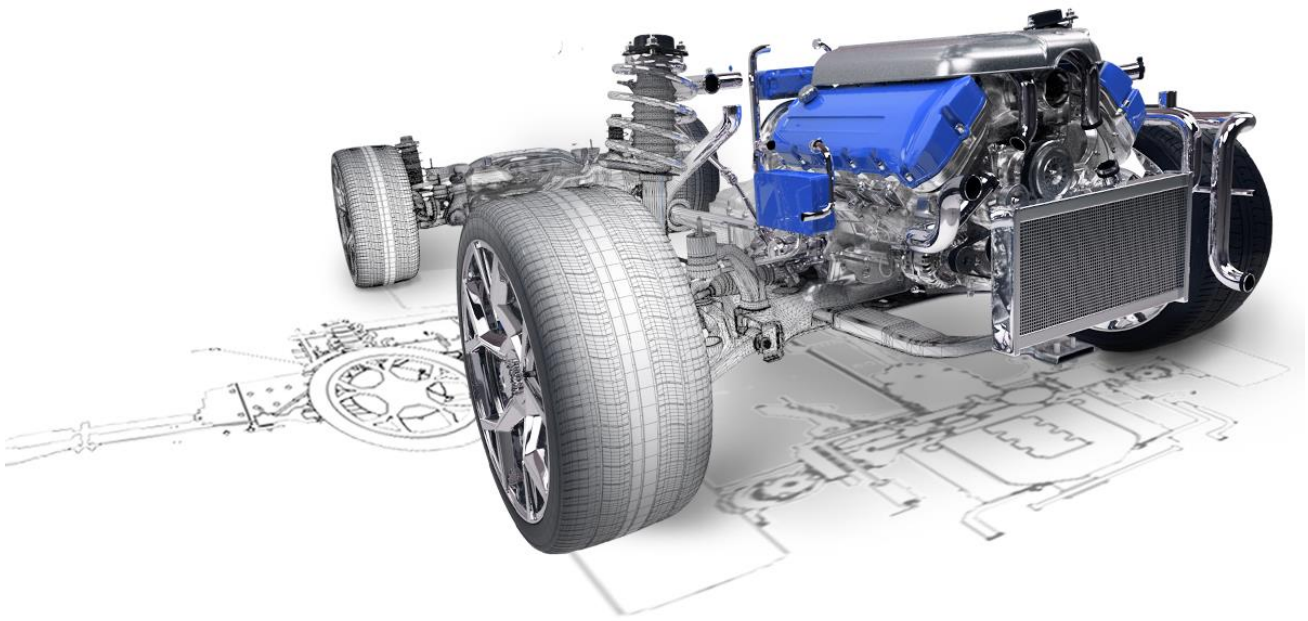


ZW3D From Entry to Master

Assembly Design

Contents

Assembly Design 1
1.1 Components Management 1
1.1.1 Inert the Component 1
1.1.2 Change the Component 5
1.1.3 Edit the Component 5
1.1.4 Merge the Component 6
1.1.5 Extract Shape 8
1.1.6 Geom to Part 10
1.1.7 External Part 11
1.2 Add Common Constraint 14
1.2.1 Anchor the Component 14
1.2.2 Add the Constraint 15
1.3 Mechanical Constraint 20
1.3.1 Gear Constraint 20
1.3.2 Path Constraint 21
1.3.3 Linear Couple Constraint 21
1.3.4 Rack and Pinion Constraint 22
1.3.5 Screw Constraint 22
1.4 Assembly Editing 23
1.4.1 Pattern the Component 23
1.4.2 Move the component 23
1.4.3 Mirror the Component 24
1.4.4 Assembly Cut 25
1.4.5 Assembly Hole 26
1.5 Assembly Check 27
1.5.1 Interference Check 27
1.6 Assembly Manager 29
1.6.1 Display Mode 29
1.6.2 Blank/Suppress the Component 30
1.6.3 Disable/Enable the Alignment 31
1.7 Case---Assembly 32
1.7.1 Case 32

Assembly Design

Key Points:

- ✧ Insert Component into Assembly
- ✧ Define Alignment
- ✧ Interference Check and Exploded View
- ✧ Animation the Motion

ZW3D Assembly module provides full sets of functions for the user to finish both top-down design and bottom-up design. Assembly module includes basic component function, alignment definition, edit component function, as well as other assistance function.

1.1 Components Management

1.1.1 Inert the Component

Assembly ribbon toolbar-> Component -> 

Use this function to insert a new component to the active assembly file.

STEP 01 Select the component object from specified file.

STEP 02 Define the location with the left mouse or specified value.

STEP 03 Define instanced options if needed.

1. **Placement**

For the Placement option, there are 3 types: Point, Face\Datum, and Frame;

- **Point**

Use this option to locate the component with selected point. The component will be located at position that the selected point and component origin point coincided.

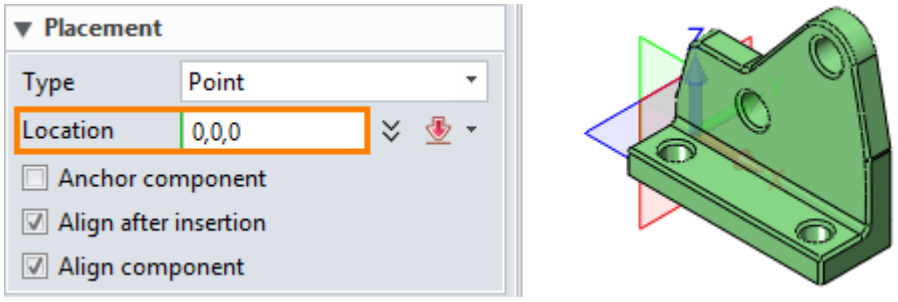


Figure1 Locate Component with Point

- **Face\Datum**

Use this option to locate the component with selected face or datum. The component will be located at position that the select faces (datum) and component origin face (XY) coincided.

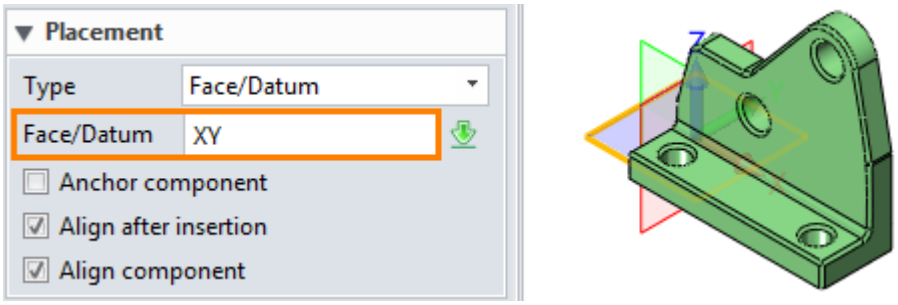


Figure2 Locate Component with Face\Datum

- **Frame**

Use this option to locate the component with selected Frame. The component will be located at the position that the selected coordinate and component coordinate coincided.

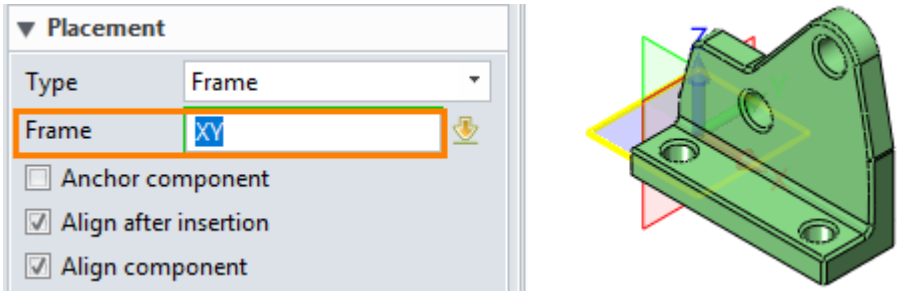


Figure3 Locate Component with Frame

- **Other**

With option “Anchor Component”, it will make the component fixed.

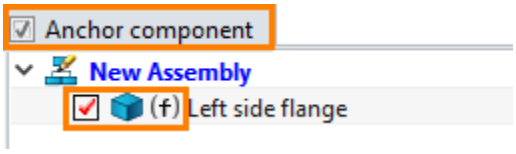


Figure4 Anchor Component

With option “Align after Insertion”, the align window will pop-up after inserting the component.

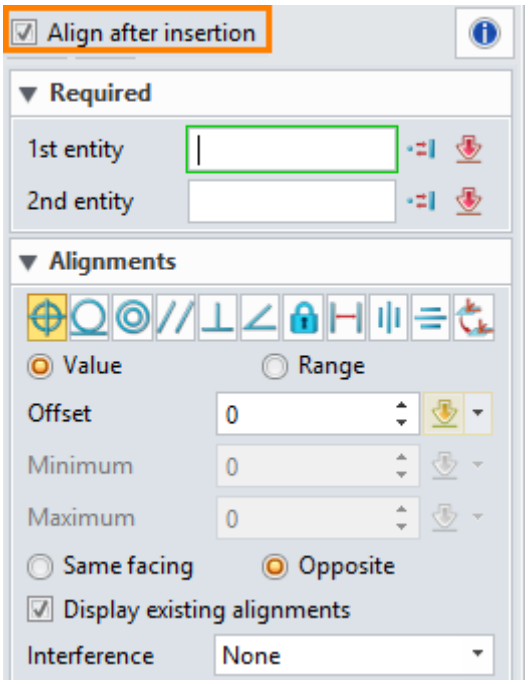


Figure5 Align After Insertion

With option “Align Component”, the constraint will be automatically created at the insertion position.

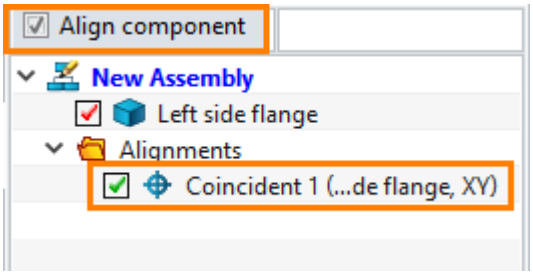


Figure6 Align Component

2. Instance

- **Copy Part**

With this option, when we insert a component from the external file, we can directly copy the original part and insert into active assembly file. The copy is not associated with the original component and will not change if the original changes.

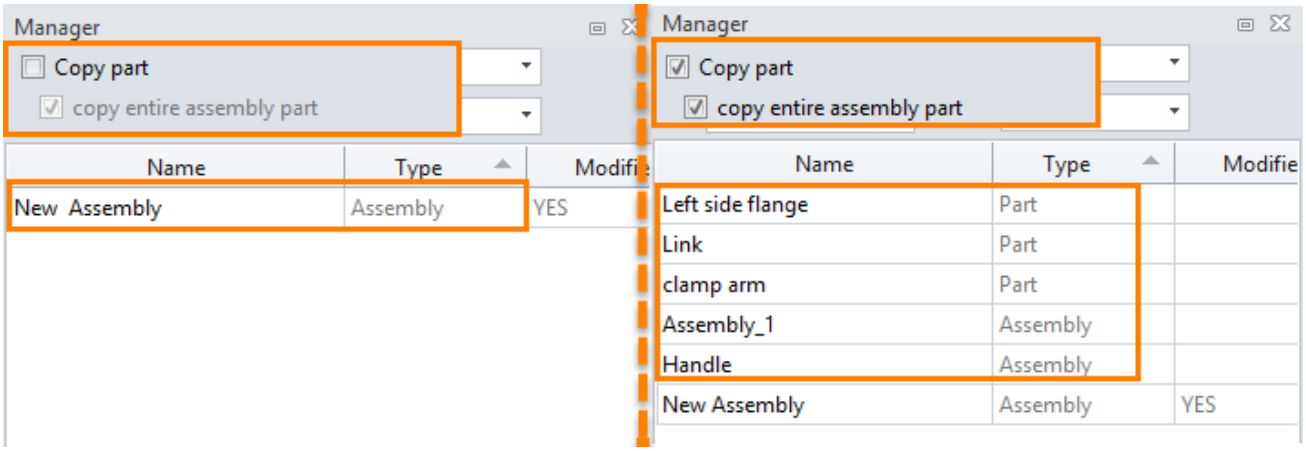


Figure7 Copy Part

With “Copy Entire Assembly Part”, it will copy all the files especially for sub-assembly part. Else it will only copy the insert file.

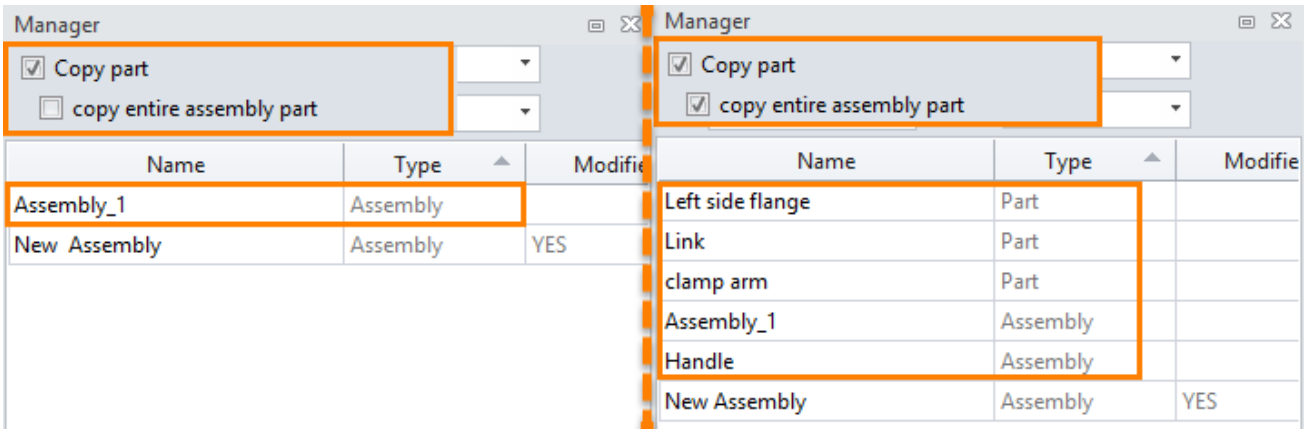


Figure8 Copy Entire Assembly Part

- Part Name

With this option, define a name for the new copy part.

- Regen

With this option, define the regen order for the new copy component. There is three option to select None, Before Assembly Regen, After Assembly Regen.

- Auto delete instanced part

With this option, the inserted component will be deleted when its parent part is deleted.

Assembly ribbon toolbar-> Component -> 

Use this function to insert multi-components at one time. It shares same functions with insert one component, Please refer to content in above.

1.1.2 Change the Component

Assembly ribbon toolbar-> Component -> 

Use this command to change the component in the active assembly. Alignment constraints placed on the original part may no longer be valid on the new part unless the new part is a copy of the original part. In this case, you need to delete the invalid constraints and replace them with valid constraints.

STEP 01 Select the component that need to be changed.

STEP 02 Select the new component to replace the old component.

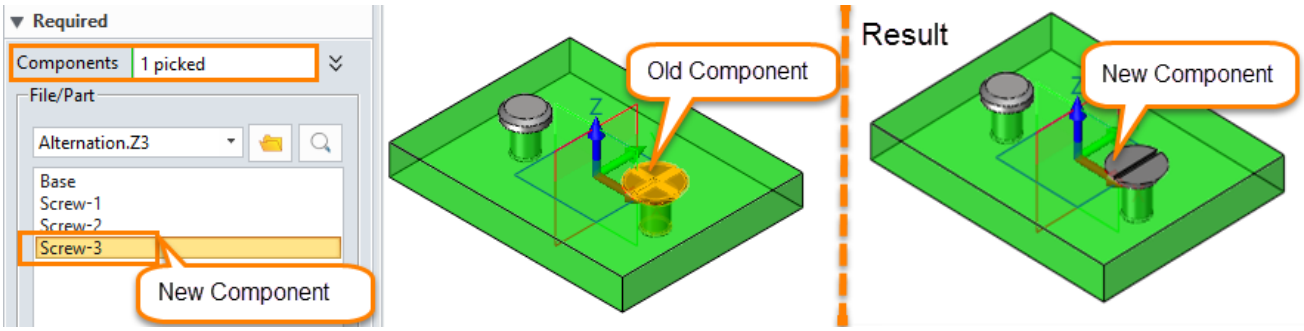


Figure9 Change Component

1.1.3 Edit the Component

Use this command to activate an existing component for editing and enter into Part Level. This is the same as right-clicking on a component and selecting Edit Part or directly double click on the component in assembly Tree. After that we can modify the part with solid or free form function.

Assembly ribbon toolbar-> Component -> 

STEP 01 Select the component that needs to be edited.

After we finish the modification, we can back to the assembly level with Exit function  in the Right Menu or directly double click on the assembly level in the history tree.

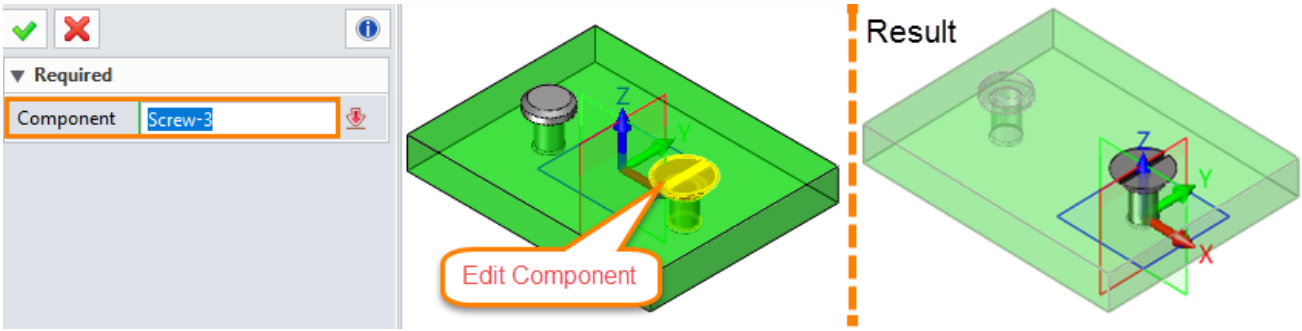


Figure10 Edit Component

1.1.4 Merge the Component

Assembly ribbon toolbar-> Component -> 

Use this command to create a base shape (using the Base option) or feature (using the Add, Remove or Intersect options) from a component.

STEP 01 Select the Merge option.

STEP 02 Select the component that needs to be merged.

STEP 03 Select the part shape that needs to do the boolean operation.

- **Base**

Use this option to convert components to separate shapes. If Base option is actived, the "Boolean Shapes" option will be automatically forbidden.

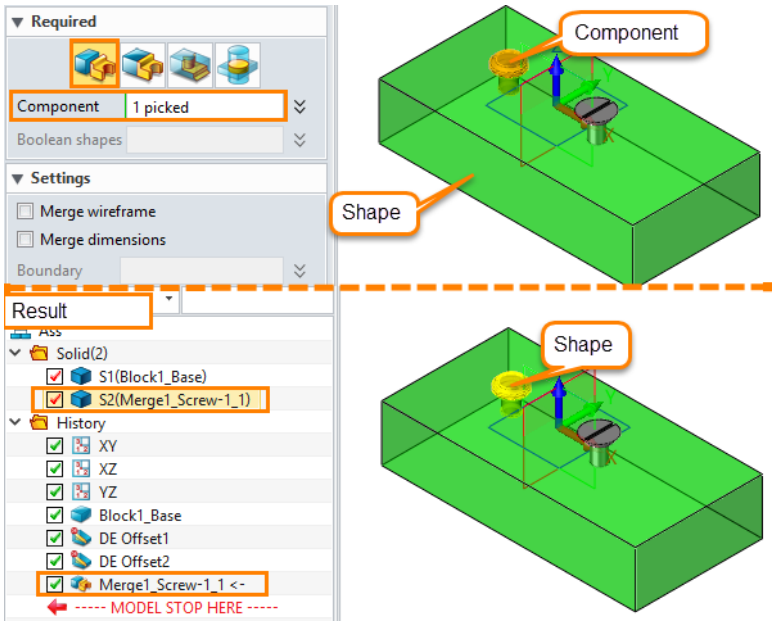


Figure11 Merge with Base

- **Add**

Use this option to add the components and shapes together.

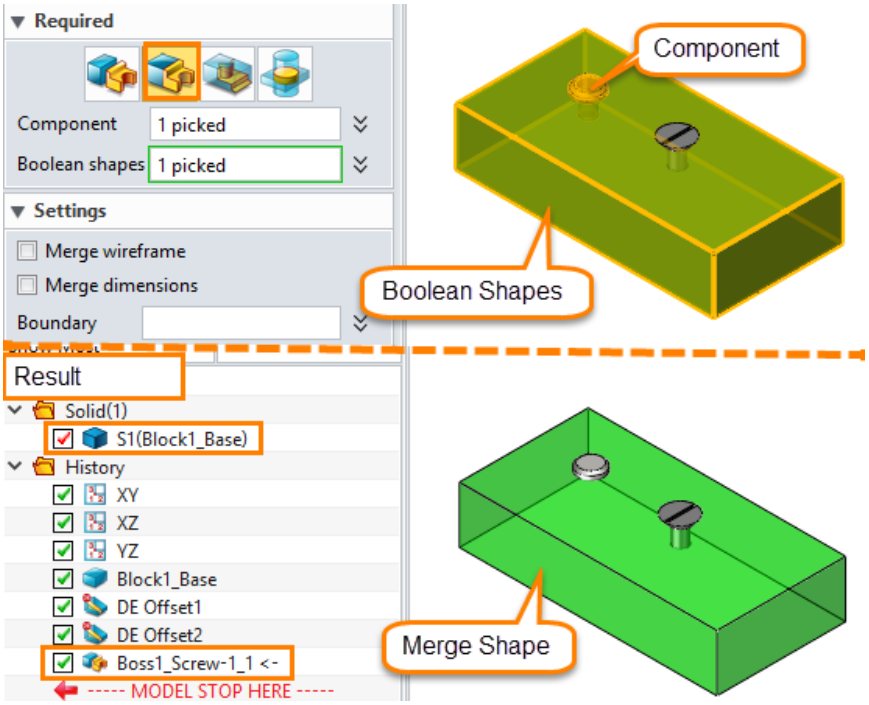


Figure12 Merge with Add

- **Remove**

Use this option to remove the Component shape from selected Boolean shapes.

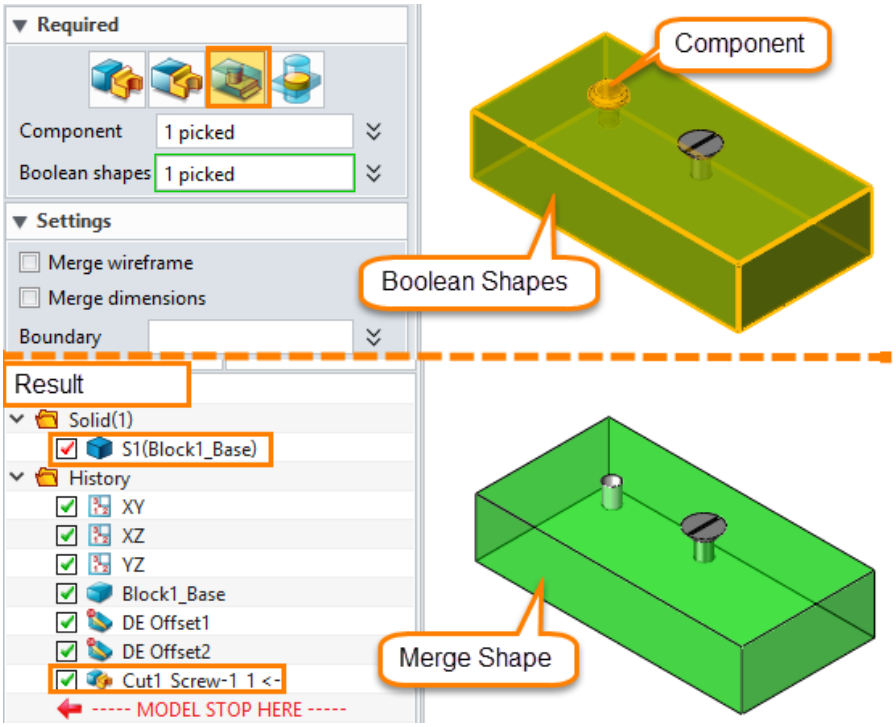


Figure13 Merge with Remove

- **Intersect**

Use this option to get the intersection between component shapes and selected Boolean shapes.

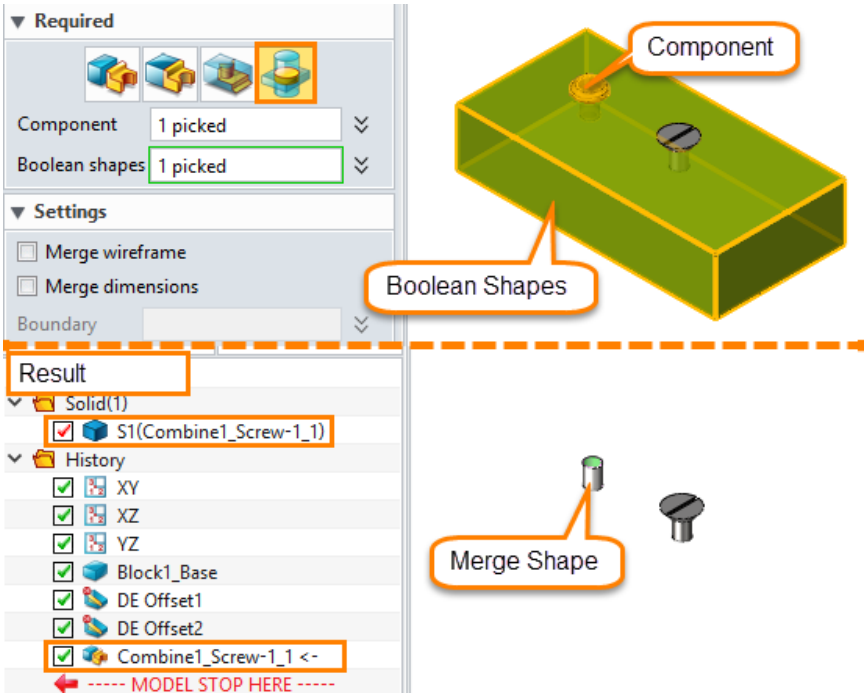


Figure14 Merge with Intersect

1.1.5 Extract Shape

Assembly ribbon toolbar-> Component -> 

Use this command to extract a "stand-alone" shape out of the active part and convert into the component. This command can be an alternate method of Assembly Design where you build all the shapes for an assembly in one part, then extract them out to separate components for CAM and 2D drawing.

STEP 01 Select the extract shapes.

STEP 02 Define the new object file frame and name.

STEP 03 Define the extract shape Settings.

- **Extract Mode**

If **Encapsulation** is selected, the newly created shape will be independent. It will not affect by the original shape. If **Associative extract** is selected, the newly created shape will be an imported geometry which will be affected by the original one.

- **Extract as component**

Check this box to extract the shape as a component in the original file, and then extract the component to a new shape.

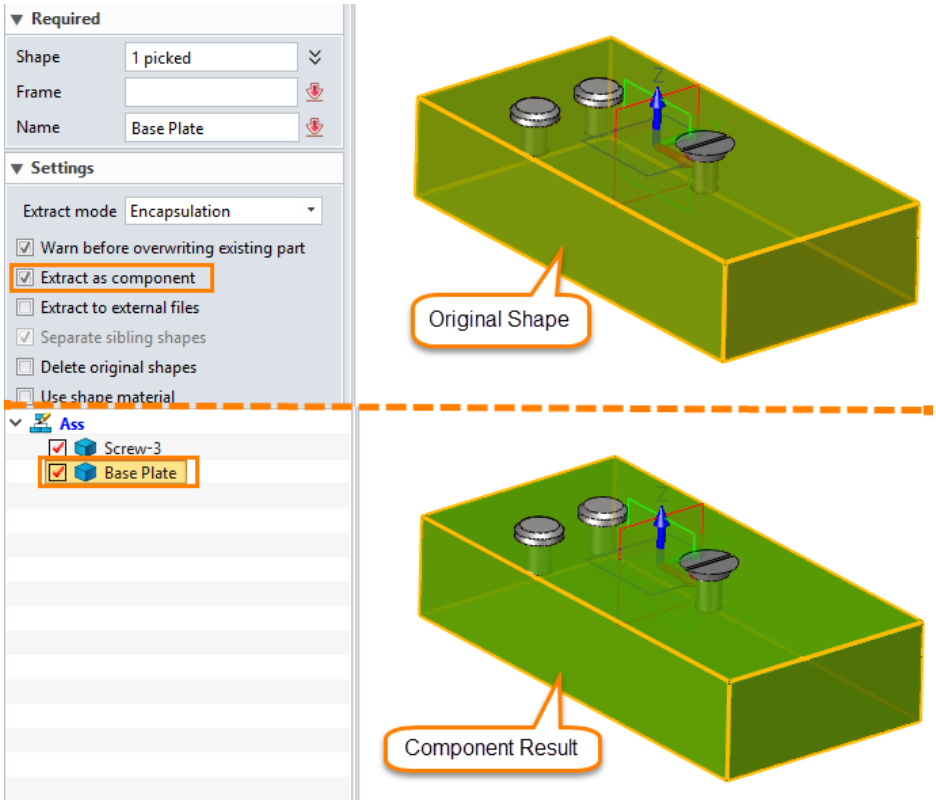


Figure15 Extract as Component

- **Extract to External File**

With this option to create new ZW3D files for each shape and users need to enter a File prefix. For “Template” option, if you have a part templates defined, you can enter its name here.

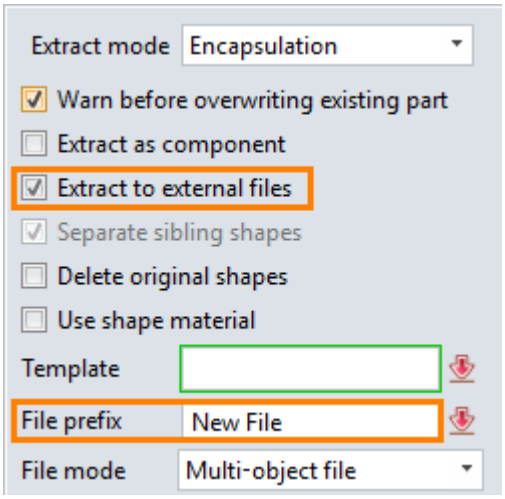


Figure16 Extract to External File

- Separate Sibling Shapes

With this option, shapes that were merged from the same component (sub-assembly) are extracted to separate components.

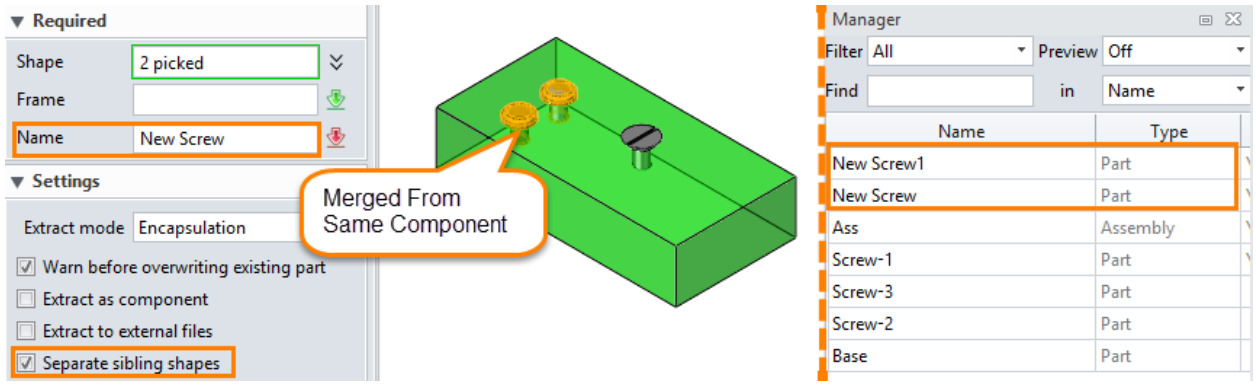


Figure17 Separate Sibling Shapes

- Delete Original Shapes

With this option, the original shapes will be delete after extract.

- Use Shape Material

With this option to use the original shape material for the newly object file.

1.1.6 Geom to Part

Assembly ribbon toolbar-> Component -> 

Use this command to copy geometry from the active part to a destination part (new or existing).

STEP 01 Select the extract shapes.

STEP 02 Select the destination File and Object.

STEP 03 Define the extract shape Settings.

- **Extract Mode**

If **Encapsulation** is selected, the newly created shape will be independent. It will not affect the newly created shape by the original shape. If **Associative extract** is selected, the newly created shape will be an imported geometry which will be affected by the original one. If Extract History is selected, the newly created shape will have entire history.

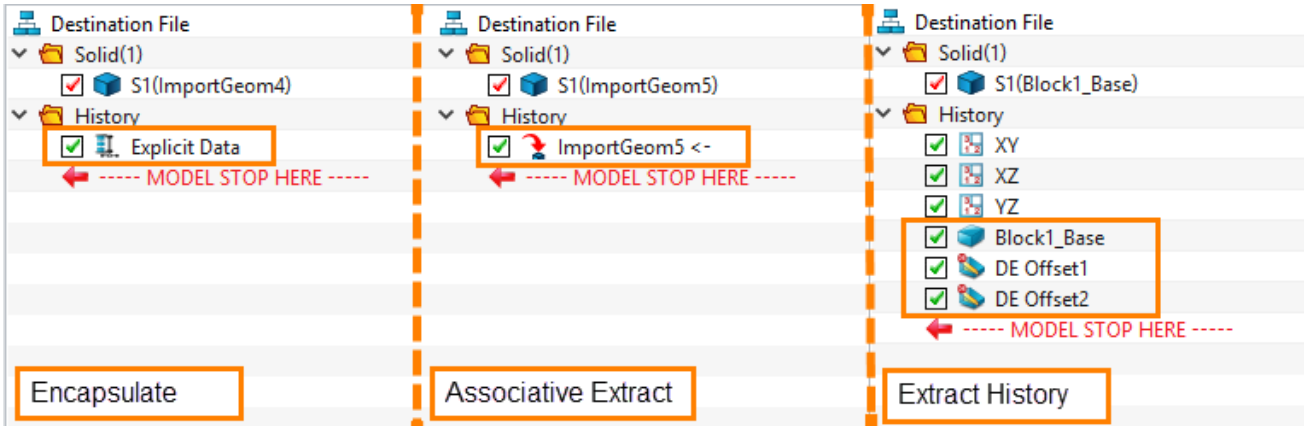


Figure18 Extract Mode

When we use **Associative extract** option, if we want to unlink the relation between the two shapes, we can use the option **Create Sub-Part in destination Part**.

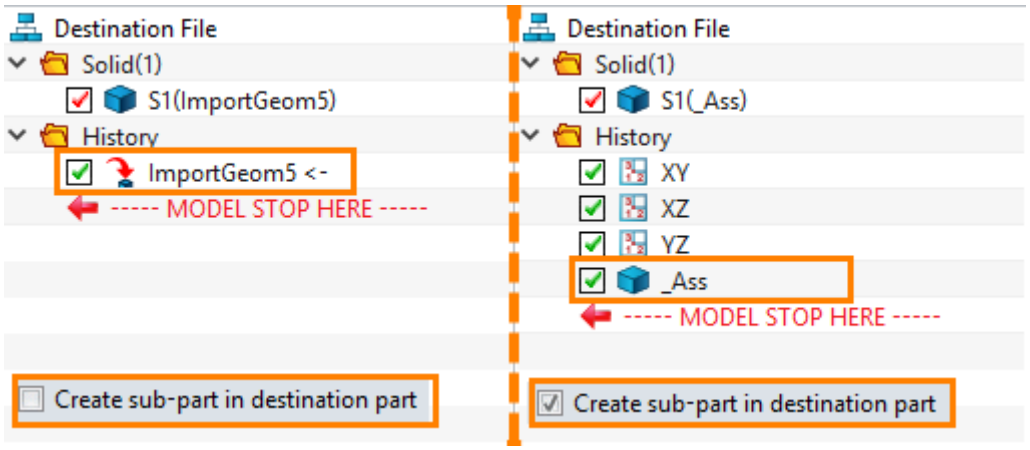


Figure19 Create Sub-Part in destination Par

For the rest options, please refer to the function in Extract Shape above.

1.1.7 External Part

Assembly ribbon toolbar-> Component -> 

Use this command to copy an external part and then insert it into the active part as shapes. The shape is associated original part. If the source part is modified, the shape is also modified the next time the active part is regenerated.

STEP 01 Select the shapes from destination part.

STEP 02 Define the location.

STEP 03 Define the shape Settings.

- **Frame & Flip Direction**

Select a face, the align direction will be calculated with the location and the face. Those two options will be activated when History has been set to **Associative Copy in this Part** or **Sub-Part with History**.

- **History**

  - **Sub-part with associative copy**: This option creates a sub-part with a copy of the geometry of the external part. When the parent part is modified, the sub-part will not update unless right-click it and select the **"Enable\Disable sub-part regeneration"** command to set the Auto-Regen Flag. When "Auto-Regen" is enabled for the sub-part, it will update when we regen the history. By default, "Auto-Regen" is disabled for a new sub-part.

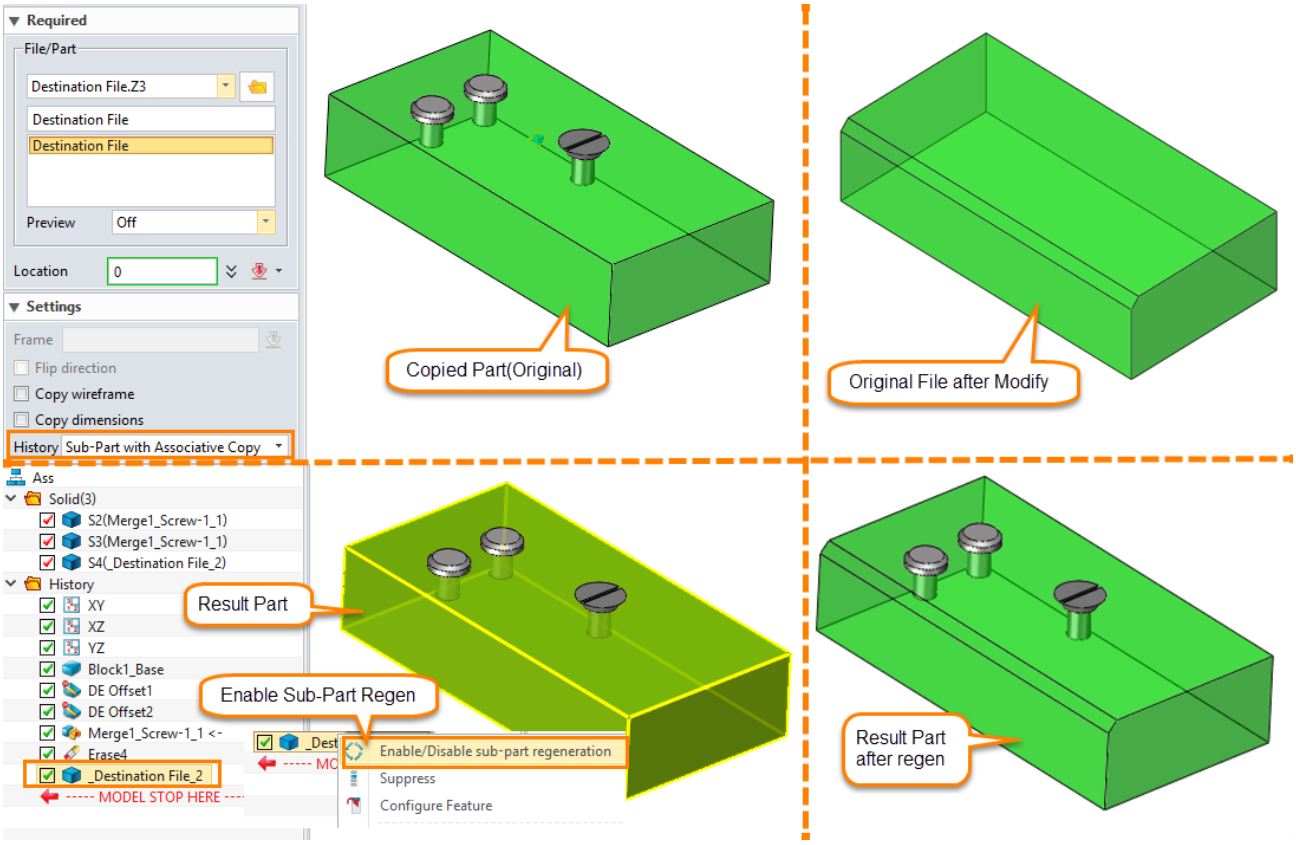


Figure20 Sub-part with associative copy

- **Sub-part with history:** This option creates a sub-part with a copy of the full history of the external part. User has a local copy to edit, but that history is unlink from the parent part's history.

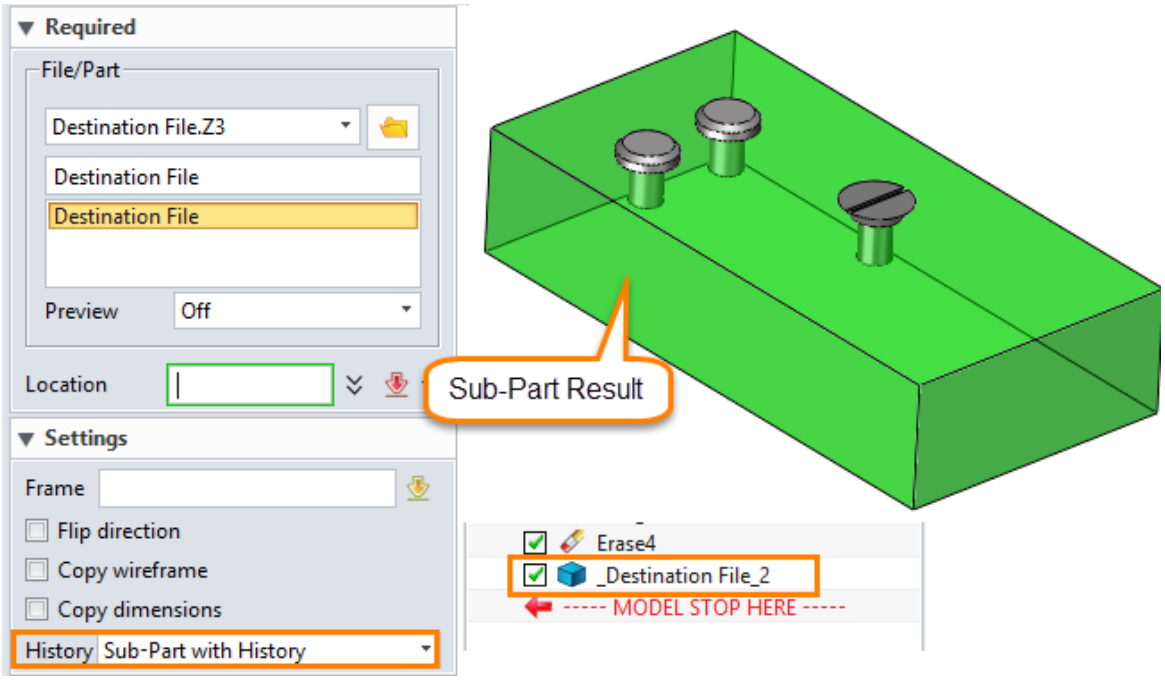


Figure21 Sub-part with history

- **History copied into this part**: This option copies the history of the external part into the active part and appends the external part's history to the end of the active part's history. Imported history operations are renamed as needed so they do not conflict with pre-existing operations in the active part. This option can be used in place of "sub-part with history" if you do not want the imported part separated as a sub-part.

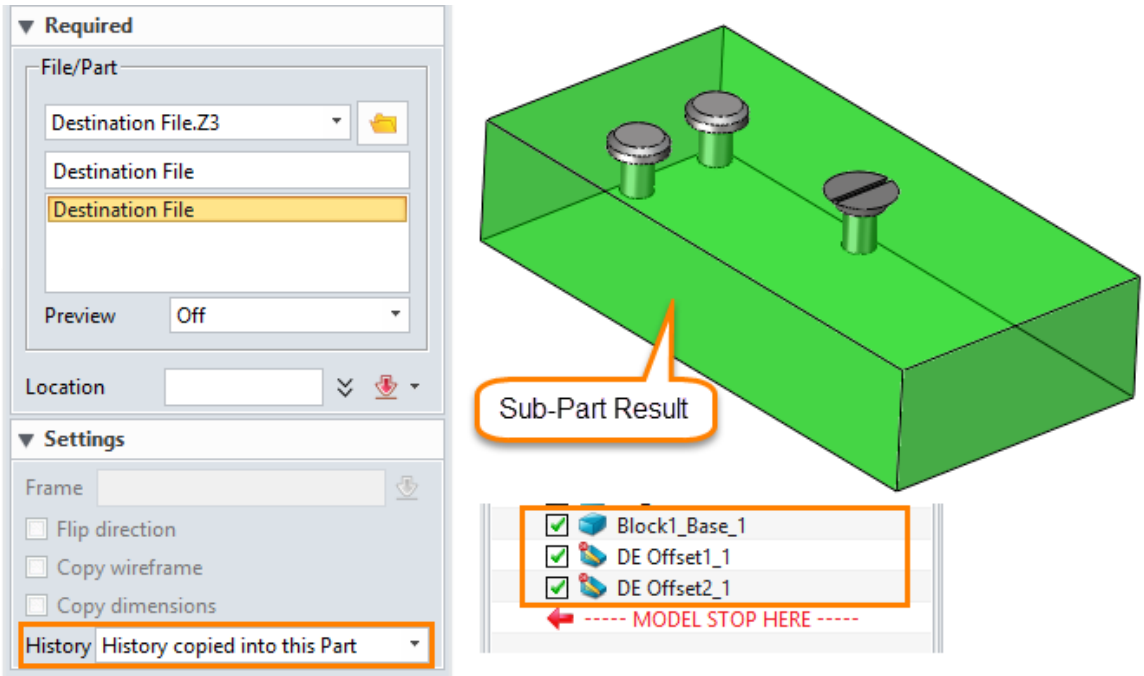


Figure22 History copied into this part

- **Associative copy in this part:** It adds an "Associative Copy" operation to the history of the active part that imports the geometry of the external part. When the parent shape changes, it will update automatically after regen the history. If the external part cannot be found, the "Associative Copy" operation will fail.

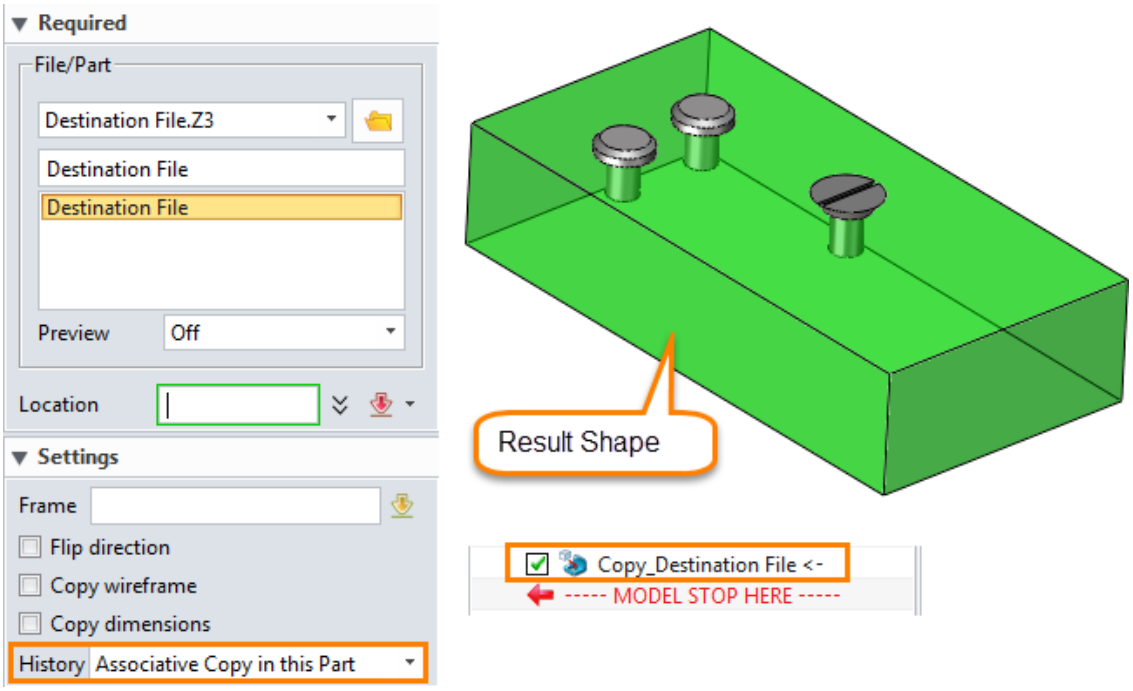


Figure23 Associative copy in this part

1.2 Add Common Constraint

1.2.1 Anchor the Component

Assembly ribbon toolbar-> Constraint -> 

Use this function to fix the current position for the selected component if the component is already anchored, this command will remove the anchor. The status of the component is indicated in the message area.

STEP 01 Select the component that needs to be anchored.

STEP 02 Click OK.

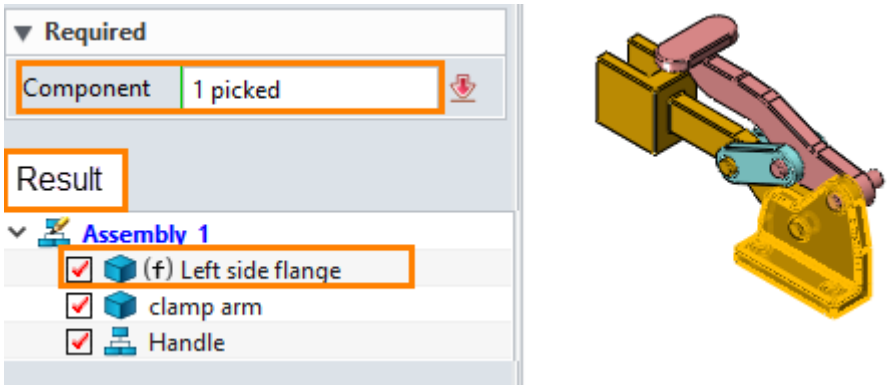


Figure24 Anchor Alignment

1.2.2 Add the Constraint

Assembly ribbon toolbar-> Constraint -> 

Creates an alignment constraint that associates two components in the active part/assembly.

STEP 01 Select first and second entity from the component you want to constrain separately.

STEP 02 Select proper alignment.

STEP 03 Define the alignment setting like direction, offset, etc.

- Coincident Constraint

  Create a coincident constraint. The Offset option is supported.

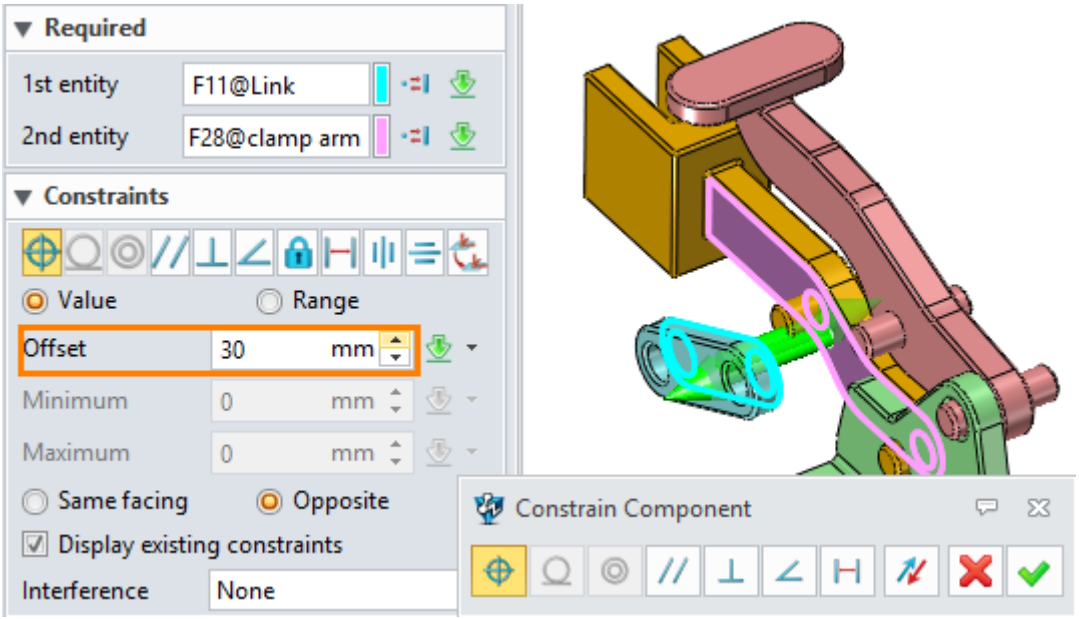


Figure25 Coincident Constraint

- Tangent Constraint: Create a tangent constraint. Supports the Offset option
- Concentric Constraint： Create a concentric constraint. Supports the Offset option.

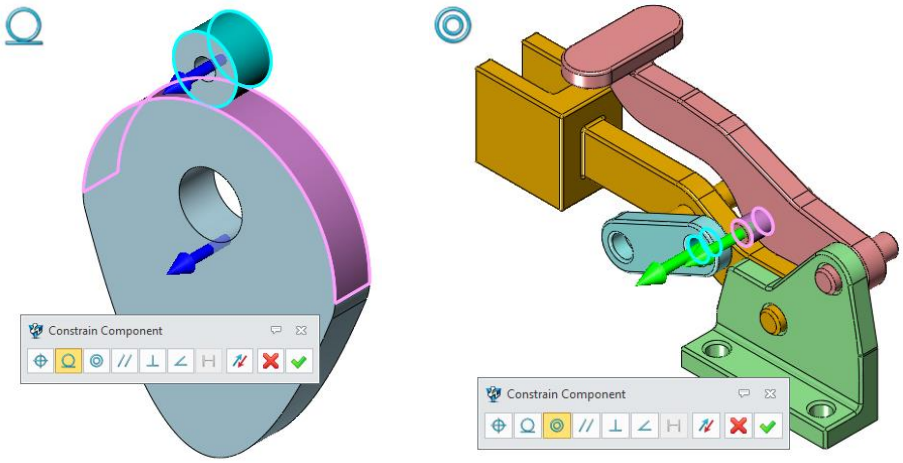


Figure26 Tangent and Concentric Constraint

- Parallel Constraint

  Create a parallel constraint. When this option is active, the offset function is disabled.

- Perpendicular Constraint

  Use this option to constrain the selected surface to be perpendicular.

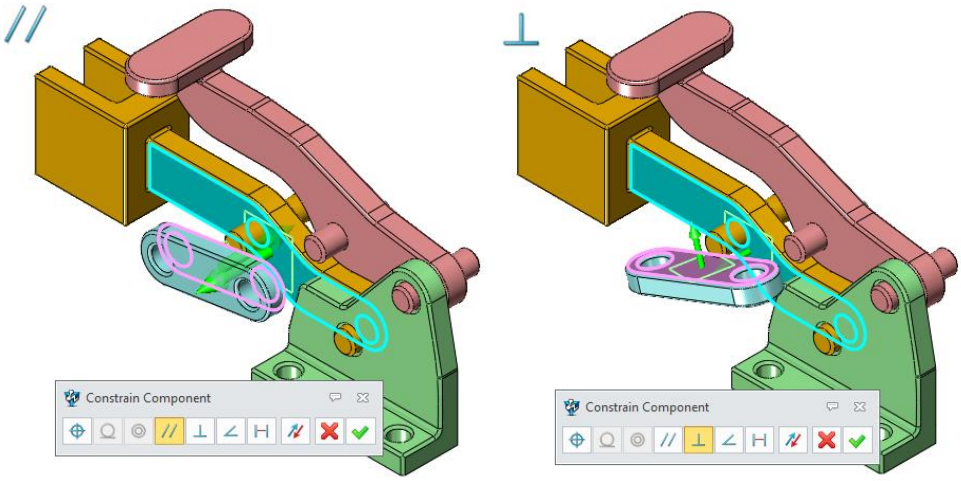


Figure27 Parallel and Perpendicular Constraint

- Angular Constraint

  Create an angular constraint to make the selected faces to be a certain angle.

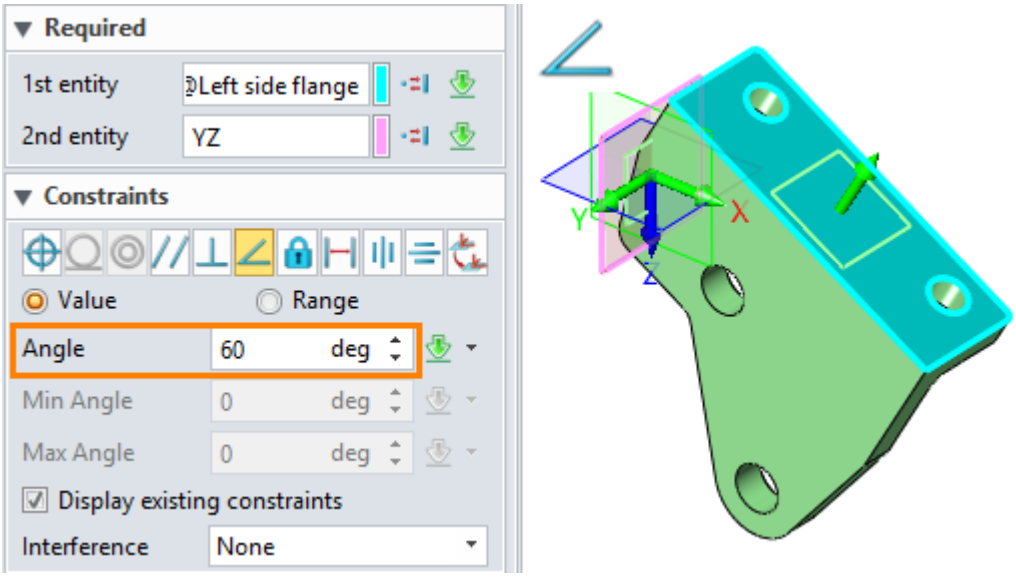


Figure28 Angular Constraint

- Lock Constraint

  Create lock constraint to lock the relative position of two components.

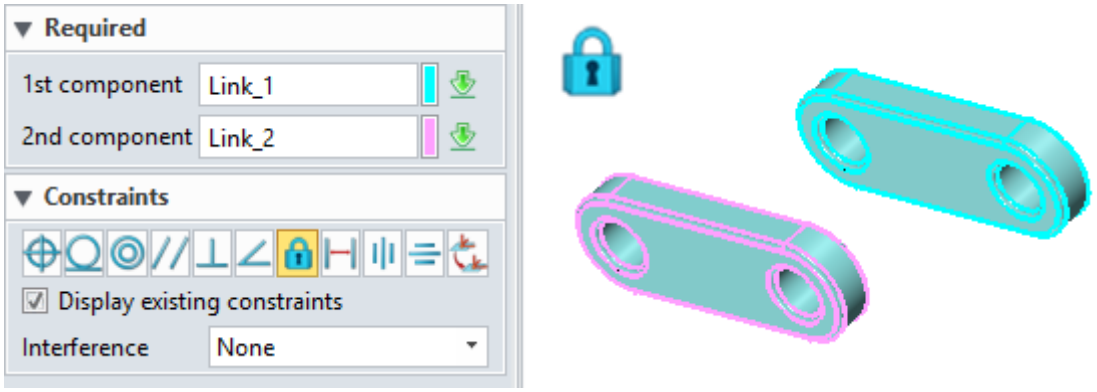


Figure29 Lock Constraint

- Distance Constraint

  Create a distance constraint. If the constraint objects are two parallel faces, the offset value is the distance between the faces by default. Otherwise, the offset is ZERO.

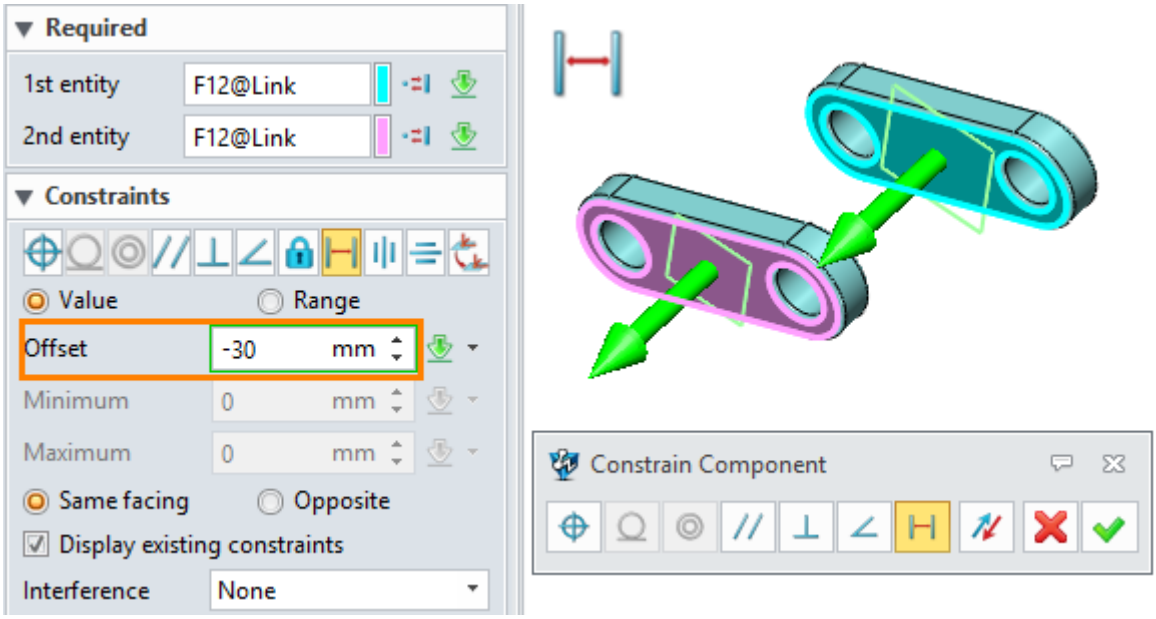


Figure30 Distance Constraint

- Middle Constraint

  Create a middle constraint to align the selected faces between base entities.

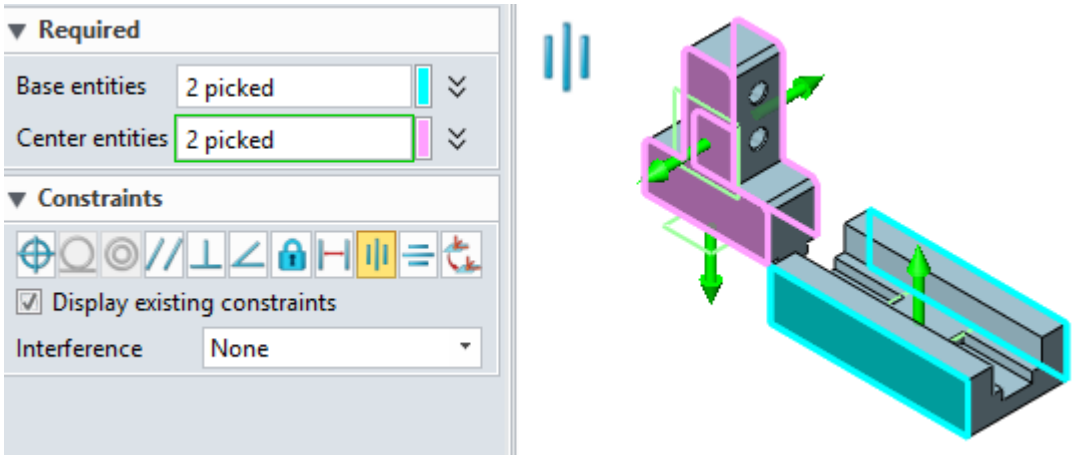


Figure31 Middle Constraint

- Symmetry Constraint

  Create a symmetry constraint to place the selected face symmetry with the plane.

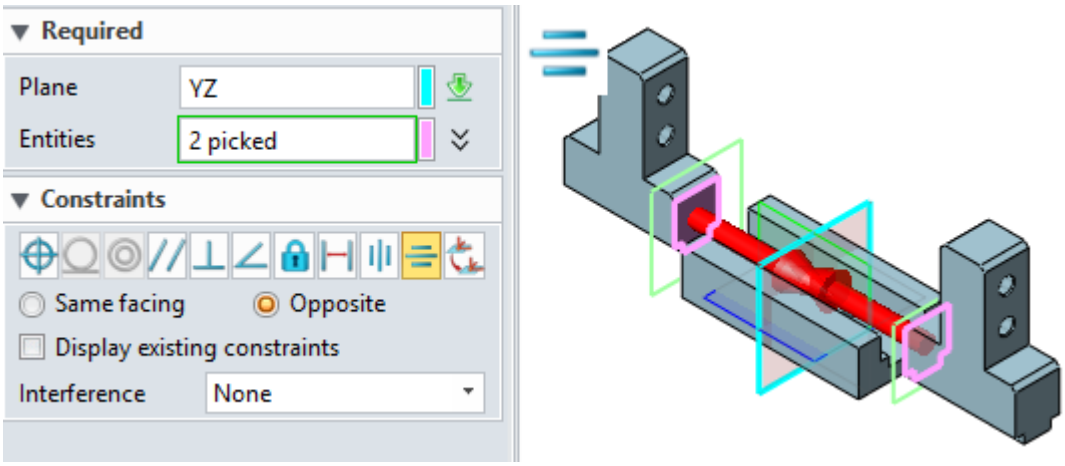


Figure32 Symmetry Constraint

- Frame Constraint

  Create a frame constraint to constrain the components. Once Frame constraint is created, the component can't move.

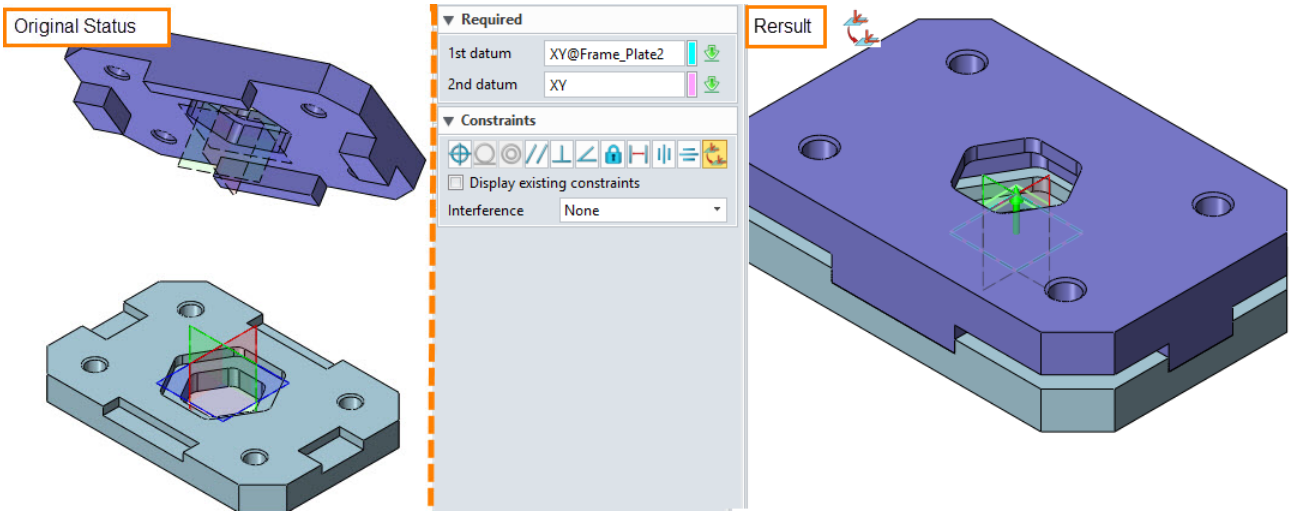


Figure33 Frame Constraint

How to show the datums for the components?

When defining the alignment, sometimes we need to use the datum from the component. In this case, we can display the external datum in the visual manger like below.

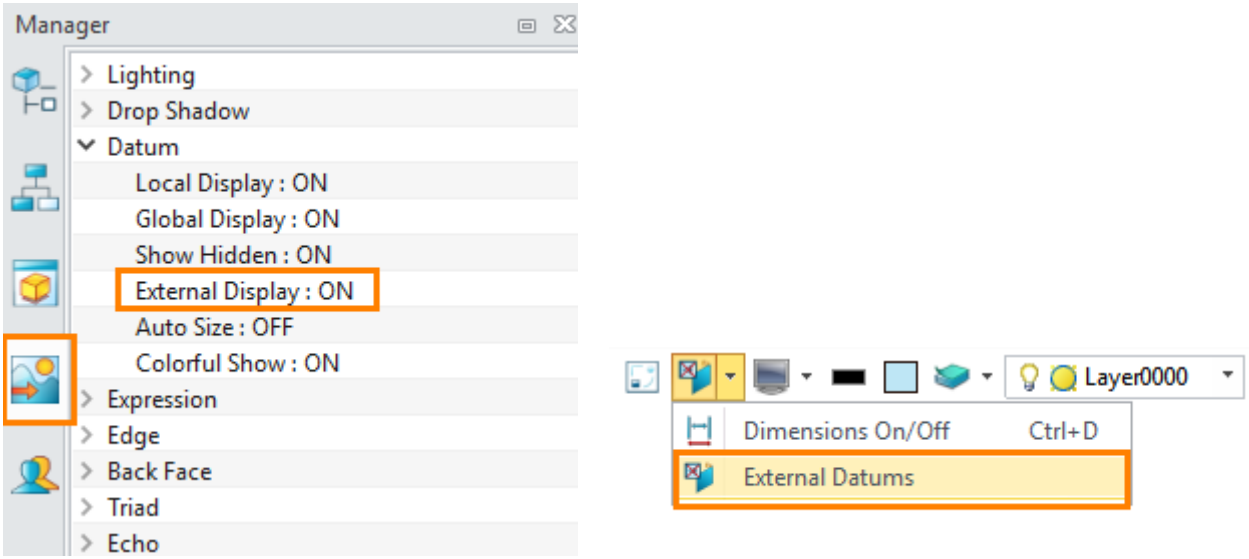


Figure34 Display External Datum

1.3 Mechanical Constraint

1.3.1 Gear Constraint

Create a gear constraint between two components. The entities can be face or line.

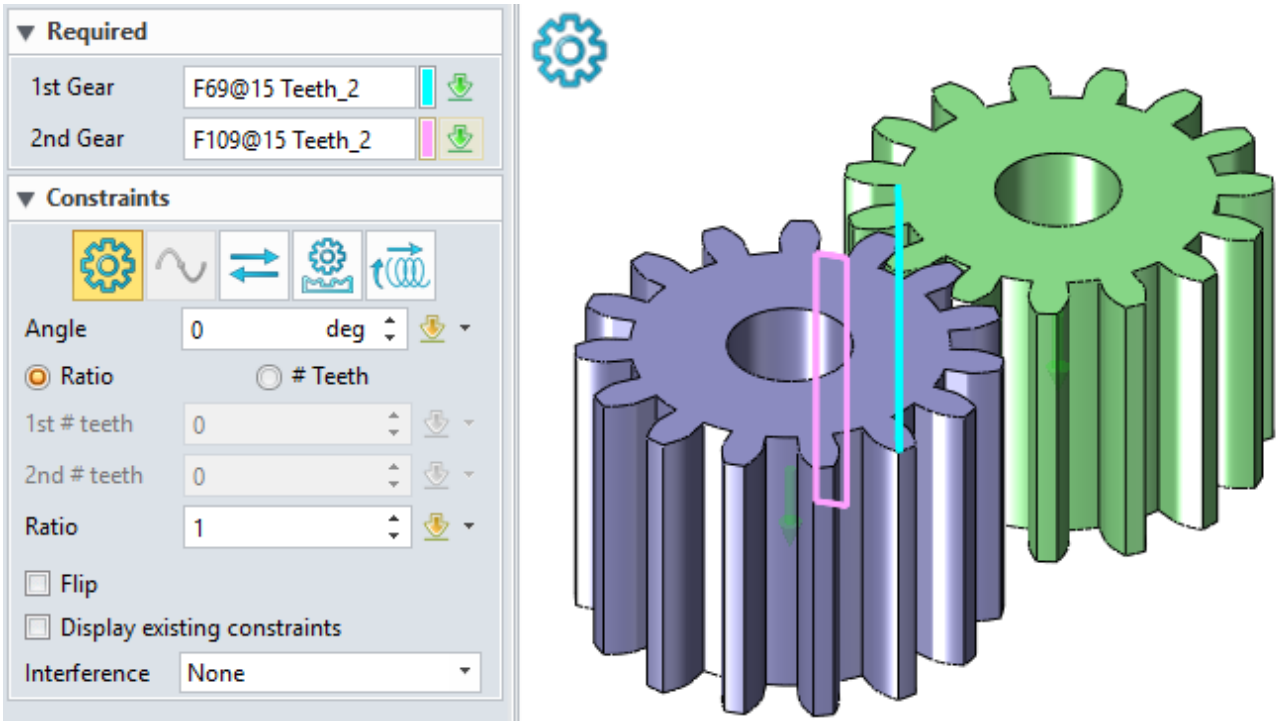


Figure35 Gear Constraint

1.3.2 Path Constraint

Create a path constraint to control the component move along the selected path. Notes that now the current path only can be a straight line.

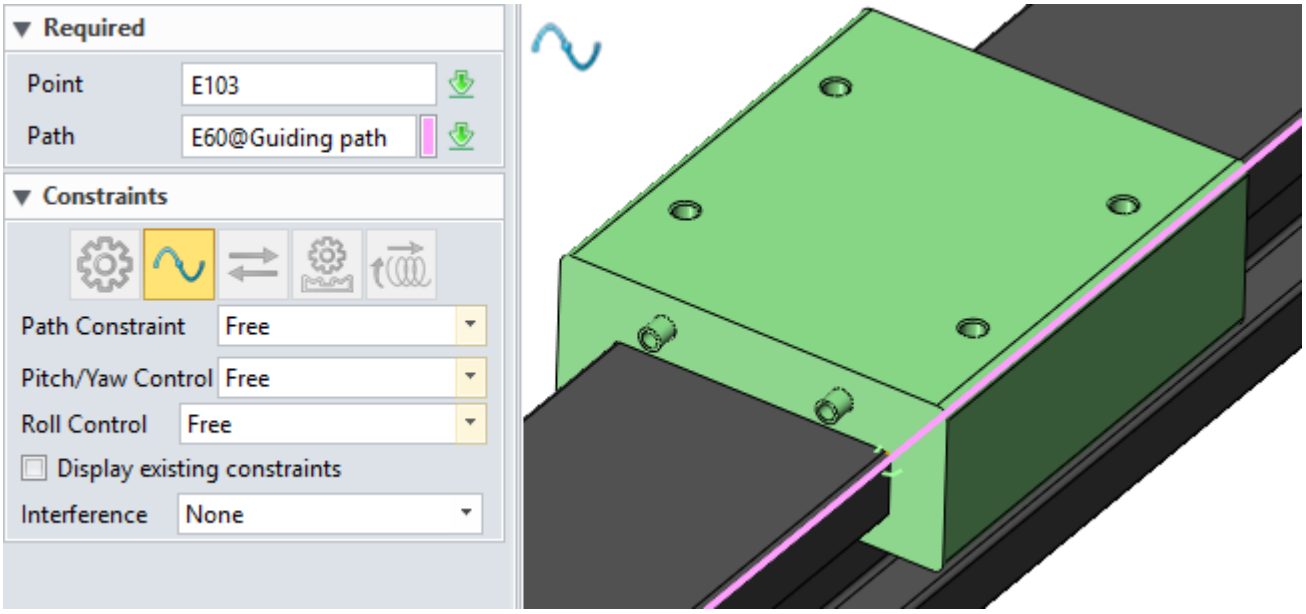


Figure36 Path Constraint

1.3.3 Linear Couple Constraint

Create a linear couple constraint to align two couple of component relative moving.

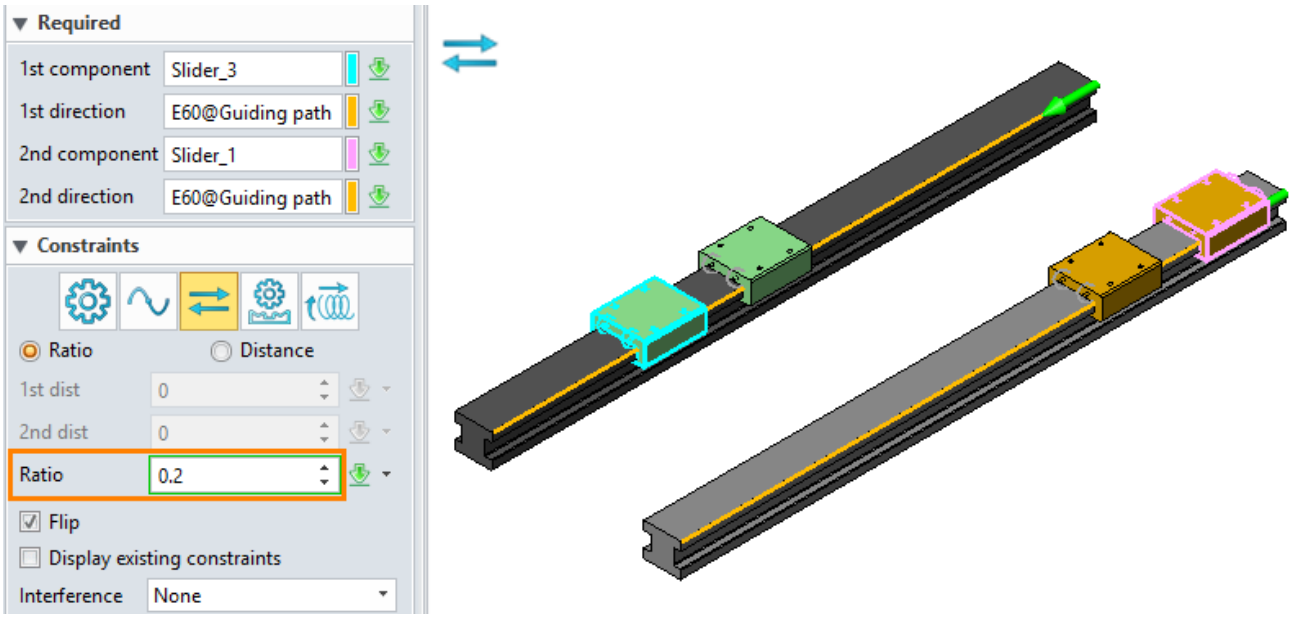


Figure37 Linear Couple Constraint

1.3.4 Rack and Pinion Constraint

Create a constraint between rack and pinion.

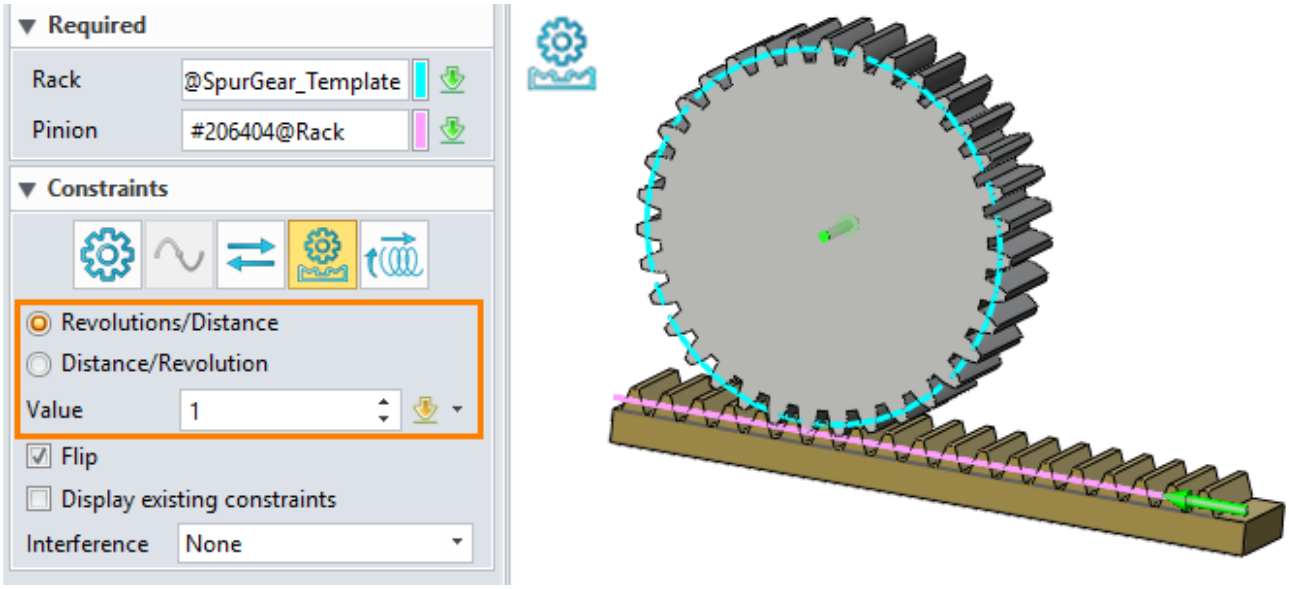


Figure38 Rack and Pinion Constraint

1.3.5 Screw Constraint

Create screw constraint between two different components. The Screw entity component will be rotated and the linear entity component will move along rotated axis.

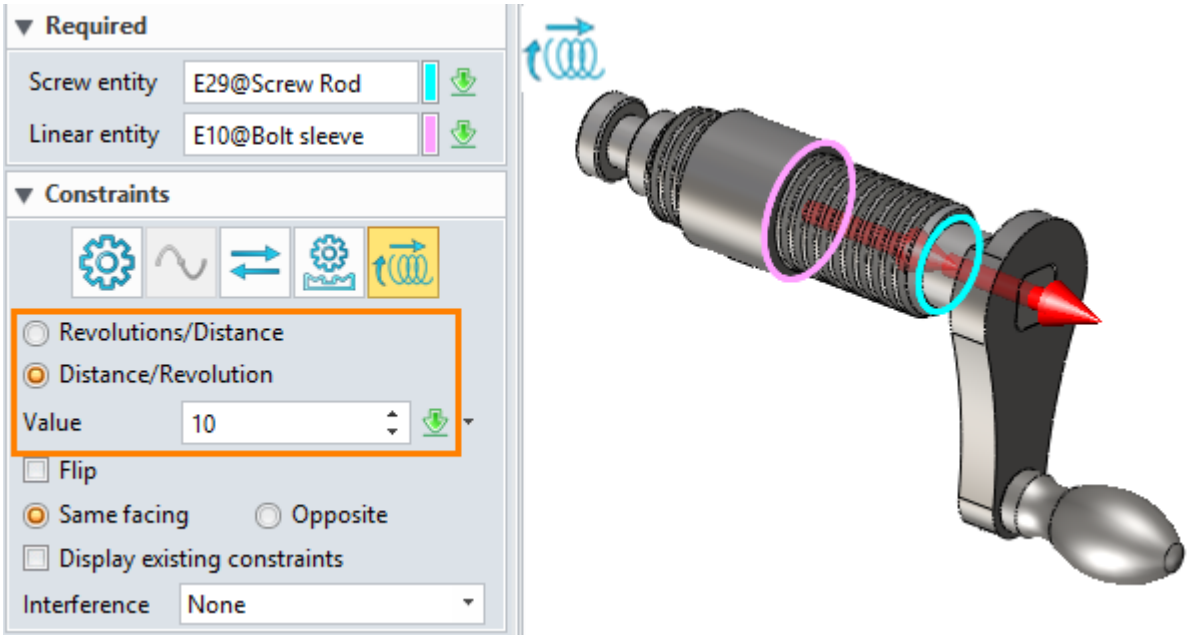


Figure39 Screw Constraint

1.4 Assembly Editing

1.4.1 Pattern the Component

Assembly ribbon toolbar-> Basic Editing-> 

Use this command to pattern components. This command is similar to the Pattern under the Shape Ribbon, but this command can only be used to pattern components. Six different methods of patterning are available (Exclude Polygon). More details please refer to Pattern function in Solid Modelling section.

STEP 01 Select entity from the component you want to pattern.

STEP 02 Define the direction, number, etc.

STEP 03 Define other setting like toggle type, orientation.

- **Instanced as component**

With this option, the pattern entity will be inserted as a component in the assembly tree. Otherwise, this operation will be recorded in the assembly tree as a pattern node.

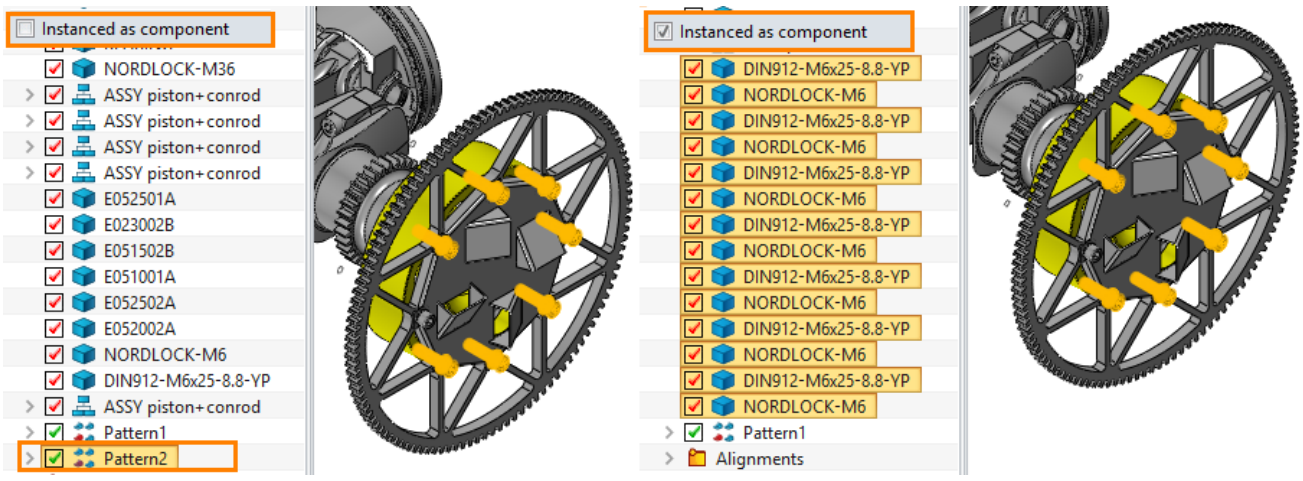


Figure40 Instanced as Component

1.4.2 Move the component

Assembly ribbon toolbar-> Basic Editing-> 

Use this command to move the components within an assembly. Various methods are supported including directions, points, and frames. This command is similar to the Move under the Shape Ribbon, but this command can only be used to move components. More details please refer to that in Solid Modelling section.

1.4.3 Mirror the Component

Assembly ribbon toolbar-> Basic Editing-> 

Use this command to mirror the components along a datum plane, planar face or sketch. Mirroring an assembly component creates a new part and inserts it as a component in the active assembly.

STEP 01 Select entity from the component you want to mirror.

STEP 02 Define the mirror face.

STEP 03 Define another setting like Center, etc .

- **Duplicate mirrored geometry**

  Click the option to create a new part when mirroring a component. The options listed as below are available only if this option isn't checked.

- **As a whole to mirror**

  When it is checked on, it will calculate mirror center as a whole. When it is checked out, it will calculate mirror center and transformed one by one.

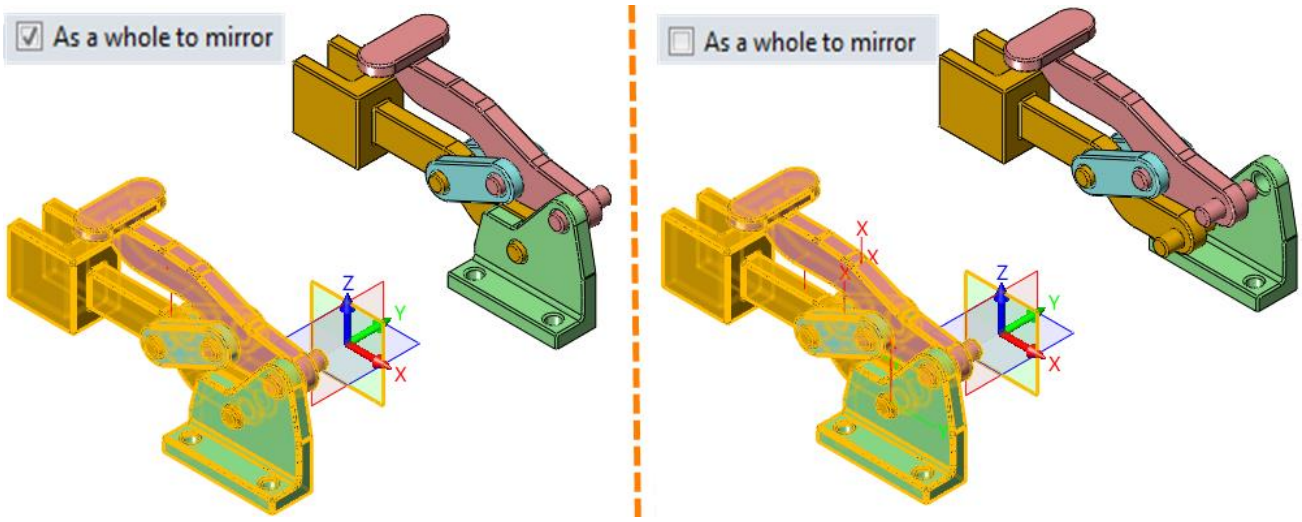


Figure41 Mirror as a Whole

- **Center of**

Set the center of rotation for mirror component to. Bounding box or Mass option is supported.

- **Self-symmetric**

Specify the component's self-symmetry plane to create mirror component.

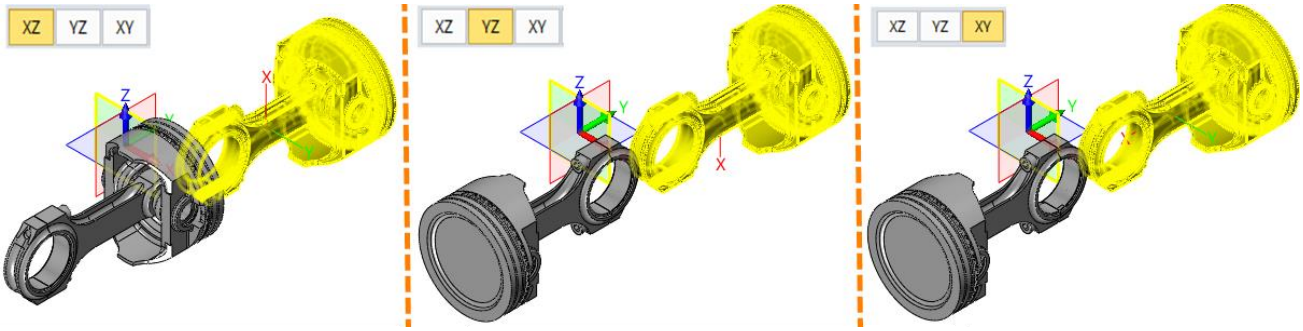


Figure42 Mirror with Self-Symmetric

1.4.4 Assembly Cut

Assembly ribbon toolbar-> Basic Editing-> 

Use this command to cut one or more components in an assembly with another component or shape. The feature is listed on the assembly modeling history tree and has all functionalities that work on modeling history features, like redefinition, suppression or conditional suppression and so on.

STEP 01 Select Cutter.

STEP 02 Define the component that needs to be cut.

STEP 03 Define other settings.

- **Propagate feature to components**

Check this option to transfer the assembly cut feature into the modeling history of the selected components to change their original parts directly. At this time, the component is under an inter-part editing mode on which its existing modeling history will be locked to avoid any modification. If you want to modify the part, you need to dissolve this mode by unlinking all assembly feature propagated from its parent assembly to break the association.

- **Hidden Cutter**

Check this option to hide the cutter.

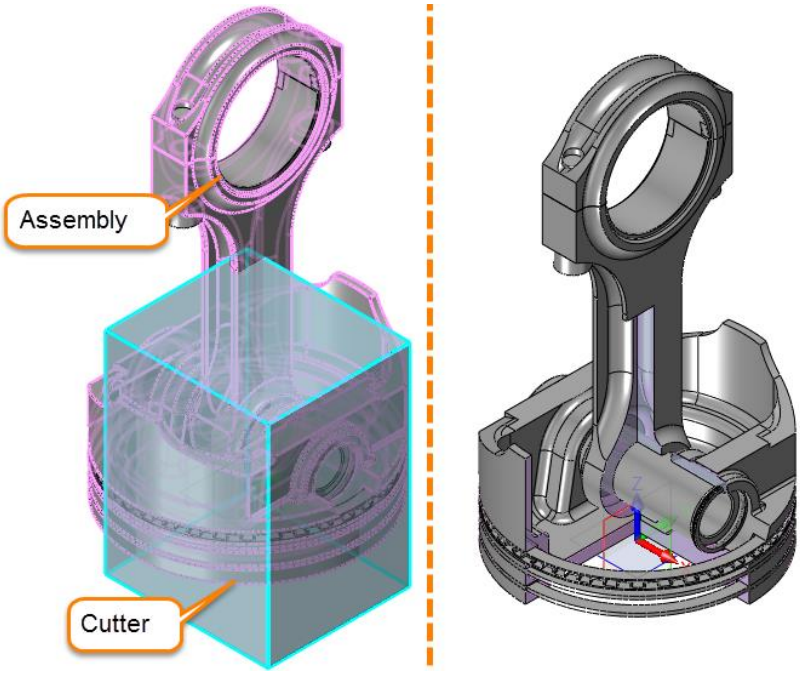


Figure43 Assembly Cut

1.4.5 Assembly Hole

Assembly ribbon toolbar-> Basic Editing-> 

Use this command to create hole from picked components. It has the same hole types as the "Hole Feature" in part, and it can only exist within the assembly, which will not affect any original modeling course of the picked components. Please refer detail information in Hole Feature in Solid Modeling tutorial.

STEP 01 Select location for holes and hole type.

STEP 02 Define the component that needs to be removed material.

STEP 03 Define other settings like hole depth, diameter, etc.

1.5 Assembly Check

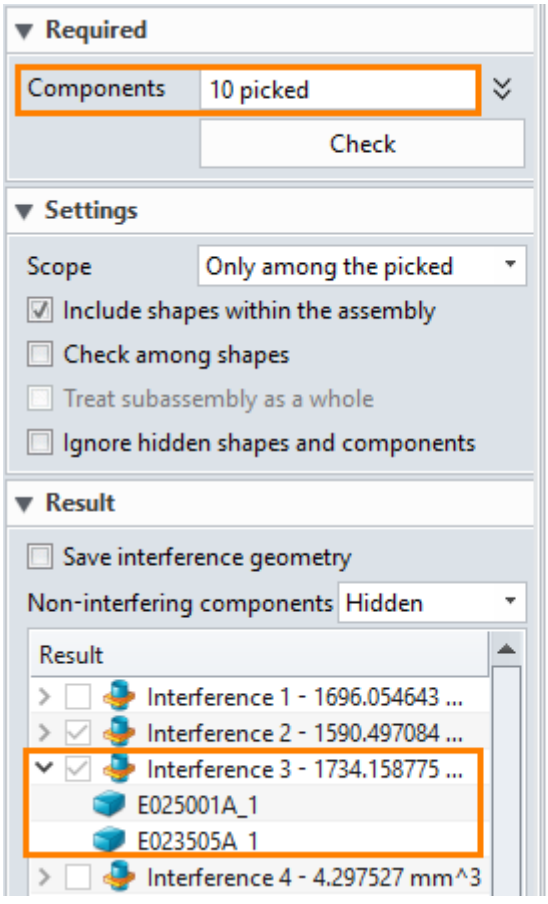
1.5.1 Interference Check

Assembly ribbon toolbar-> Inquire-> 

Use this command to check the interference between components or the assembly. Suppressed components in an assembly will be ignored during these calculations.

STEP 01 Select component that needs to be check.

STEP 02 Define Settings and display mode.



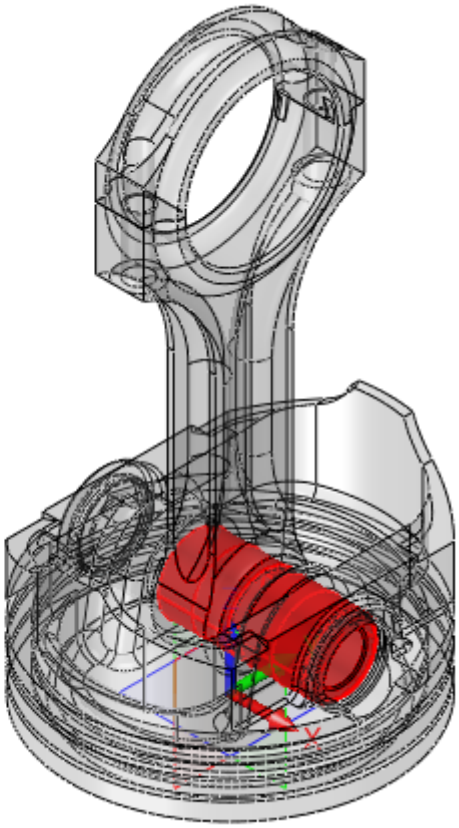


Figure44 Interference Check

- Include shapes within the assembly

  Check this box, it will check the interference between the picked components and the shapes.

- Check among shapes

Check this box, it will check the interference among all the shapes.

- Treat subassembly as a whole

  This box is available if a subassembly is picked. Check this box, it will not check the interference within the subassembly.

- Ignore hidden shapes and components

  Check this box, the hidden parts and assemblies are not involved in interference check.

- Save interference geometry

  Use this option to keep the interference shapes in the history list. And a new sub-part feature is created. Otherwise, the interference shapes and the sub-part feature will not be recorded in history tree.

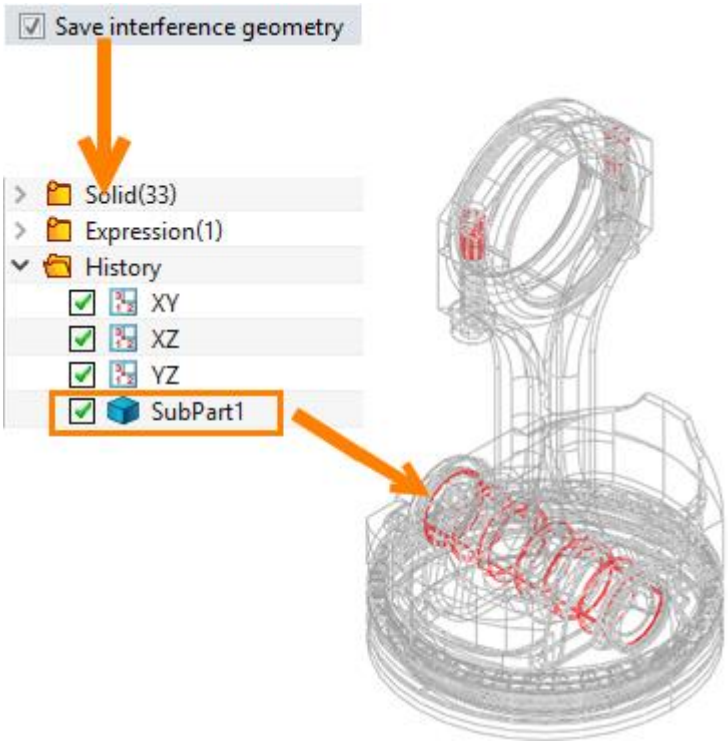


Figure45 Save Interference geometry

How to show the details of the interference shapes?

After we calculate the interference geometry, we can find more details in the result. We can check the box to display different interference shapes as well as the interference volume. Also in the drop-down list, you will find which two components have interference.

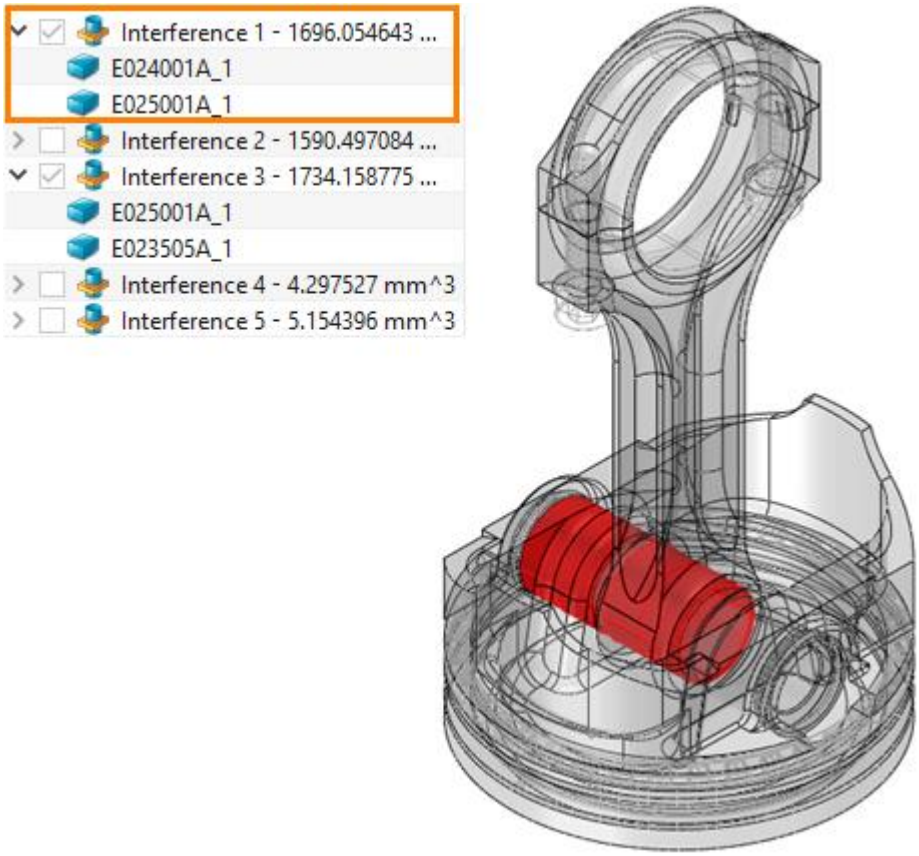


Figure46 Interference Result

1.6 Assembly Manager

1.6.1 Display Mode

Assembly Manager-> Right Click Menu

There are two types of display mode in ZW3D assembly manager, separated and combined mode. With this two mode, users can display the constraint with different locations.

Separated Mode:

With this option, all the components and all the constraints are displayed separately. And component displayed first, then constraint displayed in a bottom at the same level.

Combined Mode:

With this option, each component and its constraints are displayed together.

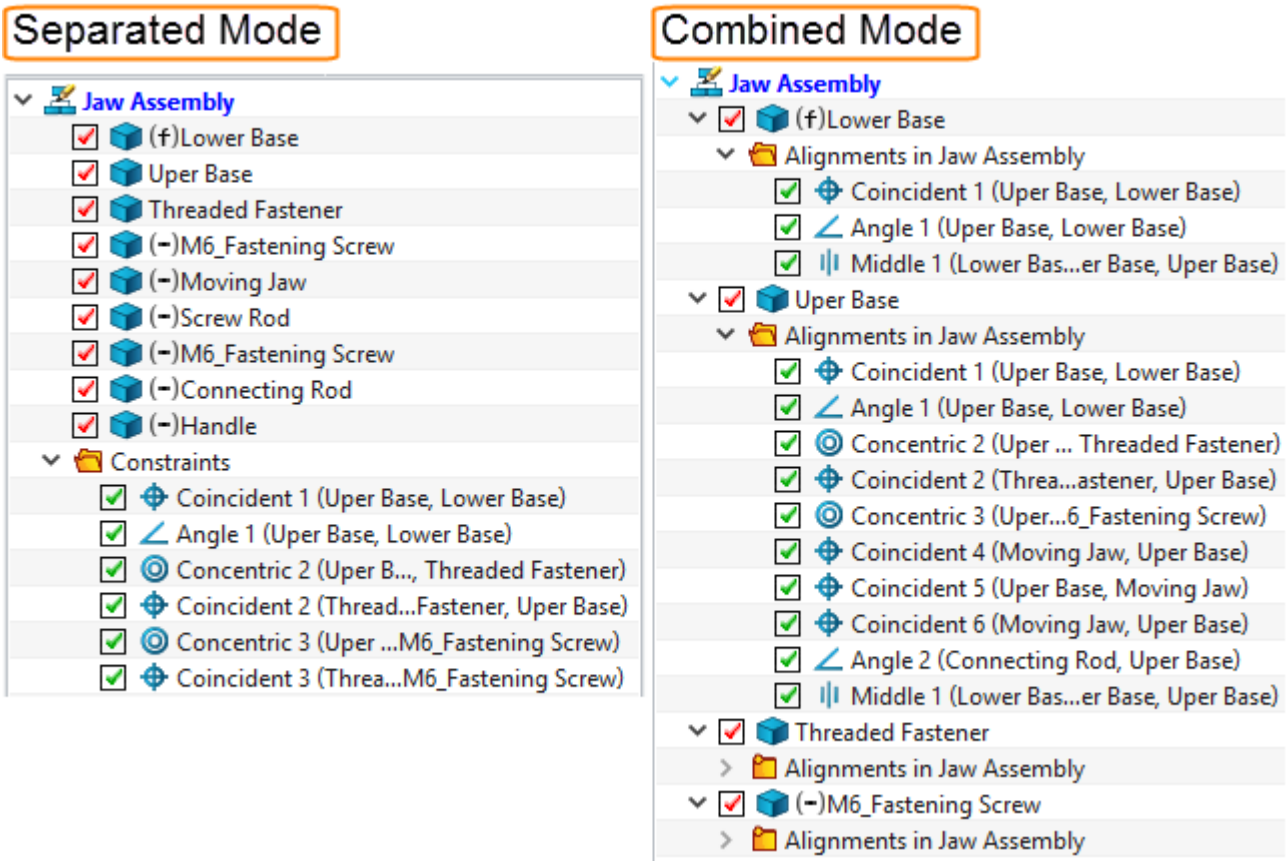


Figure47 Constraint Display Mode

1.6.2 Blank/Suppress the Component

Assembly Manager-> Right Click Menu

When right click on certain components, in the right menu, users can select blank or suppress to modify the status for selected components.

Also, you can use the Blank function in Document Aware Toolbar to do the hidden operations.

How to quickly blank the component and suppress the constraint ?

In history manager, users can quickly blank the component and suppress the constraint by check-box in the assembly tree.

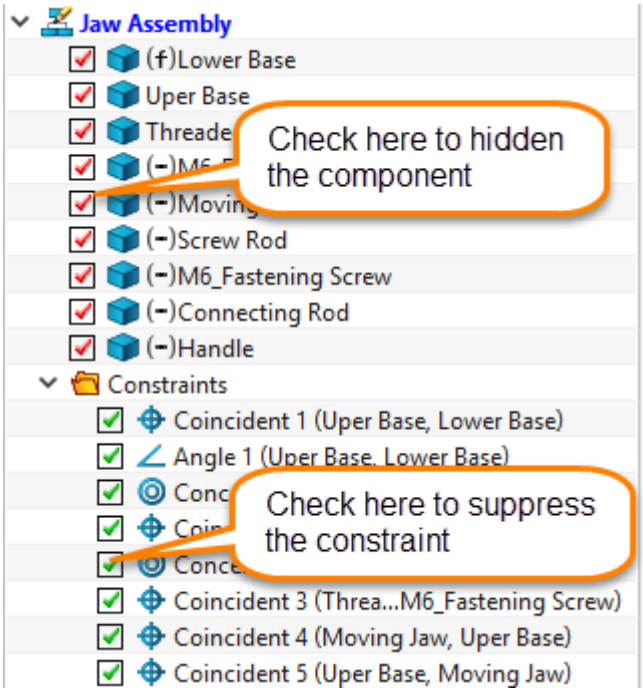


Figure48 Blank/Suppress by Check-Box

1.6.3 Disable/Enable the Alignment

Assembly Manager-> Right Click Menu

When right click on the certain constraint, in the right menu, user can disable\Enable the highlight constraint. After that, the constraint will not take effect until user enables it.

1.7 Case---Assembly

In this module, you can learn how to use above assembly functions to design your own product. With below case, it will show you the workflow in ZW3D assembly.

1.7.1 Case

In this case we will use below example to show you how to use insert, constraint, etc.

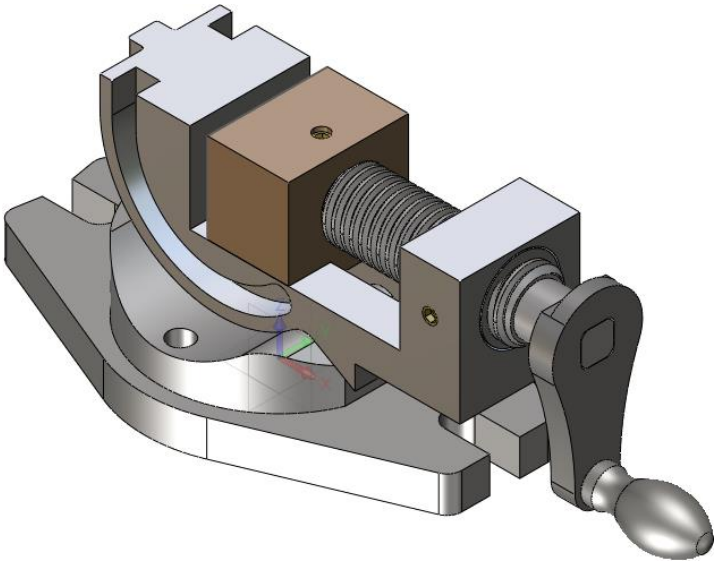


Figure49 Case-Jaw

1. Create New Assembly File

STEP 01 Open the file “Assembly Case.Z3”, and create a new object with name “New Example”.

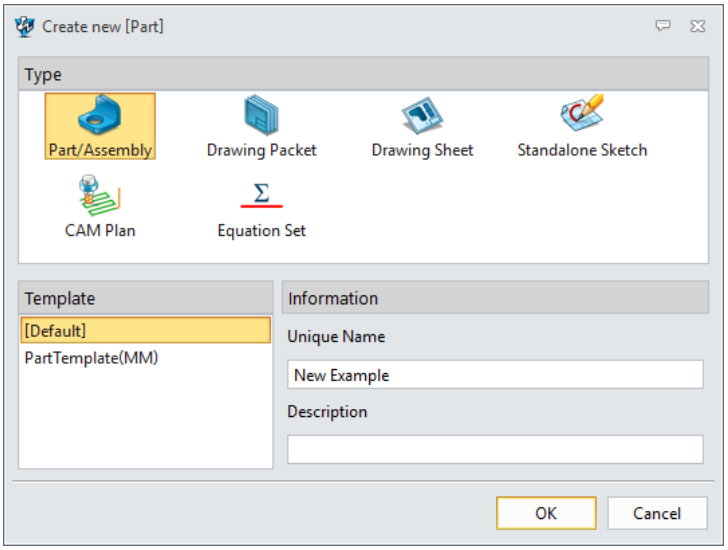


Figure50 Create New Assembly Object

2. Insert First Component

STEP 01 Right click on the blank area in the main window and select "Insert component" command.

STEP 02 In the window, select “Lower Base” component.

STEP 03 In the main window, pick the coordinate origin(0,0,0) as the location point and click “OK” to insert this component.

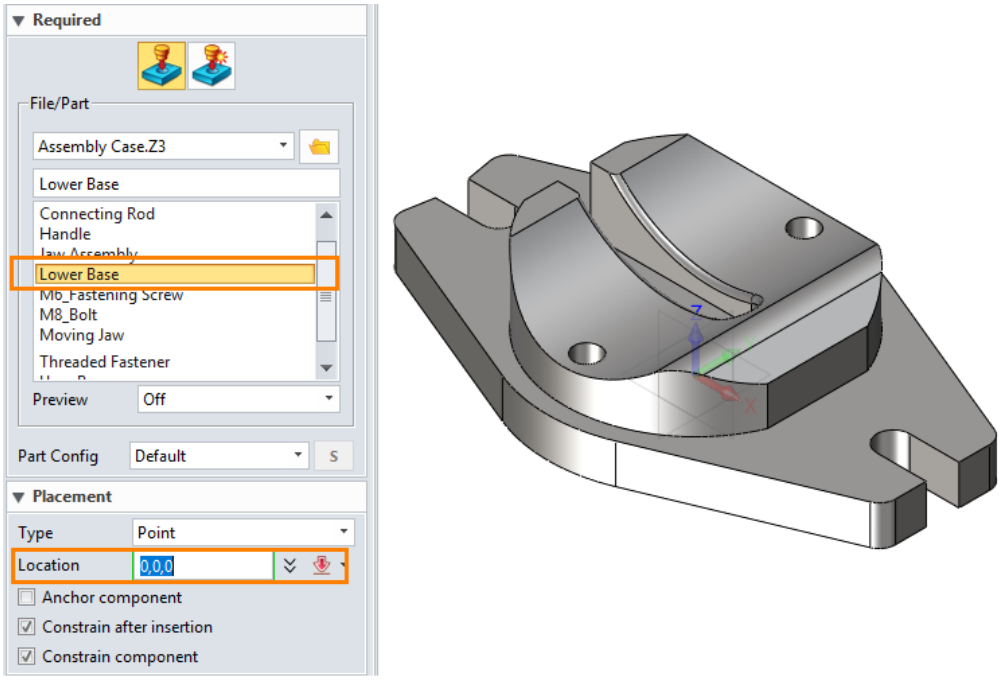


Figure51 Insert First Component

3. Anchor Component

STEP 01 Right click on the component in the main window, and select Anchor to fix this component.

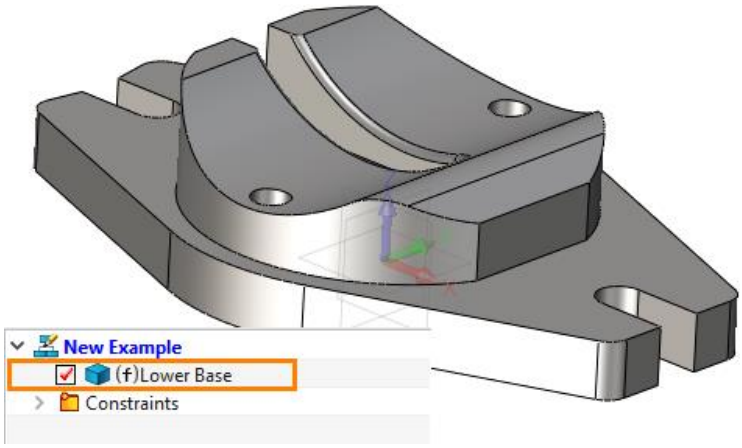


Figure52 Anchor Component

Note: After the component is anchored, it can not be moved or rotated.

4. Insert Second Component

STEP 01 Right click on the blank area in the main window and select "Insert component" command.

STEP 02 In the window, select "Uper Base" component.

STEP 03 In the main window, pick any point as the location point and click "OK" to insert it.

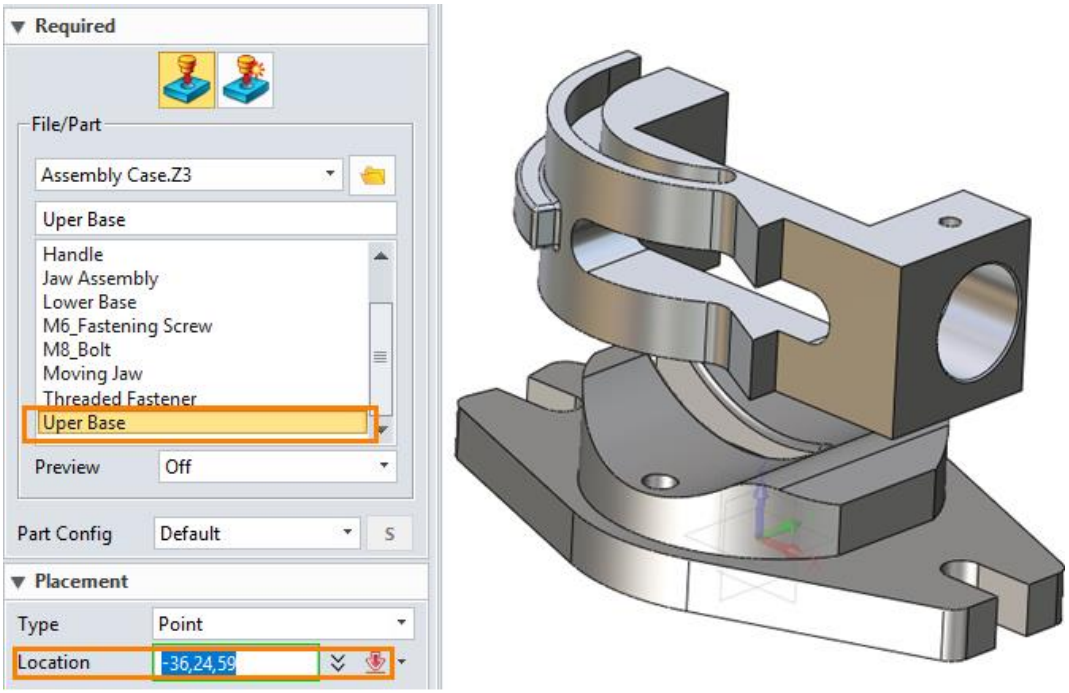


Figure53 Insert Second Component

5. Define the Constraint

STEP 01 The constraint dialog is pop-up automatically after the component insertion. Or right click on the blank area in the main window, and select "Constraint" command.

STEP 02 In the window, select face F3@Uper Base (Light Green color)and F35@Lower Base (Pink Color) as the constaint face.

STEP 03 In the Constraints or Mini bar, select the Concentric type, then click OK.

Note: If the auto constaint direction is not what want, we can use Same facing or Opposite to change the direction.

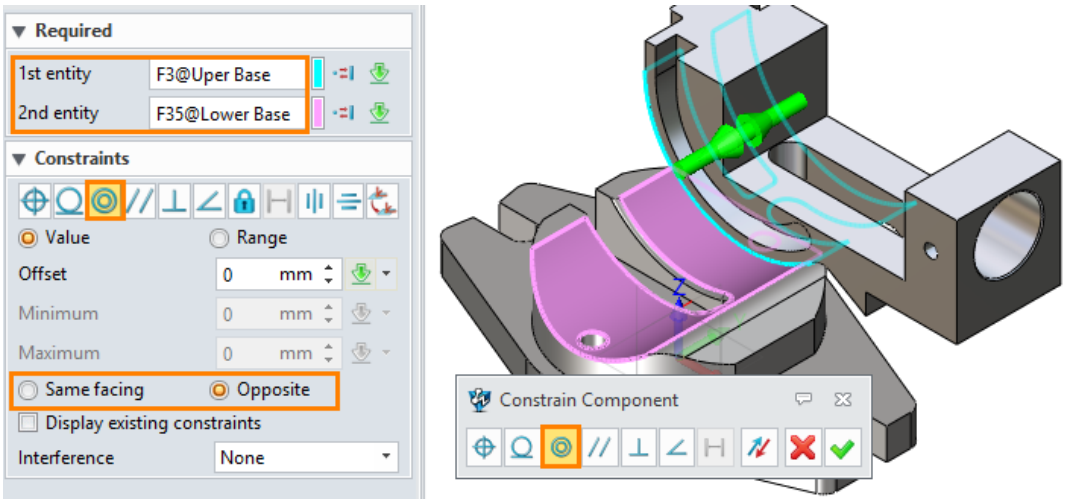


Figure54 Define Concentric Constraint

STEP 04 In the following, select "Middle" type.

STEP 05 Use the same method to select two groups of surface to finish the middle constraint.

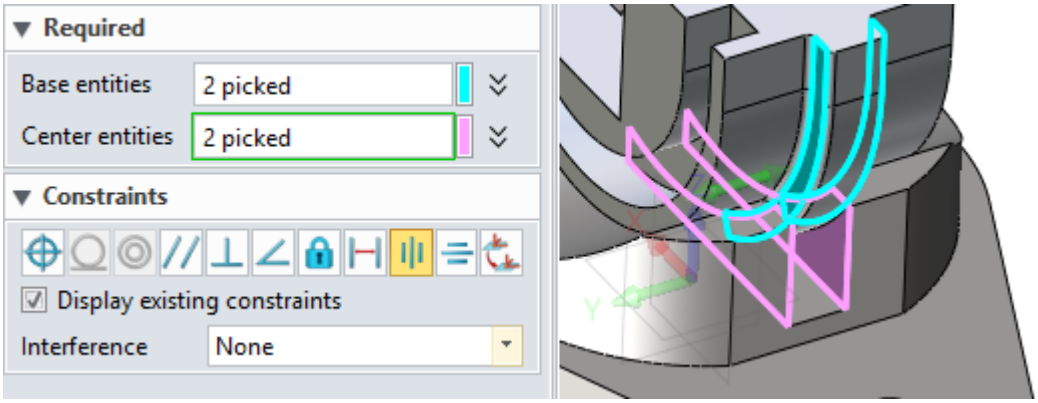


Figure55 Define Middle Constraint

STEP 06 In the following window, select "Parallel" type.

STEP 07 Use the same method to select face F12@Uper Base and YZ to finish the Parallel constraint.

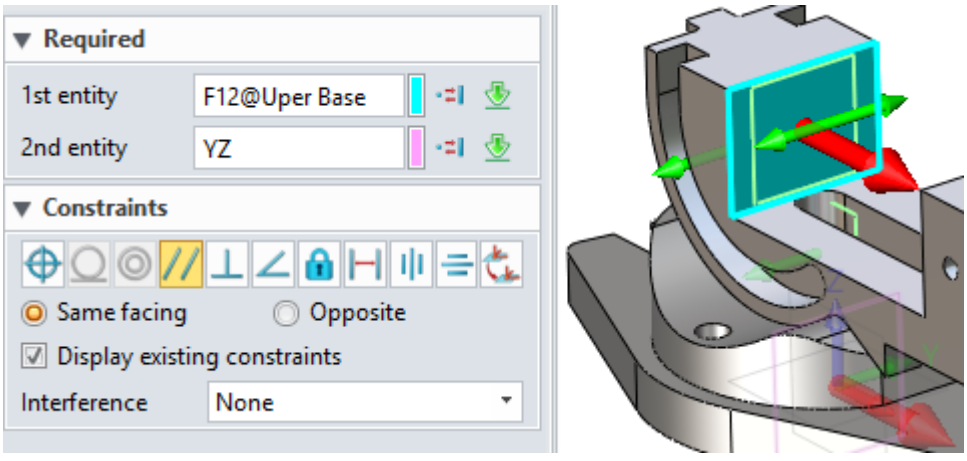


Figure56 Define Parallel Constraint

6. Insert Moving Jaw Component

STEP 01 Use the same method to insert the component Moving Jaw.

STEP 02 Use the same method to define the coincident constraint between F7@Moving Jaw and F33@Uper Base.

STEP 03 Define Parallel Contraint between F6@Moving Jaw and F37@Uper Base.

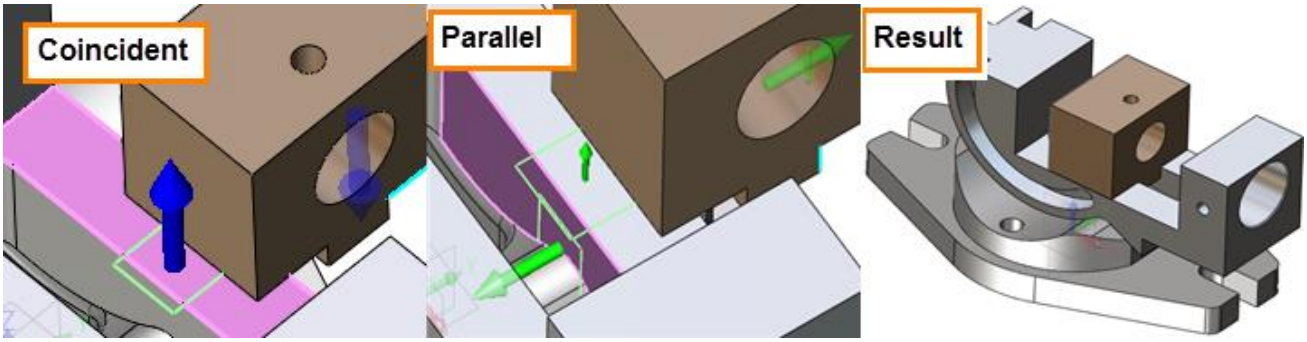


Figure57 Insert Moving Jaw

7. Insert Threaded Fastener Component

STEP 01 Insert the component Threaded Fastener.

STEP 02 Define the Concentric, Parallel and Coincident constraint.

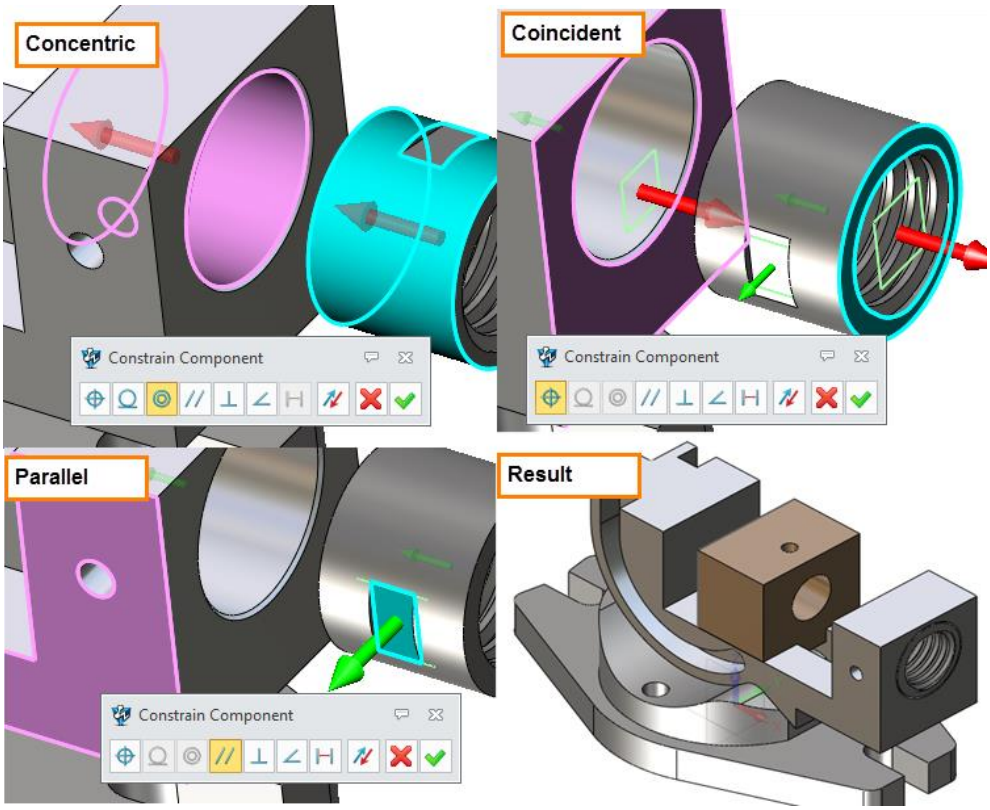


Figure58 Insert Threaded Fasetener

8. Insert Screw Rod Component

STEP 01 Insert the component Screw Rod.

STEP 02 Define the Concentric and Coincident constraint.

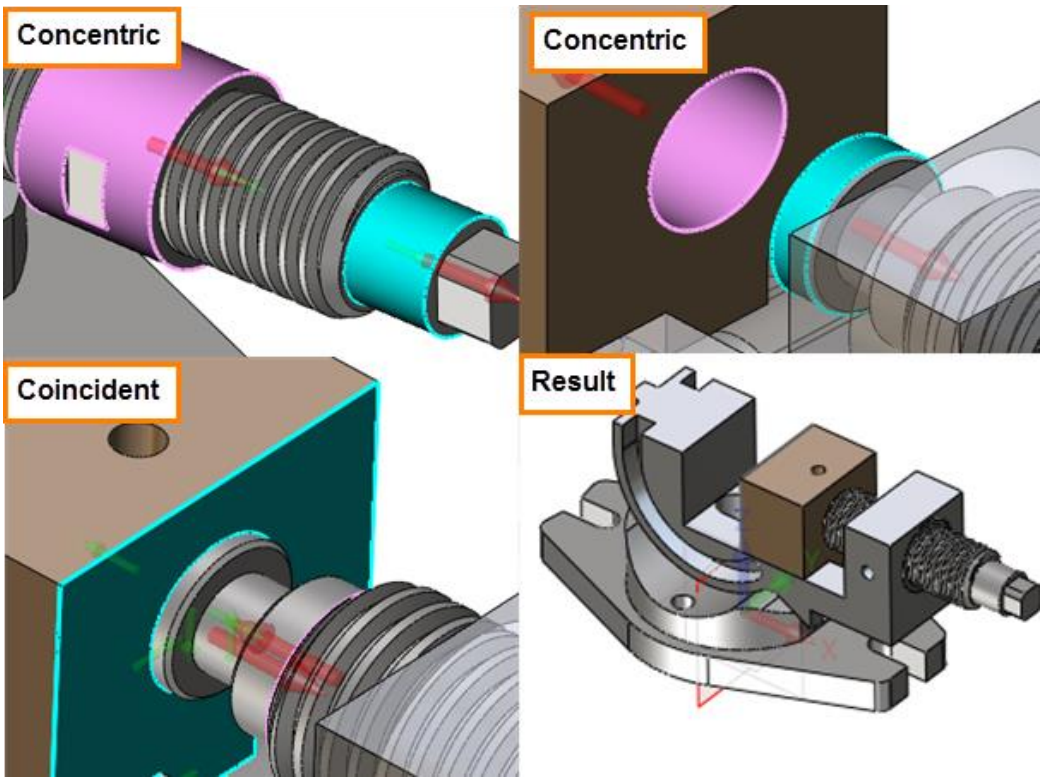


Figure59 Insert Screw Rob Component

9. Insert Connecting Rod Component

STEP 01 Insert the component Connecting Rod.

STEP 02 Define the Concentric and Coincident constraints.

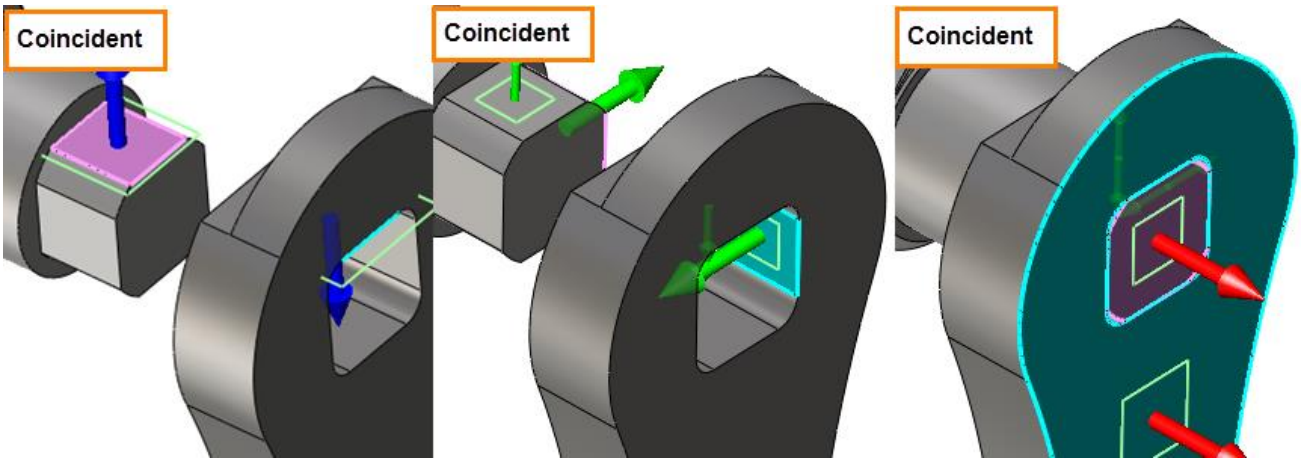


Figure60 Fig.64 Insert Connecting Rod

10. Insert Handle Component

STEP 01 Insert the component Handle.

STEP 02 Define the Concentric and Coincident constraint.

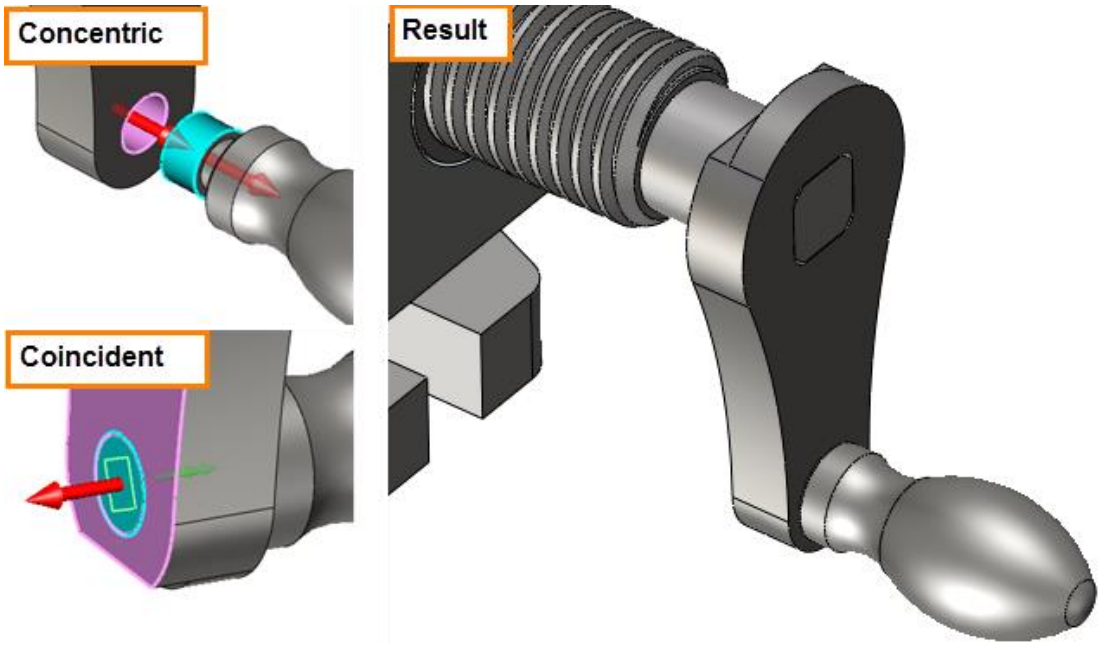


Figure61 Insert Handle

STEP 03 Turn on external datum. Then define the Coincident constrain between external components.

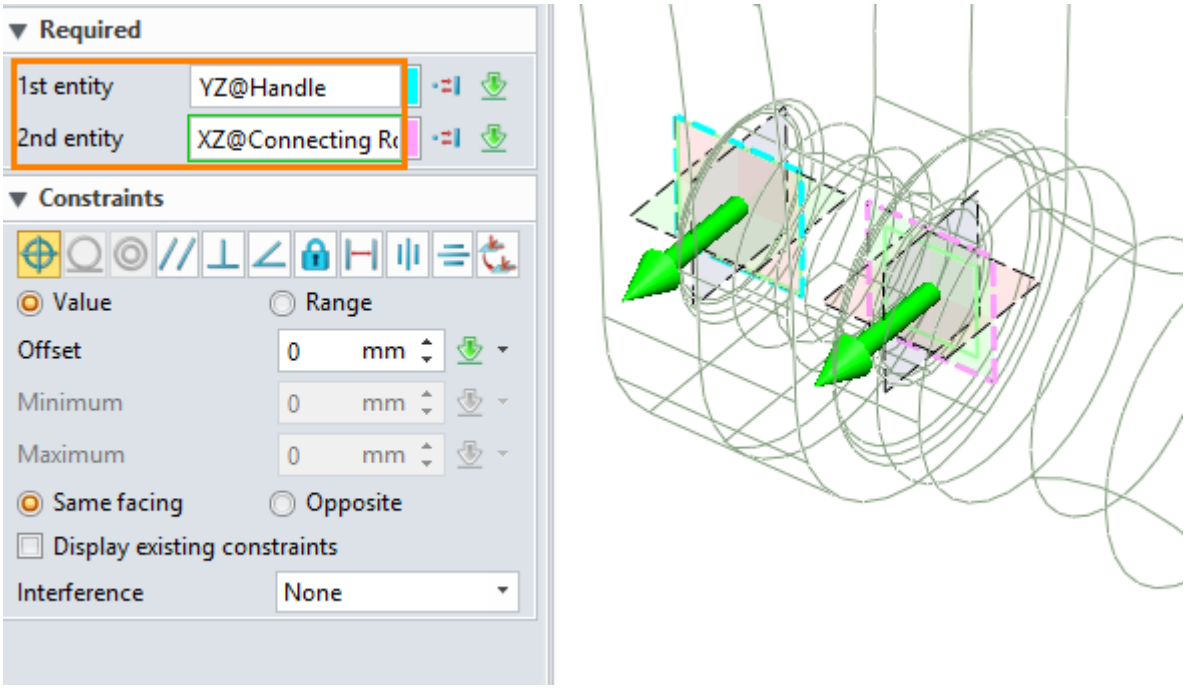


Figure62 Define Coincident Constraint with External Datum

Now we have finished the assembly modeling. With above steps, we can quickly to design our own assembly part.

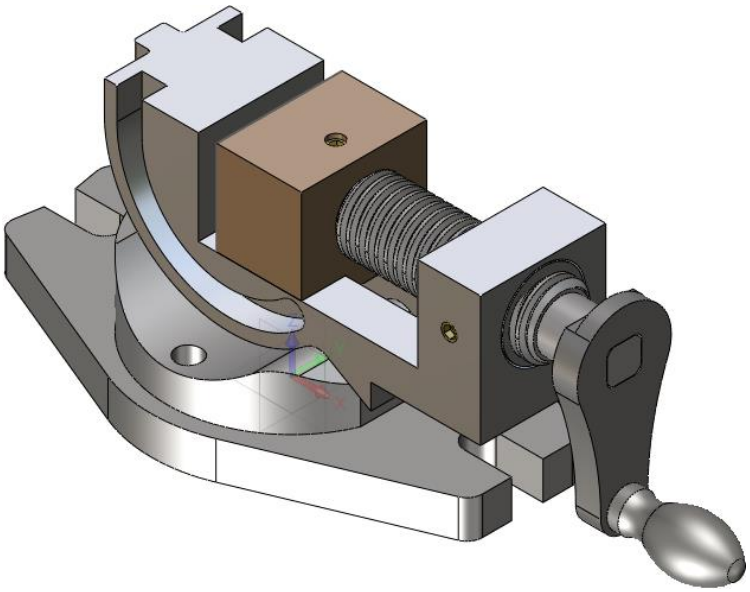


Figure63 Assembly Product