Advanced CATIA

AMET – 213

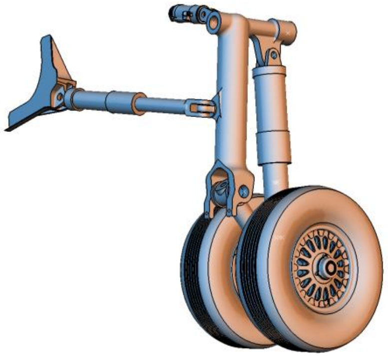
Lecture 1

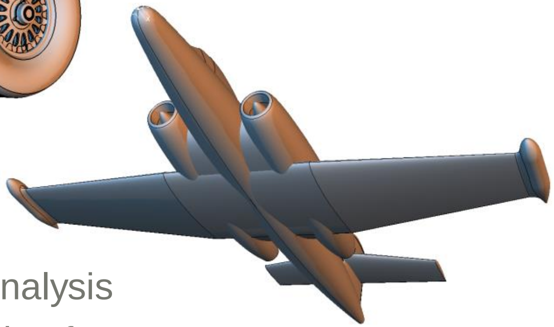
Kinematic I

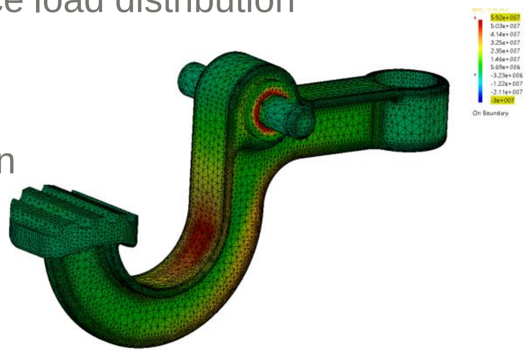
Introduction to Kinematics

Semester Agenda

- 14-week course divided into 4 sections
  - Kinematic mechanisms
    - Create assemblies and apply motions
    - Extract trace elements
  - Advanced surface wireframe applications
    - Use images to trace profiles
    - Use advanced surfacing, wireframe and analysis methods
  - Introduction to Finite Element Analysis using CATIA Structural analysis
    - Build a static singular part and apply forces to introduce design features
    - Build an assembly with fasteners, apply a force load to introduce load distribution
  - Sheet metal, Surface Un-wrap for composite applications
    - Introduce sheet metal design and how it differs from Part design
  - Human Factors, Fitting Analysis, Computational Fluid Dynamics
    - Fitting analysis demonstrates part removal in a confined space
    - Human Factors demonstrates manikin placement, vision view-point and reach
    - Describe how surface design is used by Computational Fluid Dynamics (CFD) for aerodynamic study and improvement







Semester Agenda

- Mid Term exam will include Kinematic design, and parts of Surface design
  - Starting the week of October 16
  - Represent 25% of final grade
- Final exam will focus on FEA and flat pattern and a surface problem.
  - Represent 25% of final grade
  - Starting the week of December 11
- Homework will include multiple parts, assemblies and features
  - Upload EVERYTHING to homework drop box
    - *Remember to always use Save Management*
  - Geometry not handed in to the drop box cannot be graded.
  - Please do not email homework for grading
  - Homework will represent 50% of your final grade
- Bonus questions will be handed out to add additional marks to a previously submitted assignment
- Teaching emphasis will be on this semesters new subject matter
  - Creating sketches, building assemblies, adding constraints had been taught last semester, it is therefore expected you know how to complete that portion in assignments
- Attendance matters.
  - Please make every effort to uphold the code of conduct on academic integrity expected for Centennial College.
  - Please put the phone away

Today's Agenda

- Re-familiarization with assembly constraints
  - Use soft and hard constraints
  - Re-position the geometry with the compass
  - Use Manipulate and discover advantages and disadvantages
- Familiarize the various Kinematic mechanism joints and features
  - Similar to constraints but combining soft and hard constraints
- Build a Kinematic set from Scratch
  - Manoeuvre geometry “close” to the finished placement
- Build from a pre-designed Constrained model
  - Use the Wizard tool automatically convert Constraints to kinematics
- Build a Single component Kinematic motion model
  - Add limits to define maximum and minimum movement
- Build a Simulation
  - Motion determined by speed and rate to display
- Build a Reply
  - Motion stored as a defined replay able kinematic motion.
- Step by Step
  - Convert an assembly model to a Kinematic model

Assembly Constraints

- Soft Constraints
  - Snap
  - Compass

Manipulation

Snap

Explode

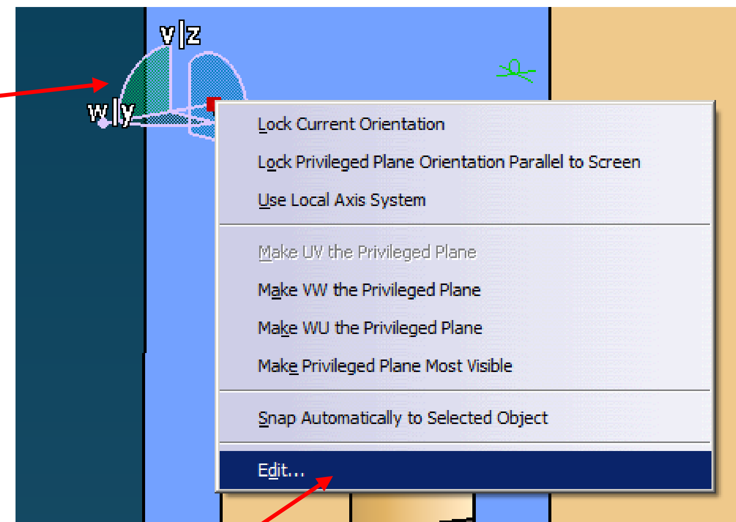
Stop Manipulate on Clash

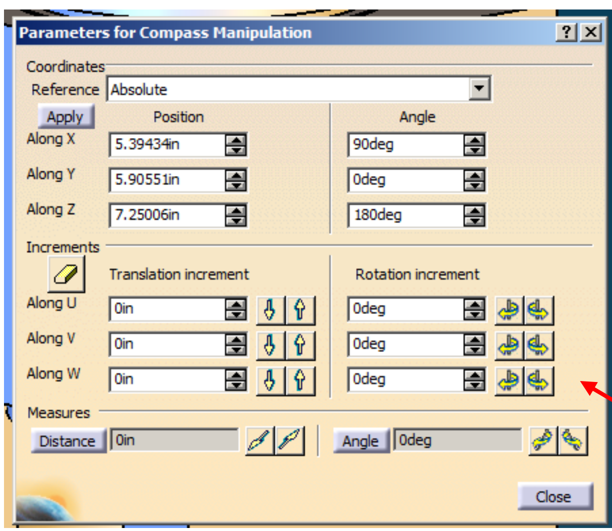
Cumulative Snap

Snap

Smart Move

Moving the Compass to a component allows free movement





With the compass placed on a component, RMB and select EDIT to adjust part position using Increments

Assembly Constraints

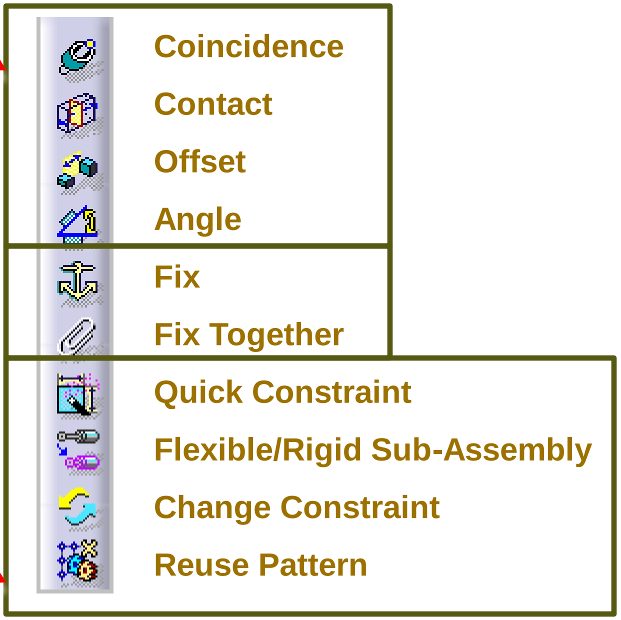
- Hard Constraints confine the movement to match the rule
- Assembly constraint apply limited degrees of freedom.
  - How much movement will the constrained joint provide
- Coincidence will allow rotation and translation
  - 2 degrees of freedom, each infinite in dimension
  - Angular rotation and distance traveled are two degrees of freedom
- Contact, Angle and Offset provides one motion on a plane, but infinite possible directions
- Fix is an anchor. Restricts movements and locks a position
- Fix Together acts as an soft anchor to connect two part together.
- Quick Constraint is a wizard tool. Seldom correct
- Flexible / Rigid will permit movement in a sub assembly
- Change constraint and Reuse pattern are repair and visual tools to assist building a constraint.

Hard Constraints

Coincidence
Contact
Offset
Angle
Fix
Fix Together
Quick Constraint
Flexible/Rigid Sub-Assembly
Change Constraint
Reuse Pattern

Assembly Constraints

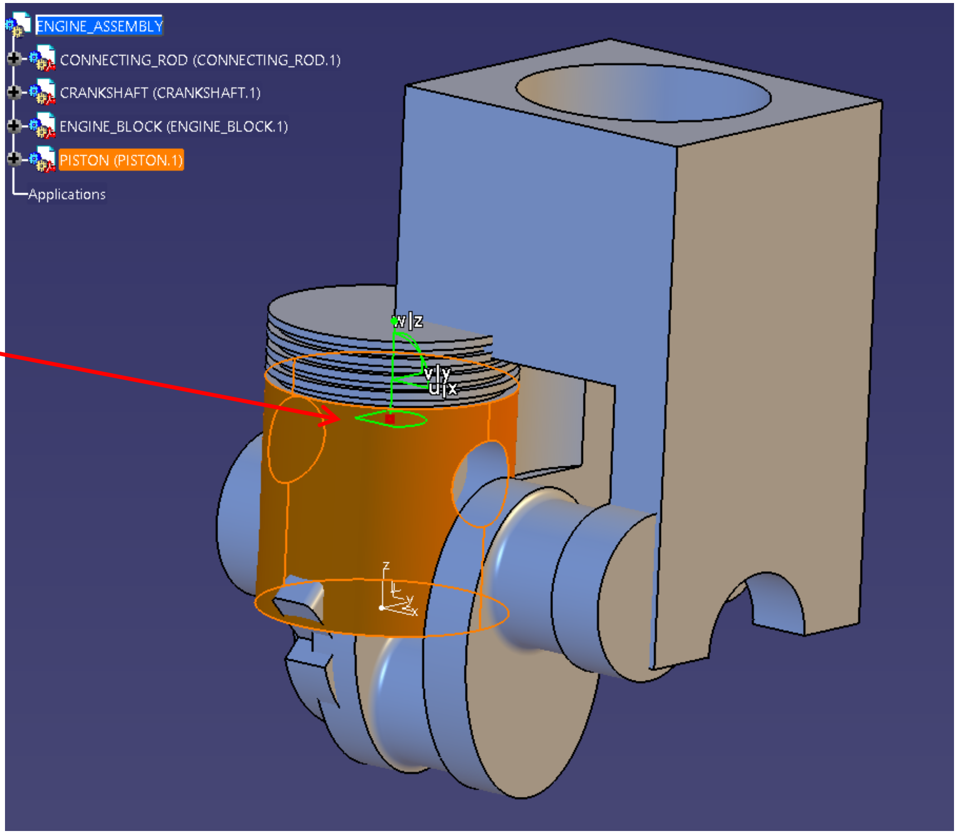
- Coincidence, Contact, Offset and Angle will either impose or remove a degree of freedom.
- Fix anchors movement. Fix Together ties on motion to another part. It follows another parts behaviour, either moving or rigid
- Quick constraint, Flexible, Change constraint and re-use pattern are applications to increase constraint placement and usage. Constraints pre-exist prior to applying these functions



Compass

- The parts to build an assembly needn't be positioned in placed, however, doing so will assist how the piece will eventually be connected
- The Compass, and Snap tools are beneficial to applying Soft Constraints. They are not secured tightly and can move freely once vacating the function
- Using the explode option also can improve visualization

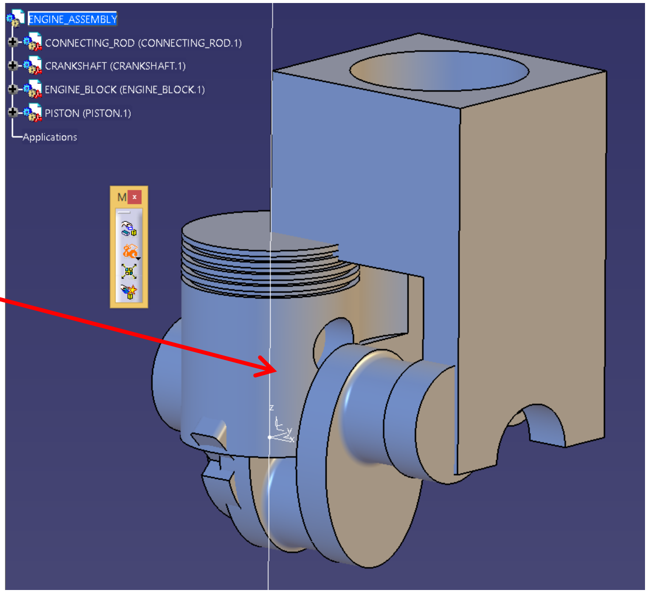
With the Product selected in Blue, the Orange component will move with the Green compass

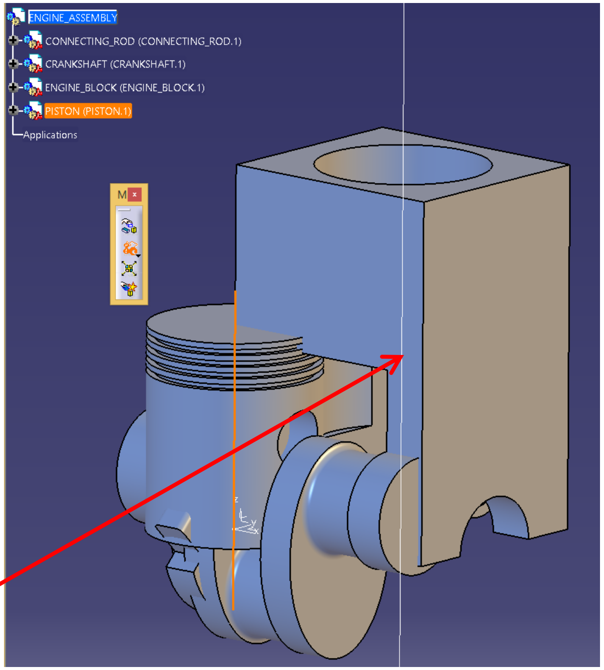


Snap

- Snap is a useful tool to reposition parts quickly
- Snapping a feature from one part to snap atop a feature of the second part
- As with Compass, the parts are merely positioned, not locked. As such, they are free to move after departing the function

Following selecting the function, passing over a part, the feature beneath the cursor will scan if there is a "snap-able" feature, it is highlighted. Select the feature with the cursor. Pass over the similar feature on the second part to snap in place



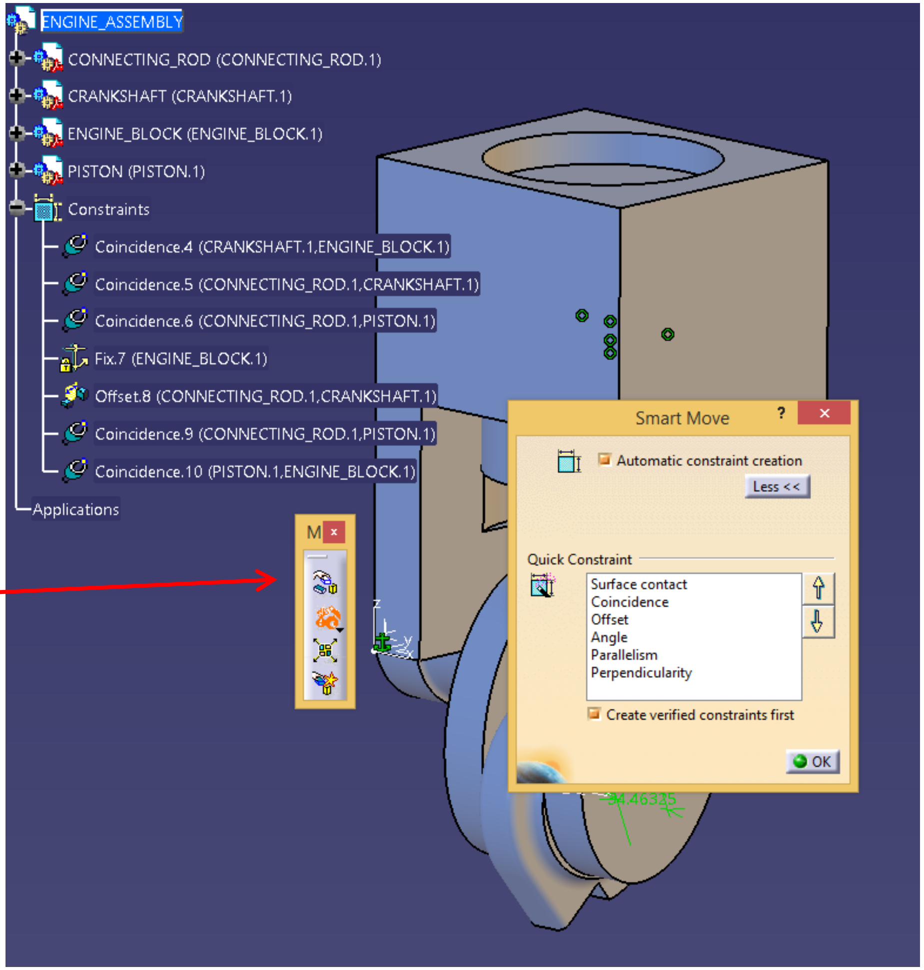


Snapping to the second part to align centerlines in this case

Smart move

- Smart Move combines Snap and Constraints to move, position and define assembly parts movement, path and freedom in an assembly

By activating the Automatic Constraint Creation feature, as you Snap parts into place, constraints are being created.



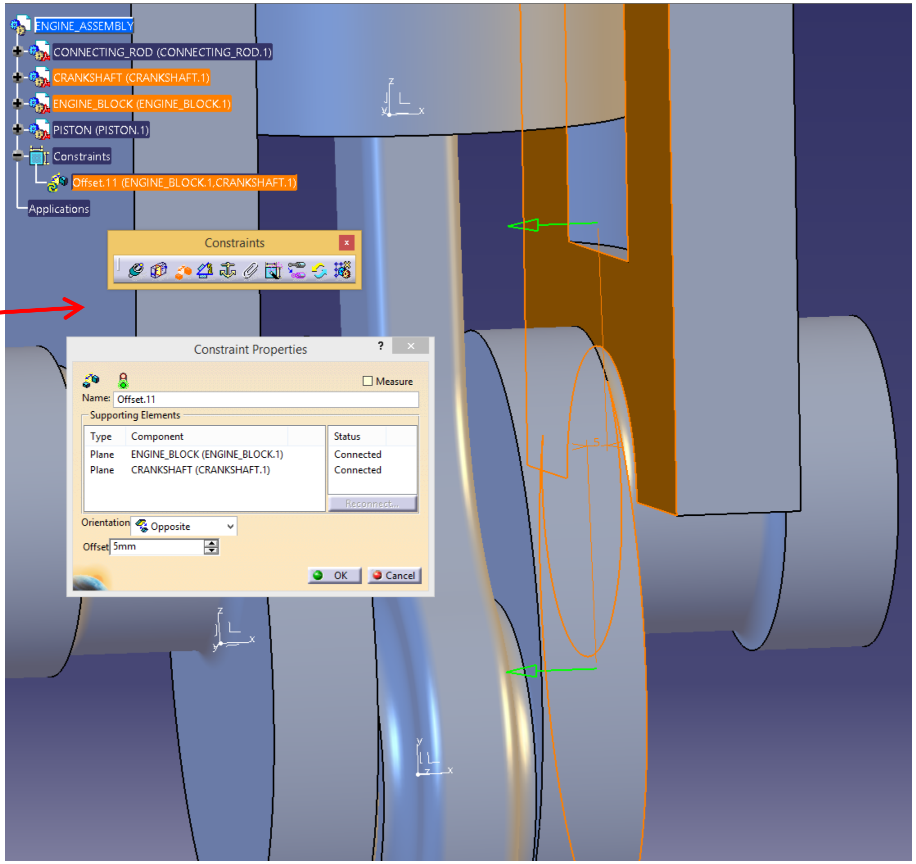
Applying Constraints

- If Smart Move was not used, applying constraints are required to validate a mechanisms movement
- As the parts are positioned in very close proximity to their initial position using the compass or snap, adding constraints now will be less surprising
- Use the Update button to refresh the parts position.

Be mindful when selecting parts to align, be offset or in contact.

Coincidence command does more than just centerlines

Don't forget to add the FIX command.



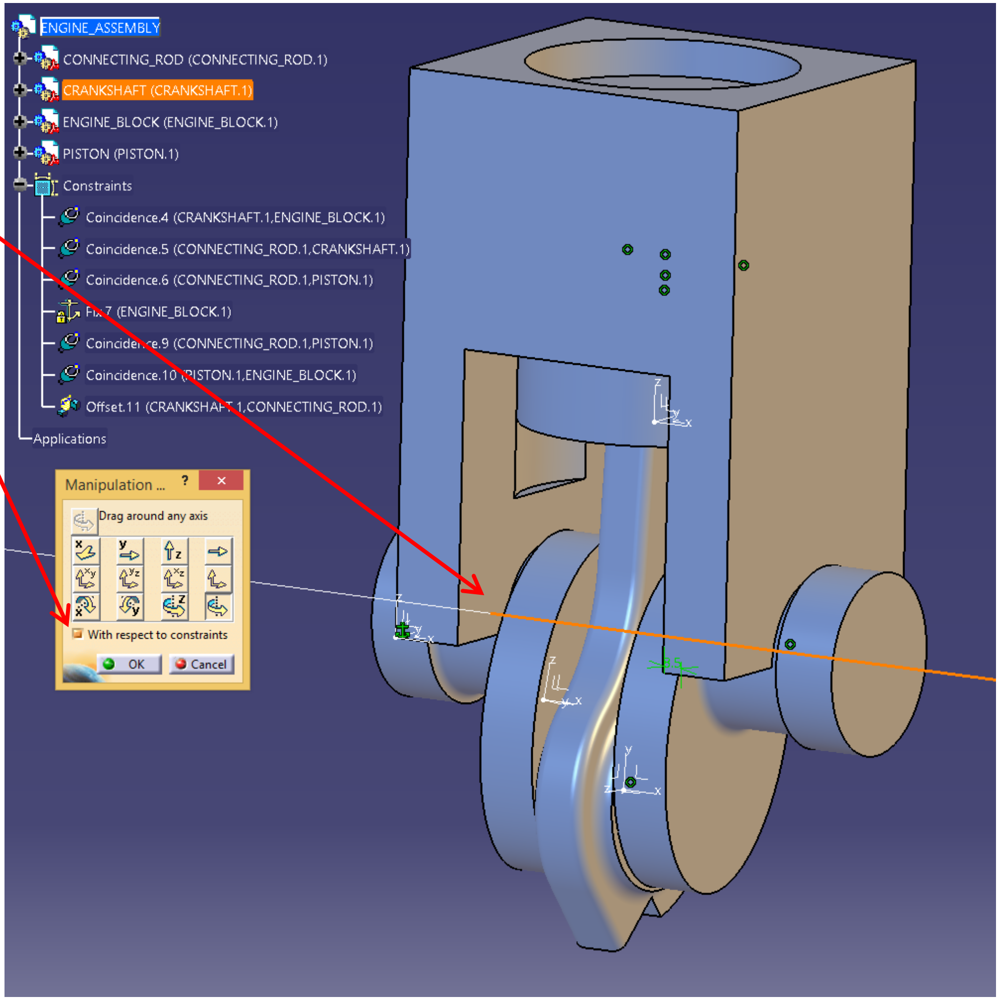
Manipulation

- After you believe constraints have been applied, using the manipulation command will demonstrate the mechanisms movements

Rotating the crankshaft around its shared axis on the engine block

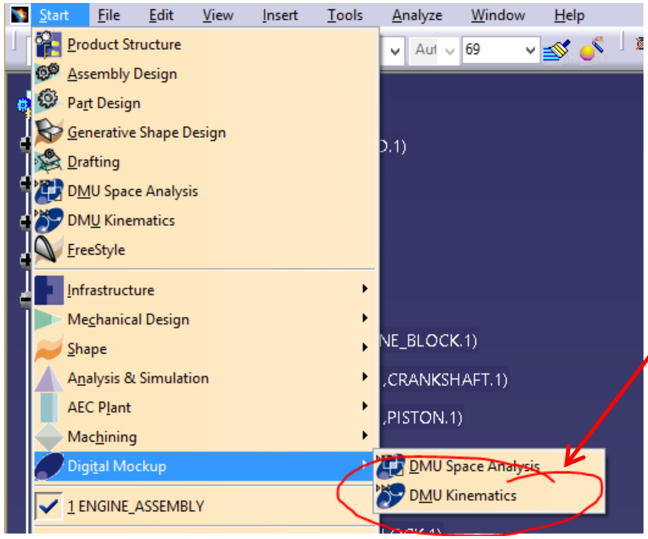
Manipulation is a good tool to ensure motion will occur

It is limited not showing sustained motion Distance bands, clashing, and output analysis is restricted to a stationary setting



Kinematic WB

- Similar to Constraints, Kinematic Work Bench joints limit, add or restrict degrees of freedom.
- Where Manipulation requires you to select an axis or direction and grab a part to move with respect to the motion, Kinematics will only apply motion commands to a variable that is definitive in direction and magnitude.
- Their descriptions are similar to Assembly Constraints and act in a similar manner. An added feature is the motion limits can be applied where as assembly constraints are infinite
- The preferred entry for Kinematics is an already opened assembly
- Select from the Start menu, Digital which commands can be used to apply motion

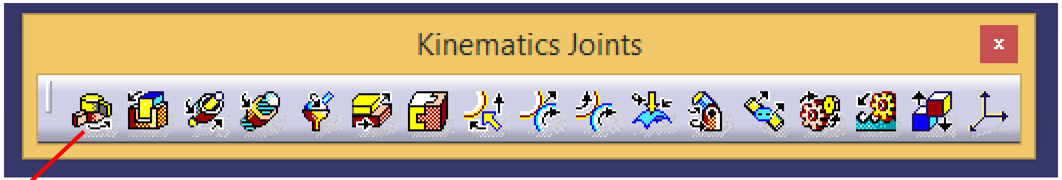


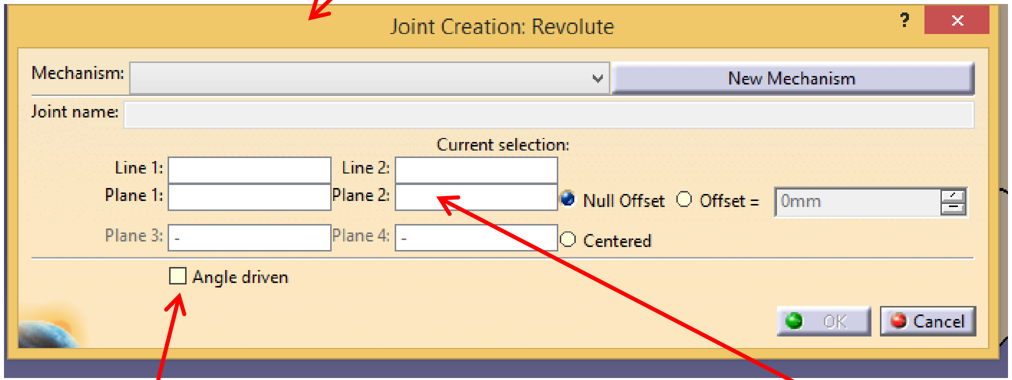
Since Kinematics requires an assembly to function, it will overlay the current assembly on the screen

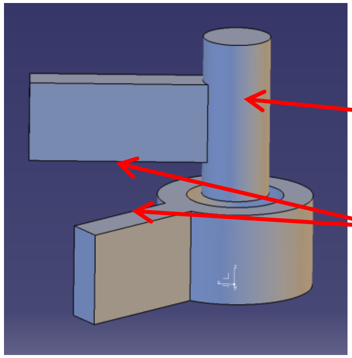
Kinematic WB

- As with all new functions in CATIA, I strongly recommend you expand the menus to illustrate the sub functions and menus

Revolute joint







Axially aligned

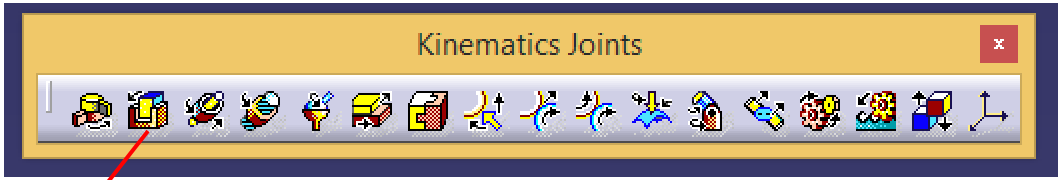
Distances maintained

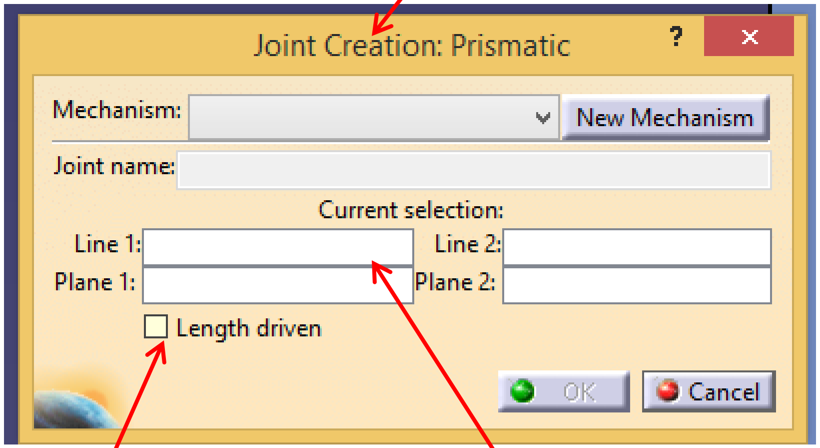
This command can drive with One degree of freedom, angular motions are defined

A Revolute will spin about a center line and requires a distance to maintain. The distance must be planar

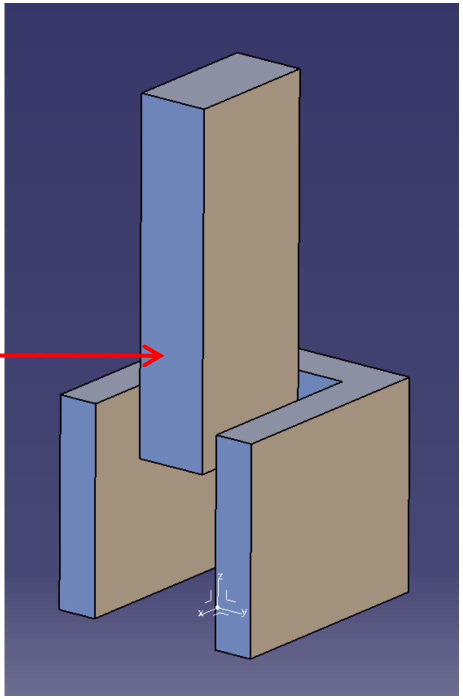
Kinematic WB

Prismatic joint





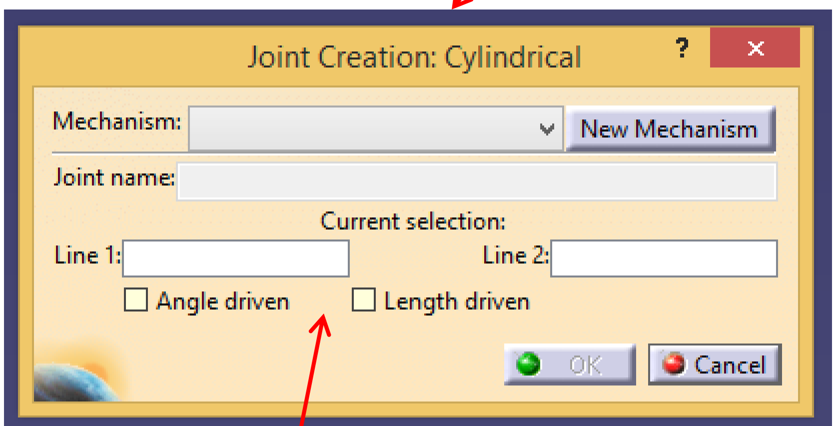
The edge of part1 and part2 are parallel and the sides are planar

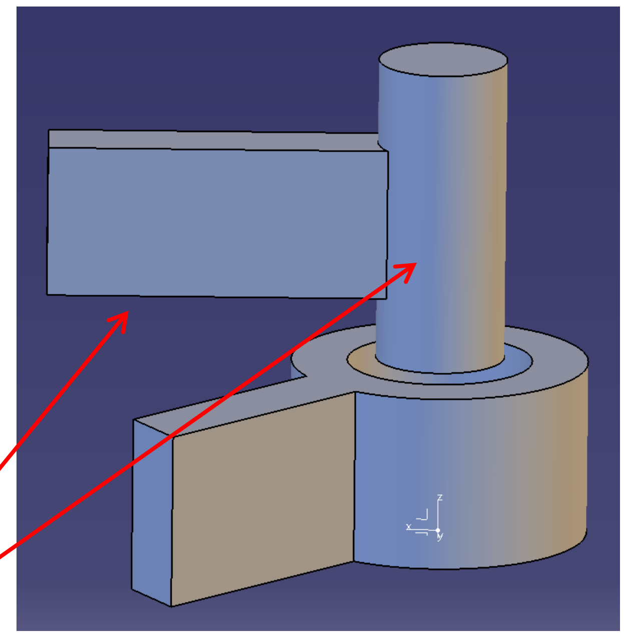


It can be moved along a length

A prismatic joint slides parallel to a line and on a planar surface

Cylindrical joint



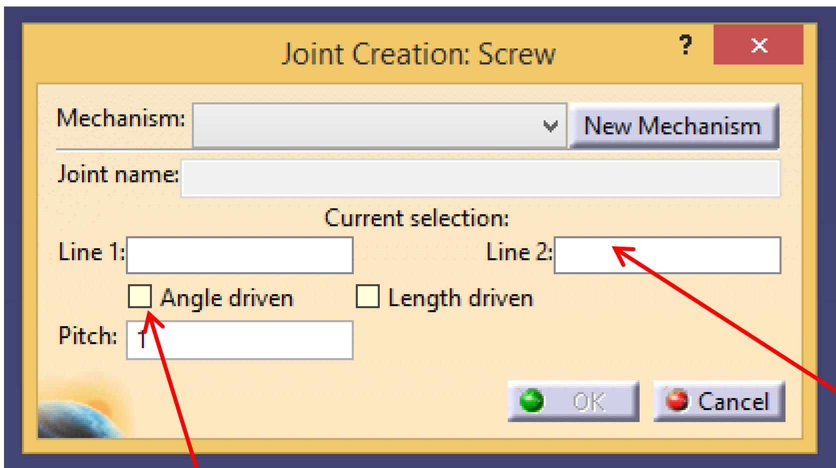


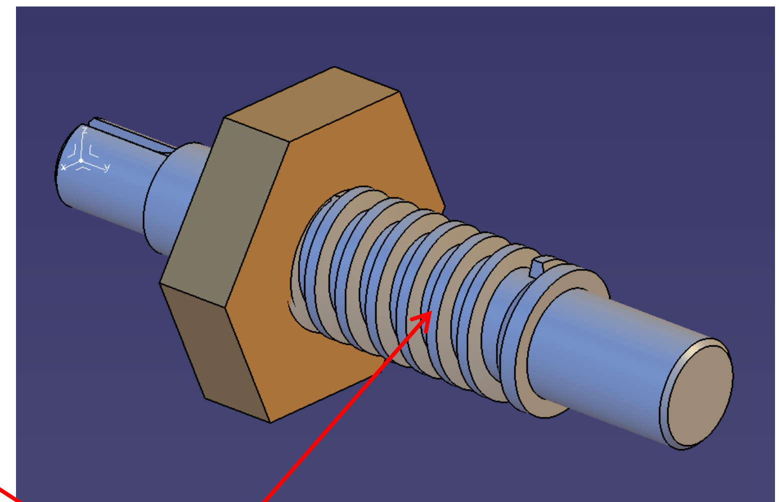
Either Angular or Length driven

Similar to Revolute however, both the centerline rotation or the distance can be driven, As the part moves, the centerline will remain coincident and the distances will remain planar

Kinematic WB

Screw joint



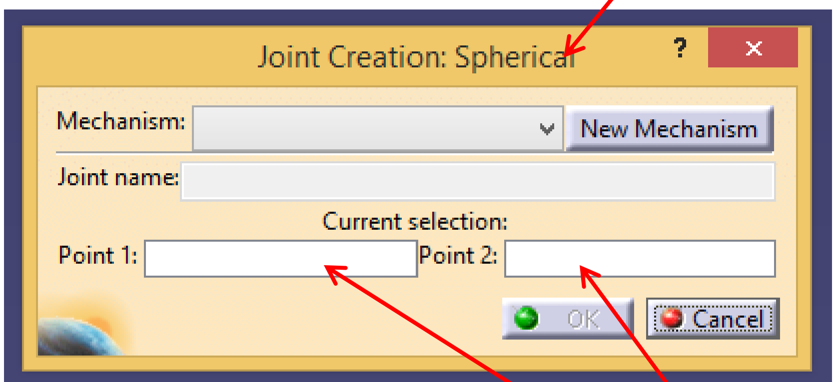


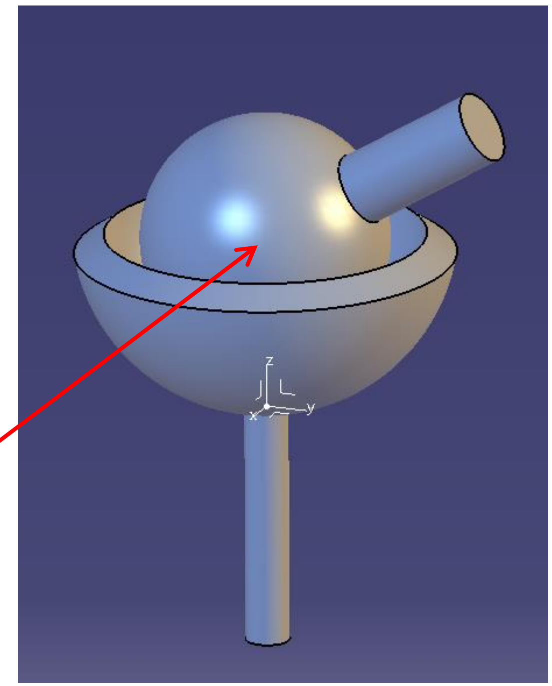
This can be driven in an angular motion or a length along the center lines

Two center lines are required and a pitch to regulate the rate of axial travel. A pitch of 10 = 1 revolution per 10 mm

Kinematic WB

Spherical joint

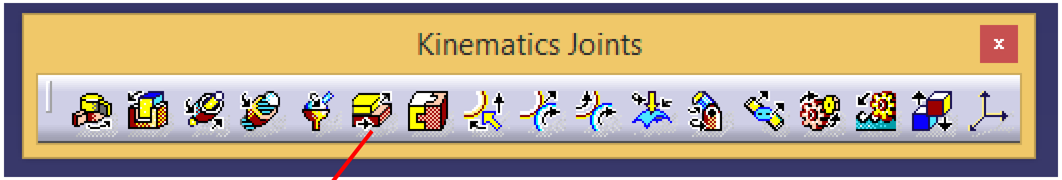


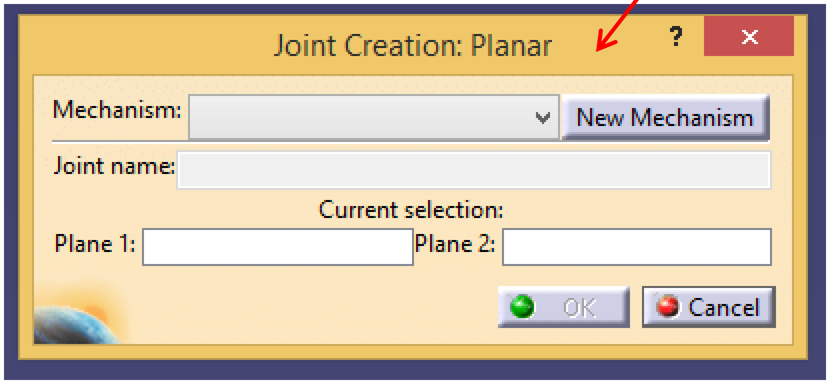


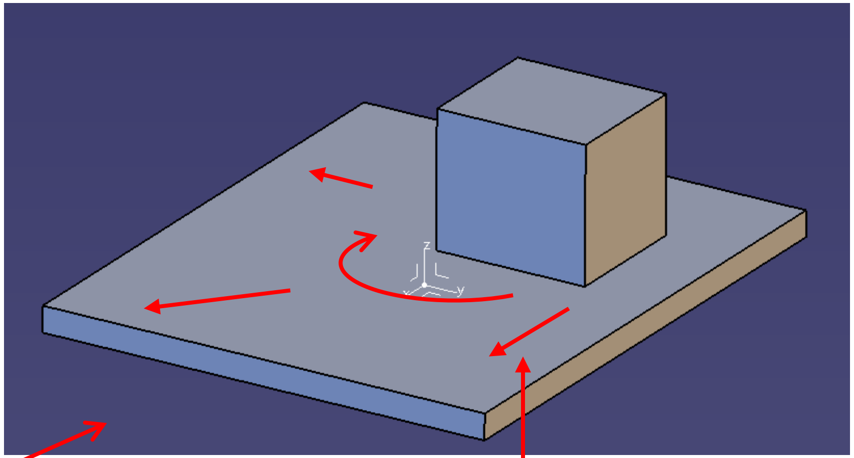
This joint cannot be driven, only followed

Two points are required; one on each part to declare how one part pivots around the other

Planar joint



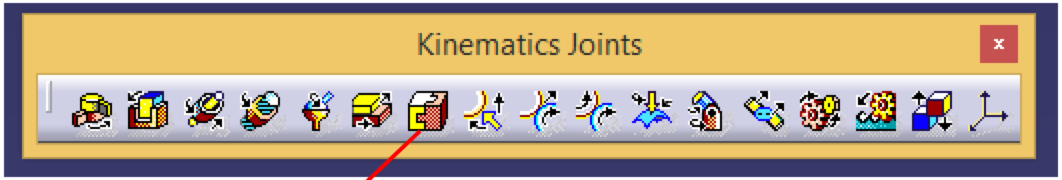


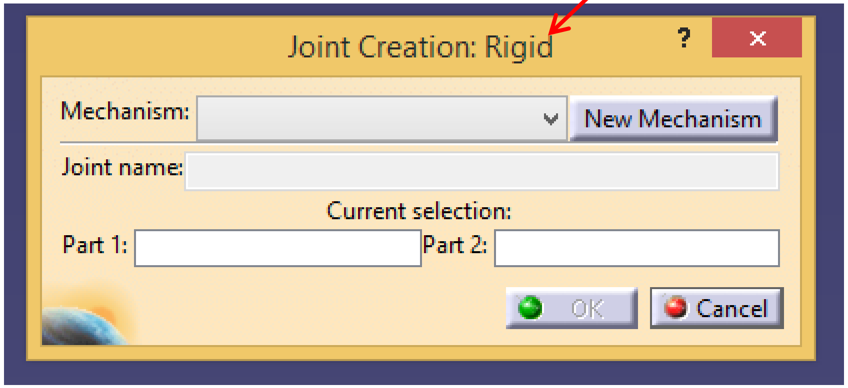


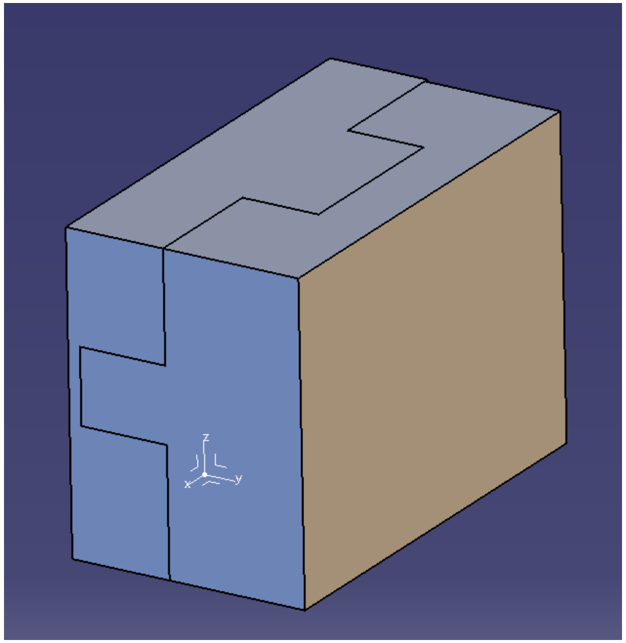
A planar joint must be opposite planes. You can not drive either planar surface as they can slide in any infinite length

The two parts have an infinite array of movements sliding on each other

Rigid joint



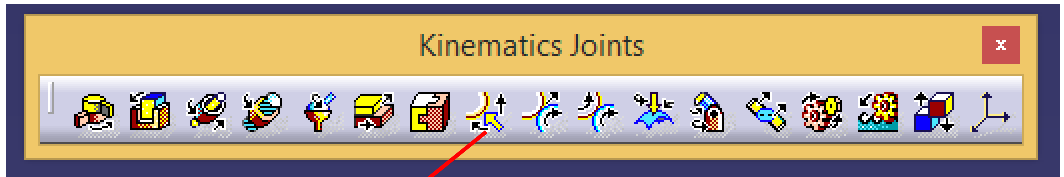


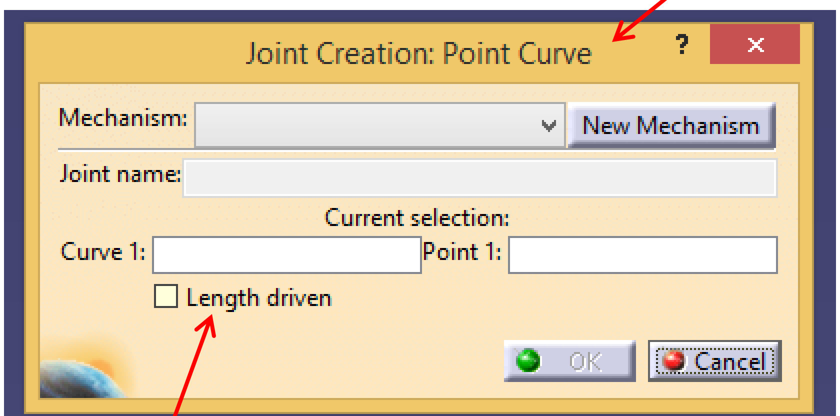


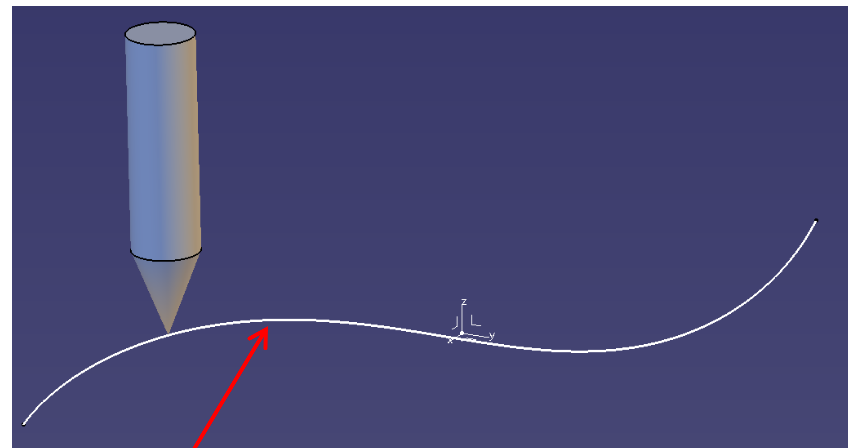
A Rigid joint connects two parts together regardless of edges, center lines or faces.

It cannot be driven, only follows

Point Curve joint



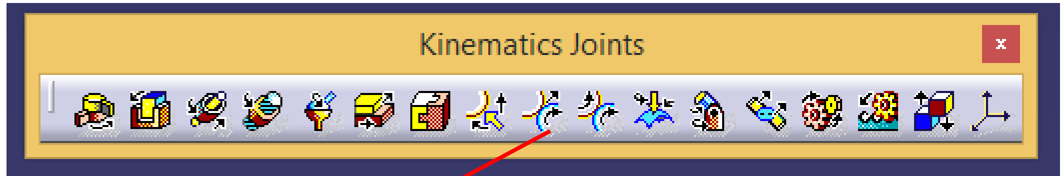


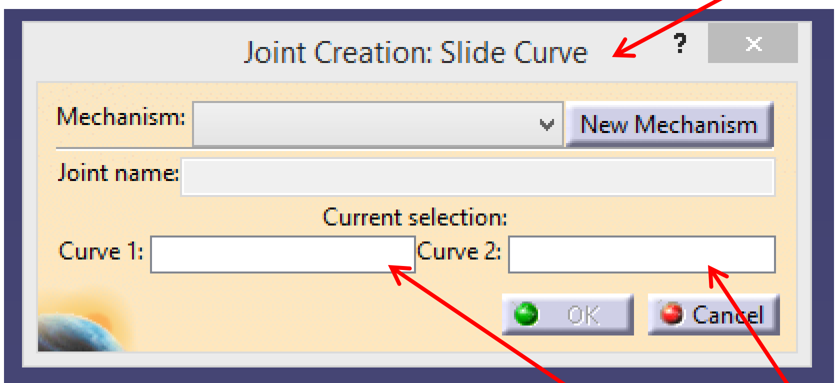


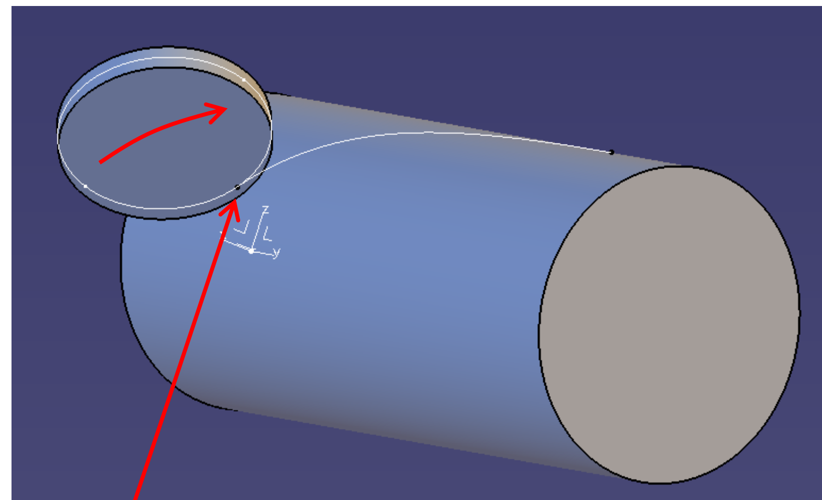
It can be driven along the length

Any curve can be the curve to drive along. It need not be a planar either, however, the point must be on the curve

Slide Curve joint



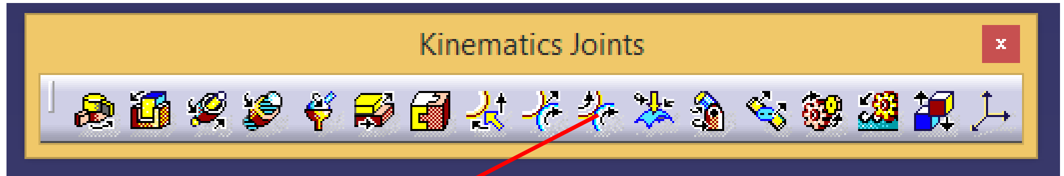


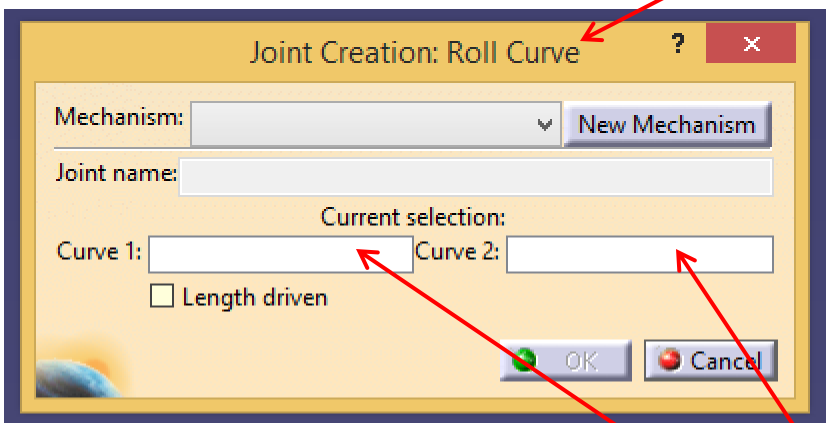


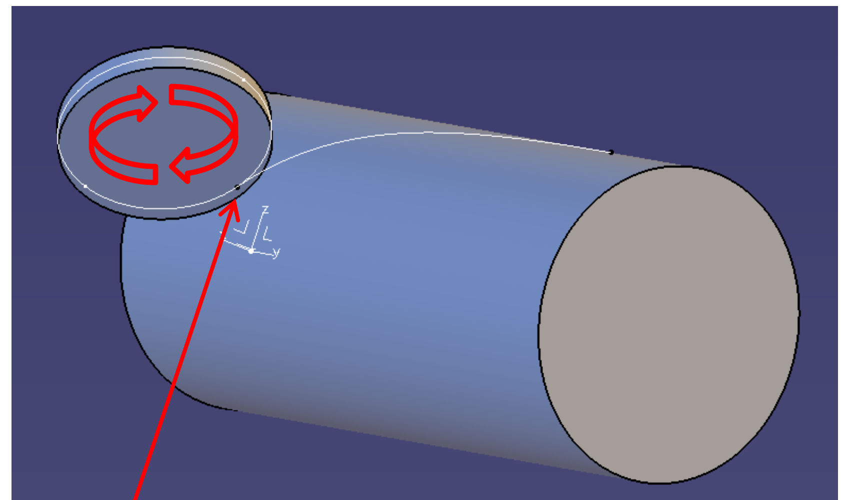
It cannot be driven, it can only follow

Any curve can be the curve to slide along. It need not be a planar either, however, the curves must be tangent to each other.

Roll Curve joint



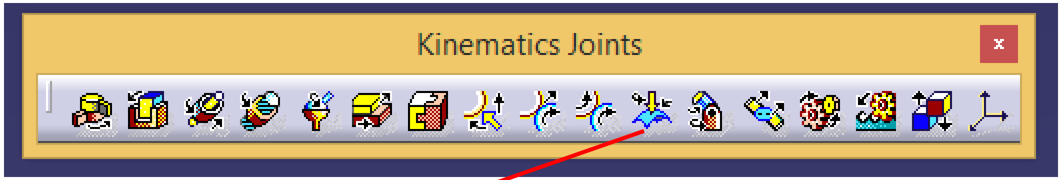


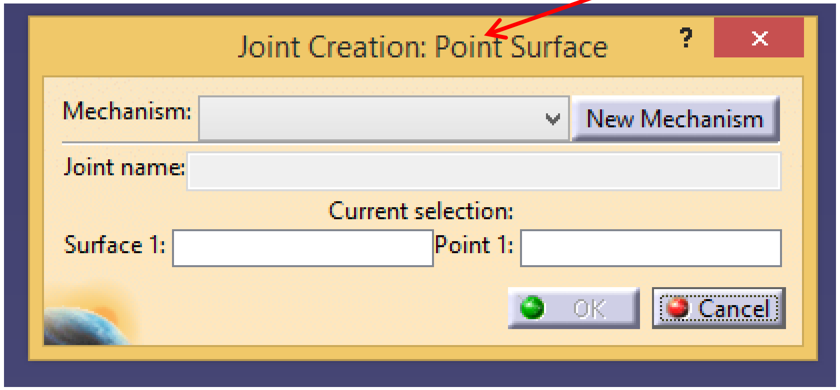


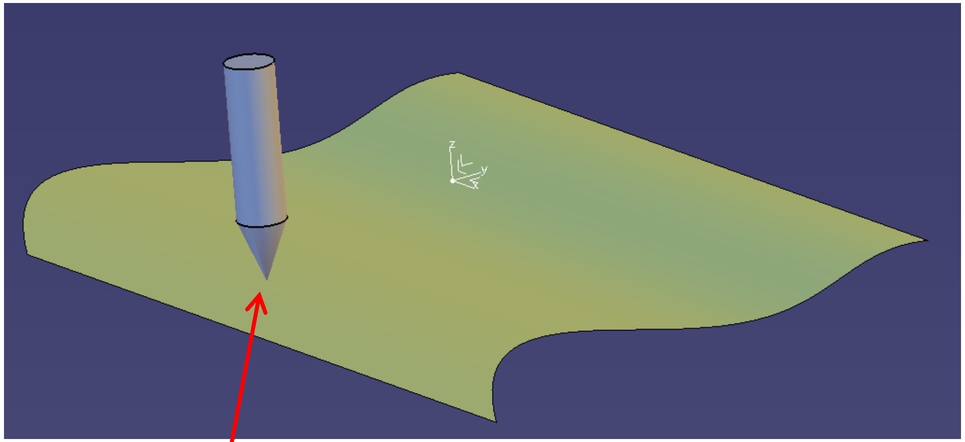
It can be driven along the length

Any curve can be the curve to drive along. It need not be a planar either, however, the curves must be tangent to each other.

Point Surface Joint



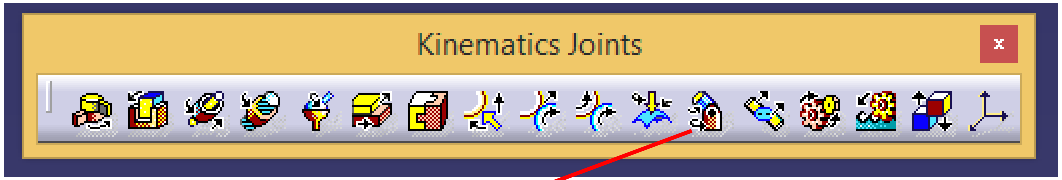


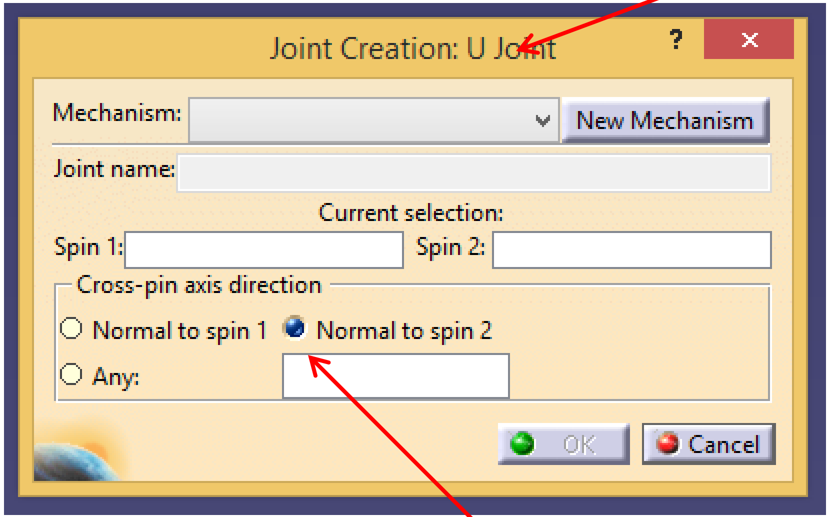


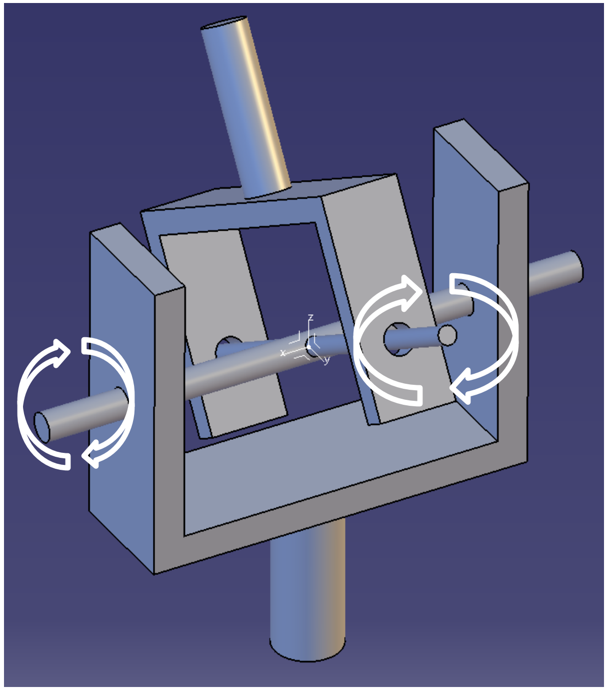
It cannot be driven, only follow

Any point can be the point but it must be on the surface

Universal Joint



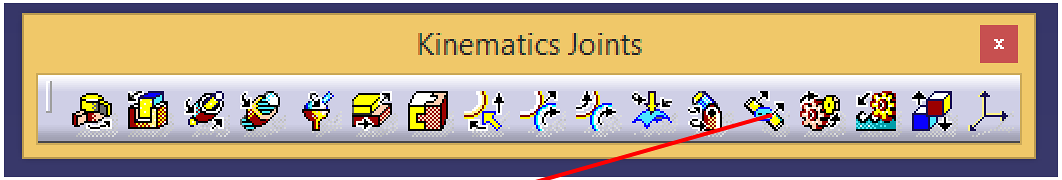


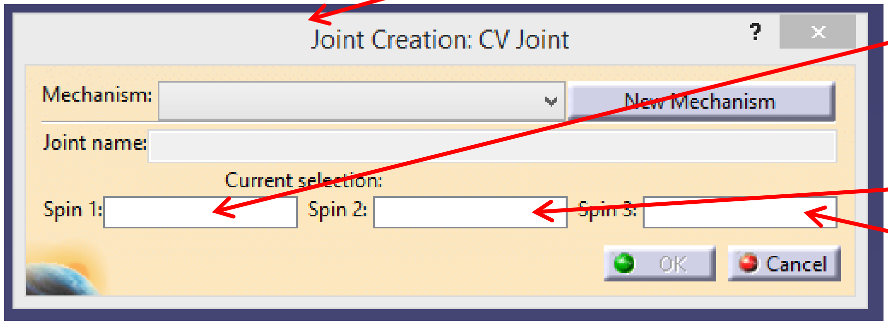


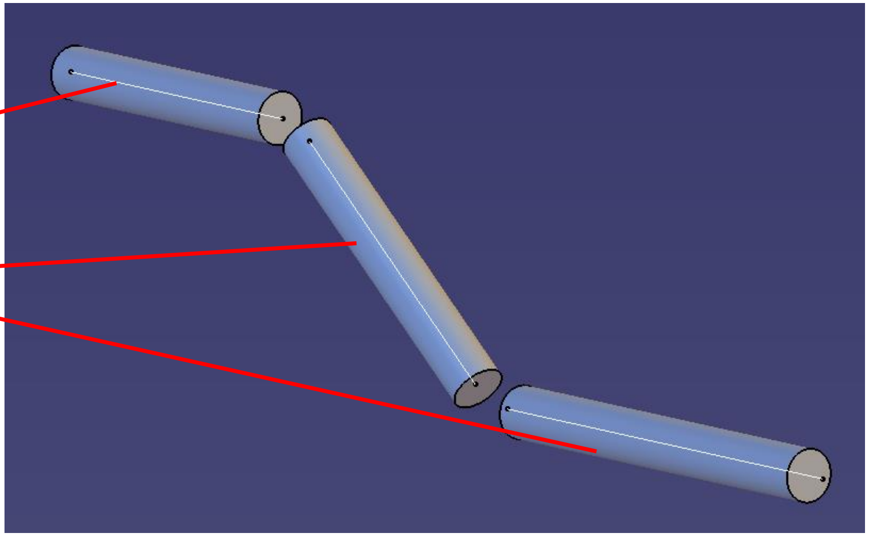
It cannot be driven, but you can declare which line the spin will spin and which will hold

Revolves one part through a second part along a spin line on both parts

CV Joint



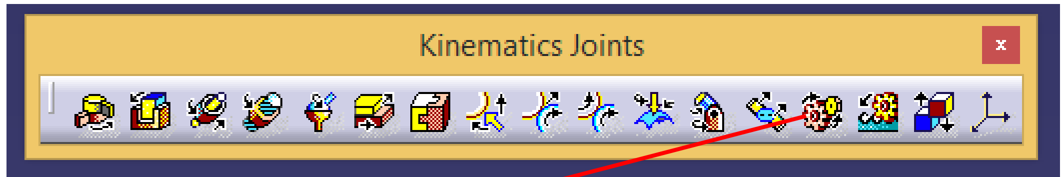


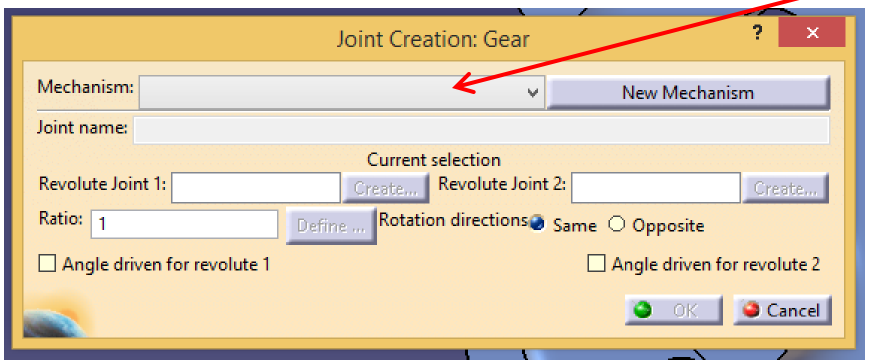


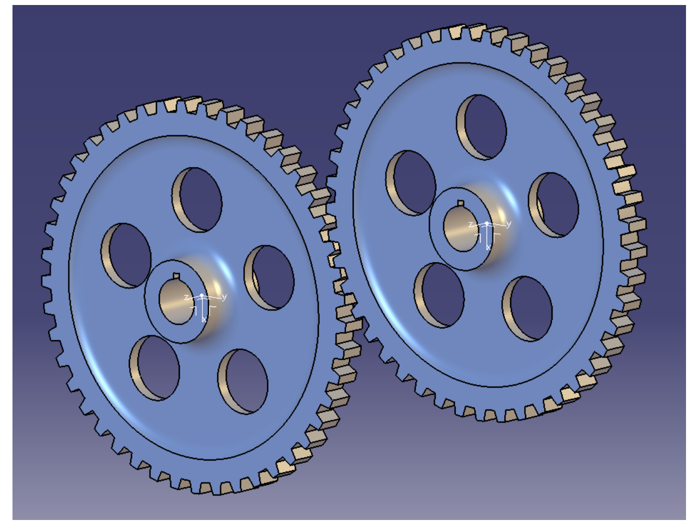
A CV joint transfers motion on 3 lines. The speed on all three lines is constant. It cannot be driven

Each line is the centerline extracted from 3 separate parts

Gear Joint

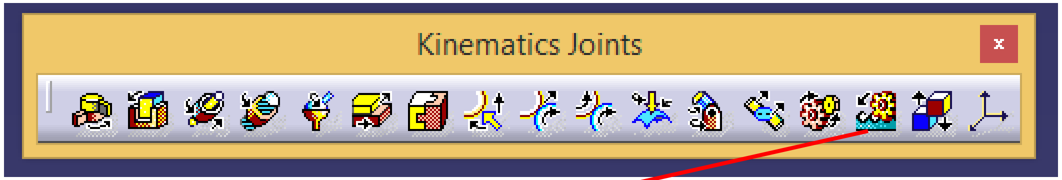


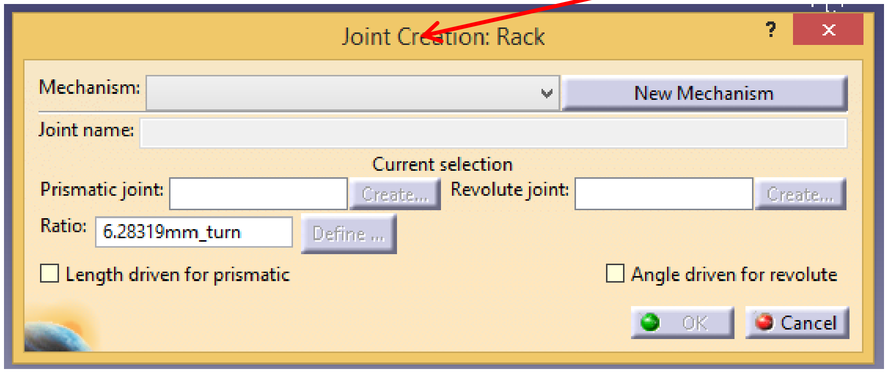


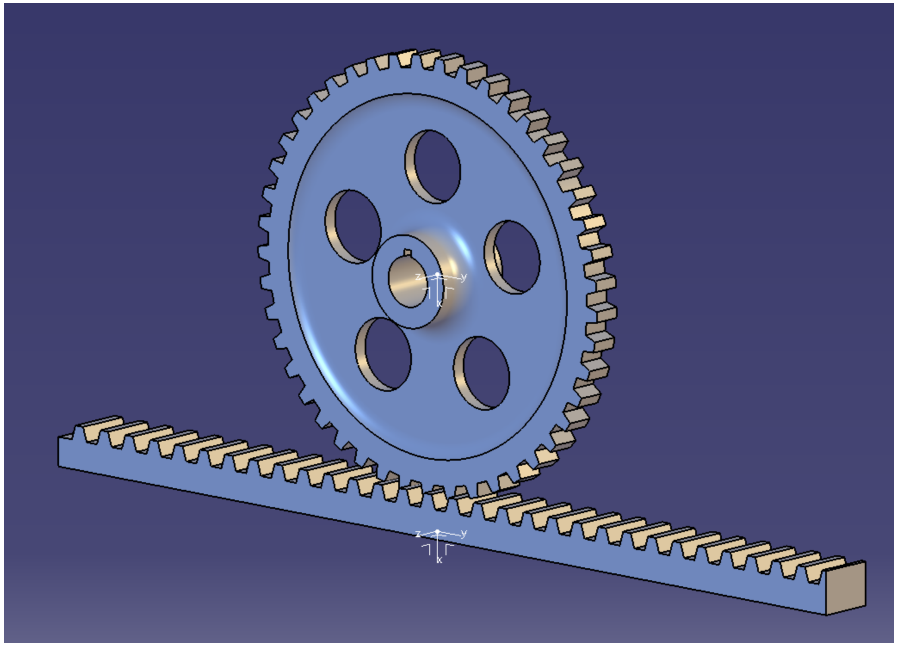


A Gear joint requires 2 revolute joints to exist. They need not be gear shapes, mathematics determines the ratio. Either gear can be driven

Rack Joint

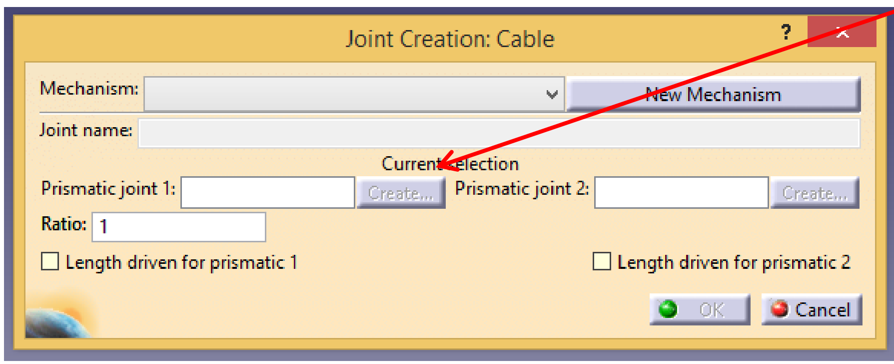


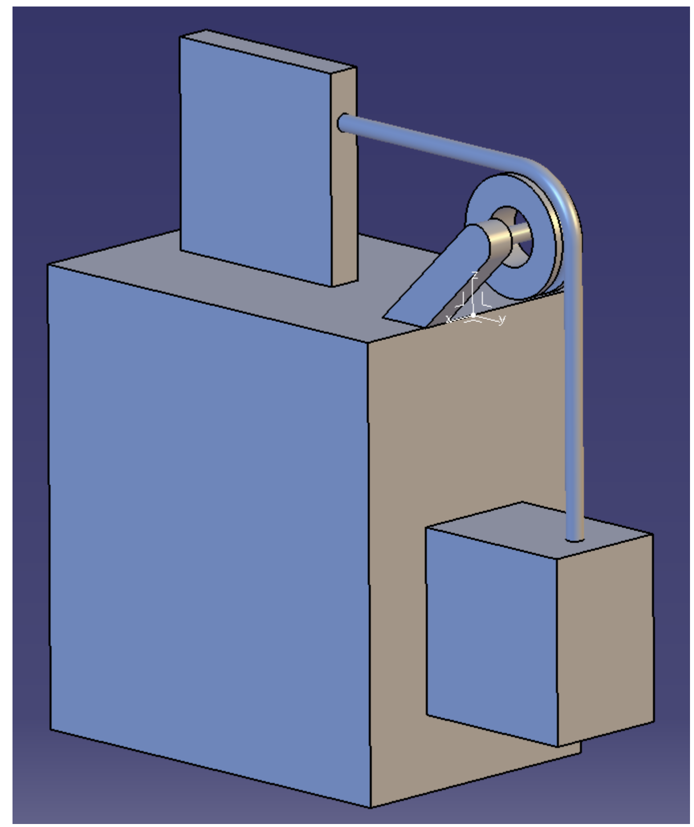




A Gear joint requires a Prismatic joint and a revolute joint to exist. The ratio is a linear distance per revolution. The length or the distance can be driven

Cable Joint

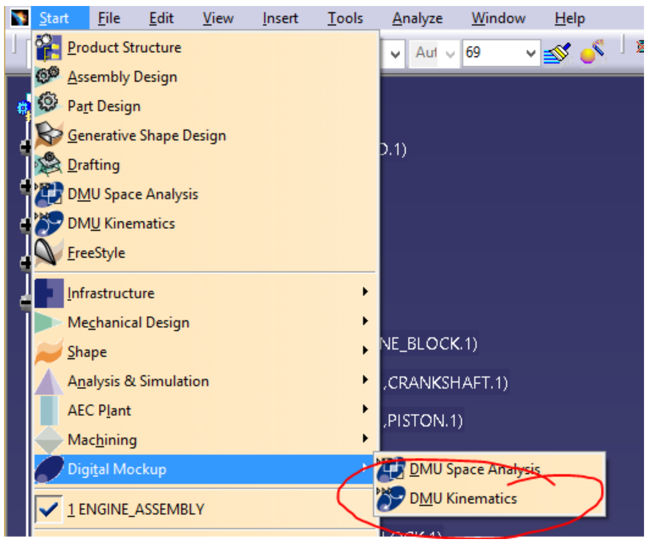


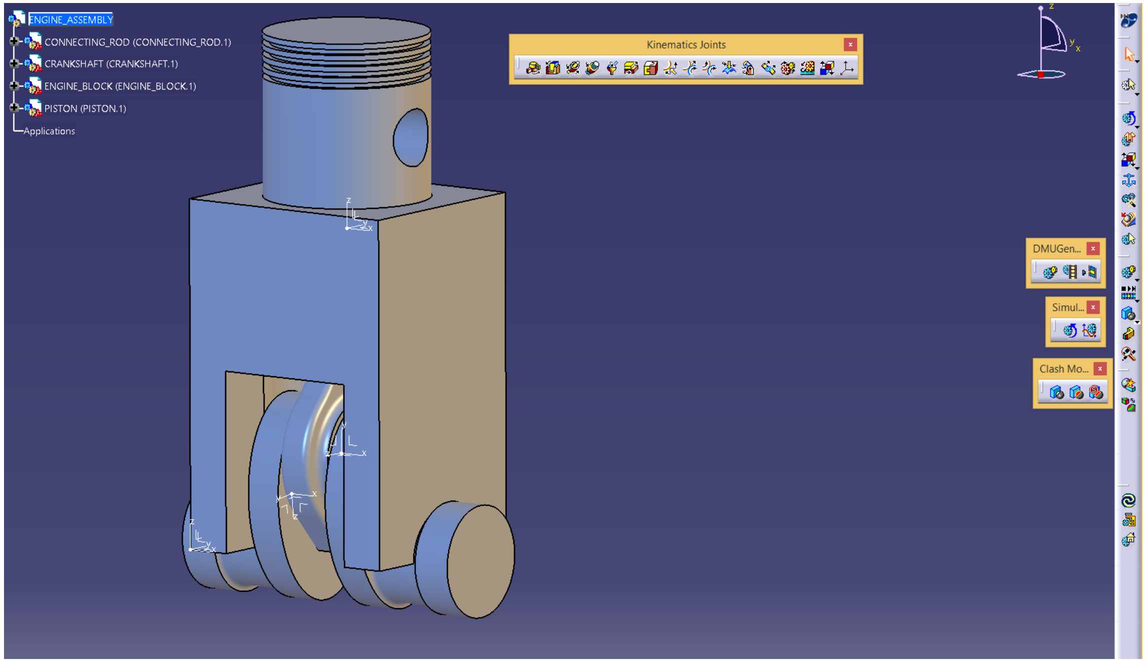


A cable joint requires two prismatic joints. A ratio of 1:1 is the default. The cable and the pulley are not needed or required. Each prism joint can command the motion

Kinematic Start

- If the Constraints are incomplete or missing, Kinematics can be launched through the Start menu
- The Mechanism title will display a rolling count for the degrees of freedom (DOF)
  - Every degree of freedom requires a “Command” to apply a movement
  - DOF = 0 means the machine cannot move on its own. It has no other motion other than what you tell it to do

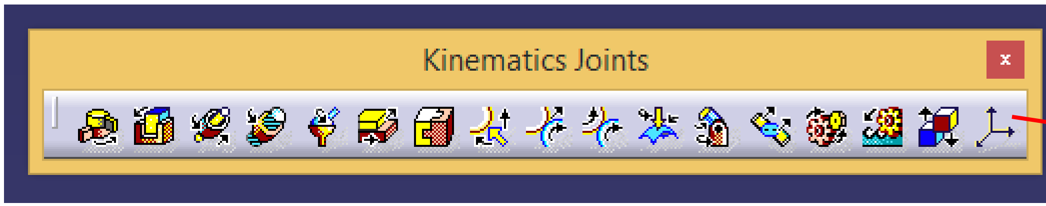




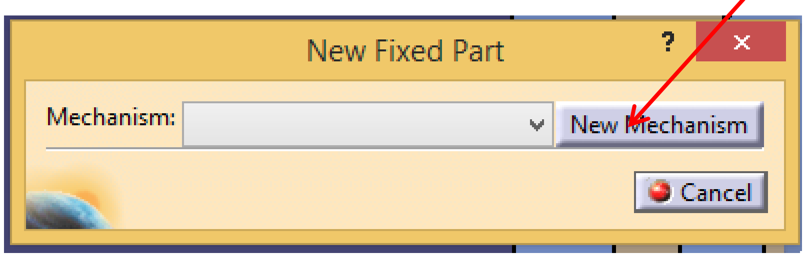
- As with other new workbenches, please explore the icons and arrange to ensure you can see the commands and functions

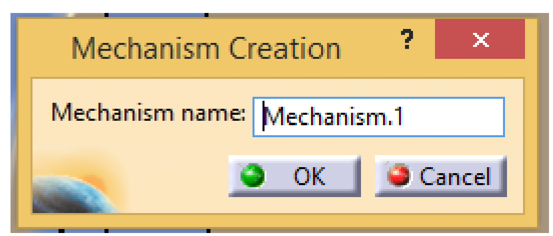
Kinematic Start

- Expanding the functions will reveal all hidden commands especially for the Kinematic joints definition



- The options described provide a condensed illustration to assist what they do in a mechanism
- When starting from scratch, meaning no constraints have been defined, you first declare an anchor or stationary part.
- If no mechanism has been initiated, CATIA will ask you for a name.
  - The anchor function prompts you to first declare a machine name
- Selecting New Mechanism will begin the installation



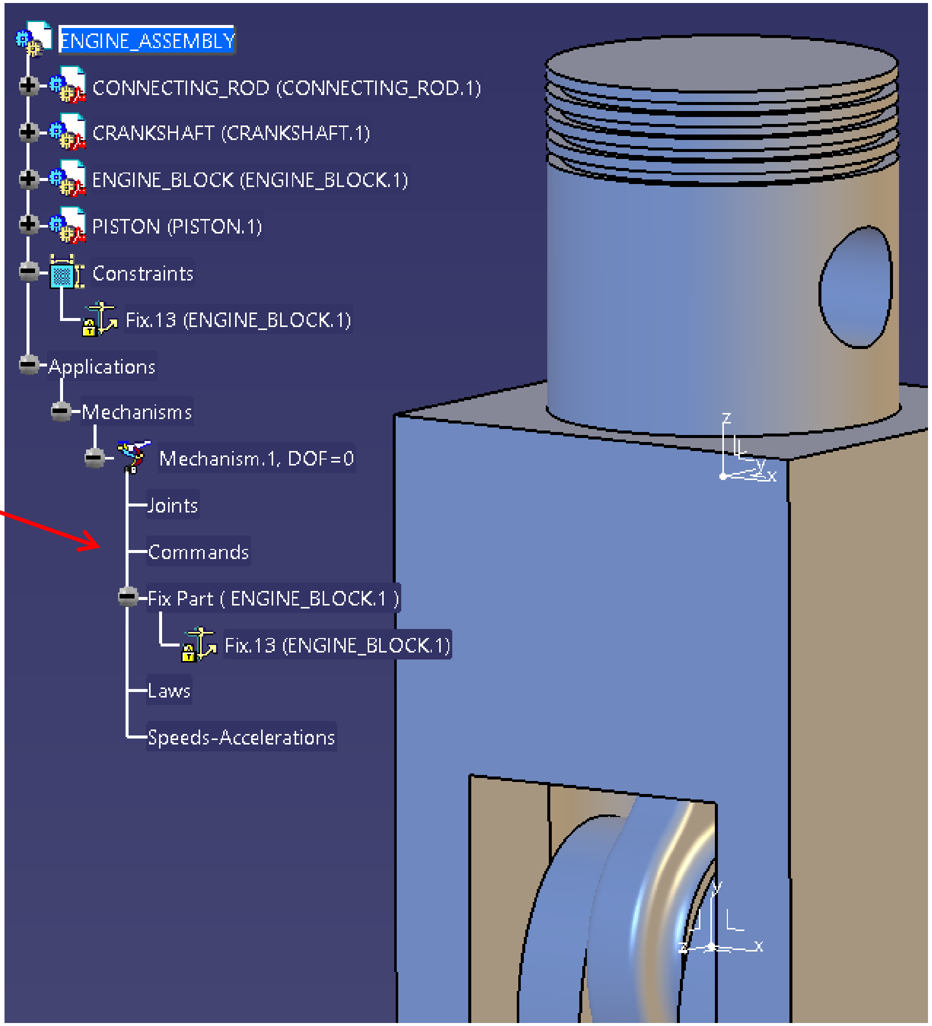


Kinematic Start

- Select a Fixed part after the mechanism has been started and inserted as a new branch into the tree

Mechanism added branch with DOF = 0. The machine cannot move as it is only fixed in one position

Fix part is defined. Expanding the + sign on any branch provides a detailed description



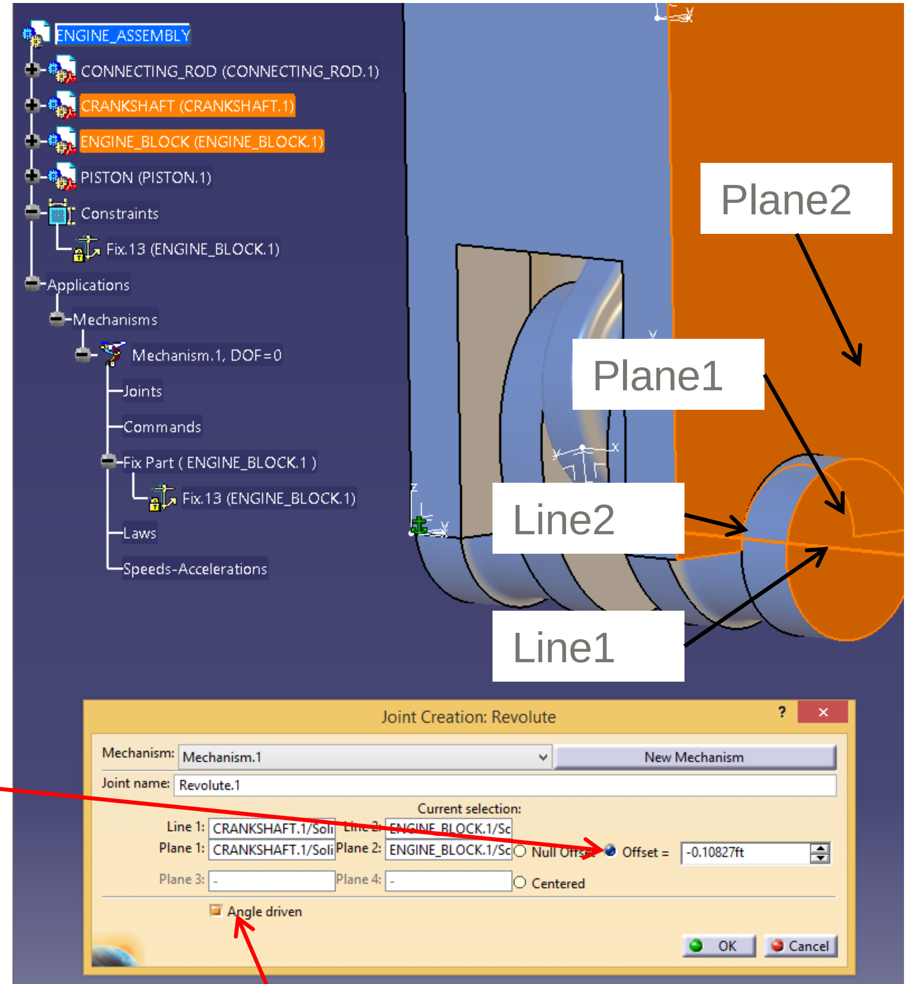
Kinematic Start

- Add a Revolute to define the crankshaft revolving on the engine and maintaining the distance from the engine sidewall

Select the circular crankshaft portion as Line 1, the circular portion removed on the engine block as line 2.

Plane 1 is on the crankshaft end plate and plane 2 is the engine block side wall

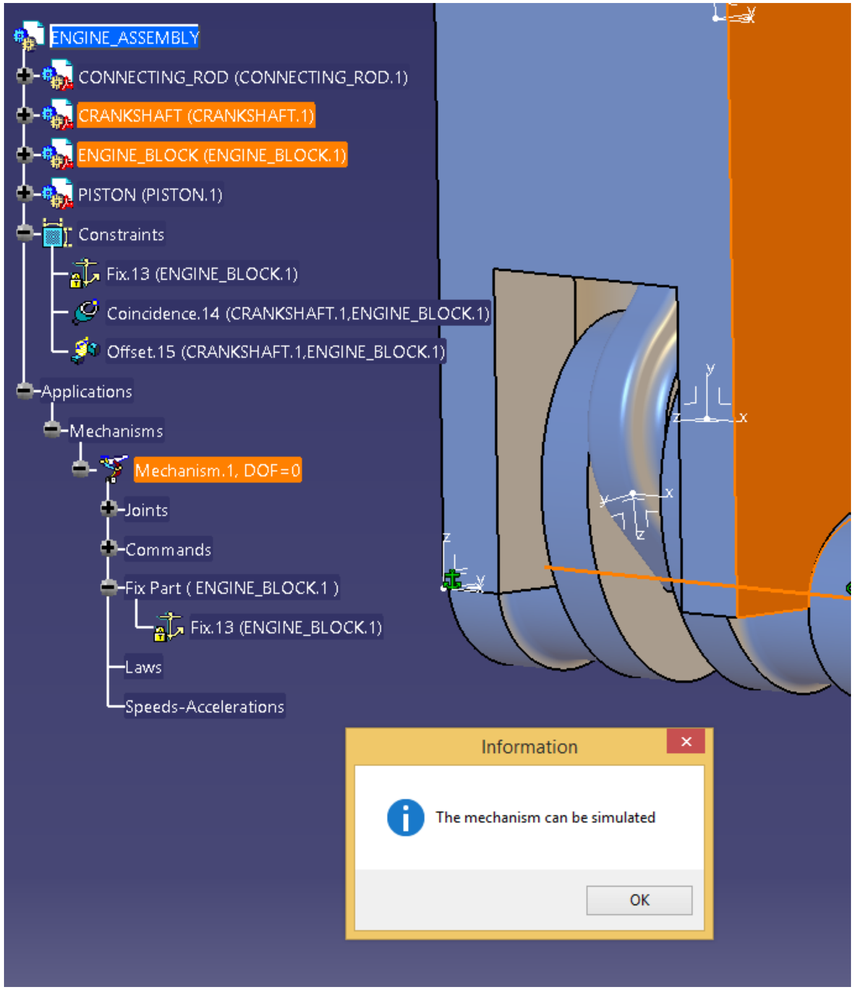
CATIA will assume the plane distance to be Zero. Selecting the Offset button will bring in the current measured distance



We will drive the crankshaft to move the remaining parts

Kinematic Start

- After selecting OK, CATIA will confirm the action and declare if it is currently ready to simulate a motion. If not, the DOF will be a number greater than zero
- In our case, we have one joint which we are driving in a angular motion, combined with a Fixed part, the DOF is 0. The machine can be simulated but only the crankshaft will move.
- We need to add the other parts to include all parts working



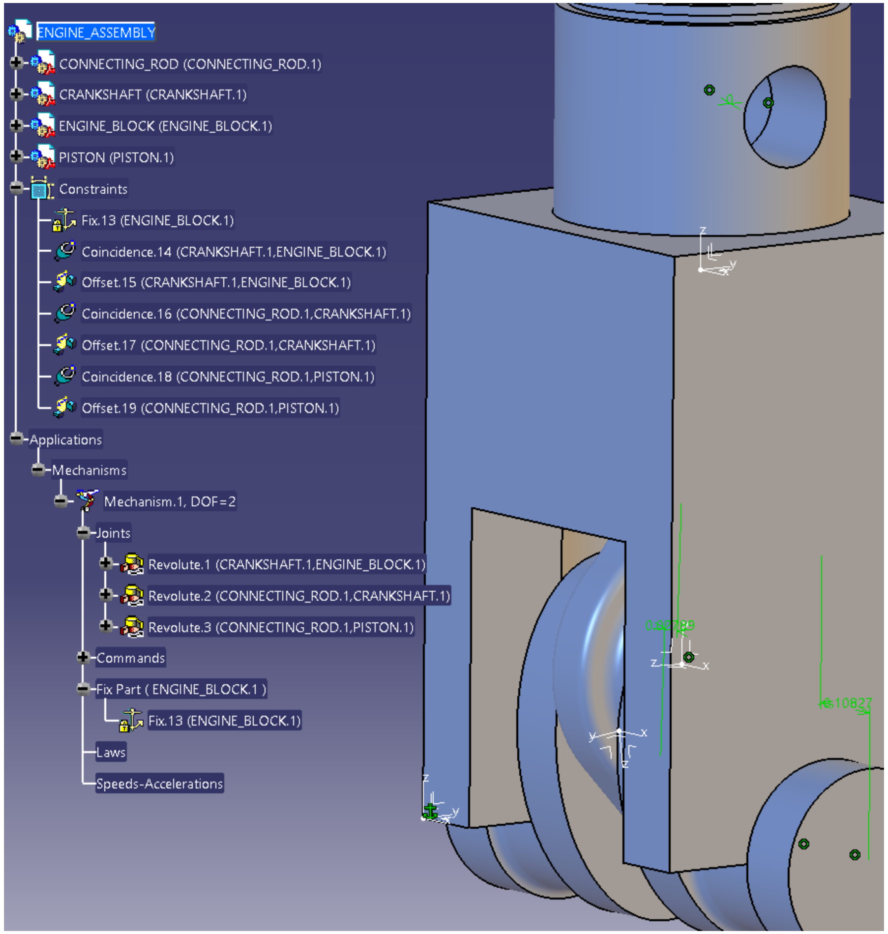
Kinematic Start

- Continue to add kinematic joints similar to those in constraints. As joints are added, the DOF increases and decreases to account for joint followers.
- Expanding tree branches help to see the progress and monitor if the DOF is too high or locked too soon

Three revolution joints have been defined.

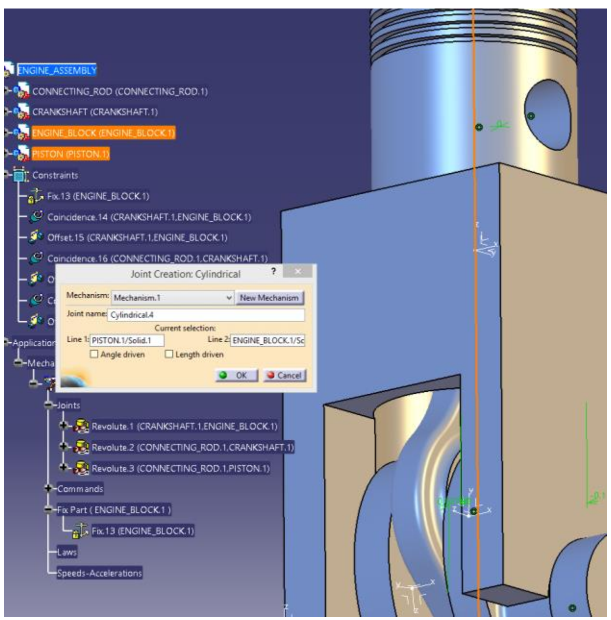
One; the crankshaft and engine block.
Two; the connecting rod and the crankshaft
Three; the piston and connecting rod.

Notice the DOF = 2.
There is no command to control the connecting rod spin on the crankshaft. And there is no command to govern the piston on the connecting rod. Only the crankshaft on the engine block is controlled from step 1

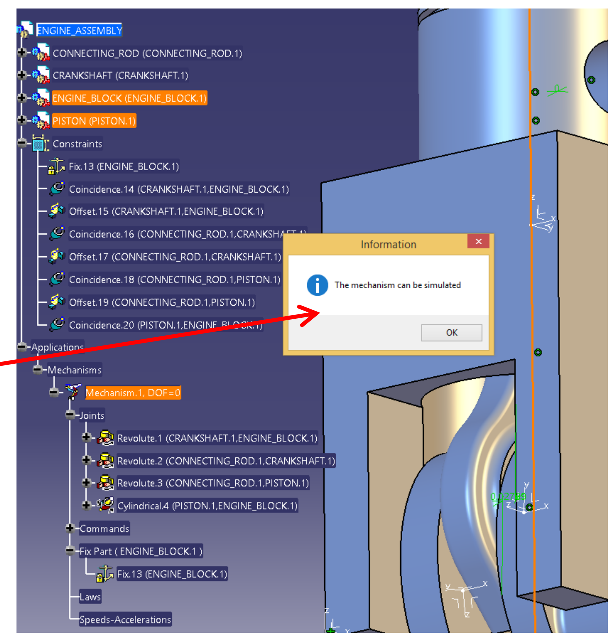


Kinematic Start

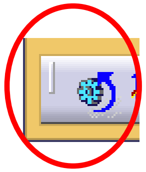
- To control the pistons relationship to the engine block, we have three options
  - Prismatic joint
  - Revolution joint
  - Cylindrical joint
- Prismatic and Revolution require both a line and a plane. The center line of the piston is coincident to the center line on the engine block piston cut-out, but there isn't a plane that can be selected on either
- Cylindrical joint requires a line on each. It can also be driven but we will decline selecting the drive options as we are driving with the crankshaft rotation



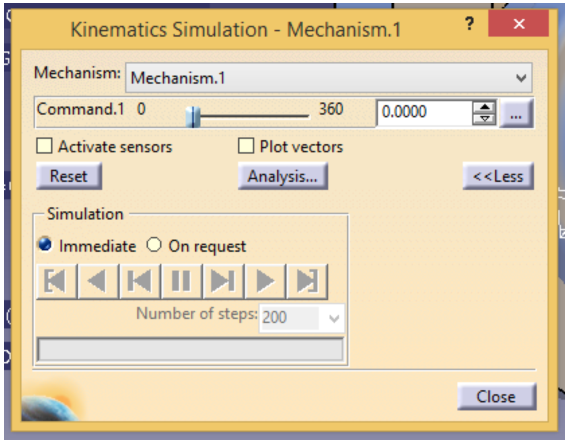
CATIA has enough information to simulate a mechanism motion



Kinematic Simulation

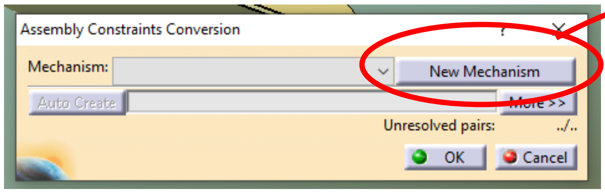


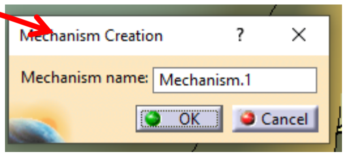
- To simulate a motion, select Simulation with commands
- From the previous slide, the lower limit is 0 and the upper limit is 360
- The simulation defaults to Immediate.
  - As we move the slider from 0 to 360, the machine moves immediately
- On request means we slide the Command.1 from 0-360, the machine remains still until play is pressed.
- The radio buttons describe the motion re-play directions.
- The number of steps is the rate it re-positions the parts as they refresh



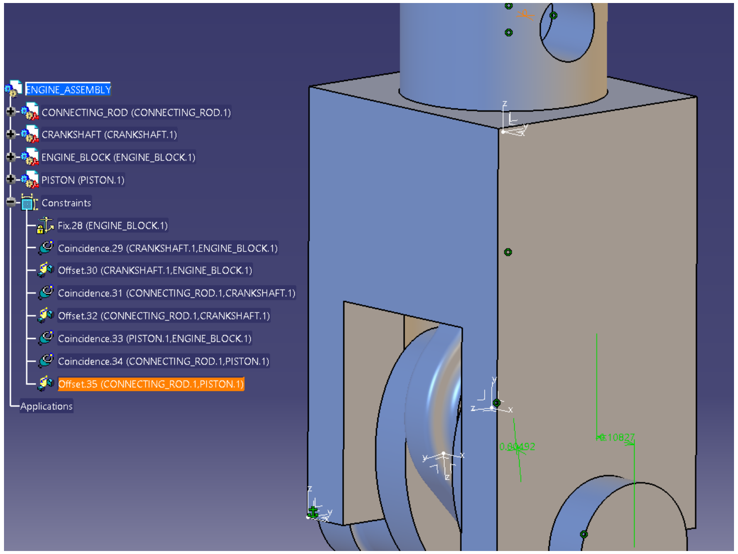
Kinematic Start

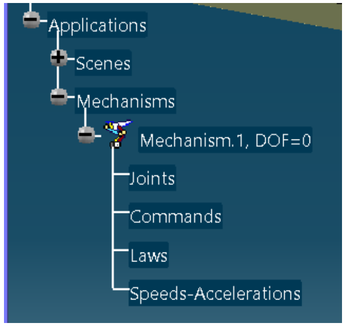
- If Constraints have been properly applied and validated with Manipulating and explode with a correct return, Kinematic motions can be constructed from an existing assembly model using the Assembly Constraints Conversion
- A panel will prompt you to declare a New Mechanism





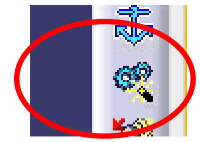
- A new branch called “Mechanisms” is added to the Product tree under an Applications branch
- 5 sub branches are automatically added
  - Joints,
  - Commands,
  - Fix Part, Laws,
  - Speed – Accelerations

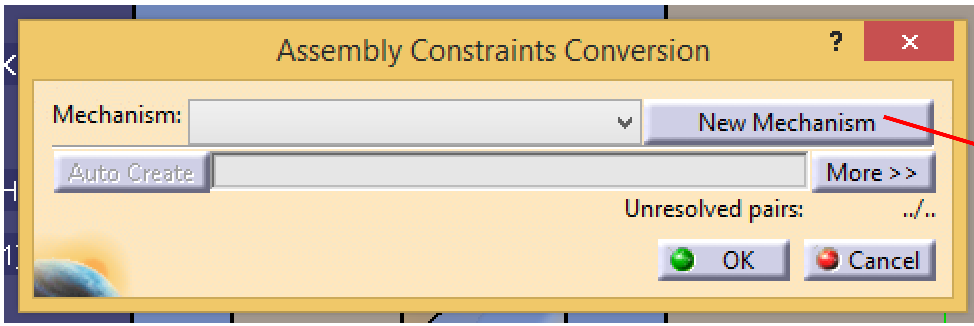


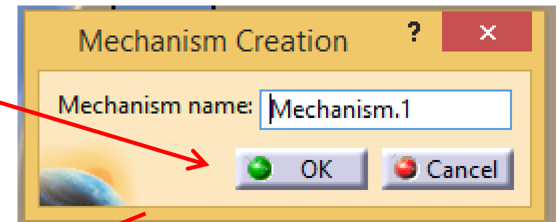


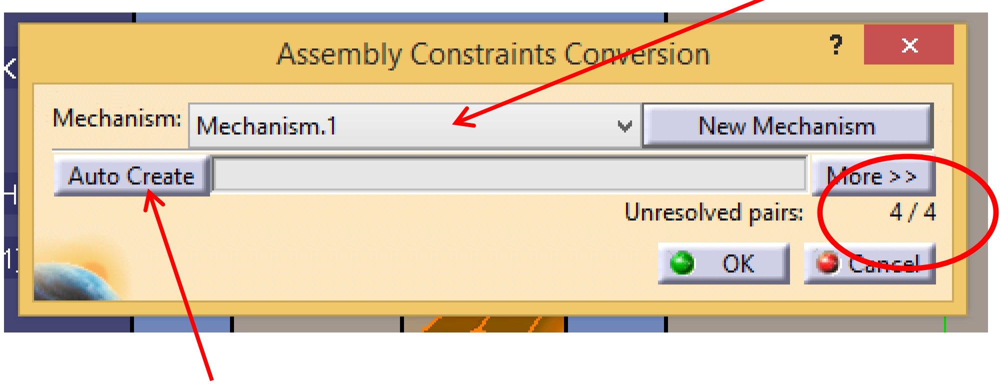
Kinematic Wizard

- Kinematic mechanisms can also be generated from existing assembly constraints using the Assembly Constraints Conversion command
- A mechanism will need to be declared







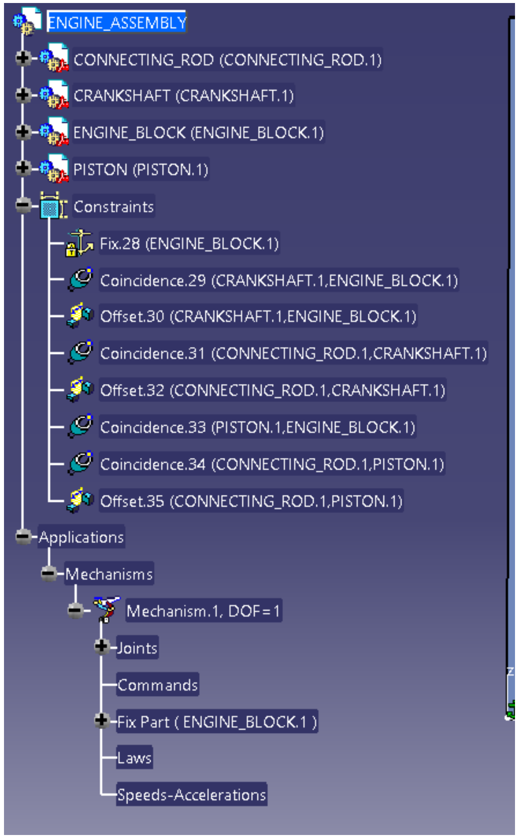


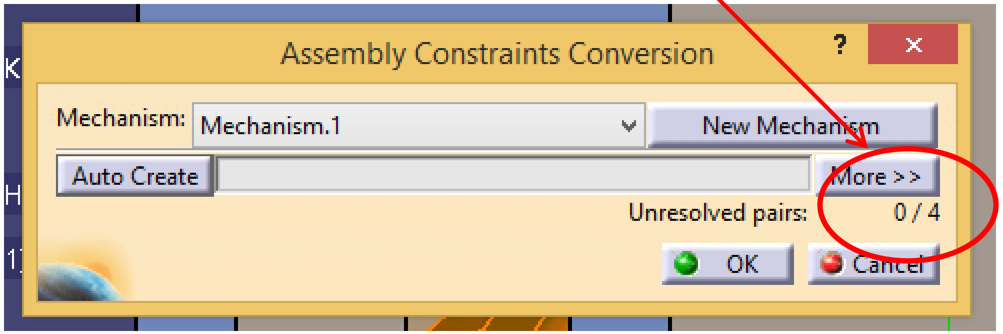
4 joints were discovered from the assembly constraints and 4 have yet to automatically be converted

Select Auto to convert the constraints into equivalent kinematic constraints

Kinematic Wizard

- Auto command converted the assembly constraints leaving 0/4.
- Assembly constraints do not have a driven component.
- Manipulation is needed
- You have to declare which command you wish to drive as DOF = 1



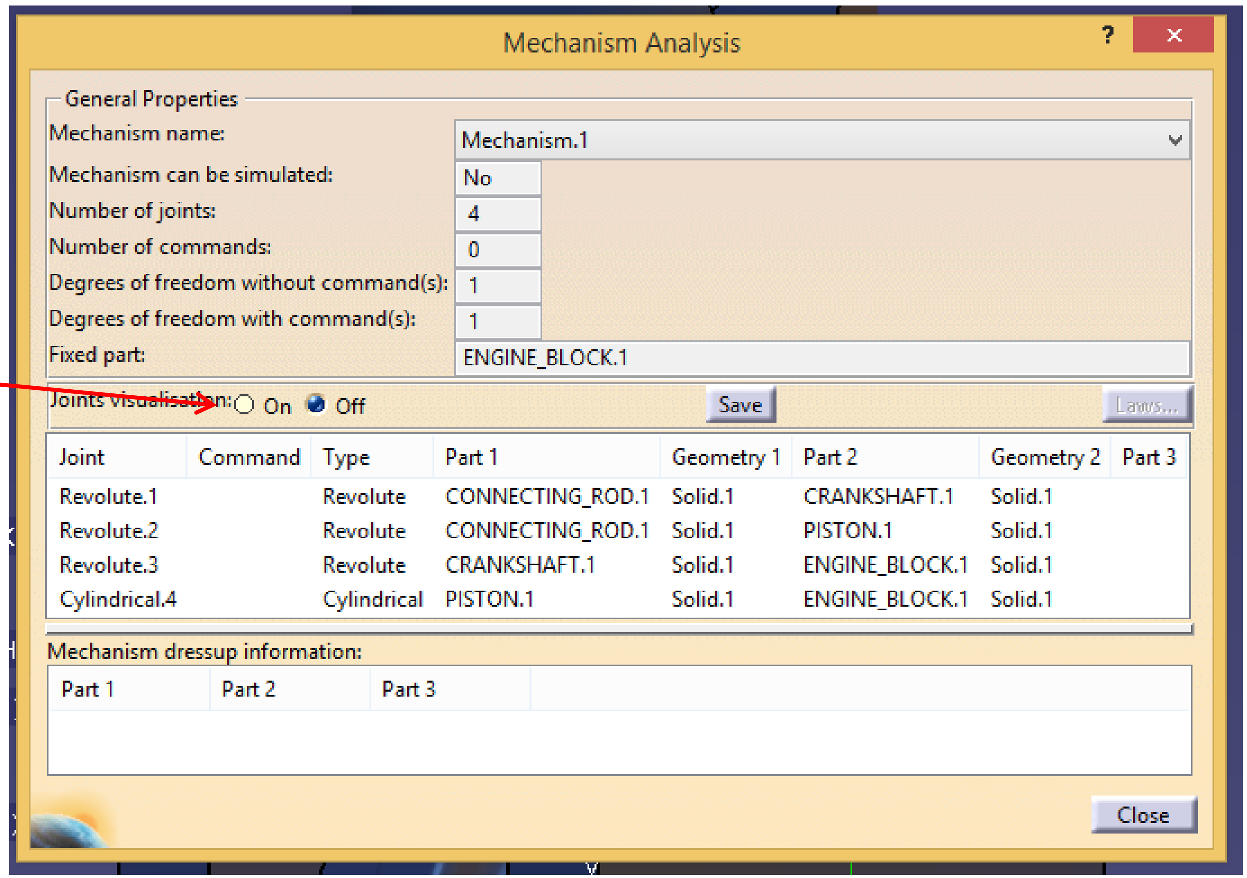


Degree of Freedom =1… It cannot be simulated

Kinematic Wizard

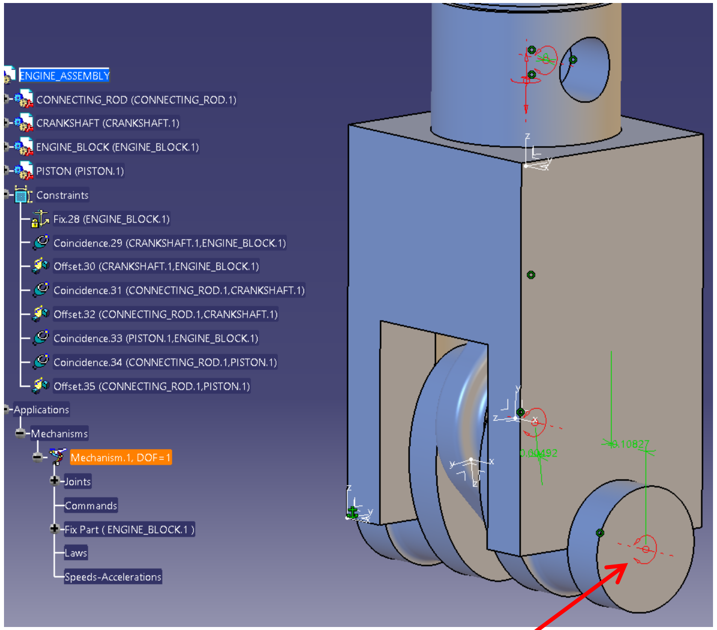
- To visualize the mechanism joints, selecting the Mechanism analysis will display a panel outlining all the joints and attachments

Select Joints Visualization to see the joints CATIA automatically attached



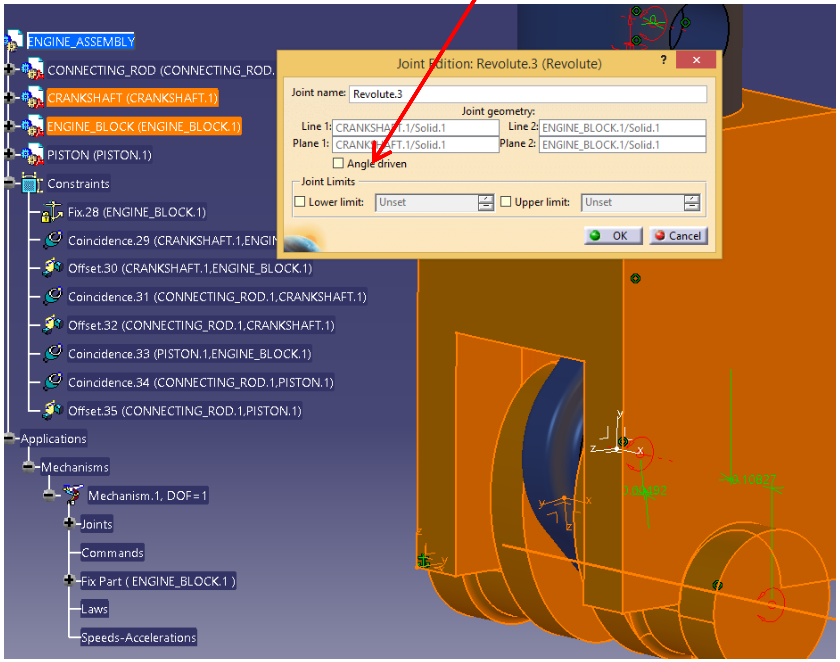
Kinematic Wizard

- The joints in Red are followers. Joints in Green are drivers. Selecting a joint will display the joint information screen and if it can be driven or not



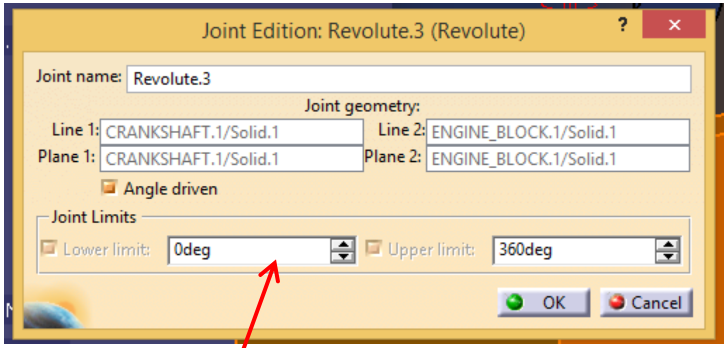
Select the Crankshaft joint to display it's parameter window

The two parts used in the joint are highlighted. Selecting the Angle Driven button will allow this joint to be adjustable

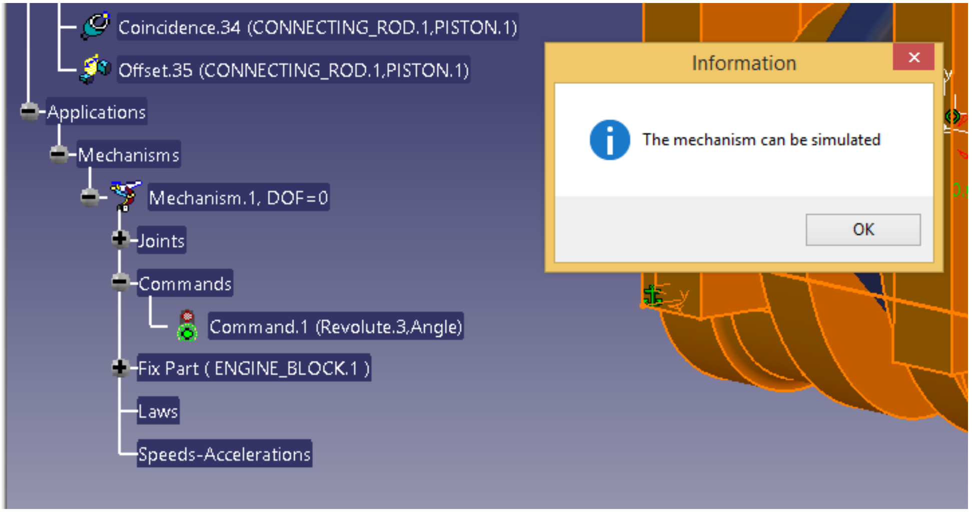


Kinematic Wizard

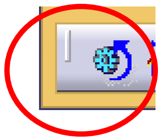
- Limits can be imposed on the machine as well.
- The revolute command for the crankshaft default to -360 to +360.
- For this example, we start at 0 and move to 360
  - Change the number for lower limit to be 0 and upper limit to be 360



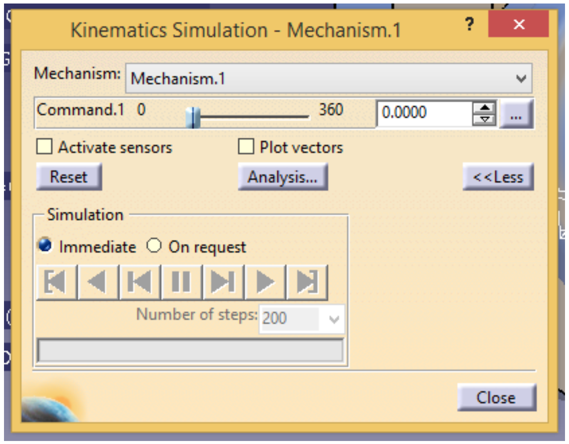
Change limits



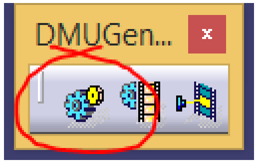
Kinematic Simulation

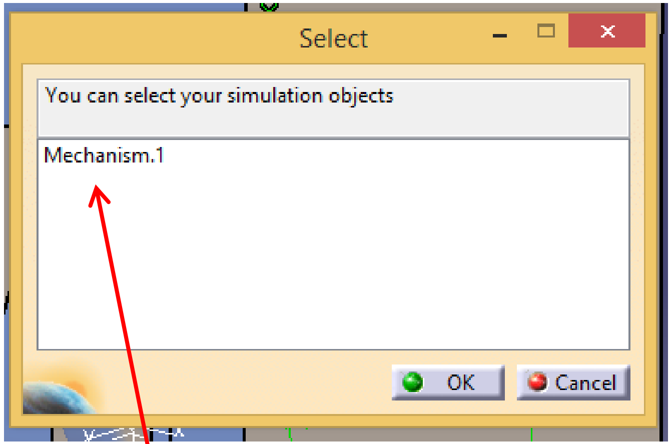


- To simulate a motion, select Simulation with commands
- From the previous slide, the lower limit is 0 and the upper limit is 360
- The simulation defaults to Immediate.
  - As we move the slider from 0 to 360, the machine moves immediately
- On request means we slide the Command.1 from 0-360, the machine remains still until play is pressed.
- The radio buttons describe the motion re-play directions.
- The number of steps is the rate it re-positions the parts as they refresh

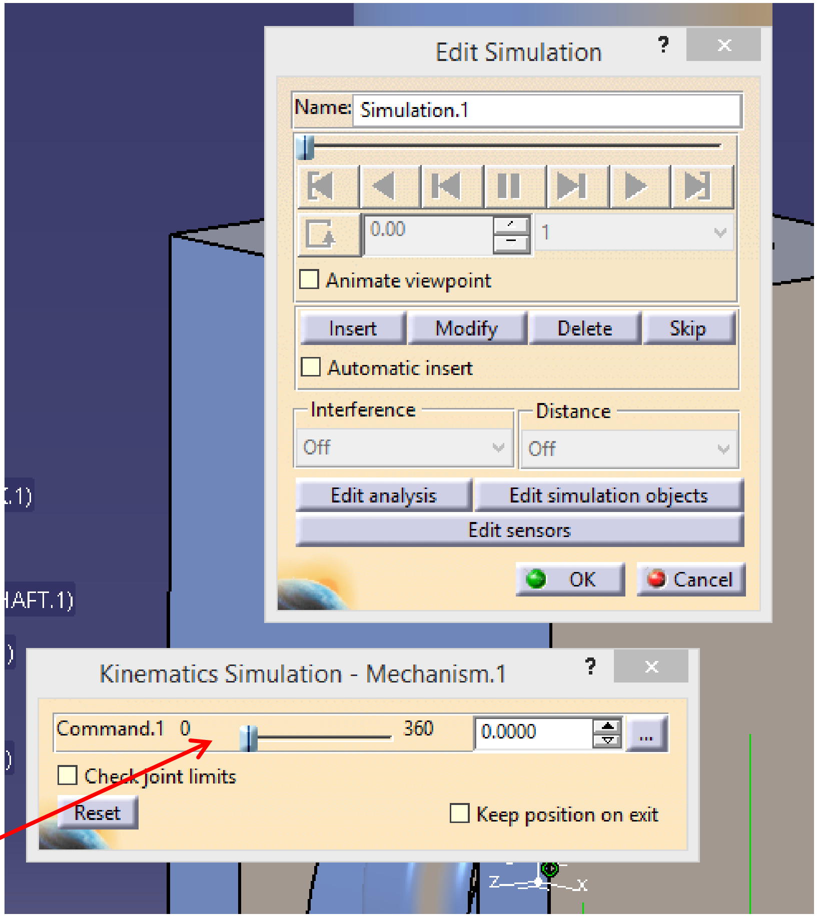


- Simulation with Command is limited to move a machine from one location to another in a linear pace.
- A Simulation is necessary to mimic varied motion steps to move

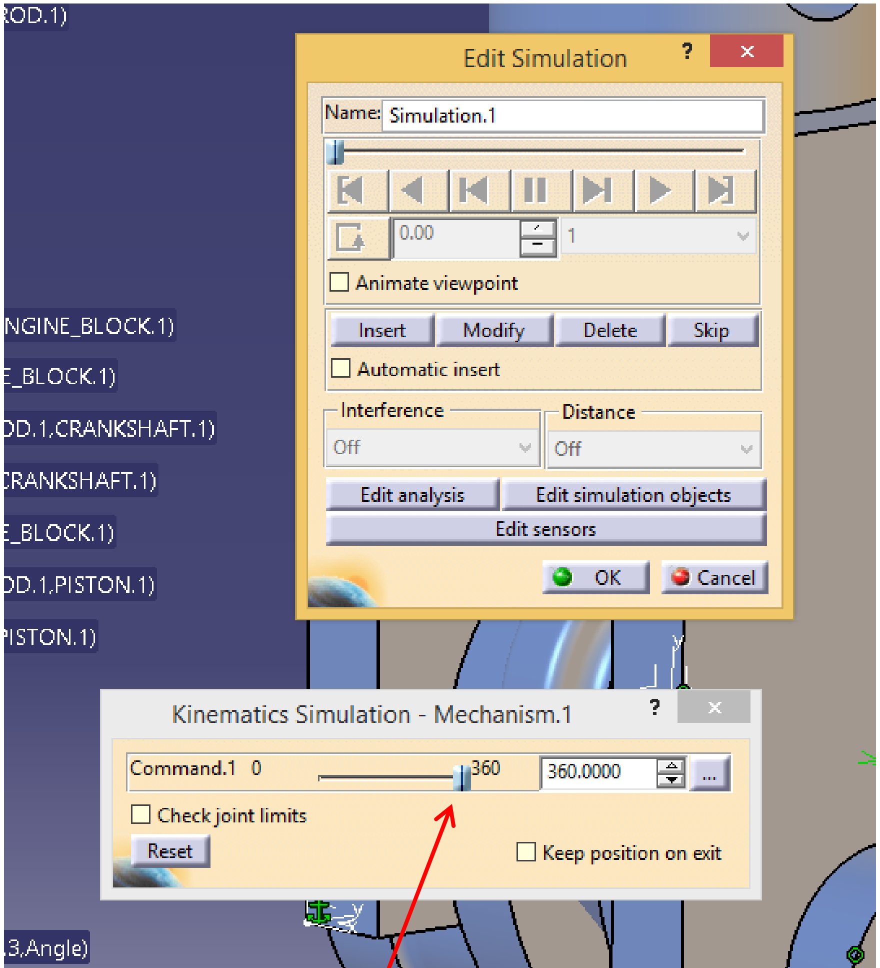




We select the mechanism to record motion

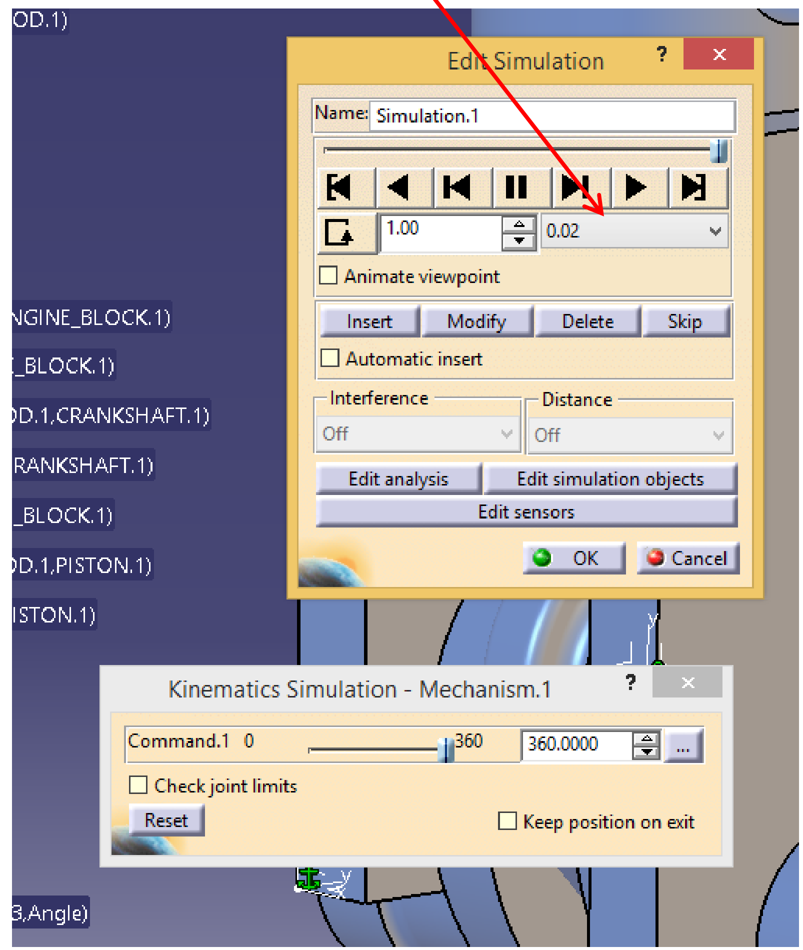


Convert each motion into a compiled simulation



1. Slide the Command to complete 1 cycle

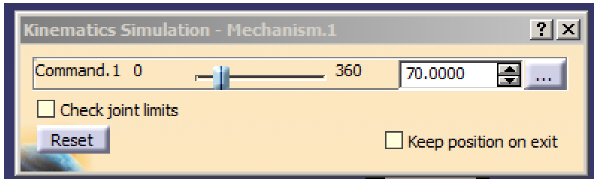
2. Select Insert and adjust the speed you want the machine to play back



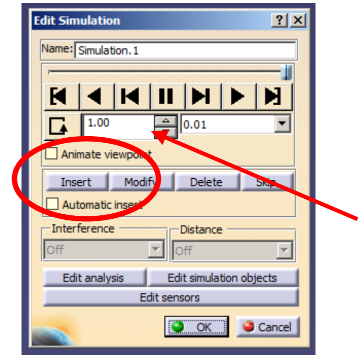
Kinematic Simulation

- Simulation with Commands only provide one motion, one command
- Simulation provides joining multiple motions into one motions
- Each command is inserted with a speed attachment
  - This example will program 0 to 360 degree
    1. Move forward from 0 to 70 degrees at 0.01 speed
    2. Move backward to 50 degrees at 0.2 speed
    3. Move forward to 120 at 0.01 speed
    4. Move backward to 90 at 0.2
    5. Move forward to 360 at 0.01 speed

- 1

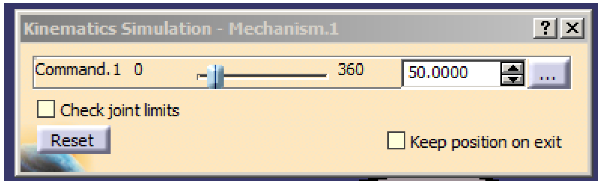


Slide from 0 to 70



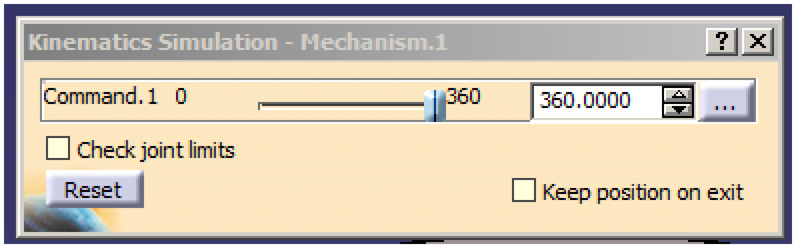
We require the piston to move from 0 to 70, then stop. Insert as Step 1

- 2



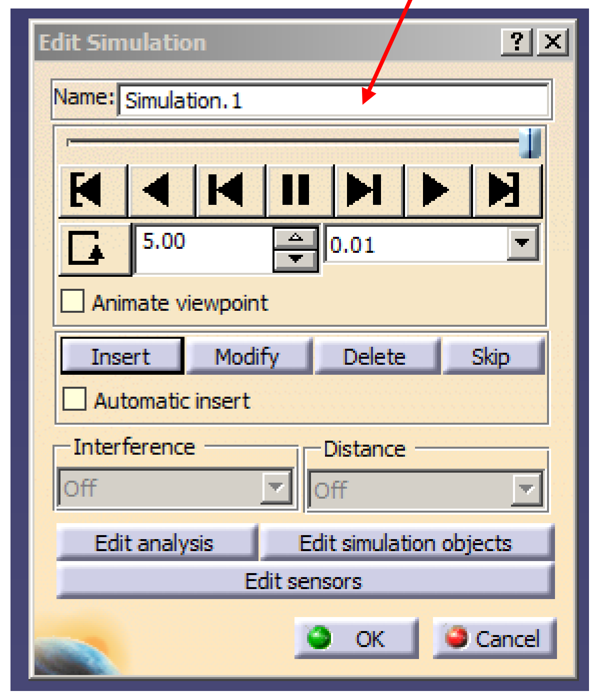
Move the slider back to 50, insert as Step 2, adjust the speed to 0.2

- Continue moving the slider forward and backward to each desired setting, select Insert after each stopped motion with the accompanying speed
- Each step is listed in the Simulation panel



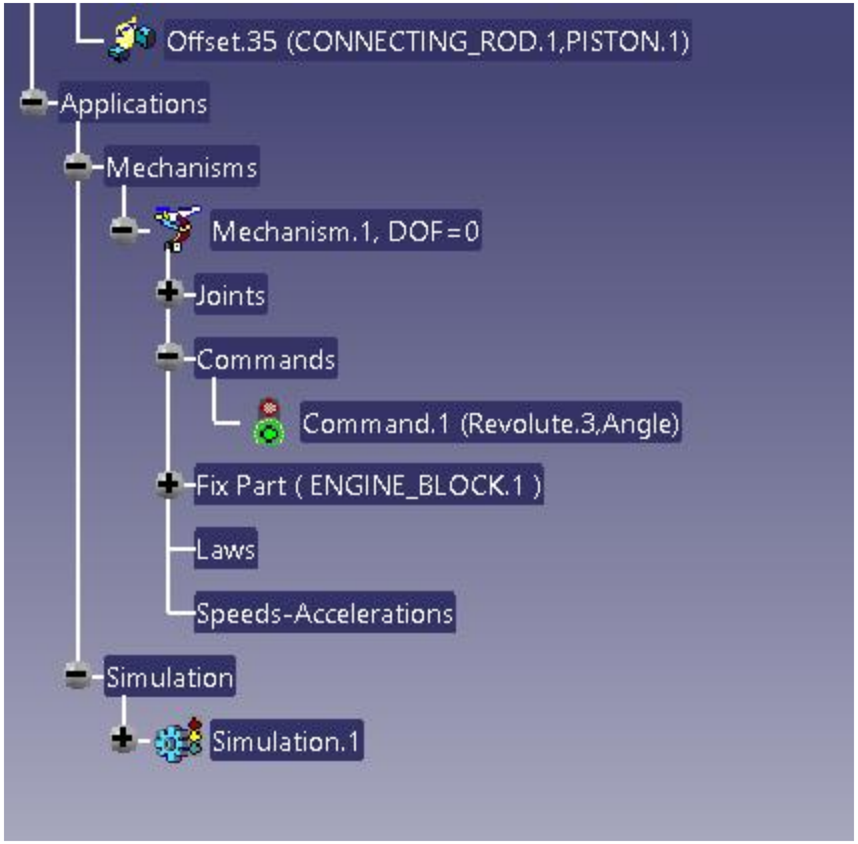
The slider controls the placement the mechanism moves. After each placement, the Simulation panel is insert with its position

Use the player commands to simulate the motion through each inserted step



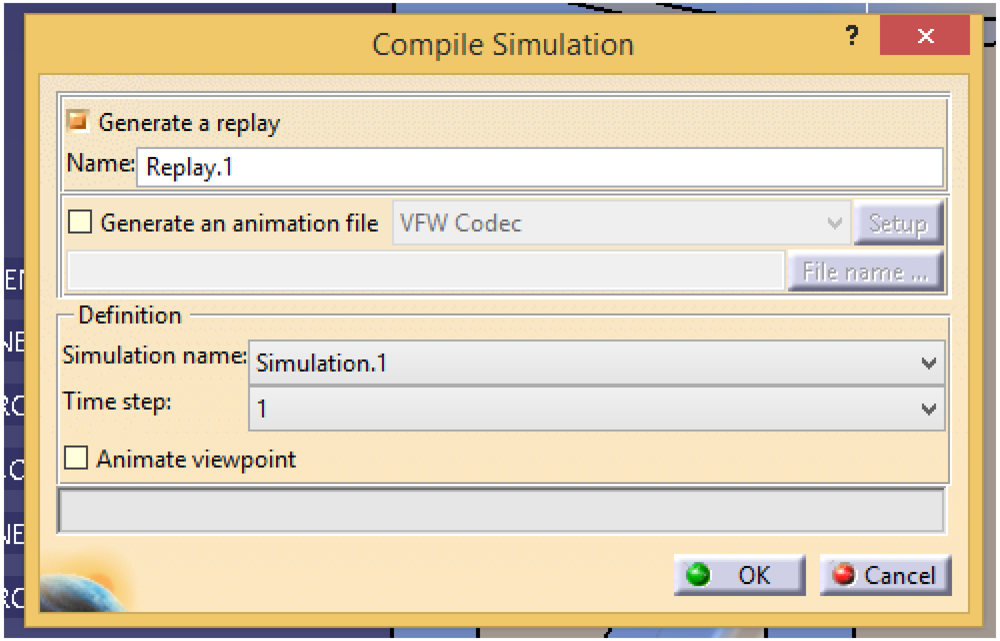
Kinematic Simulation

- The simulation can continue recording Command.1 motions. Select Insert to store each inserted start and stop position
- The tree adds another branch to denote a simulation has been inserted



Kinematic Replay

- To condense all the simulations in a replay, we convert them with Compile Simulations
- When generating a replay, we can further adjust the speed
- An animation file can also be obtained

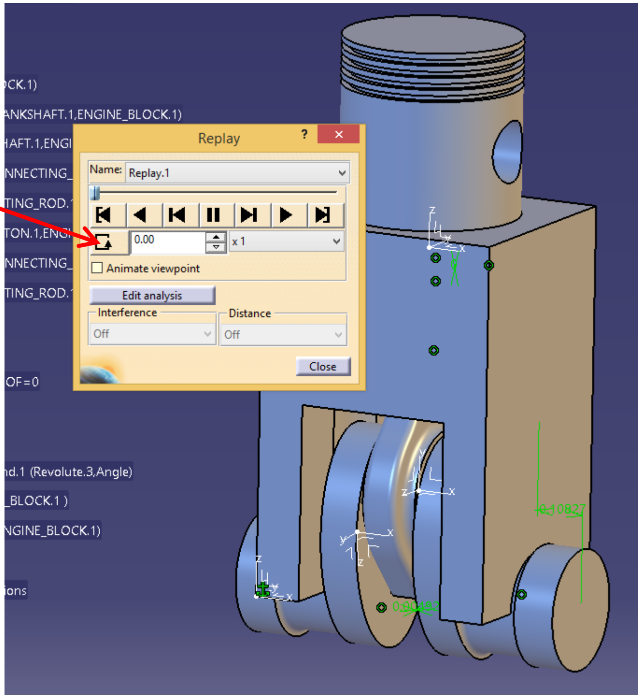


- A new branch is added to the tree to indicate a reply

Kinematic Replay

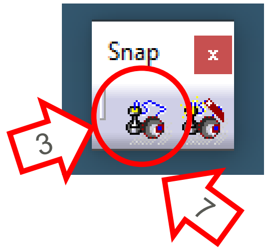
- The replay contains all the information to simulate the motion.
- All information to model the kinematic motion can be deleted if necessary, however, not advisable.
- The replay command can be made to continue in a loop, start and stop in reverse or just one cycle
- You needn't be in the kinematic workbench to run the reply

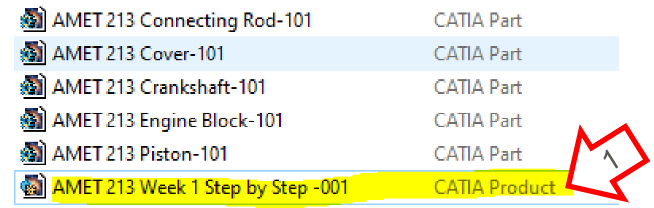
Loop modes

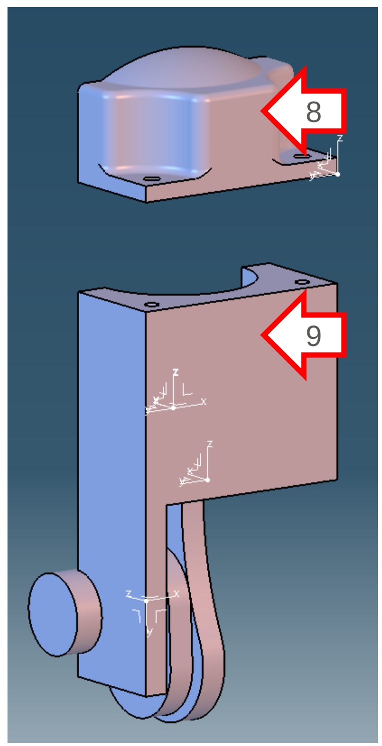


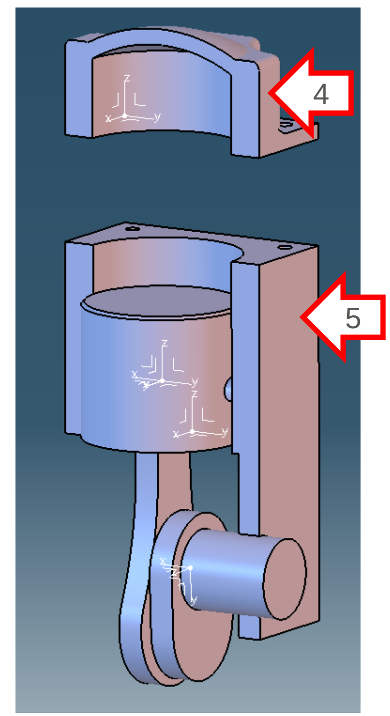
Step by Step

1. Open AMET 213 Week 1 Step by Step -001.CATProduct
2. Change workbench to Assembly Design
3. Select Snap
4. Select Face 1
5. Select Face 2
6. Press Esc
7. Select Snap
8. Select Face 3
9. Select Face 4
10. Press Esc



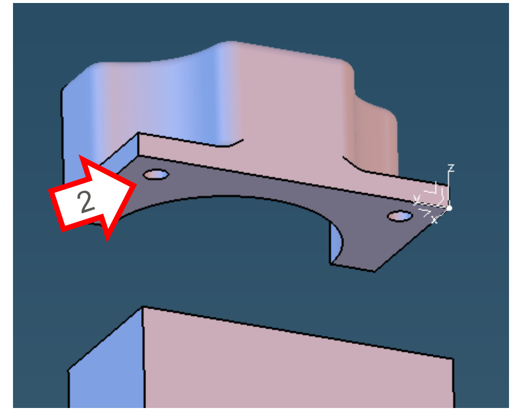


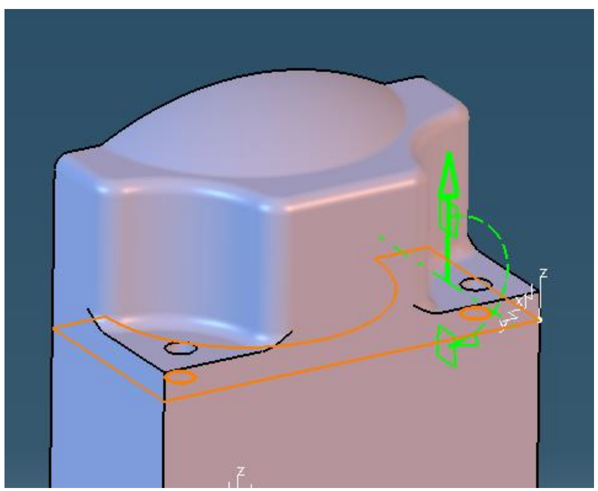


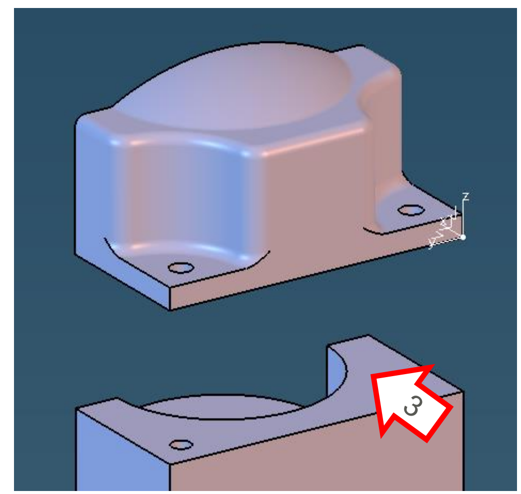


Step by Step

1. Select Snap
2. Select Face 6
3. Select Face 7
4. Select Esc

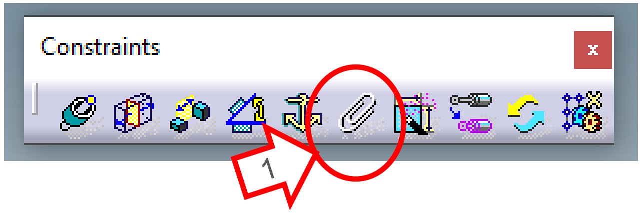


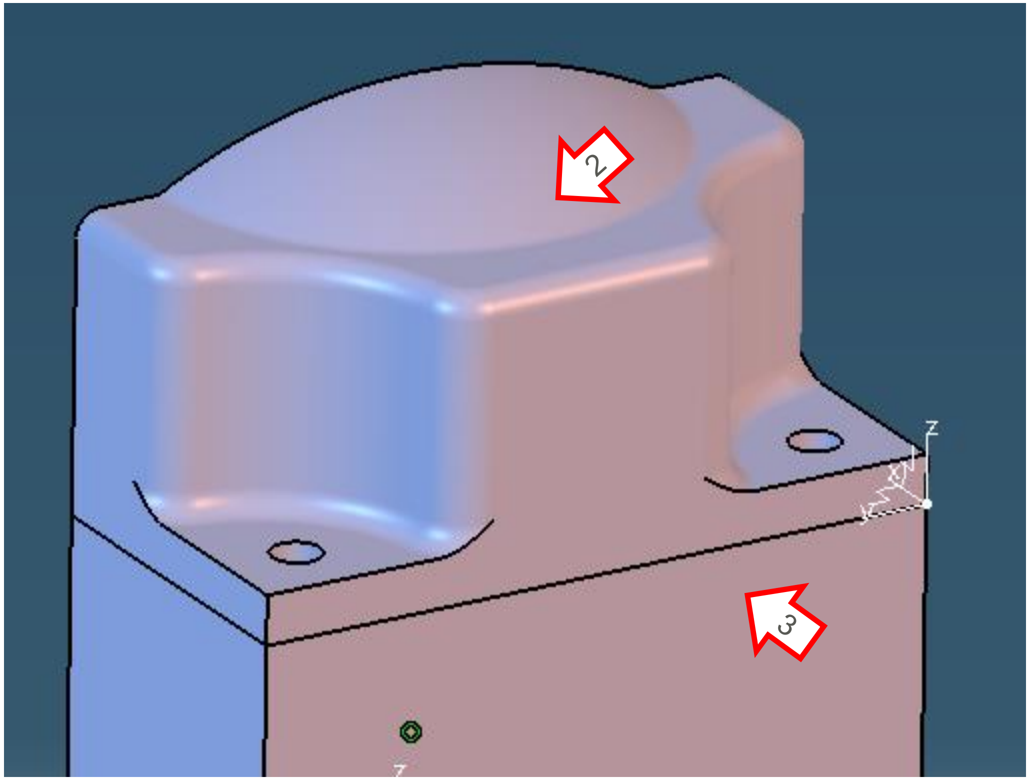


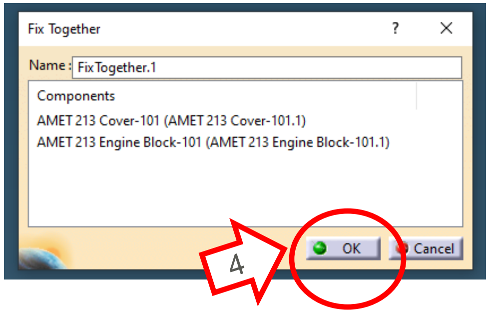


Step by Step

1. Select Fix Together
2. Select Cover
3. Select Engine Block
4. Select OK

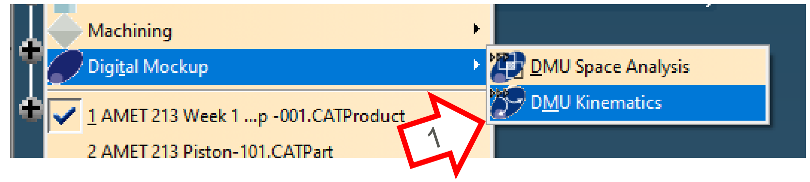


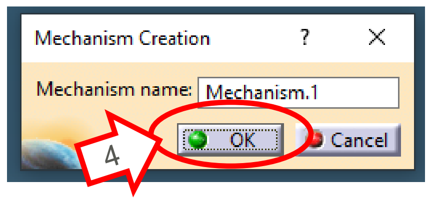


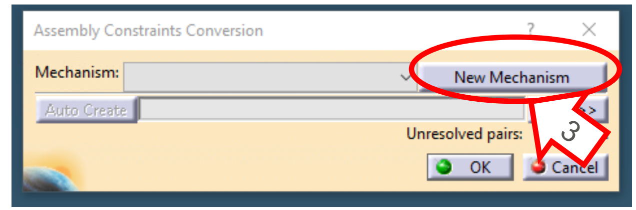


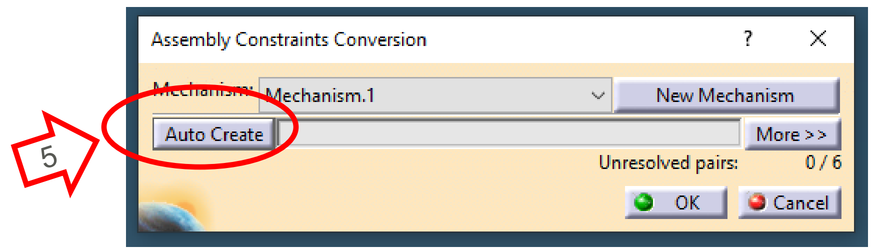
Step by Step

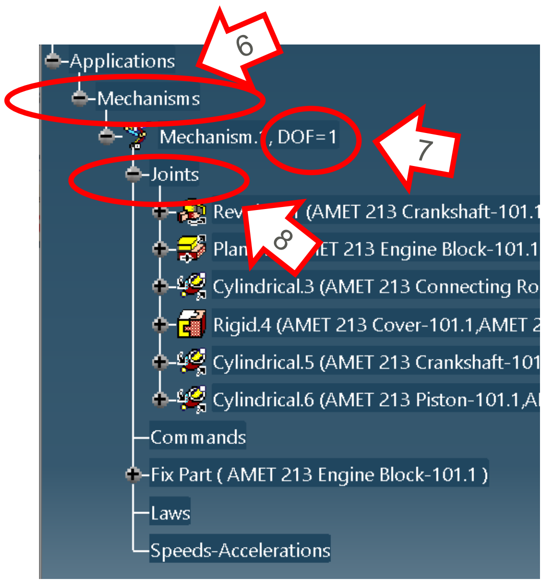
1. Select DMU Kinematics
2. Select Assembly Constraints Conversion
3. Select New Mechanism
4. Select OK
5. Select Auto Create
6. Expand the Mechanism Branch
7. Notice DOF=1
8. Expand Joints Tree





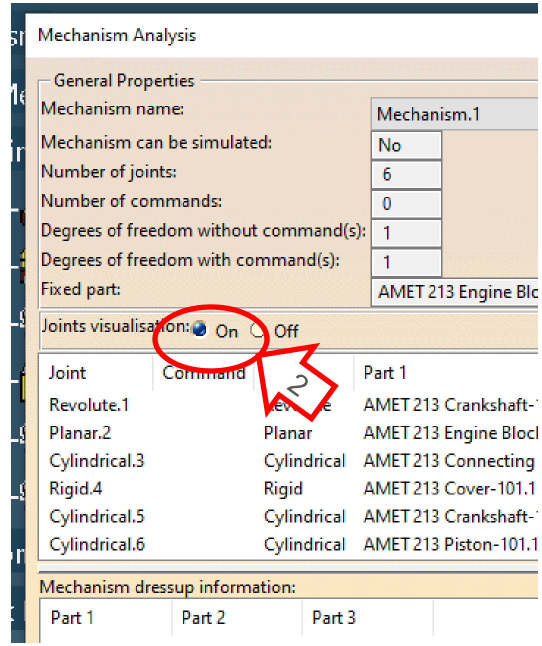


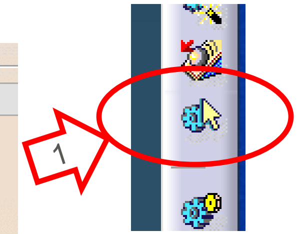


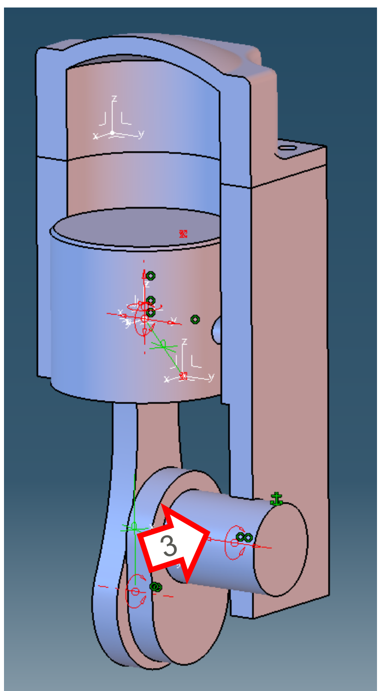


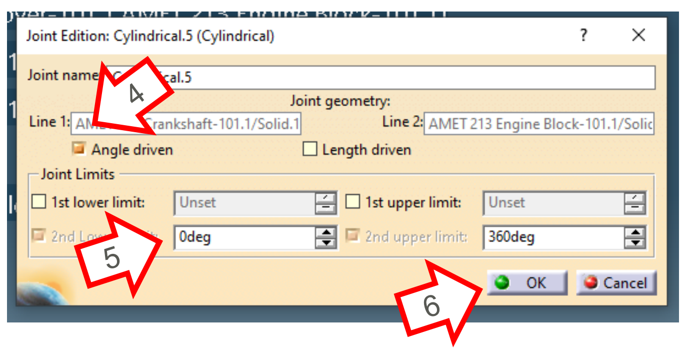
Step by Step

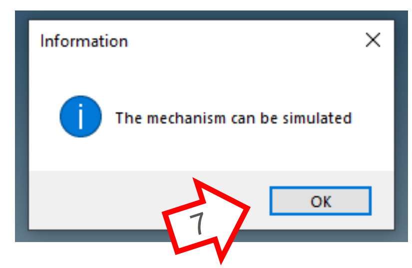
1. Select Mechanism Analysis
2. Select Joint Visualization “On”
3. Double select Revolute joint on Crankshaft
4. Select Angle Driven
5. Change lower limit to 0 degree
6. Select OK
7. Select OK





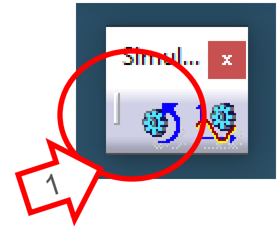


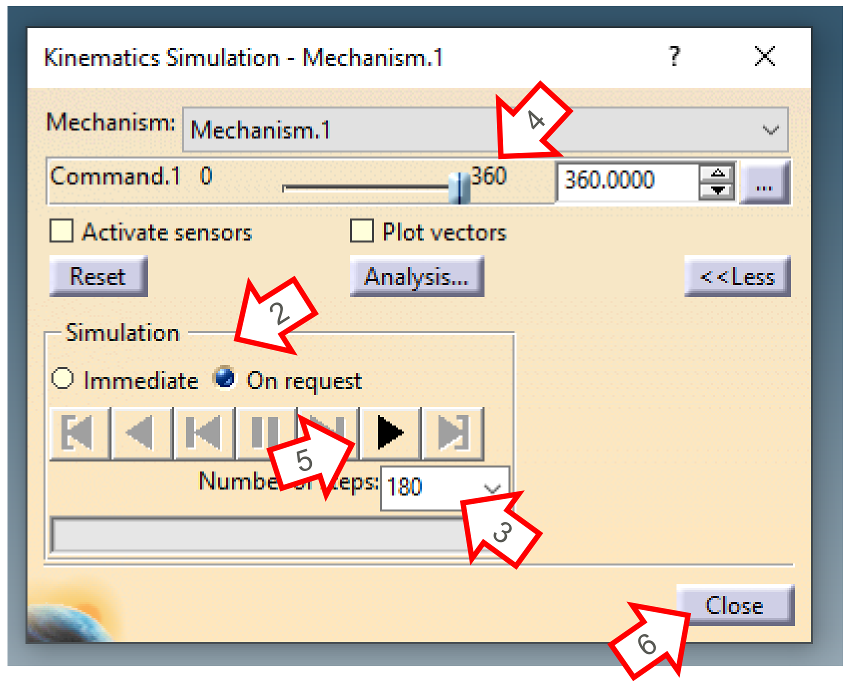




Step by Step

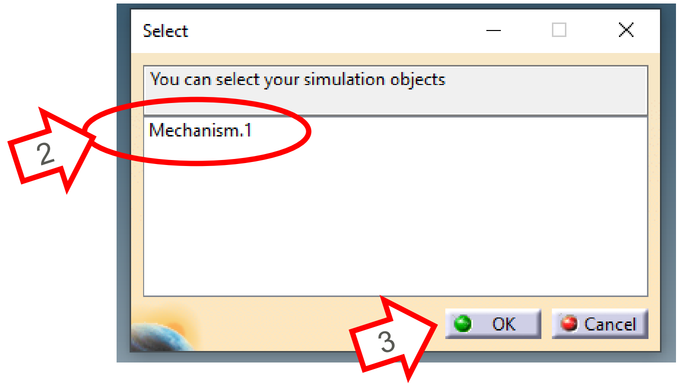
1. Select Simulation with Commands
2. Select On Request
3. Change Number of Steps to 180
4. Move slider to 360
5. Select Play
6. Select Close

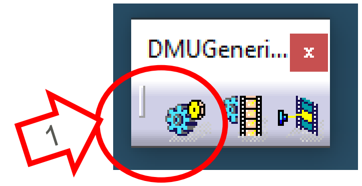


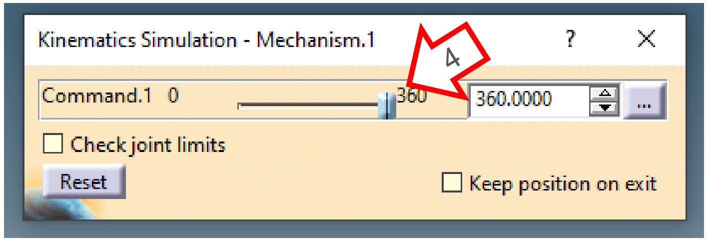


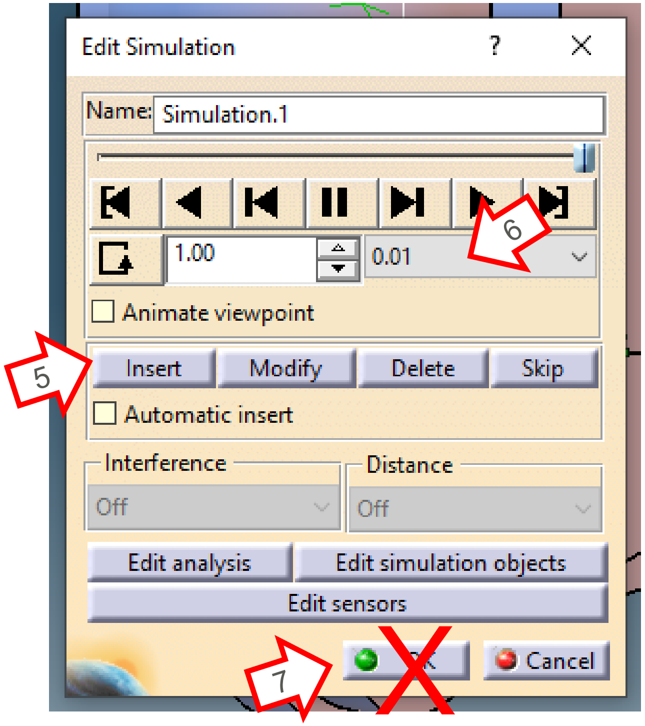
Step by Step

1. Select Simulation
2. Select Mechanism.1
3. Select OK
4. Move slide to 360
5. Select Insert
6. Change speed to 0.01
7. Do Not Select OK



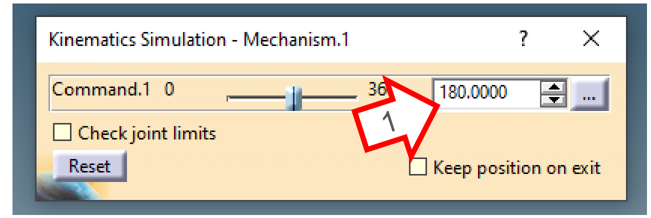


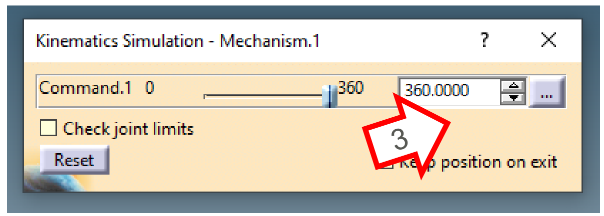


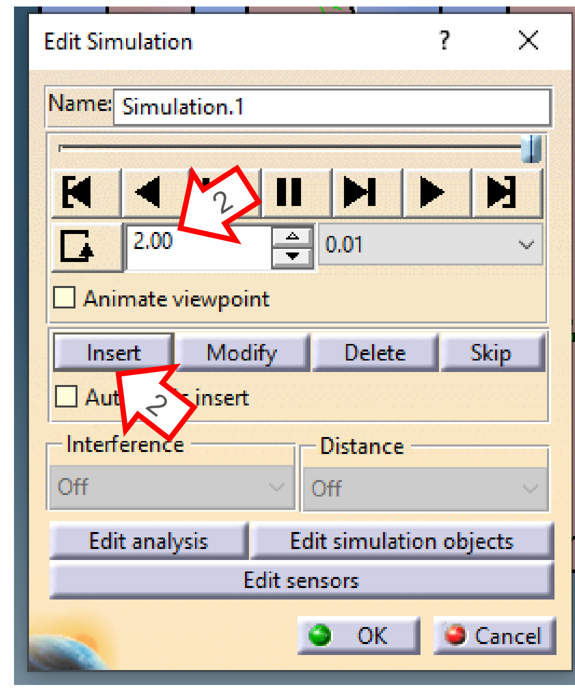


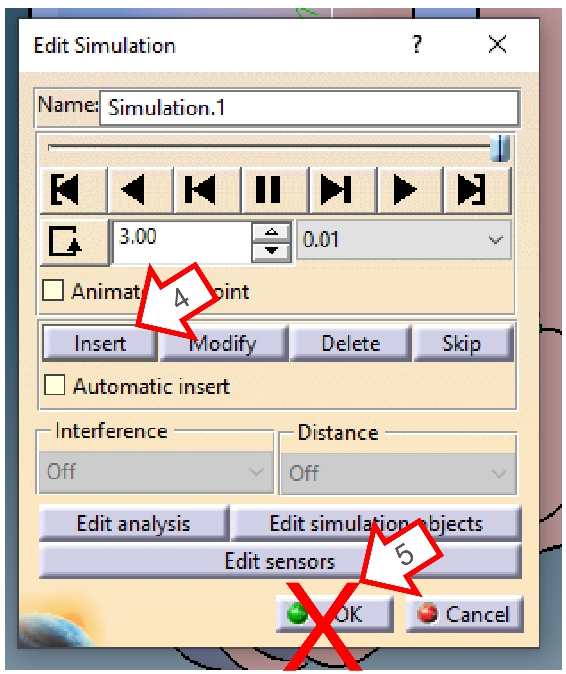
Step by Step

1. Key in slider value to 180
2. Select Insert (Notice insert number = 2)
3. Key in slider value to 360
4. Select Insert
5. Do not select OK





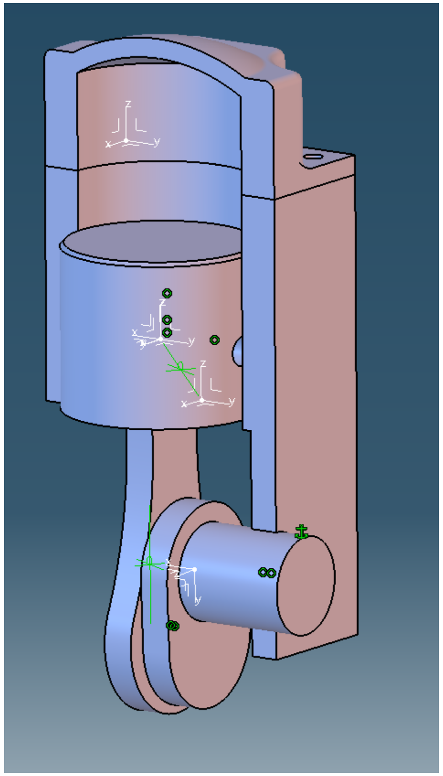


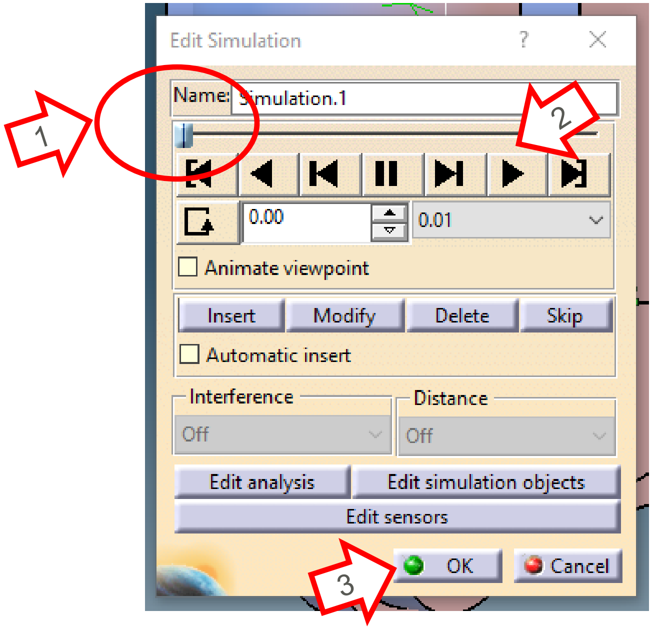


1. Move the slider to maximum left
2. Select Play

- Notice the crankshaft makes one complete revolution, then reverse for 180 degrees then reverse again for home position

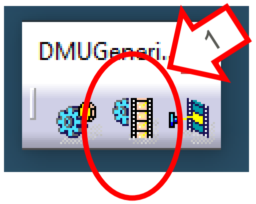
3. Select OK

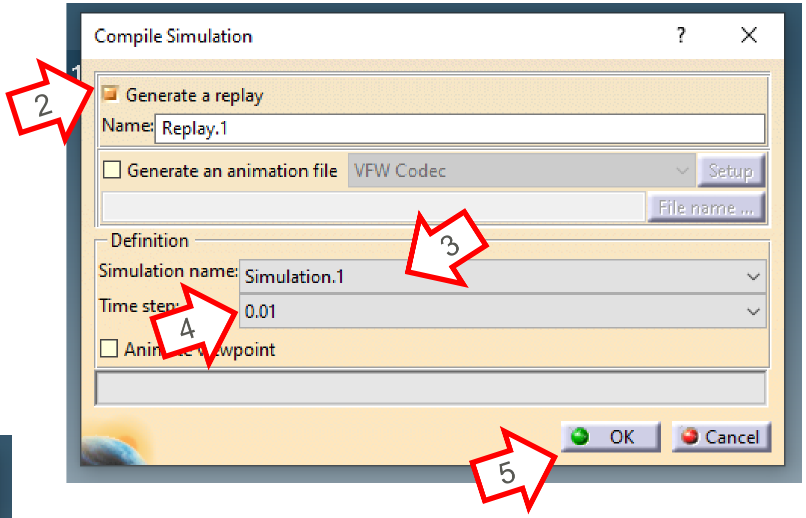


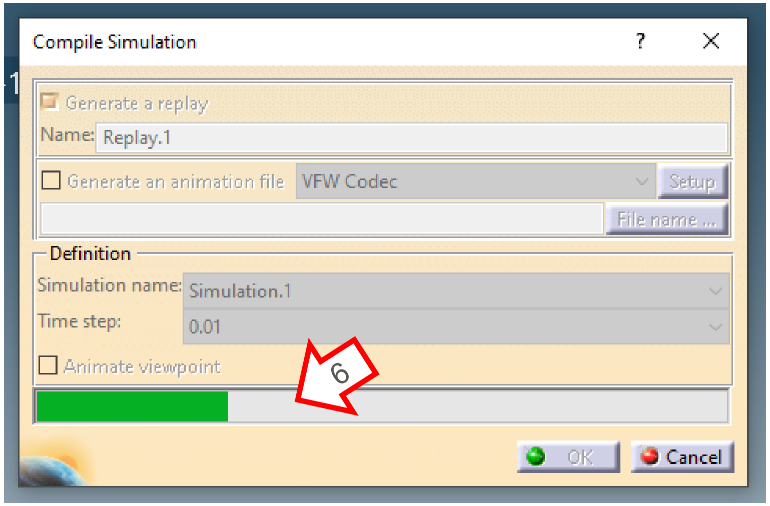


Step by Step

1. Select Compile Simulation
2. Ensure Generate a Replay is selected
3. Ensure Simulation name is Simulation.1
4. Change speed to 0.01
5. Select OK
6. The process will compile after OK

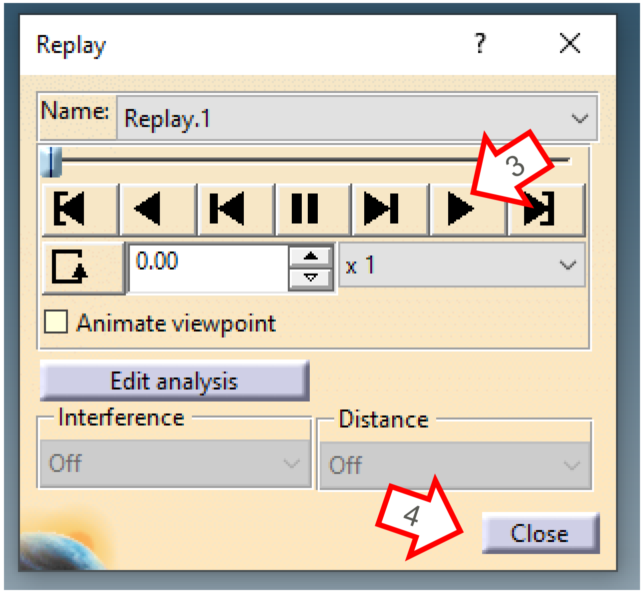


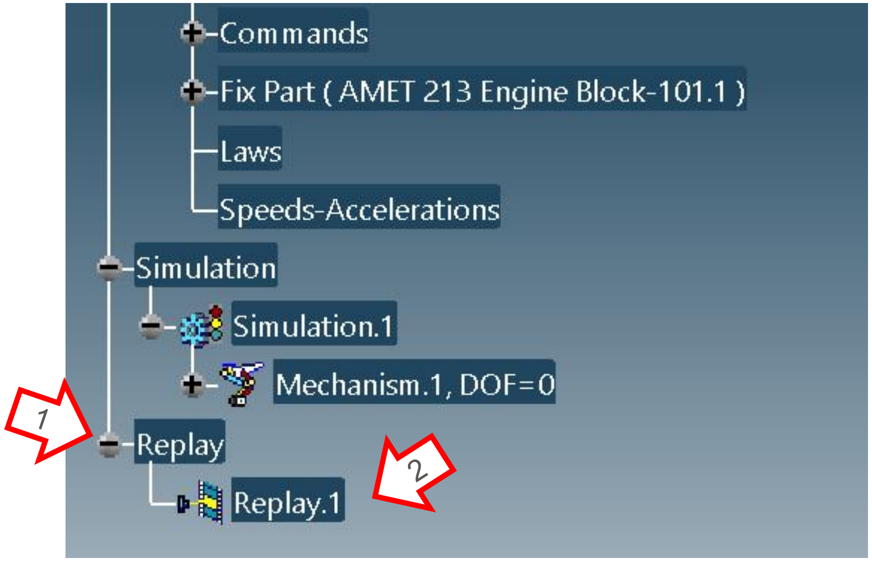




Step by Step

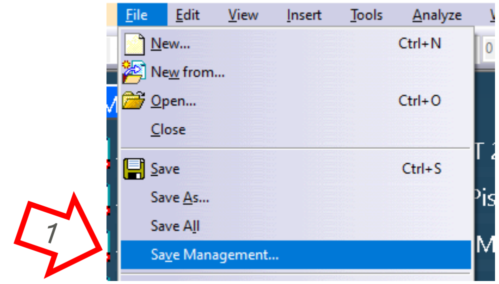
1. Expand the tree to discover Replay.1 listed
2. Double click Replay.1
3. Select Play
- The machine should repeat the Simulation
4. Select Close

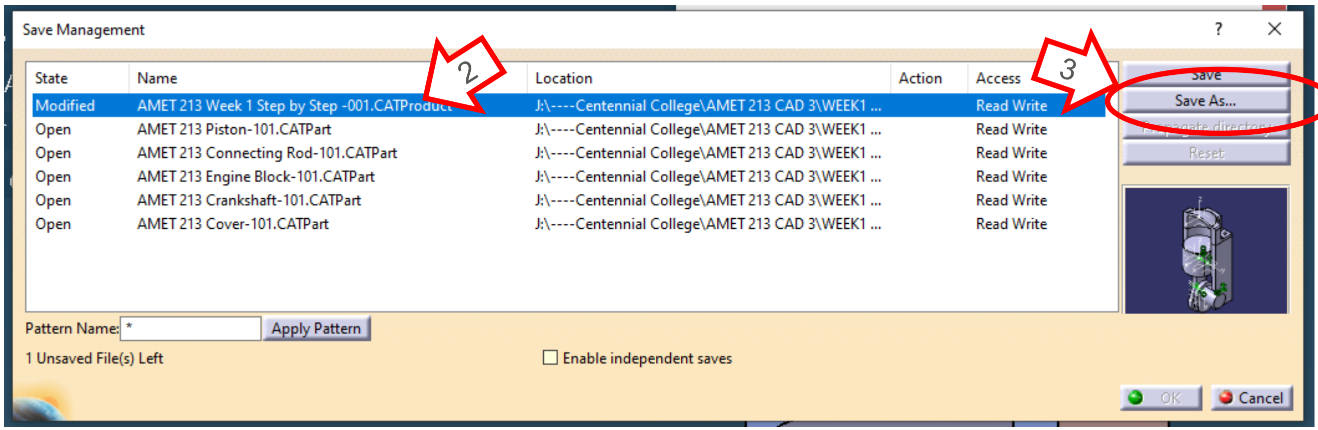


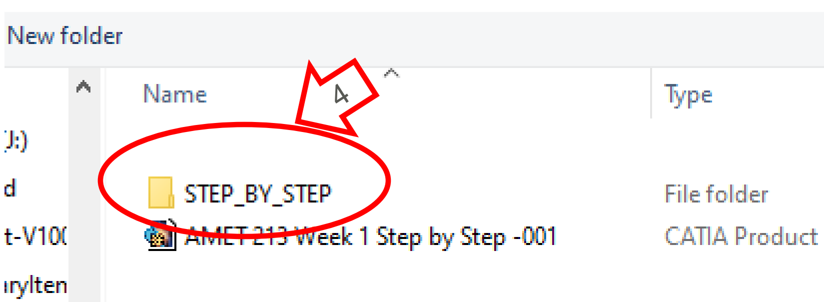


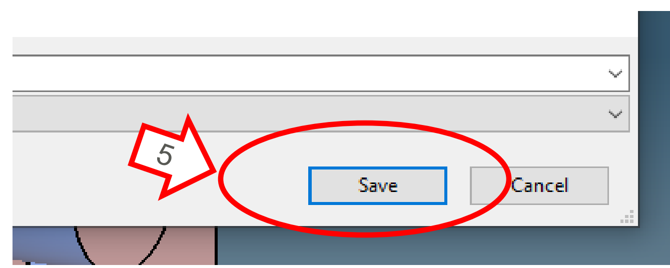
Step by Step

1. Select Save Management
2. Select the CATProduct
3. Select Save As
4. Find a directory to file the Product
5. Select Save



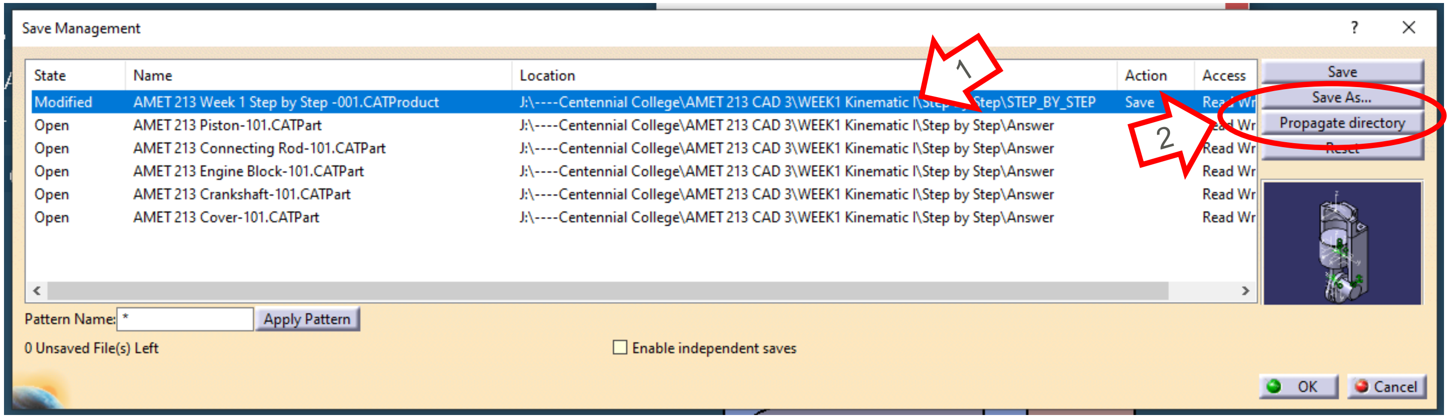






Step by Step

1. Notice the directory to file the Product is the directory previously selected



2. Select Propagate Directory
3. Notice the directory for all parts are the same as the product
4. Select OK