9. Select cgr file from the Save as type drop-down list. When done, click **Save**

**Note** you can now save your swept volume result as a STL model file

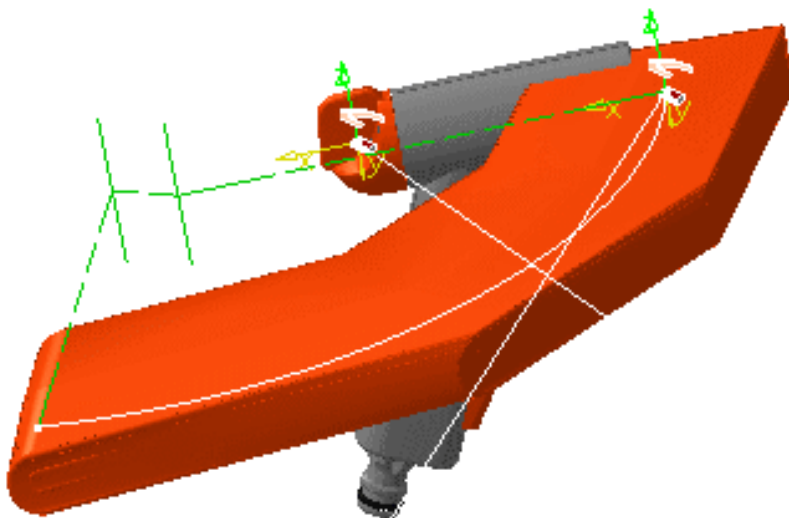


10. Click **Close** when satisfied.

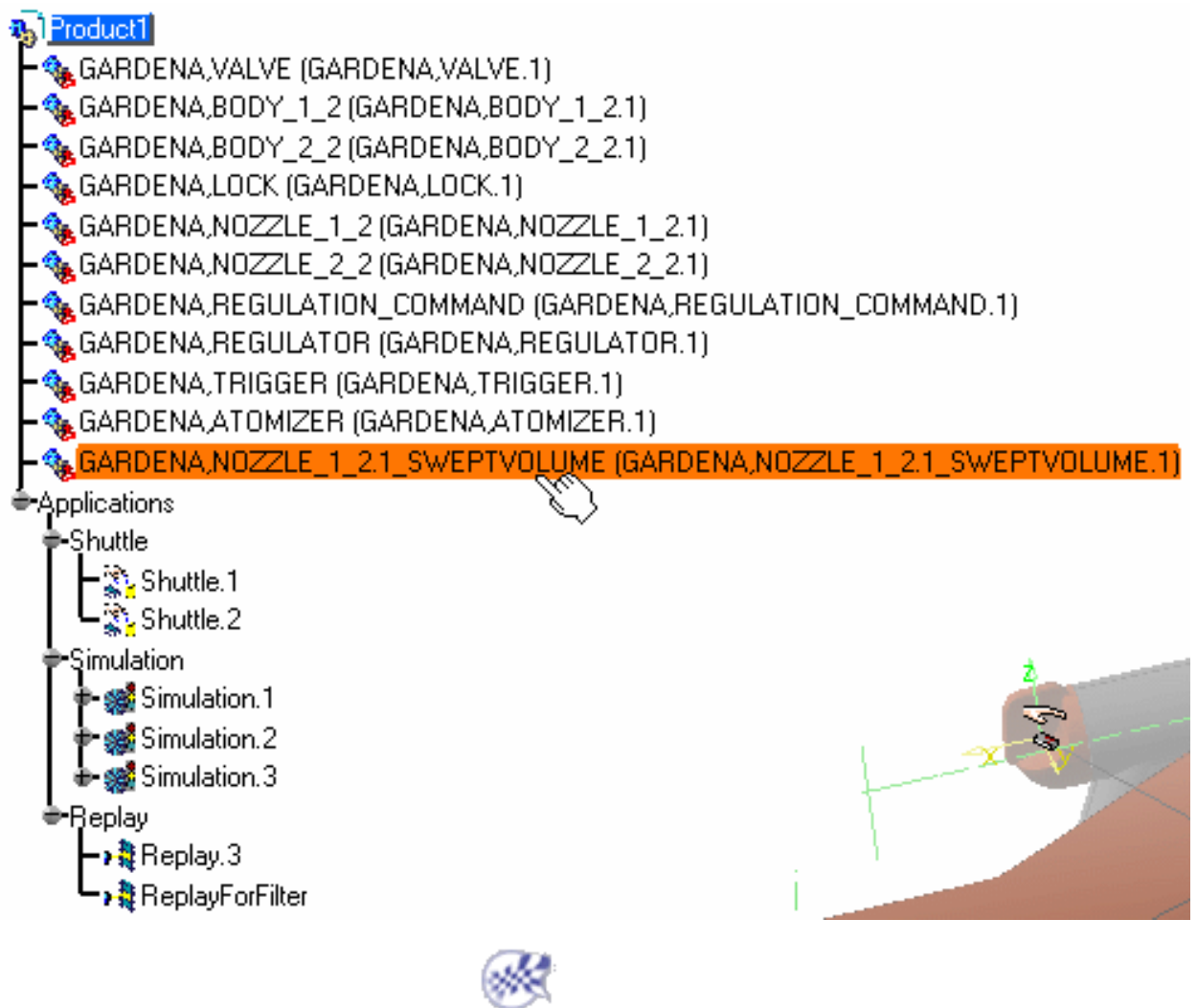
*Now, import the swept volume result into your product*

11. Right-click Product.1 in the specification tree.
12. Select **Components-> Existing component** from the contextual menu displayed.  
Select GARDENA,NOZZLE\_1\_2.1\_SWEPTVOLUME.cgr from the samples folder.


This is what you obtain:





The Swept volume result is identified in the specification tree.



# Defining a Swept Volume from any Moving Reference

 This task shows how to define a swept volume from a moving reference. In other words: the swept volume of a moving part in the axis system of another moving part. It can be very useful if you need to perform clash detection or if you need to calculate the minimum distance between two products moving within the same replay object.

 Open the [FIT\\_SWEPT\\_VOL.CATProduct](#) document.  
You recorded a multi-shuttle simulation and compiled it. You obtained a Replay object. Please refer to [Generating a Replay](#).

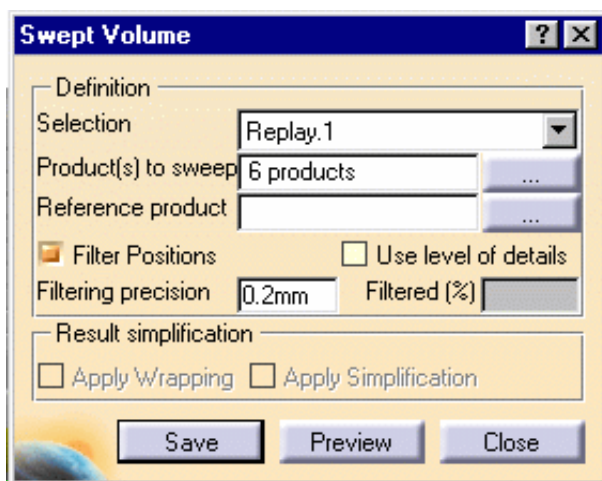
 You need to perform clash analysis between two items. You generated a simple swept volume. Please refer to [Defining a Swept Volume](#). You are not satisfied with the swept volume result as the clash detection is not clear enough.

You need to generate another swept volume using the DEMO\_CGE\_CHAINSAW\_ENGINE\_CLUTCHDRUM.1 as reference product.


1.

Click the Swept Volume icon .

The Swept Volume dialog box is displayed.



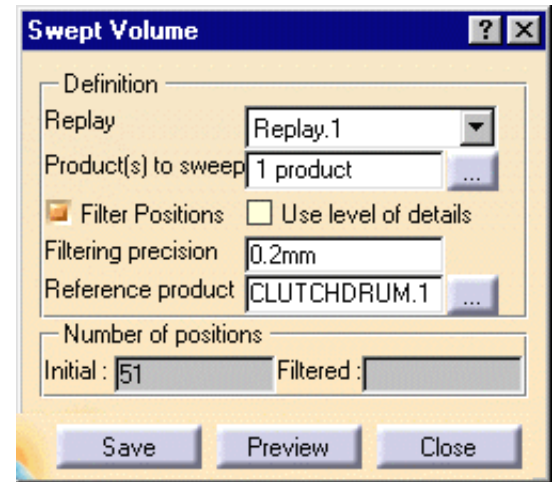
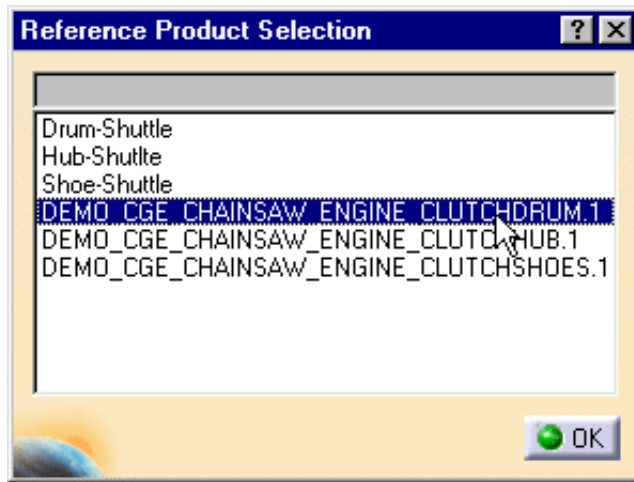
2. Select DEMO\_CGE\_CHAINSAW\_ENGINE\_CLUTCHHUB.1 as product to sweep.

3. In the Reference product field, click  to display the Reference Product Selection dialog box:

4. Select DEMO\_CGE\_CHAINSAW\_ENGINE\_CLUTCHDRUM.1 as reference product.

5. Click **Ok** to confirm your operation.

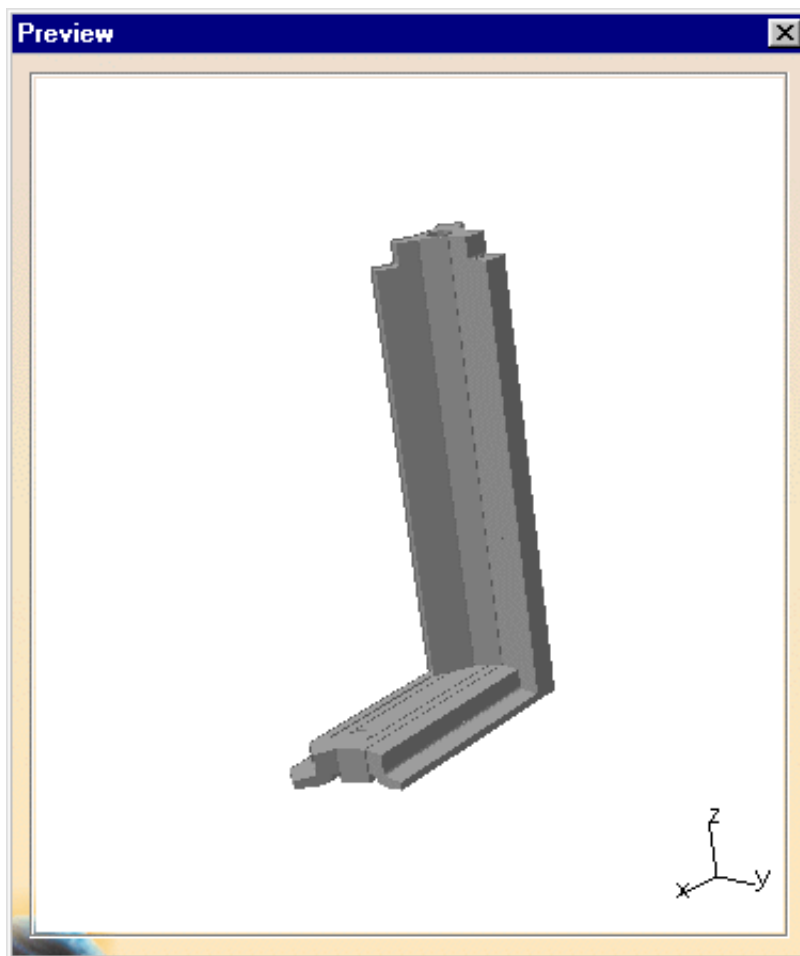
The Swept Volume dialog box is automatically updated.



6. Click **Apply** to generate the swept volume.

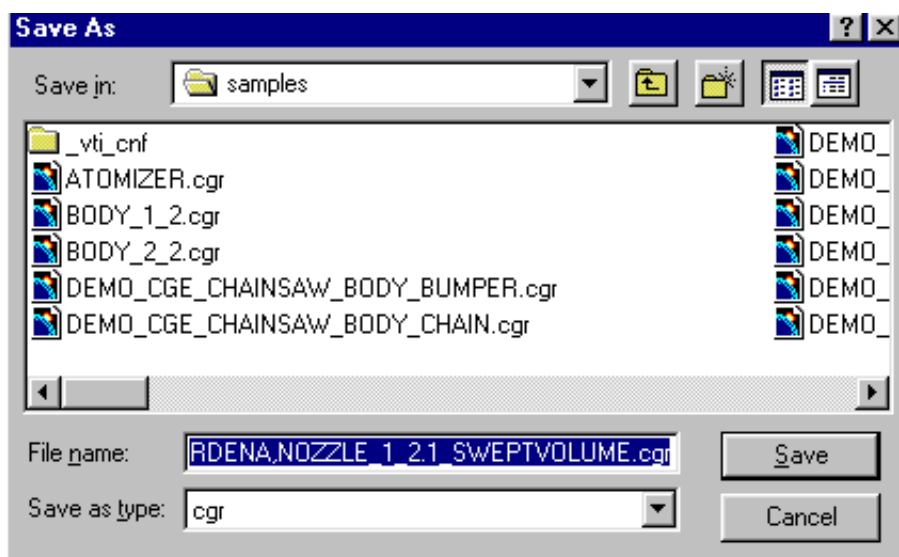
The progress bar is displayed letting you monitor and, if necessary, interrupt (**Cancel** option) the calculation.

The preview window shows the result:



7. Click **OK**.  
The Save As dialog box appears:





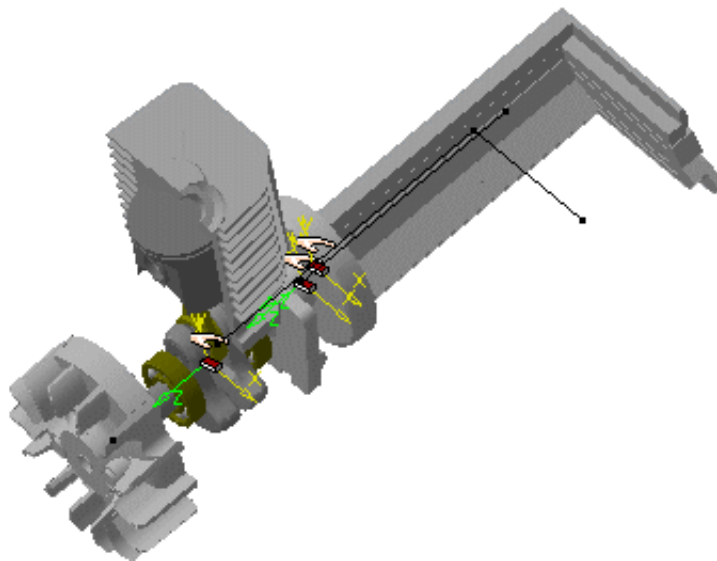
8. Select cgr file from the Save as type drop-down list. When done, click **Save**.

**Note** you can now save your swept volume result as a STL model file

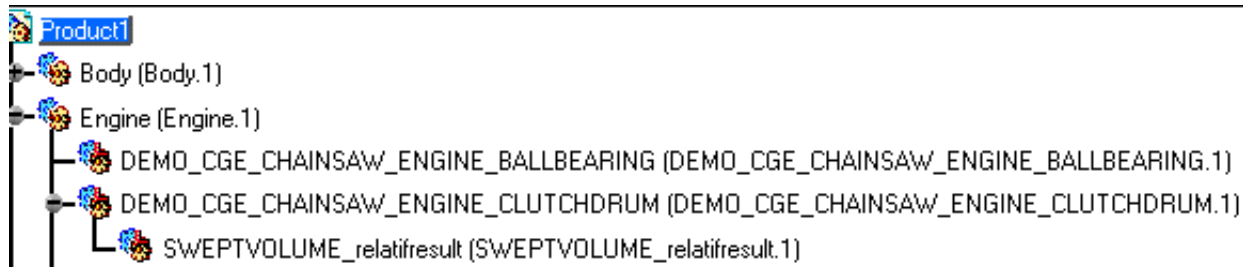
*Now, import the swept volume result into your product .*

9. Right-click DEMO\_CGE\_CHAINSAW\_ENGINE\_CLUTCHDRUM.1 in the specification tree
10. Select **Components->Existing Component** from the contextual menu and select SWEPTVOLUME\_relativeresult.cgr from the samples folder.

This is what you obtain:



The Swept volume result is identified in the specification tree





# More About Swept Volume



## About Filter positions option:

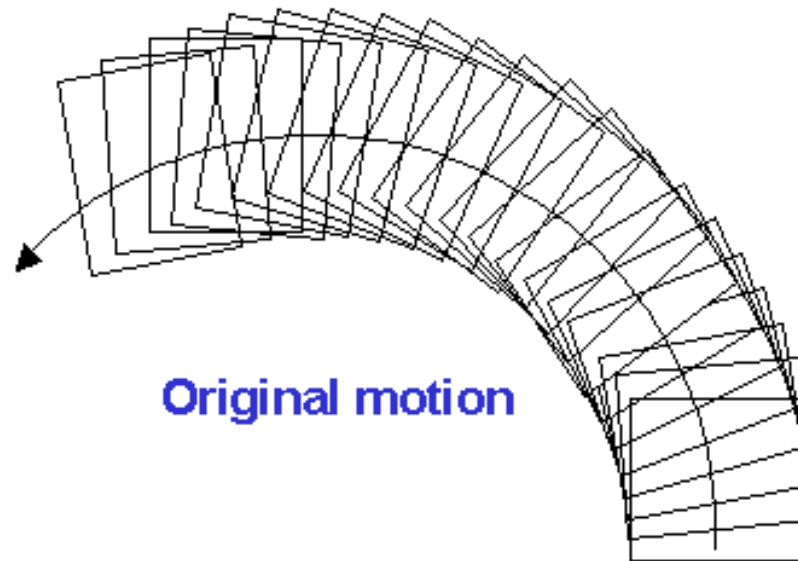
This option can be used to simplify the swept volume computation when the replay object contains many positions or when you know what precision level you need to obtain.

The "filter precision" defines the maximum distance allowed between the simplified trajectory and the initial one  
(= discretization precision)

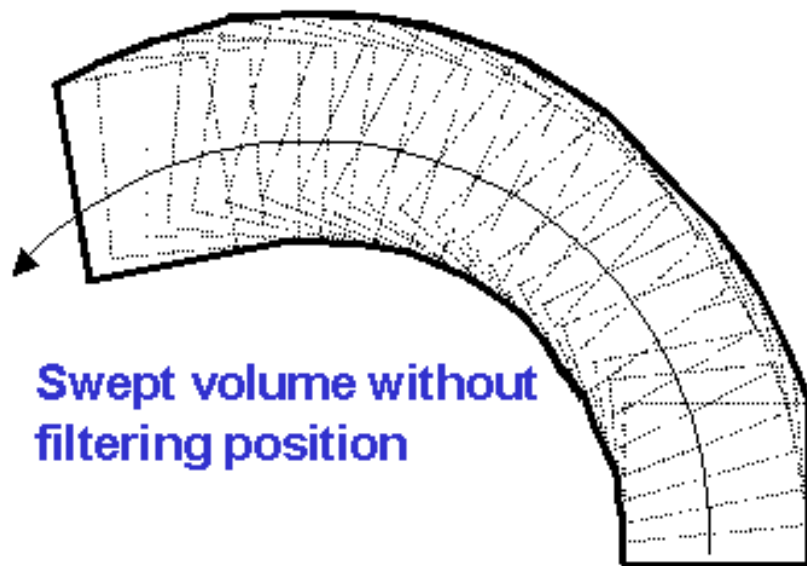


## 1- Filtering swept volume positions

*The following example aims at illustrating the impact of the filter positions option on the final result*

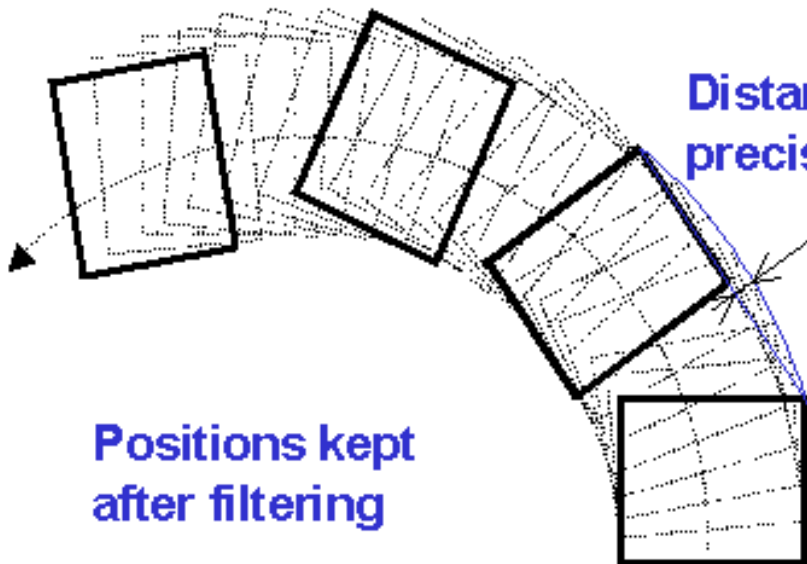


**Case 1:** the Filter positions option is not checked:



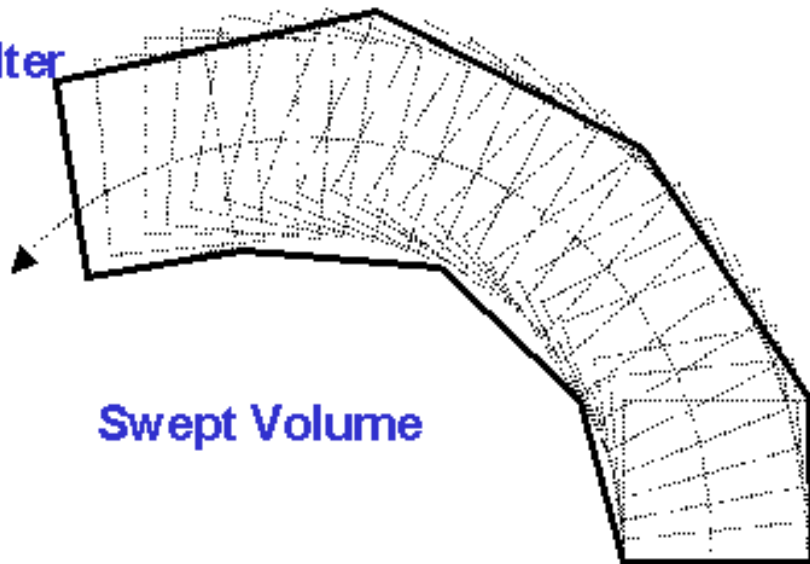
**Swept volume without  
filtering position**

**Case 2:** the Filter positions option is checked  
Filtering precision = 5mm



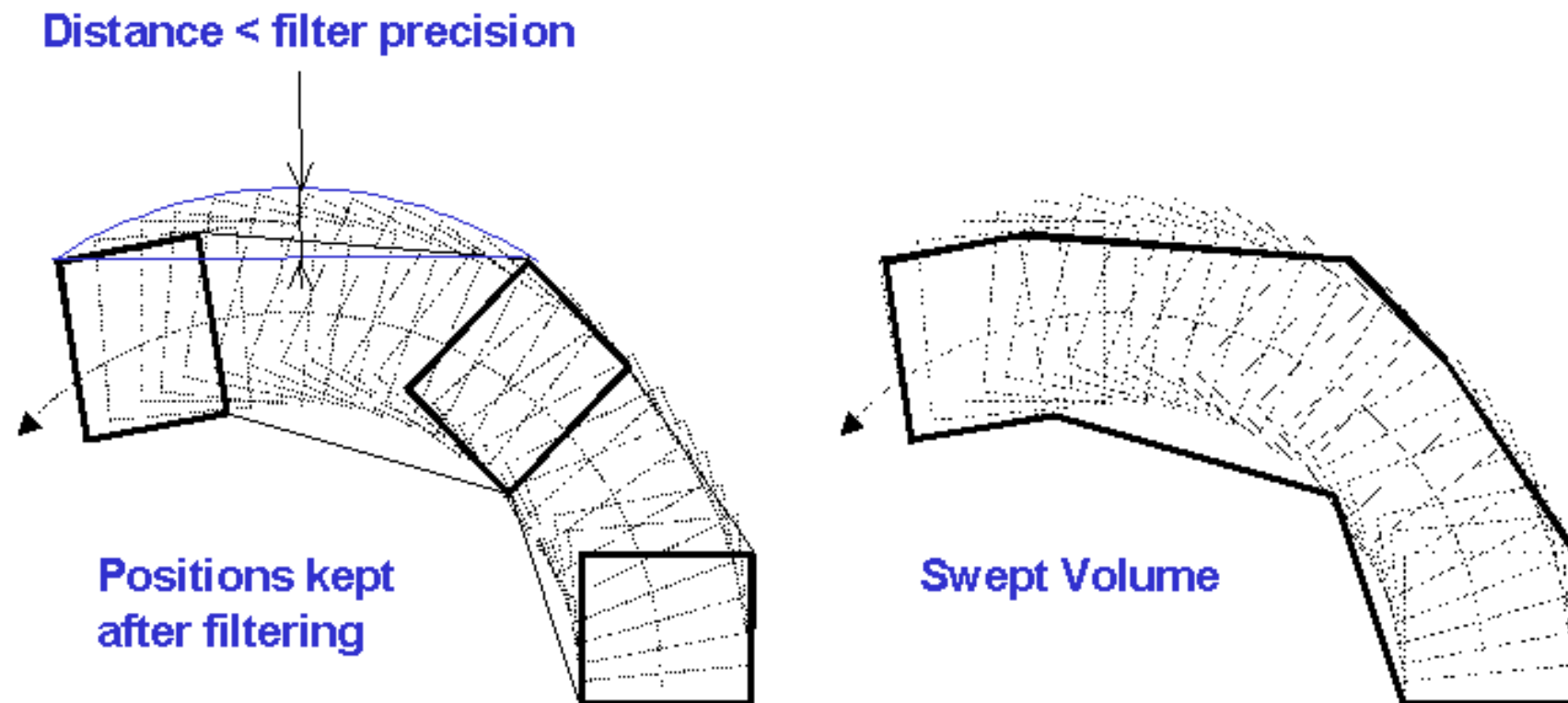
**Positions kept  
after filtering**

**Distance < filter  
precision**



**Swept Volume**

**Case 3:** the Filter positions option is checked  
Filtering precision = 10mm



2- Relative swept volume

**About Relative swept volume:**

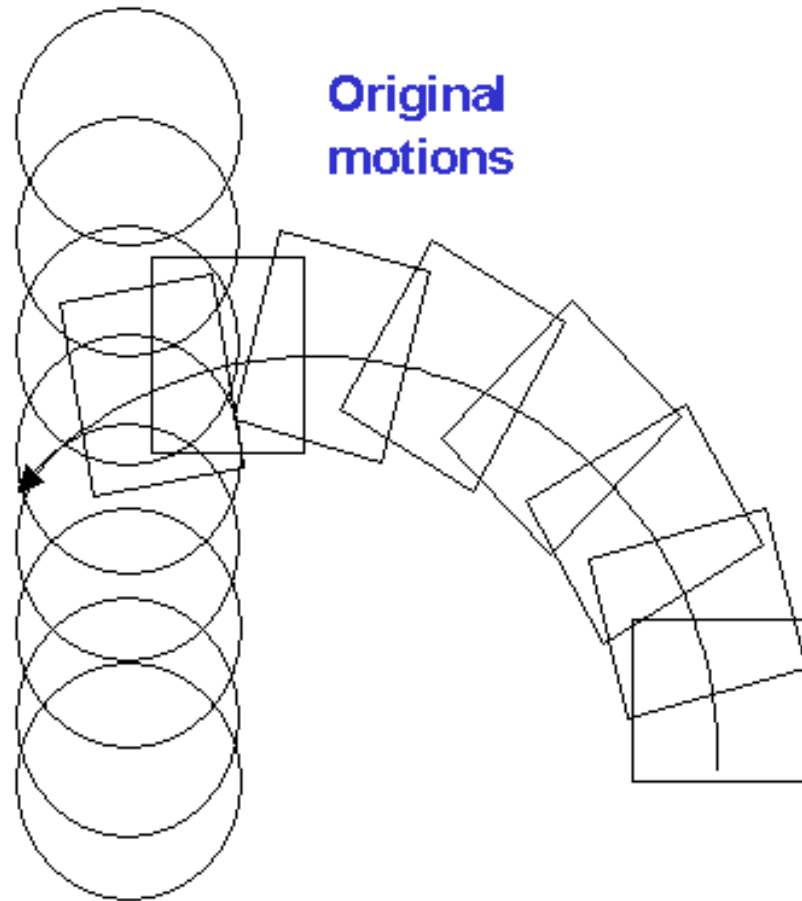
You can compute the swept volume of a moving part in the system axis of another moving part.

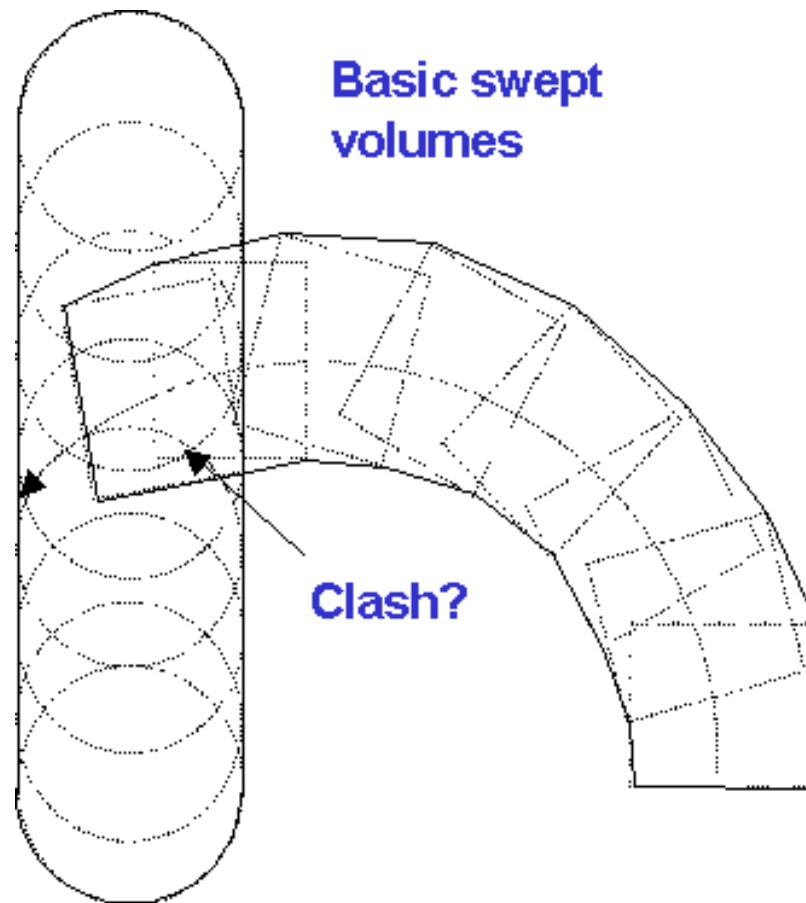
You can use this option when you need to analyze the swept volume of a product versus another product (moving or not)

Example: two moving parts: circle and square

With the basic computation of the swept volumes, the clash analysis is not relevant:

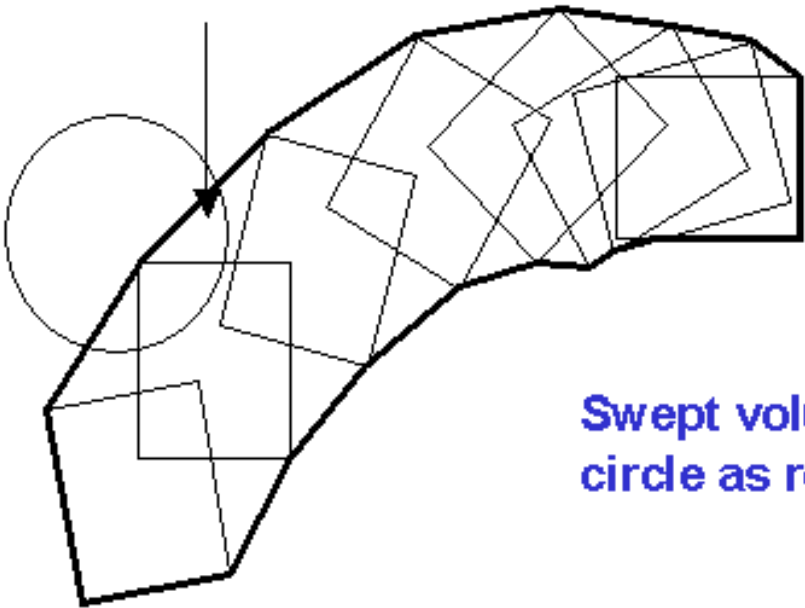
the swept volumes clash but two objects may not be in the same clash area at the same time.





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.



**Swept volume of the square with  
circle as reference product**





# Interoperability

This section provides information on how to use the DMU Fitting product with other Dassault products.

## Interoperability With Catia V4

### How to do a Safe Save in ENOVIA LCA from CATIA V5

# Interoperability With CATIA V4

Write Axis Systems From a V5 track to a V4 Model



## **Licensing**

The following capabilities are available only with a V4I license.

Read a V4 route (track + shuttle) in a V5 Session

# Writing Axis Systems from a V5 Track to a V4 Model



This task shows how to export a V5 track definition in V4. Each V5 recorded position is exported as an Axis in one V4 model document. It is then possible to rebuild a V4 track from these positions.

This functionality is very useful when you are designing products in CATIA V4 and reviewing your DMU in Fitting V5. You can easily retrieve information from your V5 review to perform modifications in CATIA V4.



**Note:** The user needs to rebuild a route in V4 from the exported axis systems.



Open the [EXPORT\\_V5\\_TRACK.CATProduct](#) document.



1. Double-click Track.2 in the specification tree.

2.



Click the arrow within the Clash Detection icon from the DMU Check toolbar. Undock the toolbar if necessary.

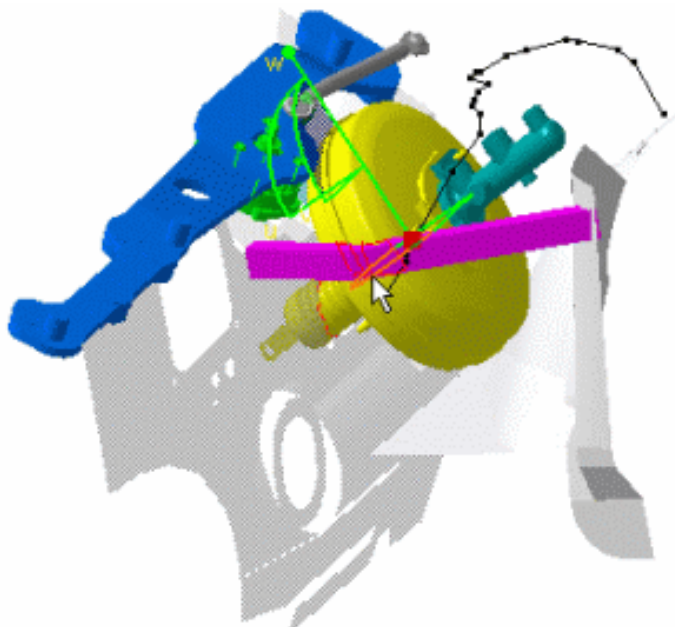


3.

Set the Clash detection to on

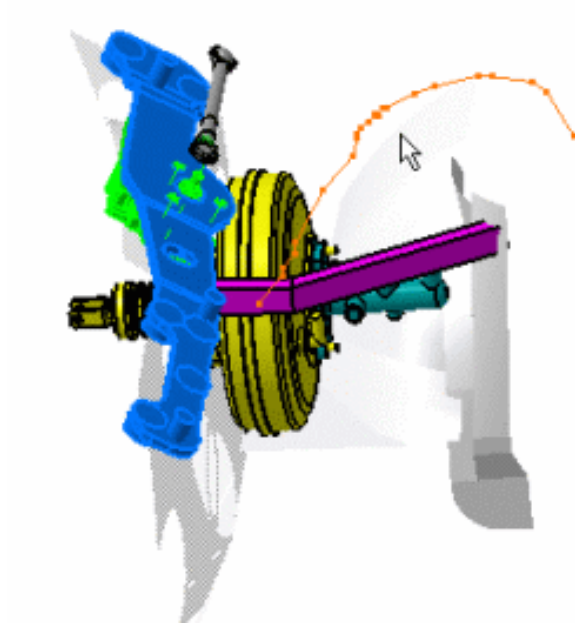


4. Play your track simulation using the Player

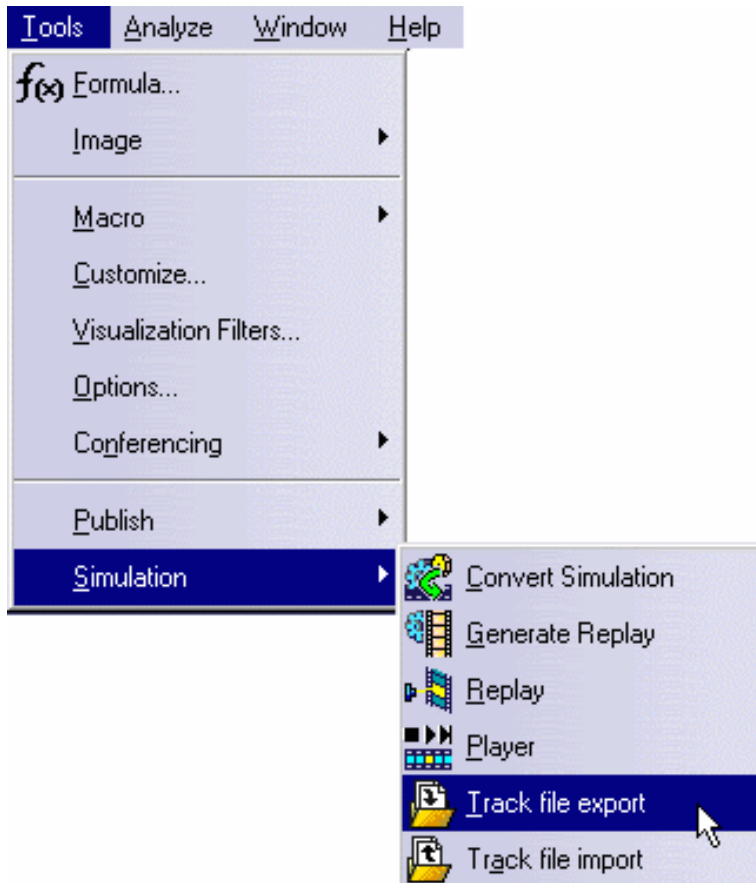


5. There is a collision detected at recorded position 3. You need to modify your design.

6. Select the track.2 either in the geometry area or in the specification tree .

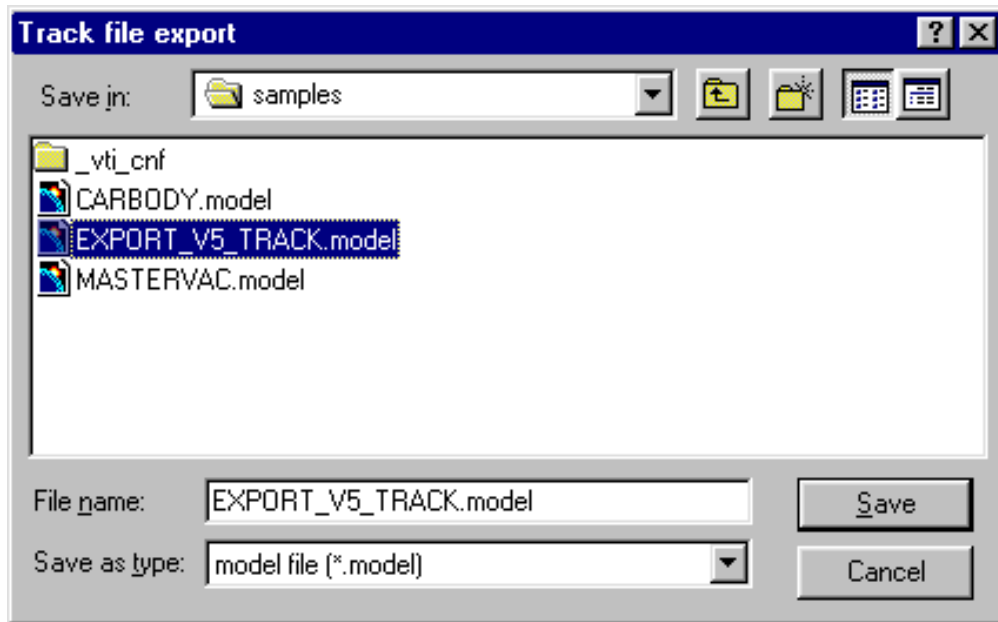


7. Select **Tools->Simulation->Track File Export**



The Track File Export dialog box automatically appears:

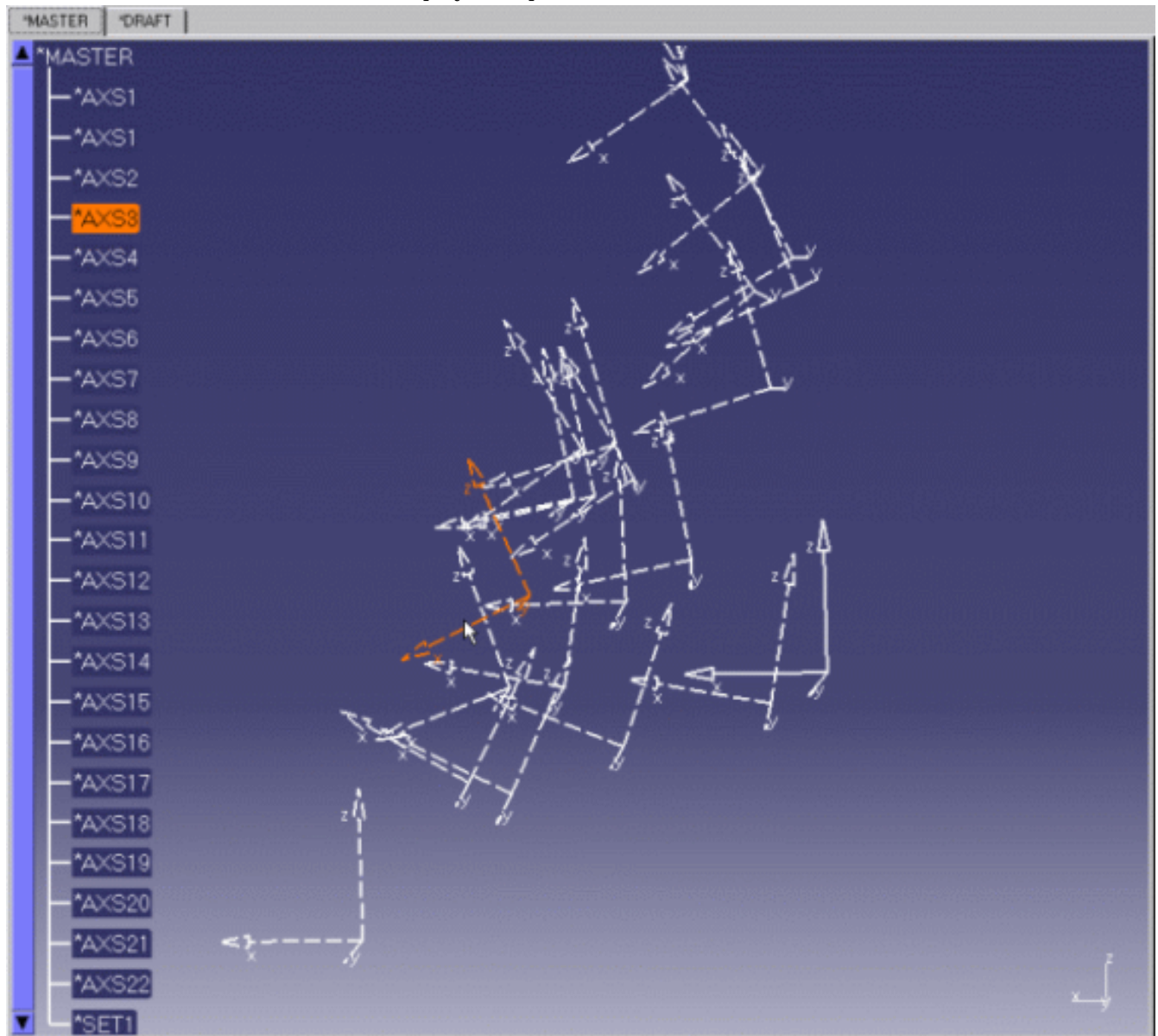
8. Enter a meaningful name and select .model format from the Save as Type drop-down list:



9. Open the [EXPORT\\_V5\\_TRACK.model](#) document you have just created (from cfyug/samples)

This is what you obtain:

**Note:** Double-click Master node to display the specification tree





- 10.** In this V4 model document we selected axis system 3 (highlighted) which corresponds in the dismounting review to the position 3 where a collision was detected.

You are now ready to modify your design using this information.



### **Remember though:**

There are not any V4 Fitting entities created.

- V4 tracks do not exactly share the same behaviors with V5 tracks (such as interpolation, etc.)



# Reading V4 Routes and Sequences in a V5 Session



This task shows how to read a V4 route (track + shuttle) as well as a sequence in DMU Fitting V5.

When inserting a V4 model in V5 product, if the model contains V4 routes and shuttles, the routes and shuttles will be read. Similarly, sequences will also be read.

They will be selectable in the navigation tree under the \*MASTER container.

A copy/paste operation from the V4 Master container to the V5 Tracks container will convert the V4 Routes into V5 tracks. Similarly, sequences will convert to V5 sequences.

A V4 Route is the association of: 1 Shuttle, 1 Home position and n Trajectories.

A V5 Track is the association of: 1 Shuttle or 1 Product, and 1 Trajectory.

The conversion will concatenate the positions in the V4 Home and Trajectories to generate a single V5 path.



Insert the following .model files in the samples folder:

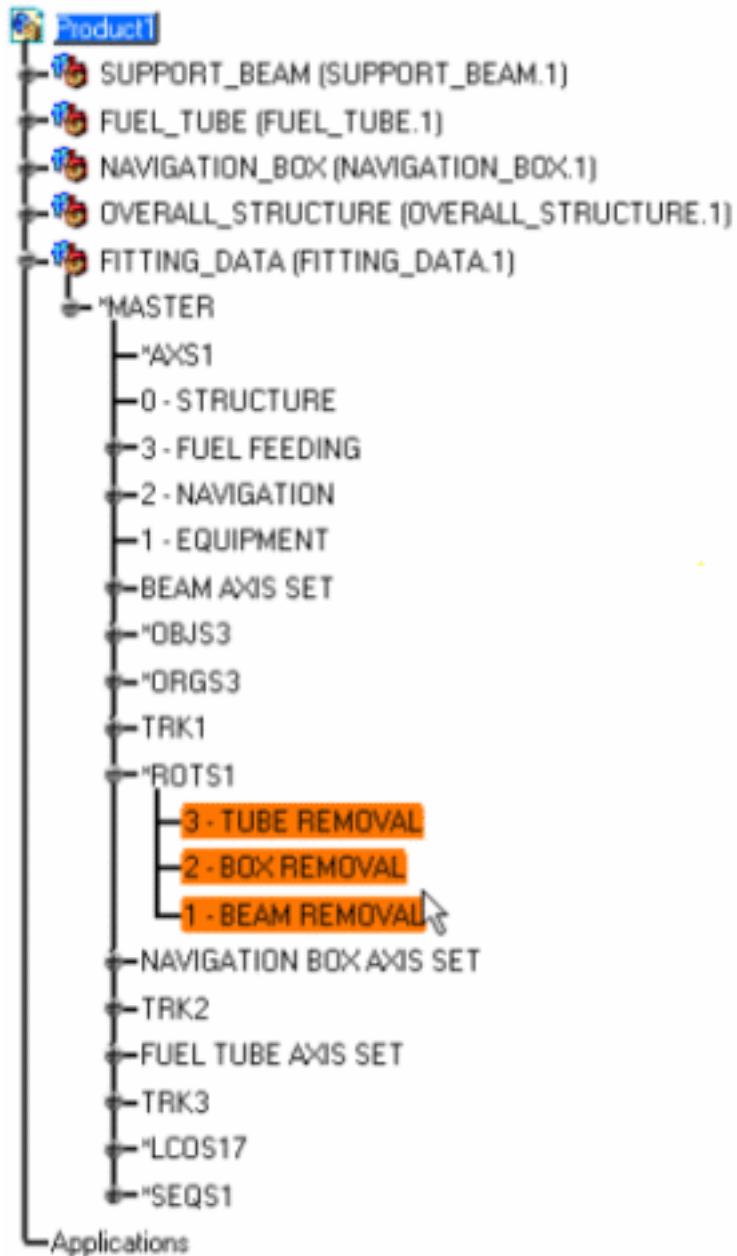
- FITTING\_DATA.model
- FUEL\_TUBE.model
- NAVIGATION\_BOX.model
- OVERALL\_STRUCTURE.model
- SUPPORT\_BEAM.model



1. In the specification tree where the Version 4 model is displayed, select the item you wish to copy into the Fitting Simulator Version 5.

In our example, expand FITTING\_DATA and \*MASTER items and multi-select.

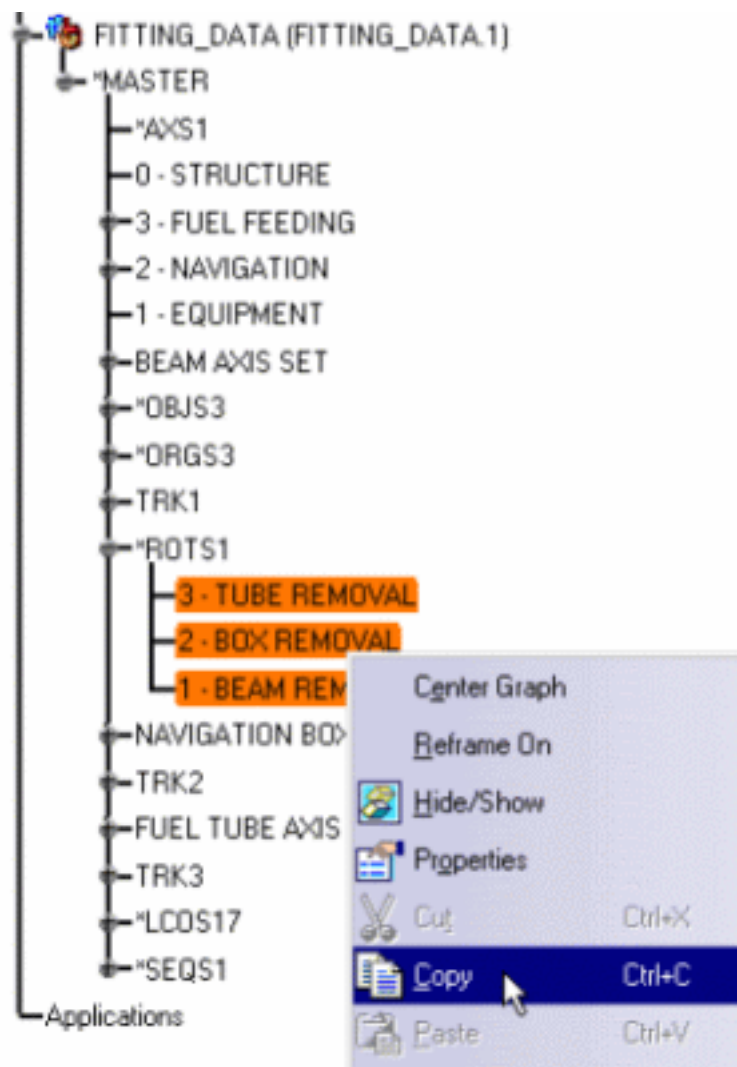
3-TUBE REMOVAL  
2-BOX REMOVAL  
1-BEAM REMOVAL



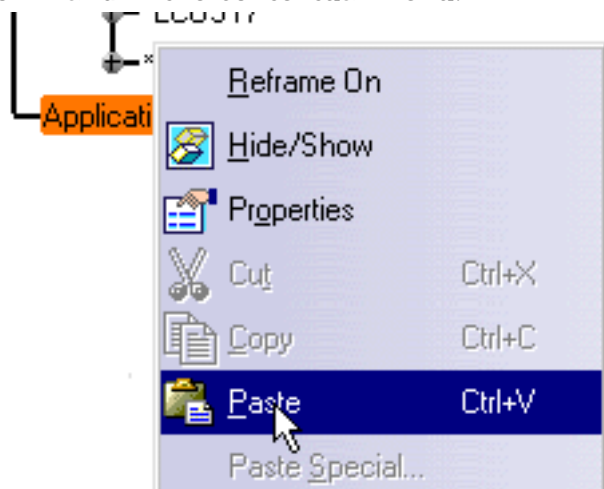
2. Put the data you have selected in the clipboard. To do this, either click the Copy icon, select the **Edit->Copy** command or select the **Copy** command in the contextual menu.

To obtain the sequences, copy the items under the SEQs1 node.

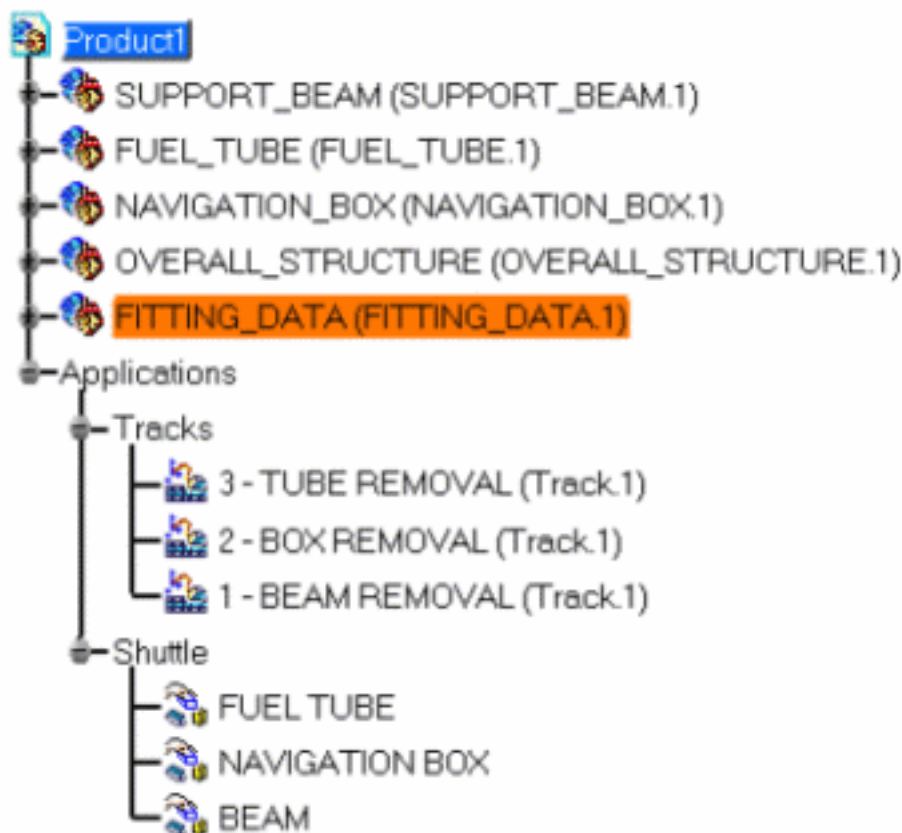




3. Select Application in the specification tree.
4. Now either click the Paste icon, select the **Edit->Paste** command or select the **Paste** command in the contextual menu.



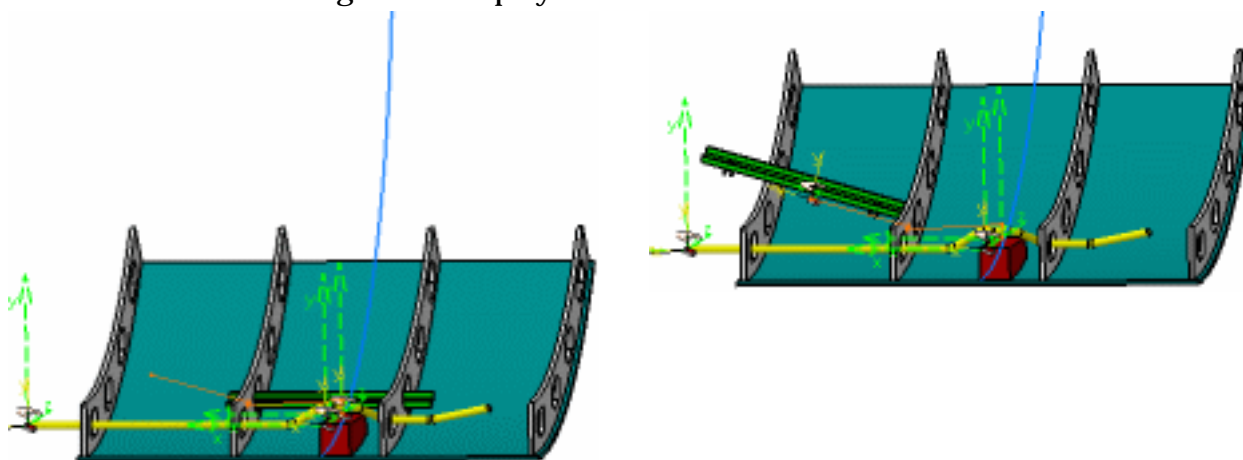
This operation recovers the data previously put in the clipboard  
The result appears as shown below:



5. Open the [READ\\_V4\\_ROUTE.CATProduct](#) to check your result
6. For instance, select 1- BEAM REMOVAL in the specification tree and click the Player icon



Simulate the track using the DMU player buttons:



**Note:** the V4 model referenced by the V4 shuttle must not be renamed. Otherwise, it may create broken links, which would disable the V4/V5 conversion.





# How to do a Safe Save into ENOVIA LCA from CATIA V5



The objective of Safe Save is to prevent the user from building / editing data in CATIA V5 if they cannot be saved in ENOVIA LCA. Therefore, in interoperability mode, some CATIA V5 commands are grayed out / hidden in the DMU Fitting Simulator workbench.

Only commands subject to restrictions are listed below. And in some cases, rules are applied to restricted commands.

Workbench	Feature	Command	Accessibility in LCA mode	Warning / Comment	Save in LCA / Rules
DMU FITTING	DMU Simulation	Track	NO (grayed out)		
		Color Action	NO (grayed out)		
		Edit Sequence	NO (grayed out)		
		Edit and Perform Experiment	NO (grayed out)		
		Simulation Player	NO (grayed out)		
		Swept Volume	NO (grayed out)		
		Explode	NO (grayed out)		
		Shuttle	NO (grayed out)		
		Reset position	YES		
		Path finder	YES		
		Smooth	YES		

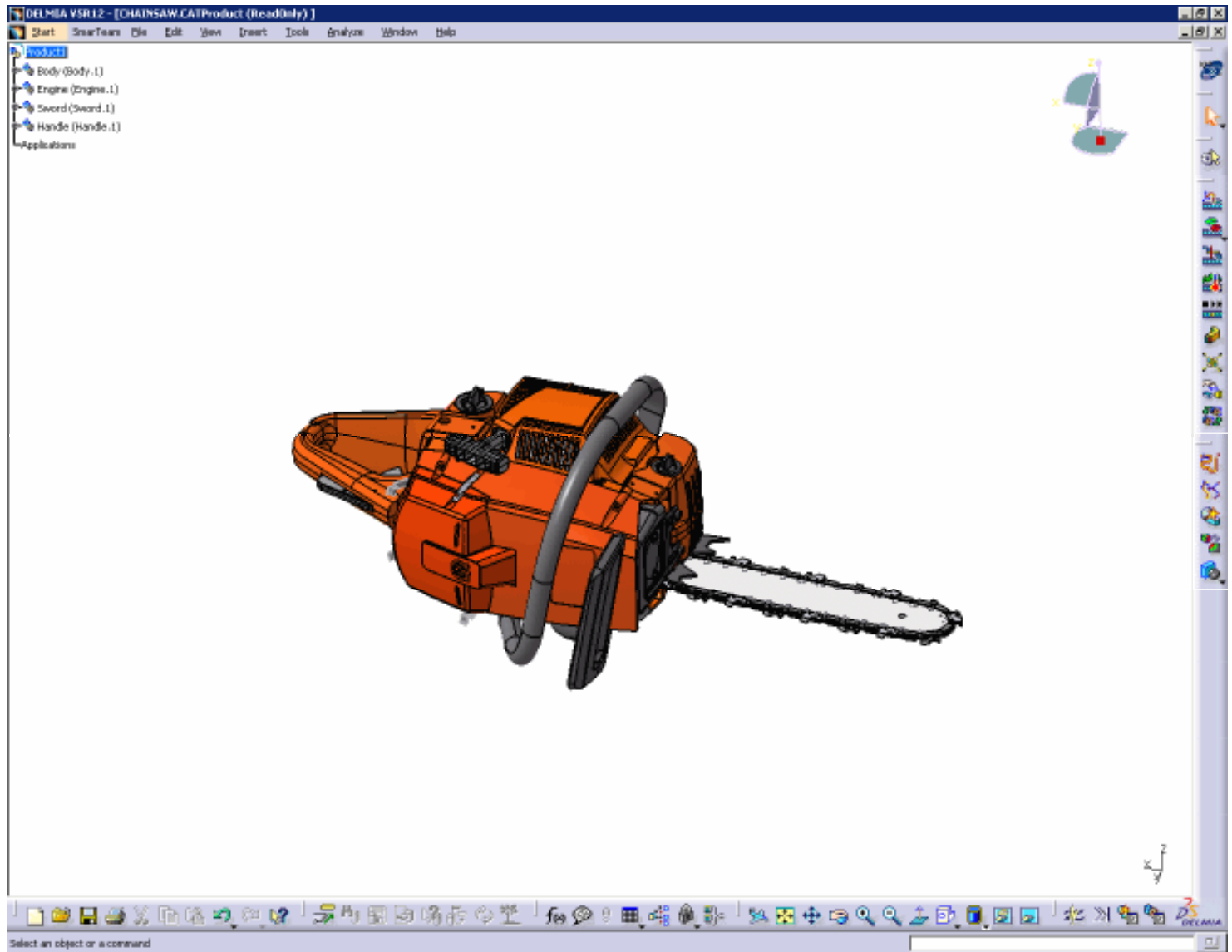
G	DMU Check	Clash	YES	Clash results cannot be saved unless the Retrieve Information from PDM option is set in the DMU Clash tab page ( <b>Tools - &gt; Options - &gt; Digital Mockup - &gt; DMU Space Analysis</b> ).	See <i>DMU Space Analysis User's Guide</i> for more information.
		Distance and Band Analysis	YES	Distance data is not saved; there is a warning displayed.	
		Clash Detection	YES		
G E N E R I C  C O M M A N D S	Generic commands (Generic tools)	Publish	YES		
		Camera	YES		

# Workbench Description

This section contains the description of the icons and menus which are specific to the DMU Fitting Simulator workbench.

The DMU Fitting Simulator window is shown below.

Click the sensitive areas (toolbars) to see the related documentation.



[Menu Bar](#)  
[Simulation Toolbar](#)  
[DMU Check Toolbar](#)  
[Manipulation Toolbar](#)  
[Automatic Clash Detection Toolbar](#)  
[Specification Tree](#)  
[Recorder](#)  
[Player](#)

# DMU Fitting Simulator Menu Bar

Here we will present the various menus and menu commands that are specific to DMU Fitting Simulator version 5.



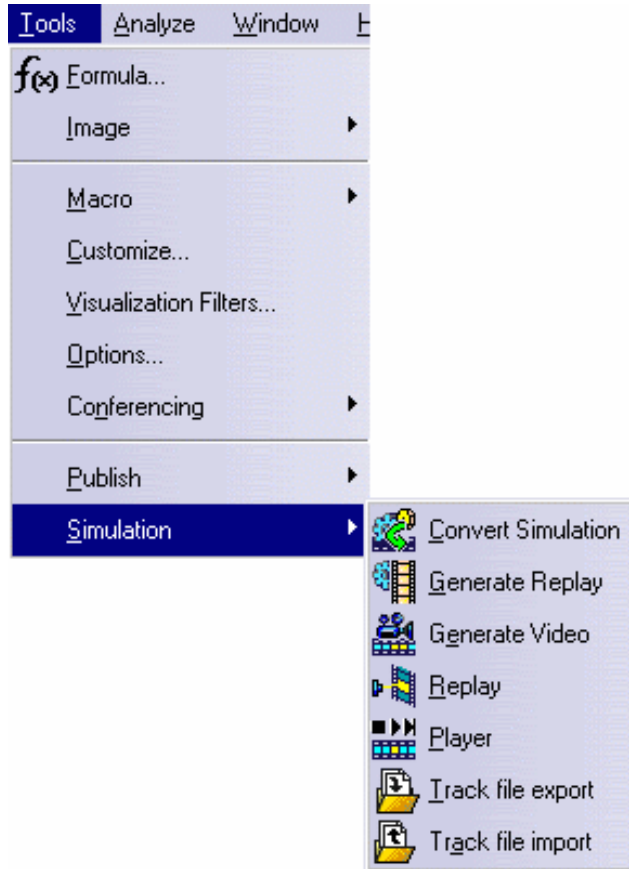
## Insert



For...	See...
<b>Sequences &amp; actions</b>	<a href="#">Measuring Distances</a>
<b>track</b>	<a href="#">Detecting Interferences</a>
<b>color Action</b>	<a href="#">Defining/Editing Several Shuttles</a>
<b>Visibility Action</b>	
<b>Edit Sequence</b>	<a href="#">Moving and Recording a Multi-Shuttle Simulation</a>
<b>Shuttle</b>	<a href="#">Entering in the DMU Navigator and Selecting Models</a>
<b>Simulation</b>	
<b>Clash</b>	
<b>Distance and band Analysis</b>	
<b>Group</b>	

## Tools

For...	See...
<b>Convert Simulation</b>	<a href="#">Measuring Distances</a>
<b>Generate Video</b>	<a href="#">Using DMU Video Generation Tool</a>
<b>Generate Replay</b>	<a href="#">Detecting Interferences</a>
<b>Replay</b>	<a href="#">Defining/Editing Several Shuttles</a>
<b>Player</b>	<a href="#">DMU Player</a>



Track file export

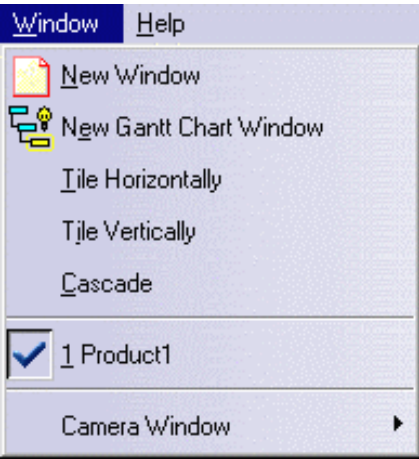
[Exporting and Importing Tracks in Neutral Format](#)

[Writing Axis Systems From a V5 track to a V4 Model](#)

Track file import

[Exporting and Importing Tracks in Neutral Format](#)

# Window



For...

See...

**new Gantt Chart Window** [Displaying Gantt Chart](#)

**Camera Window**

See [Using Camera Capabilities in DMU Navigator User's Guide](#)

Tasks corresponding to the general menu commands are described in the *Infrastructure User's Guide*.

# DMU Simulation Toolbar

The DMU Simulation toolbar contains a number of tools that are useful for DMU Fitting Simulator.



**Other Actions toolbar:** is displayed when you click the arrow within the Color action icon and undock it



See: [Tracks](#)



See: [Defining Sequences](#)



See: [About Sequence Capabilities](#)

See: [Sequences](#)



See [About Sequence Capabilities](#)

See: [Sequence Editor](#)

See: [Defining a Sequence](#)



See: [Performing Experiments and Generating Simulation Reports](#)



See [DMU Player](#)



See [Defining a Swept Volume](#)

See [Defining a Swept volume from a Moving Reference](#)



See [Exploding](#)



See [Shuttle](#)



See: [Exploding a Constrained Assembly](#)

See: [Copying and Pasting Tracks](#)







# DMU Check Toolbar



## Licensing

The following functionalities: "Clash"  and "Distance and Band Analysis"  are available only with a DMU Space Analysis license.



See [Path finder](#)



See [Using the Smooth Command](#)



See [Detecting Interferences](#)



See [Measuring Distances](#)



See [Detecting Clashes Automatically](#)

# Manipulation Toolbar

The Manipulation toolbar contains a number of tools that are useful for DMU Fitting Simulator. It displays when using the Shuttle, Explode, Track and Simulation commands (if based on one or various shuttle(s)).



Track-based manipulation toolbar



Shuttle-based manipulation toolbar



For more detailed information about these commands, please refer to:

[More About Manipulation Toolbar](#)



See [Using the Smart Target](#)

See [Target and Snap Capabilities](#)



See [Using the Smart Snap](#)



See [Moving the Shuttle](#)



See [Moving the Shuttle](#)



See [Snapping Objects in Approach Phase](#)

See [Editing a Shuttle](#)



See [Moving the Shuttle](#)



See [Moving the Shuttle](#)

# Automatic Clash Detection Toolbar

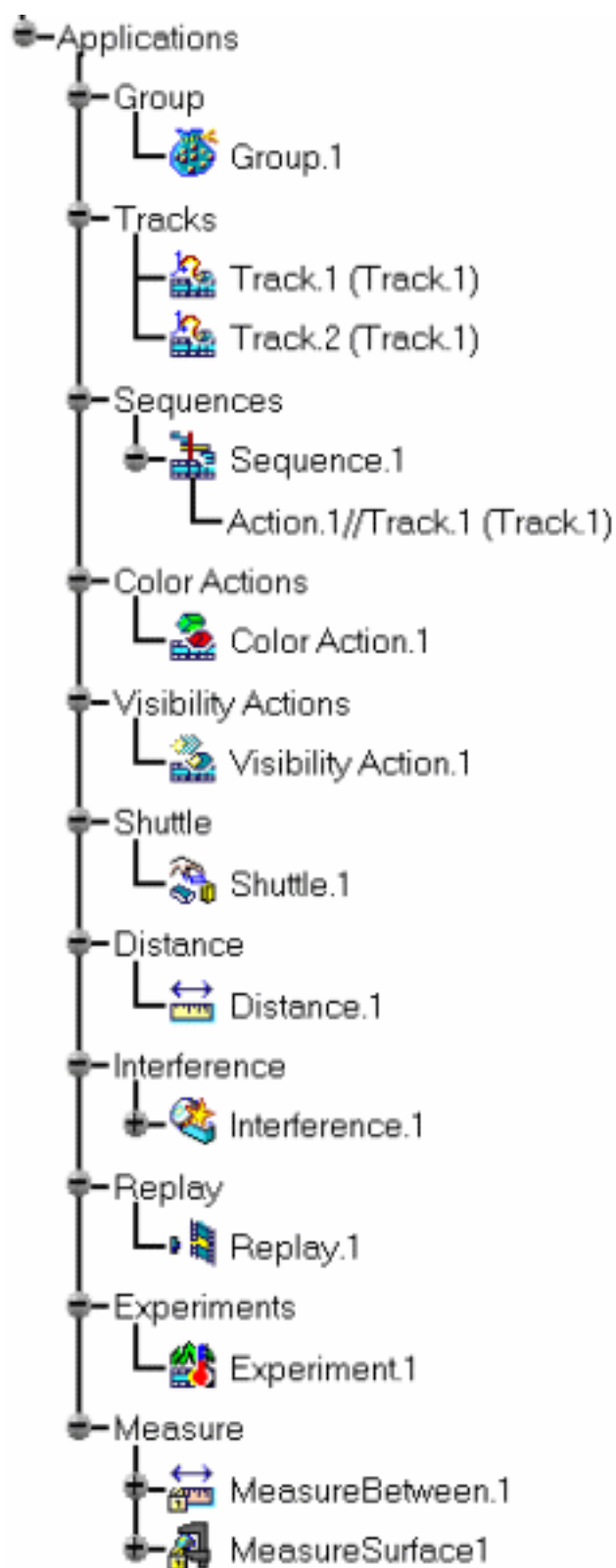


See [Validating Positions Automatically](#)



# Specification Tree

Within the DMU Fitting Simulator workbench, you can generate a number of features that are identified in the specification tree by the following icons:



Icons displayed in the specification tree and specific to the DMU Fitting Simulator workbench identify:



Color actions



Tracks



Sequences



Visibility actions



Experiments



Shuttles



Simulations



Replays



Groups



Clash entries

Distance and band analysis entries



Measures made using the Measure Item command



Measures made using the Measure Between command

For standard specification tree symbols, see the *Infrastructure User's Guide*.

# Recorder Toolbar



See: [Track Editor and Recorder](#)



See: [Recording a Track](#)



See: [Track Editor and Recorder](#)



See: [Track Editor and Recorder](#)

# Player



See [Player](#)



# Customizing for DMU Fitting Simulator

[Customizing DMU Fitting Settings](#)  
[Customizing DMU Manipulation Settings](#)  
[Customizing DMU Navigator Settings](#)

# Customizing DMU Fitting Settings



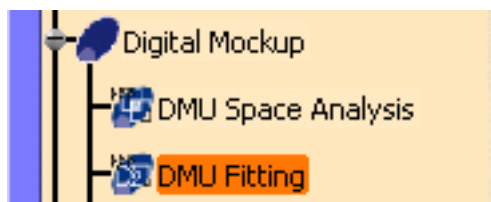
This task explains how to customize DMU Fitting settings.



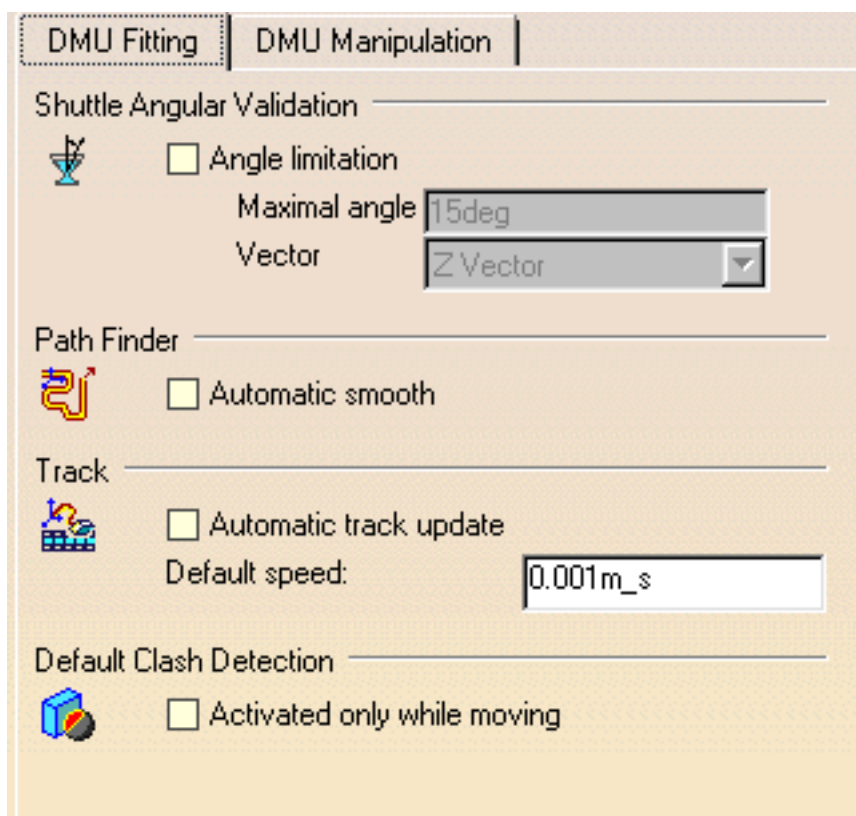
1. Select **Tools** -> **Options** from the menu bar:

The Options dialog box appears.

2. Click Digital Mockup in the left-hand frame, then DMU Fitting.



The DMU Fitting tab is active by default. If it is not on top, select it. The default values are shown below.



## Shuttle Angular Validation

Checking the box enables you set the angular shuttle validation option in the Edit Shuttle dialog box.

You can specify a maximum rotation angle for the shuttle and the vector around which the shuttle will rotate.

## Path Finder

**Automatic Smooth:** If checked, smooth is performed within the pathfinder command once you click the OK button.

## Track

**Automatic track update:** if checked, the track is automatically updated after modification.

The manual update is set by default (see the track contextual menu).

Default speed is 0.001m\_s. You can modify this value at any time. When you modify the default speed, the modification applies to tracks created after the modification. To modify the speed on existing tracks, use the Properties dialog box.

## Default Clash Detection

When activating the default clash detection, the clash computation can take several minutes before the simulation dialog boxes are displayed.

Sometimes it is more convenient not to compute this clash detection for the first step.

This step can already be validated by a static clash analysis.


- **Activated only while moving:** if checked, no calculation will be performed when the default clash detection is **activated** (ON, STOP). When running tracks, sequencsm, and simulations, clashes are still calculated with respect to the default clash detection state (OFF, ON, STOP)
- By default this option is not checked


3. Click **OK** in the dialog box when done.



[Up](#)   [Next](#)

# Customizing DMU Manipulation Settings

 This task explains how to customize DMU Fitting settings.

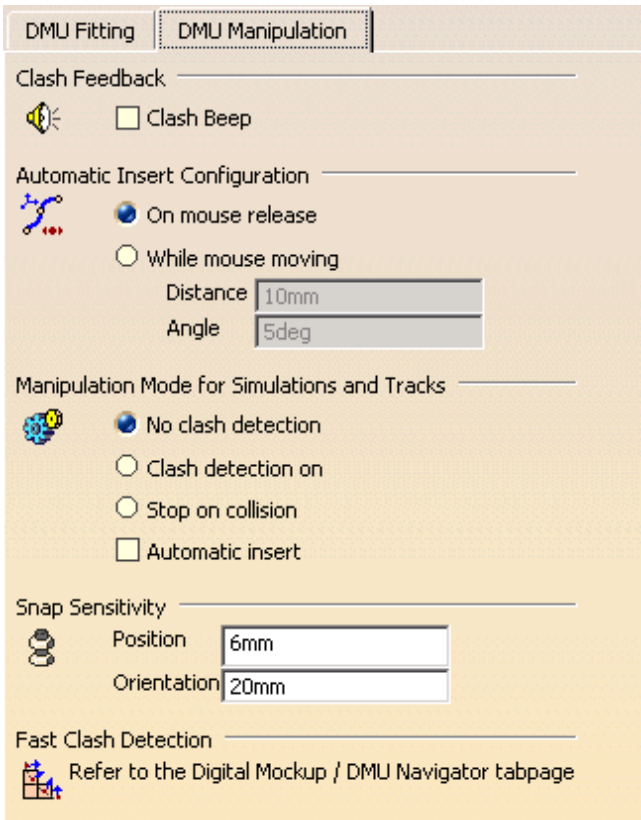
-  1. Select **Tools** -> **Options** from the menu bar:

The Options dialog box appears.

2. Click Digital Mockup in the left-hand frame, then DMU Fitting.



3. Select the DMU Manipulation tab. The default values are shown below.



## Clash Feedback


Click the Clash Beep checkbox to activate the Beep during interference analyses, simulation recordings, etc.

By default, the Clash Beep option is unchecked.

## Automatic Insert Configurations

- On mouse release (default): Lets you record positions on mouse release.
- While mouse moving:
  - Distance: Specifies the minimum distance step during simulation recording .
  - Angle: Specifies the maximum rotation angle step during simulation recording.




The Distance and Angle fields are only grayed out when **On mouse release** is checked. Once **While mouse moving** is checked, you can enter the appropriate values to record the the motion.

 Regardless of which configuration you have selected, you will not get Automatic Insert unless the Automatic Insert option is checked in Manipulation Mode, described in the section below.

## Manipulation Mode for Simulations and Tracks

Lets you validate the shuttle simulation detecting clashes.

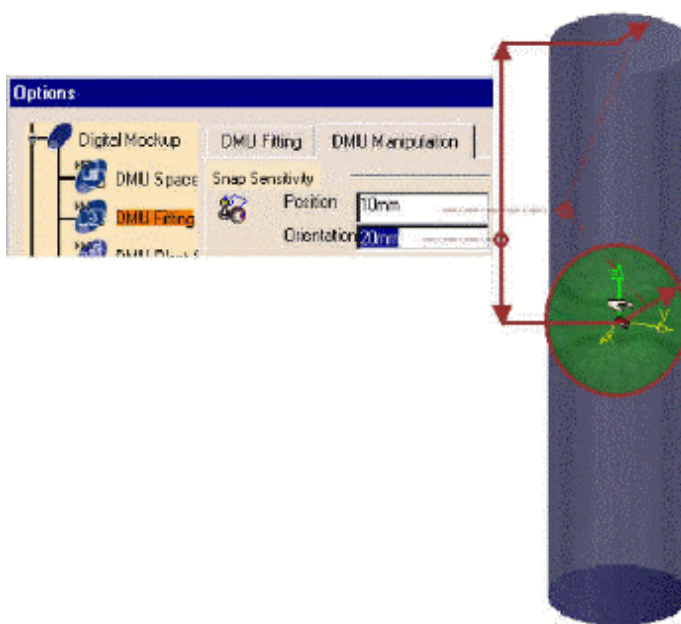
This setting is linked to the Automatic Clash Detection toolbar. The icon changes automatically with respect to the option checked.

- No clash detection (default) : Clash detection will not occur.
- Clash detection on : Activates the clash detection mode  
(The objects in clash are highlighted in the geometry area while recording the simulation )
- Stop on collision : Activates the stop mode. Consequently, the simulation stops once any objects clash. The objects in clash are highlighted in the geometry area.
- Automatic insert: When checked, it enables automatic insert mode. Automatic insert mode enable users to insert shots in a track without using the Record (Insert) command on the Recorder toolbar. The two configurations available for Automatic Insert appear above.

## Snap Sensitivity

Lets you customize the snap functionality parameters within a sphere and a cylinder:

- Position value: Specifies the sphere radius (first area) where the shuttle is snapped onto the object.
- Orientation value: Specifies the cylinder height (second area) where the shuttle is snapped onto the object.



3. Click **OK** in the dialog box when done.



# Customizing DMU Navigator Settings



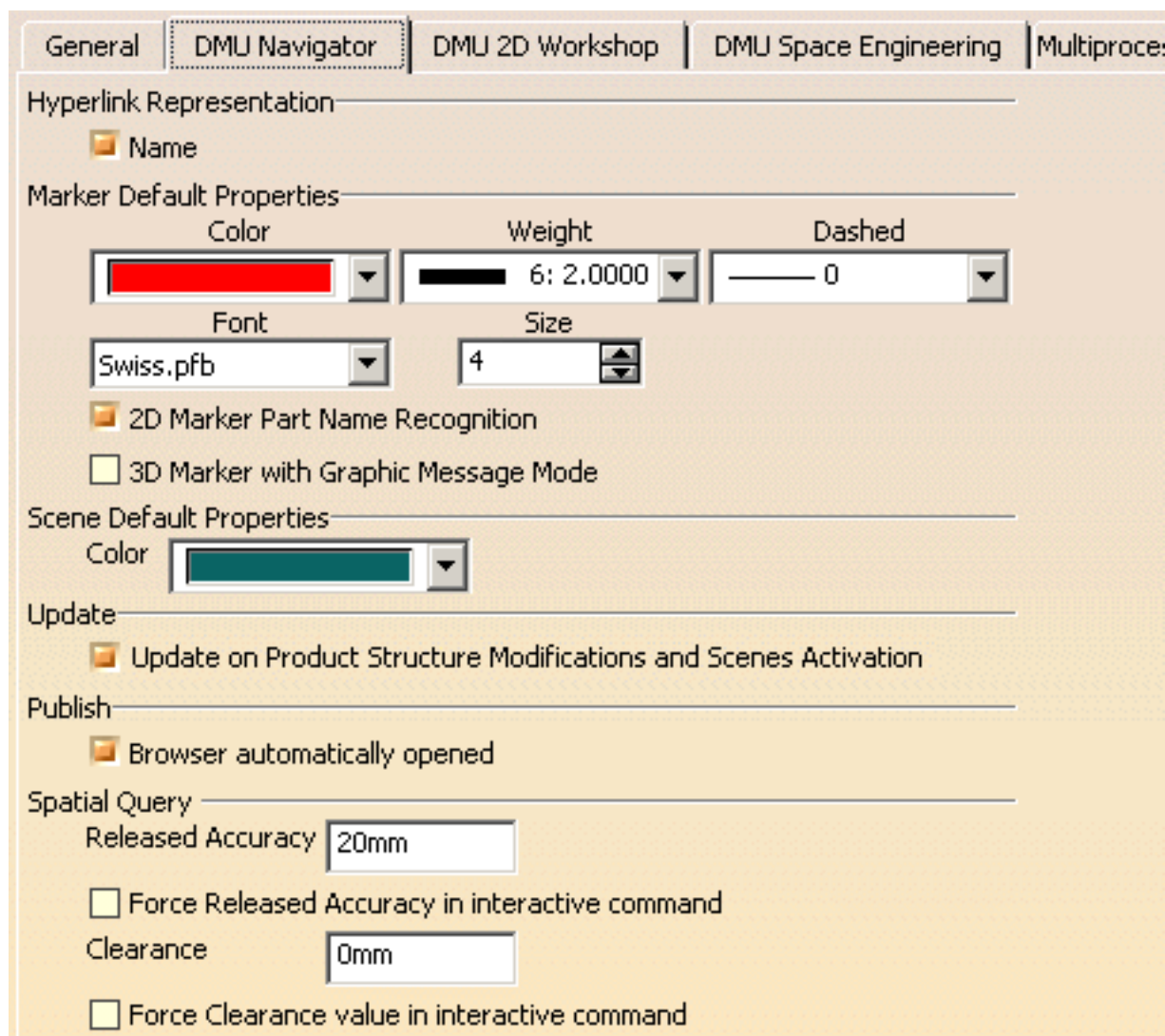
This task explains how to customize DMU Navigator settings.



1. In the menu bar, select **Tools -> Options**.

The Options dialog box appears.

2. Click the **Digital Mockup** category, then click the **DMU Navigator** tab.




The image shows the 'DMU Navigator' tab of the Options dialog box. The dialog has a tabbed interface with 'General', 'DMU Navigator', 'DMU 2D Workshop', 'DMU Space Engineering', and 'Multiprocessor'. The 'DMU Navigator' tab is selected. The settings are organized into sections: 'Hyperlink Representation' with a checked 'Name' checkbox; 'Marker Default Properties' with 'Color' (red), 'Weight' (6: 2.0000), 'Dashed' (0), 'Font' (Swiss.pfb), and 'Size' (4); '2D Marker Part Name Recognition' (checked) and '3D Marker with Graphic Message Mode' (unchecked); 'Scene Default Properties' with 'Color' (teal); 'Update' with 'Update on Product Structure Modifications and Scenes Activation' (checked); 'Publish' with 'Browser automatically opened' (checked); and 'Spatial Query' with 'Released Accuracy' (20mm), 'Force Released Accuracy in interactive command' (unchecked), 'Clearance' (0mm), and 'Force Clearance value in interactive command' (unchecked).

General | **DMU Navigator** | DMU 2D Workshop | DMU Space Engineering | Multiprocessor

Hyperlink Representation

☒ Name

Marker Default Properties


Color:  Weight: 6: 2.0000 Dashed: 0

Font: Swiss.pfb Size: 4

☒ 2D Marker Part Name Recognition

☐ 3D Marker with Graphic Message Mode

Scene Default Properties

Color: 

Update

☒ Update on Product Structure Modifications and Scenes Activation

Publish

☒ Browser automatically opened

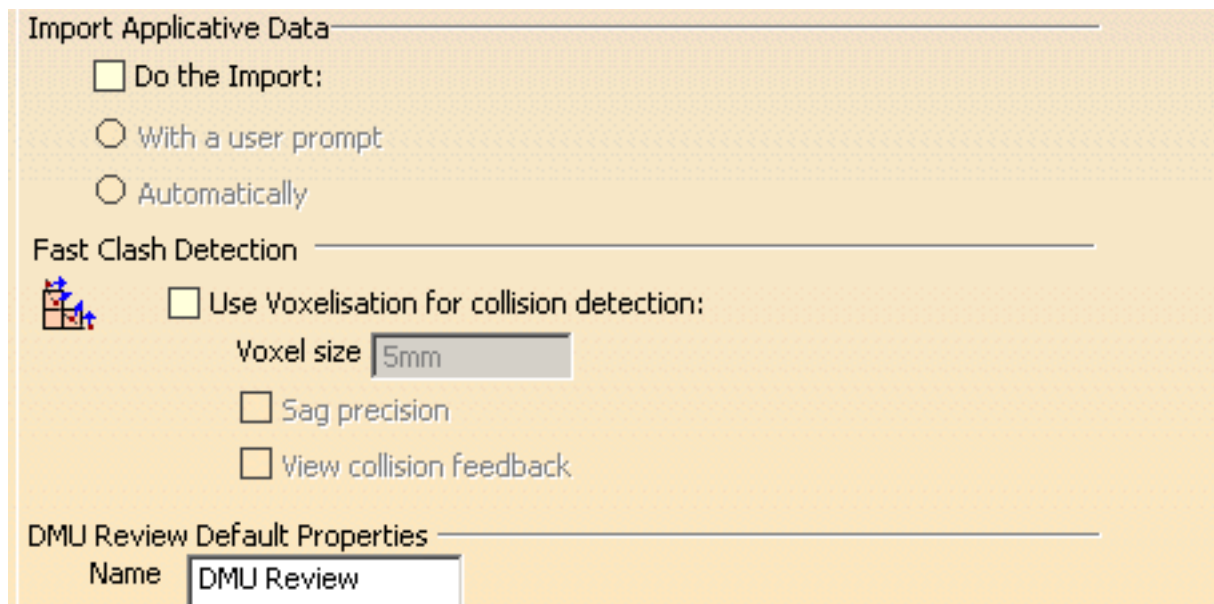
Spatial Query

Released Accuracy: 20mm

☐ Force Released Accuracy in interactive command

Clearance: 0mm

☐ Force Clearance value in interactive command




Import Applicative Data

☐ Do the Import:

☐ With a user prompt

☐ Automatically

Fast Clash Detection

 ☐ Use Voxelisation for collision detection:

Voxel size

☐ Sag precision

☐ View collision feedback

DMU Review Default Properties

Name

The **DMU Navigator** tab enables you to customize:

- Hyperlink representation (name)
- Marker Default Properties
- Scene Default Properties
- 3D annotation update (when moving the product)
- Spatial Query parameters
- Import of applicative data



You can add hyperlinks to your document and then use them to jump to a variety of locations, for example, to a marketing presentation, a Microsoft Excel spreadsheet or a HTML page on the intranet.

3. In the **Hyperlink Representation** area, check the **Name** selection box.

By default, hyperlink names are displayed : all hyperlink will be textual. The name you give the link in the Manage Hyperlink dialog box when you create it will appear when using go to hyperlink.

4. Click preview checkboxes as appropriate to change the automatic display setting of preview windows when creating cameras, manipulating objects, etc.

By default, preview windows are automatically displayed.

5. Change the color, font type and font size of 2D markers.

Note: Defining the Marker default properties in the **Tools->Options->DMU Navigator** sets the selected properties as default properties and changes how new annotations will look when you create them.

6. Click the **2D Marker Product Name Recognition** checkbox in order to activate the option to automatically use a Part's Name as the default for the creation of text annotations.

7. Click the **3D Marker with Graphic Message Mode** checkbox in order to activate the mechanism that enables you to transform temporary markers into persistent 3D annotations.

8. Change the Scene background color as desired.

9. Uncheck the **Update on Product Structure Modifications and Scenes Activation** option if necessary. This option manages a publish/subscribe mechanism which will manage the automatic updates of certain events:

- move
- shape activation
- insert
- replace product
- delete product
- UI-activation modification
- graphic attributes modification
- Enhanced Scene activation

If the option is unchecked, some applicative data will not be automatically updated upon the above events. However, if the option is unchecked, performance will be faster.

10. To activate the automatic launching of Publish results in a browser upon the stopping of the Publish process, in the **Publish** area, check the **Browser automatically opened** checkbox .



## 11. Spatial Query Released Accuracy

The default value for released accuracy is 20mm.  
This value can also be defined by the administrator.

How do you, as administrator, define the released accuracy value?

- you **set** a precise and required value in the **Released Accuracy** field.
- you then **run** the CATDMUUtility batch process without defining the -vox option, the released accuracy value is taken into account. The data is pre-tessellated.
- you **check** the **Force Released Accuracy in interactive command** checkbox if you wish to force users of the **Spatial Query** command to use the defined **Released Accuracy**.

## 12. Spatial Query Clearance

The default value for clearance is 0mm.  
This value can also be defined by the administrator.

How do you, as administrator, define the clearance value?

- you **set** a precise and required value in the **Clearance** field.
- you **check** the **Force Clearance in interactive command** checkbox if you wish to force users of the Spatial Query command to use the defined Clearance value.

## 13. Import Applicative Data

- Do the Import:
  - activated: the applicative data will be inserted
  - not activated: the applicative data will not be inserted
- with user prompt: a dialog box will appear enabling you to select any subset of applicative data to be inserted
- automatically: all applicative data will be inserted, no dialog box will appear
  - activated: the applicative data will be inserted

## 14. Fast Clash Detection

- If you select a voxel size and check the View Collision Feedback button, then you obtain a clash detection with a voxel precision of the size selected, with the clashing parts

highlighted, and the center of the voxel points appearing in red.

- If the View Collision Feedback button is unchecked, then you will obtain a clash detection with only the clashing parts highlighted.
- If you select sag precision and check the View Collision Feedback button, then you obtain a clash detection with the precision of the sag. The clashing parts are highlighted and the intersection curve of the clashing parts appears in red.
- If the View Collision Feedback button is unchecked, then you obtain clash detection with with the clashing parts highlighted.



#### 15. DMU Review Default Properties

In the **Name** text-entry field, enter the desired default name for DMU Reviews.

16. Note that you need to exit and restart to see the effects of customization, so click **OK** to confirm.



# Glossary

## C

- clash** A logical process checking whether the shuttle and the environment are in intersection or in contact.
- clearance** A logical process checking whether the minimum distance existing between the shuttle and its environment is greater than a specified threshold distance or not.
- collision** A logical process checking whether the shuttle and the environment are in intersection or in contact.

## D

- distance** A logical process measuring the minimal distance between the shuttle and its environment (considered as geometrical entities).

## G

- group** A entity to group models together. This entity is recognized during shuttle, distance and clash definition.

## I

- interference** A logical process checking whether the intersection of the shuttle and its environment (considered as geometrical entities) is null or not.

## L

- location** A position in the space used to define the trajectory of the shuttle during its motion.

## M

- manipulator** An graphic editor for positioning the shuttle. It is composed with symbols and a dialog box. The symbols are used to drive the positions of the shuttle in a specific direction.

## P

- position** The temporary location of a shuttle during manipulation.

## S

- shuttle** A shuttle is an element representing the object to be fitted or unfitted from an assembly. It is composed of an axis and list of models. The shuttle can be moved about to simulate the fitting or unfitting of the object within the assembly.

- shuttle axis** The position of a particular axis attached to the shuttle, by convention. This axis is called shuttle axis. This axis is choose by the system at the center of the shuttle bounding sphere.
- simulation** A record and replay of a scenario in which a shuttle is dismounted from an assembly according to some constraints like interferences. During the definition and replay [interference](#) detection and [distance](#) measurement can be performed.

## T

- track** A sequence of locations forming a single path for a shuttle to follow. A track can be defined by creating [locations](#). A track can then be simulated. During this simulation, [interference](#) detection can be performed.

# Index

[A](#)[B](#)[C](#)[D](#)[E](#)[F](#)[G](#)[I](#)[M](#)[O](#)[P](#)[R](#)[S](#)[T](#)[U](#)[V](#)[W](#)

## A

- About
- Actions
- Adding
- Advanced Tasks
- Analyzing
- Animating
- Automatic Clash Detection
- Automatic Path Finder
  - Using














## B

- Basic Tasks










## C

- Clashes
  - Detecting
- commands
  - automatic clash detection
  - clash
  - explode
  - Gantt Chart
  - measure
  - Player
  - replay

reset	
sequence	
simulation	
smooth	
swept volume	
track	
Constrained Assembly	
Exploding	 
Converting	
Copying Pasting	
customizing	
DMU navigator settings	



D

Dedicated tools	
Player	
Recorder	
Track Editor Recorder	
Defining	            
Detecting	
Dismounting	
Dismounting Operation	
Finalizing	
Illustrating	
Displaying	
Distances	
Measuring	
DMU Check	







E

- Editing 
- Entering Workbench 
- Exploding      



F

- Film
- Generating  
- Filtering 
- Finalizing 



G

- Generate a Replay  
- Generating  
- Getting Started 
- Glossary 







I

- Illustrating 
- in Track Context
- Analyzing     




M

- Manipulation 
- More About 
- Measuring 
- Menu Bar 

Mono-Shuttle Fitting	
More About	 
More About Path Finder	
Motion	
Validating	
Moving	
Multi-Shuttle Simulation	
Recording	




























O

Objects	
Snapping	








P

Path Finder	
path finder	
interactive mode	
more about	
publishing	 
writing macros	
Path Finder in Interactive Mode	
Using	
Path Finder in Macro	
Writing	
Path Finder Results	
Publishing	 
Perform an Experiment	
Player	 
Positions Automatically	
Validating	 
Preface	         



Product

- Dismounting 
- Exploding  
- Publishing  




















R

- Recorder 
- Recording     
- Replaying   



S

- Sequence
  - Defining 
- Sequence Capabilities
  - About 
- Sequence Editor 
- Sequence Gantt Chart
  - Displaying 
- Sequences  
- Setting Up Your DMU Fitting Simulator Session 
- Several Shuttles
  - Defining 
- Shuttle  
  - Defining  
  - Editing 
  - Exploding 
  - Moving 
- Shuttle in a Simulation
  - Adding 
- Shuttle made of Shuttles
  - Defining 

## Shuttle Motion

Defining



## Shuttle Motion Using Another Shuttle as Reference

Defining



## Shuttle Motion using Product as Reference

Defining



## Shuttles

About



Using



## Simulation

Mono-Shuttle Fitting



Recording



Replaying



## Simulation automatically

Recording



## Simulation into Sequence

Converting



## Smooth

Using



## Snapping



## Specification Tree



## Starting Session



## Swept Volume

Defining



More About



## Swept volume

defining



filtering positions



from any moving reference



more about



## Swept Volume from any Moving Reference

Defining



## Swept Volume Positions

Filtering





T

Target Capability


Using 

Toolbar

Automatic Clash Detection 

DMU Check 


Manipulation  

Simulation 


Track

Recording 

Replaying 

Selecting name 

Selecting speed 


Selecting time 

Updating 

Track and Sequence Capabilities


Using 

Track Capabilities

About 


Track Editor Recorder 

Track Operators

About 


Tracks  

Copying Pasting 

Using 



U

Updating 

Using        



V

Validating



Viewpoints

Animating



W

Workbench Description



Writing

