CATIA V5 Mechanical Design Expert

DASSAULT SYSTEMES

See what you mean

CATIA V5 Mechanical Design Expert

Student Handbook
Version 5 Release 19

40 Hours

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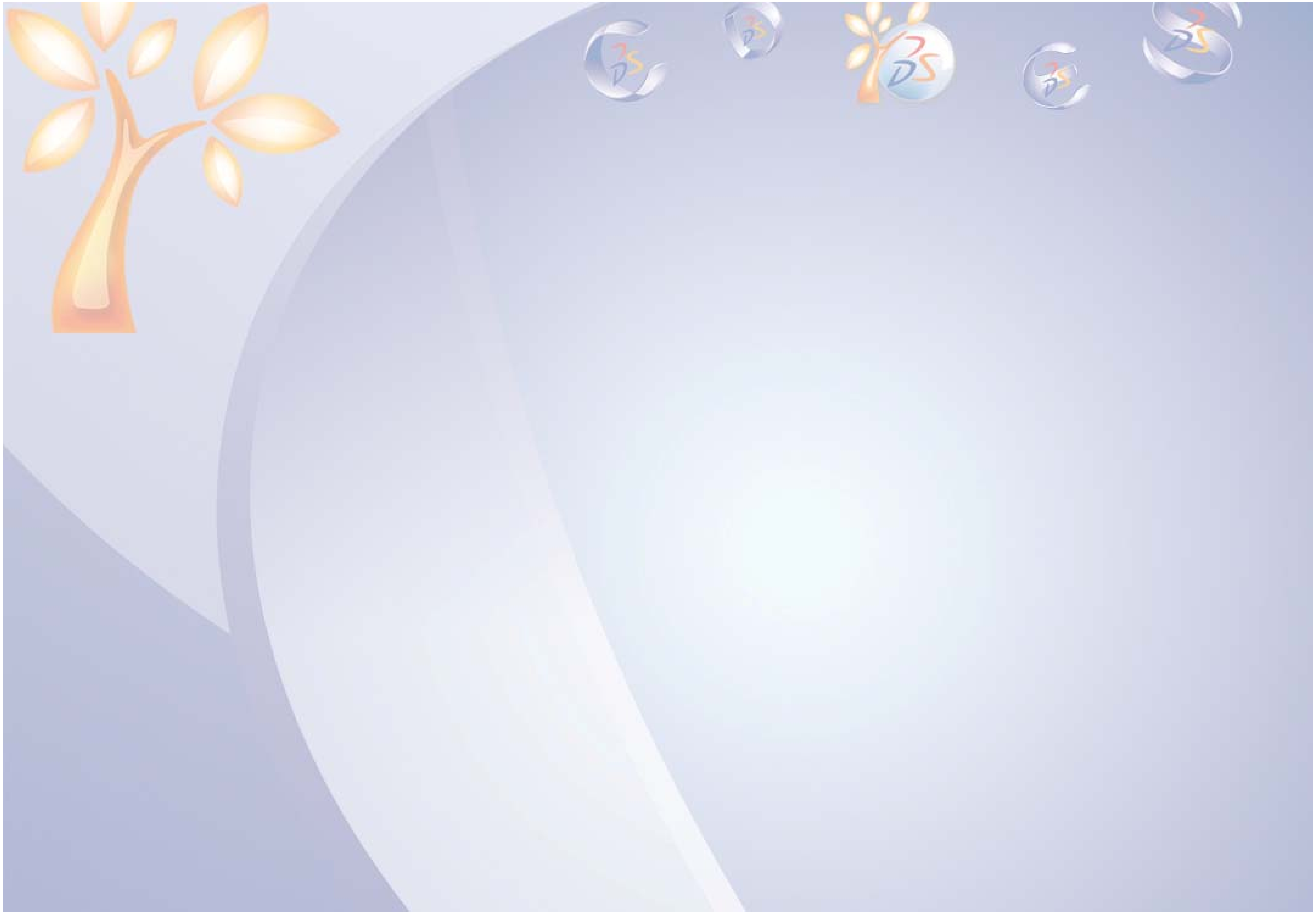
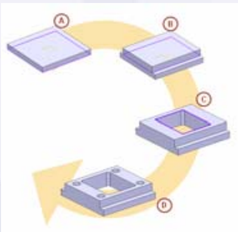


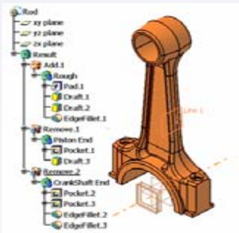
Introduction 1

Learning Objectives

Upon completion of this lesson you will be able to:

- ✓ Understand the importance of Parent/Child relationships.
- ✓ Review the model creation steps.
- ✓ Modify the design using the Define in Work Object command.
- ✓ Organize the features of the model into various bodies and geometrical sets.





4 Hours

Case Study

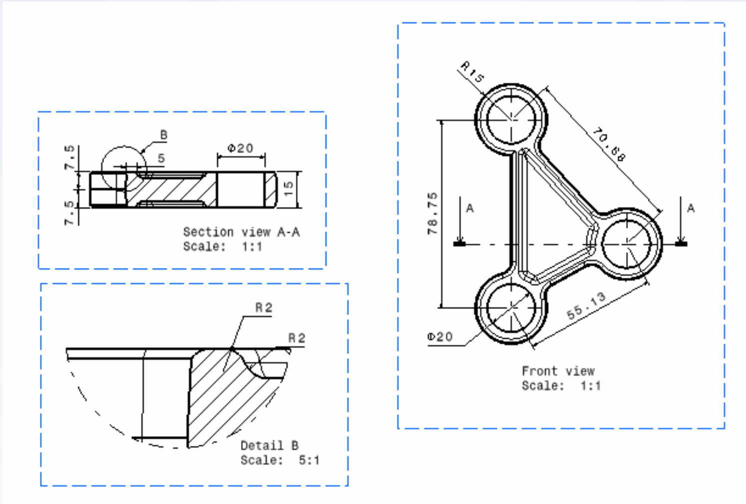
The case study for this lesson is the Hinge.

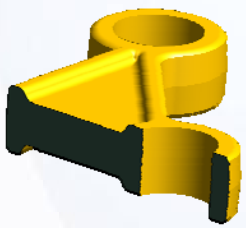
Design Intent

- ✓ The Hinge is a molded part that is used in an assembly.
- ✓ The part is symmetrical.
- ✓ The holes are centered on the bearings.

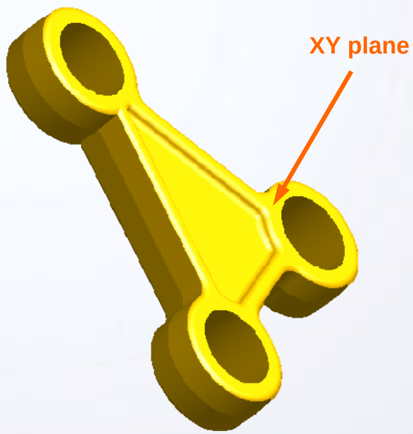
Stages in the Process

1. Review the User Interface.
2. Understanding importance of Parent/Child Relationships.
3. Organizing a Model.





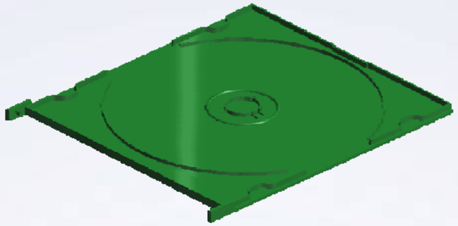
X-section of handle block

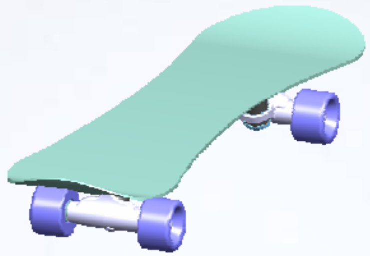


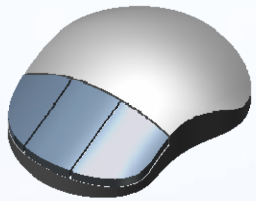
Review the User Interface

CATIA is mechanical design software. It is a feature-based, parametric solid modeling design tool that takes advantage of the easy-to-learn Windows graphical user interface. You can create fully associative 3D solid models with or without constraints while utilizing automatic or user-defined relations to capture design intent.

- ✓ The Part Design workbench lets you build solid 3D geometry. From the Part Design workbench you can access the Sketcher workbench and create 2D profiles that will become 3D model.
- ✓ The Assembly Design workbench allows you to bring components together to create the final product. You can design parts in the assembly context and use methods of designing assemblies that will aid in concurrent engineering, such as Skeleton models and publishing elements.
- ✓ The Generative Shape Design workbench lets you create surface and wireframe geometry. The surface and wireframe geometry allows you to create more complex solid models and gives more control over the shape of a model.



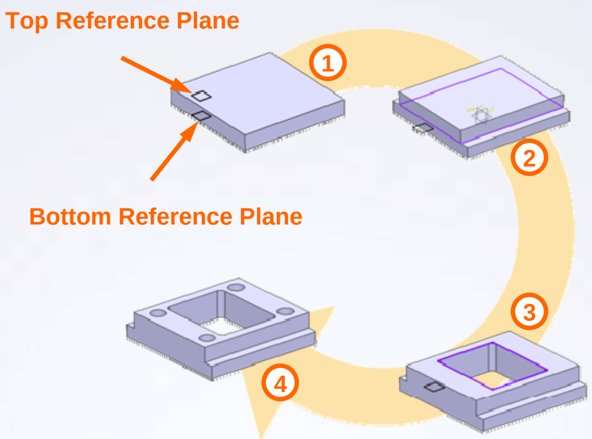




Importance of Parent/Child Relationships (1/2)

Design intent is a plan to construct solid model of a part, in order to convey its visual and functional aspects. The way a solid model is built can affect many aspects, including its flexibility to changes, its stability during the change process, and the resource requirements to compute a new result. Therefore, it is important to take the design intent into account to achieve an efficient solid model of the part.

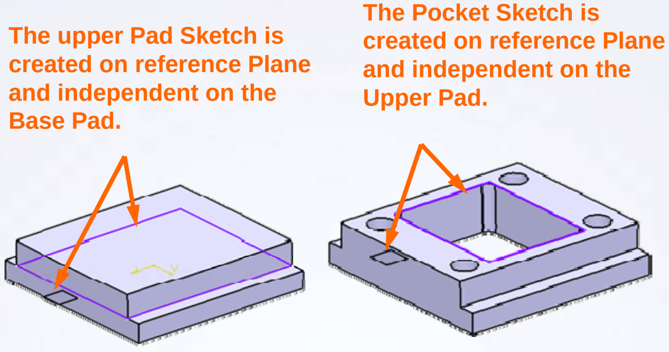
The dependency between one feature and the other is known as a parent/child relationship. Parent/Child Relationships are important in maintaining the design intent of the part. You should carefully consider choosing the best base feature, which parent/child relationships should exist, and what dimensions and feature order best reflect the planned design intent.



Importance of Parent/Child Relationships (2/2)

Many design practices are derived from company standards and need to be considered before modeling. Some common design practices are:

- ✓ Always choose the most stable feature in the model as the base feature.
- ✓ Try to avoid creating references to dress-up features such as fillets and chamfers. These features may be removed in downstream applications.
- ✓ Choose the best depth option for the application. For example, decide if a pocket is required to cut through the entire model. Creating the pocket with a dimensional depth is not recommended, because the depth of the feature it is cutting through may change; instead, create the pocket with an Up to Last depth.



On deletion of the Upper Pad, the pocket is not affected.

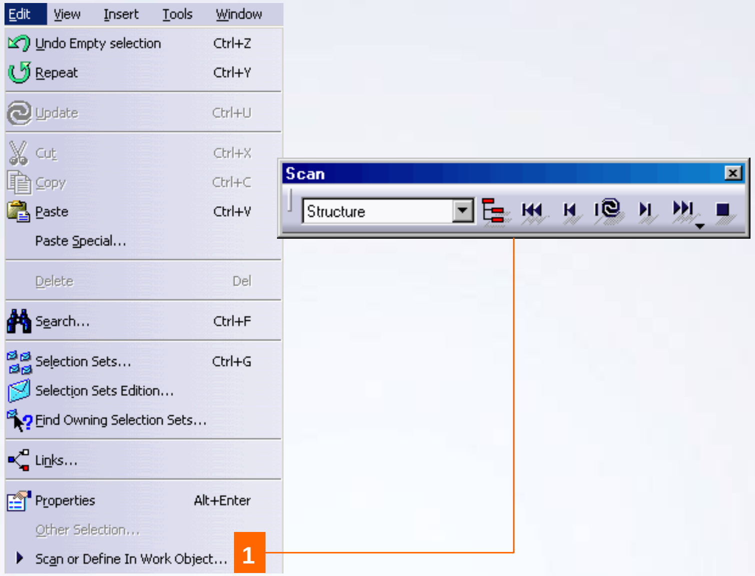
Main Tools / Menus (1/2)

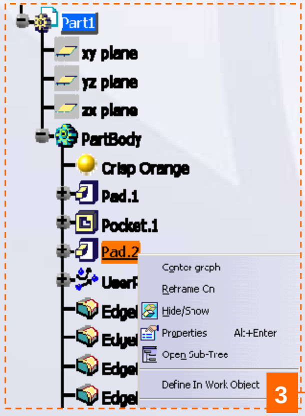
Investigating the Model

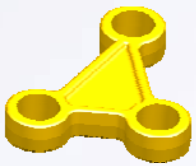
1 **Scan or Define In Work Object:** Helps you review how the model was created, feature by feature.

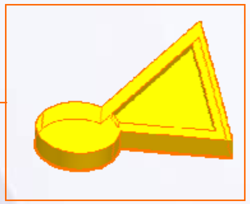
2 **Parent/Children:** Displays the parent and children of the selected feature and hence helps to display the relationships that exist in the model.

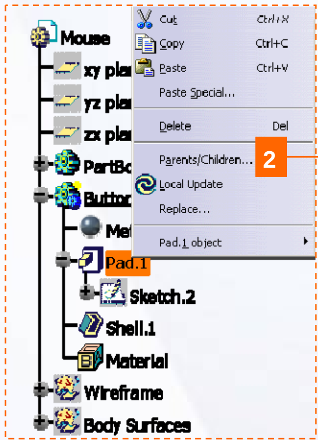
3 **Define in Work Object:** Activates the current selected feature disabling all child features. You can use Define in Work Object to review the feature, edit it or modify the design.

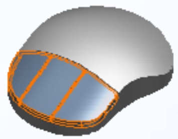


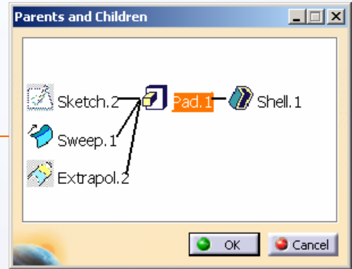












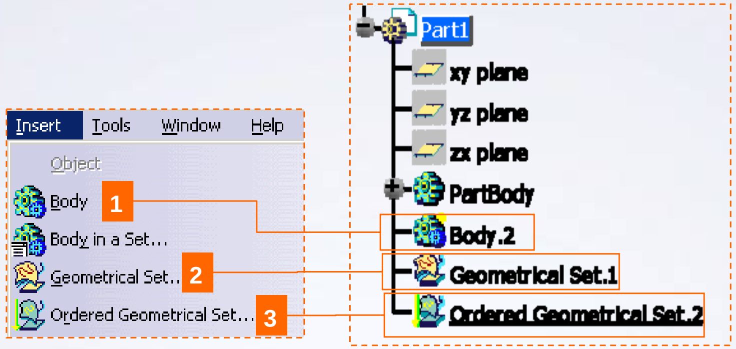
Main Tools / Menus (2/2)

Organizing a Solid Model

1 **Body :** Inserts a new body feature in the part.

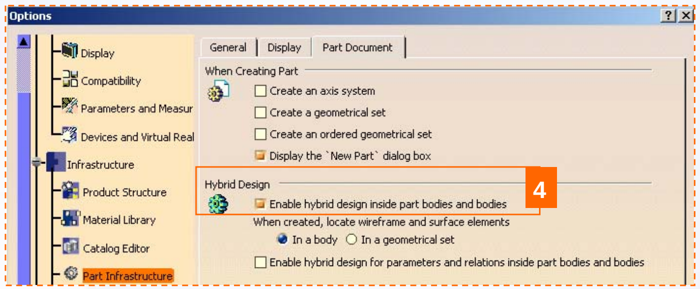
2 **Geometrical Set:** Inserts a new Geometrical Set in the part.

3 **Ordered Geometrical Set:** Inserts a new Ordered Geometrical Set in the part.



Tools > Options Setting

4 **Hybrid Design Option:** You can chose to embed hybrid design in your part by activating the **Enable hybrid design inside part bodies and bodies** option by accessing **Tools > Options > Infrastructure > Part Infrastructure > Part Document**.



Case Study: Introduction

Recap Exercise

40 min

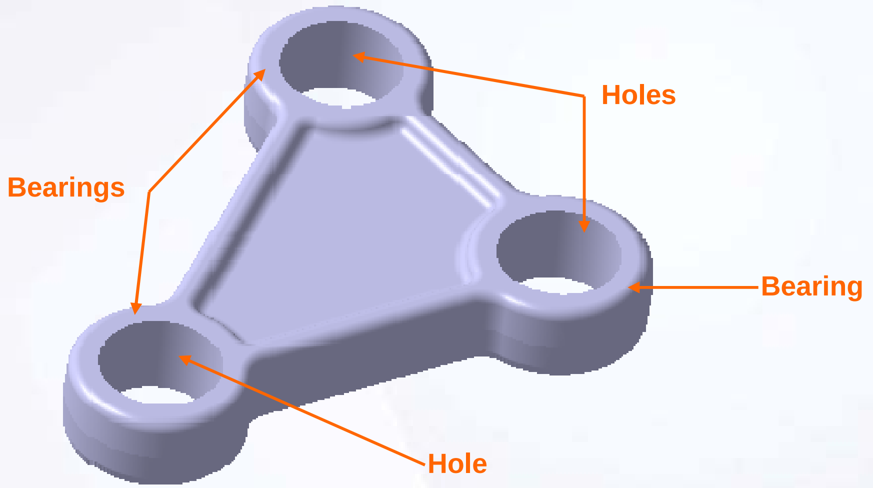
In this exercise you will create a model. The skills needed to create this model were covered in the Fundamentals course. There are no detailed instructions for this exercise. Instead a detailed drawing has been provided for your reference.

Case Study: Hinge

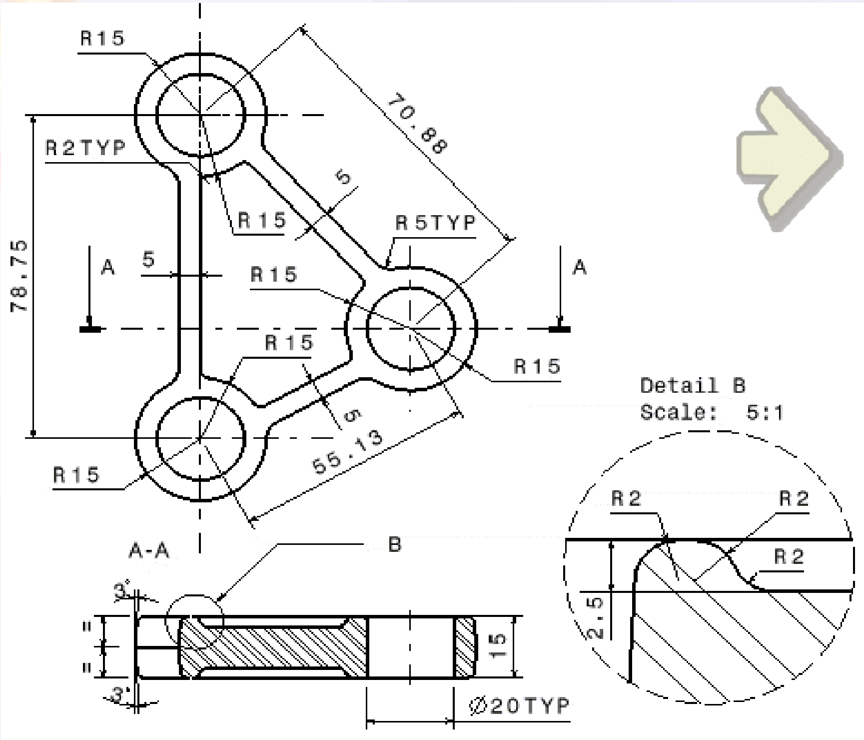
Before you begin to design the model you must analyze what is the best base feature? Which features will make good parent features? Which features will not make good parent features? What is the best orientation of the model? Which reference plane will you choose as the sketch support for the base feature? How will you constraint the base feature?

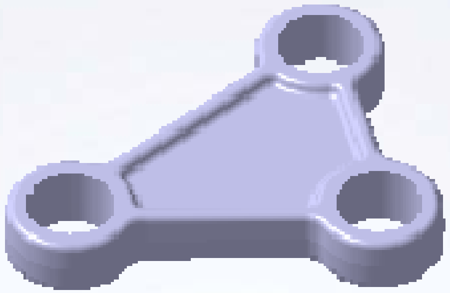
Consider the following:

- The Hinge is a molded part that is used in an assembly.
- The part is symmetrical.
- The holes are centered on the bearings.



Do It Yourself: Hinge (1/3)

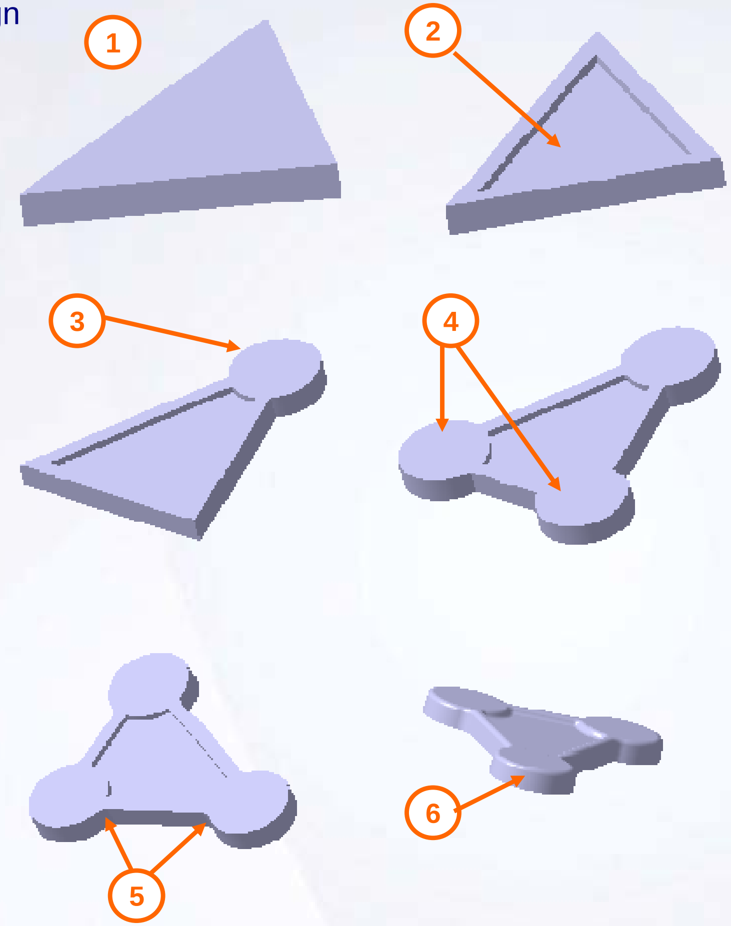




Do It Yourself: Hinge (2/3)

The following is a suggested method to design the Hinge:

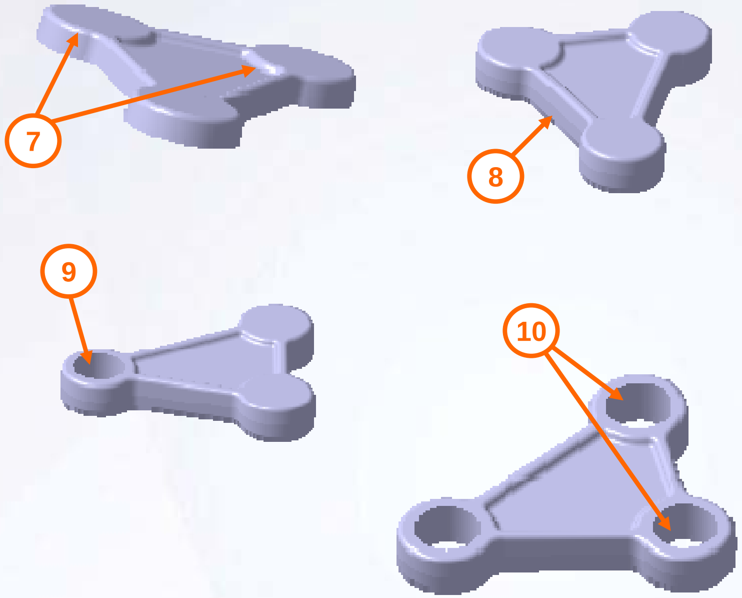
1. Create the base feature.
2. Create the pocket.
3. Create a second pad feature.
4. Create a user pattern.
5. Apply fillets.
6. Apply drafts.



Do It Yourself: Hinge (3/3)

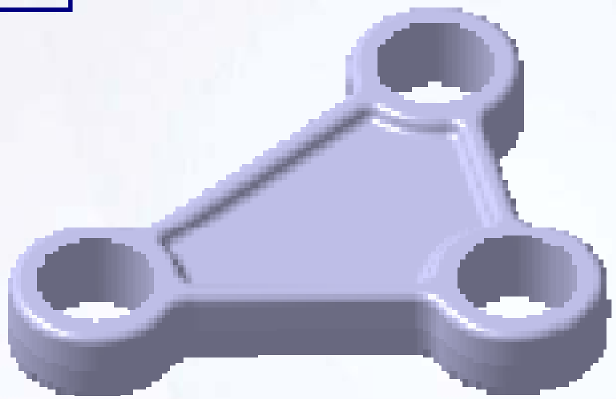
The following is a suggested method to design the Hinge (continued):

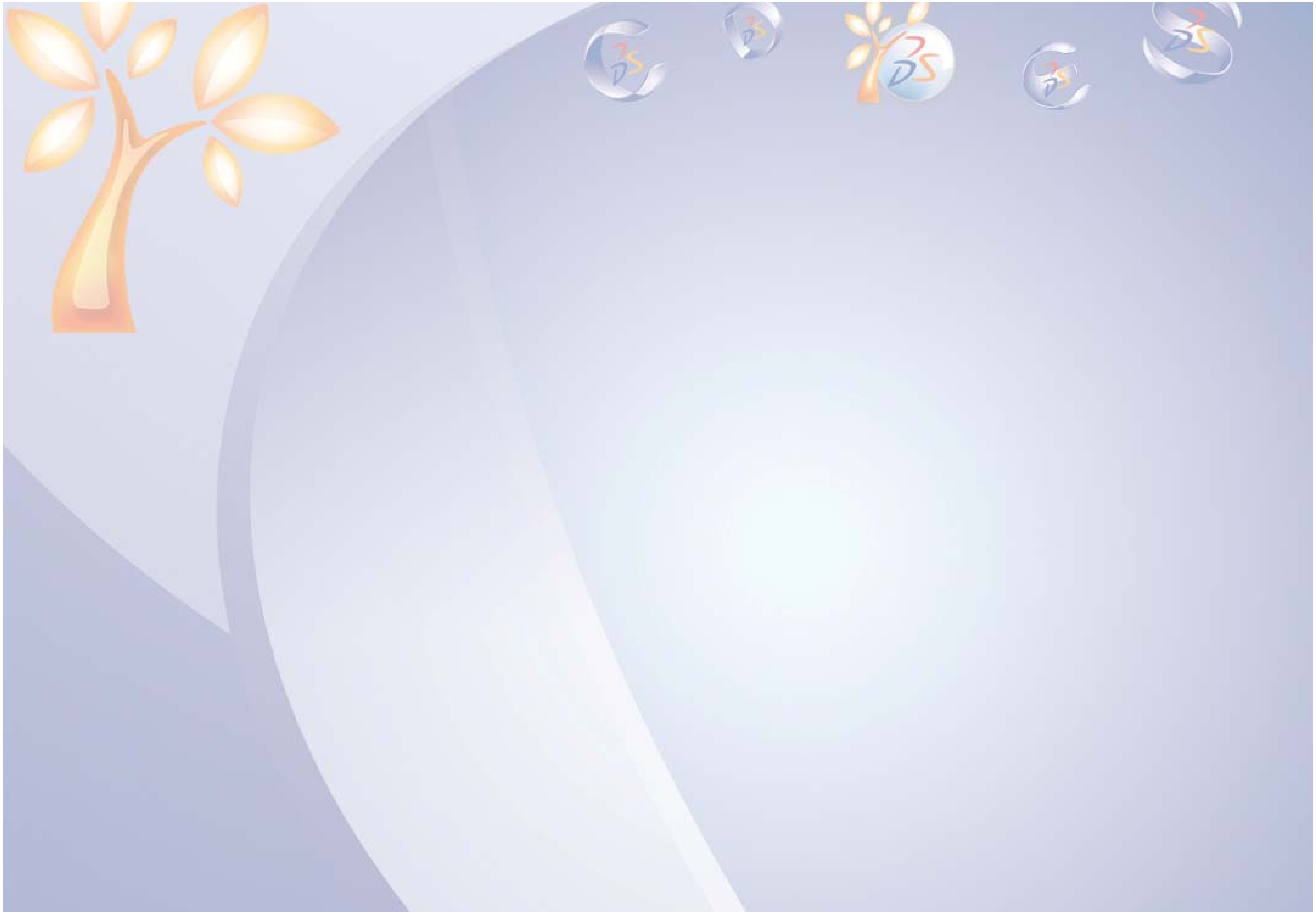
7. Create second set of fillets
8. Mirror the model.
9. Create hole.
10. Create a user pattern.



Case Study: Hinge Recap

- ✓ **Determine the best base feature.**
- ✓ **Determine the best orientation.**
- ✓ **Determine which parent/child relationships should be created and which should be avoided.**
- ✓ **Determine the best way to organize the model.**



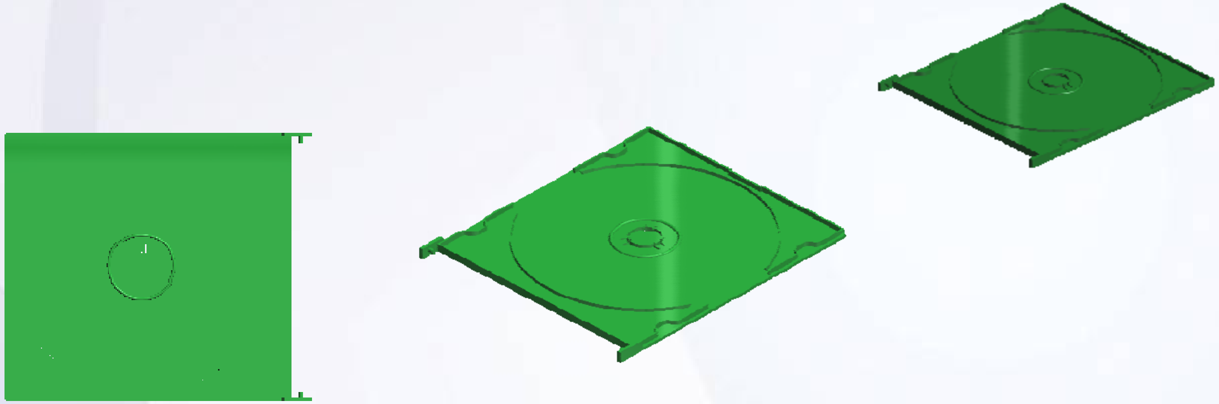


Design Complex Parts 2

Learning Objectives

Upon completion of this lesson you will be able to

- ✓ Create Advanced Sketch-Based Features.
- ✓ Multi Section solids.
- ✓ Create advanced Drafts.
- ✓ Advanced Dress-Up features.
- ✓ Use the Multi-Body Method.
- ✓ Create Multi-Model Links.



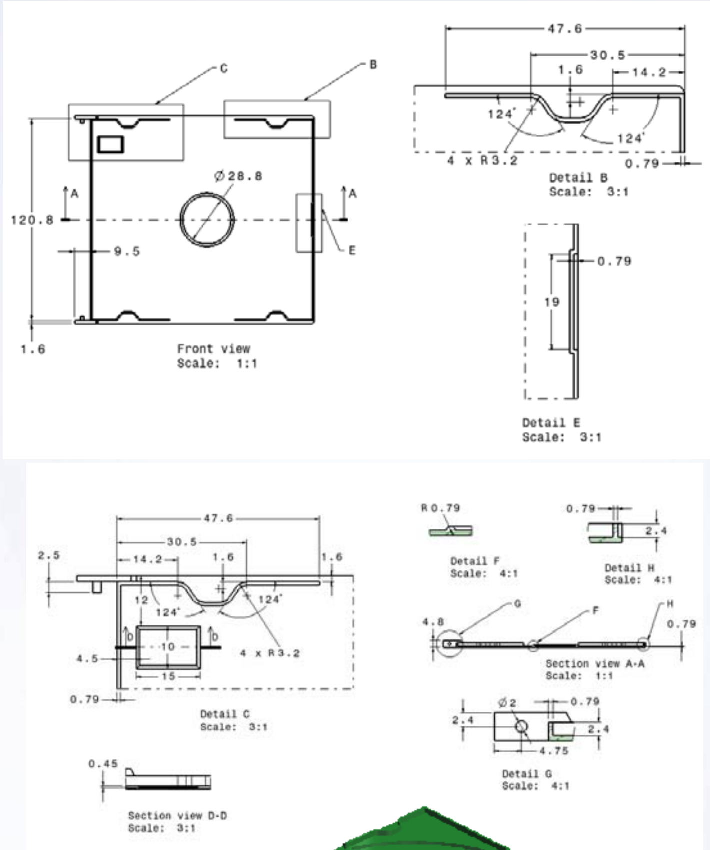
8 Hours

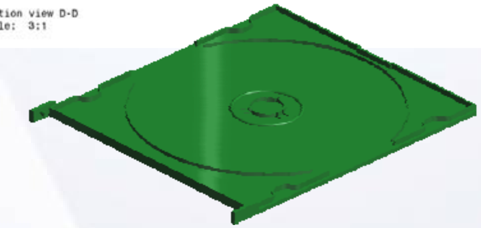
Case Study

The case study for this lesson is the Bottom Cover of a CD jewel case.

Design Intent

- ✓ Base feature must include overall dimensions supplied. Two sketches outlining the overall shape of the model are supplied to create a solid combine.
- ✓ Create each support as a single feature.
- ✓ Create a cut to simulate the logo using a removed multi-sections solid.
- ✓ By linking to disk holder and flex opening models, any changes that occur in the original source files will update in this file.
- ✓ Linked features must be kept in separate bodies.
- ✓ Do not display indented logo when it goes for manufacturing.



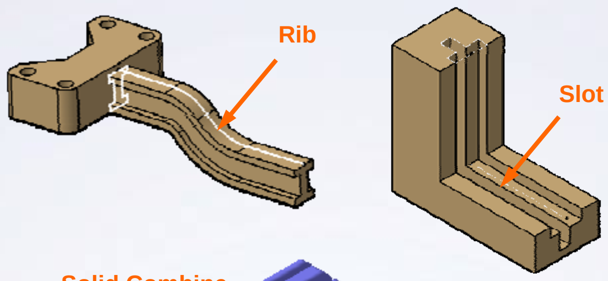


Stages in the Process

1. Create advanced sketched-based features.
2. Create dress-up features.
3. Use the Multi-Body method.
4. Create Multi-Model links.

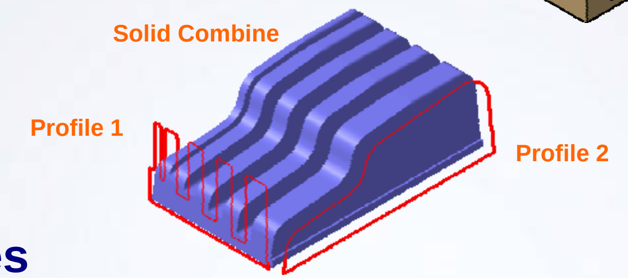
Create Advanced Sketch-Based Features

Ribs and slots are created by sweeping a profile along a center curve.



A Solid Combine feature is created by the intersection of two components. These can be:

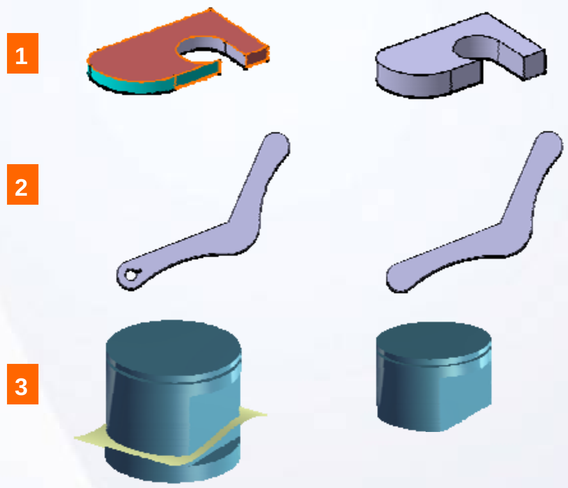
- Sketches
- Surfaces
- Sub-elements of sketches
- 3D Planar curves



Create Advanced Dress Up Features

The advanced dress up features are:

1. Thickness: Adds an over thickness to a face; used before machining the part.
2. Remove Faces: Used to simplify the geometry for downstream applications e.g. machining.
3. Replace Faces: Used to replace the planar solid surface with the surface.

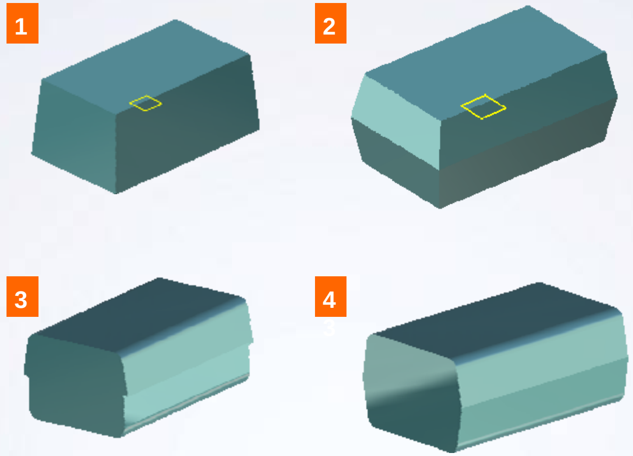


Create Advanced Draft

Advanced Drafts can be used to create basic line and reflect line drafts as well as drafts with two different angle values for complex parts.

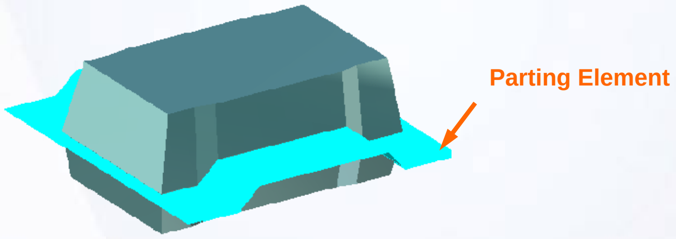
Different types of advanced Drafts are possible:

1. A Standard draft with one side draft
2. A Standard draft with two sides draft
3. A draft using a reflect line
4. A draft using two reflect lines



While creating advanced drafts, the parting element can be selected. A Parting line represents the location where two halves of the mold meet.

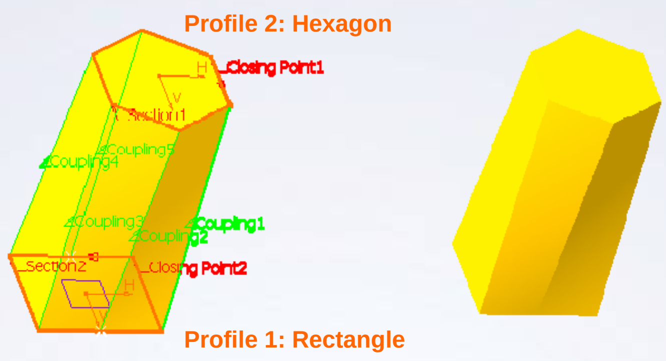
A Parting element can be a line, surface or a face.



Draft using a Parting element

Create Multi-Sections Solids (1/2)

A Multi-Sections solid can be positive (i.e., add material) or negative (i.e., subtract material). It is generated by two or more planar profiles swept along a spine.



Various types of Multi-Sections Solid are:

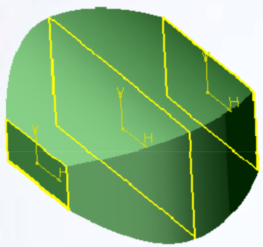
1. Simple Multi-Sections Solid: The selection order of the sections controls the shape of the result.

2. Multi-Sections Solid using Guide curve: The guide curves control the shape of the solid between the profiles. They must intersect the profile.

3. Multi-Sections Solid using Spine: The spine curve controls the shape of the features between the profiles.

4. Multi-Sections Solid Tangent to adjacent surfaces: The multi-sections solid is tangential to the adjacent solids / surfaces. Here the multi-sections solid acts as a transitional feature.

1



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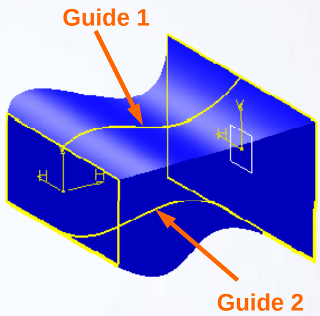

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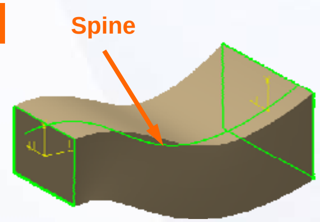

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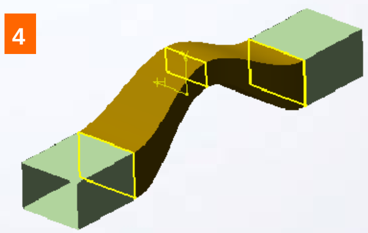


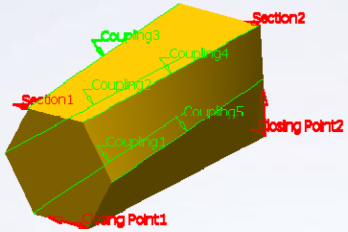
Create Multi-Sections Solids (2/2)

5. Multi-Sections Solid using Couplings: The curves are coupled according to different criteria. These are as follows:
   a. Ratio: The ratio of each section's length.
   b. Tangency: Uses tangency discontinuity points.
   c. Tangency then Curvature: uses the tangency discontinuity points first and then later the curvature discontinuity points.
   d. Vertices: Uses the section's vertices.
   e. Manual coupling: Used when various sections do not have the same number of vertices.
6. Multi-Sections Solid using Relimitations: By clearing the Relimitation options in the Relimitation tab, the result can be extended to the length of the spine or the guide curves.

Recommendations to avoid twisted surfaces:

✓ Choose appropriate Closing Points.
✓ Keep consistent directions.

5



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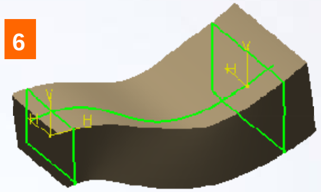
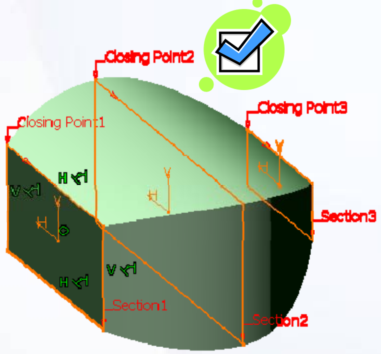
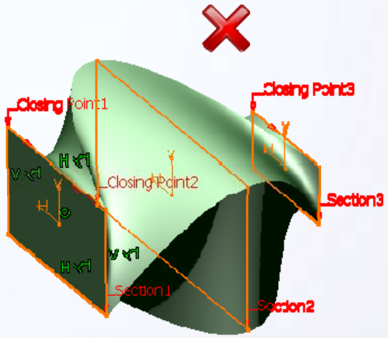




Closing Point and Direction is correct



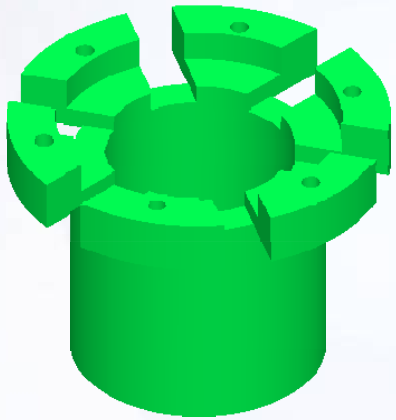
Closing Point selected is incorrect

Use the Multi-Body Method

The Multi-Body Method allows you to design a complex part using simple bodies. Each body acts independently in the model. The final part is obtained by combining these bodies using Boolean operations.

The advantages of using the Multi-Body method are as follows:

- ✓ It provides an organized approach to modeling complex parts.
- ✓ Solid features within a body can be hidden independently of the rest of the model.
- ✓ Groups of geometry can be de-activated by de-activating the body.
- ✓ Complex geometry is easier to create within a focused area of the model.
- ✓ The model will update faster due to the organized structure.



Boolean Operations

A. Assemble: The result will depend on the polarity of Body.2. A negative feature (pocket or groove), will remove material from the PartBody, a positive feature will add material.

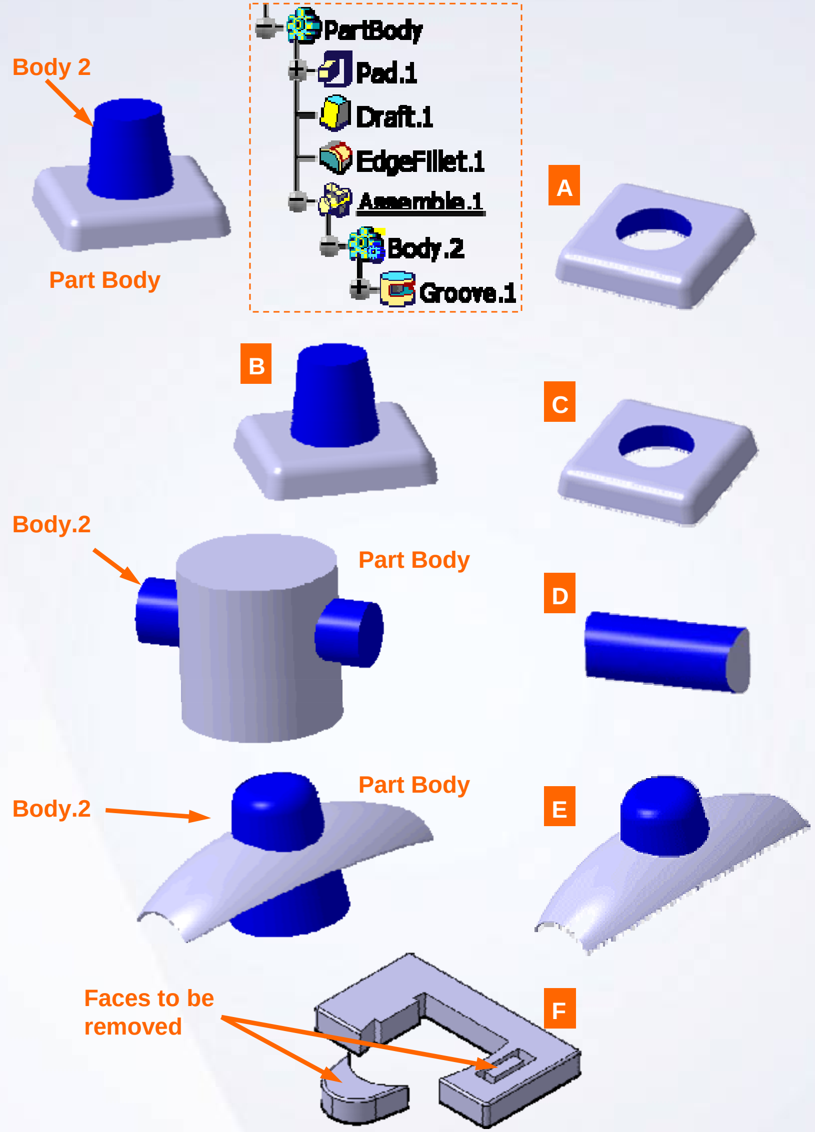
B. Add: A union of Body.2 and PartBody.

C. Remove: Body.2 will cut PartBody.

D. Intersect: The resulting solid is the material common to the intersecting elements.

E. Union Trim: This operation is a union of the two bodies with the option to remove or keep selected faces.

F. Remove Lump: A lump is material that is completely disconnected from the remainder of a single body, and may appear after certain operations. This operation is used select the faces to remove.



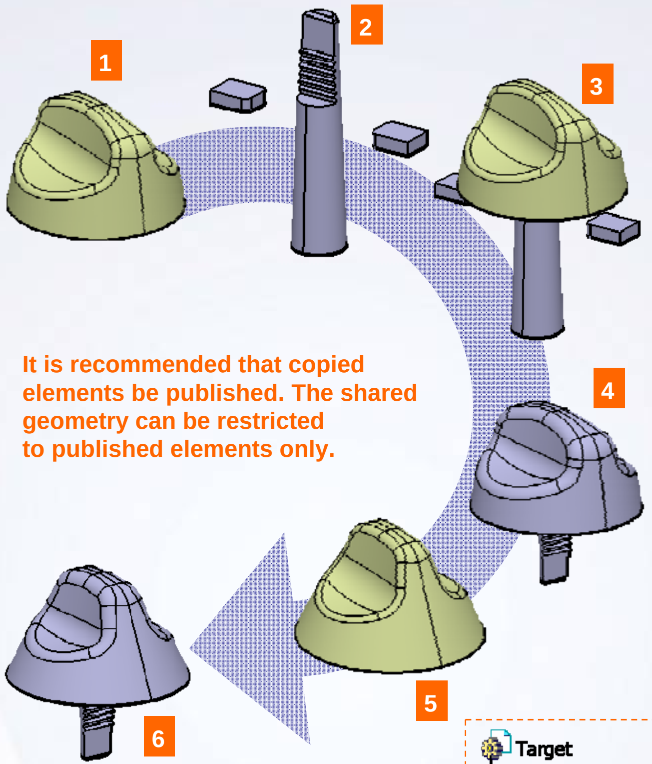
Create Multi-model Links

The use of Multi-Model Links enable you to design a model using elements from another model. This will enable you to update the part automatically if changes occur in the source model. To create links:

1. Copy a body in the source model.
2. In the target model, right-click on the Part and click Paste Special from the contextual menu.
3. Select As Result with Link. The Source PartBody is copied into the target model.
4. Complete the target model with the new body.
5. Modify the source model.
6. The target model is updated to take into account changes to the source model.

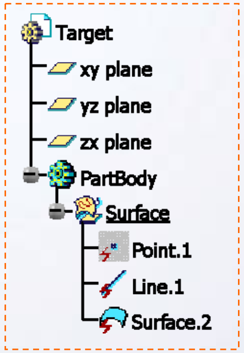
Choose the Paste Special option that best meets your design requirements:

- As Specified in the Part Document: The copied elements can be edited separately in the target part. A surface cannot be pasted in this way.
- As Result: The copied elements cannot be edited in the target part and are not linked to the source part.
- As Result with Link: The copied elements cannot be edited in the target part but are linked to the source part. A geometrical set cannot be pasted in this way.



It is recommended that copied elements be published. The shared geometry can be restricted to published elements only.

Note: If you copy a geometrical set and select the PartBody for the paste the geometrical set will be inserted under the Part Body and NOT at the same level as the PartBody.



Main Tools (1/3)

Advanced Sketch Based Features

1 **Rib:** Creates a positive solid from a profile swept along the center curve.

2 **Slot:** Creates a negative solid from a profile swept along the center curve.

3 **Solid Combine:** Creates an intersection solid from the two extruded profiles.

4 **Multi-sections Solid:** Creates a positive solid joining multiple sections.

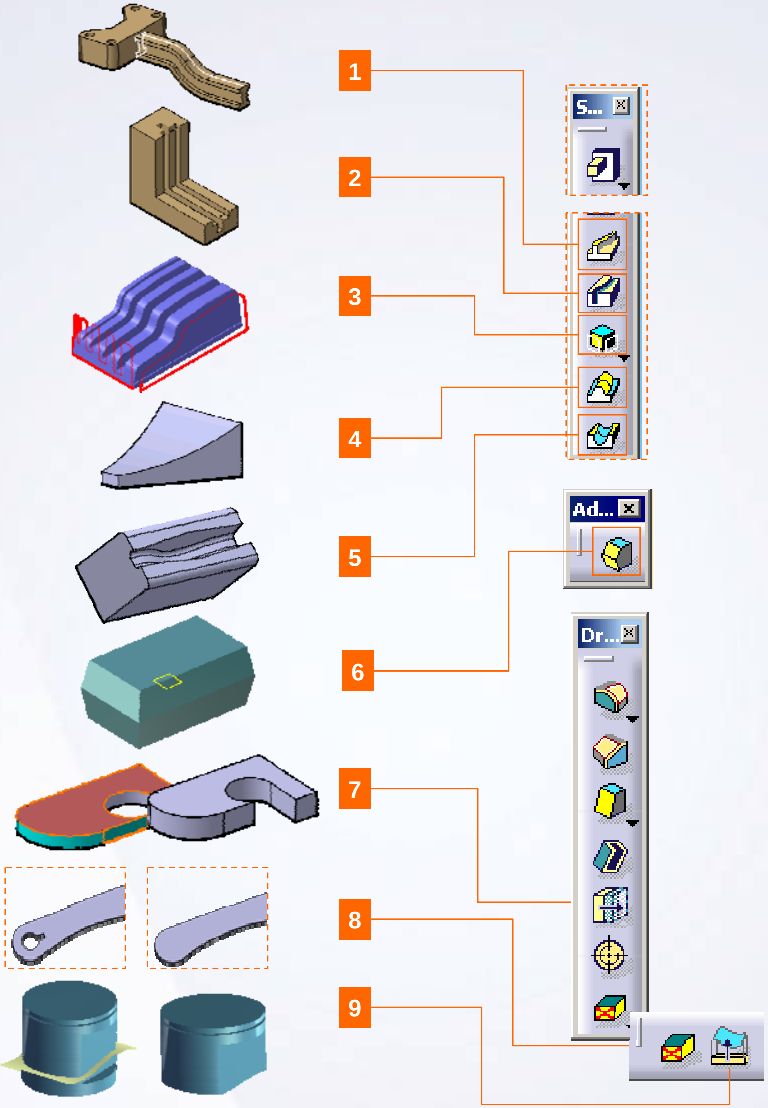
5 **Remove Multi-sections Solid:** Extrudes a solid up to a surface.

6 **Advanced Draft:** Creates a basic line and reflect line drafts with two different draft angles for complex parts.

7 **Thickness:** Adds / Removes thickness to a selected face or a surface.

8 **Remove Face:** Removes selected faces to simplify the geometry for finite element analysis / downstream applications.

9 **Replace Face:** Extrudes a solid up to a surface.



Main Tools (2/3)

Boolean Operations

10 **Assemble:** Creates a union of two bodies, the union respects the true nature of the bodies. (Positive features add material, negative features remove material).

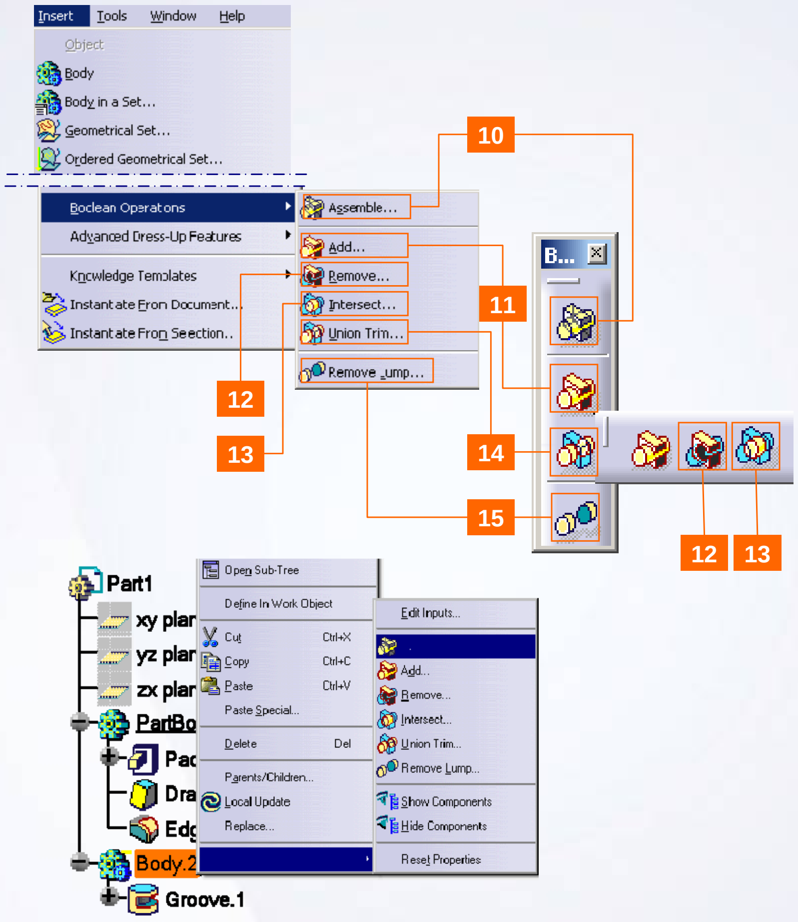
11 **Add:** Creates a union of two bodies.

12 **Remove:** Removes selected body from the PartBody.

13 **Intersect:** Creates an intersection solid from the selected bodies.

14 **Union Trim:** Creates an intersection solid from the selected bodies with an option to remove or keep one side.

15 **Remove Lump:** Removes selected faces (lumps and cavities).

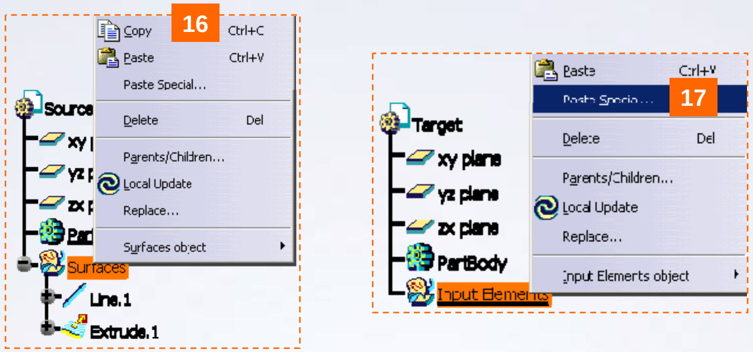


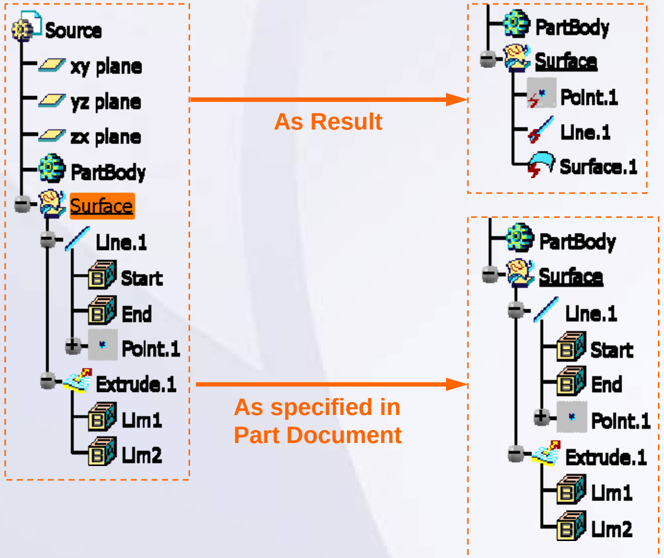
Main Tools (3/3)

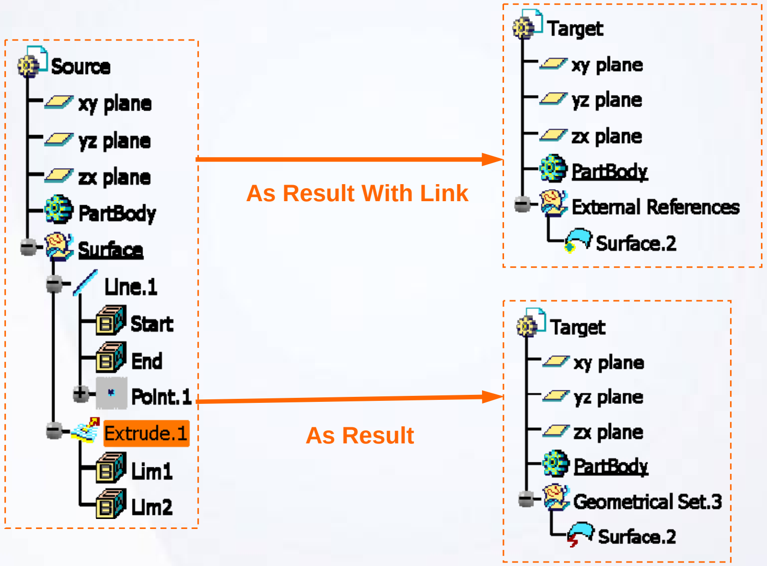
Multi-Model Links

16 **Copy:** Copies the selected features.

17 **Paste Special:** Pastes the selected features into the destination.







Exercise: Rib and Slot

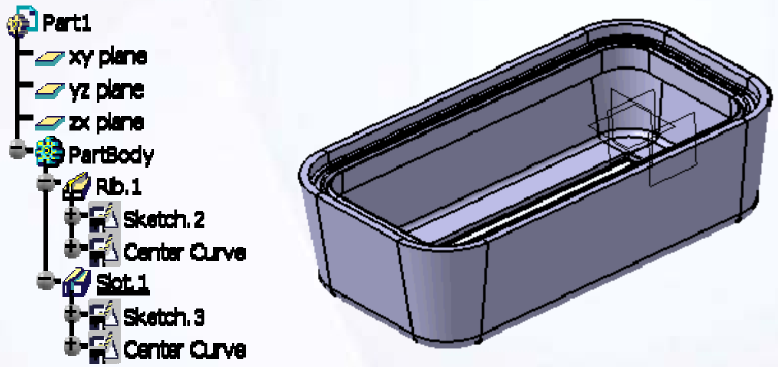
Recap Exercise

15 min

In this exercise, you will create a new model and use the tools learned in the lesson to create a rib and a slot feature. High-level instruction is provided for this exercise.

By the end of this exercise you will be able to:

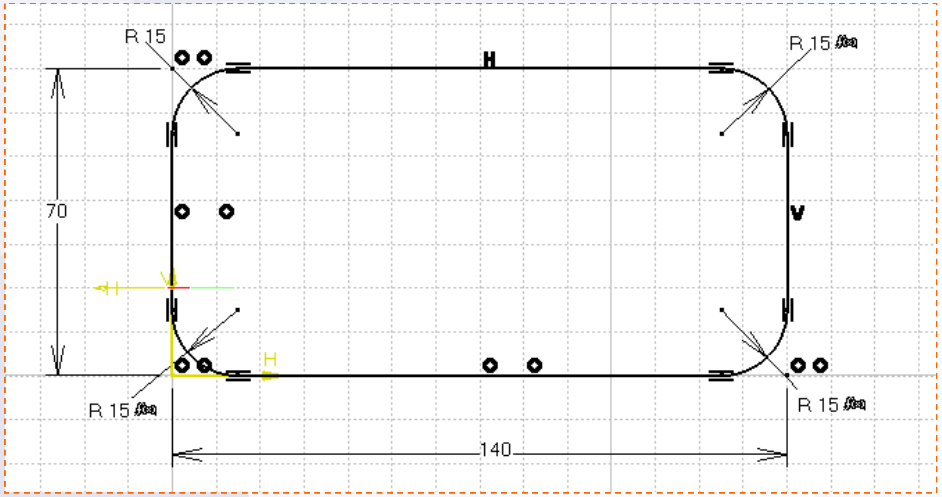
- Create a Rib Feature
- Create a Slot Feature



Do it Yourself (1/4)

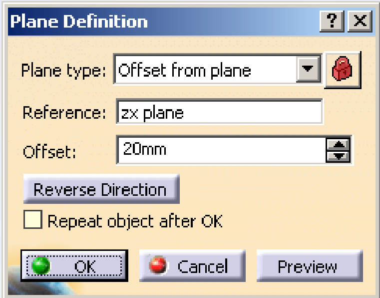
1. **Create a new part file.**
   - Create a new part file called Ex8B.

2. **Create the center curve sketch.**
   - Create a positioned sketch as shown for the center curve.
   - Rename the sketch to [Center Curve].

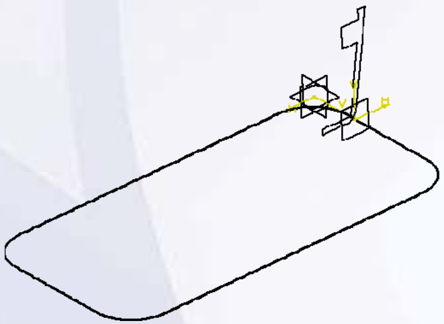


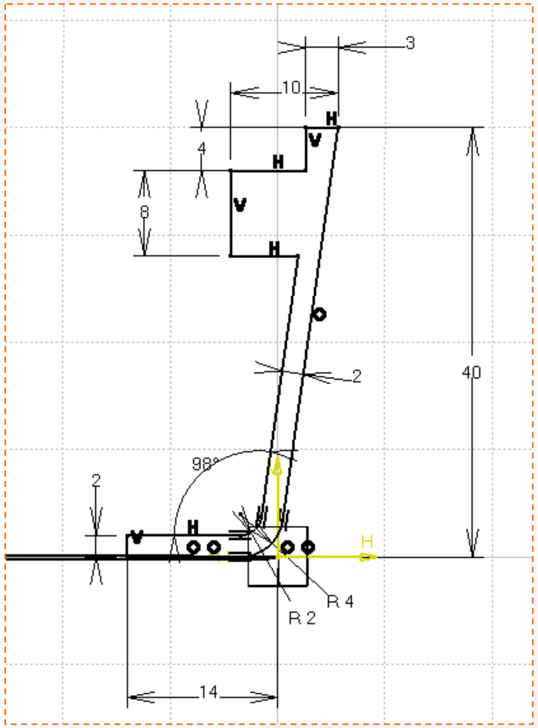
Do it Yourself (2/4)

3. **Create a reference plane.**
   - Create an offset plane as shown.



4. **Create a profile sketch for the rib.**
   - Create a positioned sketch as shown for the rib profile.

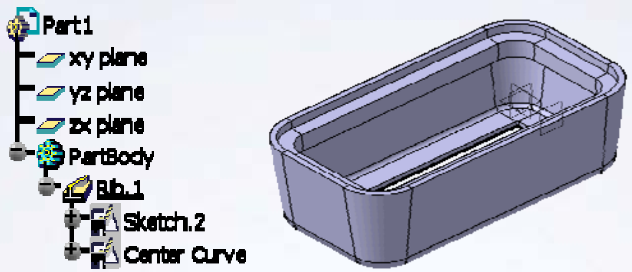




Do it Yourself (3/4)

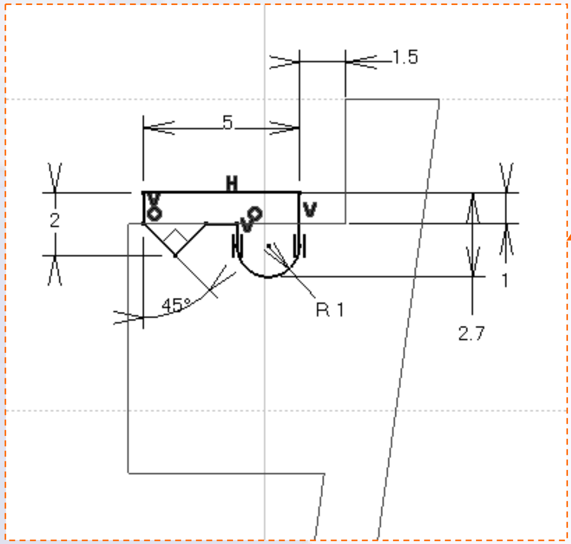
5. Create the rib feature.

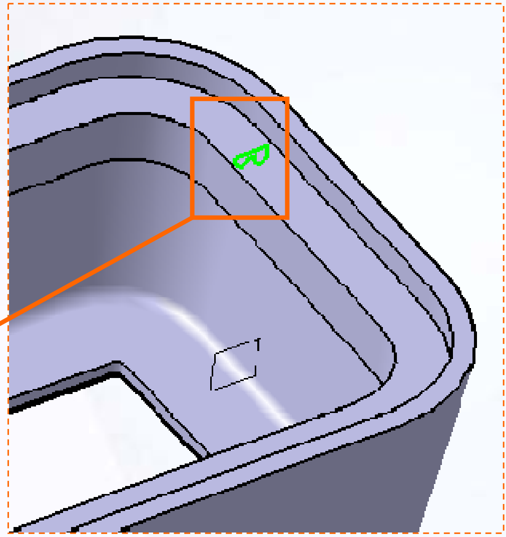
- Use the center curve and profile sketch to create a rib feature.



7. Create a profile sketch for the slot.

- Create a positioned sketch as shown for the slot profile.

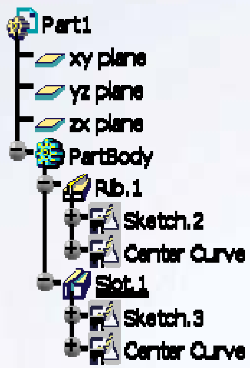


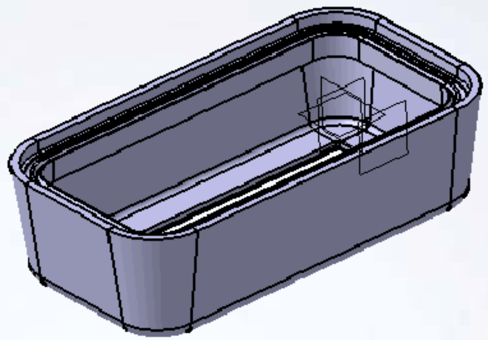


Do it Yourself (4/4)

8. Create a slot feature.

- Create a slot feature using the sketch created in the last step as the profile and the Center Curve sketch as the trajectory.

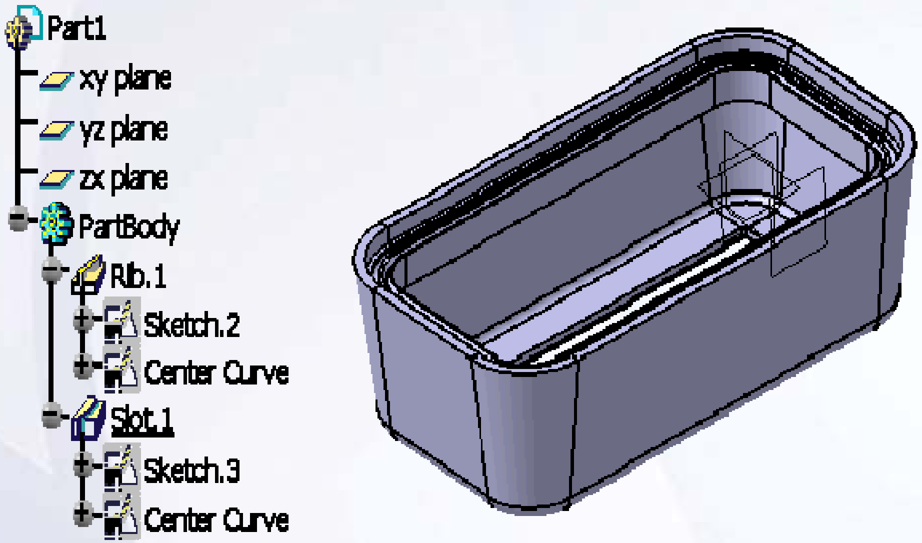




9. Close the file without saving it.

Exercise Recap: Rib and Slot

- ✓ Create a rib
- ✓ Create a slot



Exercise: Rib and Multi-section Solid

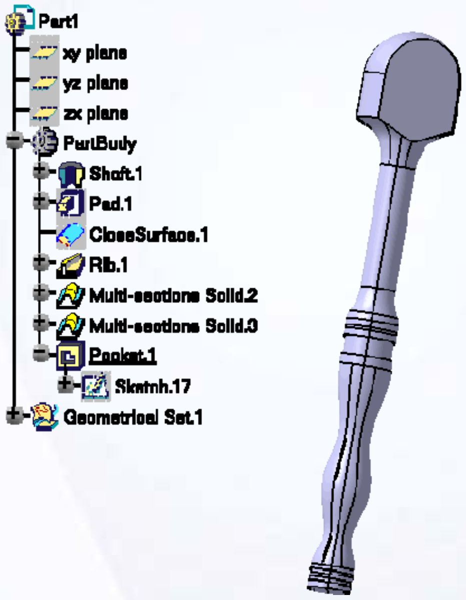
Recap Exercise

30 min

In this exercise, you will open an existing model and use the tools learnt in the lesson to create rib and Multi-sections Solid features. High-level instruction is provided for this exercise.

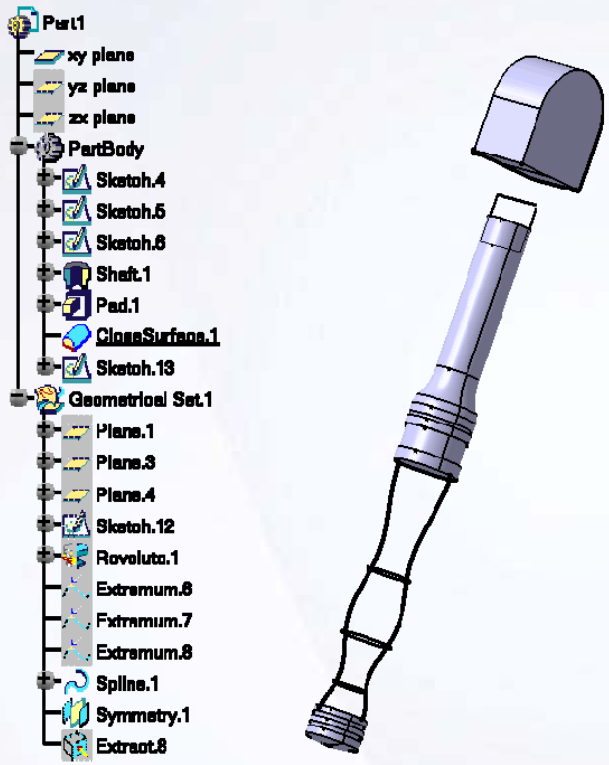
By the end of this exercise you will be able to:

- **Create a Rib Feature**
- **Create a Multi-sections Solid Feature**



Do it Yourself (1/7)

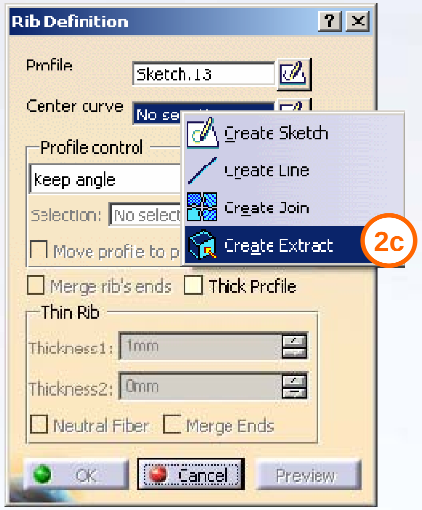
1. **Open the existing part file.**
   - Open Wrench.CATPart. Notice some features have already been created.

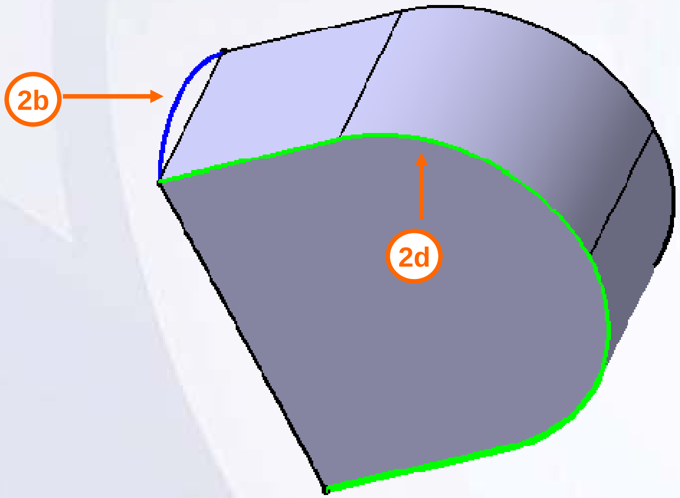


Do it Yourself (2/7)

2. Create a rib.

- Use Sketch.13 as the profile for a rib feature.
  a. Access the **Rib Definition** dialog box.
  b. Select Sketch.13 as the profile.
  c. Right-click the **Center Curve** field and click Extract from the contextual menu.
  d. Select the edge shown. Can the feature be created? Why not?





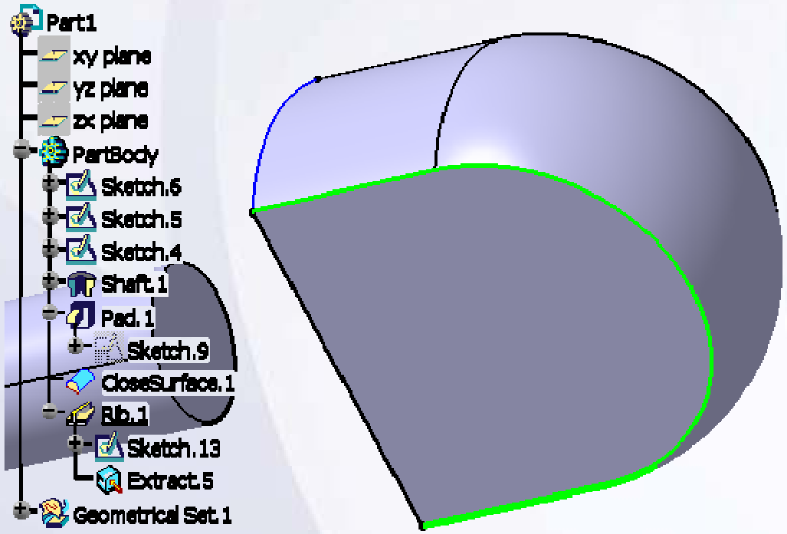
Do it Yourself (3/7)

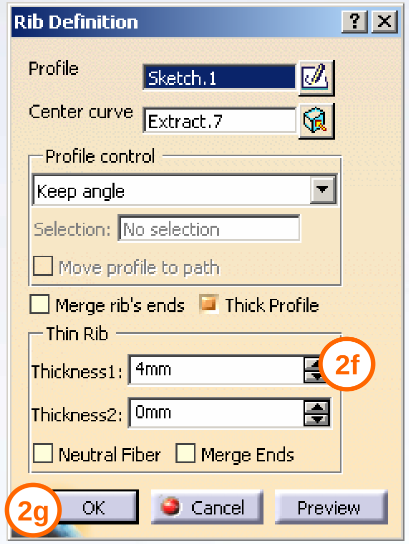
2. Create a rib (continued).

e. Select the **Thick Profile** option.

f. Type [4 mm] in the **Thickness1** field.

g. Complete the feature.





Do it Yourself (4/7)

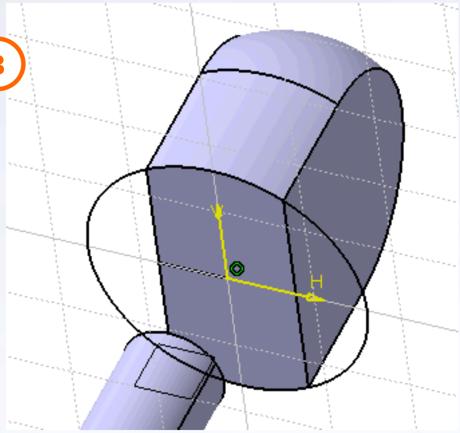
3. Create a profile for the multi-sections solid.

- Create the profile as shown using the lower face of the pad as the sketch support.

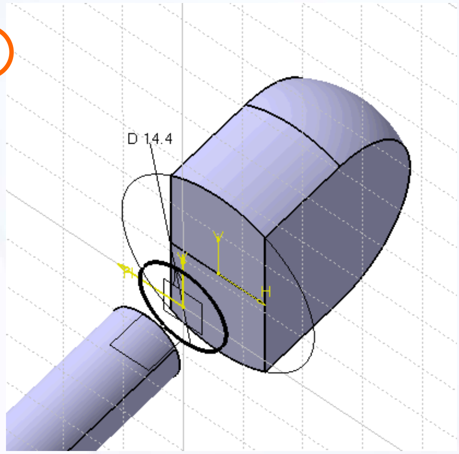
4. Create a second profile for the Multi-sections solid.

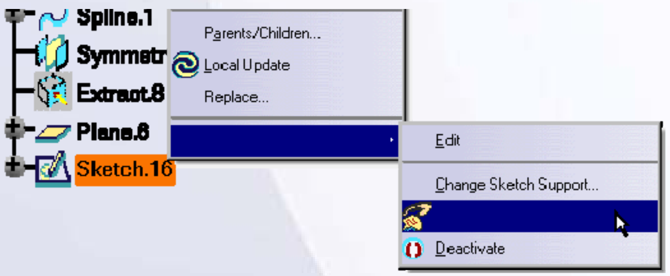
- Create a reference plane offset [7mm] from the lower surface of the pad. Create the sketch shown using the reference as the sketch support. The diameter of the sketched circle is [14.4mm].
- The sketch is created on a user-defined plane. After the plane is created, if you do not re-activate the PartBody, the sketch will be created in Geometrical Set.1. Move the sketch back to the PartBody by clicking **Change Geometrical Set** from its contextual menu, and select the PartBody from the specification tree.

3



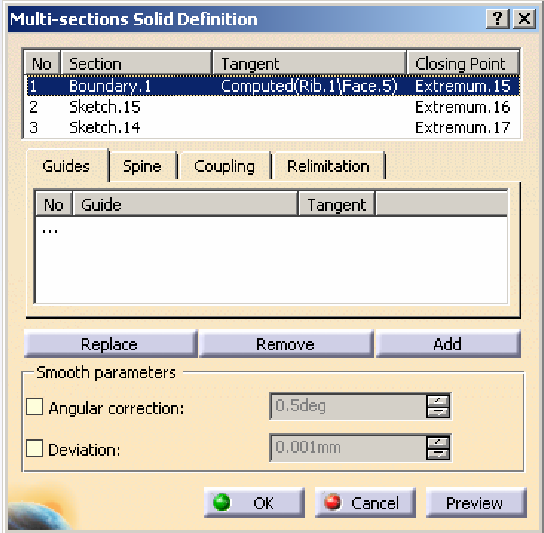
4

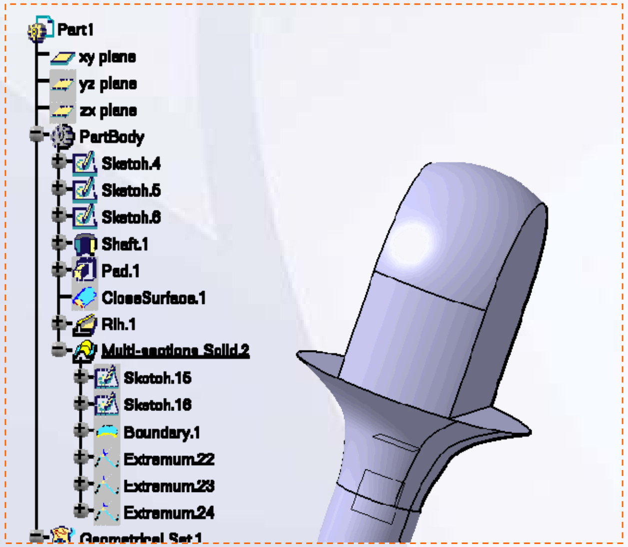


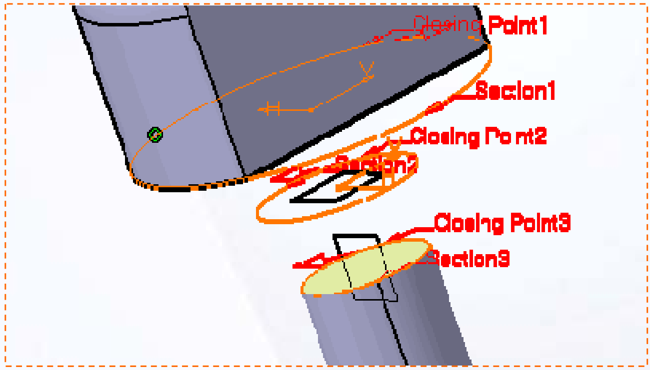


Do it Yourself (5/7)

5. **Create a multi-sections solid.**
   - Use the profiles and the lower surface of the shaft feature as the profiles for the feature. Notice that the feature is automatically tangent to the shaft.



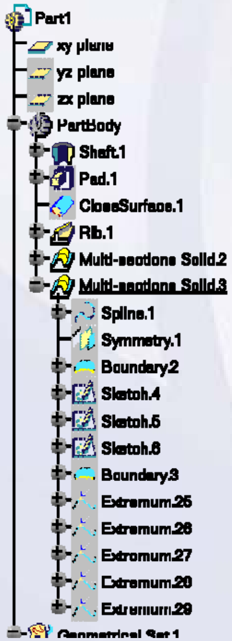


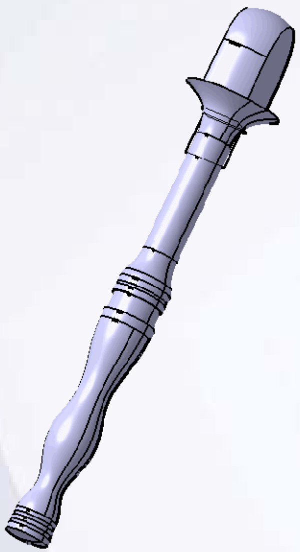


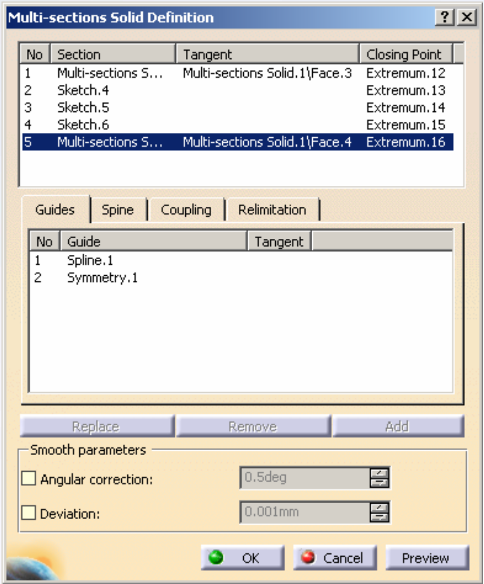
Do it Yourself (6/7)

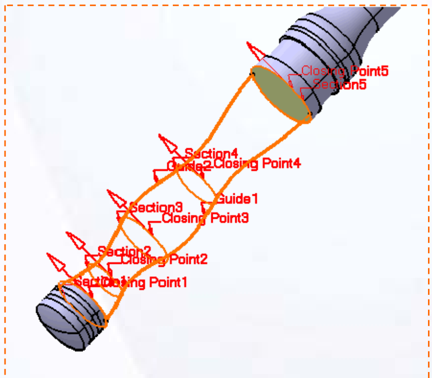
6. Create a second multi-sections solid.

- Create a second multi-sections solid to complete the handle. Use appropriate surface of the shaft, sketch.4, sketch.5, and sketch.6 as the profiles. Use Spine.1 and Symmetry.1 as guide curves for the feature.





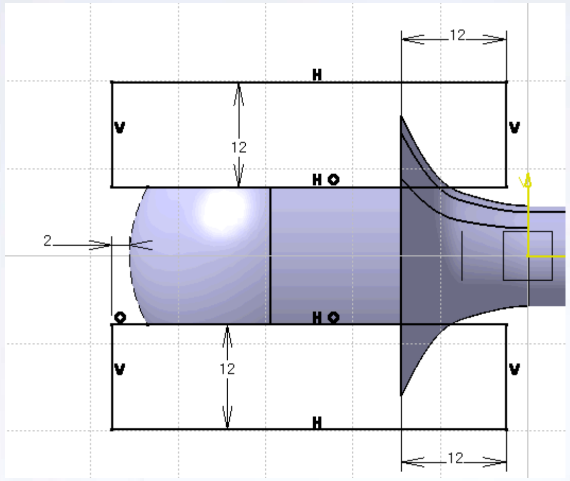


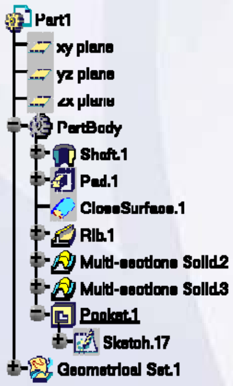


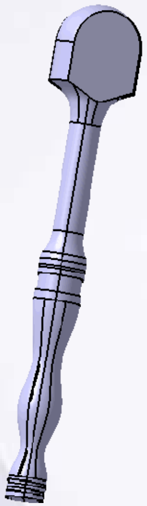
Do it Yourself (7/7)

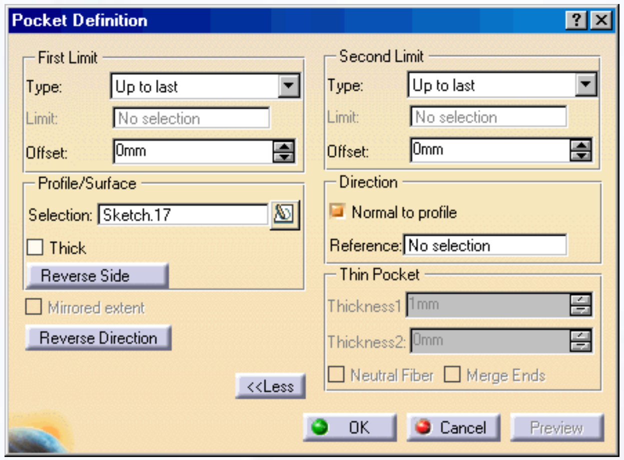
7. **Create a pocket feature.**
   - Create a pocket feature to trim away the excess material from the top of the wrench. Use the XY plane as the sketch support for the pocket feature.

8. **Clarify the display, save, and close the model.**
   - Hide all wireframe and surface elements. Save and close the model.



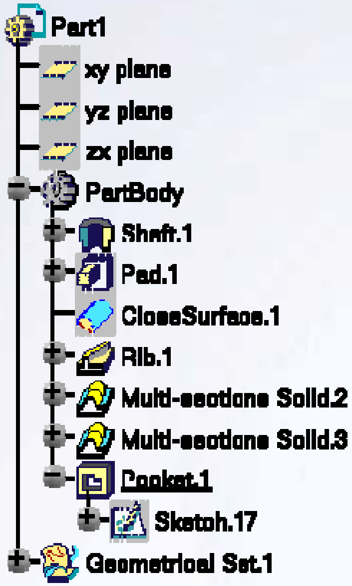


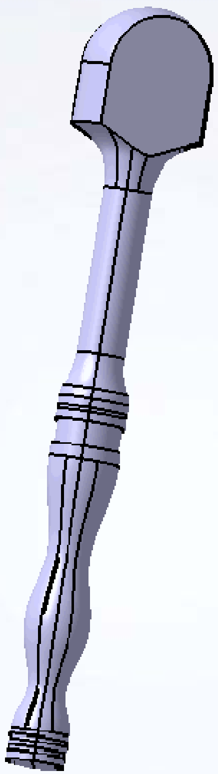




Exercise: Rib and Multi-Section Solid Recap

- ✓ Create a rib
- ✓ Create a multi-sections solid





Exercise: Advanced Draft

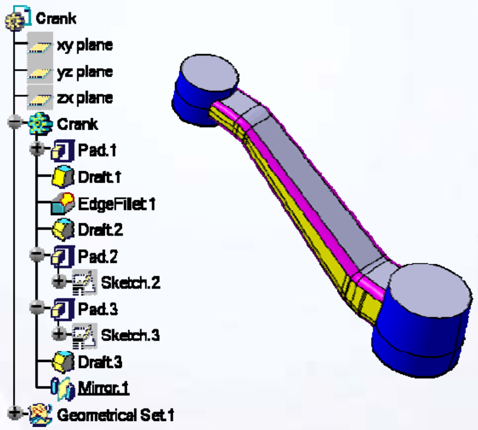
Recap Exercise

20 min

In this exercise, you will open an existing part that contains sketched wireframe elements and a surface feature. To complete this model you will have to create several advanced draft features. You will also use pads, variable fillets, and the mirror operation to complete this model. High-level instruction is provided for this exercise.

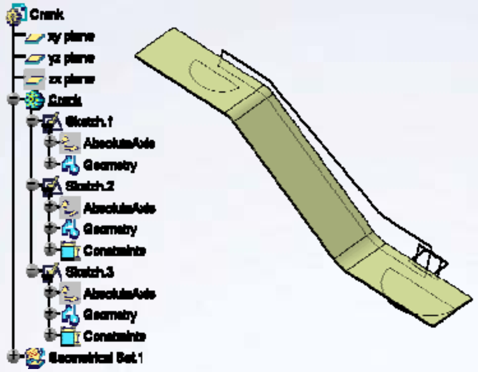
By the end of this exercise you will be able to:

- Apply advanced draft features



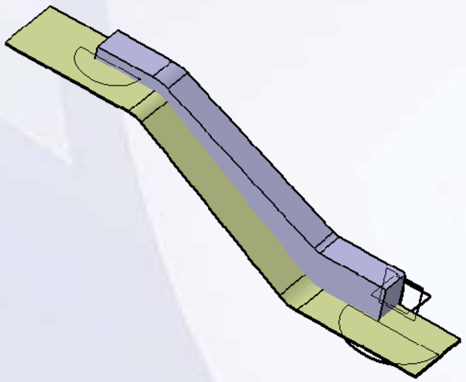
Do it Yourself (1/6)

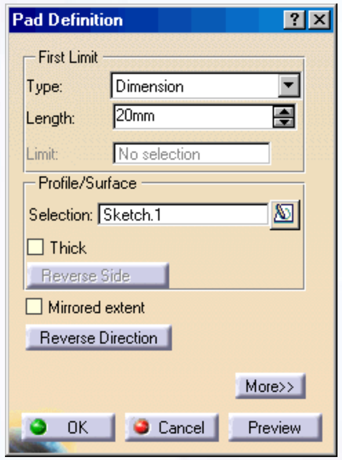
1. Load Ex8F.CATPart.



2. Create a pad Feature.

- Use Sketch.1 to create a pad feature with a depth of [20mm].

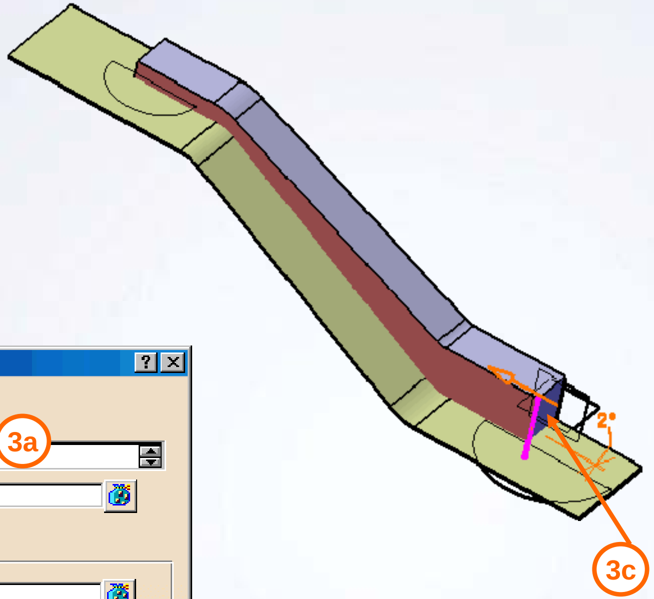


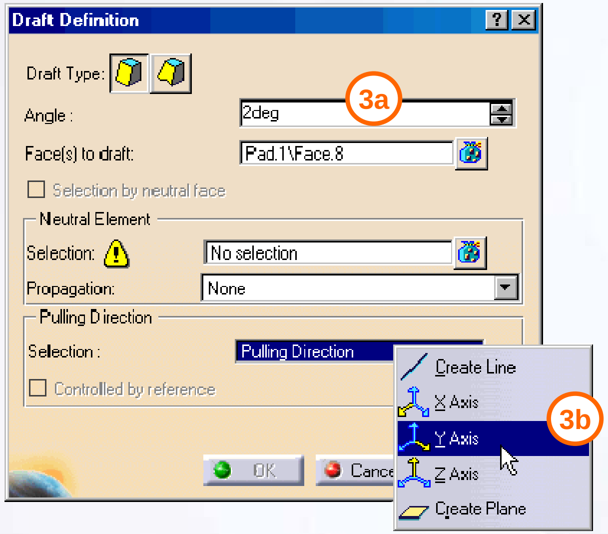


Do it Yourself (2/6)

3. Create a draft.

- Create draft on the outside vertical wall.
  a. Use a draft angle of 2 degrees.
  b. Use the positive Y direction as the pull-direction.
  c. Use the right vertical face as the neutral plane.

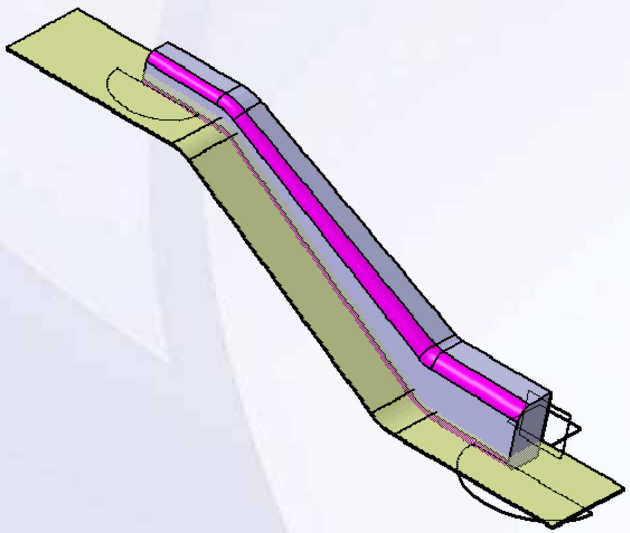


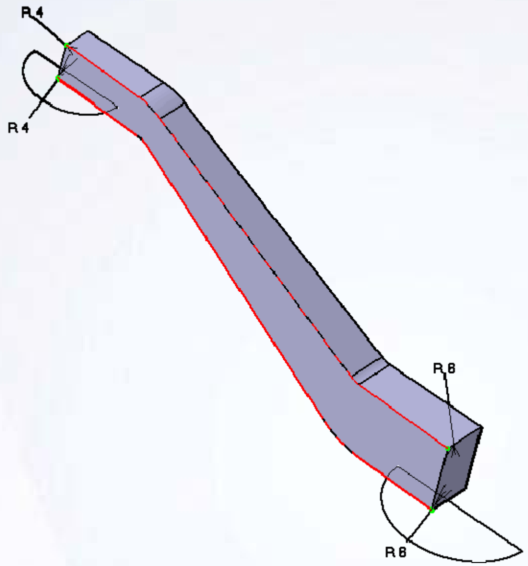


Do it Yourself (3/6)

4. Create a variable radius fillet.

- Apply a variable radius fillet to the top and bottom outside edges. Create the fillet from [4mm] to [6mm] along each side.

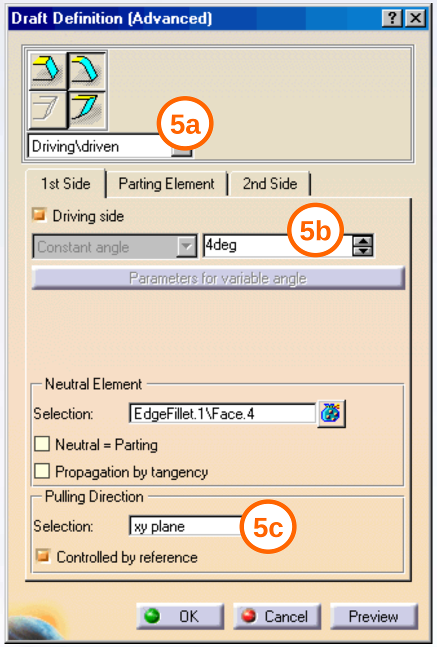


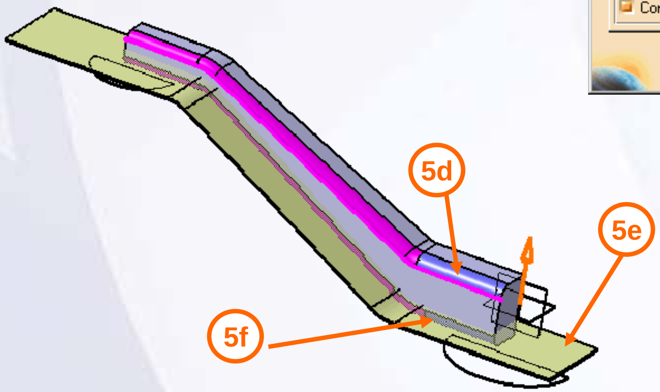


Do it Yourself (4/6)

5. Create an advanced draft.

- Create a two-sided reflect draft.
  - a. Use the **Driving/Driven** dependency option.
  - b. Set the draft angle to 4 degrees.
  - c. Use the XY plane as the pulling direction for the first side.
  - d. Use the top fillet as the neutral element for the side one.
  - e. Select the Extruded surface as the parting element.
  - f. Use the bottom fillet as the neutral element for the side two.



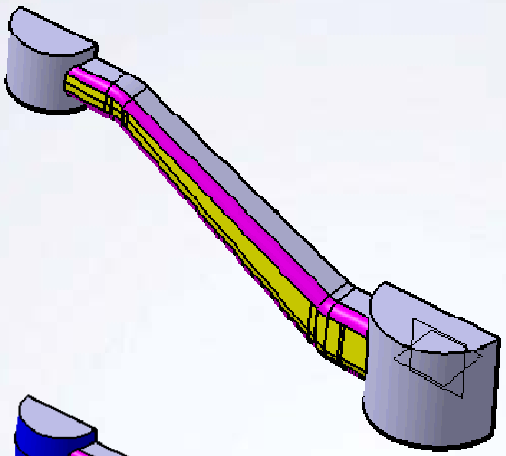


Do it Yourself (5/6)

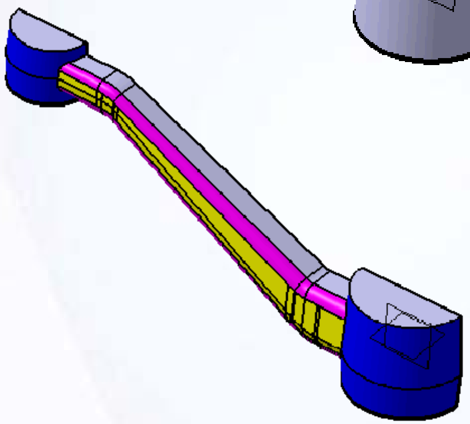
6. **Create two pad features.**
   - Use Sketch.2 to create a pad feature with a depth of [30mm].
   - Use Sketch.3 to create a pad feature with a depth of [50mm].

7. **Apply an advanced draft.**
   - Apply an advanced draft feature to the two pads.
     a. Create the draft with a 4 degree draft angle on the first side.
     b. Use the XY plane as the pulling direction for side one.
     c. Use a 6 degree draft angle on the second side.
     d. Use Extrude.1 as the parting element.
     e. Set the Neutral element on both sides equal to the parting element.

6



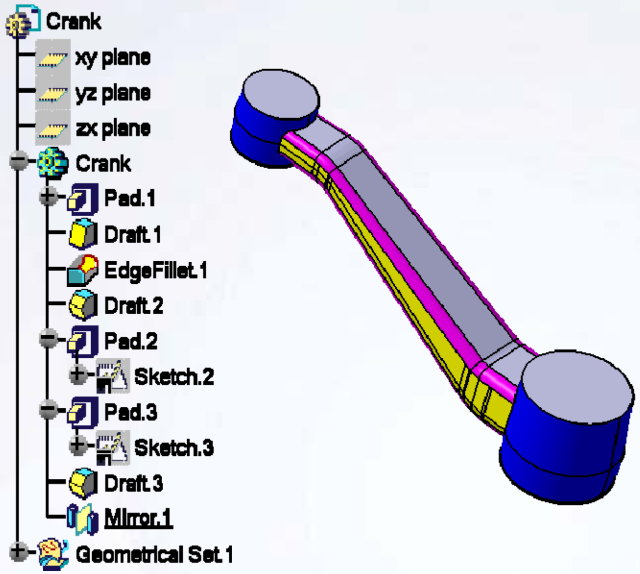
7



Do it Yourself (6/6)

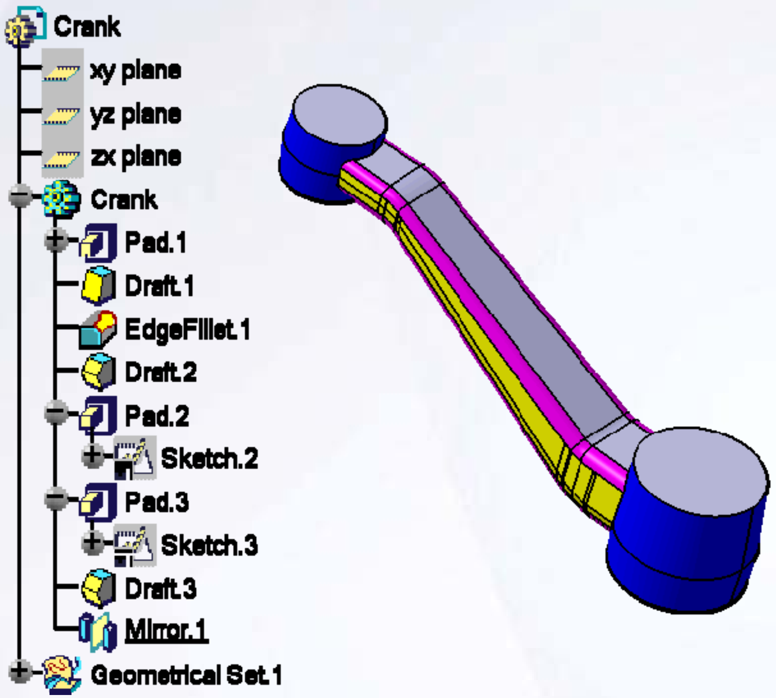
8. **Mirror the model.**
   - Complete the model by mirroring the part body about the YZ plane.

9. **Clear the model, save and close it.**
   - Hide all wireframe and surface elements and save the model.



Exercise Recap: Advanced Draft

✓ Create an advanced draft



Exercise: Multi-Body Work

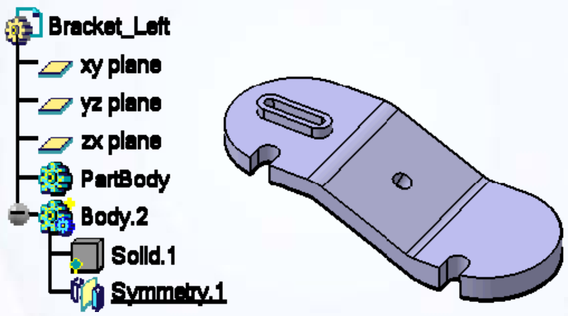
Recap Exercise

15 min

In this exercise, you will open an existing part that contains a single feature. You will use the tools learned in this lesson to perform a Boolean operation, and create a multi-model link. High-level instructions for this exercise are provided.

By the end of this exercise you will be able to:

- **Create Multi-Model links.**
- **Perform Boolean Operations.**
- **Modify Multi-Linked Models.**



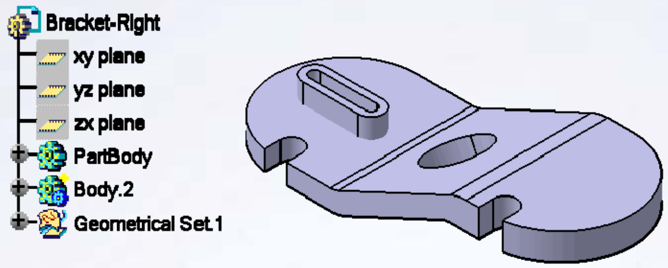
Do it Yourself (1/2)

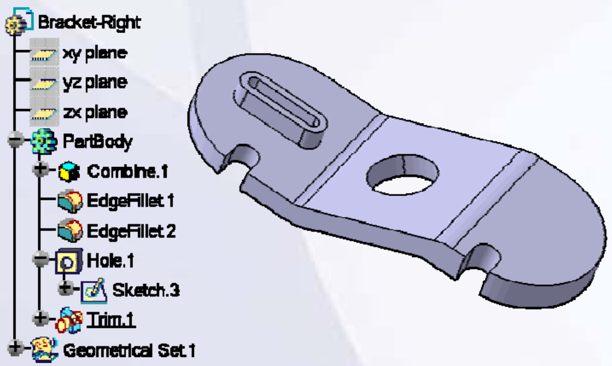
1. Open the part file.

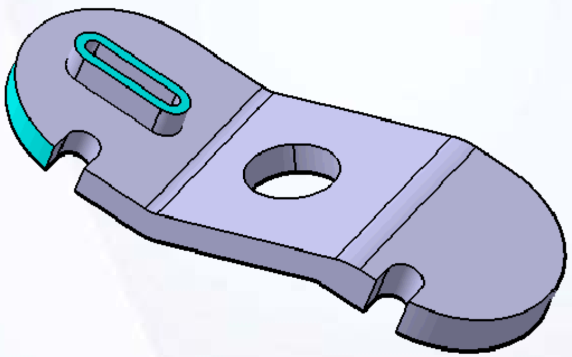
- Open the existing part file, Bracket_right.CATPart. There are two bodies in this file.

2. Perform a Union Trim operation on the PartBody using Body.2.

- Use the **union trim** operation to trim Body.2 from the PartBody. Keep the top surface of Body.2 and the cylindrical surface from the PartBody.





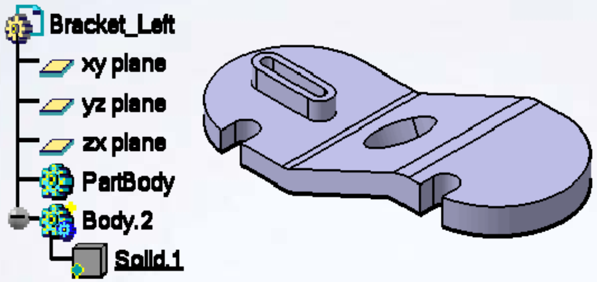


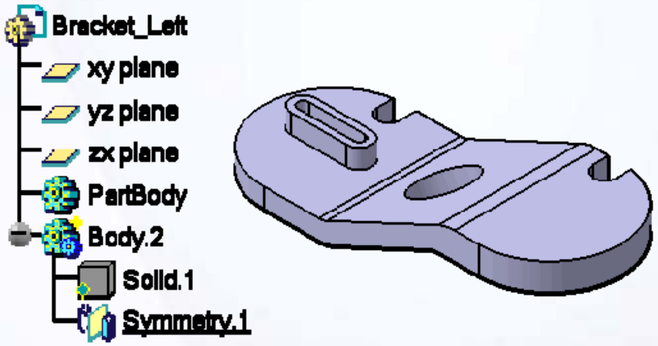
Do it Yourself (2/2)

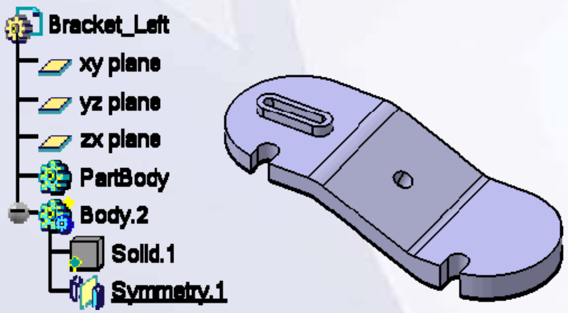
3. **Create a new part file.**
   - Create a new PartBody called Bracket_Left. Create a multi-model link to the PartBody in Bracket_right.

4. **Transform features.**
   - Use the **symmetry** tool to transform the notches in the Bracket_Left model. Perform the symmetry operation about the YZ plane.

5. **Modify the hole in Bracket_Right.**
   - Modify the hole dimension in Bracket_Right to [5 mm] and update both the models.

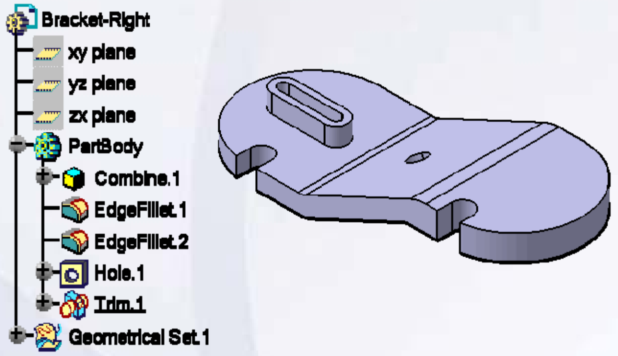


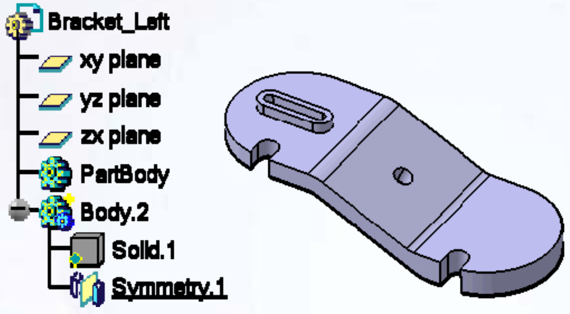




Exercise: Multi-Body Work Recap

- ✓ Create multi-model links
- ✓ Perform a Union Trim operation
- ✓ Modify multi-model link models





Case Study: Design Complex Parts

Recap Exercise

15 min

In this exercise you will create the case study model. Recall the design intent of this model:

- ✓ Base feature must include overall dimensions supplied.
- ✓ Create each support as a single feature.
- ✓ A cut is to be created to simulate the logo. The cut profile varies.
- ✓ Links must be created to the Disk holder and the flex opening models to ensure conformance to standards.
- ✓ Linked features must be kept in separate bodies.
- ✓ An indented logo should not be displayed when it goes for manufacturing.

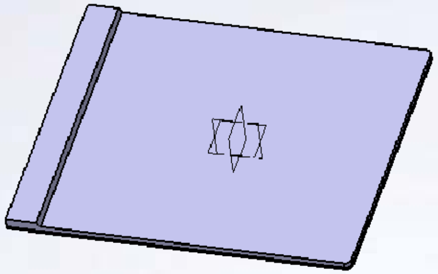
Using the techniques you have learnt in this and previous lessons, create the model without detailed instructions.

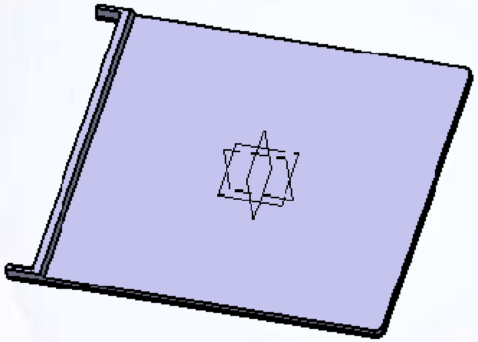
Do It Yourself: CD Jewel Case (1/7)

You must complete the following tasks:

1. **Create a solid combine.**
   - Load JewelCase.CATPart
   - Use the two sketches supplied to create a solid combine feature.

2. **Create a pocket.**
   - Create a pocket using the dimensions shown on the front view of the drawing.



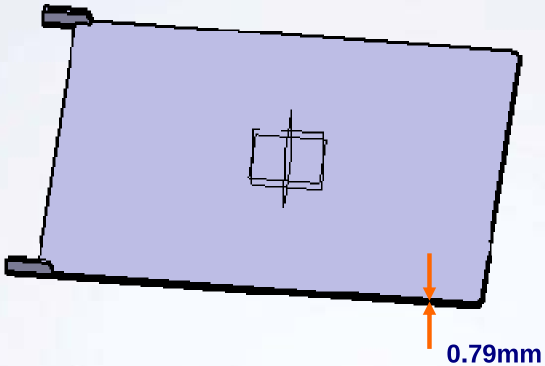


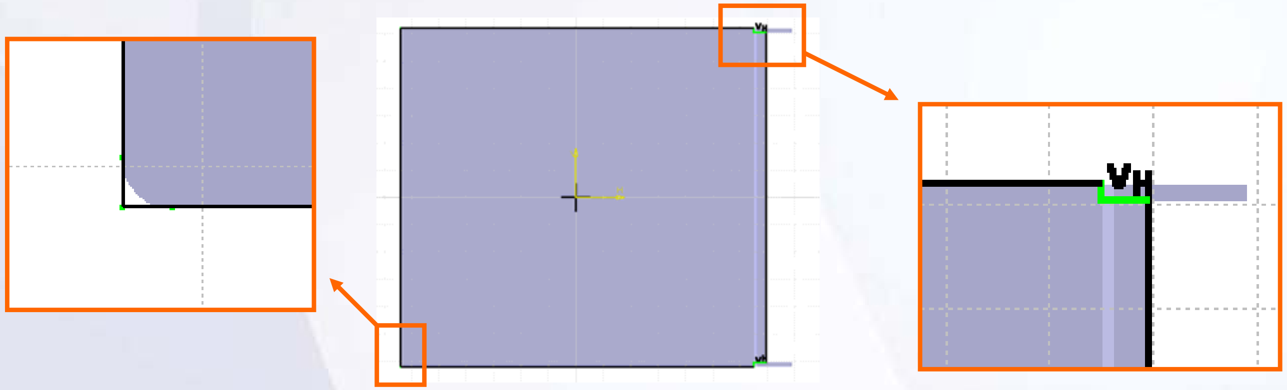
Do It Yourself: CD Jewel Case (2/7)

You must complete the following tasks (continued):

3. Create a pocket.

- Create a second pocket using the dimensions shown. The cut is symmetrical about the ZX plane. This pocket needs to cut the material such that only a 0.79mm thickness is left.





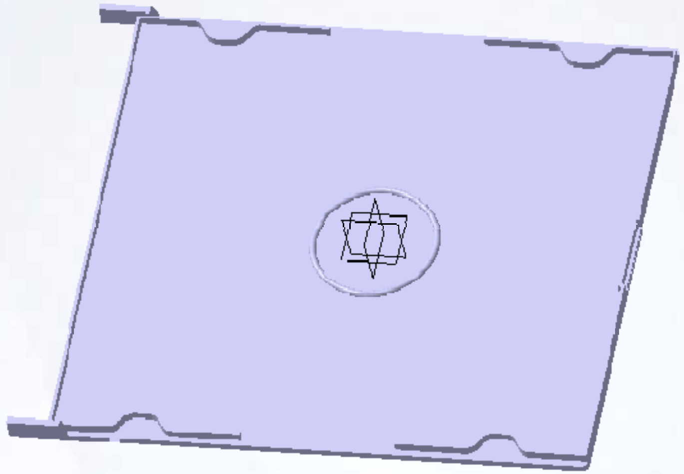
Do It Yourself: CD Jewel Case (3/7)

You must complete the following tasks (continued):

4. **Create a rib feature.**
   - Create a rib feature using the dimensions shown on Detail view C and G of the drawing. The rib is symmetric about the ZX plane.

5. **Create a second rib feature.**
   - Create a rib feature using the dimensions shown on Detail view B, E and H of the drawing. The rib is symmetric about the ZX plane.

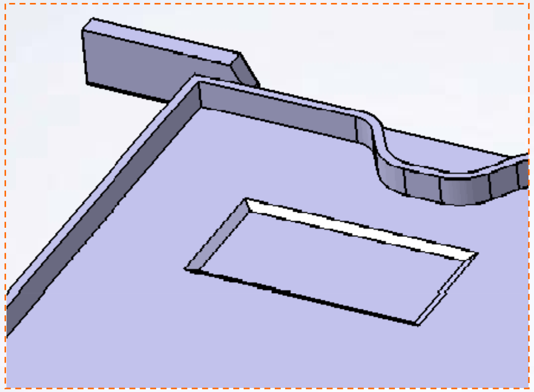
6. **Create a third rib feature.**
   - Create a rib feature using the dimensions shown on detail view F and the front view of the drawing.

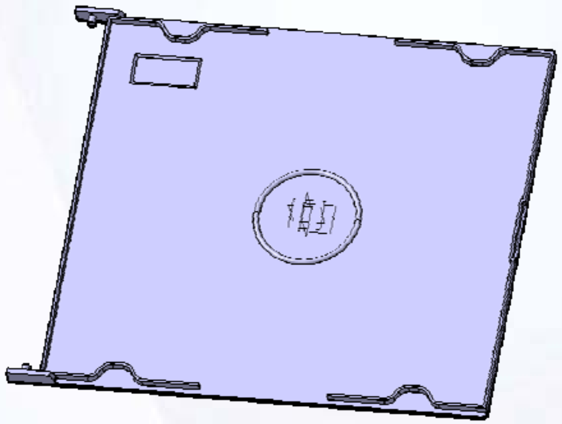


Do It Yourself: CD Jewel Case (4/7)

You must complete the following tasks (continued):

7. **Create a removed multi-sections solid.**
   - Create the logo using a removed multi-sections solid. The lower profile is created on a reference plane that is offset 0.45mm below the top surface of the case. Use Detail view C and Section view D-D for the dimensions.
8. **Create two pad features.**
   - Create two pad features using the dimensions shown on Detail views C and G. Consider creating only one Pad feature and mirroring it to create the other.





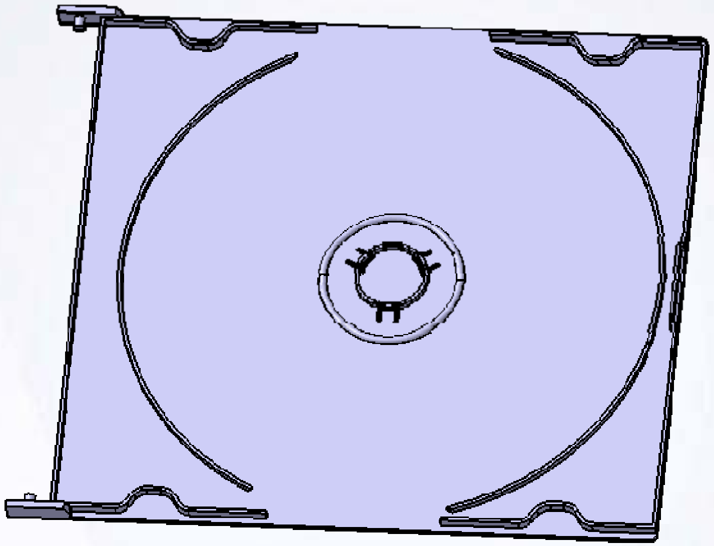
Do It Yourself: CD Jewel Case (5/7)

You must complete the following tasks (continued):

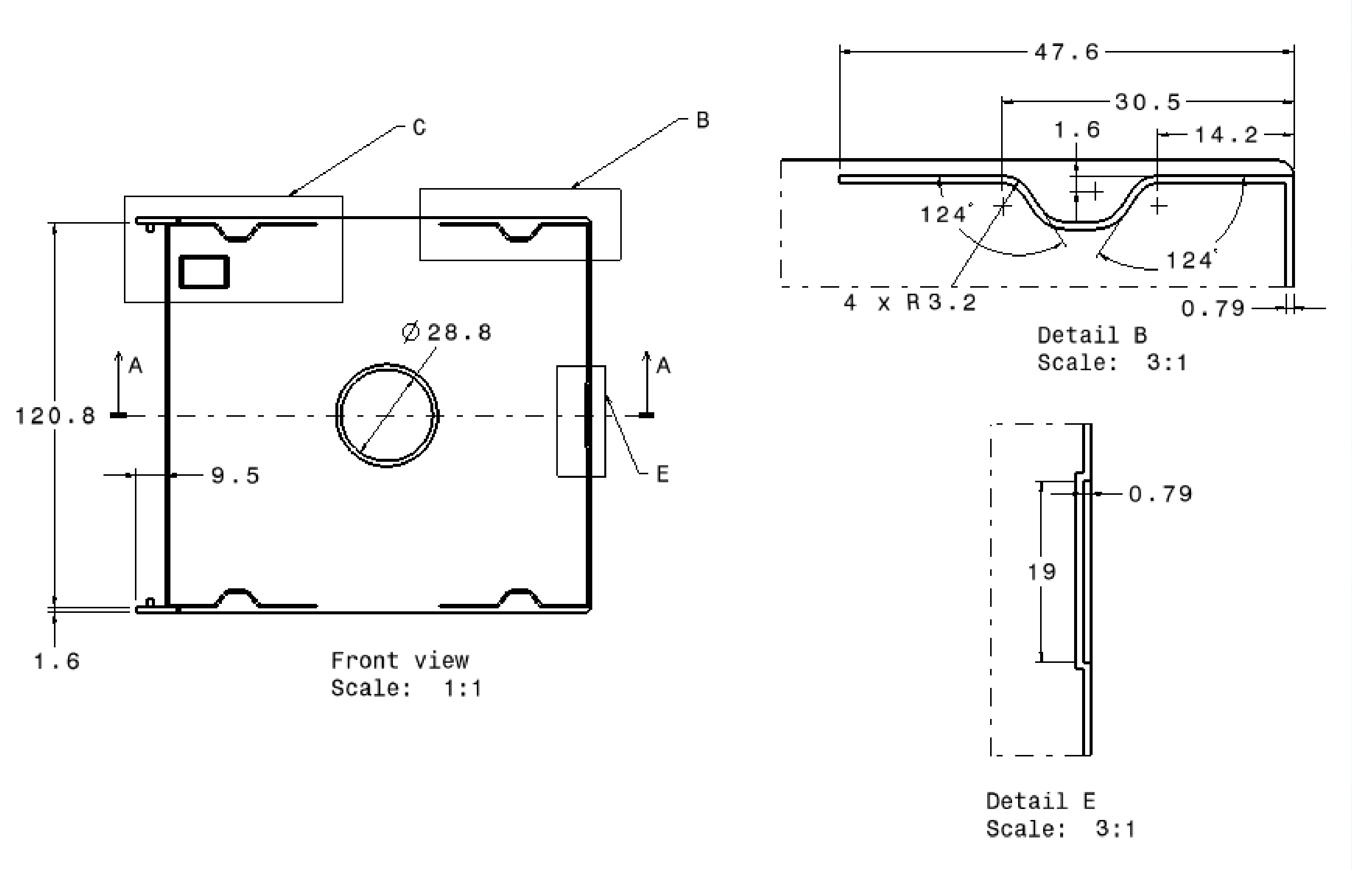
9. **Copy the DiskHolder and FlexOpening bodies.**
   - Copy the DiskHolder and the FlexOpening bodies from JewelCaseSubPart.CATPart using the **Paste Special** option **As Result With Link**.

10. **Assembly the FlexOpening body to the main body.**

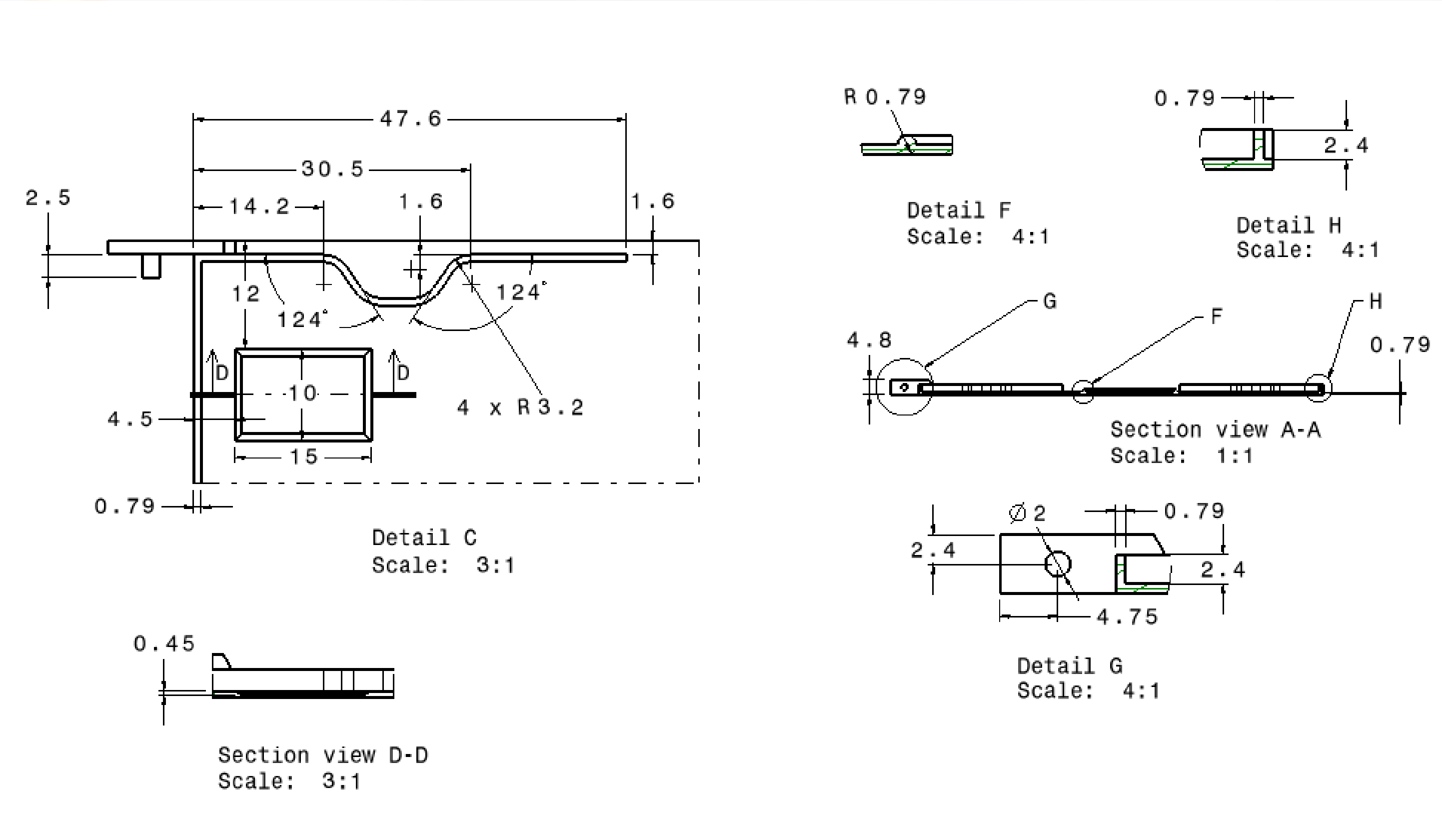
11. **Use the remove face tool to remove the logo from display.**



Do It Yourself: CD Jewel Case (6/7)



Do It Yourself: CD Jewel Case (7/7)



Case Study: Jewel Case Recap

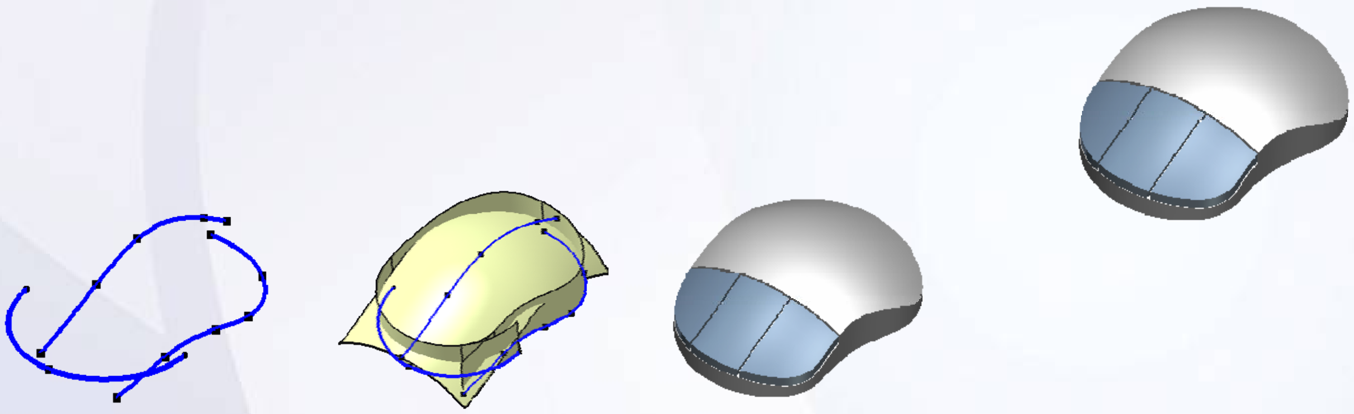
- ✓ Create a solid combine
- ✓ Create rib features
- ✓ Create a removed multi-sections solid
- ✓ Create multi-model links
- ✓ Perform Boolean operations
- ✓ Remove a face

Surface Design 3

Learning Objectives

Upon completion of this lesson you will be able to:

- ✓ Access the Surface Design Workbench
- ✓ Create the Reference Geometry
- ✓ Create the Basic Surface Geometry
- ✓ Create the Complex Surface Geometry
- ✓ Perform Operations on Surfaces
- ✓ Solidify the Model



4 Hours

Case Study

The case study for this lesson is the Computer Mouse.

Design Intent

✓ Model contours are likely to change.

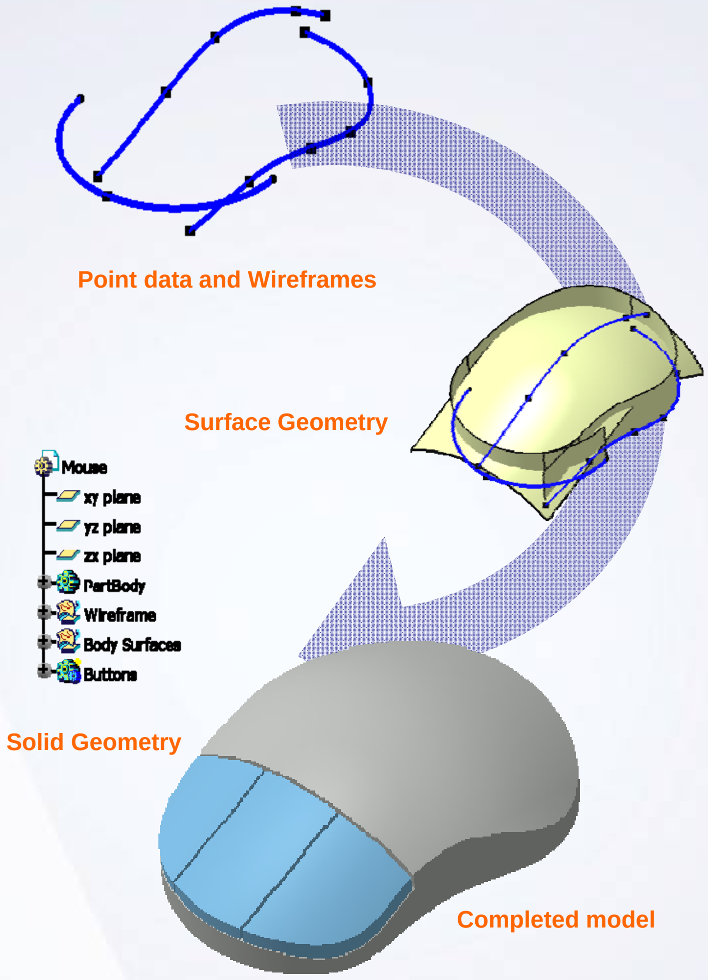
- This model is created from point data so that the geometry can quickly be changed simply by adjusting point locations.

✓ Wire-frame, surface and solid geometry must be kept separate.

- By creating separate Geometrical Sets the model can be kept organized to help other users quickly identify the different elements making up the model.

✓ Buttons must be built as a separate body but update when the changes are made to the main body.

- The button geometry can be created in a separate body will use surfaces from the main body as limiting elements.



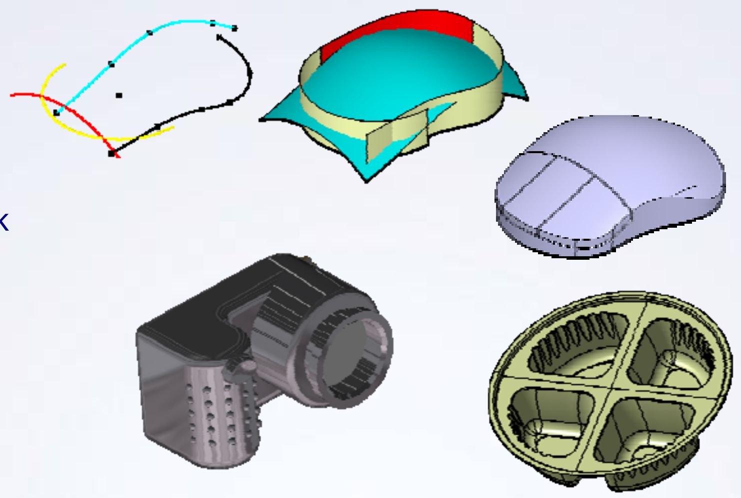
Stages in the Process

1. Access the Generative Surface Design workbench.
2. Create the wire-frame geometry.
3. Create the surface geometry.
4. Perform operations.
5. Solidify the model.

Introduction to Generative Shape Design

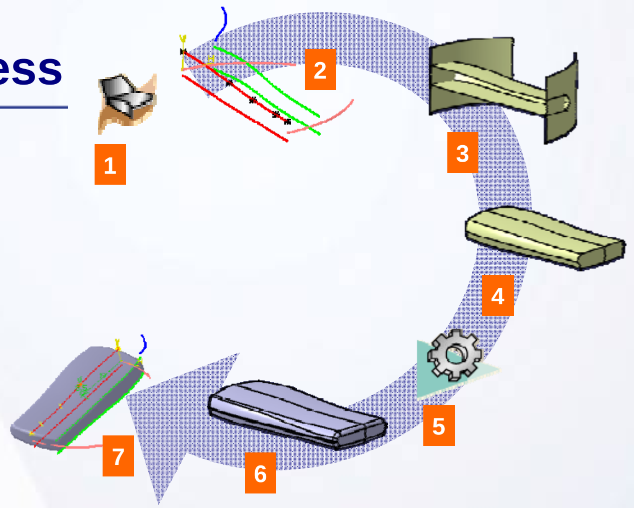
Wireframe and surface geometry is created with Generative Shape Design workbench to define complex shapes.

- ✓ Can be used by novice as well as advanced users.
- ✓ Provides a set of comprehensive tools for making quick changes in the preliminary design and keeping the accuracy needed for the detailed design.
- ✓ Lets you control the propagation of modifications when designing in context. You can reuse existing surfaces and other surface models.
- ✓ Datum curves or skins can be used to drive the design and can be quickly replaced if required.



Surface Design Workbench General Process

1. Access the Generative Surface Design workbench.
2. Create the wireframe geometry.
3. Create the surface geometry.
4. Trim and join the body surfaces.
5. Access the Part Design workbench.
6. Create a part body.
7. Modify geometry as needed.

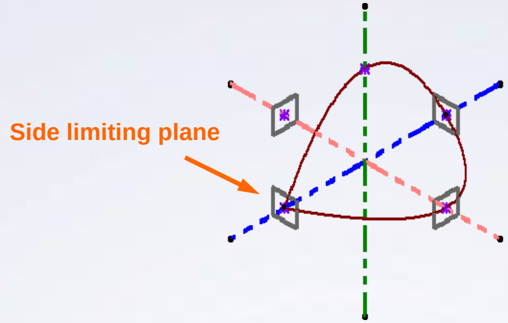


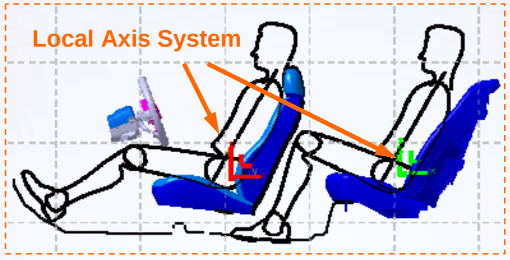
Create the Reference Geometry

Reference geometries are the basic elements which provide a stable geometric support. They can be used to limit and control the overall size of the part. Examples are: Points, Lines, Planes, and Axis systems.

CATIA has a fixed coordinate system called the Absolute Axis System. A point in the model will have coordinates specific to this axis system.

You can also define user-defined axis systems known as Local Axis Systems. These can be anywhere in 3D space. There can be multiple axis systems in a single part.





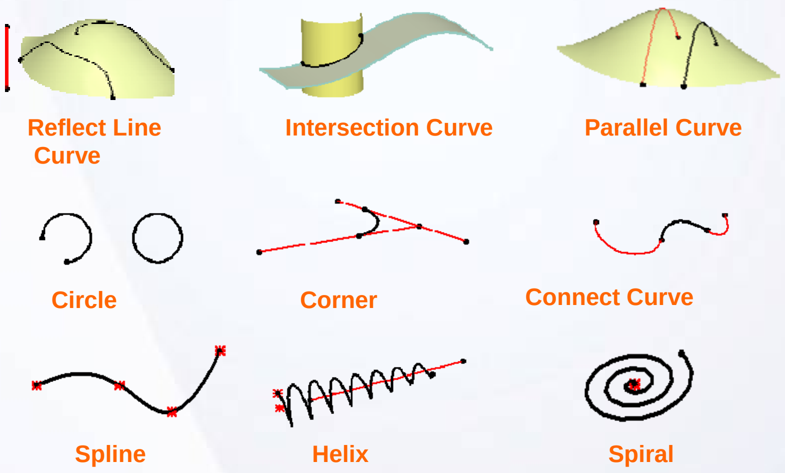
Create Curves

Curves are geometrical elements used as limiting elements (lines, planes), guides or references to create other elements. Some examples are:

A. Project-Combine curves (Projection curve, Reflect Line Curve, Intersection Curve, Parallel Curve)

B. Circular-Conic curves (Circle, Corner, Connect Curve, Conic)

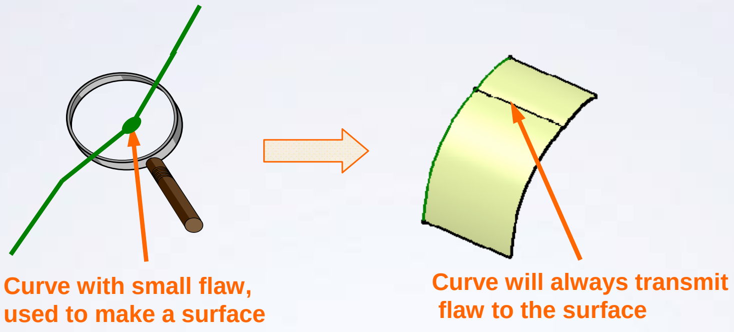
C. Curves (Spline, Helix, Spiral)



Create Curves (continued)

Great care should be taken while constructing the wire-frame geometry since surfaces inherit any flaws within the parent curves or wire-frame geometry,

In a product development cycle a surface would be further used in downstream operations such as prototyping, machining, tooling, etc. and the final product would be adversely affected.

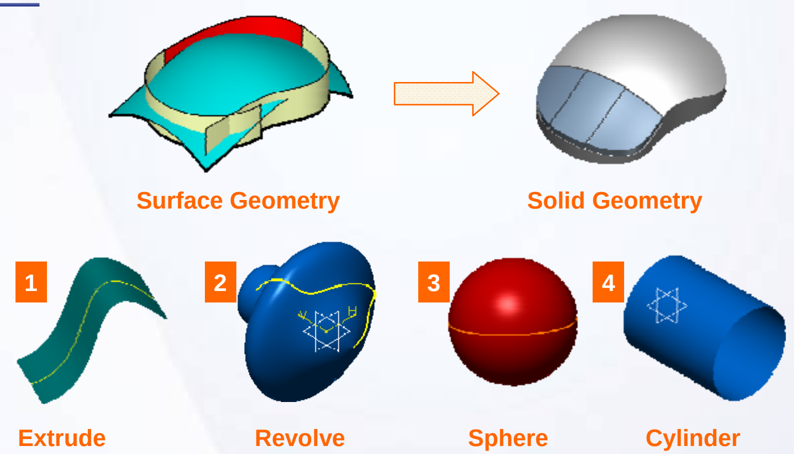


Create the Basic Surface Geometry

Complex 3D shapes often need to be defined using surface geometry which is created based on explicit wire-frame construction geometry.

Some examples of basic surfaces are:

1. Extruded Surface
2. Revolve
3. Sphere
4. Cylinder



Create the Complex Surface Geometry

Some examples of complex surfaces are:

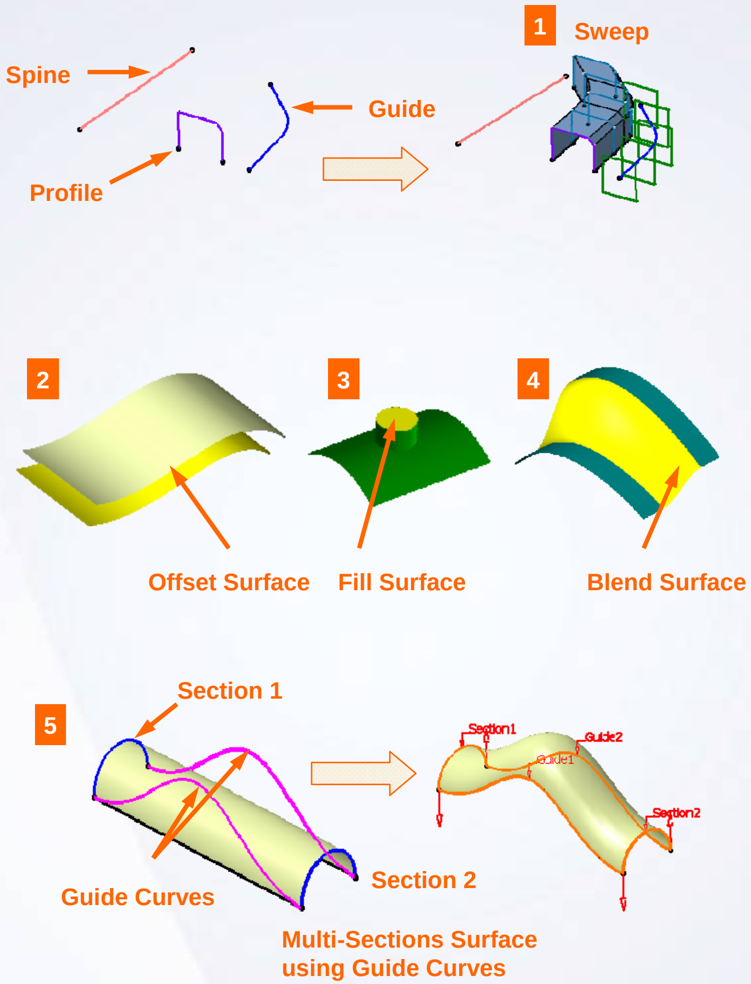
1. Sweep: A surface generated by sweeping a profile along a guide curve with respect to a spine. The profile can be a user-defined or pre-defined profile. The shape and quality of the sweep depends upon the spine.

2. Offset Surface: A surface which is offset from the reference surface.

3. Fill Surface: Created from a closed boundary. The boundary can consist of wire-frame elements or edges of existing surfaces.

4. Blend Surface: Created between two wire-frame elements.

5. Multi-Sections Surface: Computed by passing through two or more sections along a spine. The spine defines the shape of the surface between two sections. Various options for defining multi-sections surface exists - Guides, Spine, Re-limitation, and Canonical elements.



Perform Operations on Surfaces

Operations such as trim, join, extrapolate, and transform are performed to produce the required finished geometry. Transformations, such as scaling and affinity, help to resize the part. Transformation operations, such as translate and rotate, are required on the wireframe elements to change the positioning of the part in the co-ordinate axis system. Boundary operation extracts internal or external edge of the surface. Extract operation extracts sub-elements of a surface (edge or surface).

Solidify the Model

Completing the geometry in Part Design, with hybrid modeling capability of V5, enables the complex surface geometry to shape the solid part. Use the Part Design workbench to integrate surface geometry into a solid part.

You can create the following surface based features in Part Design using the surface geometry.

1. Split
2. Thick surface
3. Close surface
4. Sew surface

Main Tools (1/3)

Wireframe Geometry

1 **Points:** Creates a point or multiple points.

2 **Line-Axis:** Creates lines, axis or polyline.

3 **Plane:** Creates planes using different options.

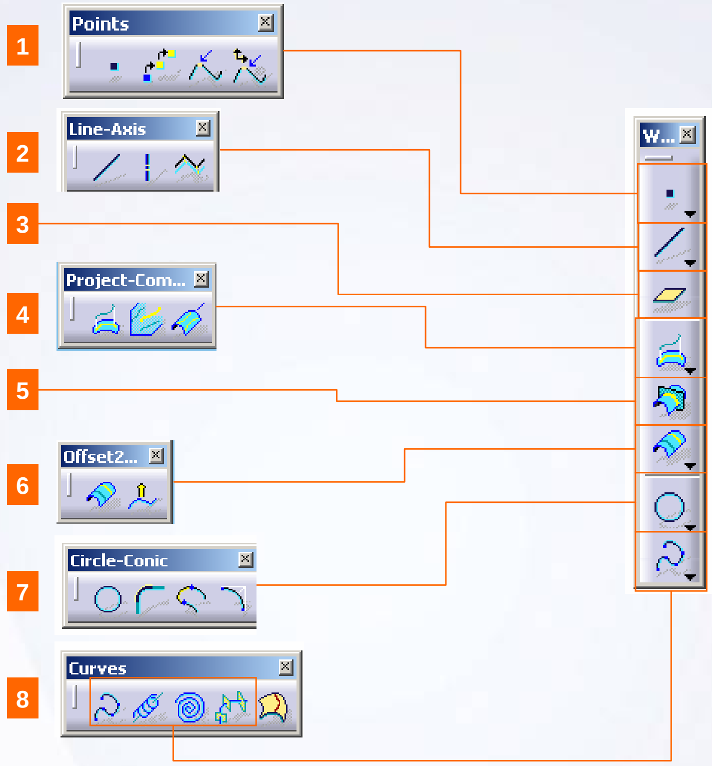
4 **Project-Combine:** Projection curve, Combine curve and Reflect Line Curve

5 **Intersection Curve:** Creates a curve at the intersection of two elements.

6 **Offset 2D3D:** Creates a parallel curve and offset curve.

7 **Circle-Conic:** Creates circle, corner, connect curve, and conic curve.

8 **Curves:** Creates a spline, helix, spiral, and spine.



Main Tools (2/3)

Surfaces

9 **Extrude-Revolution:** Creates extrude, revolution, sphere, and cylinder surface.

10 **Offset:** Creates an Offset Surface.

11 **Sweep:** Creates a swept surface.

12 **Fill:** Creates a fill surface.

13 **Multi-Sections Surface:** Creates a surface passing through multiple sections along the spine.

14 **Blend Surface:** Creates a blend surface between wireframe elements.

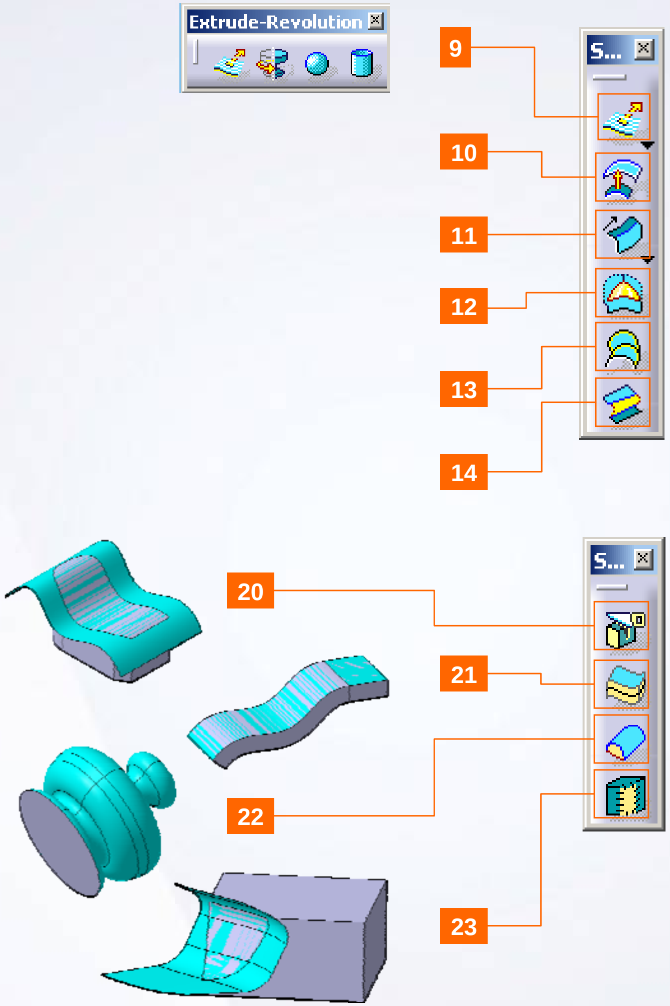
Surface Features

20 **Split:** Splits a solid using a surface.

21 **Thick Surface:** Creates a solid from existing surface with thickness specified.

22 **Close Surface:** Creates a solid by closing the sides of the surface.

23 **Sew Surface:** Creates a Boolean operation and combines surface and solid.



Main Tools (3/3)

Operations

15 **Join:** Joins curves or surfaces.

16 **Healing:** Heals surfaces by filling in small gaps between the surfaces.

17 **Trim-Split:** Creates a Split surface and Trim surface.

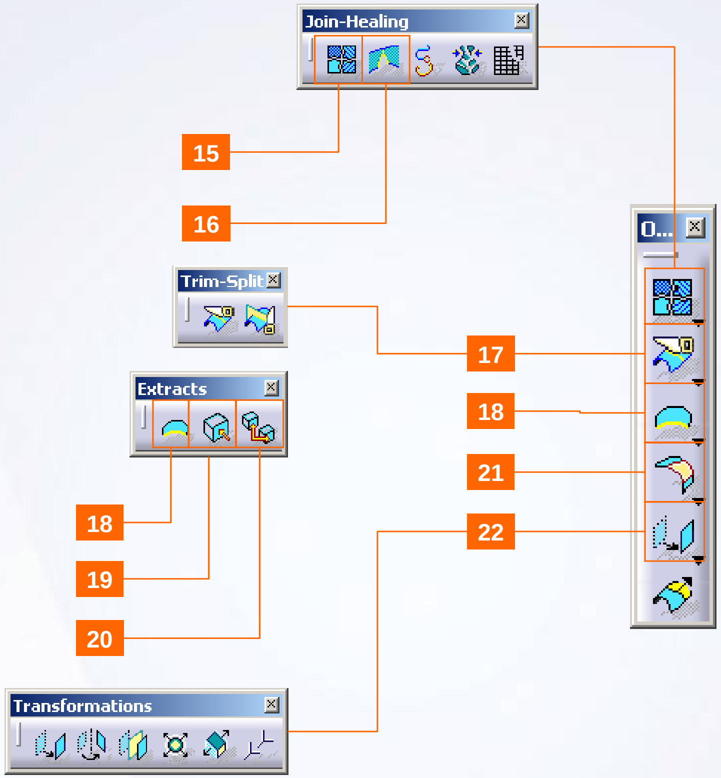
18 **Boundary:** Creates a boundary from edge of the surface.

19 **Extract:** Extracts a face or a surface edge.

20 **Multiple extract:** Extracts a group of elements.

21 **Fillets:** Creates various types of surface fillets.

22 **Transformations:** Creates transformation features – translation, rotation, symmetry, scaling, affinity, and Axis to axis.



Exercise: Join, Trim and Close Surface

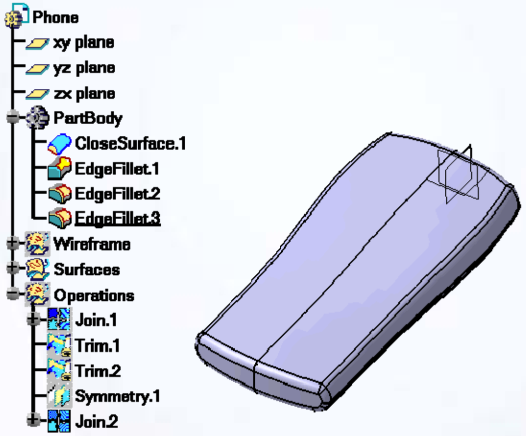
Recap Exercise

20 min

In this exercise, you will open an existing file that contains the wireframe and surface geometry necessary to complete the model. You will use the tools learnt in this lesson to perform operations and solidify the model. High-level instructions for this exercise are provided.

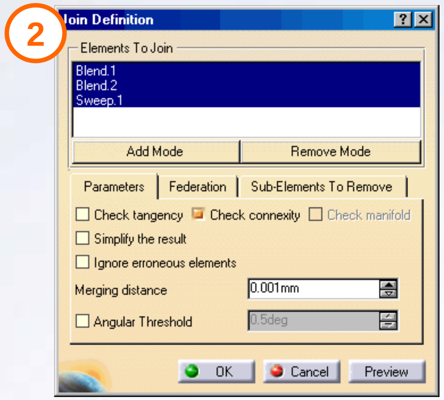
By the end of this exercise you will be able to:

- **Join Surfaces**
- **Trim Surfaces**
- **Mirror**
- **Close a Surface**

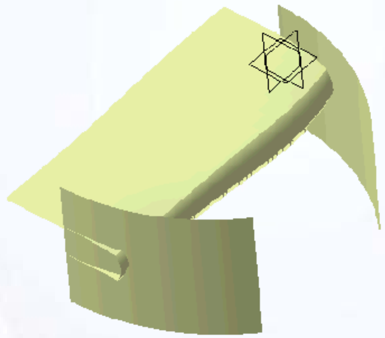


Do it Yourself (1/5)

1. **Open part file.**
   - Open the existing part file, Operations_Phone.CATPart. The wireframe and surface geometry has been created for you.
     a. Create a new geometrical set called Operations.
     b. Ensure that the Operation geometrical set is active.



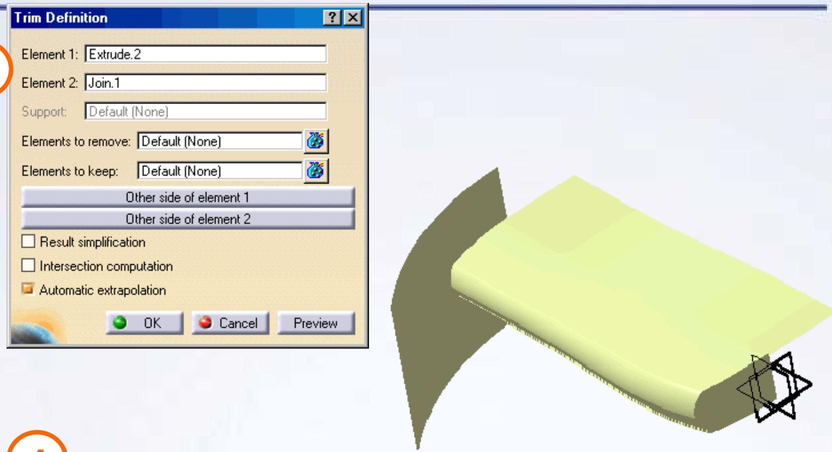
2. **Join the top and side surfaces.**
   - Join Blend.1, Blend.2 and Sweep.1 to create the top and side surface.
     a. Select the **Join** icon.
     b. Select Blend.1, Blend.2, and Sweep.1.
     c. Click **OK**.



Do it Yourself (2/5)

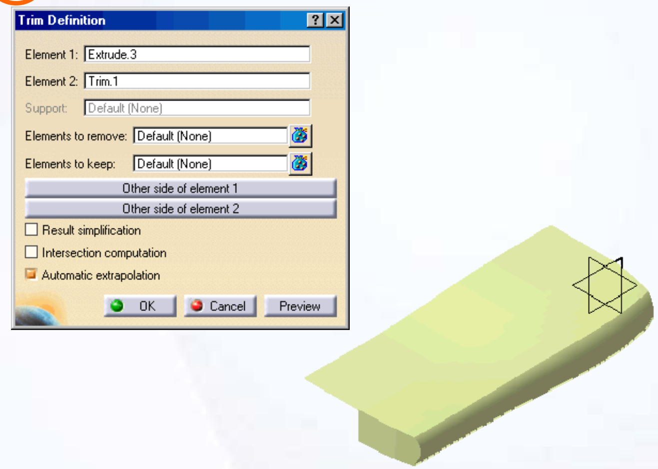
3. **Trim the surfaces.**
   - Trim the top extrude and the join.
     a. Select the **Trim** icon.
     b. Trim Extrude.2, and Join.1
     c. Use the **Other side of element** buttons to create the trim as shown.
     d. Click **OK** to complete the operation.

3



4. **Trim the surfaces.**
   - Trim the bottom extrude and the trim feature.
     a. Select the **Trim** icon.
     b. Trim Extrude.3, and Trim.1.
     c. Use the **Other side of element** buttons to create the trim.
     d. Click **OK** to complete the operation.

4

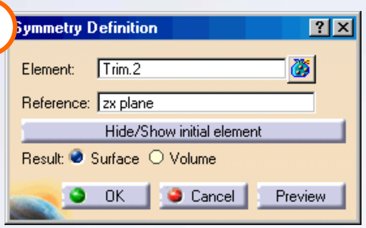


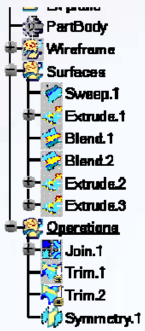
Do it Yourself (3/5)

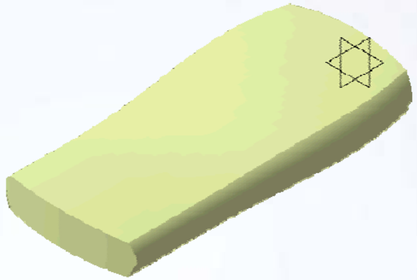
5. Mirror the model.

- Use the Symmetry tool to create the other side of the model.
  - a. Select the **Symmetry** icon.
  - b. Mirror Trim.2 about the ZX plane.
  - c. Click **OK** to complete the operation.
  - d. Hide Extrude.1 from the Surface geometrical set.

5



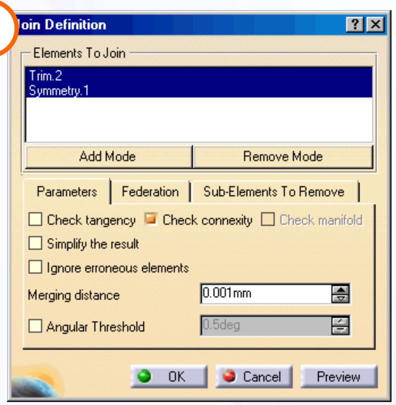




6. Join the surface.

- Complete the surface model by joining the two halves.
  - a. Select the **Join** icon.
  - b. Select Trim.2 and Symmetry.1.
  - c. Click **OK** to complete the operation.

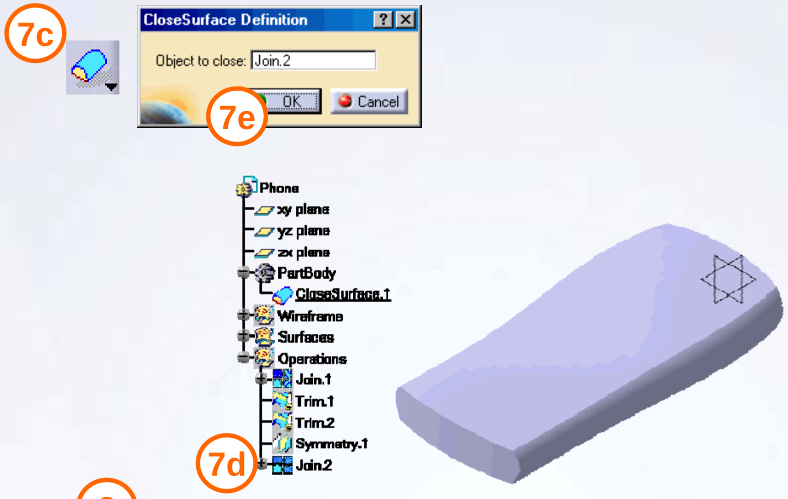
6

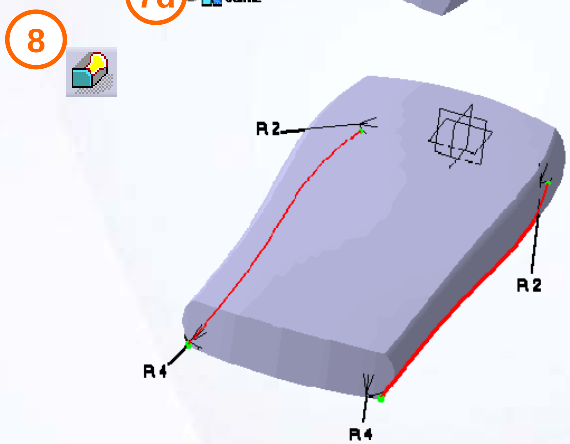


Do it Yourself (4/5)

7. **Solidify the model.**
   - Use the Close surface tool to solidify the model.
     - a. Access the Part Design workbench.
     - b. Activate the PartBody.
     - c. Select the **Close Surface** icon.
     - d. Select Join.2 as the object to close.
     - e. Click **OK** to complete the operation.

8. **Add variable fillets.**
   - Complete the model by adding fillets. In this step, add variable fillets to the bottom side edges.
     - a. Select the **Variable Edge Fillet** icon.
     - b. Select both the edges of the bottom of the model.
     - c. Create the fillets with a [2mm] radius at the top and [4mm] radius at the bottom.
     - d. Click **OK** to complete the operation.





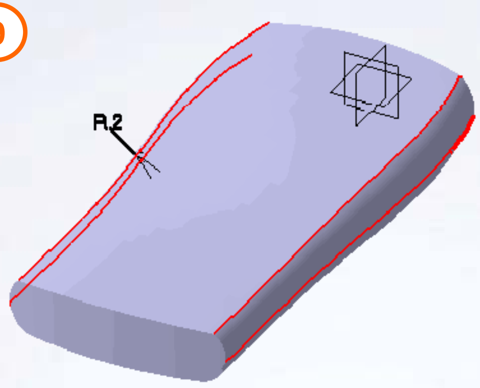
Do it Yourself (5/5)

9. **Apply edge fillets.**
   - Create edge fillets for the top and middle side edges.
     a. Select the **Edge Fillet** icon.
     b. Select the top and middle edges on both sides (four edges).
     c. Use a [2mm] radius.

10. **Apply edge fillets.**
   - Complete the model by adding 2mm edge fillets to the top and bottom faces of the model.

11. **Save and close the model.**

9



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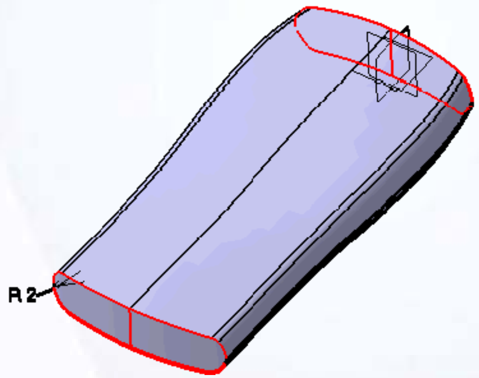
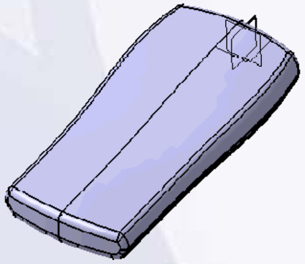
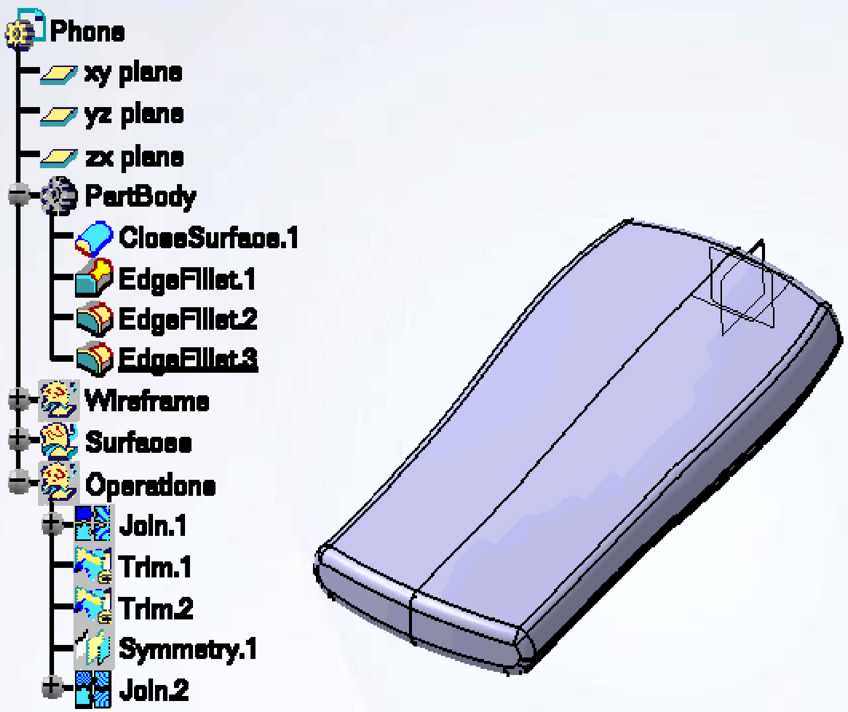




Exercise: Join, Trim and Close Surface Recap

- ✓ Join surfaces
- ✓ Trim surfaces
- ✓ Mirror
- ✓ Close a surface



Case Study: Surface Design

Recap Exercise

40 min

In this exercise, you will create the case study model. Recall the design intent of this model:

- ✓ Model contours are likely to change.
- ✓ Wireframe, surface, and solid geometry must be kept separate.
- ✓ Buttons must be built as a separate body, however it must be updated when changes are made to the main body.

Using the techniques you have learned in this and previous lessons, create the model with only high-level instruction.

Do It Yourself: Model of Computer Mouse (1/15)

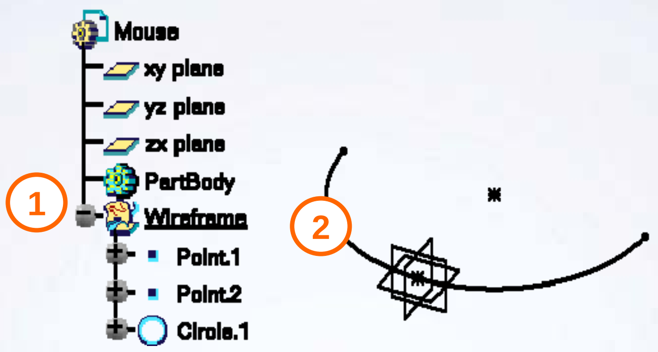
You must complete the following tasks:

1. Create a new part file.

- Create a new part file. Create a geometrical set inside the part called Wireframe and make the Wireframe geometrical set active.

2. Create a semi-circle.

- Create a semi-circle.
- Select XY plane as support.
- Create the center-point for the circle at: X = -44.45, Y = 0, Z = 0.
- Have the circle run though a point located at: X = 0, Y = 0, Z = 0.
- Create the circle starting at –90deg and ending at 90deg.

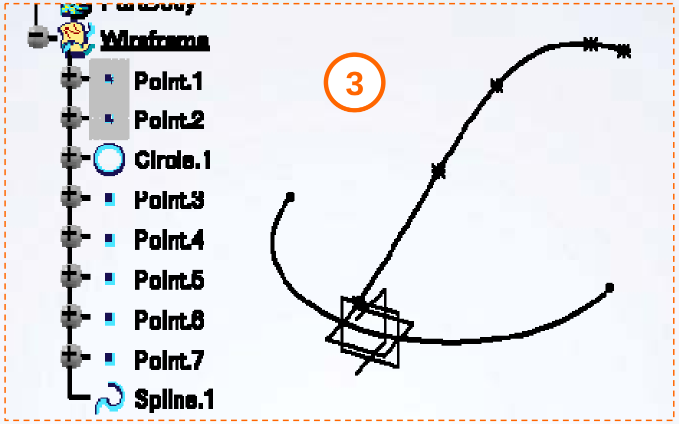


Do It Yourself: Model of Computer Mouse (2/15)

You must complete the following tasks (continued):

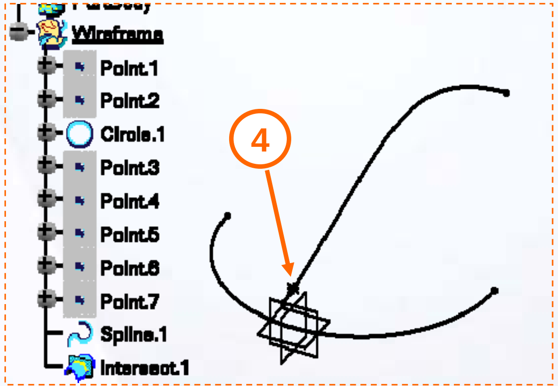
3. Create a spline.

- Create a spline through the following points:
  Pt1: X = 6.65, Y =0.00, Z = 12.70.
  Pt2: X = -38.10, Y = 0.00, Z = 25.40.
  Pt3: X = -69.85, Y = 0.00, Z = 31.75
  Pt4: X = -121.92, Y = 0.00, Z = 12.70
  Pt 5: X = -139.70, Y = 0.00, Z = 0.00



4. Intersect elements.

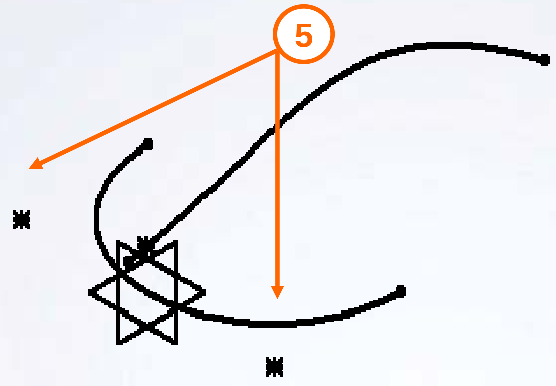
- Using the Intersect tool to intersect the Spline with the YZ plane.



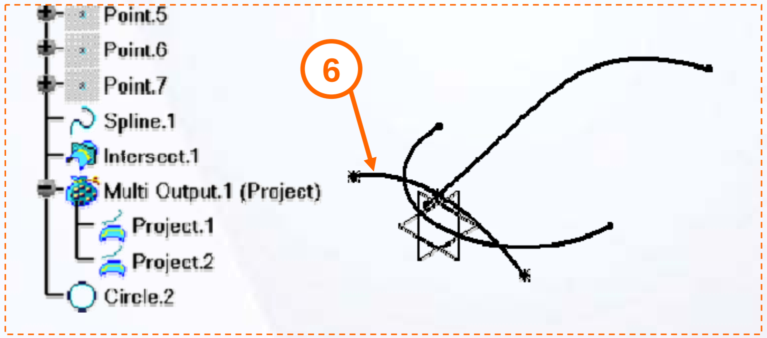
Do It Yourself: Model of Computer Mouse (3/15)

You must complete the following tasks (continued):

5. **Project elements.**
   - Project the semicircle end points onto the YZ plane.



6. **Create trimmed circle.**
   - Create another circle using the Trimmed Circle option. Create the circle through the intersected and projected points.



Do It Yourself: Model of Computer Mouse (4/15)

You must complete the following tasks (continued):

7. Create a spline.

- Create a spline through the following points:

  Pt1: X = 0.00, Y =38.10, Z = 0.00.

  Pt2: X = -38.10, Y = 38.10, Z = 0.00.

  Pt3: X = -68.58, Y = 44.45, Z = 0.00

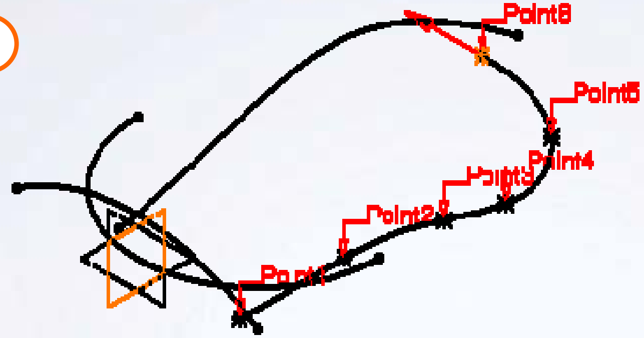
  Pt4: X = -85.09, Y = 50.80, Z = 0.00

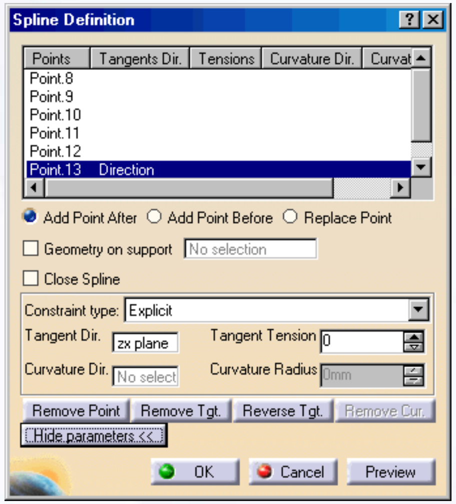
  Pt 5: X = -114.30, Y = 38.10, Z = 0.00

  Pt 6: X = -127.00, Y = 0.00, Z = 0.00

- Create the last point tangent to the ZX plane.

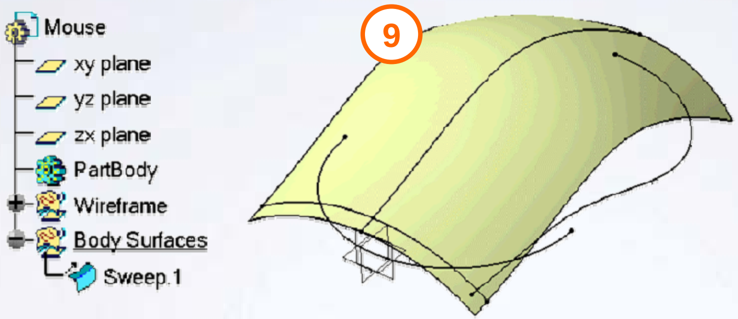
7





Do It Yourself: Model of Computer Mouse (5/15)

You must complete the following tasks (continued):



8. Create a new geometrical set.

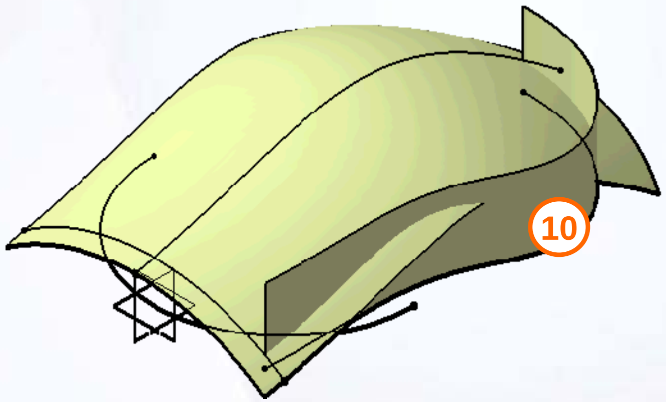
- Create a new geometrical set called Body surfaces and ensure it is active.

9. Create a swept surface.

- Create a swept surface using circle.2 as the profile and spline.1 as the guide curve.

10. Create an extrude.

- Create an extruded surface using Spline.2 as the profile. Extrude the surface [24.5mm] in the direction of the XY plane.

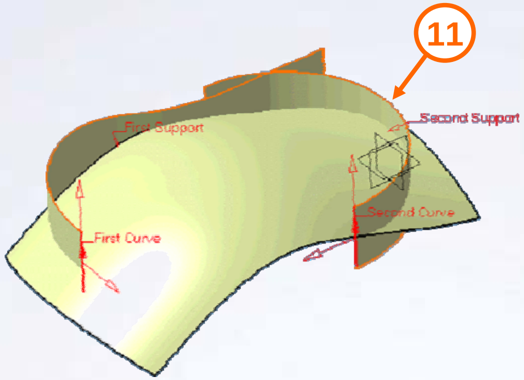


Do It Yourself: Model of Computer Mouse (6/15)

You must complete the following tasks (continued):

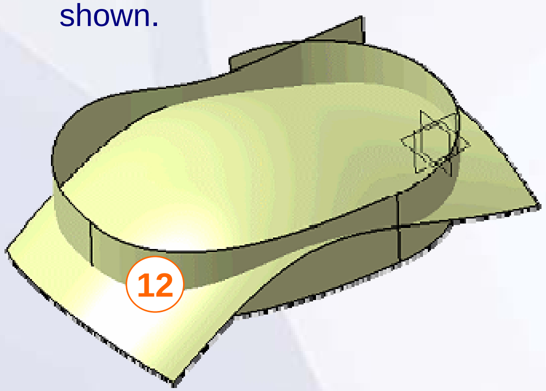
11. Create an extrude.

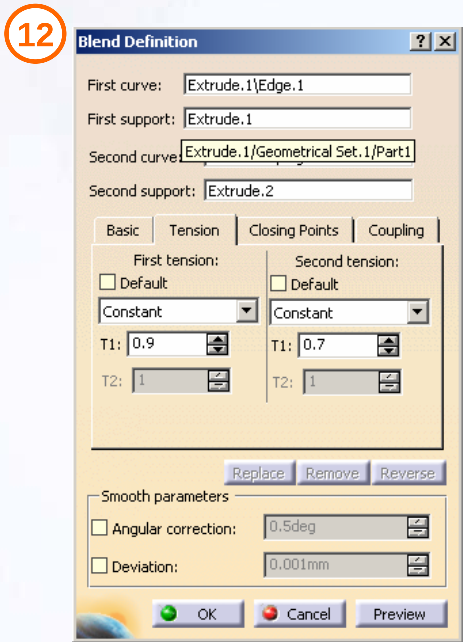
- Create an extruded surface using Circle.1 as the profile. Extrude the surface [24.4mm] in the direction of the XY plane.



12. Create a blend.

- Create a blended surface to connect the two extruded surfaces.
- Apply tensions on the blend as shown.





Do It Yourself: Model of Computer Mouse (7/15)

You must complete the following tasks (continued):

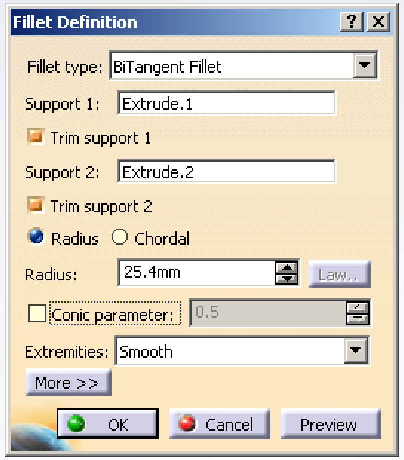
13. Create a shape fillet.

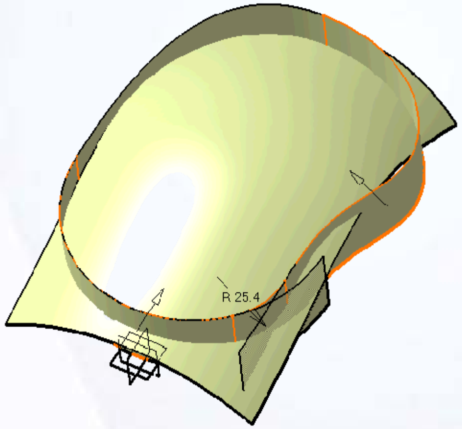
- Create a [25.4mm] shape fillet between the two extruded surfaces.

14. Perform a join operation.

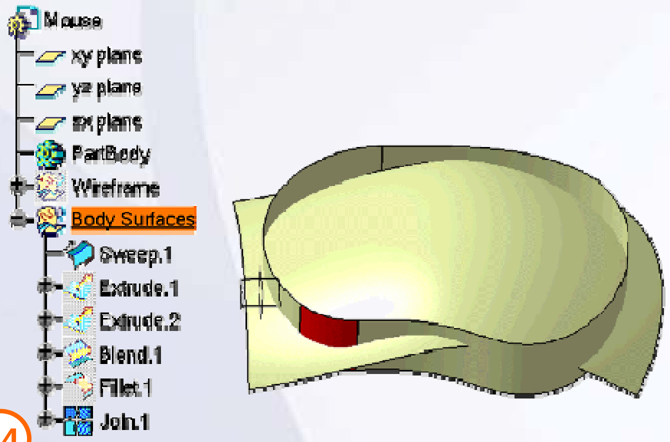
- Join Blend.1 and Fillet.1 using the Join operation.

13





14

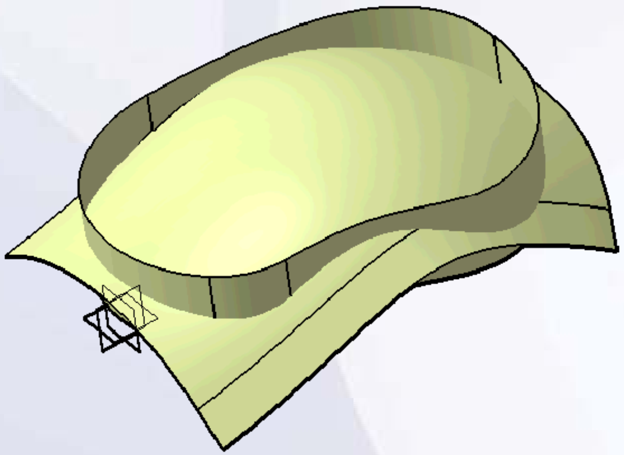


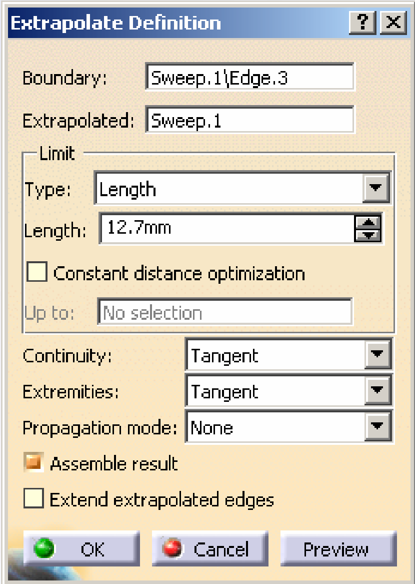
Do It Yourself: Model of Computer Mouse (8/15)

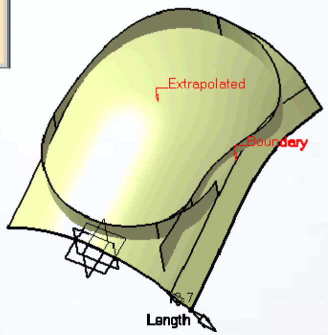
You must complete the following tasks (continued):

15. Extrapolate an edge.

- Extrapolate the edge of the sweep [12.7mm].







Do It Yourself: Model of Computer Mouse (9/15)

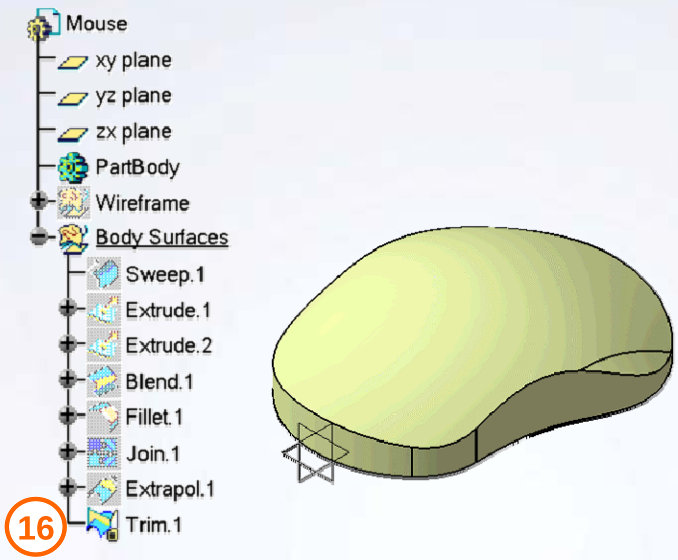
You must complete the following tasks (continued):

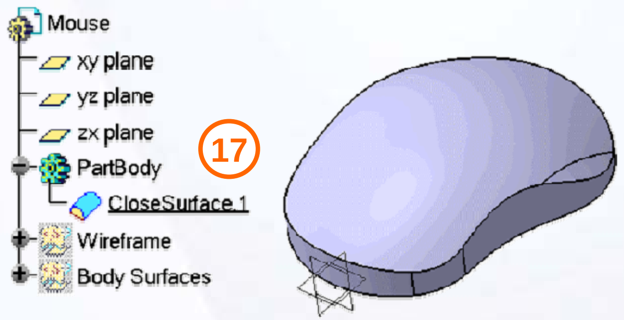
16. Trim surface.

- Trim Join.1 and Extrapol.1.

17. Solidify the model.

- Activate the PartBody and use the close surface tool to solidify Trim.1.





Do It Yourself: Model of Computer Mouse (10/15)

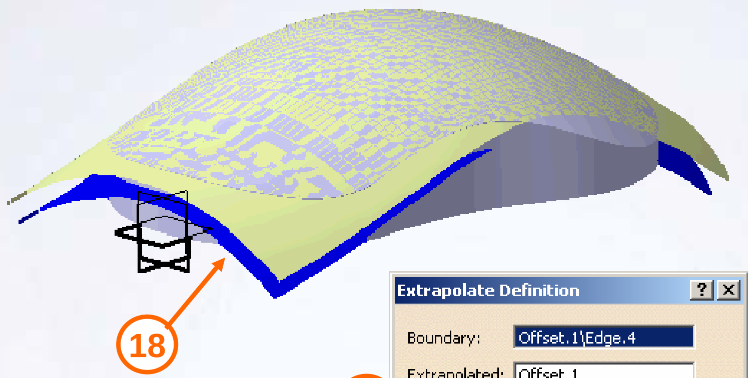
You must complete the following tasks (continued):

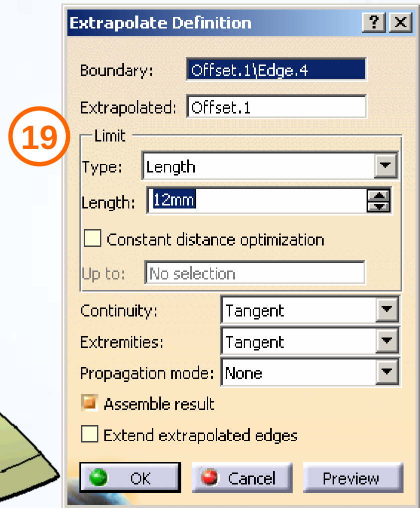
18. Offset a surface.

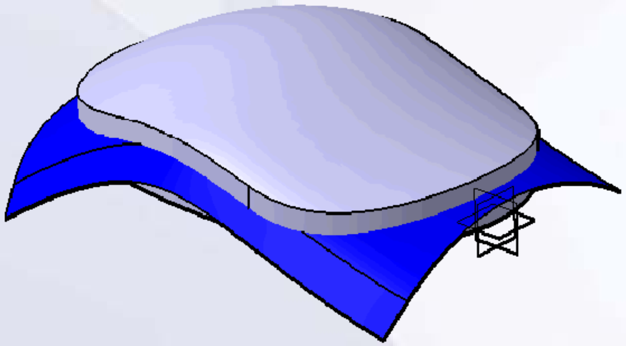
- Reactivate the Body Surfaces geometrical set.
- Offset Sweep.1 using the offset tool [5mm].

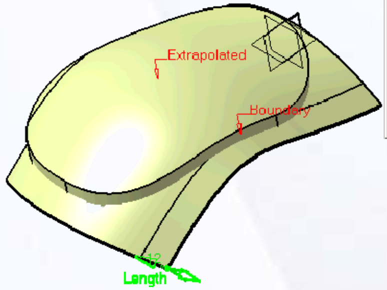
19. Extrapolate the boundary.

- Extrapolate the edge of the offset surface [12mm].







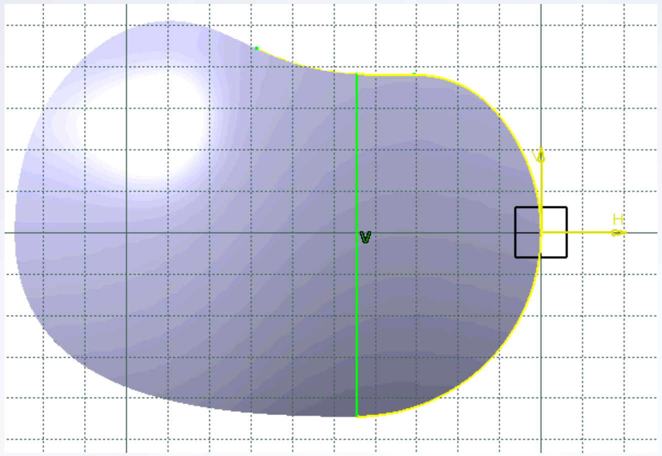


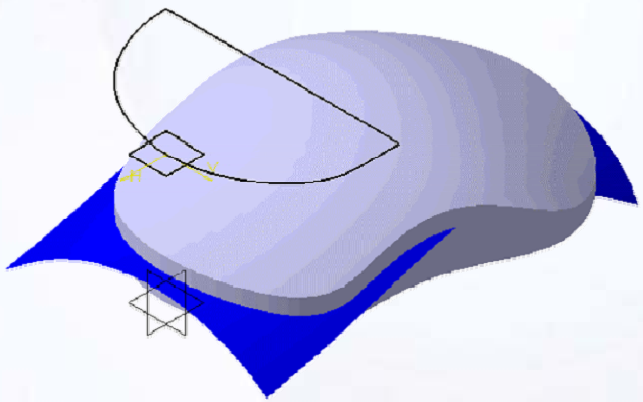
Do It Yourself: Model of Computer Mouse (11/15)

You must complete the following tasks (continued):

20. Create a sketch.

- Create a plane [46mm] above the XY plane. Use this plane as the sketch support to create the sketch shown.
- Project the three curves along the front of the mouse and create a vertical line from the lower curve endpoint to a location on the upper curve. Use the trim tools to trim the upper projected line to the vertical line.



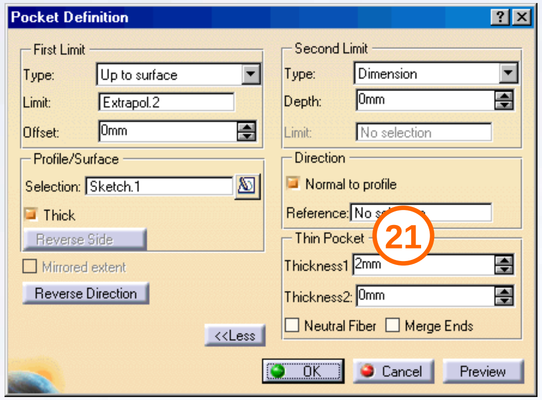


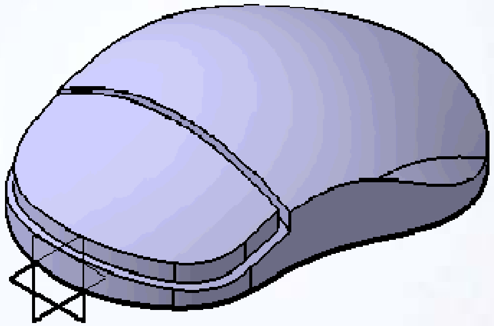
Do It Yourself: Model of Computer Mouse (12/15)

You must complete the following tasks (continued):

21. Create a pocket.

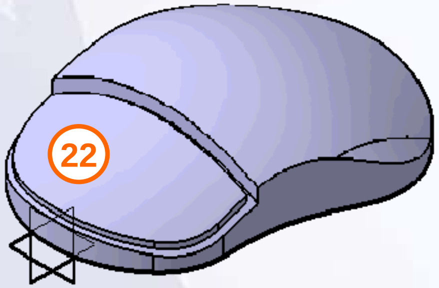
- Use the sketch as the profile for a pocket feature. Extrude the pocket up to the extrapolated offset surface.
- Create the pocket using **Thin Pocket** options with 2mm thickness.





22. Add thickness.

- Use the Thickness tool to add [–3mm] of thickness to the top of the pocket surface.



Do It Yourself: Model of Computer Mouse (13/15)

You must complete the following tasks (continued):

23. Add thickness.

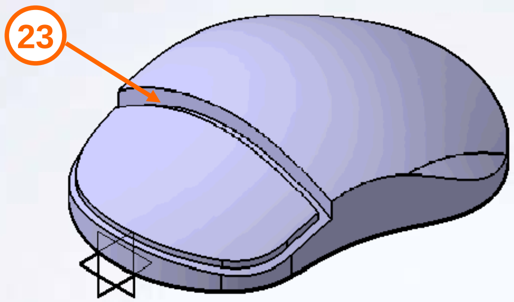
- Use the Thickness tool to add [–1mm] to the back surface.

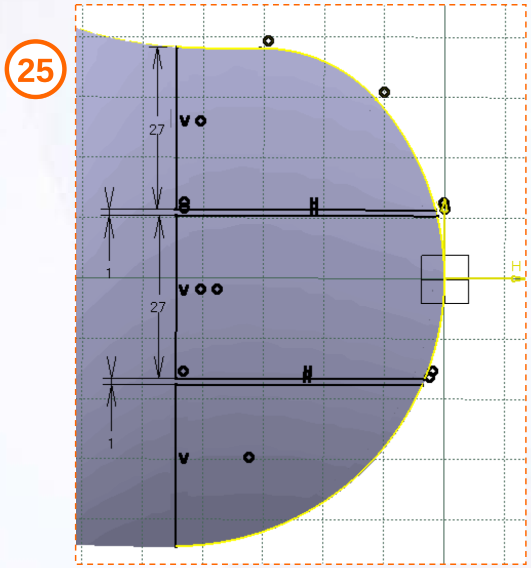
24. Create a new body.

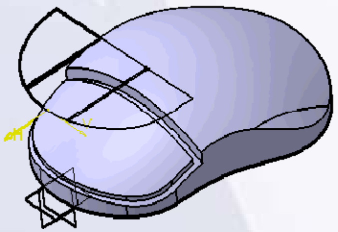
- Create a new body called Button.

25. Create a sketch.

- Copy Sketch.1 from the pocket into the Button body.
- Edit the sketch as shown.







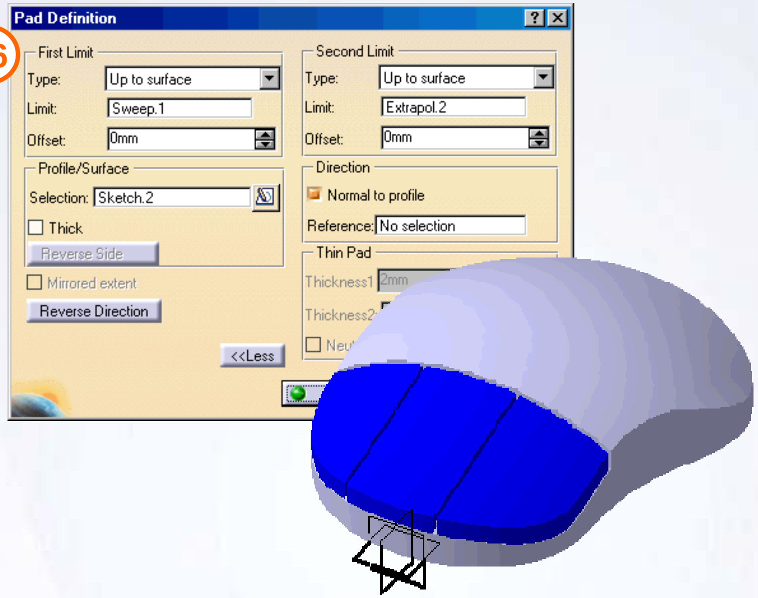
Do It Yourself: Model of Computer Mouse (14/15)

You must complete the following tasks (continued):

26. Create a pad feature.

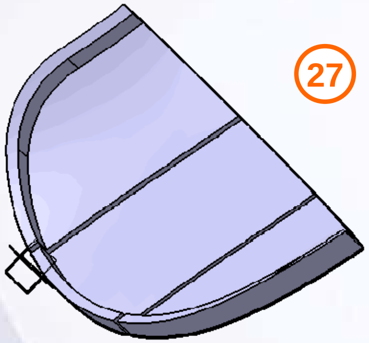
- Create a pad feature using the copied sketch.
- Limit the pad feature between Sweep1and extrapolate.2.

26

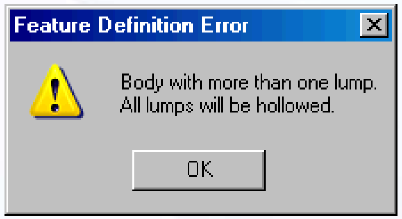


27. Shell the buttons

- Hide the PartBody and shell the buttons to a [2mm] inside thickness.
- Click OK to the warning message.
- Remove all the lower and inside faces from the pad feature.



27



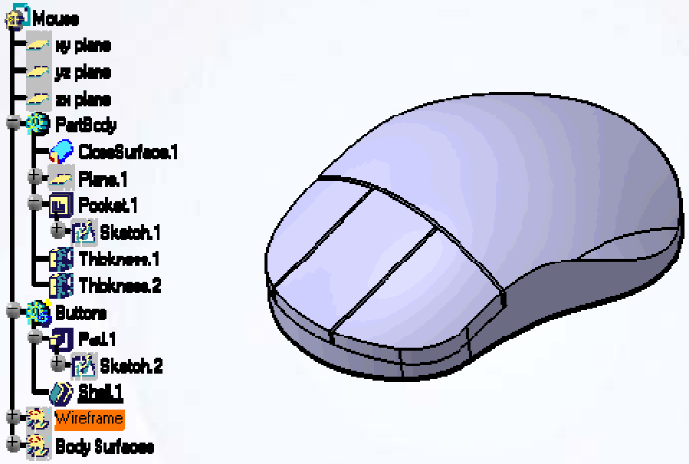
Do It Yourself: Model of Computer Mouse (15/15)

You must complete the following tasks (continued):

28. Clarify the display.

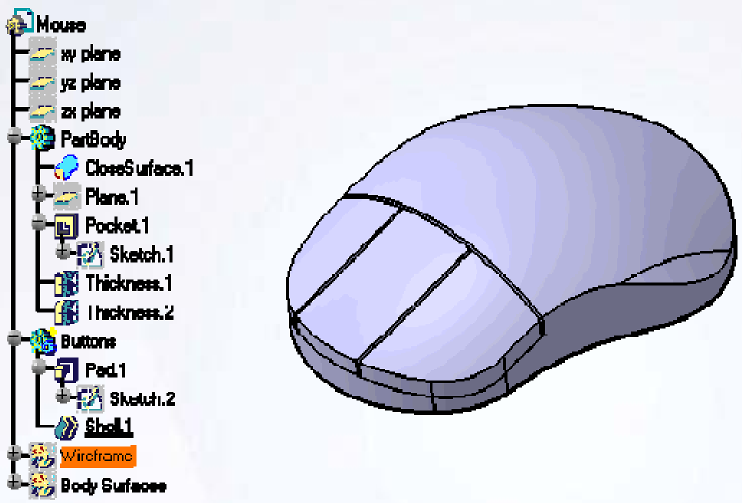
- Show the PartBody and the Buttons body. Hide the Wireframe and Body Surfaces geometrical sets.

29. Save and close the model.



Case Study: Surface Design Recap

- ✓ Create points
- ✓ Create splines
- ✓ Create projections
- ✓ Create intersections
- ✓ Create circles
- ✓ Create swept surfaces
- ✓ Create extrudes
- ✓ Create blends
- ✓ Create fillets
- ✓ Perform a join operation
- ✓ Extrapolate a boundary
- ✓ Trim elements
- ✓ Offset elements
- ✓ Close a surface

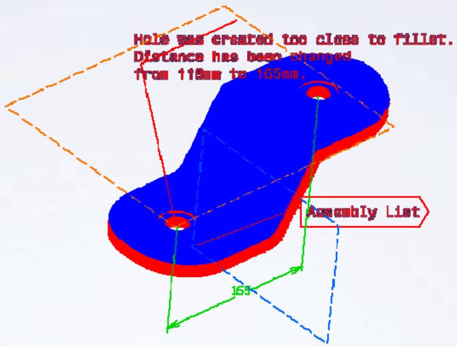


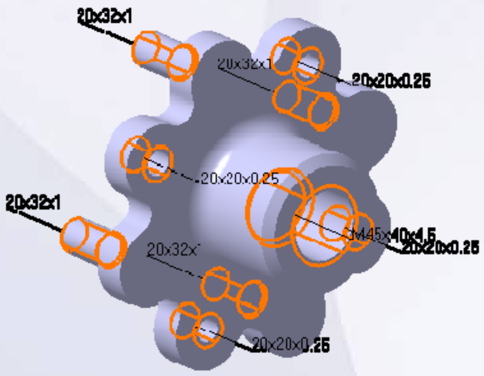
Analyze and Annotate Parts 4

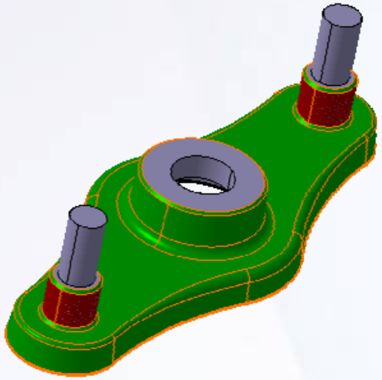
Learning Objectives

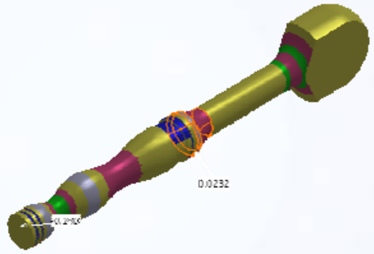
Upon completion of this lesson you will be able to:

- ✓ Analyze the Part
- ✓ Create 3D Constraints
- ✓ Annotate the Part









3 Hours

Case Study

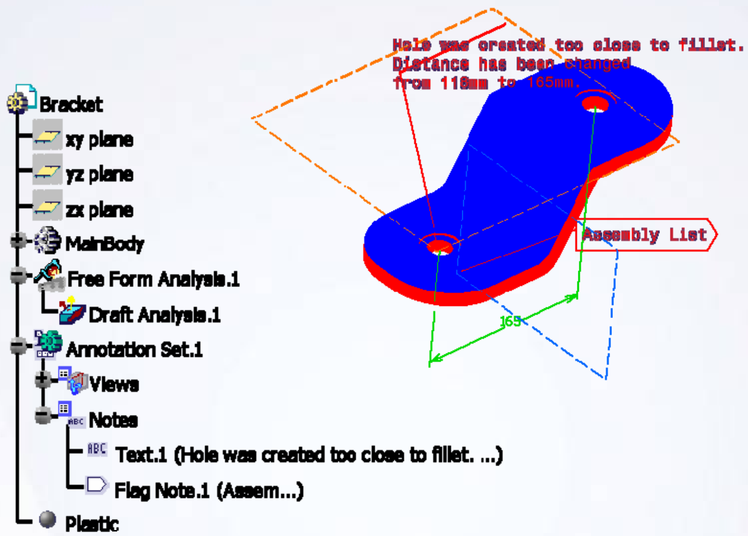
The case study for this lesson is a Bracket part.

Design Intent

1. The model needs to be analyzed and scanned to verify that it follows company policy.
2. The model needs to be constrained. The creator of this model incorrectly constrained the holes. You need to make corrections.
3. Notes must be added. Your company policy requires that you make note of changes that you have made to the part.

Stages in the Process

1. Analyze the Part
2. Create 3D Constraints
3. Annotate the Part



Bracket

Analyze the Part

The following types of analysis can be done:

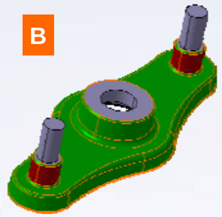
A. Thread and Taps Analysis: Used to visualize threads and tap information.

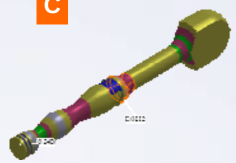
B. Draft Analysis: Used to analyze the ability of a part to be extracted for mold design.

C. Surfacic Curve Analysis: Used to analyze high quality surfaces.

A

Thread and Tap Analysis

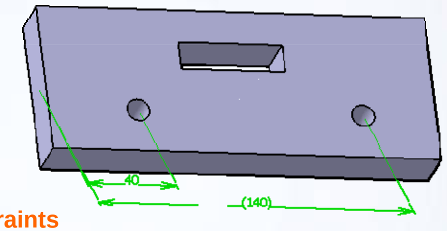
B

Draft Analysis

C

Curvature Analysis

Select View > Render Style >Customized View and check the Material option to view the Draft and Curvature Analysis

Create 3D Constraints

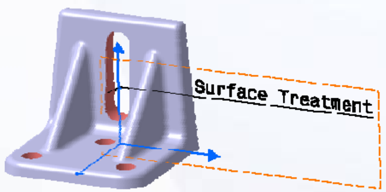
A 3D constraint is created on the 3D model. There are two types of 3D Constraints. Regular Constraints drive the design of the part, and can be driven if necessary. Reference Constraints are created and displayed in parentheses in place of Regular constraints if there a dimension exists which constrains the same aspect of the part.



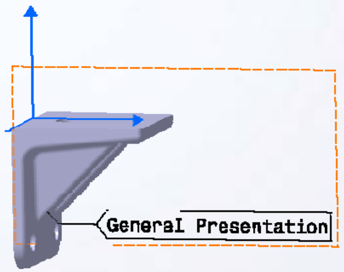
3D Constraints

Annotate the Part

Annotations are added to display the additional information about the part. Some of the annotations that can be added to the part are Text, Text with Leader, and Flag Note. A Flag note links to the information stored in an external file.



Text with Leader



Flag Note

Main Tools

Analysis Toolbar

1 **Draft Analysis:** Analyzes the drafts on a part.

2 **Curvature Analysis:** Analyzes the curvature.

3 **Thread and Tap Analysis:** Creates planes using different options.

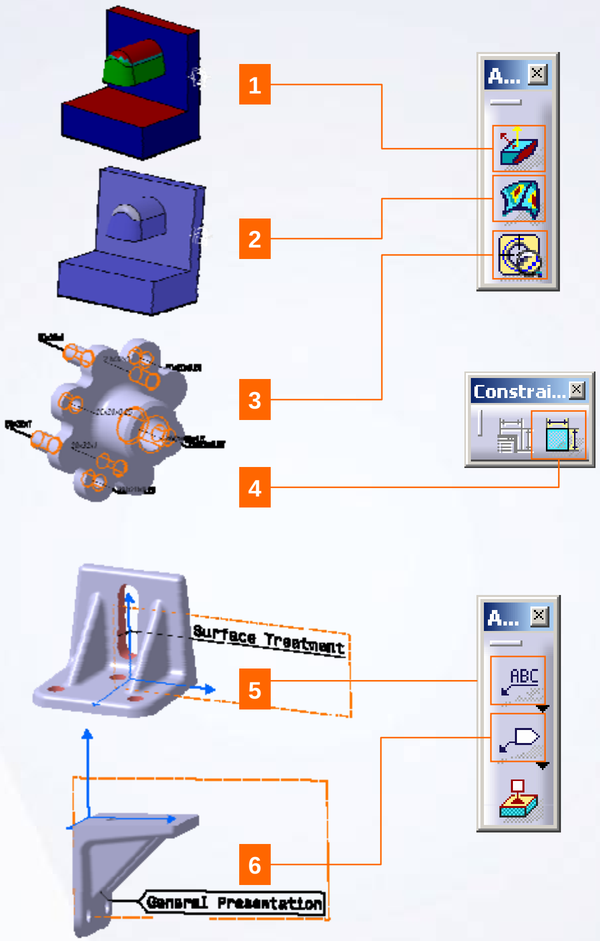
Constraints

4 **Constraint:** Creates a 3D Constraint in a part.

Annotations Toolbar

5 **Text with Leader:** Creates annotation on a part with a leader.

6 **Flag Note:** Creates annotation on a part which links to an additional information stored in an external file.



Case Study: Analyze and Annotate Parts

Recap Exercise

40 min

In this exercise, you will create the case study model. Recall the design intent of this model

- ✓ Model needs to be Analyzed
- ✓ Model needs to be constrained
- ✓ Notes must be added

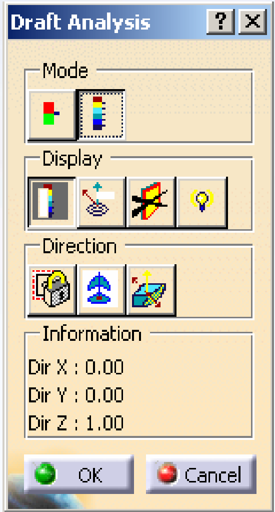
Using the techniques you have learned in this and previous lessons, create the model without detailed instructions.

Do It Yourself: Annotate the Bracket (1/3)

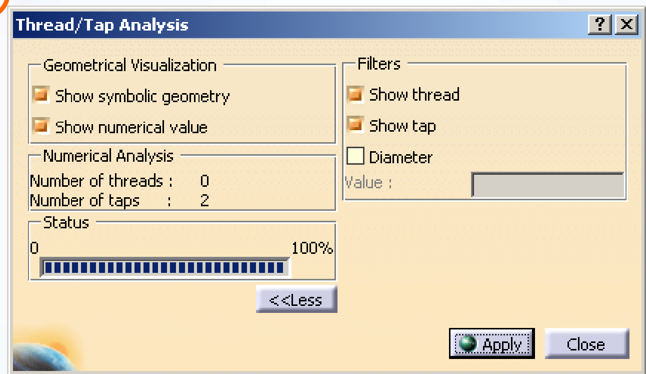
You must complete the following tasks:

1. **Open a part file.**
   - Open Bracket-complete.CATPart.
2. **Scan the part.**
   - Scan the part using the Scan tool.
3. **Perform a Draft Analysis.**
   - Convert the view mode to Material.
   - Perform a Draft Analysis on the part.
     a. Use the Quick Analysis mode
     b. Select the Color Scale and On The Fly options.
4. **Perform a Tap-Thread Analysis.**
   - Perform a Tap-Thread Analysis on the part.
     a. Clear the Show thread option.
     b. Verify that all other options are selected except for Diameter.

3



4



Do It Yourself: Annotate the Bracket (2/3)

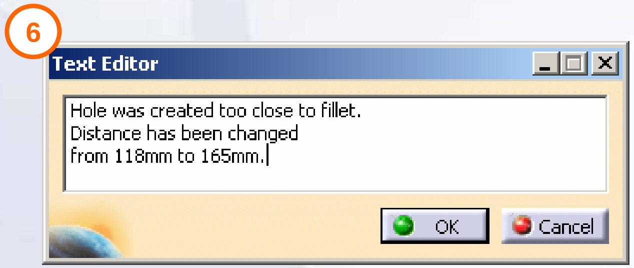
You must complete the following tasks:

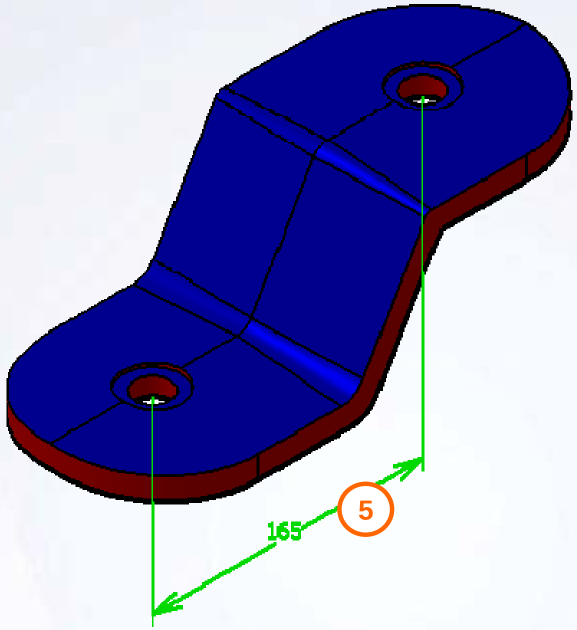
5. Constrain the part.

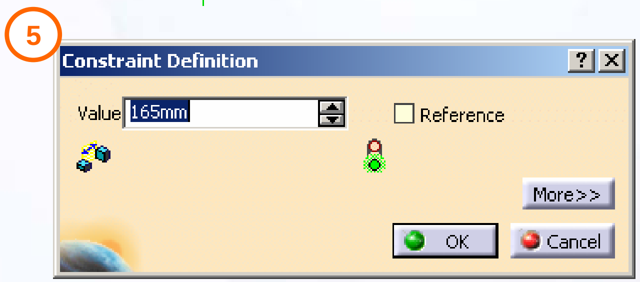
- One of the holes was created too close to a fillet. Constrain this hole to be [165mm] away from the other.

6. Annotate the part.

- Create a text note indicating that the hole has been shifted. Enter the following text: [Hole was created too close to fillet. Distance has been changed from [118mm] to [165mm].





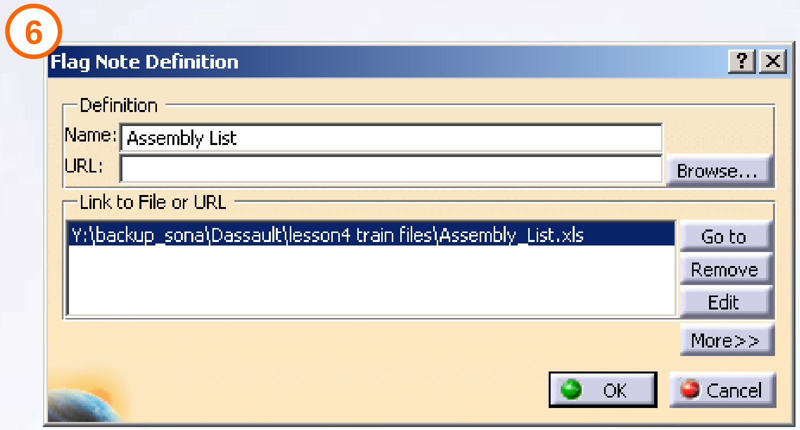


Do It Yourself: Annotate the Bracket (3/3)

You must complete the following tasks:

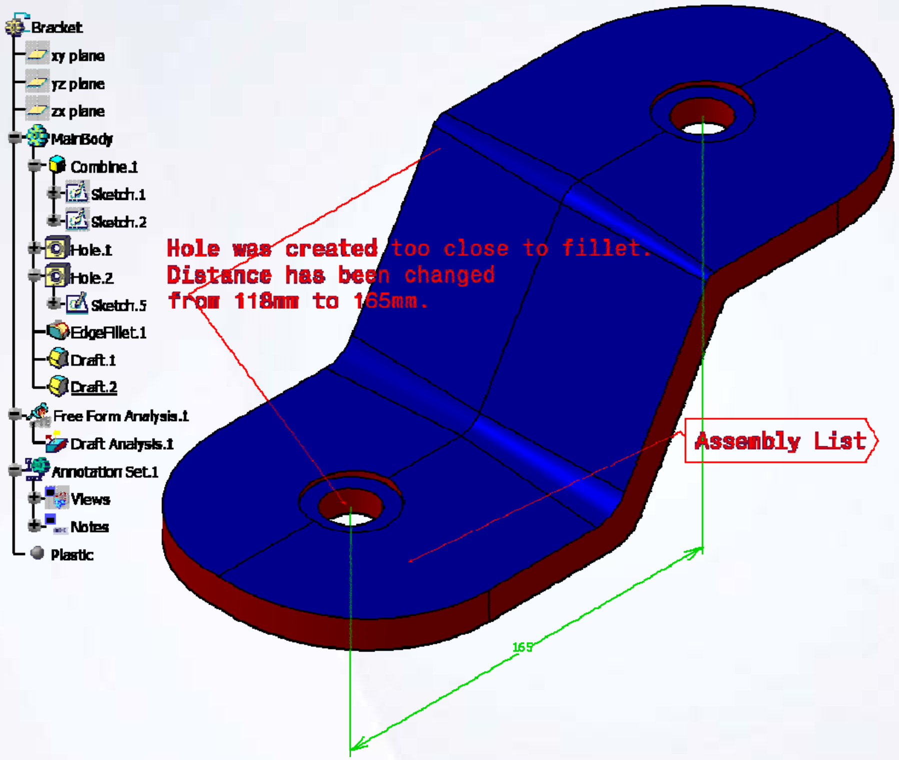
6. Annotate the part (continued).

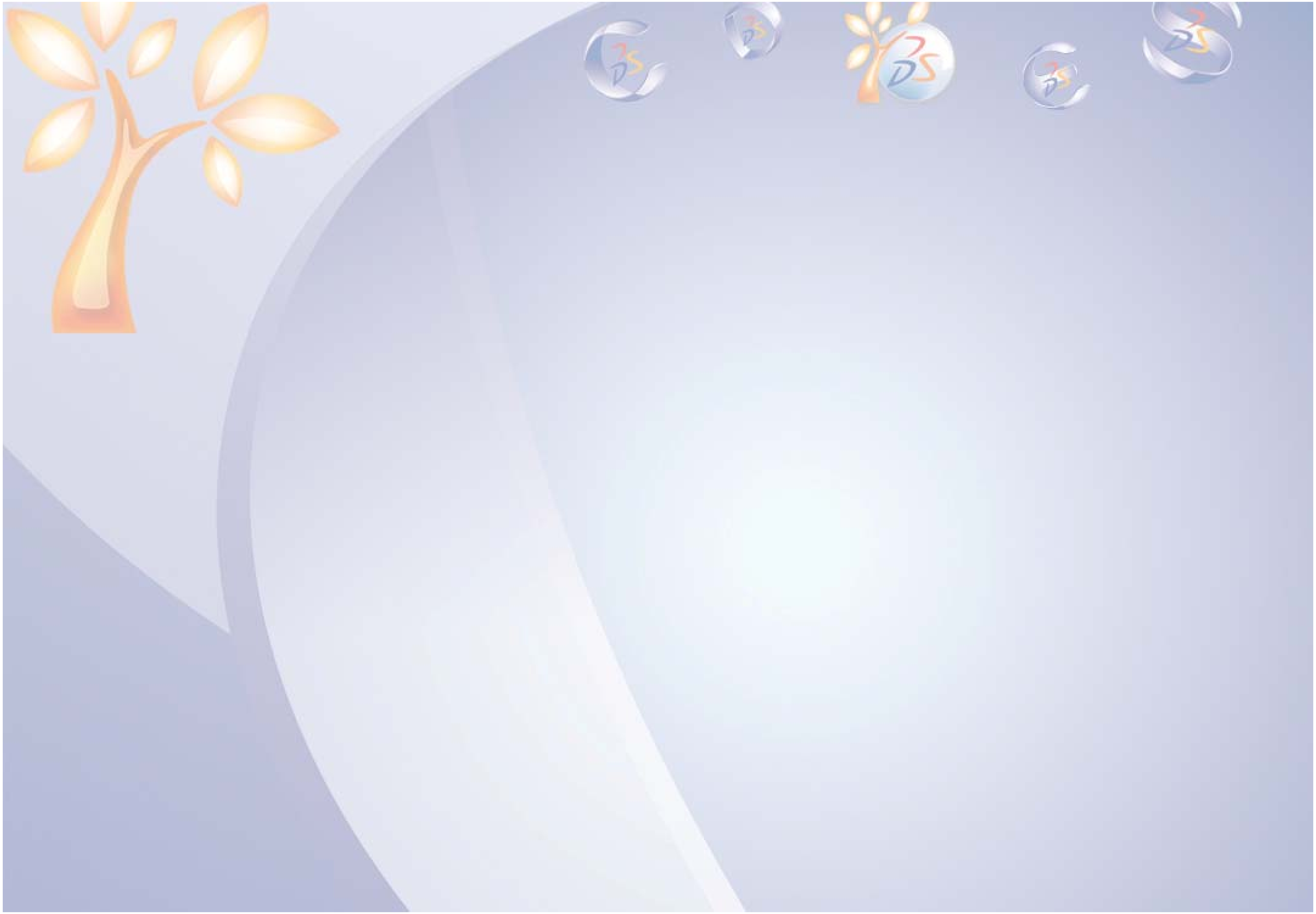
- Create a flag note, referencing a file that lists the assemblies in which this part will be used in.
  a. Name: Assembly List
  b. Link to File or URL: Assembly_List.xls
  c. Modify the notes such that they are always parallel to the screen.
  d. Hide the two view planes that were created while creating the two notes.



Case Study: Analyze and Annotate Parts Recap

- ✓ Scan a Part
- ✓ Perform a Draft Analysis
- ✓ Perform a Tap-Thread Analysis
- ✓ Constrain a Part
- ✓ Annotate a Part



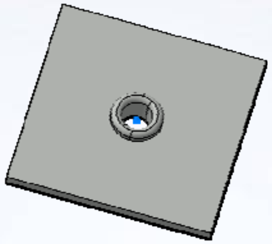


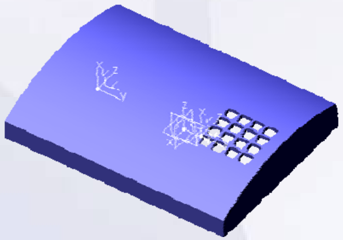
Sharing Information 5

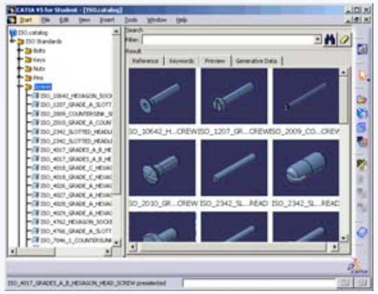
Learning Objectives

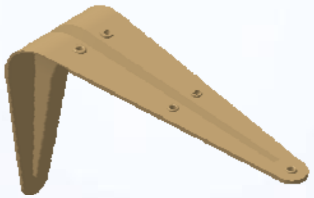
Upon completion of this lesson you will be able to:

- ✓ Create a Power Copy
- ✓ Create Parameters and Formulas
- ✓ Create a Design Table
- ✓ Create a Catalog









4 Hours

Case Study

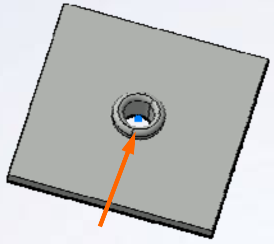
The case study for this lesson is the angle bracket catalog.

Design Intent

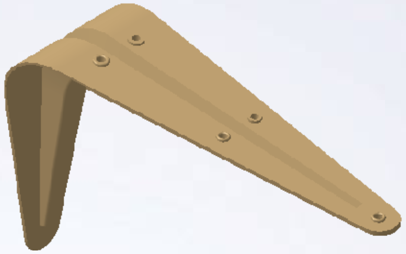
1. You must be able to modify the diameter of the boss hole.
2. The rib of the angle bracket must be related to the length.
3. A catalog of angle brackets must be available.

Stages in the Process

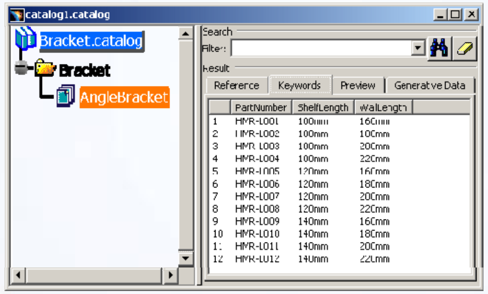
1. Create a Power Copy
2. Create Parameters and Formulas
3. Create a Design Table
4. Create a Catalog



Power Copy



Bracket



Catalog of Bracket with various design configurations

Create a PowerCopy

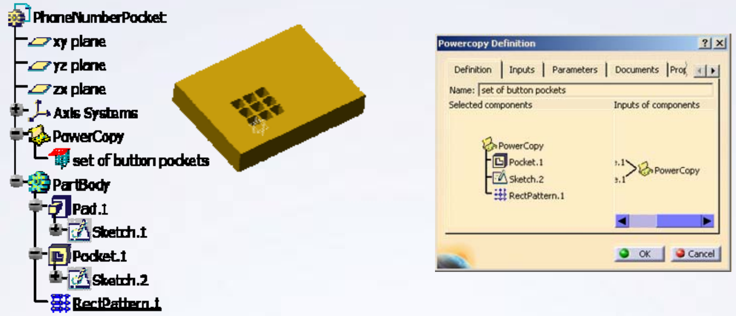
A PowerCopy consists of a group of one or more features that can be re-used in other models. It differs from a typical copy because it contains references which enable you to position the copied features when inserting it in another model.

PowerCopies are stored in the original model or in a Catalog.

While instantiating PowerCopies in the destination documents, you need to specify the necessary inputs and parameter values which will drive the feature parameters being instantiated.

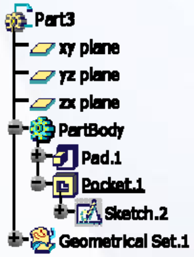
An instantiation of a PowerCopy includes all the design specifications that originally made up the PowerCopy and the features can be modified.

An instantiation of a User Feature hides the design specifications to preserve confidentiality of the features.

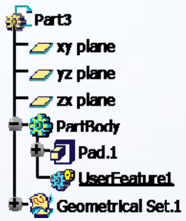


PowerCopy Creation

PowerCopy Instantiation



PowerCopy



Userfeature

Introduction to Knowledgeware

CATIA V5 Knowledgeware is a set of tools to assist engineering decisions by detecting design errors and automating design for maximum productivity. The following terminology is used:

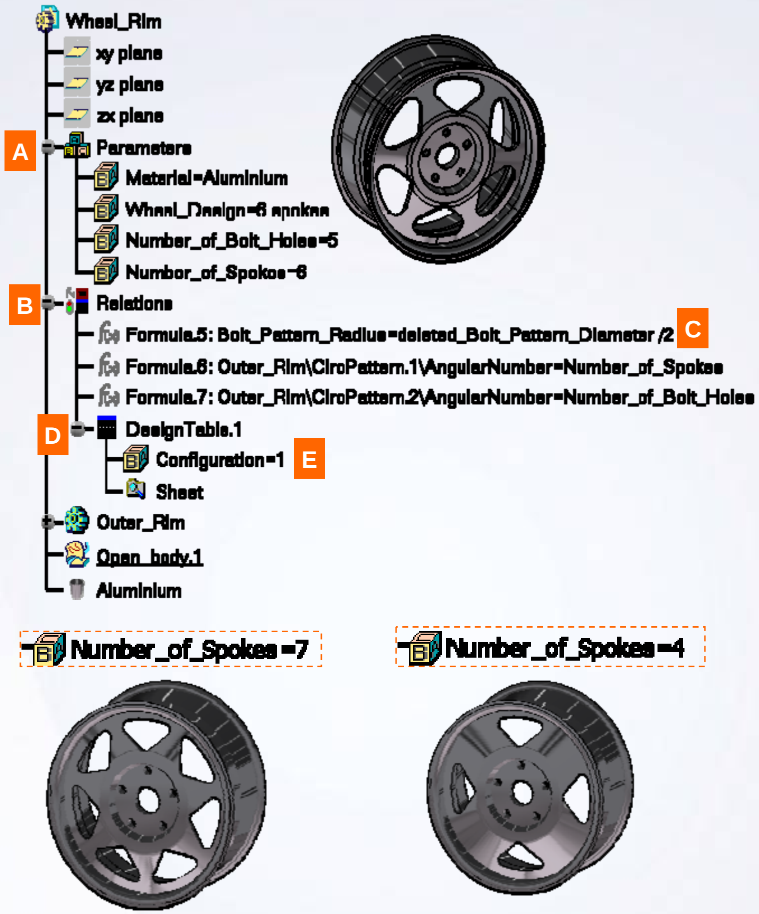
A. A parameter is a property with a given value defined as a feature in the specification tree.

B. A relation is a generic name for knowledge features, such as formulas and design tables.

C. A formula defines how a parameter is calculated with respect to other parameters.

D. A design table is an MS Excel or text table constraining a set of parameters. Each column defines parameter values. Each row defines a configuration.

E. A configuration is a set of parameter values.



The Wheel Rim has a number of user parameters. Number_of_Spokes is one such parameter. Above images show two configurations of the part created by two different values of the Number_of_Spokes.

Create Parameters

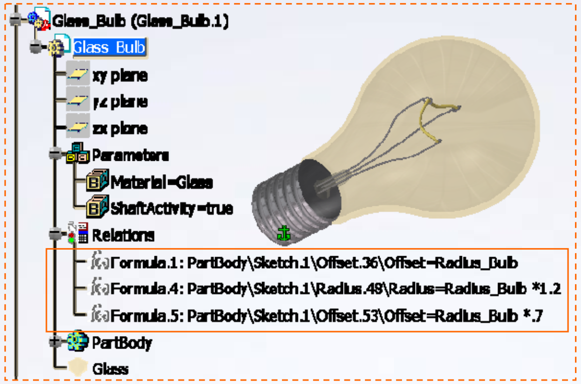
There are two kinds of parameters: intrinsic and user. Intrinsic parameters are created automatically. User parameters are created explicitly by the user.

Parameter values can be defined by relations or used as arguments in a relation.

Create Formulas

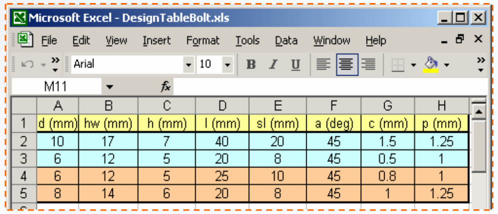
Formulas are relations used to define or constrain any parameter. They can be defined with parameters, operators, and functions. For example a formula is created the moment you attribute a user parameter to a feature. The left part of the relation is the parameter to constrain and the right part is a statement.

When you are editing a formula, you can use pre-defined functions, such as measures. The functions allow you to capture values from the geometry.

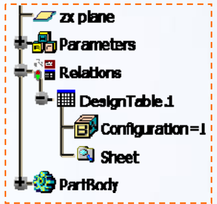


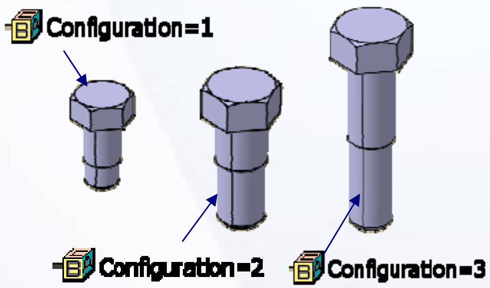
Create a Design Table

The design table provides you with a means to create and manage component families. These components can, for example, be mechanical parts differing in their parameter values. A design table can be created from the CATIA document parameters or from an external file. The values defining the design table are stored either in a Microsoft ® Excel file or in a tabulated text file.



	A	B	C	D	E	F	G	H
1	d (mm)	hw (mm)	h (mm)	l (mm)	sl (mm)	a (deg)	c (mm)	p (mm)
2	10	17	7	40	20	45	1.5	1.25
3	6	12	5	20	8	45	0.5	1
4	6	12	5	25	10	45	0.8	1
5	8	14	6	20	8	45	1	1.25





The Bolt Design uses the Design Table. Different configurations of the bolt refer to different rows in the design table. Each row has a set of parameters that drive the design of the bolt.

Create a Catalog

A catalog is a library of stored items used to avoid having to recreate frequently used geometry.

Catalogs can be created using the Catalog Editor Workbench. A catalog structure consists of the following elements:

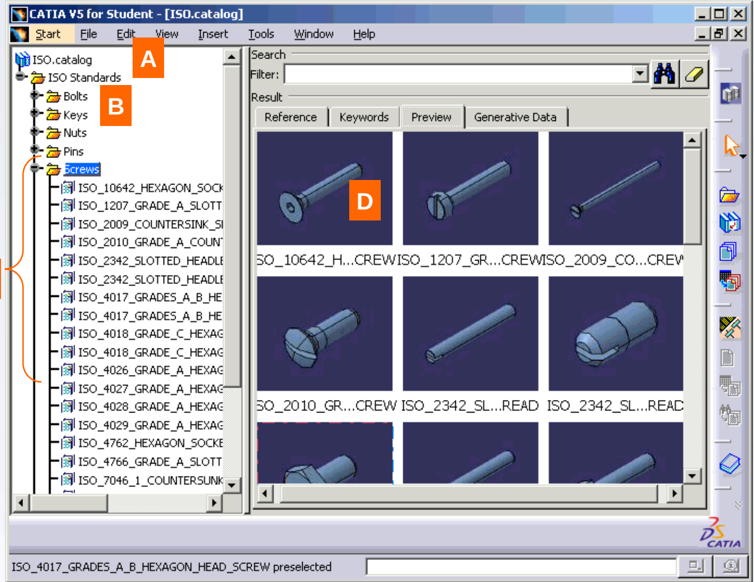
A. Document: For example, an ISO Standards catalog will contain ISO standard parts.

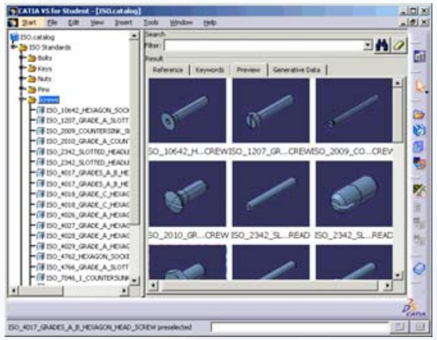
B. Chapter: Used to group entities with a common classification. i.e. Fasteners. Chapters may contain several component families such as Bolts, Pins and Nuts.

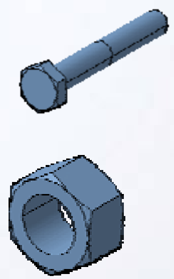
C. Family: A set of components with the same classification. For example – all types of Bolts.

D. Component: It is a reference to an entity stored in the catalog. For example – a Screw.

Once the catalogs have been created, the catalog browser allows you to preview the objects in the catalog as well as to view and sort the object descriptions.







Main Tools (1/2)

PowerCopy Tools

1 **PowerCopy / PowerCopy Creation:** Creates a new PowerCopy feature. The exact location of the PowerCopy tools varies depending on the active workbench. It can be accessed from **Insert > Knowledge Templates** menu or **Insert > Advanced Replication Tools** Menu.

2 **Save In Catalog:** Saves a PowerCopy feature in a Catalog.

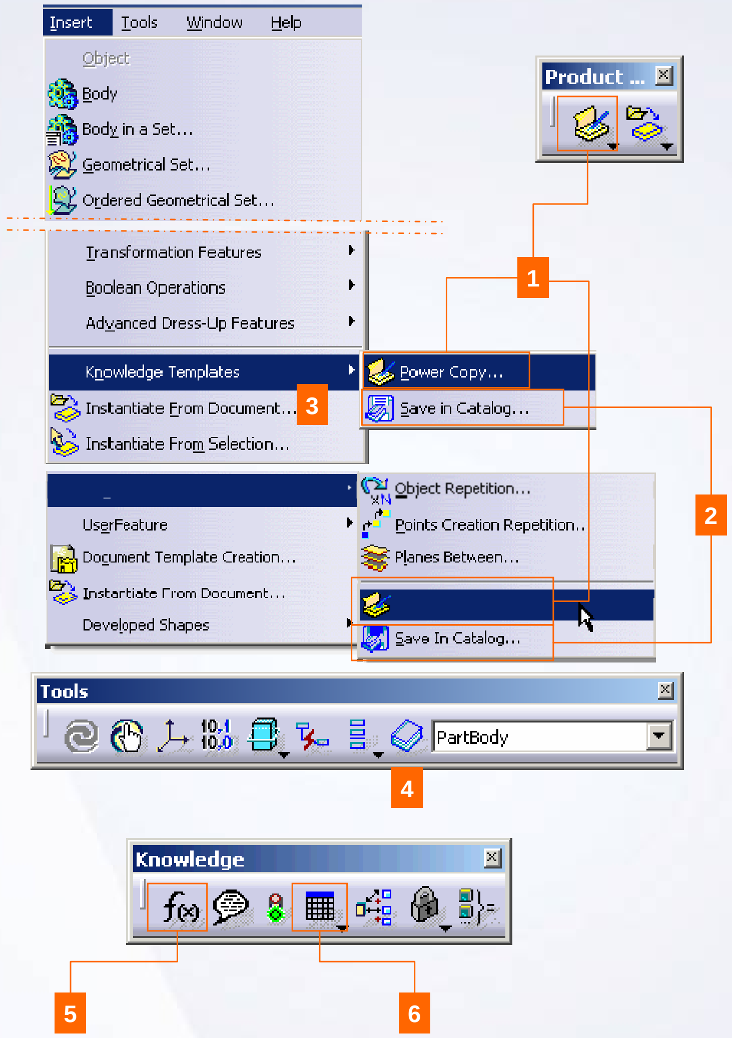
3 **Instantiate from Document:** Instantiates a PowerCopy from the existing document.

4 **Catalog Browser:** Instantiates a PowerCopy from a Catalog.

Knowledge Toolbar

5 **Formula:** Creates parameter / formula using the Formula editor.

6 **Design Table:** Creates a design table.



Main Tools (2/2)

Chapter Toolbar

7 **Add Chapter:** Adds a new Chapter in a Catalog/active Chapter.

8 **Add link to Another Catalog:** Adds link to another catalog.

9 **Add Family:** Adds family in a Chapter.

10 **Add Part Family:** Creates a Part family.

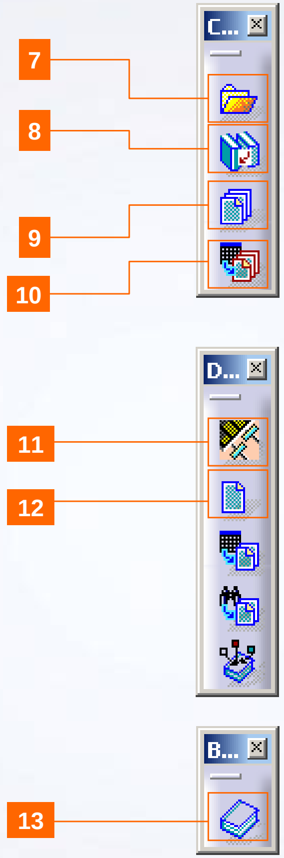
Data Toolbar

11 **Add Keyword:** Adds a keyword to a chapter/component family.

12 **Add Component:** Adds a component in a family.

Browser Toolbar

13 **Display With Browser:** Displays a catalog browser.



Exercise: PowerCopy and Catalog

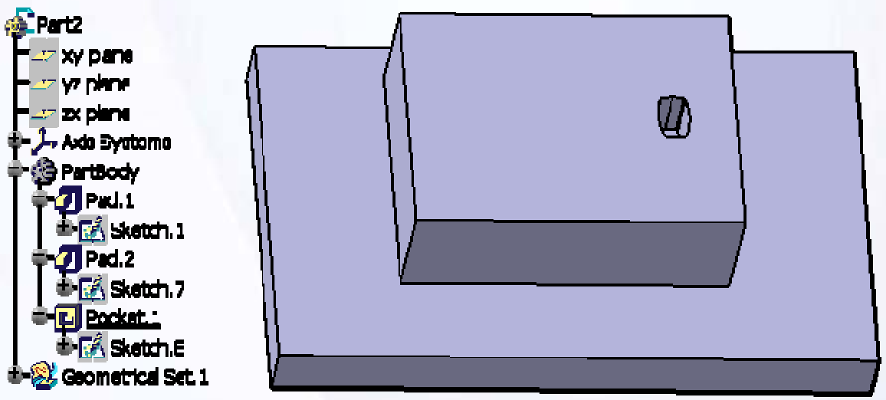
Recap Exercise

10 min

In this exercise, you will create a PowerCopy, save it, and instantiate it. The features to be included in the PowerCopy have already been constructed for you. Limited instructions are provided for this exercise.

By the end of this exercise you will be able to:

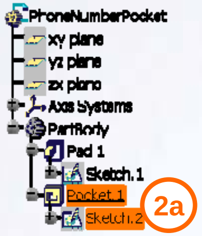
- **Create a PowerCopy**
- **Save PowerCopy in a catalog**
- **Instantiate a PowerCopy from a catalog**

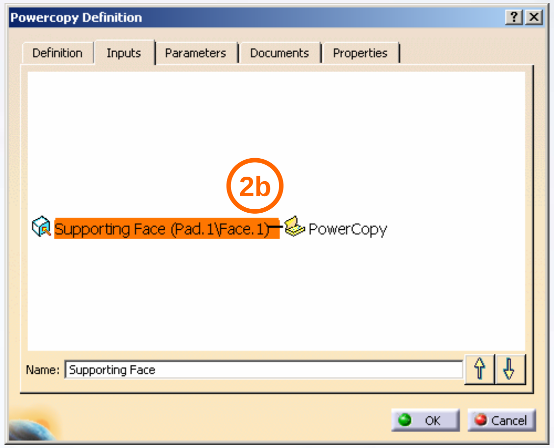


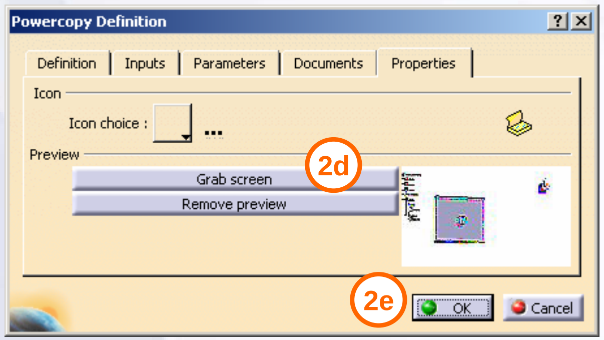
Do it Yourself (1/4)

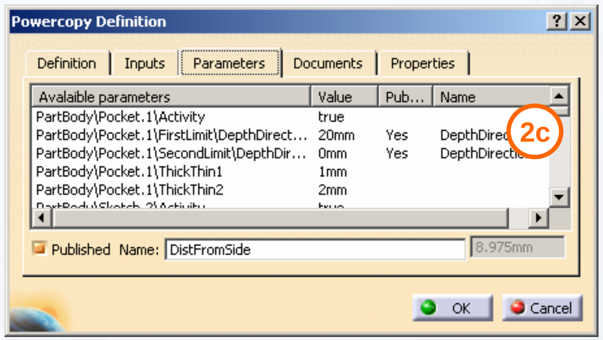
1. **Open Ex5BReference.CATPart.**

2. **Create a PowerCopy of the pocket and its sketch.**
   a. Select Pocket.1 and Sketch.2 for the PowerCopy.
   b. Rename the input to [Supporting Face].
   c. Publish the following parameters: DepthDirection1, DepthDirection2, DistFromBottom, Height, and DistFromSide.
   d. Take a screen grab of the pocket.
   e. Click **OK** to complete the PowerCopy.









Do it Yourself (2/4)

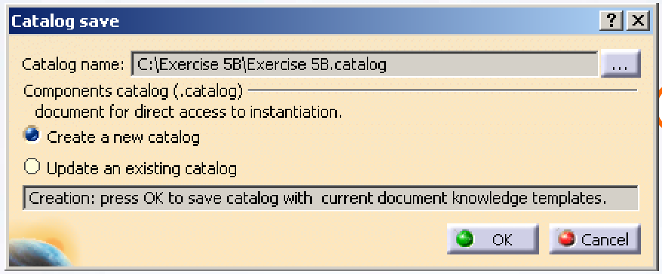
3. **Save the PowerCopy in a new catalog**.
   a. Click **Insert > Knowledge Templates > Save In Catalog.** The Catalog Save dialog box appears.
   b. Click on the… button shown to select a directory and name for the catalog. The File Selection dialog box appears.
   c. Specify a directory and a name for the catalog.
   d. Click **Open** to accept the directory and name.
   e. Click **OK** to save.

4. **Save the document.**

5. **Close the document.**

6. **Open Ex5B.CATPart.**

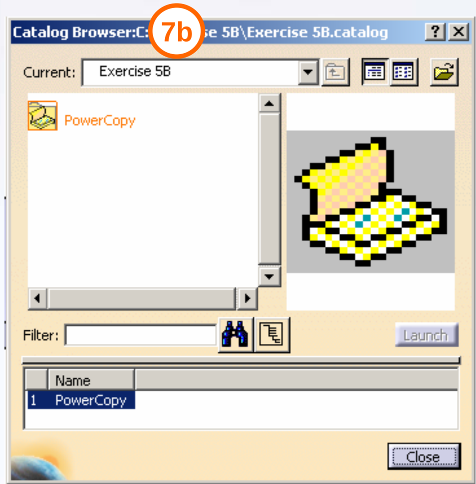
3

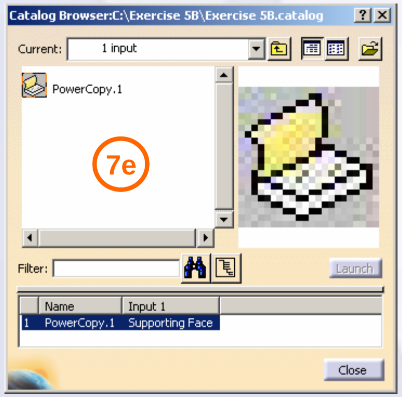


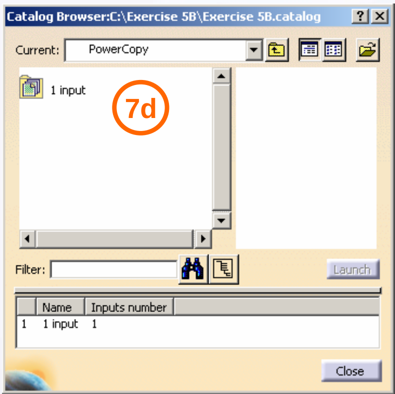
3b

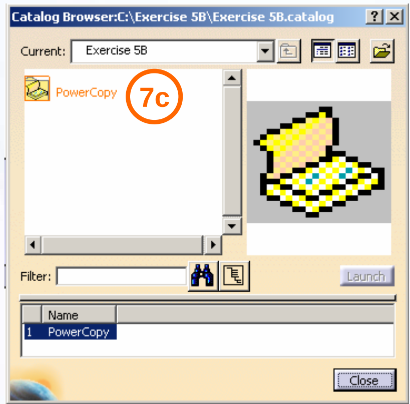
Do it Yourself (3/4)

7. **Instantiate the PowerCopy from the catalog.**
   a. Click the **Open Catalog** icon.
   b. Click the **Browse another catalog** icon, locate the saved catalog, select it, and click **Open**.
   c. Double-click on PowerCopy.
   d. Double-click on 1 input.
   e. Single-click on PowerCopy.1 preview.
   f. Double-click on PowerCopy.1 to select this object for instantiation. The Insert Object dialog box appears.



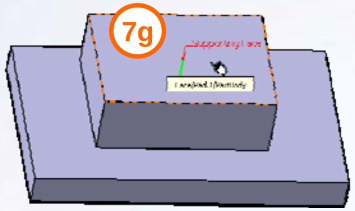


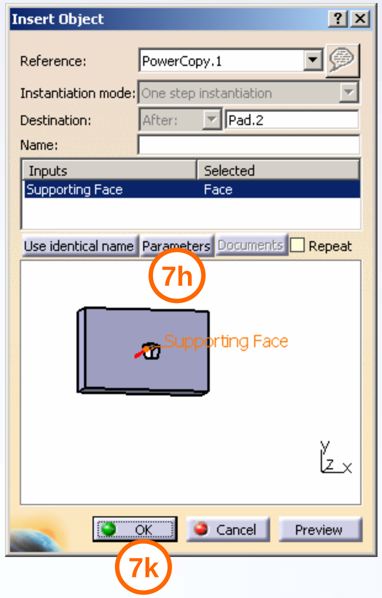


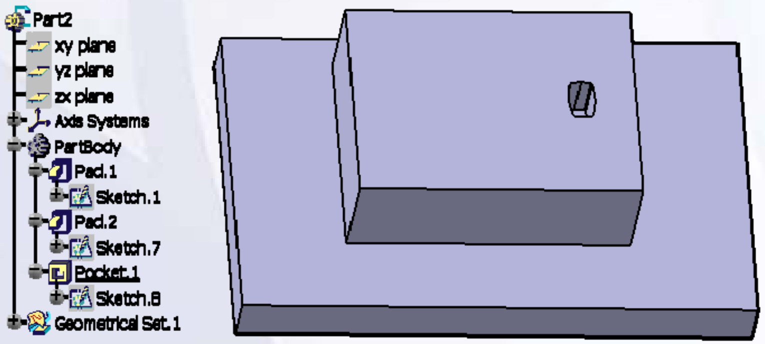


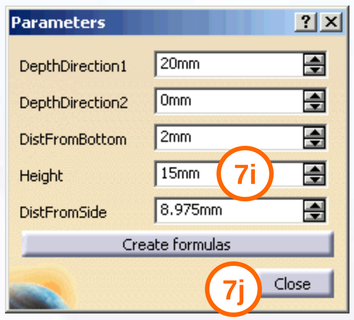
Do it Yourself (4/4)

7. **Instantiate the PowerCopy from the catalog (continued).**
   g. Select the face shown.
   h. Select the **Parameters** button.
   i. Change the value of the Height parameter to [15mm].
   j. Click **Close**.
   k. Click **OK** in the **Insert Object** dialog box.



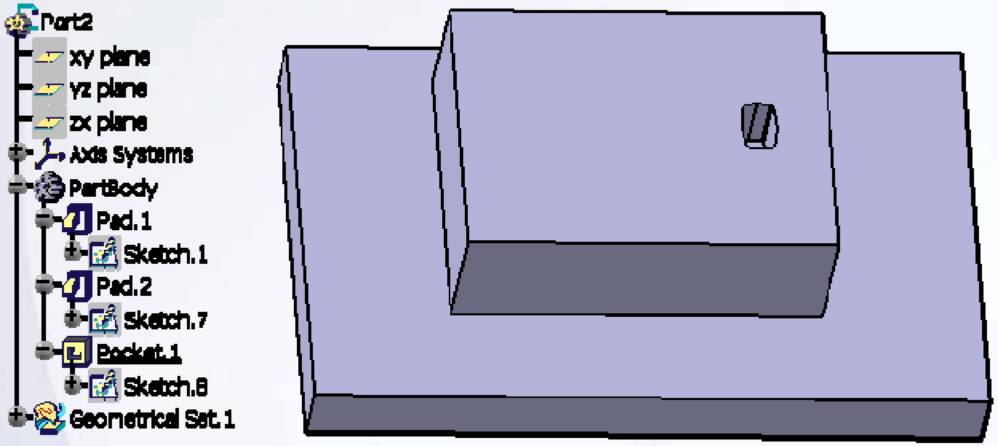






Exercise: PowerCopy and Catalog Recap

- ✓ Create a PowerCopy
- ✓ Save a PowerCopy in a catalog
- ✓ Instantiate a PowerCopy from a catalog



Case Study: Sharing Information

Recap Exercise

30 min

In this exercise, you will create and add a PowerCopy to the case study model. Then you will add the case study model to a part family catalog. Recall the design intent of this model:

- ✓ You must be able to modify the diameter of the boss hole.
- ✓ You must be able to access all features of the template geometry in the specification tree.
- ✓ The rib of the angle bracket must be related to the length.
- ✓ A catalog of angle brackets must be available.

Using the techniques you have learned in this and previous lessons, create the model with only high-level instruction.

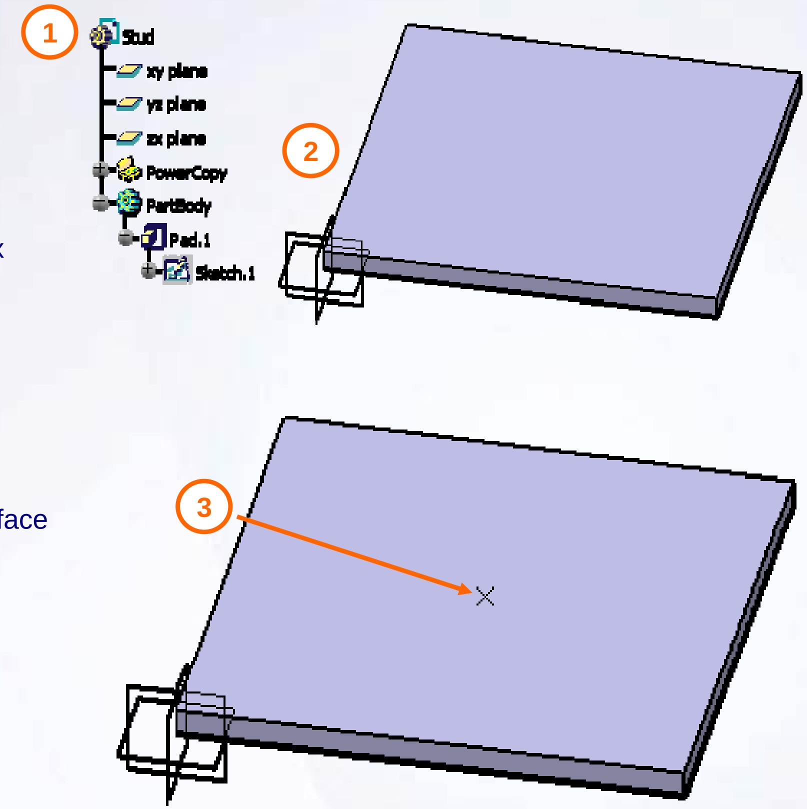
Do It Yourself: Angle Bracket Catalog (1/8)

You must complete the following tasks:

1. **Create a new part file.**
   - Create a new part file named Boss.

2. **Create a Pad.**
   - Create a Pad with dimensions 20mm x 20mm x 1mm.
   - Sketch on the XY plane.
   - Constrain the lower, left corner of the square section to the sketch origin.

3. **Create a Point.**
   - Create a point at the center of the top face of the pad.

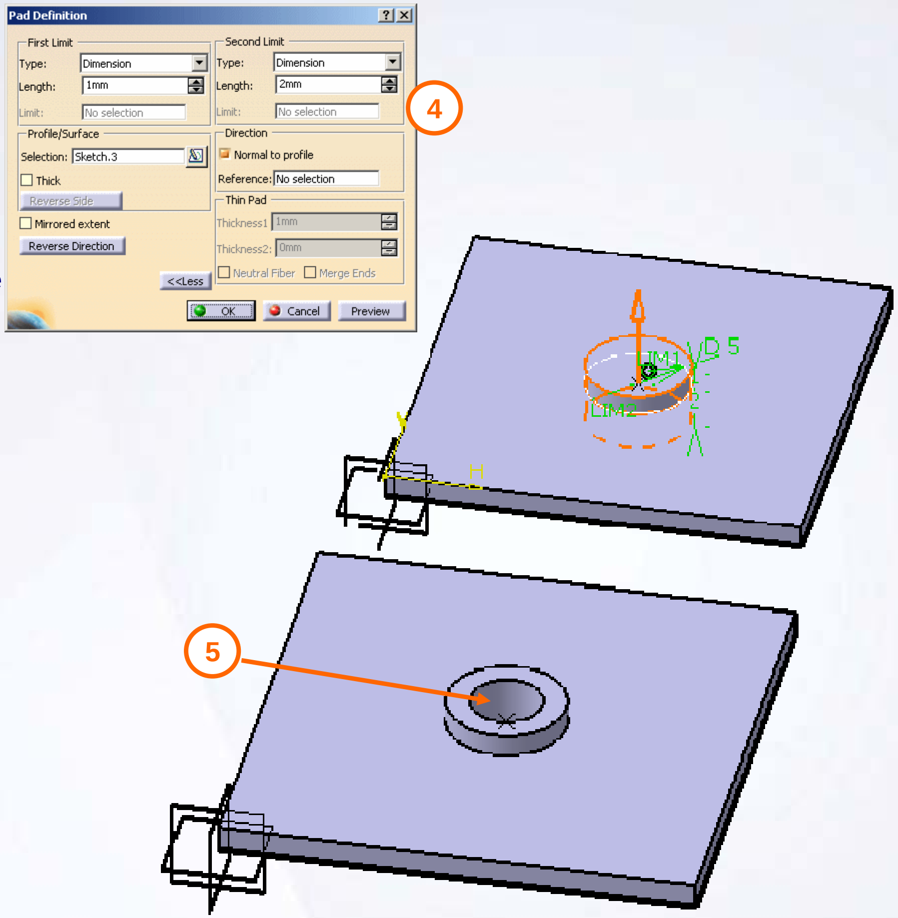


Do It Yourself: Angle Bracket Catalog (2/8)

You must complete the following tasks (continued):

4. Create a Pad.
- Select the top of Pad.1 as the sketch support.
- Create a cylindrical pad and constrain the circular sketch to Point.1. The face of Pad.1 and Point.1 should be the only references selected.
- Enter [5mm] for the diameter of the circle.
- Enter [1mm] for the First Limit.
- Enter [2mm] for the Second Limit.

5. Create a Hole.
- Pre-select the top face of Pad.2 and Point.1.
- Create a 3mm diameter, simple hole that goes up to last.



Do It Yourself: Angle Bracket Catalog (3/8)

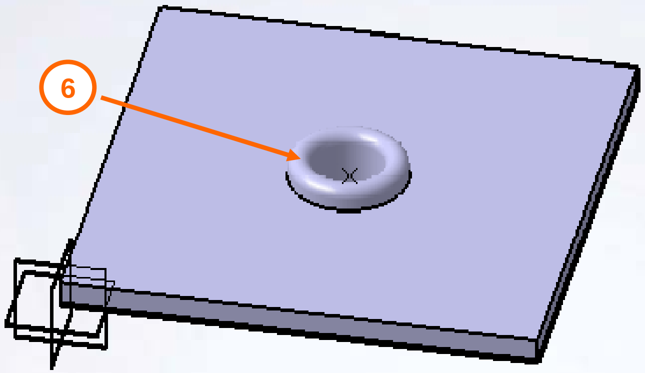
You must complete the following tasks (continued):

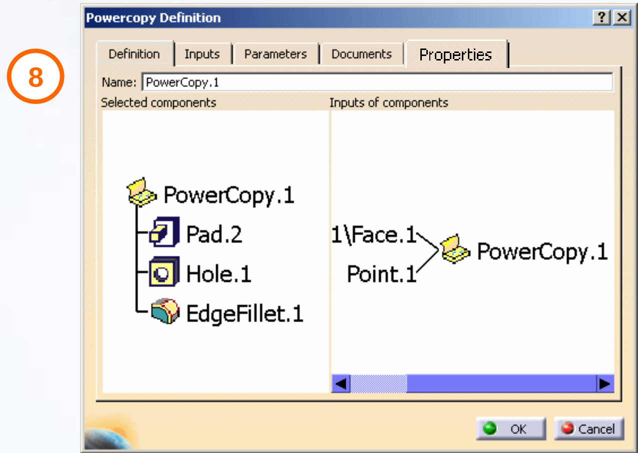
6. Create a Fillet.
- Create a 0.5mm edge fillet on the top and bottom face of the part.

7. Save the model.

8. Create a PowerCopy
- A PowerCopy is used as all features of the boss must be accessible.
- Add Pad.2, Hole.1, and EdgeFillet.1
- Rename the following Inputs:
  - Pad.1\Face.1 = PlaceSurf
  - Point.1 = PlacePnt
- Add the following Parameters:
  - Radius of circle in the sketch for Pad.2
  - Diameter of Hole.1

9. Save the model and close the window.





Do It Yourself: Angle Bracket Catalog (4/8)

You must complete the following tasks (continued):

10. Open AngleBracket.CATPart.

11. Create a Point.

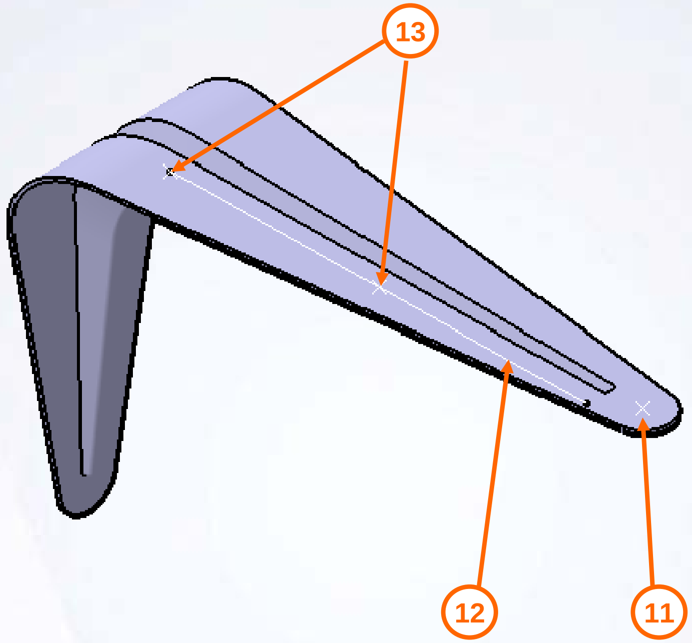
- Create a Circle / Sphere center point.

12. Create a Parallel Curve.

- Access the Generative Shape Design workbench.
- Offset the curve by [5mm].

13. Create two Points.

- Create an On curve point at the midpoint.
- Create an On curve point at the endpoint.



Do It Yourself: Angle Bracket Catalog (5/8)

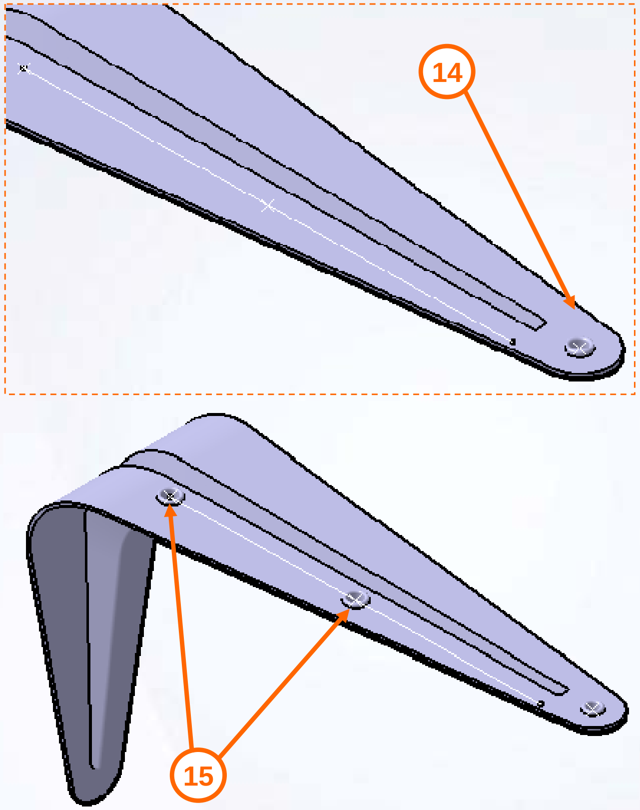
You must complete the following tasks (continued):

14. Insert a PowerCopy.

- Access the Part Design Workbench.
- Instantiate from Boss.CATPart.
- Select PlaceSurf and PlacePnt references to place PowerCopy.1.

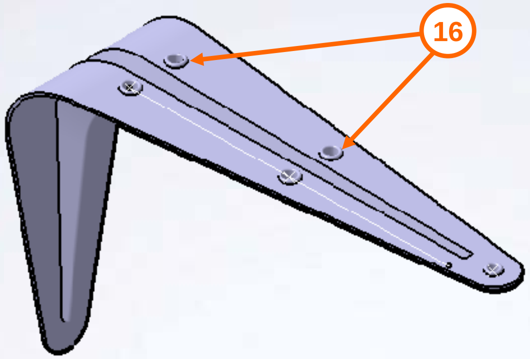
15. Insert two PowerCopys.

- Use the Repeat option to insert two instances of PowerCopy.1 from Boss.CATPart.
- Modify the parameters:
  - Pad radius = [3mm]
  - Hole diameters = [4mm]



Do It Yourself: Angle Bracket Catalog (6/8)

You must complete the following tasks (continued):



16. Mirror two PowerCopys.

17. Hide all wireframe elements.

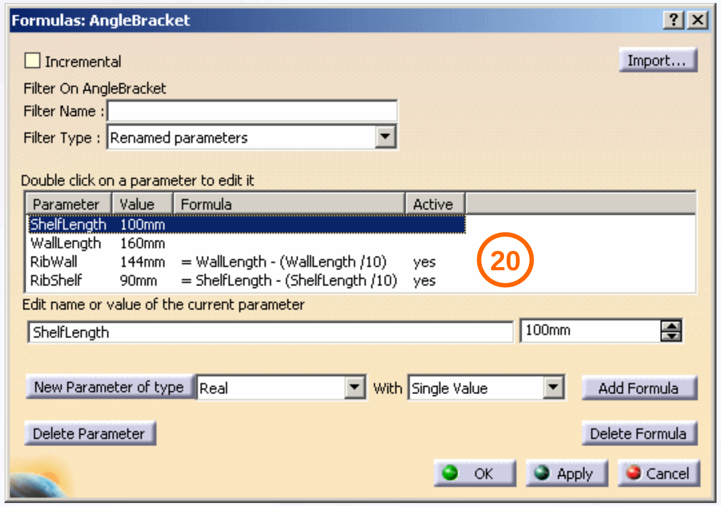
18. Save the model.

19. Create Parameters:

- WallLength = 160mm from Sketch.1
- ShelfLength = 100mm from Sketch.1
- RibWall = RibLimit1 in Geometrical Set.1
- RibShelf = RibLimit2 in Geometrical Set.1

20. Create Formulas:

- RibWall = WallLength – (WallLength/10)
- RibShelf = ShelfLength – (ShelfLength/10)



Do It Yourself: Angle Bracket Catalog (7/8)

You must complete the following tasks (continued):

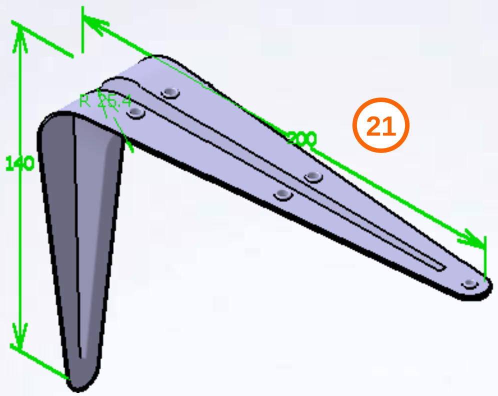
21. Flex the model.

- Modify WallLength to [200mm].
- Modify ShelfLength to [140mm].
- Update the model.

22. Create a Design Table.

- Create a design table with current parameter values.
- Add WallLength and ShelfLength
- Edit the table to add PartNumber column and 12 instances.
- Create PartNumber parameter in CATIA to associate with the PartNumber column in the design table.

23. Save the model and close the window.



22

DesignTable1.xls

	A	B	C
1	PartNumber	ShelfLength (mm)	WallLength (mm)
2	HMR-L001	100	160
3	HMR-L002	100	180
4	HMR-L003	100	200
5	HMR-L004	100	220
6	HMR-L005	120	160
7	HMR-L006	120	180
8	HMR-L007	120	200
9	HMR-L008	120	220
10	HMR-L009	140	160
11	HMR-L010	140	180
12	HMR-L011	140	200
13	HMR-L012	140	220
14			

Sheet1

Do It Yourself: Angle Bracket Catalog (8/8)

You must complete the following tasks (continued):

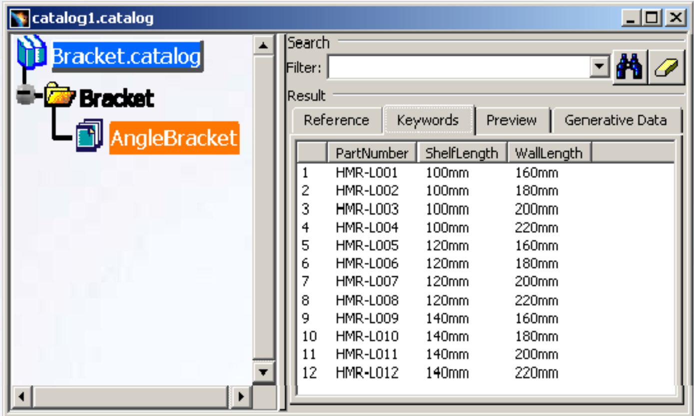
24. Create a Catalog.
- Rename Chapter.1 to [Bracket].
- Save as Bracket.catalog.

25. Add a Part Family.
- Rename the family to [AngleBracket].
- Add AngleBracket.CATPart.
- Resolve the Part Family.

26. Test the Design Table.
- Open HMR-L007 in a new window.

27. Save the catalog and close all windows.

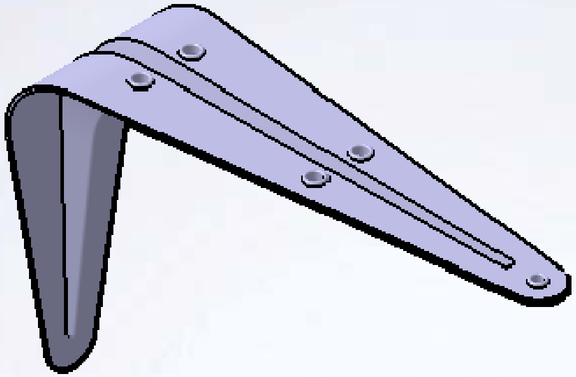
24

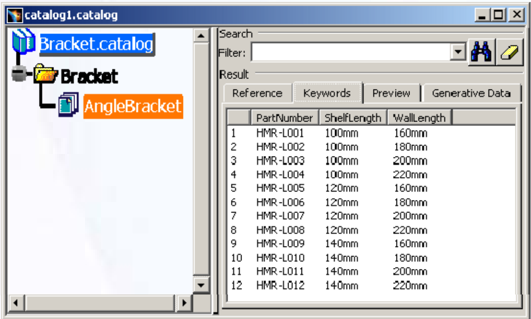


25

Case Study: Sharing Information Recap

- ✓ Create a PowerCopy
- ✓ Instantiate a PowerCopy
- ✓ Modify PowerCopy parameters
- ✓ Mirror a PowerCopy
- ✓ Create Parameters
- ✓ Create Formulas
- ✓ Create a Design Table
- ✓ Add a Part Family to a Catalog
- ✓ Resolve Catalog instances





Assembly Design 6

Learning Objectives

Upon completion of this lesson you will be able to:

- ✓ Manage the product structure
- ✓ Create flexible sub-assemblies
- ✓ Analyze the assembly
- ✓ Create scenes
- ✓ Create annotations
- ✓ Generate reports

4 Hours

Case Study

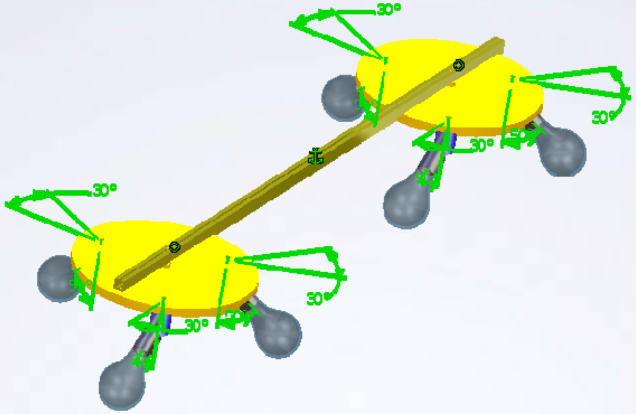
The case study for this lesson is a lighting fixture.

Design Intent

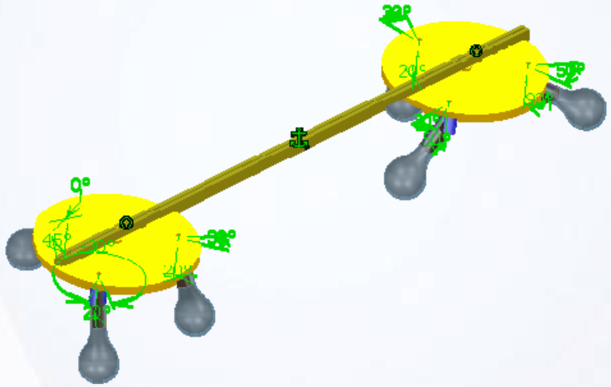
1. You must be familiar with associated files
2. Modify the position of a sub-assembly without affecting other instances
3. Check for interference
4. Define an exploded state and save it for future use
5. Add textual information

Stages in the Process

1. Manage the product structure.
2. Create flexible sub-assemblies.
3. Analyze the assembly.
4. Create scenes.
5. Create annotations.
6. Generate reports.



Lighting Fixture assembly



Lighting Fixture assembly: Flexible Assembly concept is illustrated in the above example.

Manage the Product Structure

An assembly document contains links to all related CATIA documents and external files such as text and Excel files. The File > Desk command can be used to display and manage the product structure.

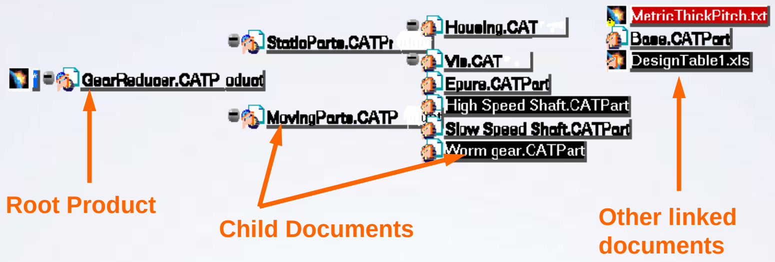
1. Visualize the structure of linked components.
2. Load / Unload individual components.
3. See the links of CATProduct.
4. View the properties of component.
5. Find missing components and re-create links.

The Edit > Links command can be used to manage the product components and the component links.

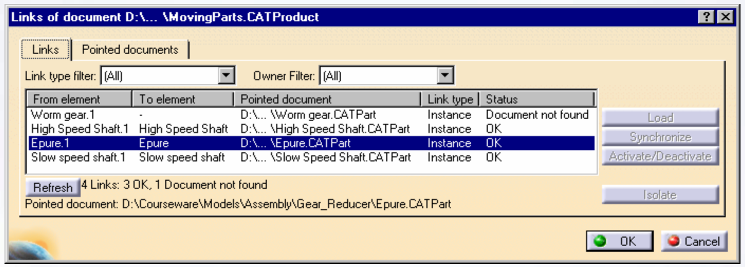
1. Quickly analyze the broken links.
2. Load / Unload individual components.
3. Activate / Deactivate components.
4. Isolate components.
5. Replace components.

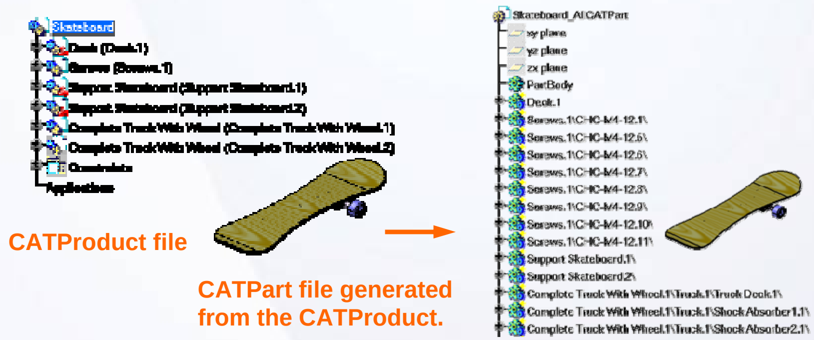
The Generate CATPart from the Product tool is used to create a non-associative CATPart file showing all active assembly nodes. The individual components are replaced by solid features.

Product Structure links viewed using File > Desk command



Product Structure links viewed using Edit > Links command

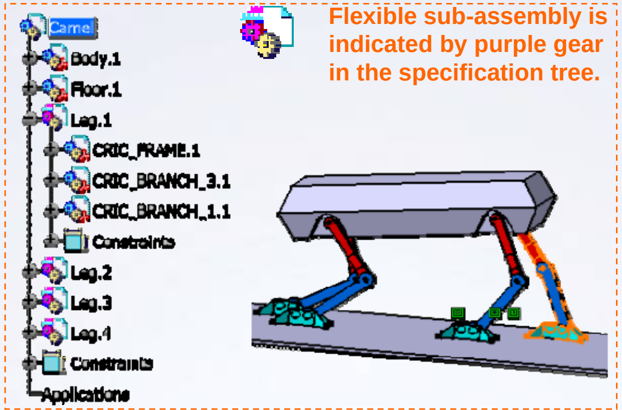




Create Flexible Sub- Assemblies

A flexible sub-assembly is a sub-assembly whose child components can be moved without affecting either the child components of other instances of the same sub-assembly or referenced CATProduct of the sub-assembly. In the example shown, by making each leg sub-assembly flexible, each leg can be have a different configuration.

If a change is made to a flexible sub-assembly and the same change needs to be made on the rigid instances of the same referenced assembly, the Propagate position to reference option must be used.

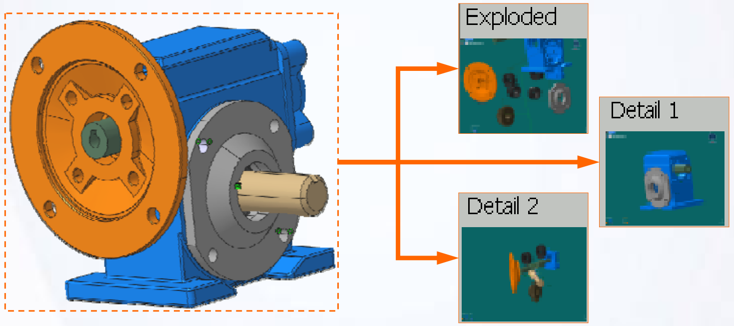


Flexible sub-assembly is indicated by purple gear in the specification tree.

Create Scenes

Scenes enable you to capture assembly configurations without modifying the root product. You can do the following:

1. Change the Hide/Show state of components.
2. Change the color of components.
3. Change the position of components.
4. Change the activation states of representations.
5. Create drafting views based on scenes.



Three different states of gear box assembly are represented.

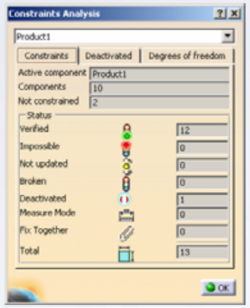
A drawing view is generated from an exploded view.

Analyze the Assembly

The types of Analysis are as follows:

1. Component constraints using Analyze > Constraints.
2. Relationships between components using Analyze Dependencies.
3. Degrees of freedom using Analyze > Degree(s) of freedom.
4. Minimum distance between components, products and groups of documents using Distance and Band Analysis tool.
5. Assembly Sections using the Section plane.
6. Clash, interference, and clearance in the assembly using the Clash and Interference analysis tools.

1



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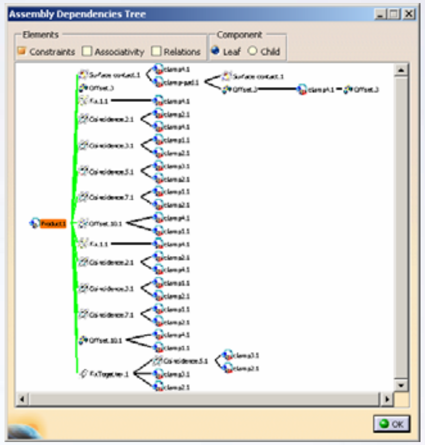

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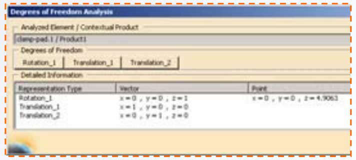

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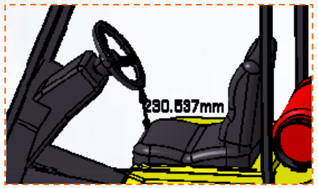

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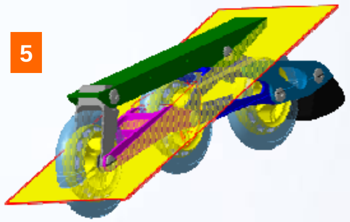

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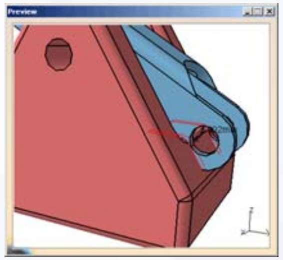
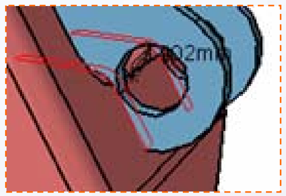




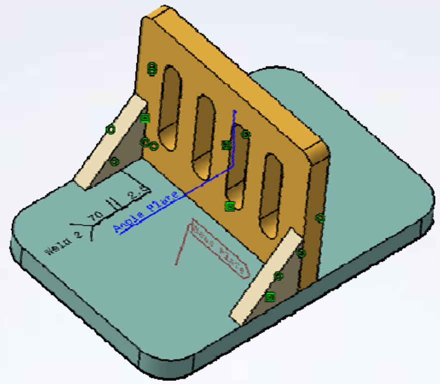
Create Annotations

Annotations are added to assembly documents to provide additional information about a part or product. For example, a brief description of the part, the material used for the part, the finish or the hardness requirements. You can add three types of annotations:

A. Weld Feature: Adds weld symbols and annotations.

B. Text with leader: Adds a brief description of the part.

C. Flag note with leader: Adds links to external documents and/or URLs, such as a link to a presentation or a specification document.

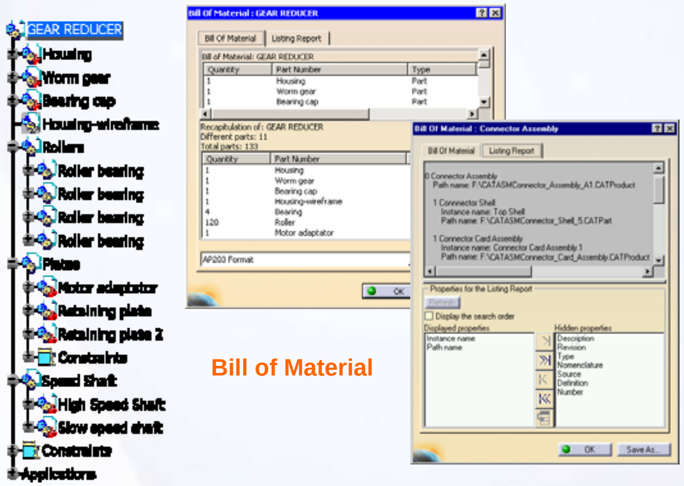


Generate Reports

Two types of reports can be generated:

A. Bill Of Material: It lists all the assembly components as well as information such as quantity, type and description.

B. Listing Report: It lists all the assembly components. They are displayed in a hierarchical format.



Bill of Material

Listing Report

Main Tools (1/2)

Manipulating Product Structure

1. **File > Desk:** Opens a window that displays a tree with a product and child documents.
2. **Edit > Links:** Displays the linked components of the CATProduct / CATPart / CATDrawing.
3. **Generate CATPart from Product:** Creates a CATPart from the CATProduct.

Constraints Toolbar

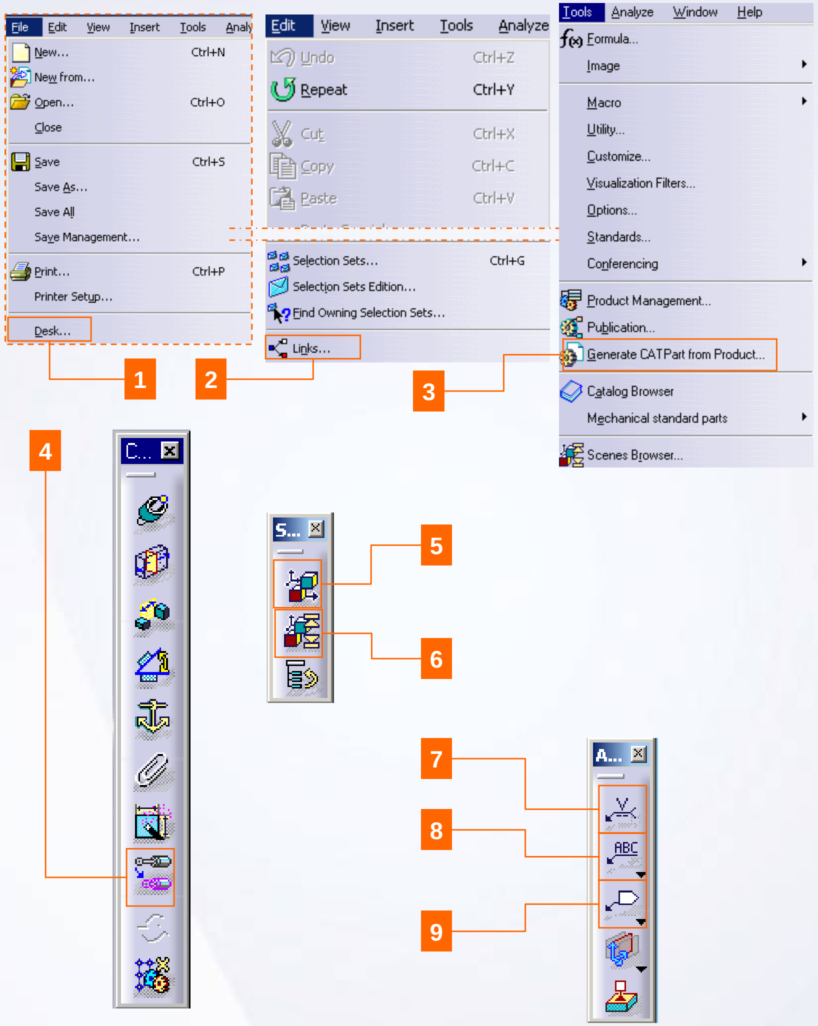
4. **Flexible/Rigid Sub-Assembly:** Toggles the sub-assembly between flexible and rigid states.

Scenes Toolbar

5. **Enhanced Scenes:** Creates a Scene.
6. **Scenes Browser:** Browses existing scenes.

Annotations Toolbar

7. **Weld Feature:** Creates a weld feature.
8. **Text with Leader:** Creates a text with leader.
9. **Flag Note with Leader:** Creates a flag note with leader.



Main Tools (2/2)

Analysis Tools

5 **Analyze > Constraints:** Analyzes the constraints of an active product.

6 **Analyze > Degrees of Freedom:** Analyzes the degrees of freedom of the active product.

7 **Analyze > Dependencies:** Displays relationships between components and constraints.

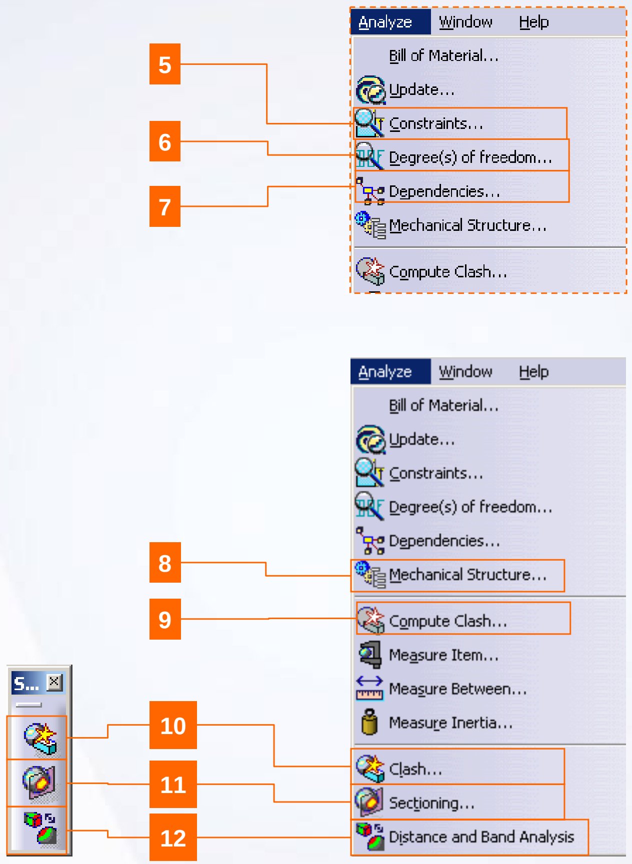
8 **Analyze > Mechanical Structure:** Analyzes the Mechanical Structure of the product.

9 **Compute Clash:** Computes clash/contact between two selected components.

10 **Clash:** Analyzes interference between the products.

11 **Sectioning:** Creates a section using a section plane.

12 **Distance and Band Analysis:** Computes minimum distance between selected components and also performs a distance band analysis.



Exercise: Flexible Sub-Assembly

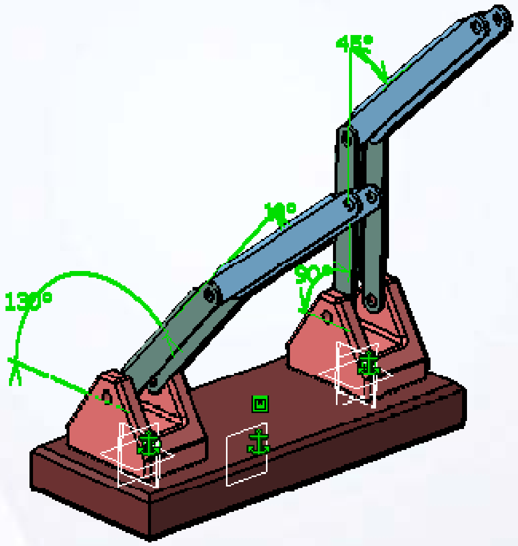
Recap Exercise

20 min

In this exercise, you will create flexible sub-assemblies within the top level assembly.

By the end of this exercise you will be able to:

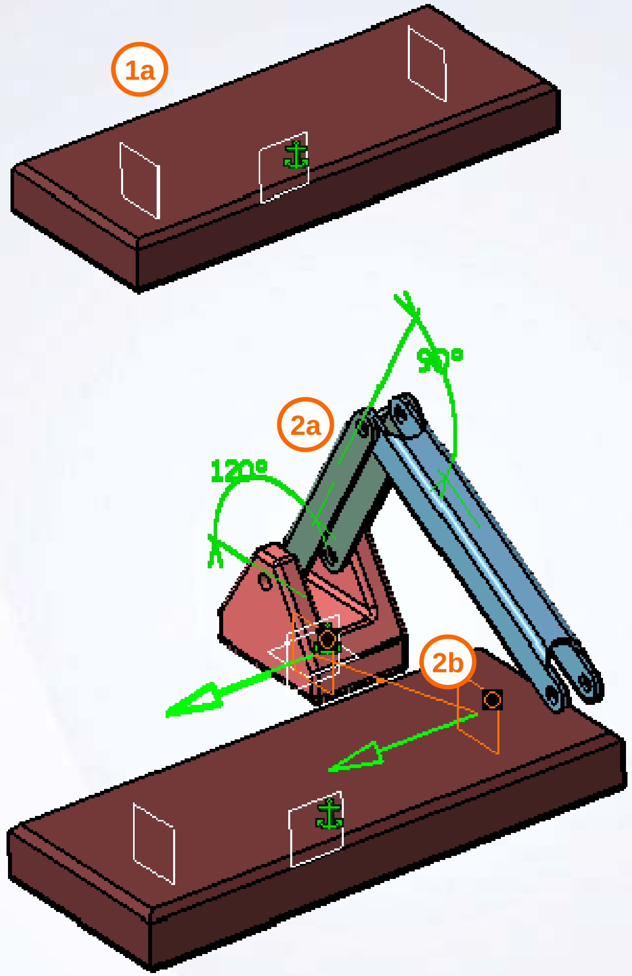
- Create flexible sub-assemblies
- Manipulate flexible sub-assemblies



Do it Yourself (1/11)

1. **Open a product file.**
   - The product file has a base component that is fixed.
     a. Open Arms.CATProduct.

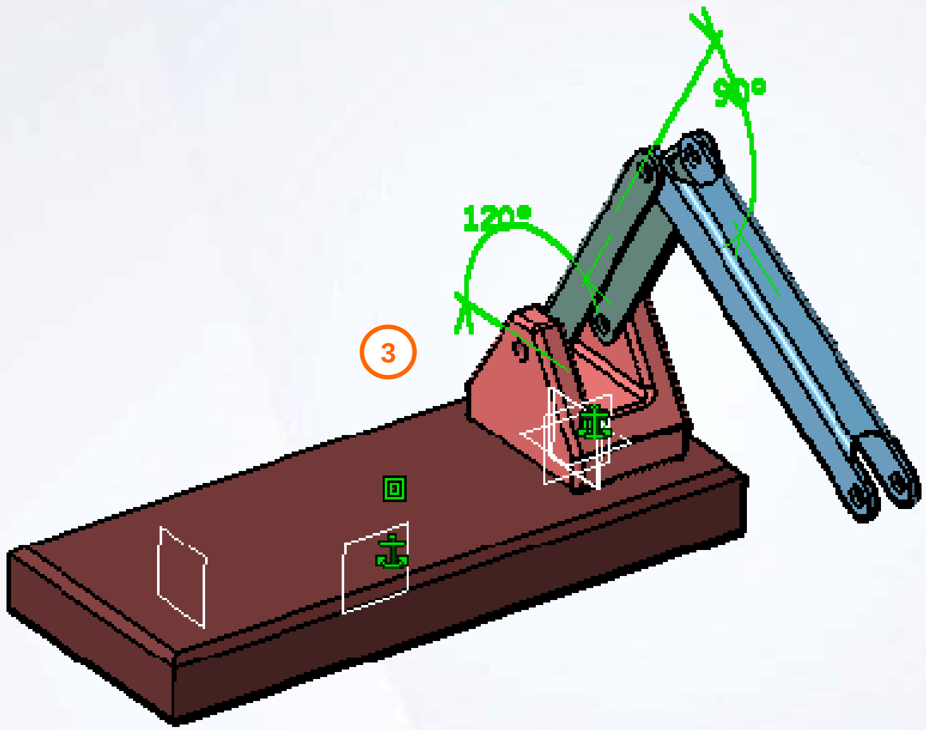
2. **Assemble a sub-assembly.**
   - Apply constraints to assemble the sub-assembly.
     a. Assemble Links.CATProduct.
     b. Apply a coincidence constraint between the reference planes.



Do it Yourself (2/11)

3. Add constraints.

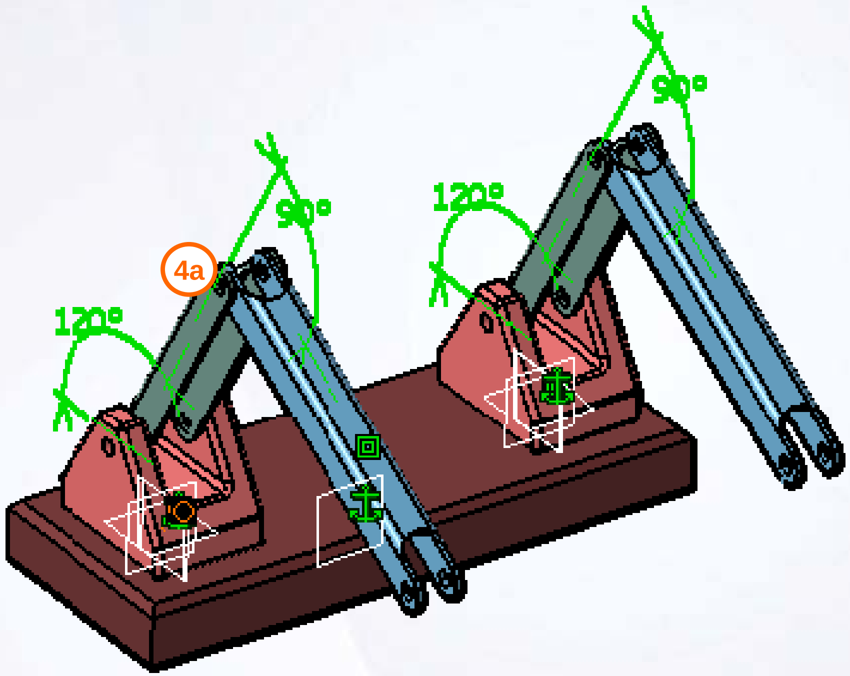
- Add a contact constraint.The updated assembly appears as shown below



Do it Yourself (3/11)

4. Assemble a second instance of a sub-assembly.

- Both sub-assemblies are rigid by default.
  a. Assemble Arms.CATProduct as shown below.

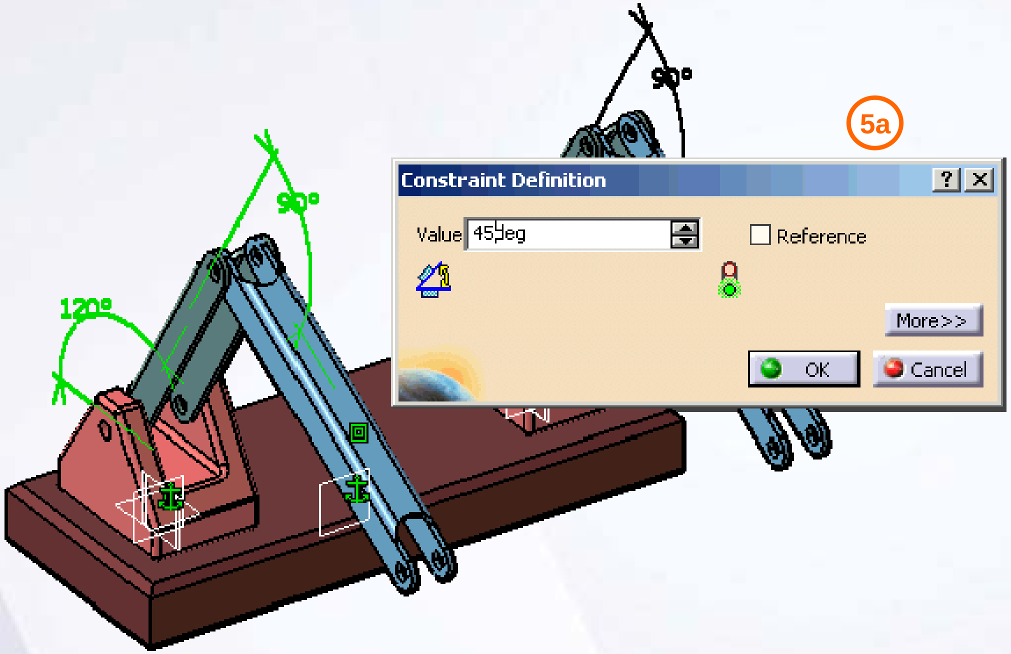


Do it Yourself (4/11)

5. Modify a sub-assembly.

- You will modify a rigid sub-assembly, both instances will be affected.

  a. Modify the 90 value of (Links.1) instance to 45deg.

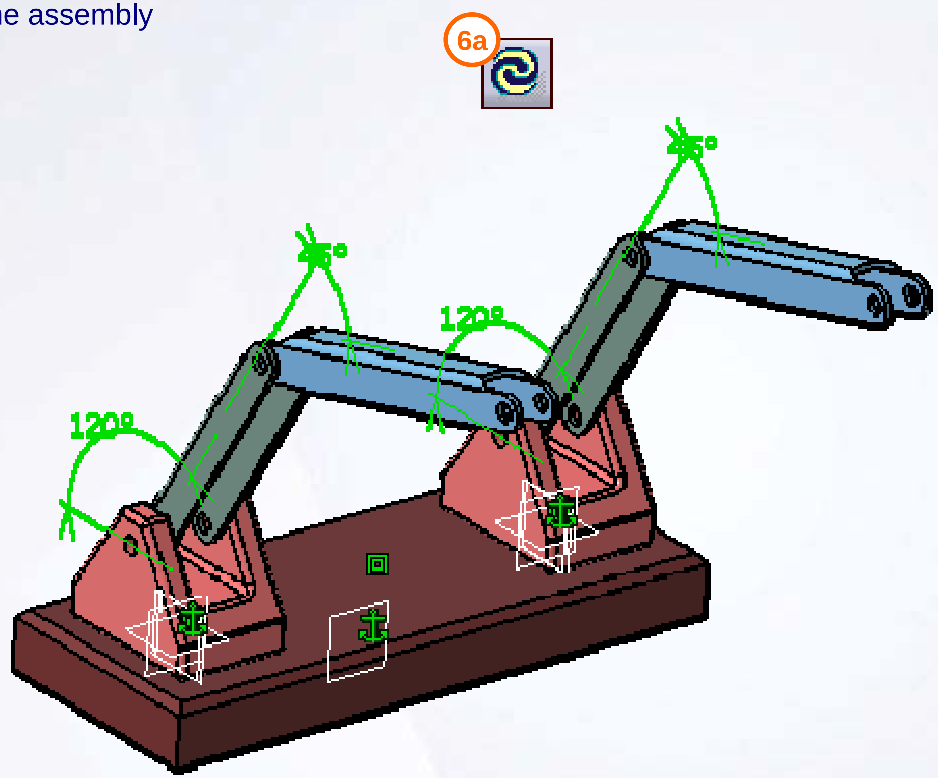


Do it Yourself (5/11)

6. Update the assembly.

- Modifications are shared between rigid sub-assemblies.

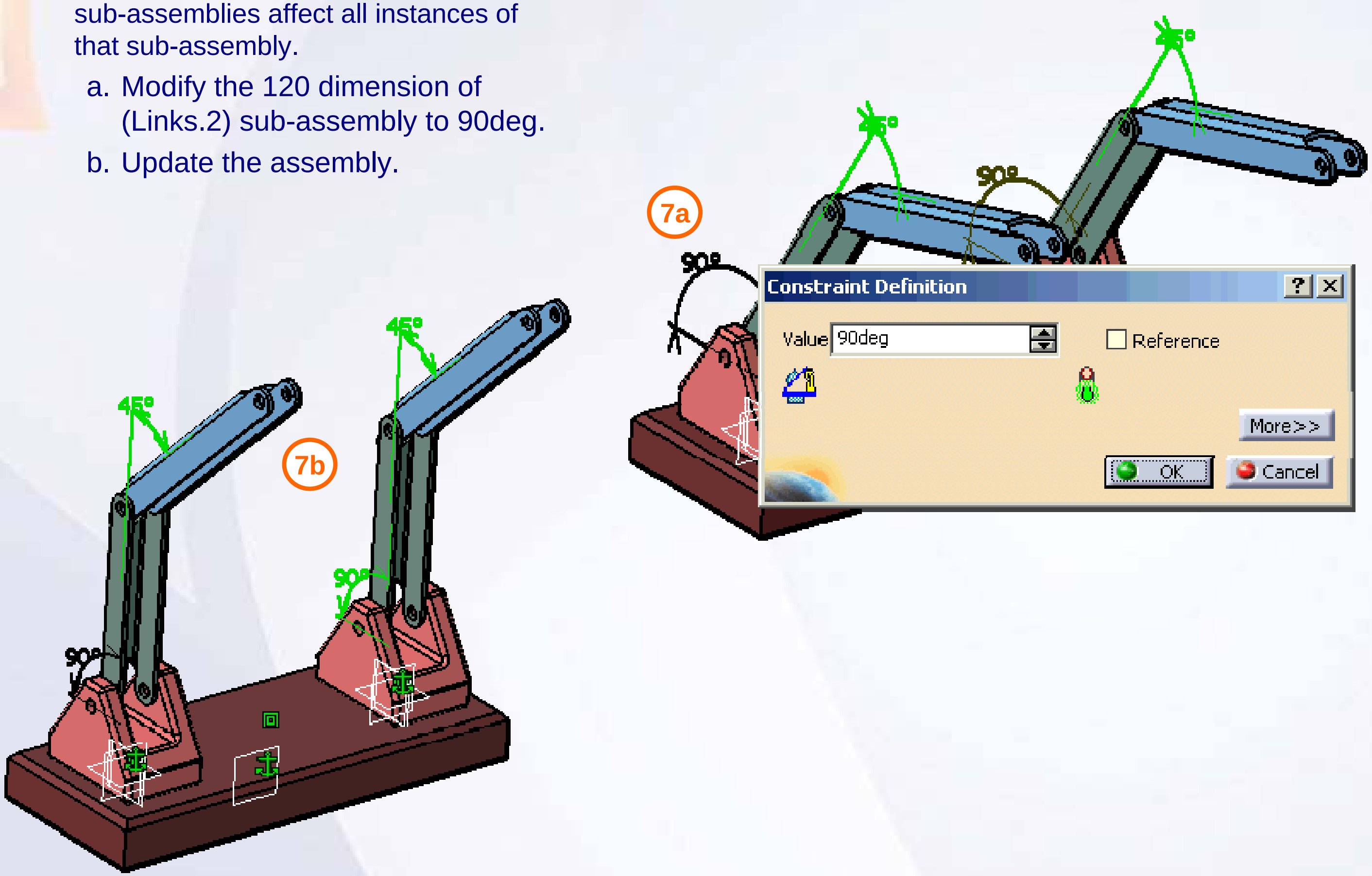
  a. Select the update icon. The assembly updates as shown below.



Do it Yourself (6/11)

7. Modify a rigid sub-assembly.

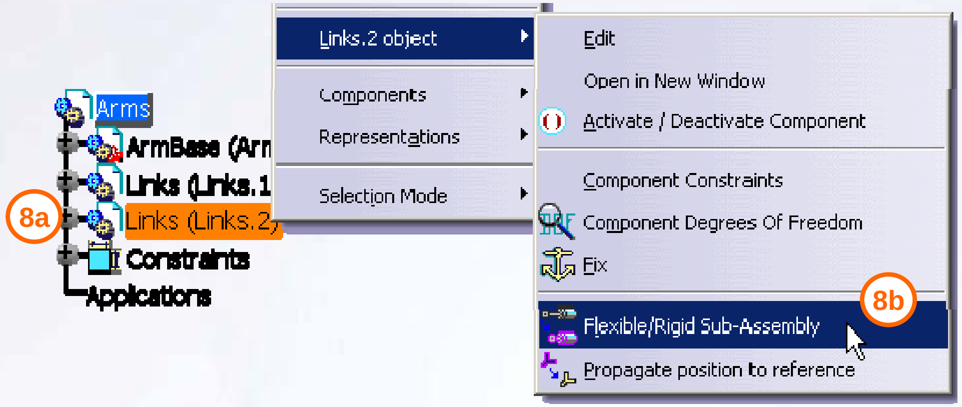
- Modifications to any instance of rigid sub-assemblies affect all instances of that sub-assembly.
  - a. Modify the 120 dimension of (Links.2) sub-assembly to 90deg.
  - b. Update the assembly.

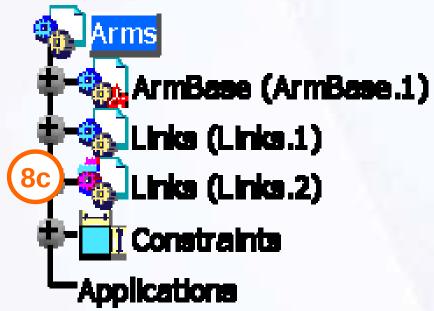


Do it Yourself (7/11)

8. Make a sub-assembly flexible.

- You will make one of the two Links sub-assemblies flexible.
  a. Select Links (Links.2) sub-assembly from the specification tree.
  b. Select the **Flexible/Rigid Sub-Assembly** from the contextual menu.
  c. The sub-assembly is now flexible, as indicated by the symbol in the specification tree.

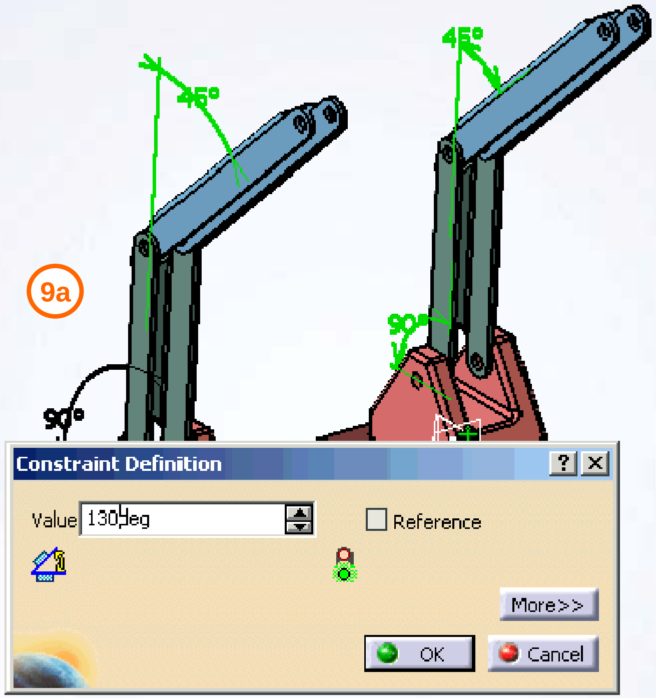


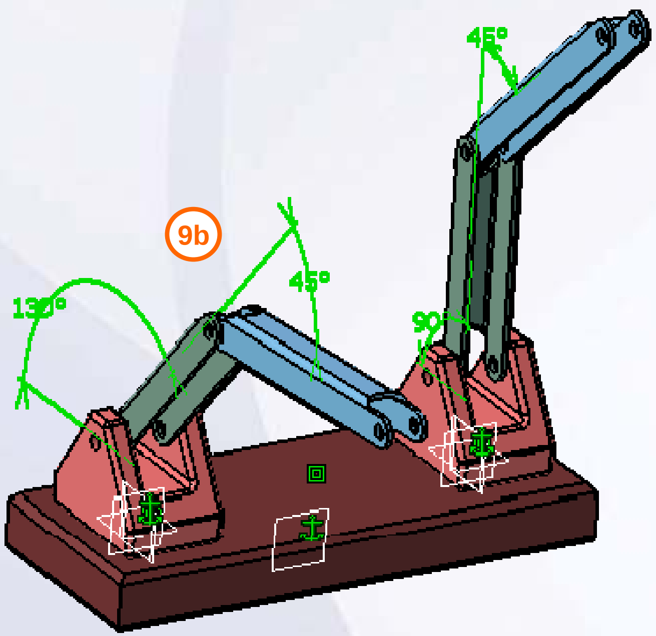


Do it Yourself (8/11)

9. Modify a flexible sub-assembly.

- Changes made to a flexible sub-assembly will not propagate to a rigid instance.
  a. Modify the 90deg dimension of (Links.2) flexible instance to 130deg.
  b. The updated assembly appears as shown below.

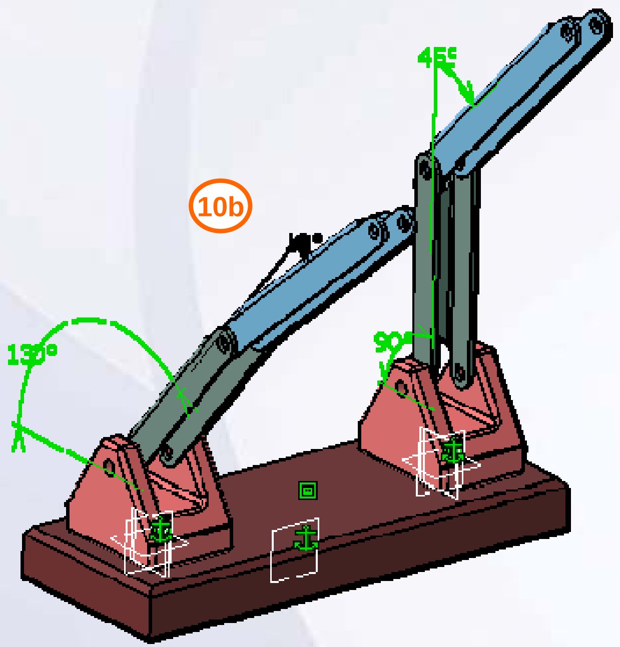


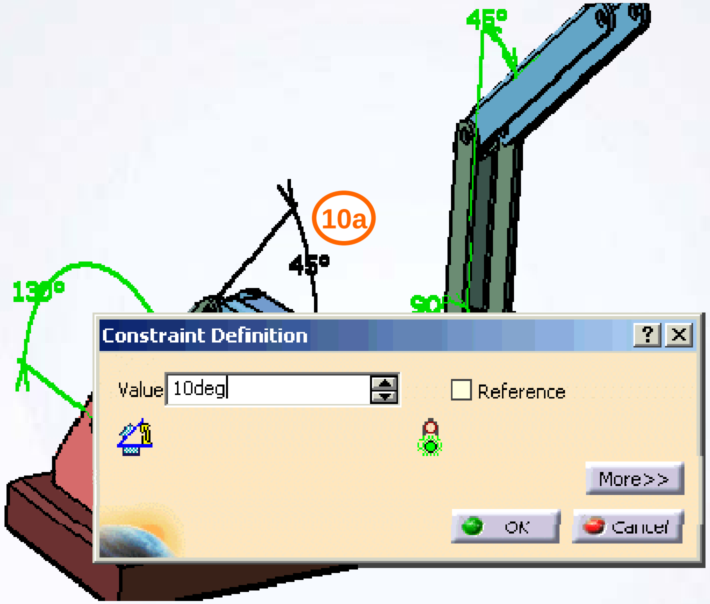


Do it Yourself (9/11)

10. Modify a flexible sub-assembly.

- Changes made to a flexible sub-assembly will not propagate to a rigid instance.
  a. Modify the 45deg dimension of (Links.2) flexible instance to 10deg.
  b. The updated assembly appears as shown below.

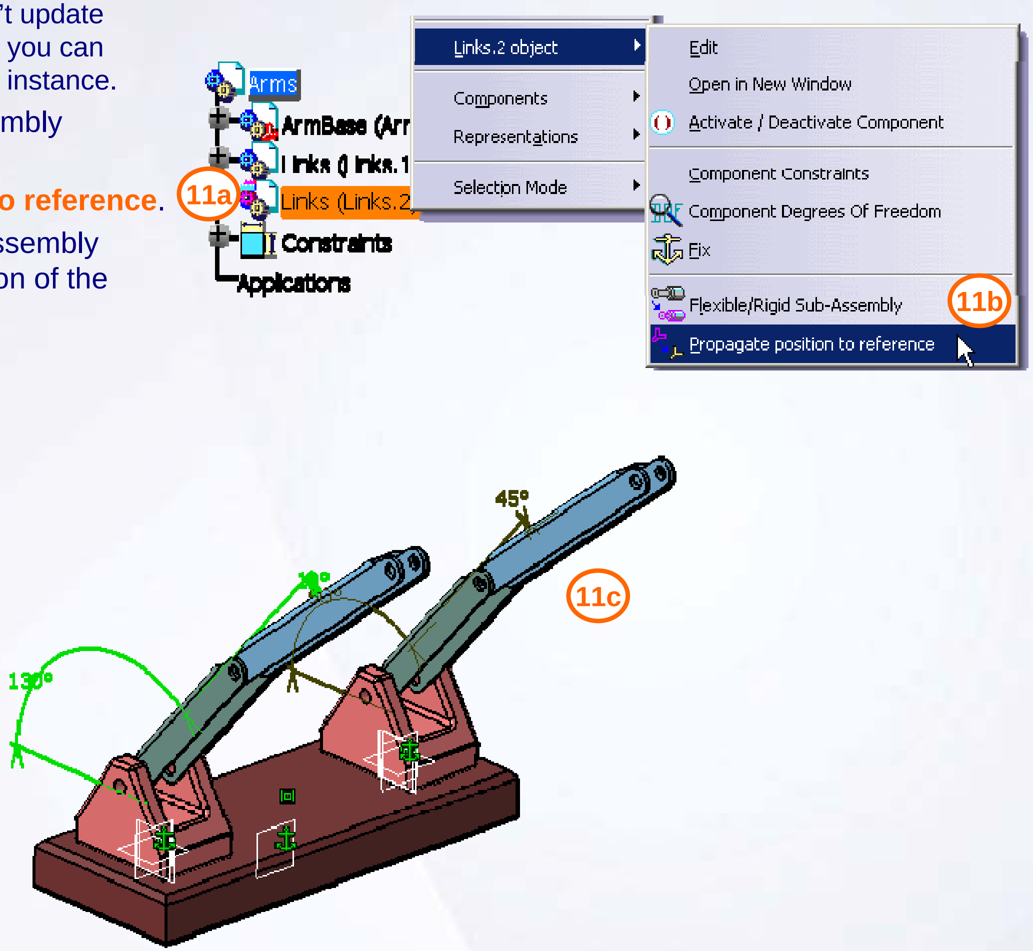




Do it Yourself (10/11)

11. Propagate position to reference.

- Although the rigid instance doesn't update position with the flexible instance, you can propagate the position to the rigid instance.
  a. Select the flexible sub-assembly (Links.2).
  b. Click **Propagate position to reference**.
  c. The rigid (reference) sub-assembly updates to reflect the position of the flexible instance.



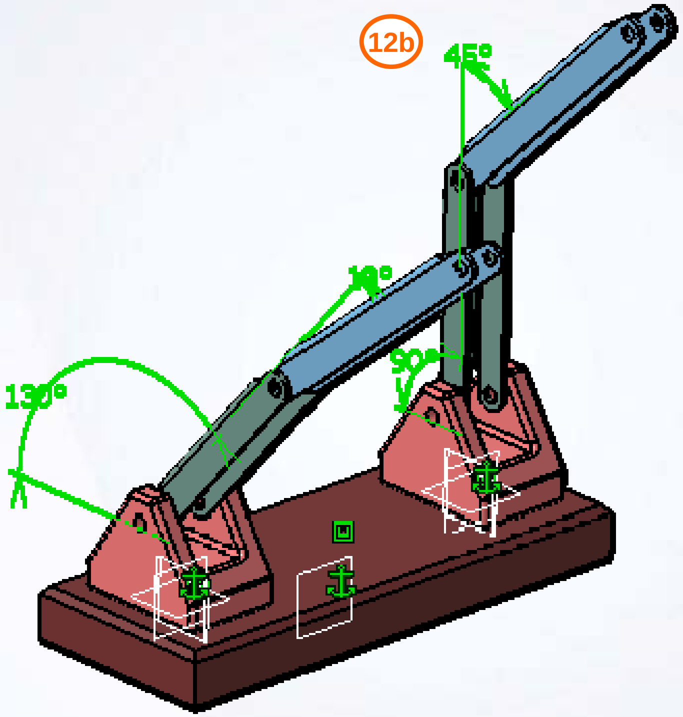
Do it Yourself (11/11)

12. Update the assembly.

- The propagation to position from a flexible sub-assembly to a rigid reference sub-assembly is temporary.
  a. Select the Update icon.
  b. The rigid sub-assembly returns to the reference position.
  c. Save the assembly and close the file.

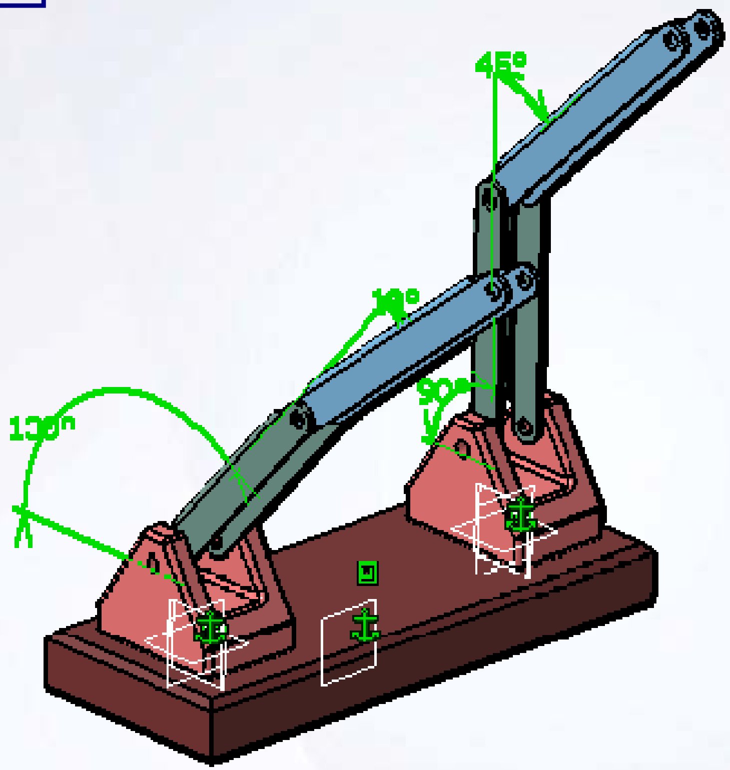
12a

12b



Exercise: Flexible Sub-Assembly Recap

- ✓ **Create flexible sub-assemblies**
- ✓ **Manipulate flexible sub-assemblies**



Exercise: Desk and Flexible Sub-Assembly

Recap Exercise

20 min

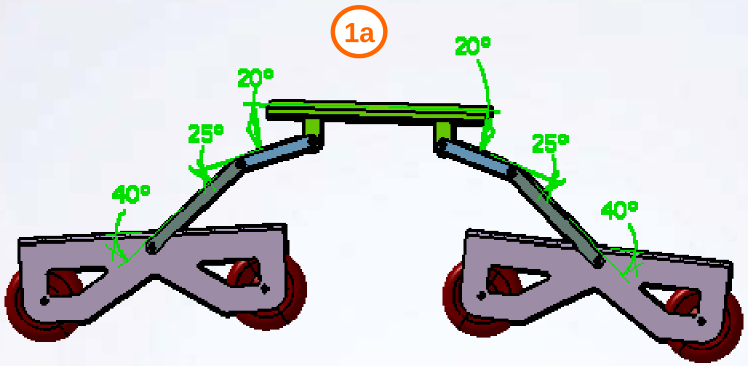
In this exercise, you will use the Desk command to rename a file. You will also create flexible sub-assemblies to achieve positional requirements.

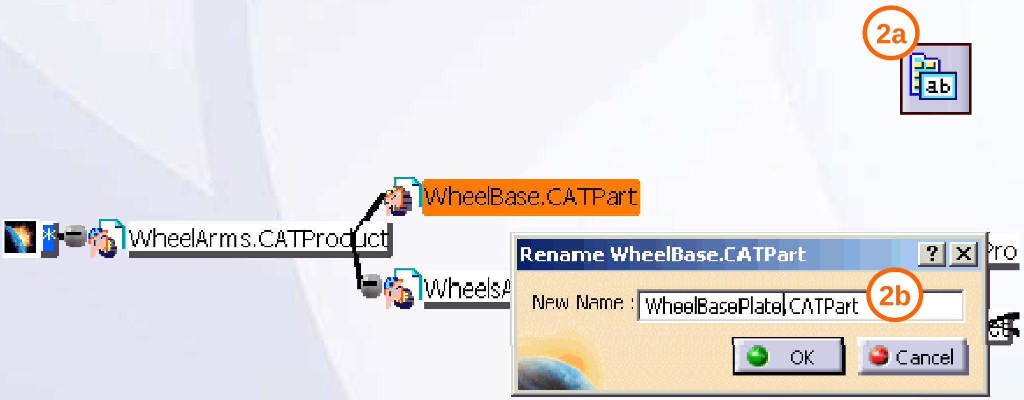
By the end of this exercise you will be able to:

- **Create flexible sub-assemblies**
- **Analyze the mechanical structure of an assembly**

Do it Yourself (1/6)

1. **Open a product file.**
   - This assembly is rigid.
     a. Open WheelArms.CATProduct.
2. **Rename a file using the Desk command.**
   - You will use the Desk command to rename a part file.
     a. Select the rename icon.
     b. Use the Desk command to rename WheelBase.CATPart part to WheelBasePlate.CATPart

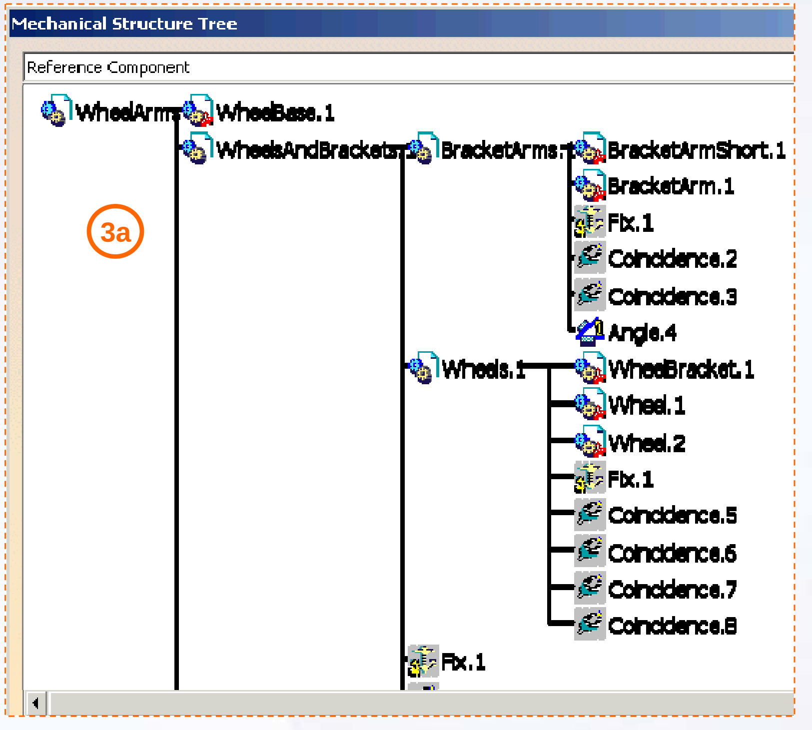




Do it Yourself (2/6)

3. Analyze the mechanical structure.

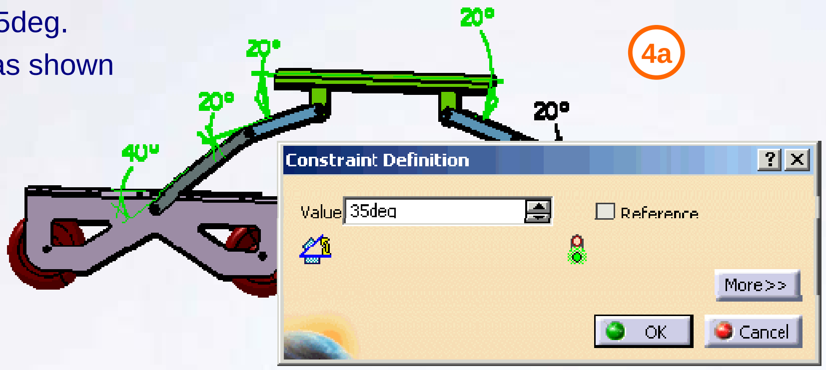
- The mechanical structure that you will analyze is for a rigid assembly.
  a. Click **Analyze > Mechanical Structure.** The mechanical structure is reported in the Mechanical Structure Tree.

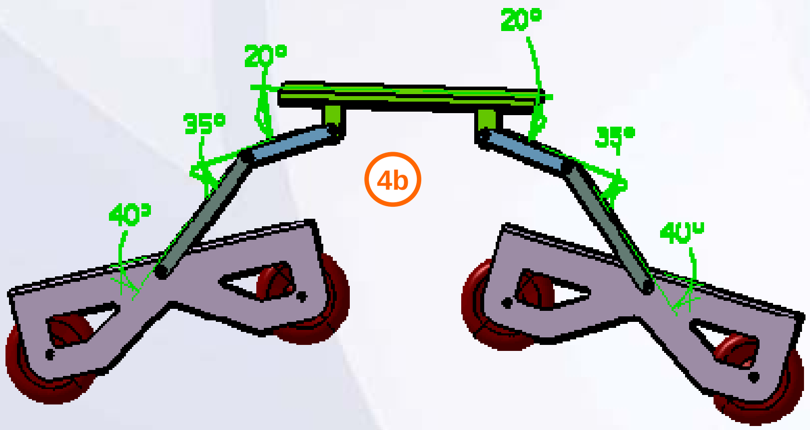


Do it Yourself (3/6)

4. Make changes to a sub-assembly.

- By default, assemblies are rigid.
  a. Modify the 20deg dimension to 35deg.
  b. The updated assembly appears as shown below.



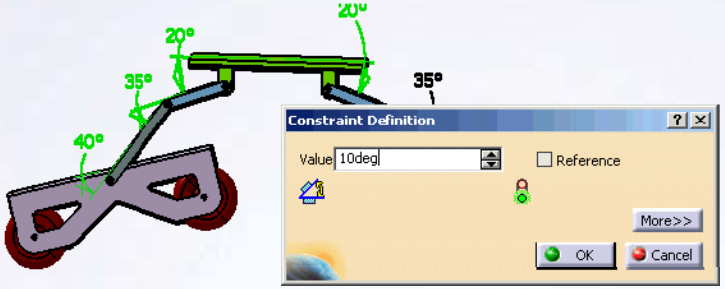


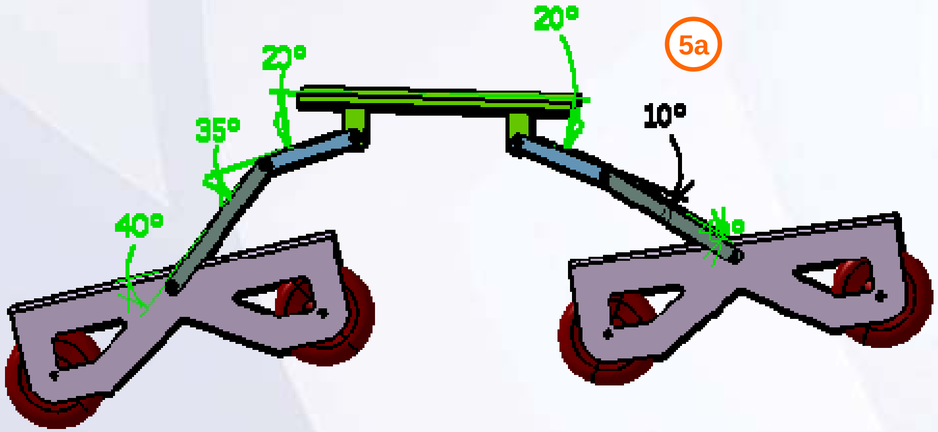
Do it Yourself (4/6)

5. Make a sub-assembly flexible.

- Think about the hierarchy of the sub-assemblies.

  Make the appropriate sub-assembly flexible so that the 35deg dimension can be changed to 10deg without affecting the other instance of the sub-assembly.



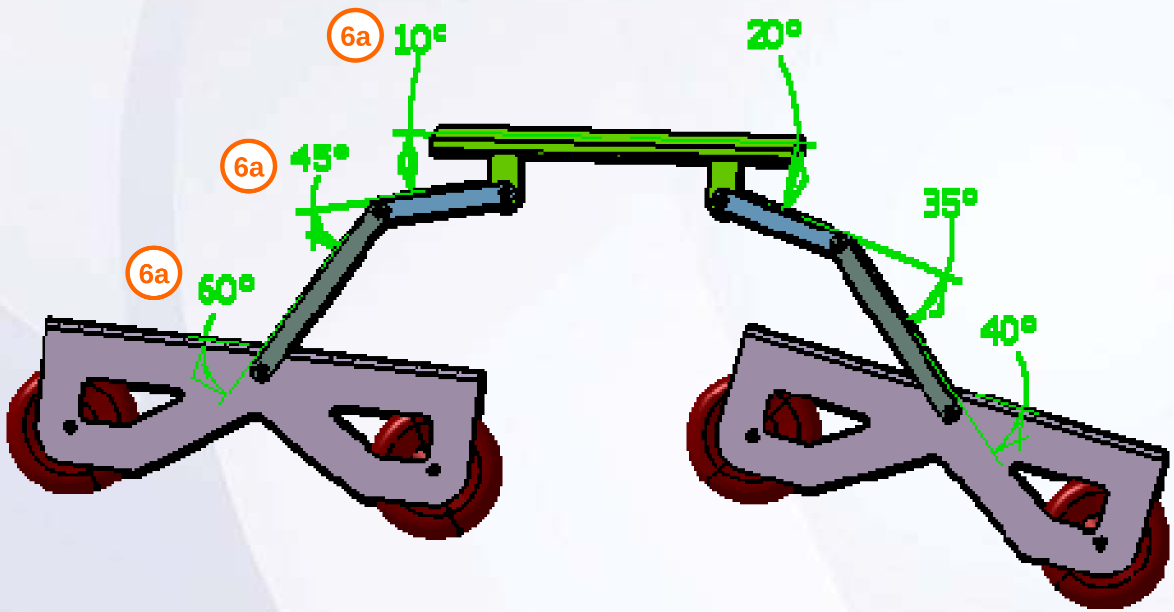


Do it Yourself (5/6)

6. Make a sub-assembly flexible.

- Think about the hierarchy of the sub-assemblies.

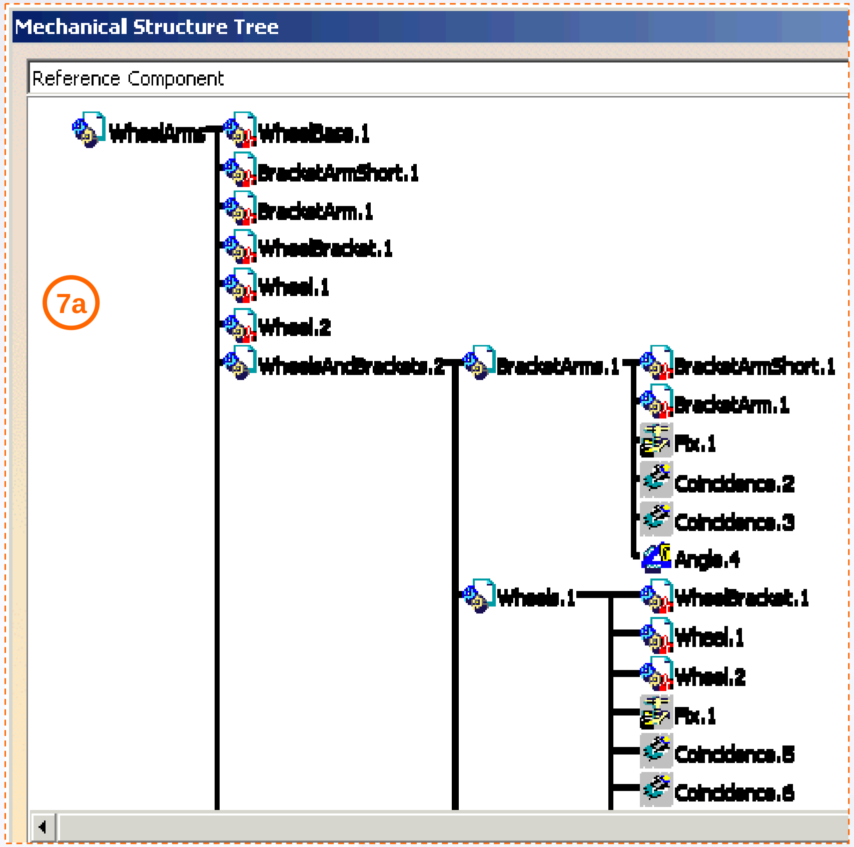
  a. Make the appropriate sub-assemblies flexible so that the dimensions can be changed to create the positions shown below.



Do it Yourself (6/6)

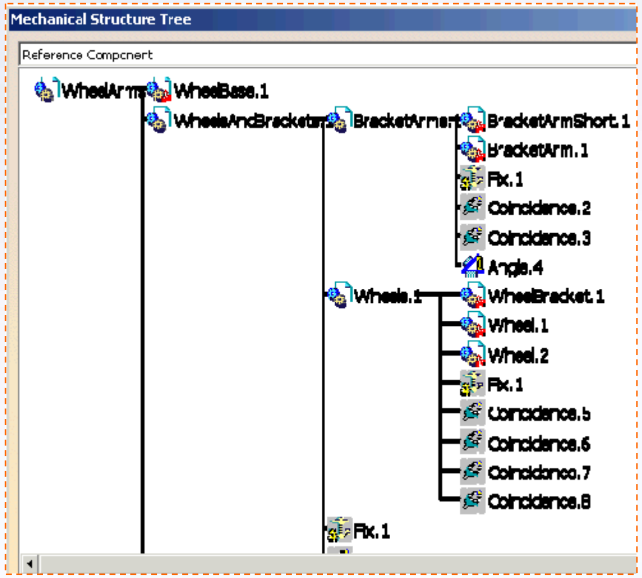
7. Analyze the mechanical structure.

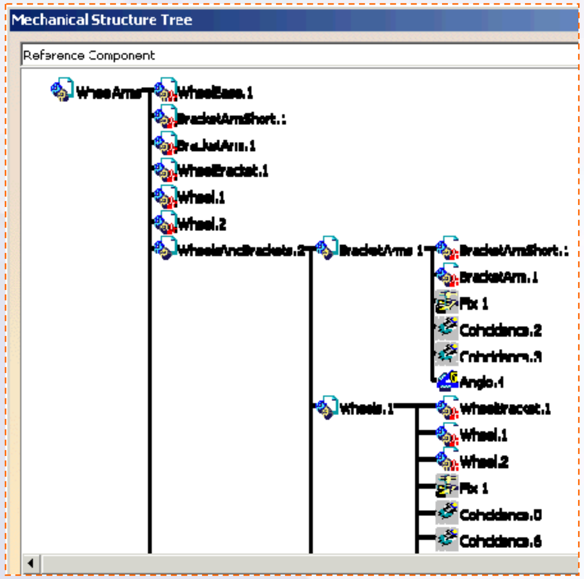
- The mechanical structure that you will analyze is for a flexible assembly. Notice how different it is to the mechanical analysis reported when the assembly was rigid.
  a. The mechanical structure is reported in the Mechanical Structure Tree.



Exercise: Desk and Flexible Sub-Assembly Recap

- ✓ Create flexible sub-assemblies
- ✓ Analyze the mechanical structure of an assembly





Case Study: Assembly Management

Recap Exercise

40 min

In this exercise, you will create the case study model. Recall the design intent of this model:

- ✓ You must be familiar with associated files
- ✓ Modify the position of a sub-assembly without affecting other instances
- ✓ Check for interference
- ✓ Define an exploded state and save it for future use
- ✓ Add textual information

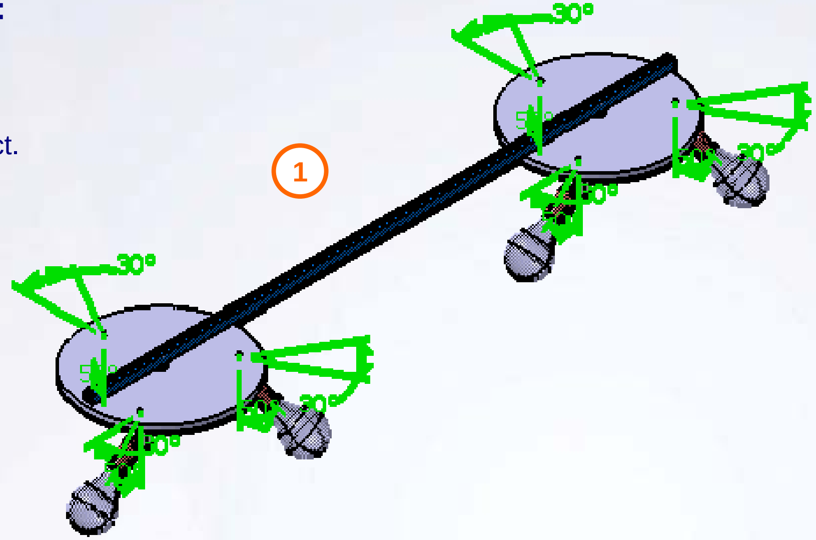
Using the techniques you have learned in this and previous lessons, create the model without detailed instructions.

Do It Yourself: Lights (1/8)

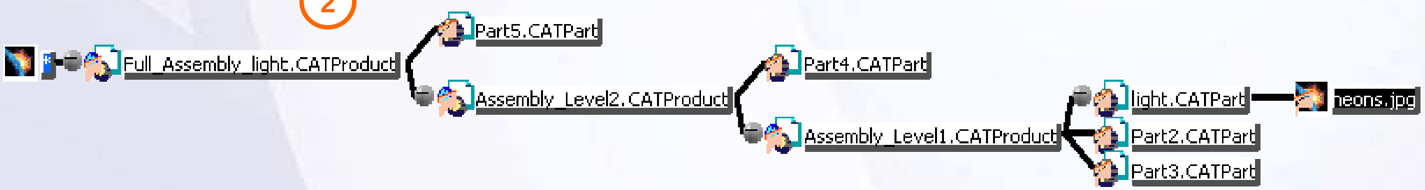
You must complete the following tasks:

1. **Open existing product file.**
   - Open Full_Assembly_light.CATProduct.
2. **View product links.**
   - Use the Desk command to view all associated files

1



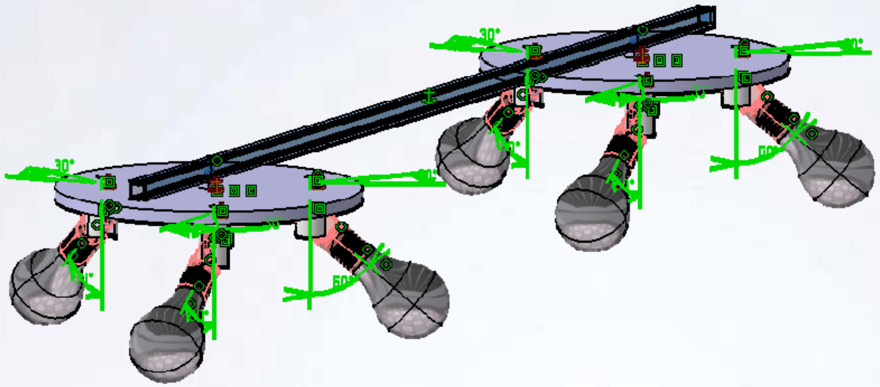
2

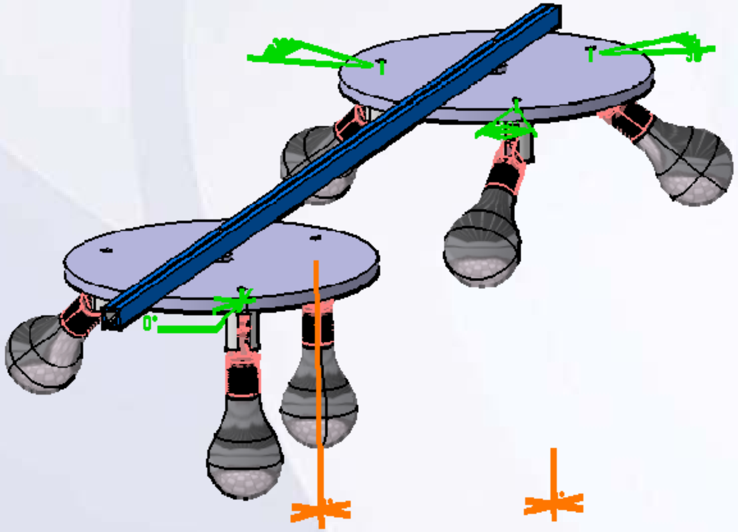


Do It Yourself: Lights (2/8)

You must complete the following tasks (continued):

3. **Define a flexible assembly.**
   - Make the appropriate sub-assemblies flexible.
   - You must achieve the configuration shown below.

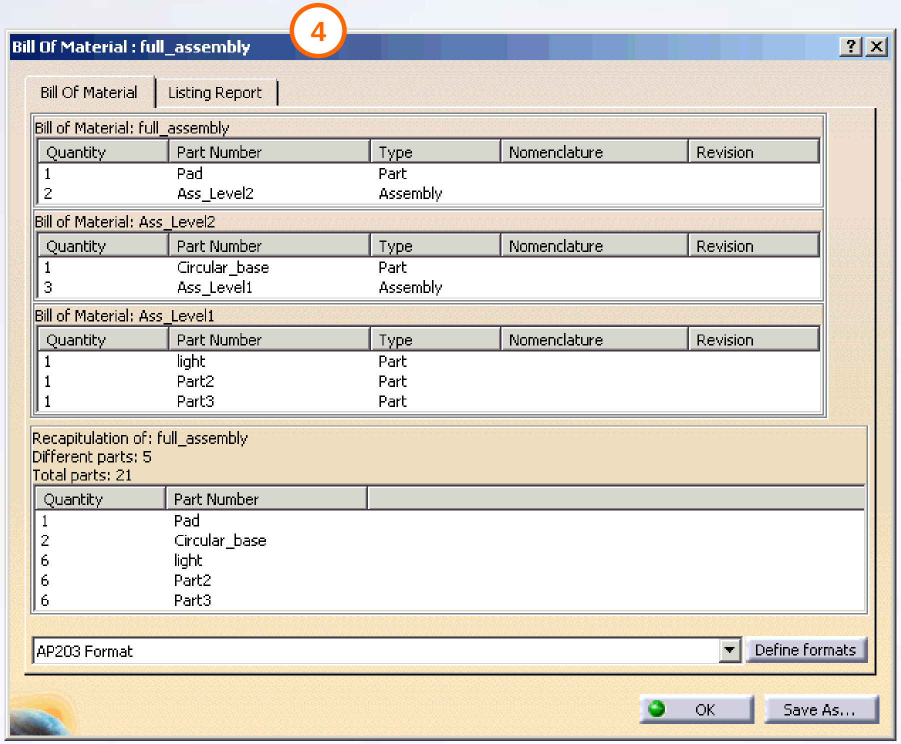


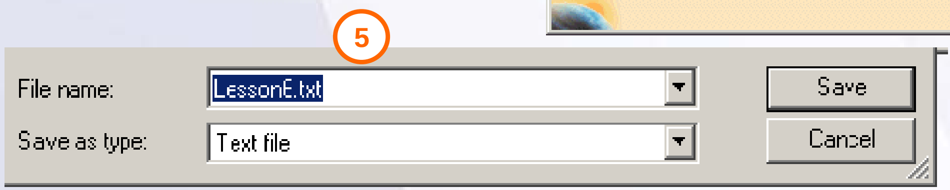


Do It Yourself: Lights (3/8)

You must complete the following tasks (continued):

4. **Analyze for Bill of Material (BOM).**
   - Analyze BOM.
5. **Save the analysis.**
   - Save the analysis as a *.txt file.

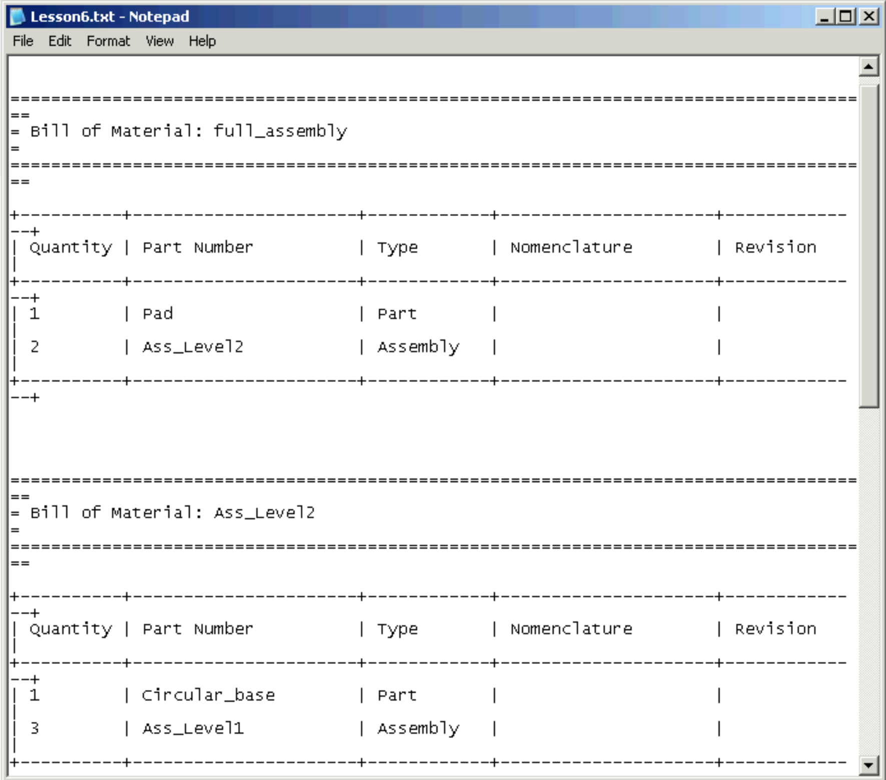




Do It Yourself: Lights (4/8)

You must complete the following tasks (continued):

6. **View the saved text file.**
   - Open Lesson6.txt.



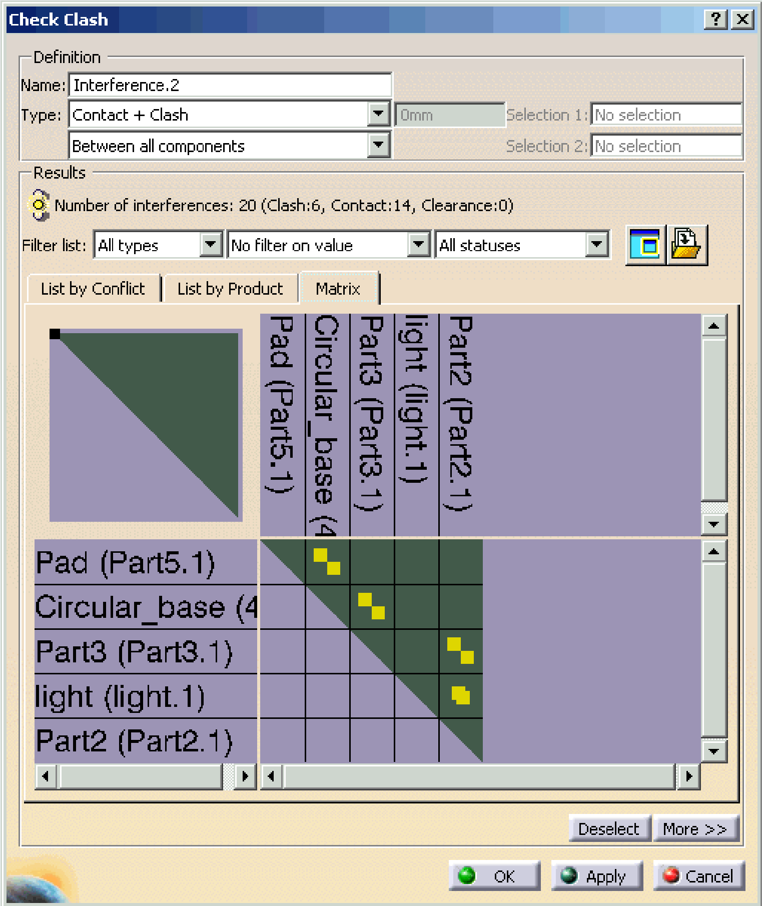

```
==
= Bill of Material: full_assembly
=
==

+----------+--------------------+-----------+-------------------+-----------
--+
| Quantity | Part Number        | Type      | Nomenclature      | Revision
|
+----------+--------------------+-----------+-------------------+-----------
--+
| 1        | Pad                | Part      |                   |
|
| 2        | Ass_Level2         | Assembly  |                   |
|
+----------+--------------------+-----------+-------------------+-----------
--+

==
= Bill of Material: Ass_Level2
=
==

+----------+--------------------+-----------+-------------------+-----------
--+
| Quantity | Part Number        | Type      | Nomenclature      | Revision
|
+----------+--------------------+-----------+-------------------+-----------
--+
| 1        | Circular_base      | Part      |                   |
|
| 3        | Ass_Level1         | Assembly  |                   |
|
+----------+--------------------+-----------+-------------------+-----------
```

Do It Yourself: Lights (5/8)

You must complete the following tasks (continued):

7. Analyze for interference.

- Analyze for Contact and Clash between all components.

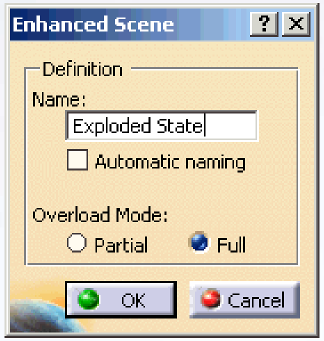


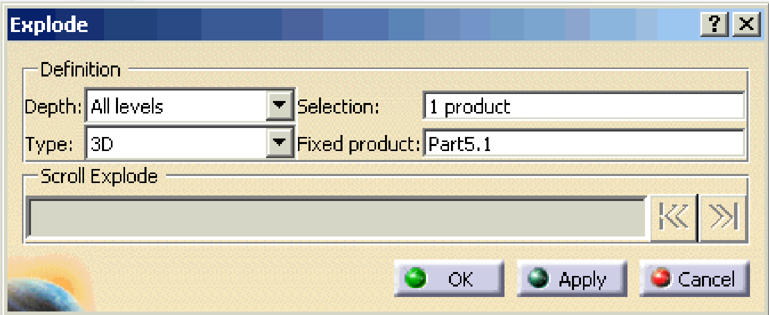
Do It Yourself: Lights (6/8)

You must complete the following tasks (continued):

8. Create a scene.

- Create a Full overload mode scene called Exploded State.
- Define Part5 to be the Fixed product



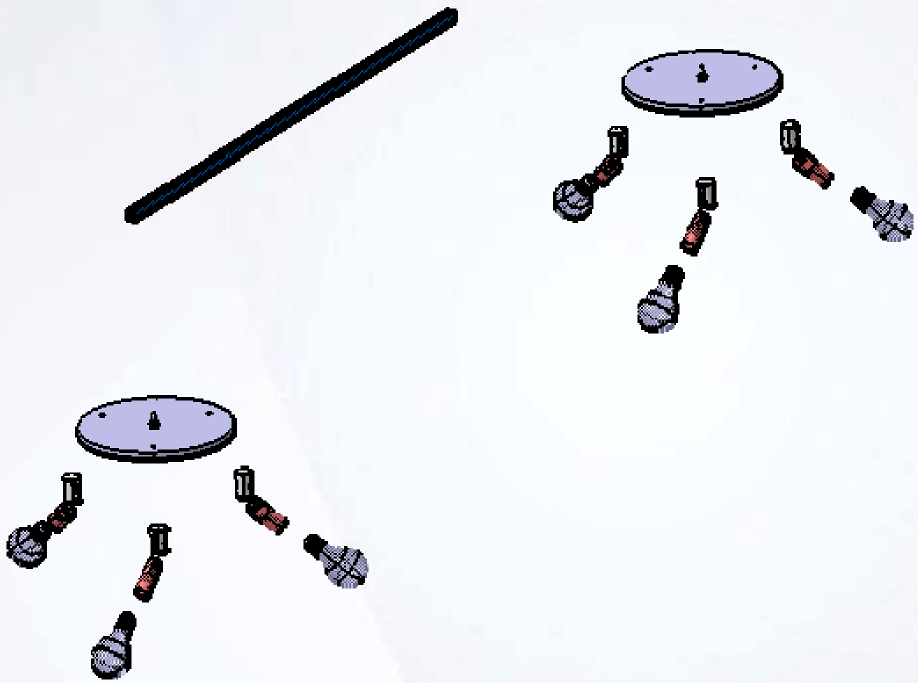


Do It Yourself: Lights (7/8)

You must complete the following tasks (continued):

9. Apply a scene.

- Apply the Exploded State scene to the assembly.

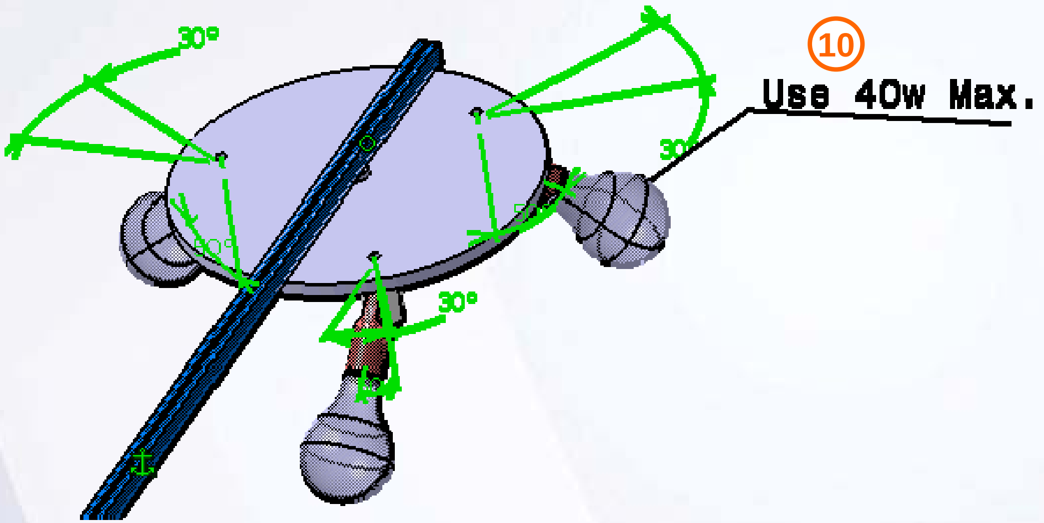


Do It Yourself: Lights (8/8)

You must complete the following tasks (continued):

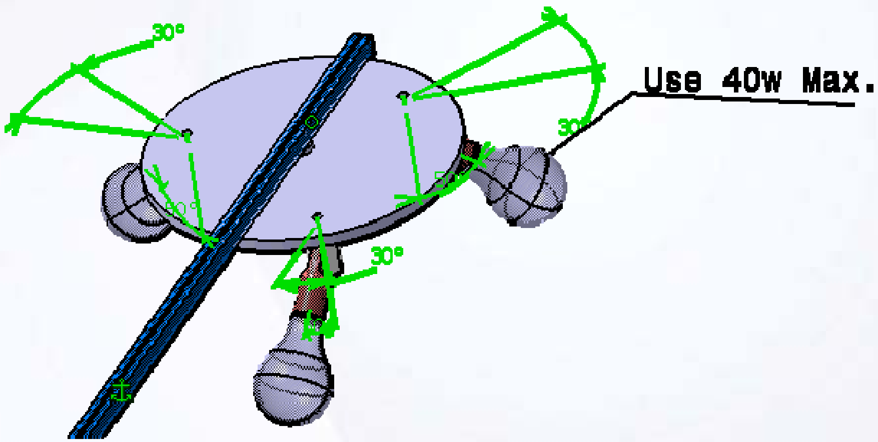
10. Add annotations.

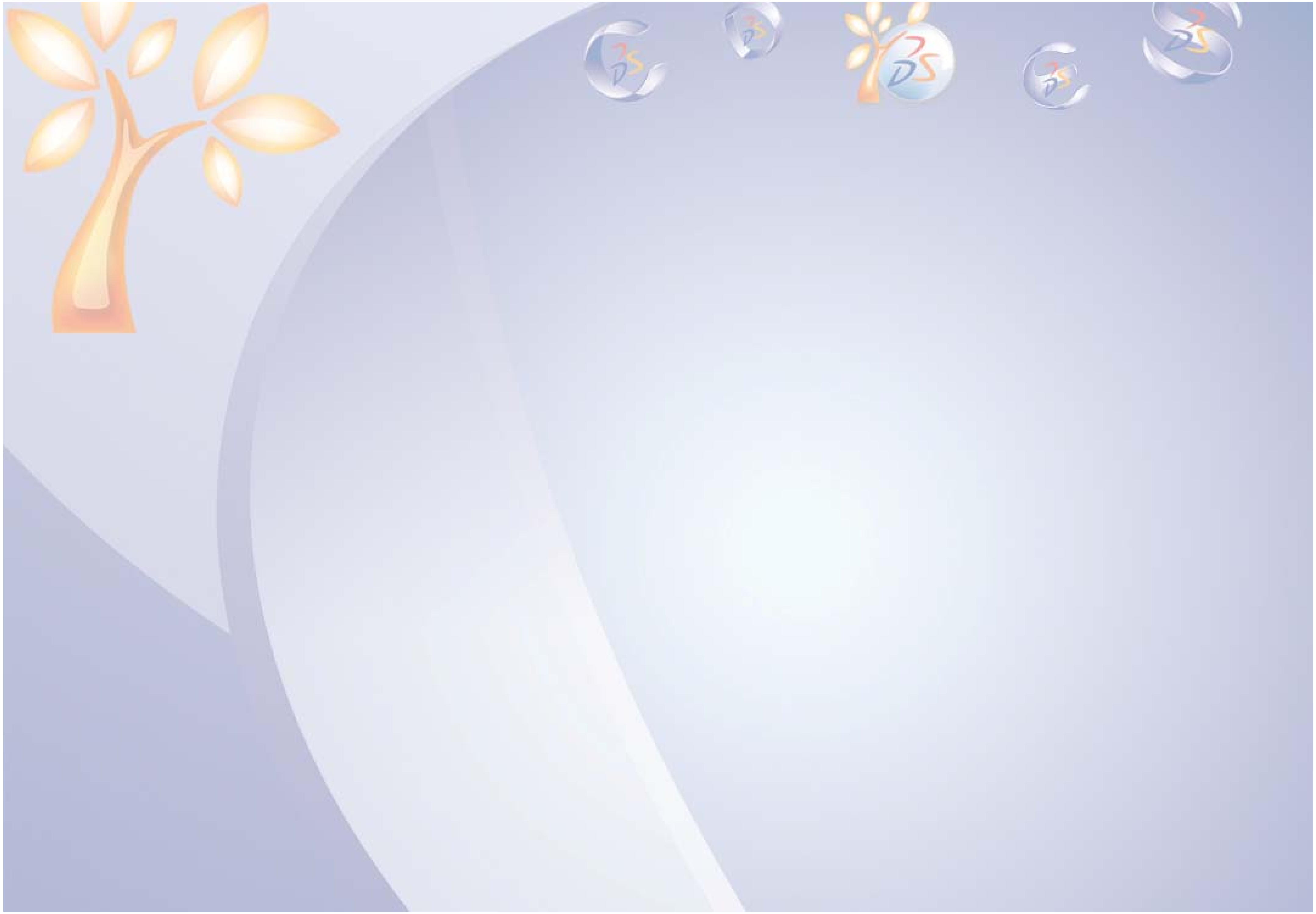
- Add the text shown below.
- The font size of the text must be 7.00mm and it should always be parallel to the screen.



Case Study: Assembly Design Recap

- ✓ View associated files
- ✓ Modify the position of a sub-assembly without affecting other instances
- ✓ Check for interference
- ✓ Define an exploded state and save it for future use
- ✓ Add textual information



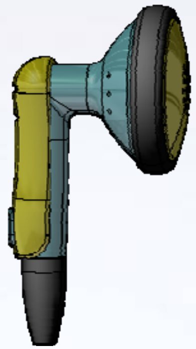


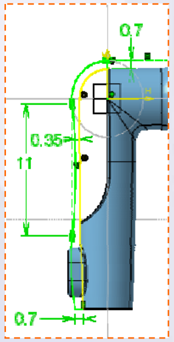
Contextual Design 7

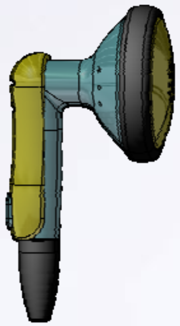
Learning Objectives

Upon completion of this lesson you will be able to:

- ✓ Clarify the display
- ✓ Create Contextual Parts
- ✓ Create Assembly-Level Features
- ✓ Manipulate the Contextual Components
- ✓ Save the Contextual Models







8 Hours

Case Study

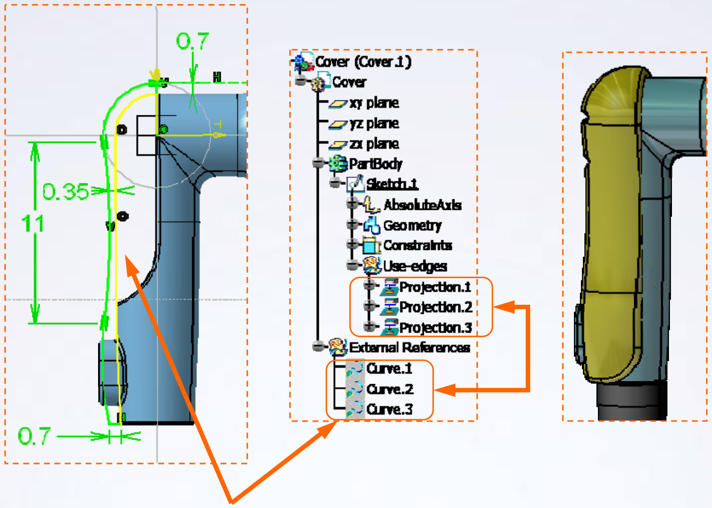
The case study for this lesson is the completion of an earphone model.

Design Intent

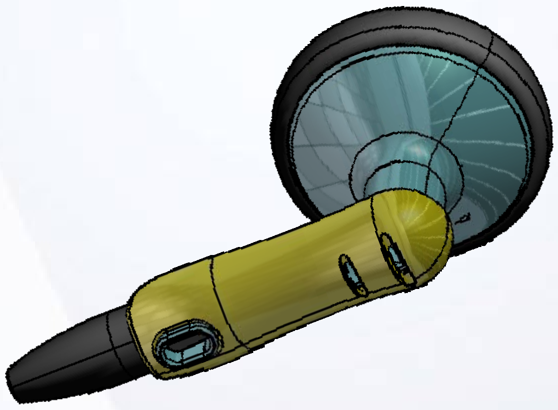
1. Contextual links must be used to ensure that the ear phone cover references the Housing.
2. Additional components may be added to this assembly depending on the model. To ensure that the oval cut intersects all components the cut must be created at the assembly level.
3. The assembly must be saved to another directory.
4. Using the Send To Directory option you can be sure that all files associated with the assembly are copied to the required directory.

Stages in the Process

1. Clarify the display.
2. Create contextual parts.
3. Create assembly features.
4. Manipulate the contextual components.
5. Save the model.



Using Projections from Housing part, you will design the sketch profile for the Ear Phone cover.



Clarify the Display

CATIA performance can be improved for large assemblies while panning, zooming, updating and saving. The following tools are used:

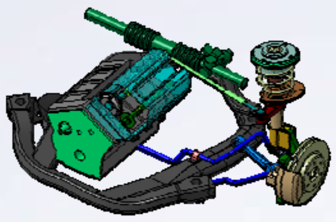
1. Visualization mode: In this mode only a light CGR representation of the model is loaded.

2. Hide: You can hide components to clarify the display and see only desired components.

3. Deactivate representations: Deactivating representations improves performance by hiding the components and excluding them from Mass Property analysis.

4. Deactivate components: It will remove the component from show and no show space, bill of Material.

5. Selective load: Used to load the assembly up to a required depth and manage progressive loading of assemblies.

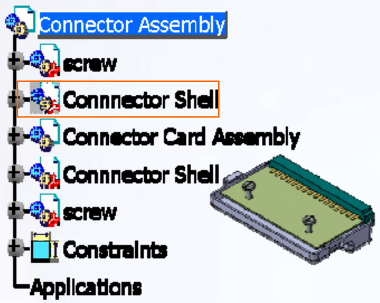


Powertrain Assembly

1

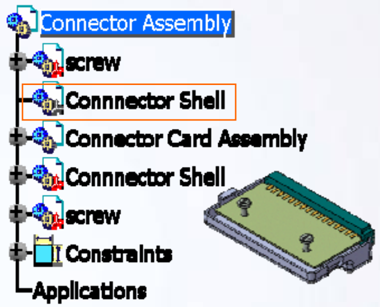
Visualization Mode

2



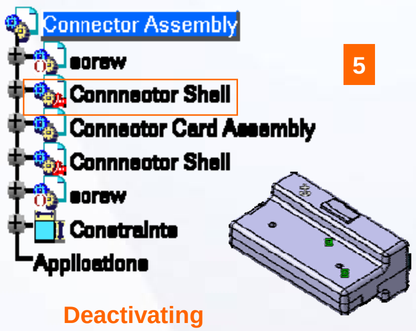
Hiding Components

3



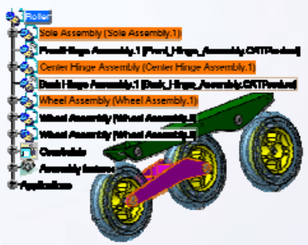
Deactivating Representations

4



Deactivating Components

5



Selective Load

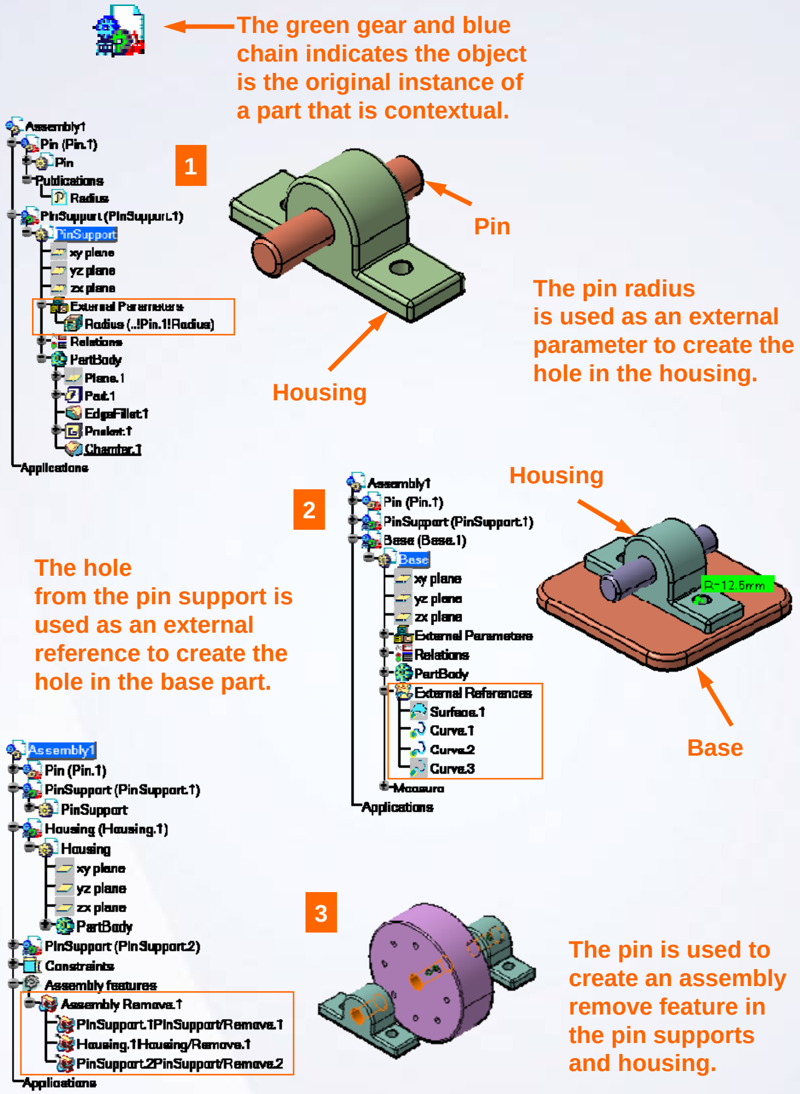
Create Contextual Parts

Contextual parts are parts that have their geometry driven by another component. There are various ways to create contextual parts:

1. Using External Parameters: Contextual links are created when the part uses a reference of parameters defined in another part.
2. Using External References: Contextual links are created when the part refers to geometrical elements from another part.
3. Using Assembly Features: Contextual links are created when there are Assembly features (Assembly Split, Remove, Hole) in a part.

The benefits of using Design in Context:

1. Reuses existing geometry.
2. Reuses parameters.
3. The contextual part is automatically updated when the geometry of the referenced part changes.



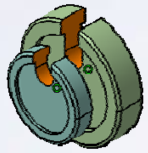
Create Assembly-Level Features

Assembly features are features that are applied not only to a single part (from within the part design workbench) but to a set of several parts of an assembly.

The following Assembly features can be created:

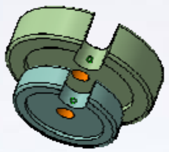
1. Split
2. Hole
3. Pocket
4. Add
5. Remove

1



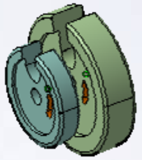
Split: The two parts are split by the surface.

2



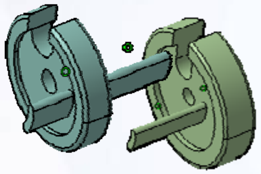
Hole: The hole feature affects both the parts.

3



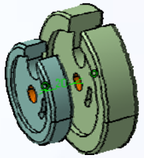
Pocket: The elliptical pocket feature affects both the parts.

4



Add: The elliptical shaped body is added to the two parts.

5

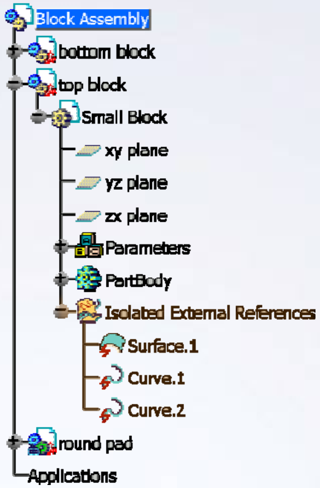


Remove: The central hole is the result of removing a shaft feature from the two bodies.

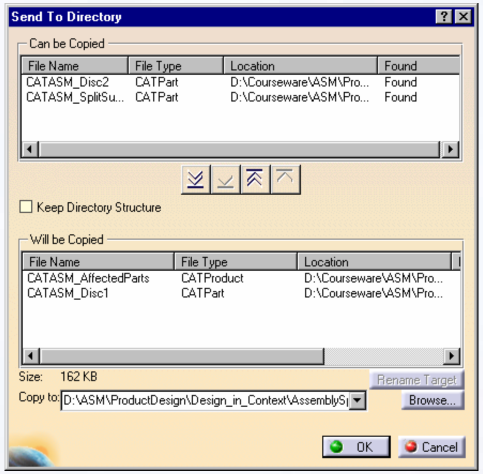
Manipulate the Contextual Components

Various operations can be done as follows:

1. Isolate: The contextual links are broken. You can choose to isolate individual or all elements in a contextual part.
2. Delete: If the original instance of a driven part is deleted a new original instance should be established. When a component that drives a contextual part is deleted, the option to delete the contextual components that are driven by the component is available.
3. Save: After saving a driving CATPart with a new filename, the driven CATParts and the parent CATProduct must be saved because of their reference to the CATPart. After saving a contextual CATPart with a new filename, the parent CATProduct will need to be saved because of its reference to the driving CATPart. After saving a CATProduct with a new filename, the contextual CATParts that were defined in context of the CATProduct will have to be saved because of the CATPart's reference to the CATProduct
4. Copy: Using the Send to > Directory tool, you can create a copy of the CATProduct along with all related components.



Isolated external references appear with the broken link,



Creating a copy of CATProduct using Send to > Directory.

Main Tools (1/3)

Tools > Options Settings

1 **Work with the cache system:** Activates the visualization mode.

2 **Do not activate default shapes on Open:** Prevents loading the representation of the Components in the CATProduct when it is opened.

3 **Keep link with selected object:** The object created with this option will keep the link with the original part/reference.

Representation Tools

4 **Design Mode:** Loads the component in the design mode.

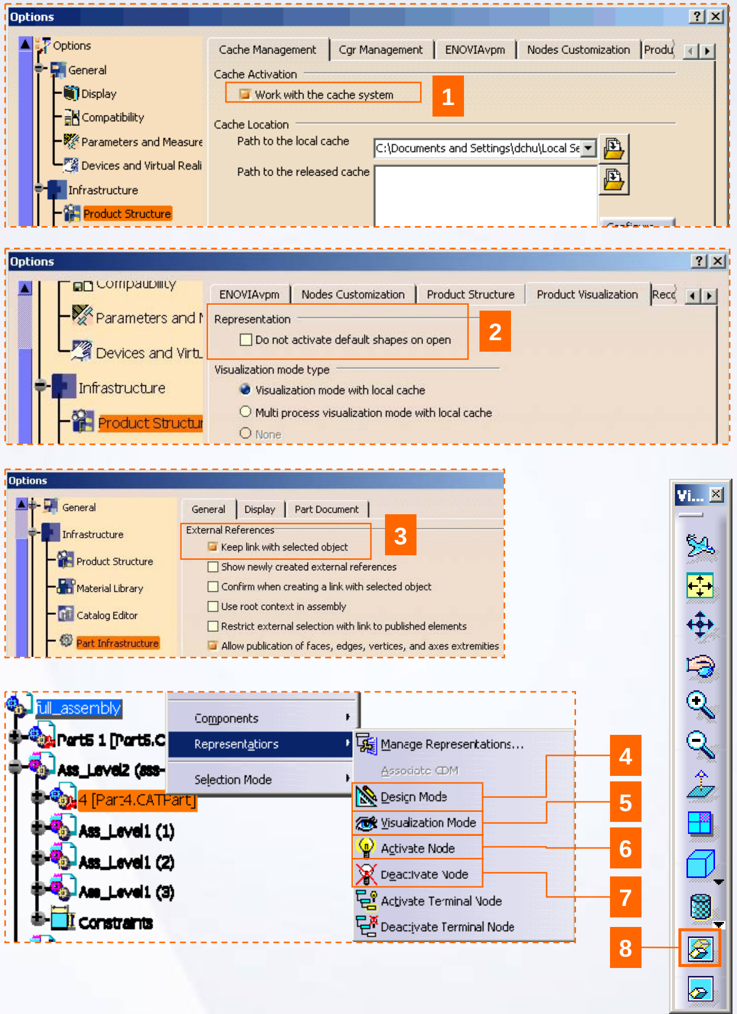
5 **Visualization Mode:** Loads the component in the visualization mode.

6 **Activate Node:** Activates the shape representation of the component.

7 **Deactivate Node:** Displays relationships between components and constraints.

View Toolbar

8 **Hide/Show:** Hides/Shows the selected components.



Main Tools (2/3)

Activate / Deactivate Tools

9 **Activate / Deactivate Component:** Activates/Deactivates a selected component.

Product Structure Tools

10 **Selective Load:** Manages the loading of sub-products level by level.

Knowledge Toolbar

11 **Formula:** Creates formulae and parameters to incorporate design constraints.

Assembly Features

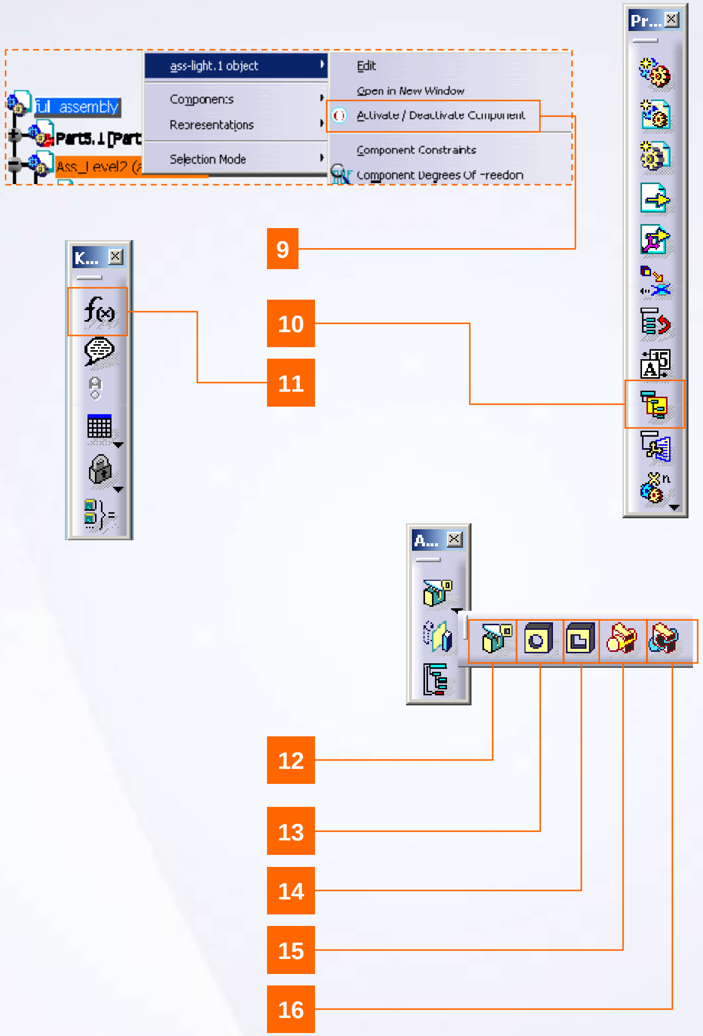
12 **Split:** Creates an assembly split feature.

13 **Hole:** Creates an assembly hole feature.

14 **Pocket:** Creates an assembly pocket feature.

15 **Add:** Creates an assembly add feature and adds the body to the selected parts.

16 **Remove:** Creates an assembly remove feature and subtracts the body from the selected parts.



Main Tools (3/3)

Manipulating Contextual Parts

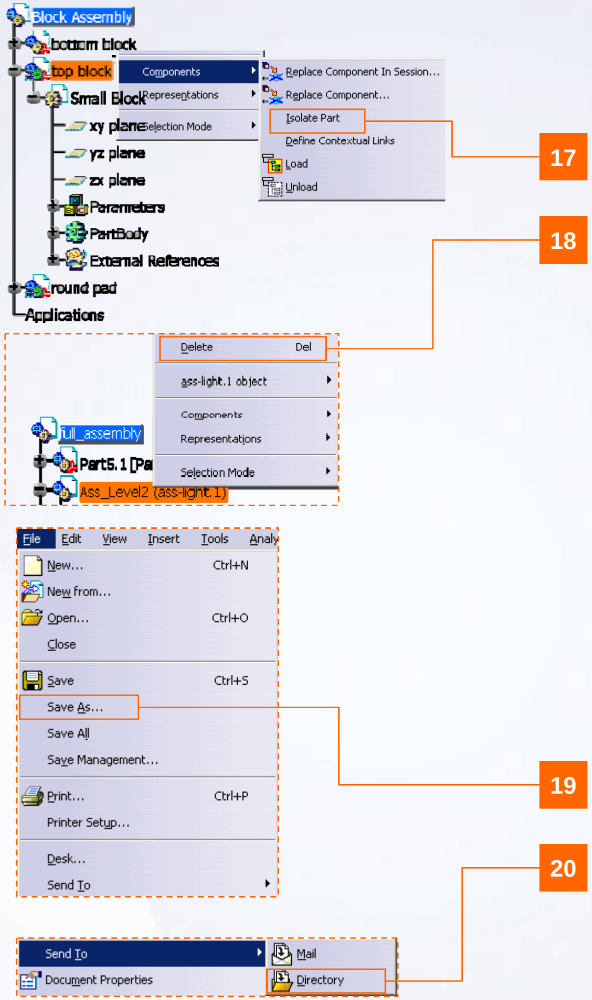
17 **Isolate:** Isolates the part by removing contextual links.

18 **Delete:** Deletes the selected component.

Save Tools

19 **Save As:** Saves the component with a new name.

20 **Send to > Directory:** Creates copy of the component in selected directory.



Case Study: Contextual Design

Recap Exercise

40 min

In this exercise you will create the case study model. Recall the design intent of this model:

- ✓ Contextual links must be used to ensure that changes to the referenced parts are reflected in the contextual part
- ✓ Contextual links can only reference the housing component
- ✓ The oval cut may need to intersect other component that have not yet been created
- ✓ Assembly must be saved to another directory in its entirety

Using the techniques you have learnt in this and previous lessons, create the model with only high-level instruction.

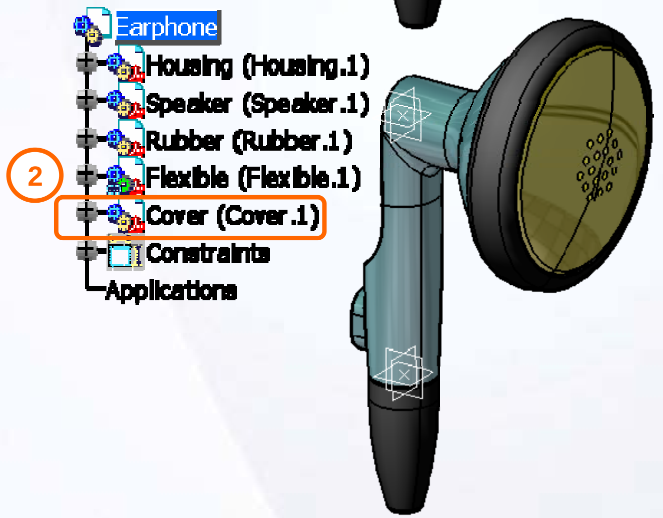
Do It Yourself: Earphone (1/11)

You must complete the following tasks:

1. **Open existing product file.**
   - Open Earphone_start.CATProduct.

2. **Create a new part.**
   - Create a new part named 'Cover'.

3. **Constrain the new part.**
   - Position the new part using reference elements of Housing component in order to center it on Bend_Point.
   - Create a coincidence between 'Bend_Point' of Housing and XY plane of Cover.
   - Create a coincidence between YZ plane of Housing and YZ plane of Cover.
   - Create a coincidence between ZX plane of Housing and ZX plane of Cover.



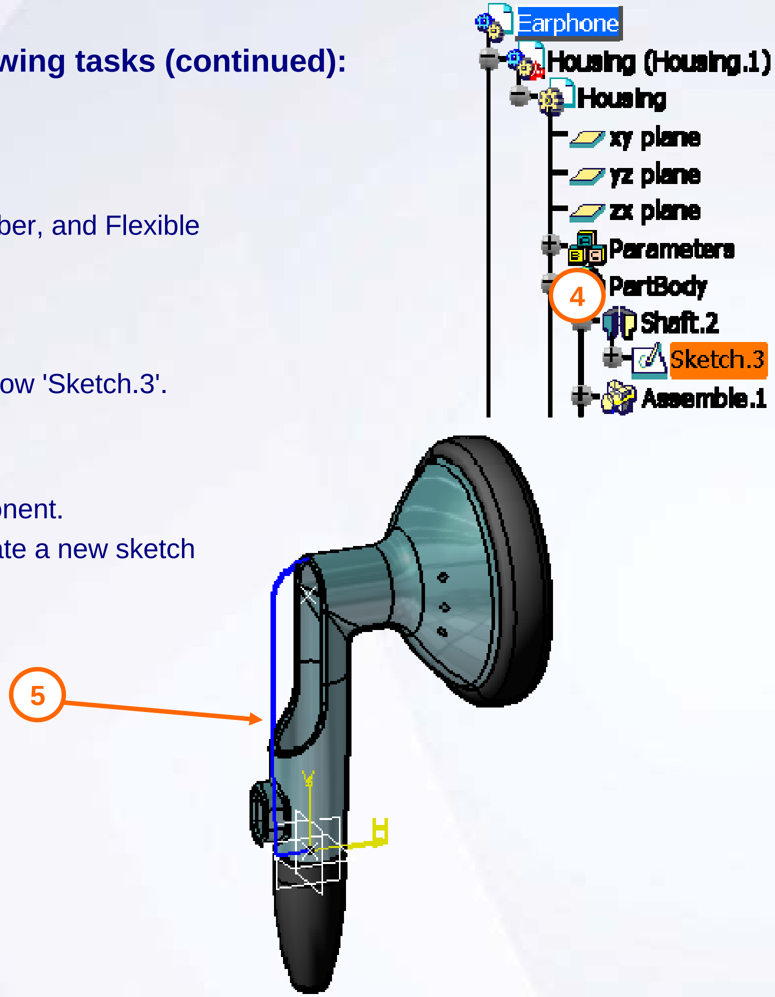
Do It Yourself: Earphone (2/11)

You must complete the following tasks (continued):

4. **Unload components.**
   - Unload the Speaker, Rubber, and Flexible components.

5. **Show a sketch.**
   - In Housing component show 'Sketch.3'.

6. **Create a sketch.**
   - Activate the Cover component.
   - In Cover component, create a new sketch lying on ZX plane.

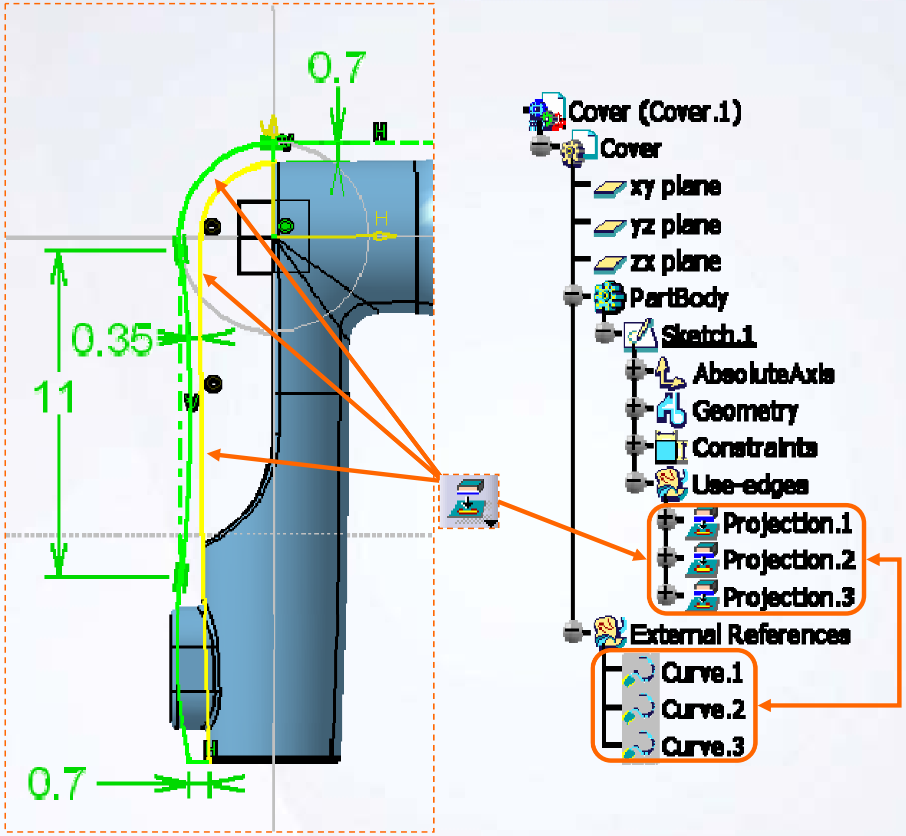


Do It Yourself: Earphone (3/11)

You must complete the following tasks (continued):

6. Create a sketch (continued).

- Project the three outlines of 'Sketch.3' in this new sketch. Make sure the link is kept with Housing Component.
- Add geometry as shown on the scheme and exit the sketcher.



Do It Yourself: Earphone (4/11)

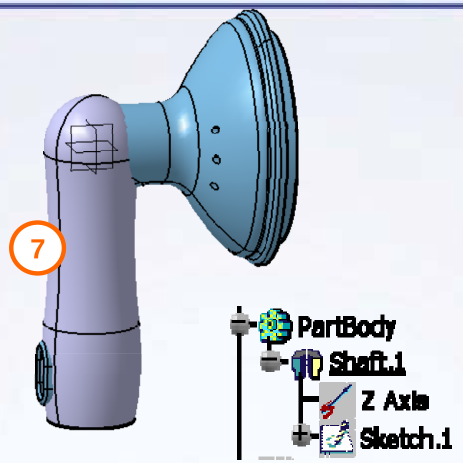
You must complete the following tasks (continued):

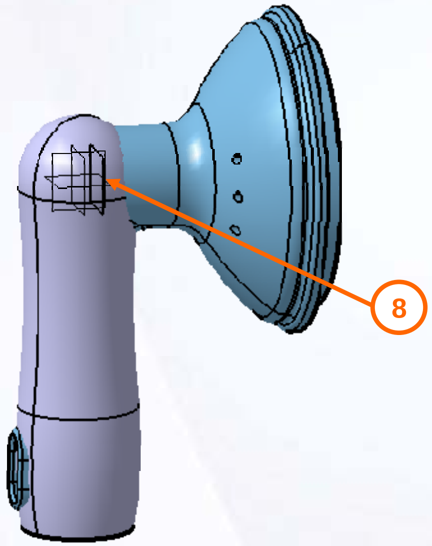
7. Create a shaft.

- Create a complete Shaft around Z Axis with the sketch previously created.

8. Create a plane.

- Create a new plane defined with an offset of 1mm from YZ plane. Reverse its direction if necessary. This plane will be used to split the shaft feature.



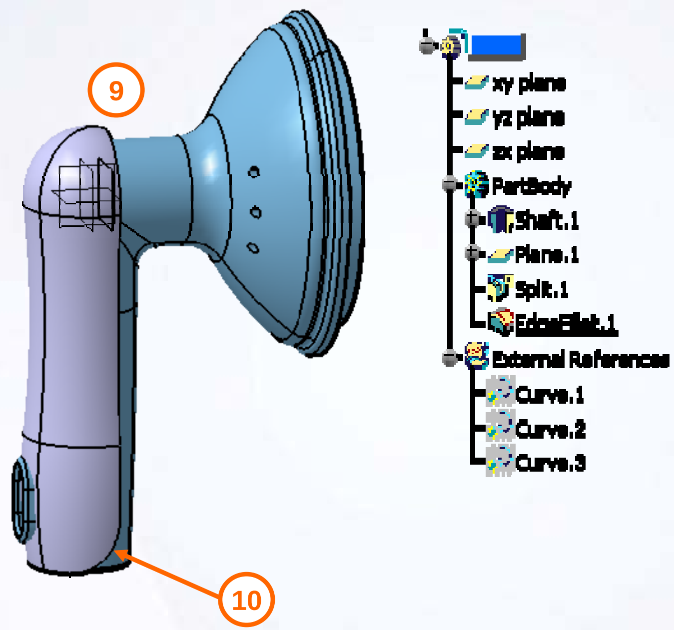


Do It Yourself: Earphone (5/11)

You must complete the following tasks (continued):

9. Create a split.
- Use this plane to split the current solid. Keep the biggest part of the solid.

10. Create a fillet.
- Define an edge fillet of Radius 2.5mm on the two corners of the Cover.



Do It Yourself: Earphone (6/11)

You must complete the following tasks (continued):

11. Create a groove.

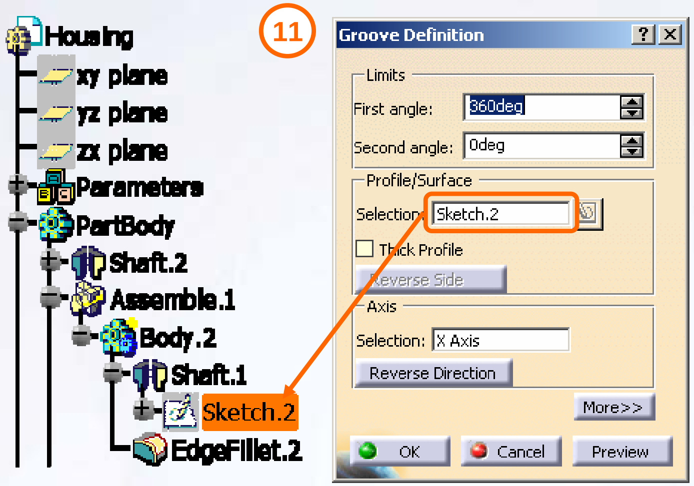
- Create a groove reusing 'Sketch.2' (with link) from Housing component.

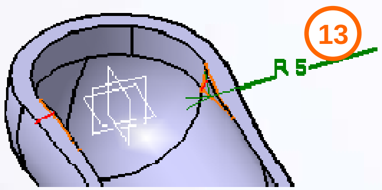
12. Hide component.

- Hide the Housing component for clarity.

13. Create a fillets.

- Create two edge fillets of 5mm to smooth the edges left by the groove.



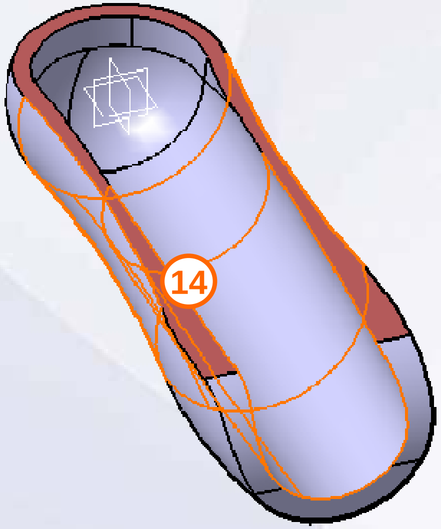


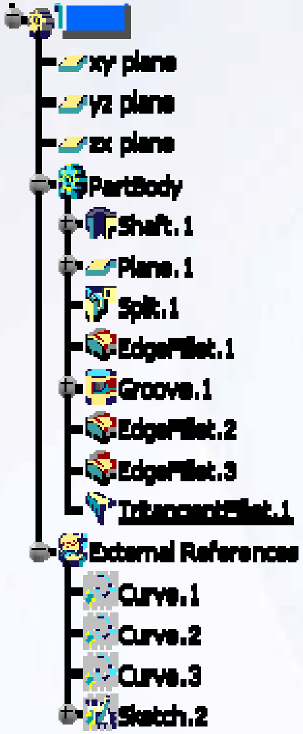
Do It Yourself: Earphone (7/11)

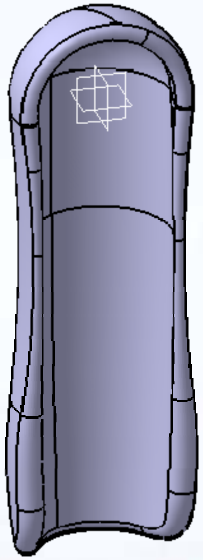
You must complete the following tasks (continued):

14. Create a tritangent fillet.

- Create a tritangent fillet to remove the planar face.







Do It Yourself: Earphone (8/11)

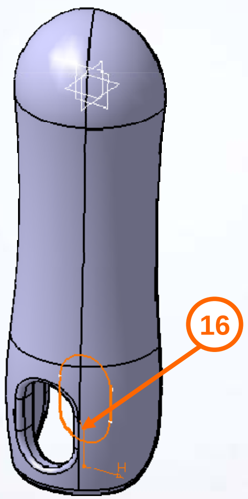
You must complete the following tasks (continued):

15. Create an assembly level pocket.

- Reuse Sketch.8 from the Housing component to create a Pocket at the assembly level that cuts through the Cover component. Use the **Up to Last** depth option.

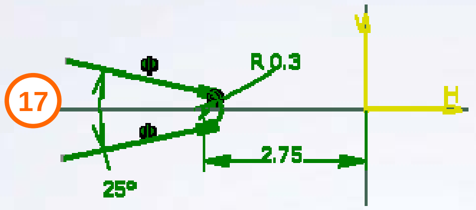
16. Create a fillet.

- Activate the Cover component.
- Fillet the inner face of the pocket (Radius = 0.2mm).



Do It Yourself: Earphone (9/11)

You must complete the following tasks (continued):

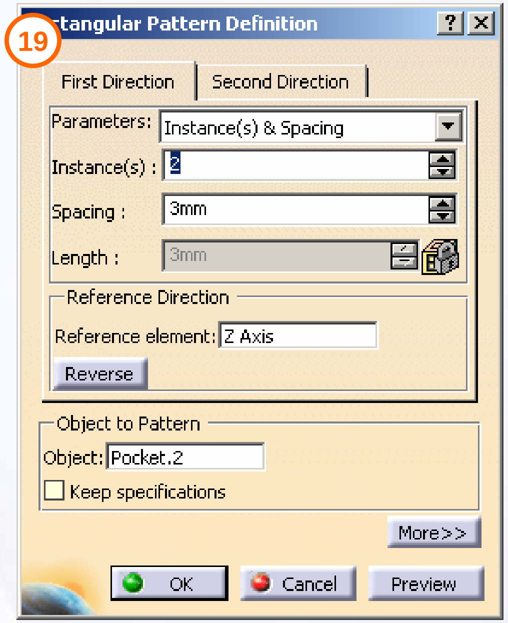


17. Create a sketch.

- Create a new sketch as shown. Use the ZX plane as the sketch support.

18. Create a pocket.

- Create a pocket using the Up to Last limit type for both the first and second limits.

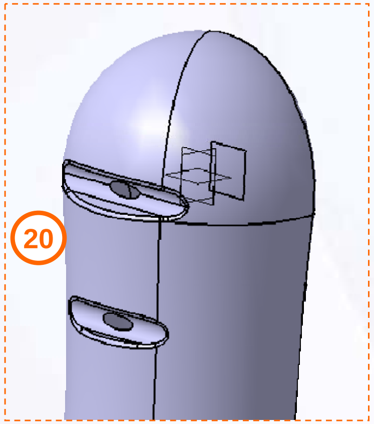


19. Create a pattern.

- Create a rectangular pattern to duplicate the pocket.

20. Create fillets.

- Fillet both pockets (Radius 0.1mm)



Do It Yourself: Earphone (10/11)

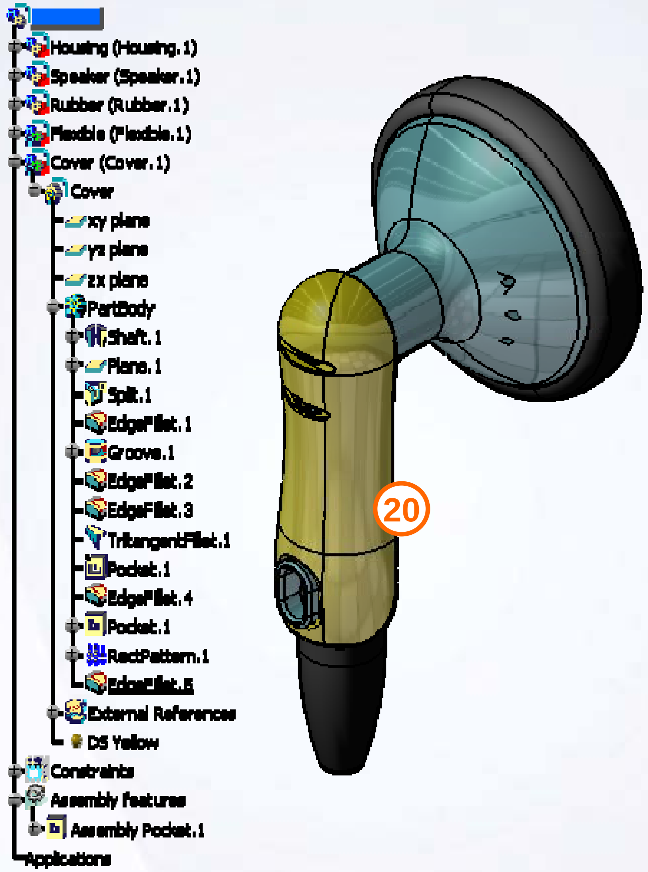
You must complete the following tasks (continued):

21. Add color to the part.

- You can apply the material of your choice (Painting for instance) on Cover component.

22. Display all components.

- Show the Housing component and Load the Speaker, Rubber, and Flexible components.



Do It Yourself: Earphone (11/11)

You must complete the following tasks (continued):

23. Verify links

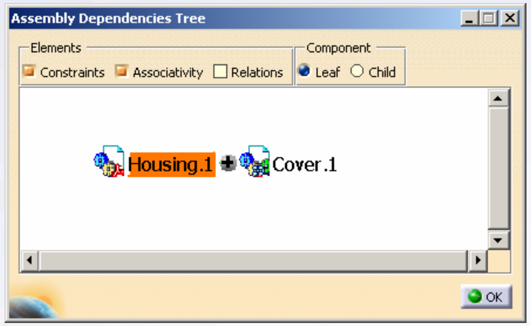
- Ensure that only the Cover component has external links to the housing component.

24. Save the assembly.

25. Send the assembly to another directory.

- Create a new folder called Earphone_Complete and save the entire assembly to this directory.

23



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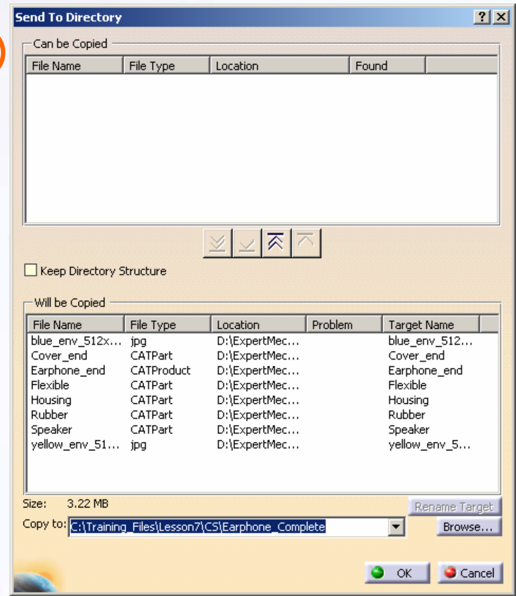
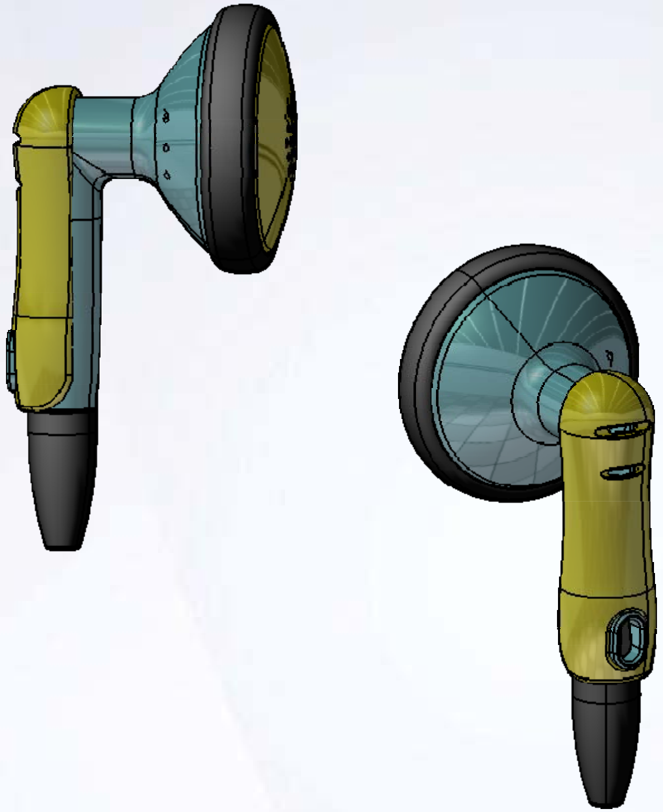


Case Study: Contextual Design Recap

- ✓ Unload components
- ✓ Create part in context
- ✓ Hide components
- ✓ Create an assembly level pocket
- ✓ Load components
- ✓ Show components
- ✓ Save the entire assembly to another directory using the Send To command

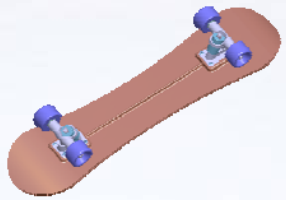


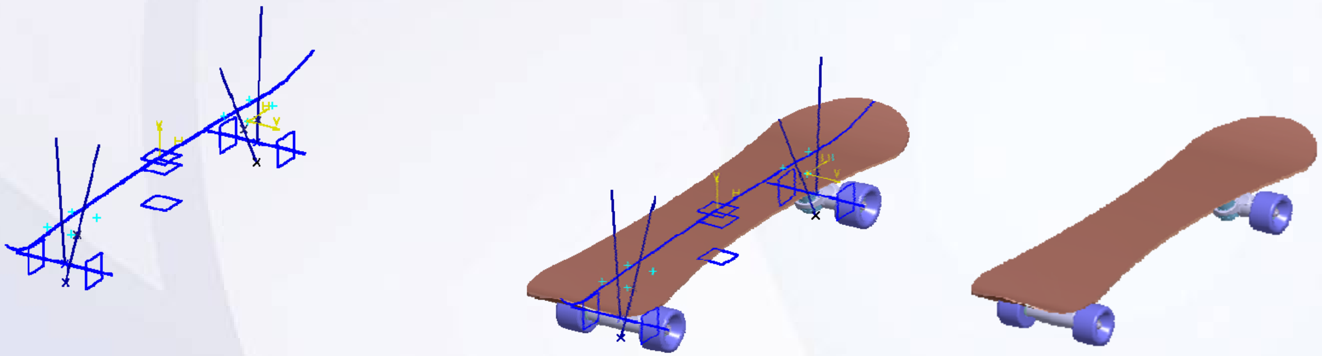
Complex Assembly Design 8

Learning Objectives

Upon completion of this lesson you will be able to:

- ✓ Stages in the Process
- ✓ Create a Skeleton Model
- ✓ Create the Published Elements
- ✓ Use the Published Elements





4 Hours

Case Study

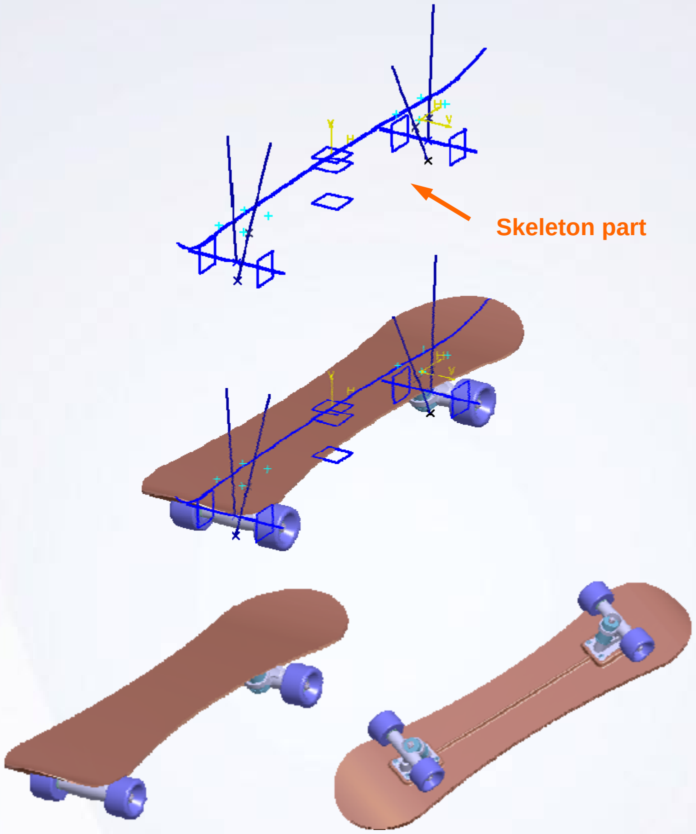
The case study for this lesson is a skateboard assembly.

Design Intent

1. Component locations must be controlled from a centralized location using the skeleton method.
2. Support geometry must update in all components using the skeleton approach.
3. References must be strictly controlled. Using published geometry, only published elements are allowed for selection when creating external references and assembly constraints.

Stages in the Process

1. Create a Skeleton model.
2. Create published elements.
3. Use the published elements.

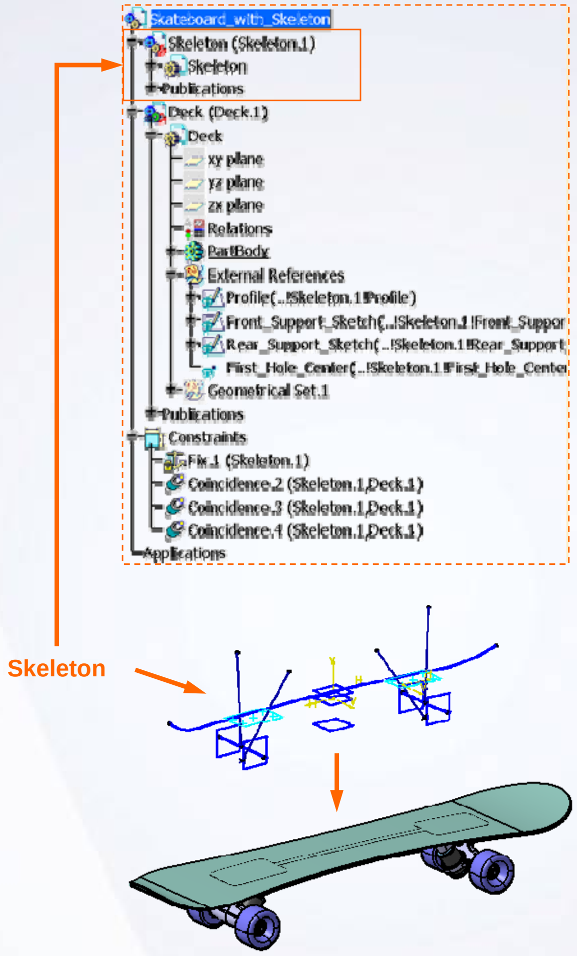


Skateboard design using skeleton approach.

Create the Skeleton Model

The skeleton method is a top down design approach used to create and reuse information stored in a single part to define the underlying design framework of components and assemblies. It has the following advantages:

1. Specification-driven design: All important information related to the design and space requirements are defined within the skeleton.
2. Design change propagation: It helps manage high-level design changes and propagate them throughout the assembly.
3. Collaborative design: Key information stored in the skeleton model can be associatively copied into the appropriate components used in the product. The components can then be edited separately by different designers.
4. Avoids update loops: All are external references point to a single source: the skeleton. Since the skeleton model does not use any external references to define its geometry update loops are avoided.

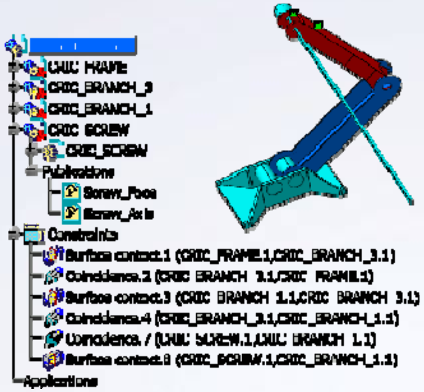


Skateboard design using skeleton approach.

Create Published Geometry

Publishing geometrical elements is the process by which geometrical features are made available to different users. It is most applicable to the geometry and parameters of a skeleton model. It has the following benefits:

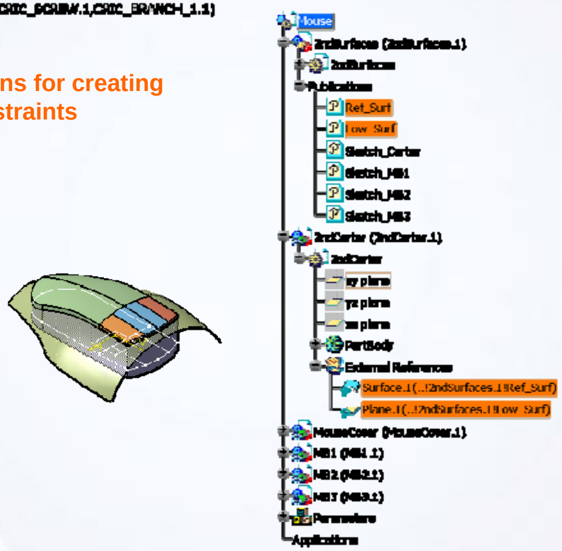
A. Label geometry to give it a name that can be easily recognized (particularly in the case of publishing edges, faces, etc.).
B. To make particular geometry easier to access from the specification tree.
C. Control external references. An option is available that lets you to select as external reference only (For the published elements).
D. Easy replacement of one feature of the part with another.



Use Publications for creating assembly constraints

Use Published Geometry

Published geometry is particularly useful when replacing components with assembly constraints or components with external links that have been designed in context.



Use Publications in contextual design

Main Tools

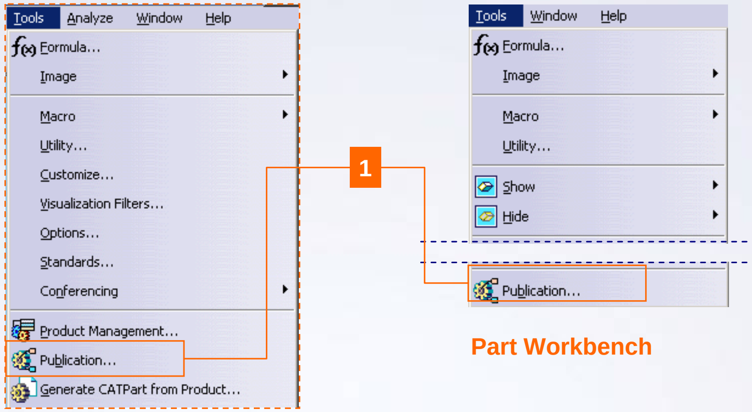
Publications

1 **Tools > Publication:** Creates publications of the selected element.

Tools > Options

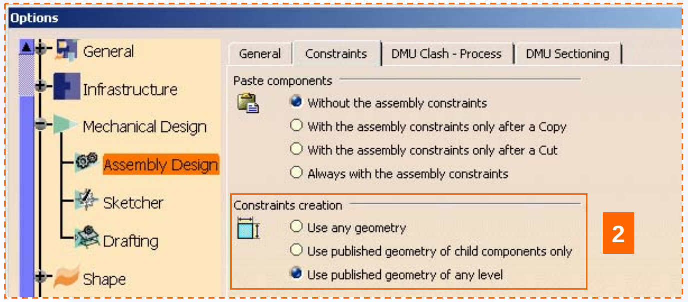
2 **Assembly Constraints:** You can choose to select options for using publications while creating assembly constraints.

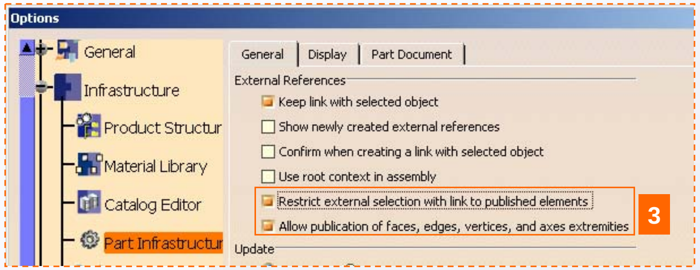
3 **External References:** You can choose options while creating external references.



Part Workbench

Assembly Workbench





Exercise: Skeleton and Design in Context

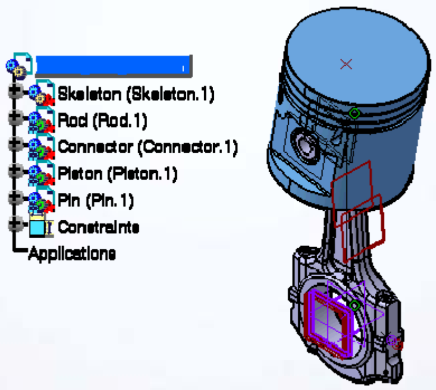
Recap Exercise

90 min

In this exercise, you will use the tools learnt in the present and previous lessons to create an assembly using the skeleton method. You will use a skeleton model to control a rod and a piston assembly, by referring to its geometry to position components and design in context. High-level instructions for this exercise are provided.

By the end of this exercise you will be able to:

- **Use the skeleton method**
- **Design a part in context using the skeleton model for external references**
- **Constrain an assembly using a skeleton model**



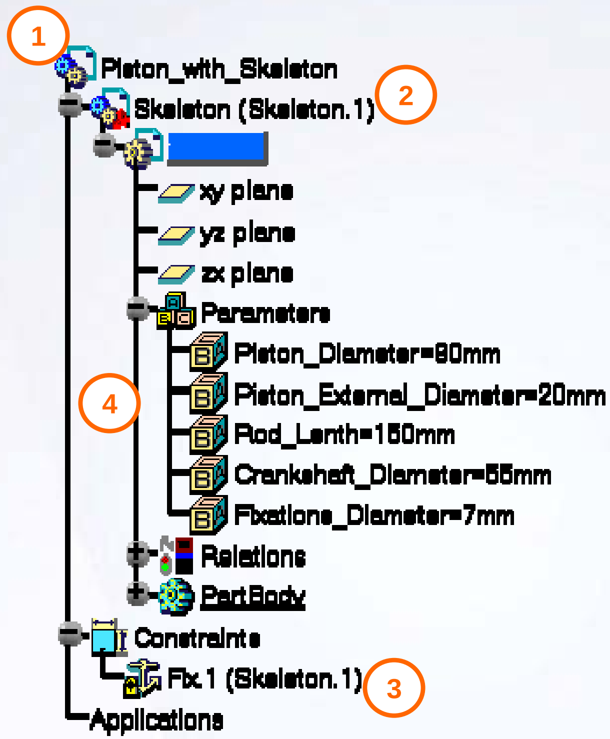
Do it Yourself (1/14)

1. **Create a new product file.**
   - Name the product [Piston_with_Skeleton].

2. **Create Skeleton.CATPart.**
   - Create a new component inside the assembly called [Skeleton.CATPart].

3. **Fix the skeleton model in the assembly.**

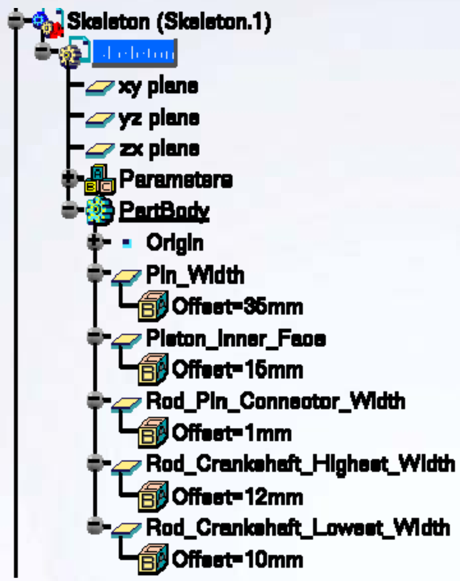
4. **Create user parameters**
   - Activate the skeleton component and create the five user parameters shown.
   - Create the new parameters of type Length.

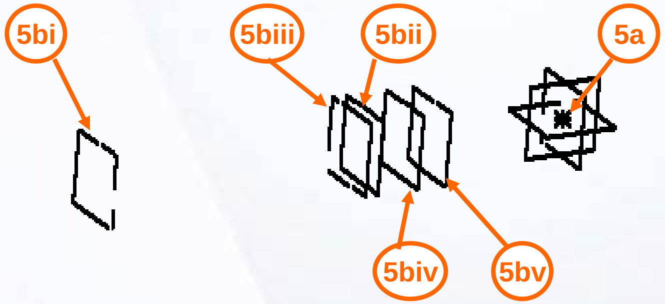


Do it Yourself (2/14)

5. Create additional skeleton geometry.

- The main dimensions of a model can be expressed not only with user parameters but also with geometry (planes, axis, points, etc.)
  - a. Create a point to locate the origin of the part.
  - b. Create five planes and rename them as shown:
    - i. Pin_Width = [35mm] offset from the YZ plane
    - ii. Piston_Inner_Face = [15mm] offset from the YZ plane
    - iii. Rod_Pin_Connector_Width = [1mm] offset from the Piston_Inner_face
    - iv. Rod_Crankshaft_Highest_Width = [12mm] from the YZ plane.
    - v. Rod_Crankshaft_Lowest_Width = [10mm] offset from the YZ plane.

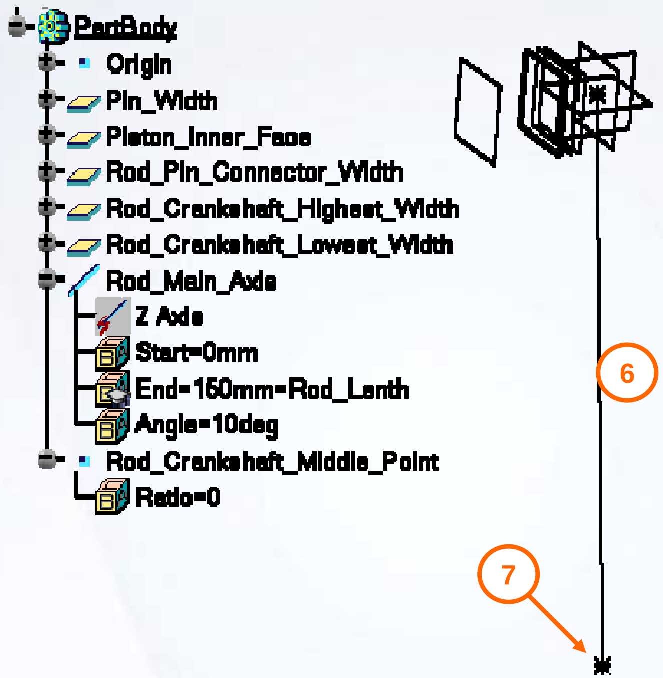




Do it Yourself (3/14)

6. Create a line.

- Create a line named [Rod_Main_Axis] using the following references:
  - a. Type: Angle/Normal to curve
  - b. Curve: Z Axis
  - c. Support: YZ plane
  - d. Point: Origin
  - e. Angle: 10deg.
  - f. Start: 0mm
  - g. End: Create a formula equal to the [Rod_Length] parameter.
- This line represents the direction of the rod.



7. Create a point.

- Create a point at the end of the Rod_Main_Axis.
- Name the point [Rod_Crankshaft_Middle_Point].

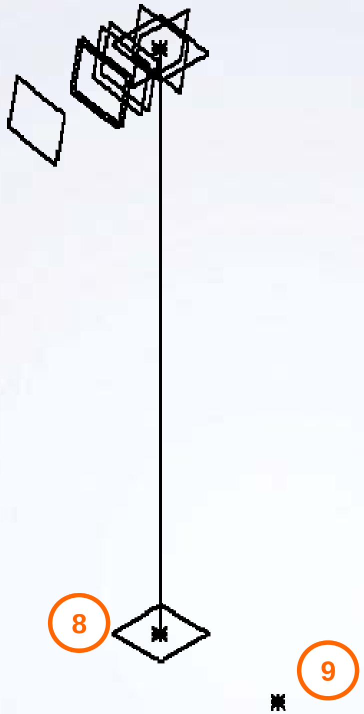
Do it Yourself (4/14)

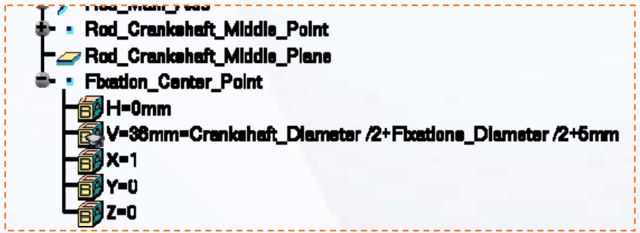
8. Create a plane.

- Create a plane normal to the Rod_Main_Axis and through the Rod_Crankshaft_Middle_Point.
- Rename the point to [Rod_Crankshaft_Middle_Plane]

9. Create a point.

- Create a point called [Fixation_Center_Point] using the following reference:
  - a. Type: On plane
  - b. Plane: Rod_Crankshaft_Middle_Plane
  - c. H = 0
  - d. V = From the contextual menu Edit Formula to be [Crankshaft_Diameter / 2 + Fixations_Diameter / 2 + 5mm]

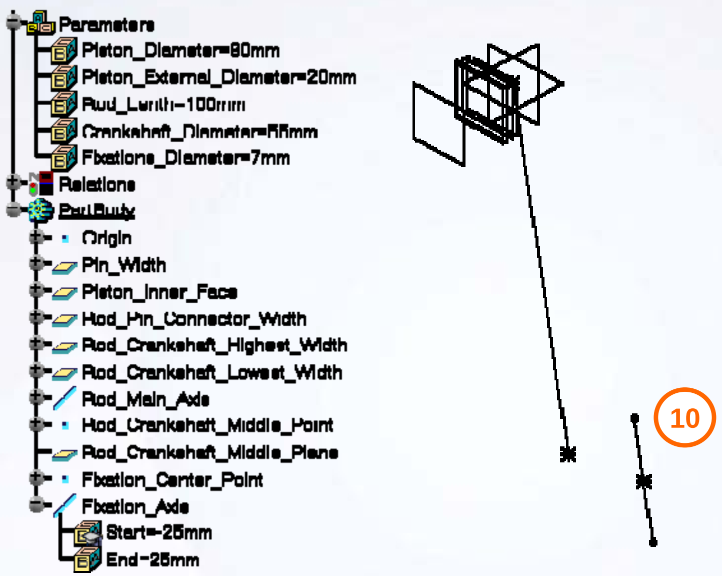




Do it Yourself (5/14)

10. Create a line.

- Create a line called [Fixation_Axis] using the following references.
  a. Type: Point-Direction
  b. Point: Fixation_Center_Point
  c. Direction: Rod_Main_Axis
  d. Support: YZ plane
  e. End: 25mm
  f. Select the Mirrored Extent option



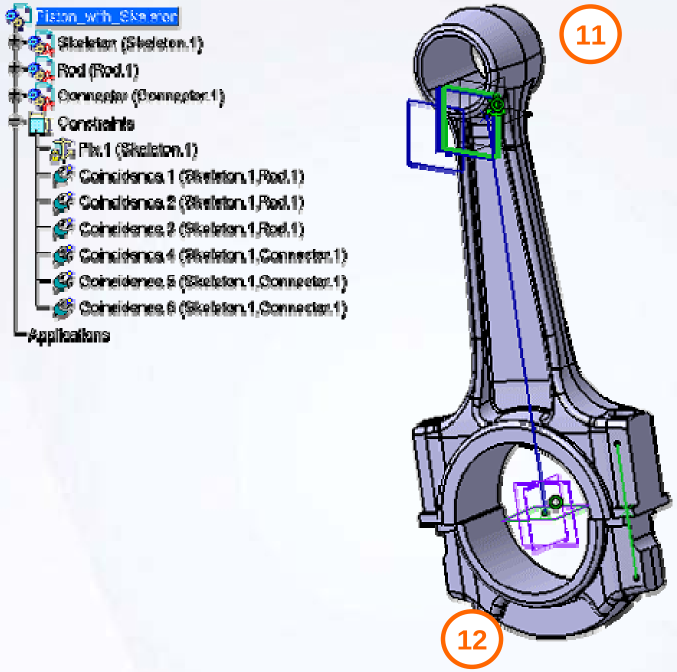
Do it Yourself (6/14)

11. Insert an existing component.

- Insert the Rod.CATPart.
- Use the following coincidence constraints to place the component:
  a. Rod_Crankshaft_Middle_Plane of skeleton with XY plane of Rod.
  b. Rod_Crankshaft_Middle_Point of skeleton with ZX plane of Rod.
  c. YZ plane of skeleton with YZ plane of Rod.

12. Insert an existing component.

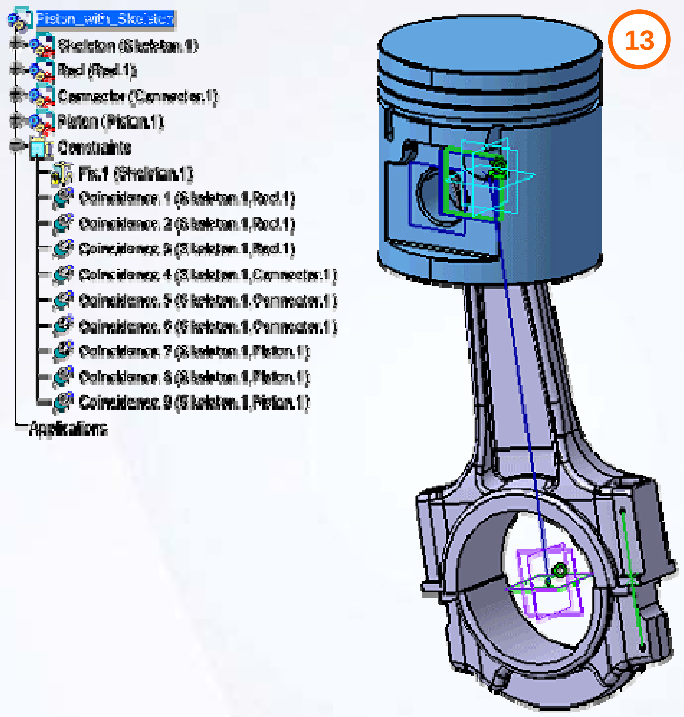
- Insert the Connector.CATPart into the assembly.
- Use the following coincidence constraints to place the component:
  a. Rod_Crankshaft_Middle_Plane of skeleton with XY plane of Connector.
  b. Rod_Crankshaft_Middle_Point of skeleton with ZX plane of Connector.
  c. YZ plane of skeleton with YZ plane of Connector.



Do it Yourself (7/14)

13. Insert an existing component.

- Insert the Piston.CATPart into the assembly.
- Use the following coincidence constraints to place the component:
  - a. XY plane of the skeleton with XY plane of the Piston.
  - b. YZ plane of the skeleton with YZ plane of the Piston.
  - c. ZX plane of the skeleton with ZX plane of the Piston.



Do it Yourself (8/14)

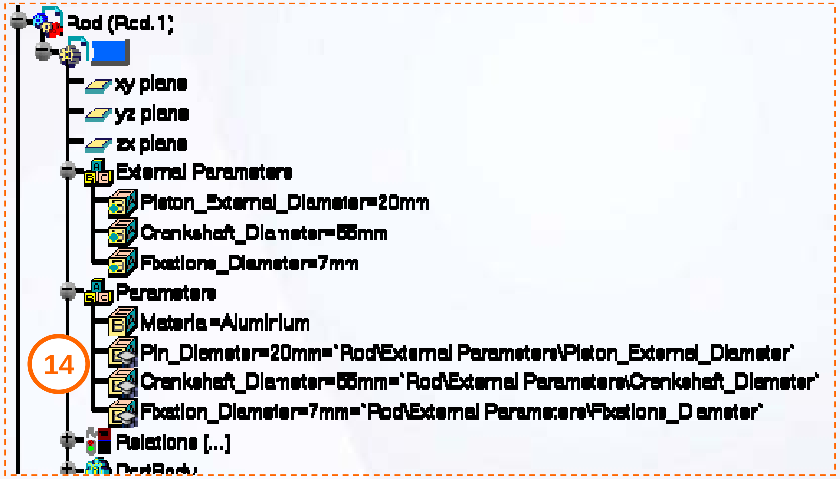
14. Link the Rod component parameters.

- Activate the rod component and link the rod’s user parameters to the skeleton’s corresponding user parameters:

a. Pin_Diameter = [Pin_External_Diameter] of skeleton.

b. Crankshaft_Diameter = [Crankshaft_Diameter] of skeleton.

c. Fixations_Diameter = [Fixations_Diameter] of skeleton.

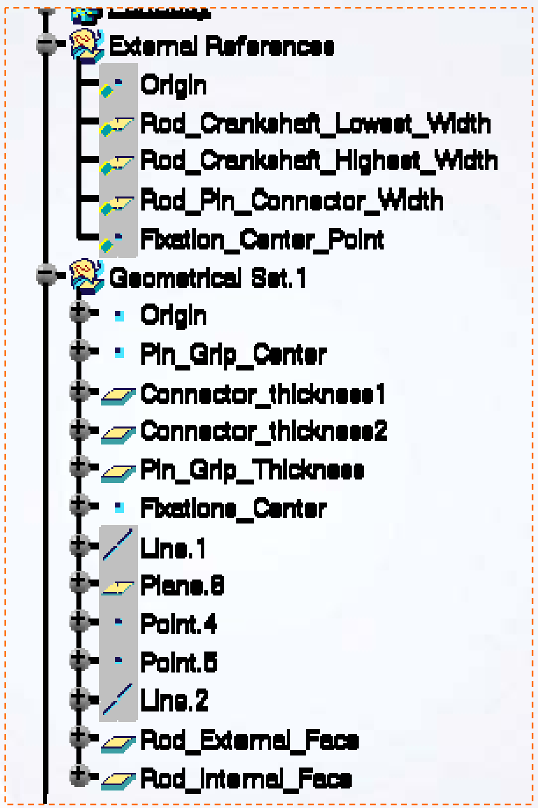


Do it Yourself (9/14)

15. Replace geometrical elements.

- From the geometrical set, replace:
  a. [Pin_Grip_Center] by [Origin] of skeleton.
  b. [Connector_Thickness1] by [Rod_Crankshaft_Lowest_Width] of skeleton.
  c. [Connector_Thickness2] by [Rod_Crankshaft_Highest_Width] of skeleton.
  d. [Pin_Grip_Thickness] by [Rod_Pin_Connector_Width] of skeleton.
  e. [Fixation_Center] by [Fixation_Center_Point] of skeleton.

15



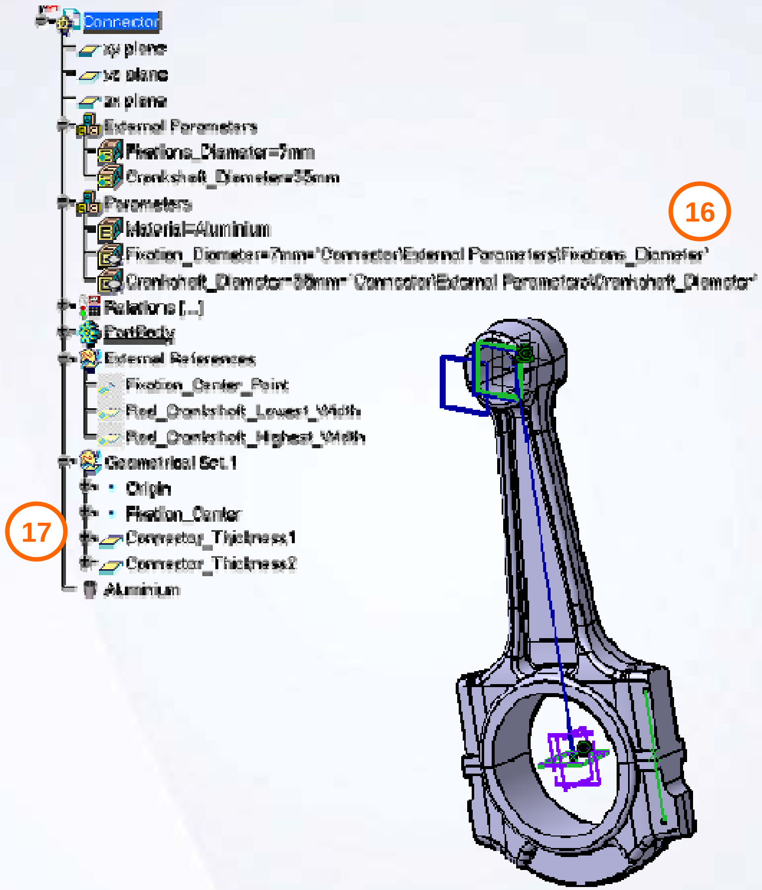
Do it Yourself (10/14)

16. Link the connector parameters.

- Activate the connector component and link the connector's user parameters to the skeleton's corresponding user parameters:
  - a. Fixation_Diameter = [Fixation_Diameter] of skeleton.
  - b. Crankshaft_Diameter = [Crankshaft_Diameter] of skeleton.

17. Replace geometrical elements.

- From the geometrical set, replace:
  - a. [Fixation_Center] by [Fixation_Center_Point] of skeleton.
  - b. [Connector_Thickness1] by [Rod_Crankshaft_Lowest_Width] of skeleton.
  - c. [Connector_Thickness2] by [Rod_Crankshaft_Highest_Width] of the skeleton.



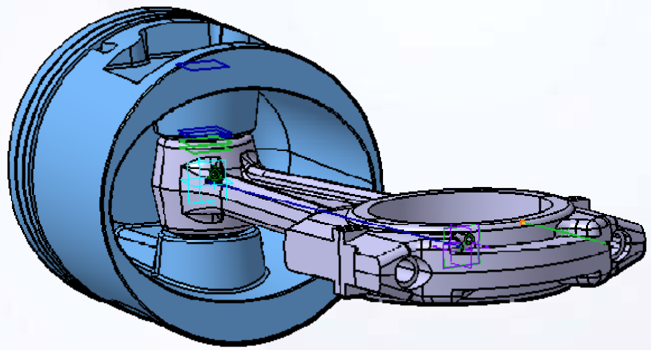
Do it Yourself (11/14)

18. Link the piston parameters.

- Activate the piston component and link the piston's user parameters to the skeleton's corresponding user parameters:
  - a. Piston_Radius = Piston_Diameter /2.

19. Replace geometrical elements.

- From the geometrical set, replace:
  - a. [Left_Face_Plane] by [Pin_Width] of skeleton.
  - b. [Left_Inner_Plane] by [Piston_Inner_Face] of skeleton.



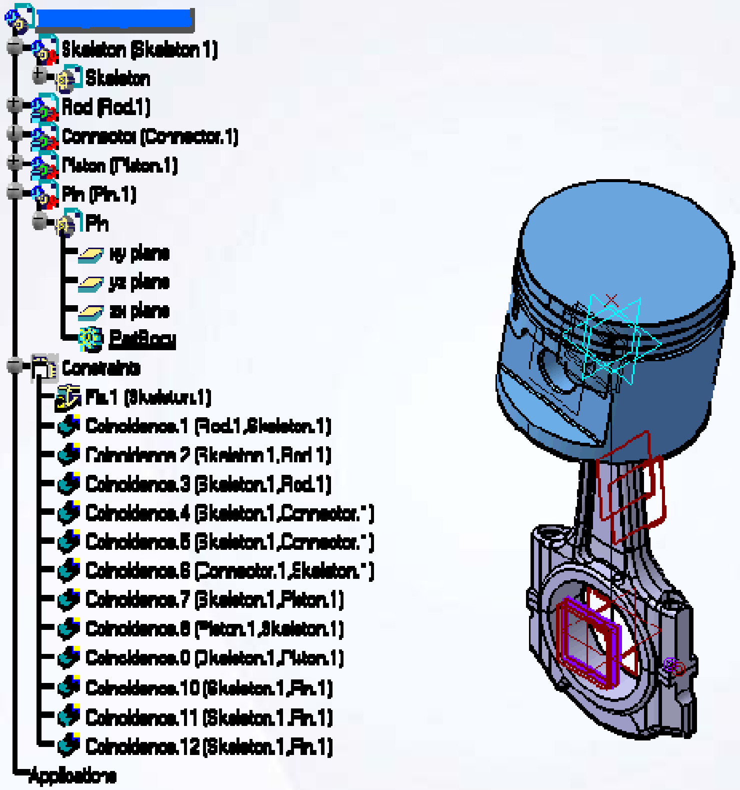
Do it Yourself (12/14)

20. Create a new part.

- Activate the top-level assembly.
- Create a new part named [Pin].

21. Position the pin.

- Constrain the pin component using the following coincident constraints.
  - a. XY plane of the skeleton with XY plane of the Pin.
  - b. YZ plane of the skeleton with YZ plane of the Pin.
  - c. ZX plane of the skeleton with ZX plane of the Pin.
- Update the assembly to place the component.



Do it Yourself (13/14)

22. Create pin geometry.

- Activate the pin component and create the following contextual geometry.

a. Activate the pin component.

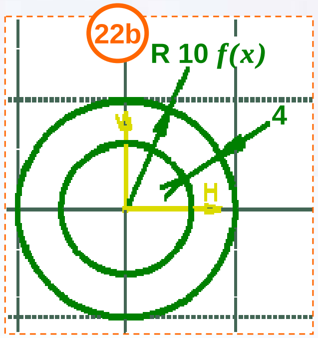
b. Create the sketch shown on the YZ plane.

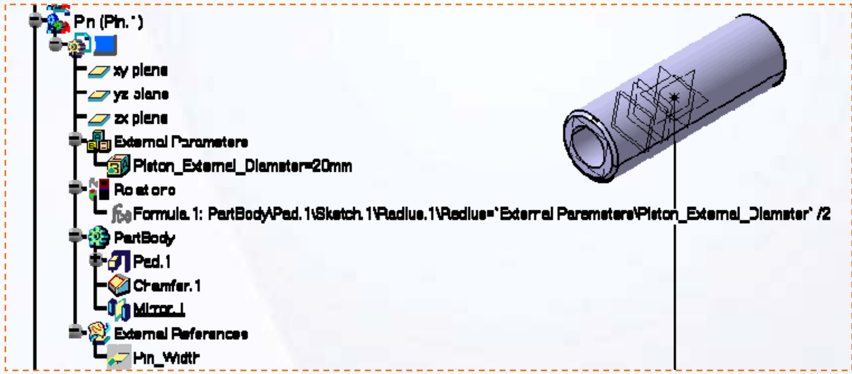
c. Create a formula to control the external radius. Make the external radius equal to half of the [Pin_External_Diameter] defined in the skeleton.

d. Create a pad feature. Create the pad up to the [Pin_Width] plane.

e. Create two chamfers [1mm/45deg] on the external face of the pin.

f. Mirror the pin geometry about the YX plane.





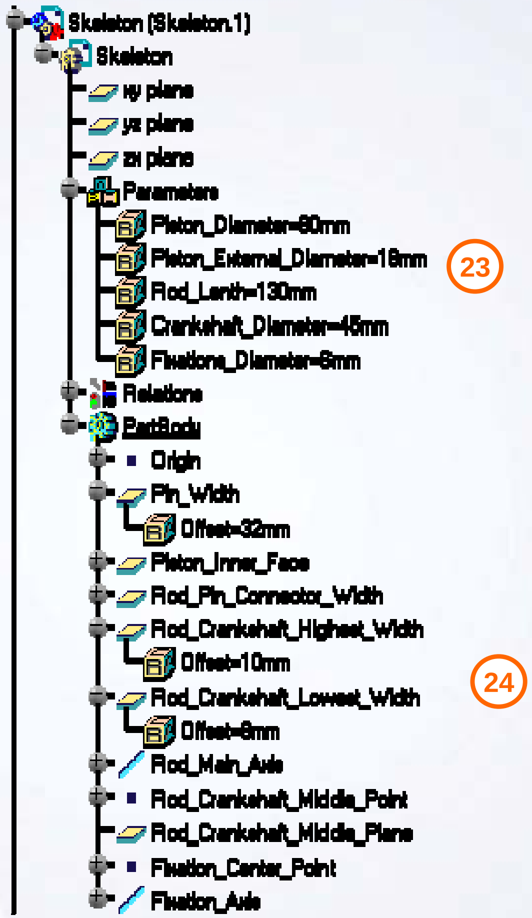
Do it Yourself (14/14)

23. Edit specifications.

- Now that all the components of the product are linked to the specifications of the skeleton, we can change the general specifications of the product just by editing the skeleton.
  - a. Activate the skeleton component.
  - b. Change the user parameter values:
    - a. Piston Diameter = 80mm
    - b. Pin_External_Diameter = 18mm
    - c. Rod_Length = 130mm
    - d. Crankshaft_Diameter = 45mm
    - e. Fixations_Diameter = 6mm
  - c. Change the value of the offsets for the following planes:
    - a. Pin_Width = 32mm
    - b. Rod_Crankshaft_Highest_Width = 10mm
    - c. Rod_Crankshaft_Lowest_Width = 8mm

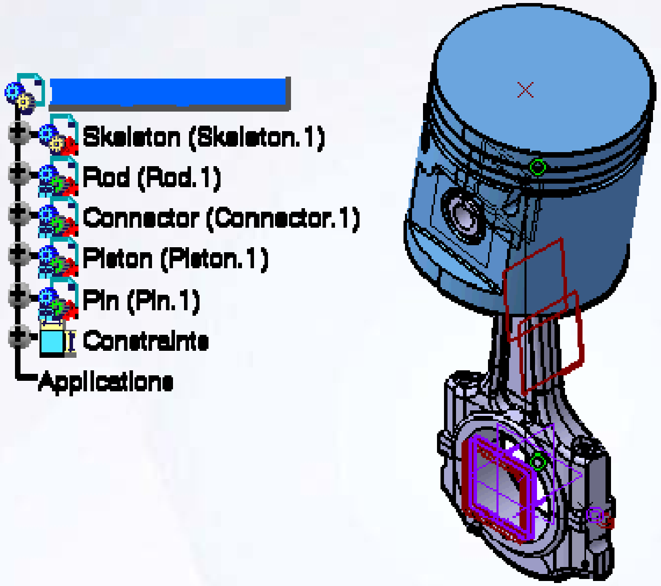
24. Update the top-level assembly.

- Activate the top-level assembly and update it. Notice the changes made to the skeleton propagate through the entire assembly.



Exercise: Skeleton and Design in Context Recap

- ✓ Use the skeleton method
- ✓ Design a part in context using the skeleton model for external references
- ✓ Constrain an assembly using a skeleton model



Exercise: Publication

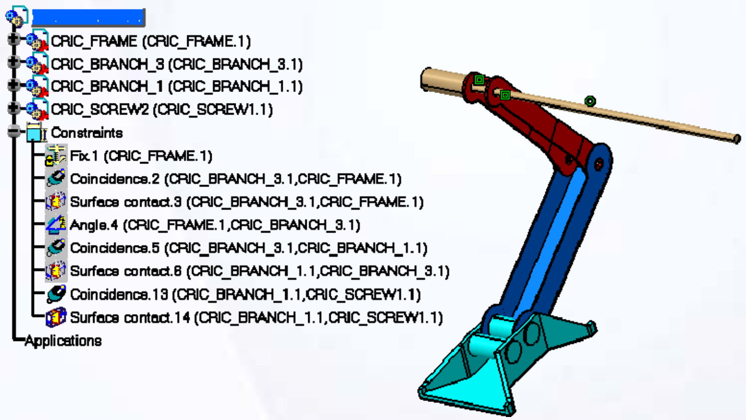
Recap Exercise

20 min

In this exercise, you will open an existing assembly and replace one of its components. To ensure constraint references are not lost, you will publish elements in the replaced and replacing components. High-level instructions for this exercise are provided.

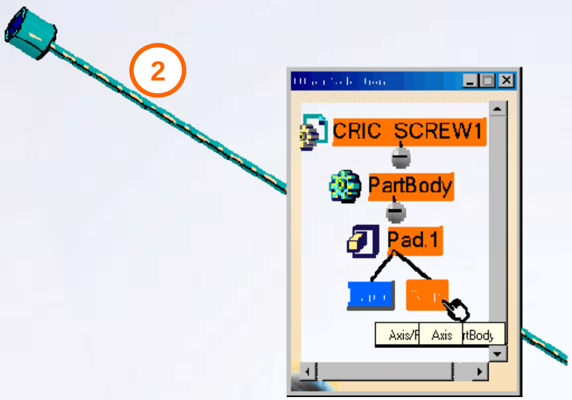
By the end of this exercise you will be able to:

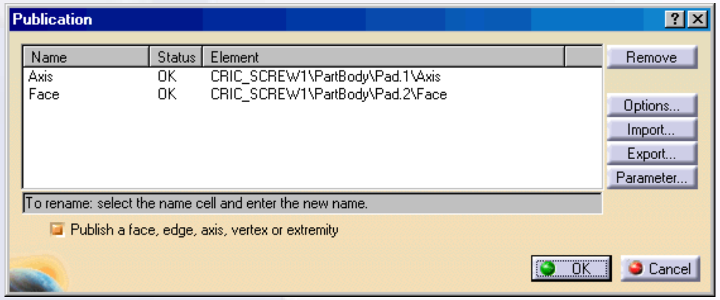
- **Create published elements**
- **Replace components**

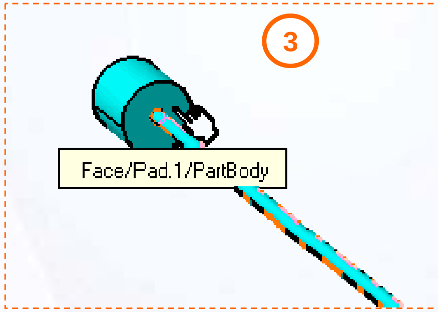


Do it Yourself (1/4)

1. **Open CRIC_SCREW1.CATPart.**
2. **Publish the axis as shown.**
3. **Publish the face as shown.**

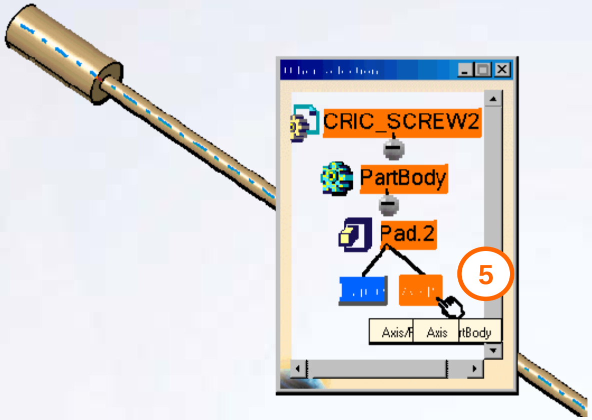


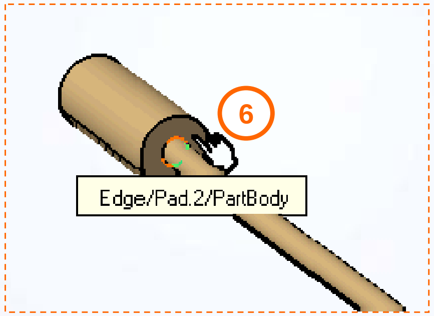


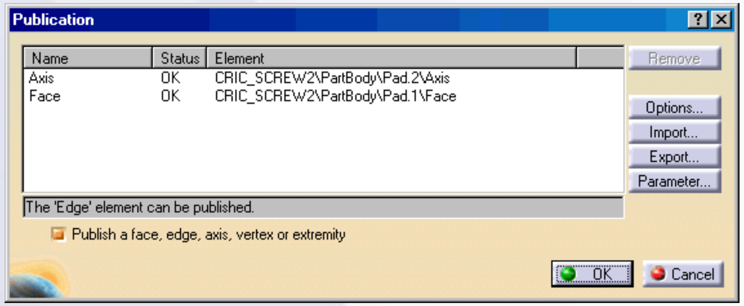


Do it Yourself (2/4)

4. Open CRIC_SCREW2.CATPart.
5. Publish the axis as shown.
6. Publish the face as shown.

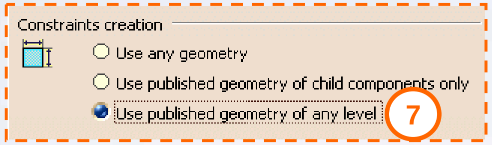






Do it Yourself (3/4)

7. Click Tools > Options > Mechanical Design > Assembly Design. From the Constraints tab set the Constraints creation option to Use published geometry of any level.



8. Open CricFirstAssembly.CATProduct.

9. Expand the CRIC_BRANCH1 component. Notice that two of its elements are already published.

Do it Yourself (4/4)

10. Assemble CRIC_SCREW1.CATPart using the published geometry.

11. Create a coincident constraint between the Axis and the BRANCH1_AXIS published elements.

12. Create a surface contact constraint between the FACE and the BRANCH1_FACE published elements.

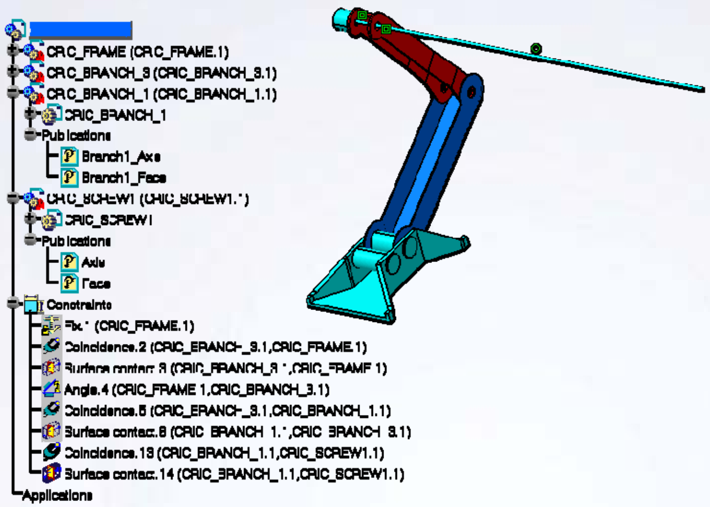
13. Update the assembly.

14. Replace CRIC_SCREW1.CATPart with CRIC_SCREW2.CATPART.

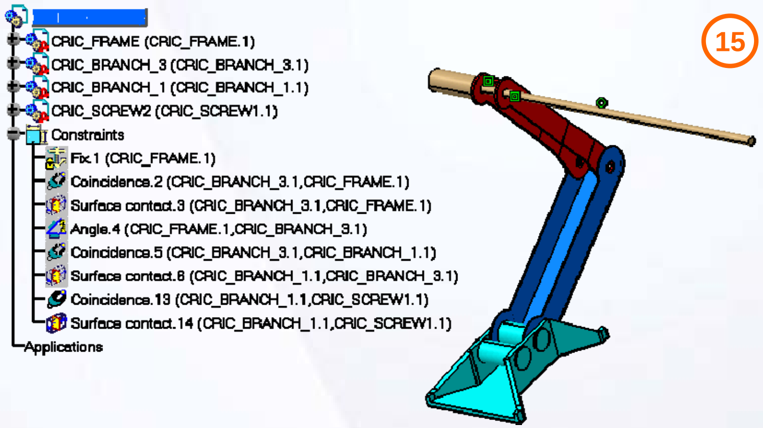
15. Update the assembly. Notice the assembly constraints are automatically replaced because of the publications.

16. Save and close the assembly and all associated files.

13

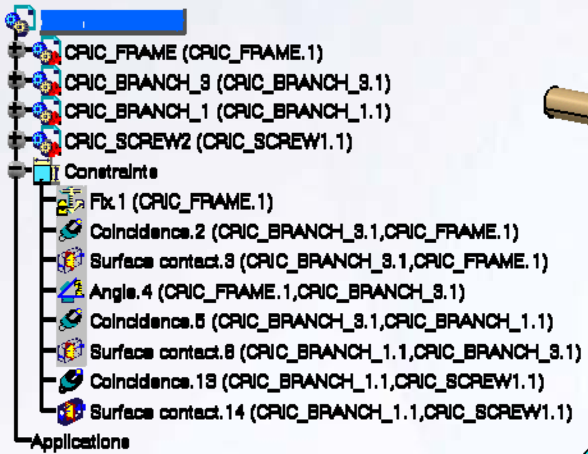


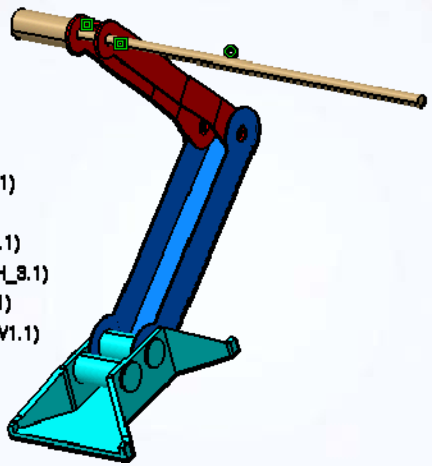
15



Exercise: Publication Recap

- ✓ Create publications
- ✓ Replace components





Case Study: Complex Assembly Design

Recap Exercise

80 min

In this exercise, you will create the case study model. Recall the design intent of this model:

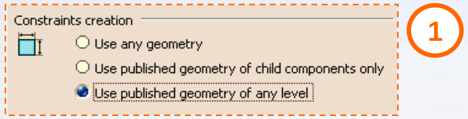
- ✓ Component locations must be controlled from a centralized location.
- ✓ Support geometry must be defined contextually.
- ✓ References must be strictly controlled.

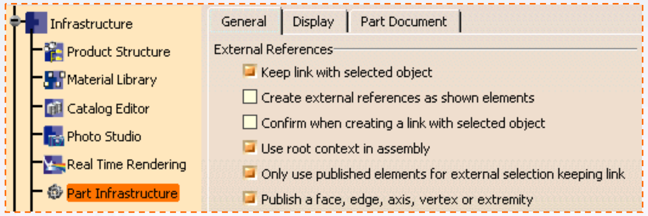
Using the techniques you have learnt in this and previous lessons, create the model with only high-level instruction.

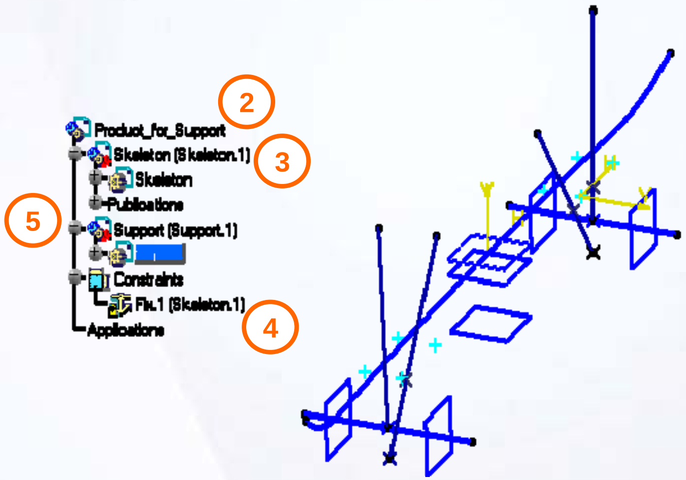
Do It Yourself: Skateboard (1/17)

You must complete the following tasks:

1. **Ensure that the options are set correctly.**
   - Set the Constraint Creation option to **Use Published Geometry of any level**.
   - Set the External References options as shown.
2. **Create a new product file.**
   - Name the product [Product_for_Support].
3. **Insert Skeleton.CATPart.**
   - This model has been created for you. Review the created geometry. Notice that the publications have already been created for you.
4. **Fix the skeleton model in the assembly.**
5. **Insert a new part called [Support].**
6. **Activate the support component.**



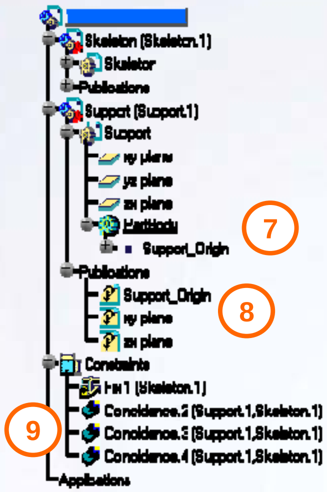


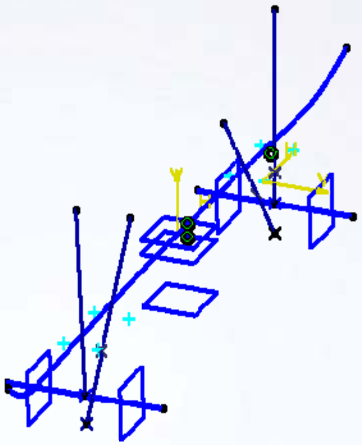


Do It Yourself: Skateboard (2/17)

You must complete the following tasks (continued):

7. **Create a new point.**
   - Create a new point of coordinates [0,0,0].
   - Rename the point to [Support_Origin].
8. **Publish elements**
   - Publish the [Support_Origin] point.
   - Publish the XY plane.
   - Publish the ZX plane.
9. **Position the support.**
   - Position the support in the product by creating the following coincidence constraints between the published elements in the support and those of the skeleton:
     a. Support_Origin with Front_Support_Middle_Point.
     b. XY plane with Support _Plane, choose the opposite orientation.
     c. ZX plane with ZX plane.





Do It Yourself: Skateboard (3/17)

You must complete the following tasks (continued):

10. Insert a body.

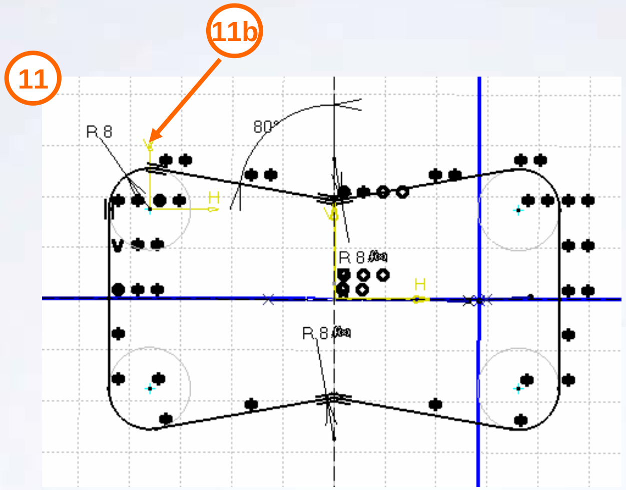
- Insert a new body into the Support component, name the body [Base].

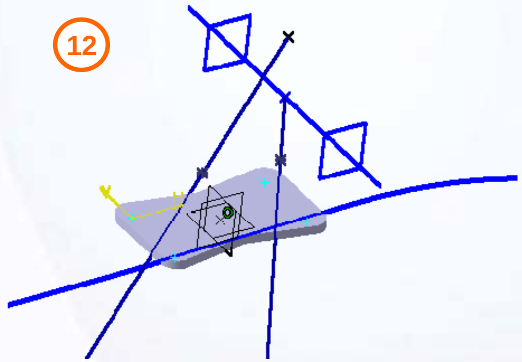
11. Create a sketch.

- Create the sketch shown on the XY plane. Ensure that all radii are equal.
- Constrain the outside radius shown to the "first_Hole_Center" point from the skeleton model.
- You need to create only the top left quarter of the sketch, and mirror it twice to create the final sketched geometry.

12. Create a pad.

- Use the sketch to create a [3mm] pad.





Do It Yourself: Skateboard (4/17)

You must complete the following tasks (continued):

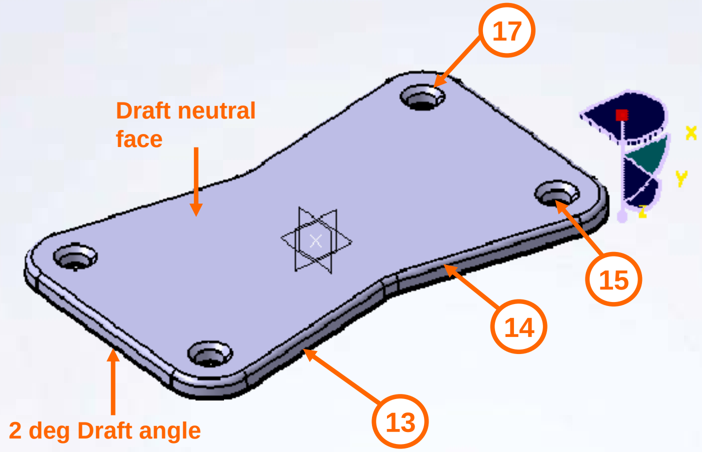
13. Apply draft.
- Create a [2deg] draft on the sides of the pad. Use the top surface as the neutral plane.

14. Create a [1mm] edge fillet on the upper face of the pad.

15. Create a hole
- Create a M6 threaded-hole centered on the First_Hole_Center point of skeleton.

16. Pattern the hole.
- Pattern the hole using the Fixation_Pattern publication from the skeleton to locate the instantiations.

17. Create a chamfer.
- Create a [1mm/45deg] chamfer on the upper edge of the four holes.



Do It Yourself: Skateboard (5/17)

You must complete the following tasks (continued):

18. Create a new body

- Create another body in the Support component called [Shock_support].

19. Create the sketch.

- Create the sketch shown on the ZX plane.
- You will need to project from the skeleton model:

a. The Front_Support_Axis

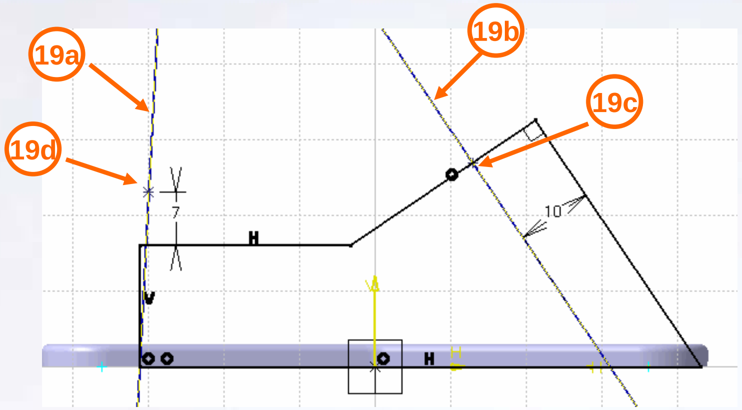
b. The Front_Shock_Absorber_axis

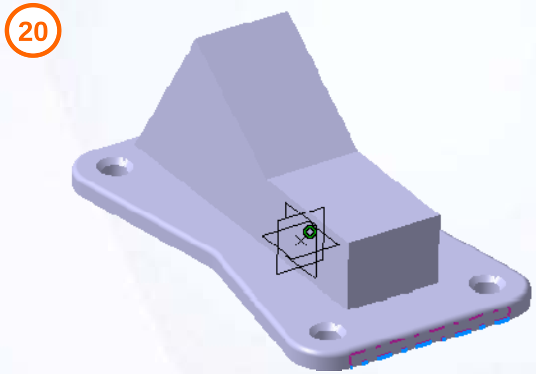
c. The Front_Shock1_Start_point

d. The Front_Axle_Connection_point

20. Create a pad.

- Create a pad of type Mirrored extent dimension. Use a length of [10mm].





Do It Yourself: Skateboard (6/17)

You must complete the following tasks (continued):

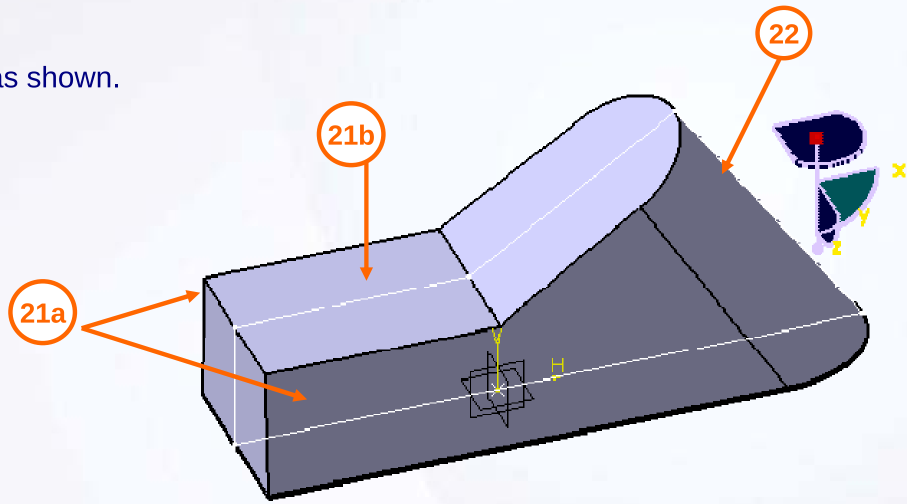
21. Apply draft.

a. Add a [3deg] draft to the two sides of the draft.

b. Use the top planar surface as the neutral plane.

22. Create a tritangent fillet.

- Create a tritangent fillet as shown.



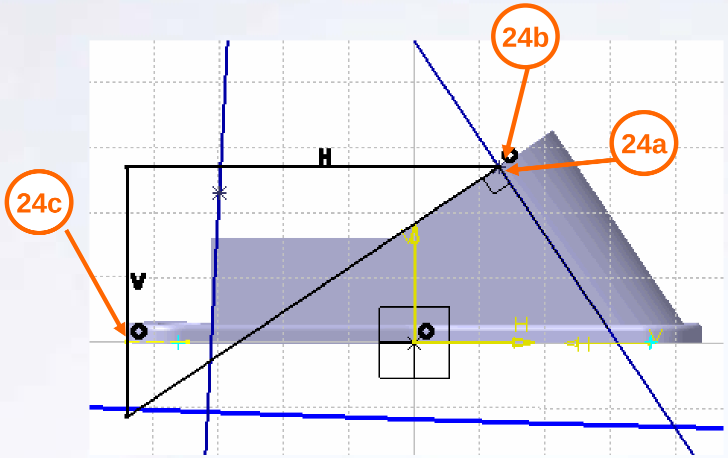
Do It Yourself: Skateboard (7/17)

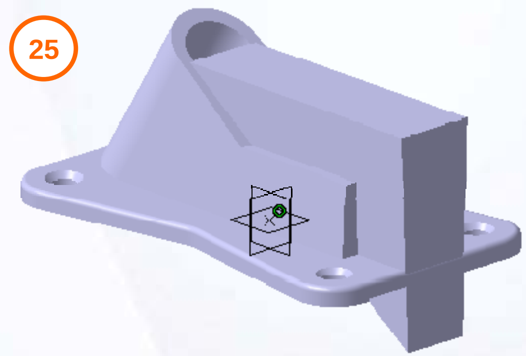
You must complete the following tasks (continued):

23. Create a new body.
- Create a new body in the Support component.

24. Create the sketch.
- Create the sketch shown on the ZX plane.
- Create the sketch such that:
  a. The angled line is perpendicular to the Front_Shock_Absorber_axis.
  b. The vertex of the horizontal line must be coincident with the Front_Shock1_Start_Point.
  c. The vertical line must be coincident with the base edge.

25. Create a pad.
- Create a pad from the sketch or type Mirrored Extend Dimension and of length [7mm].



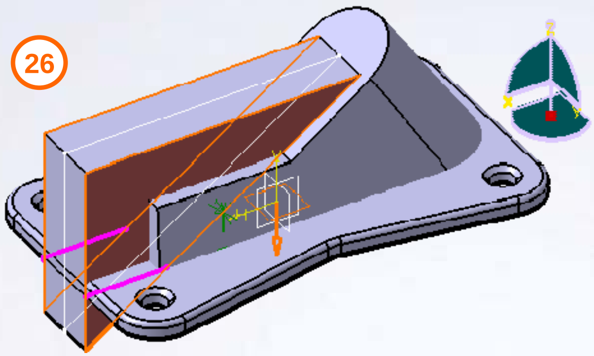


Do It Yourself: Skateboard (8/17)

You must complete the following tasks (continued):

26. Add draft.

- Add draft of [3deg] to both sides of the pad. Use the XY plane as the neutral plane.

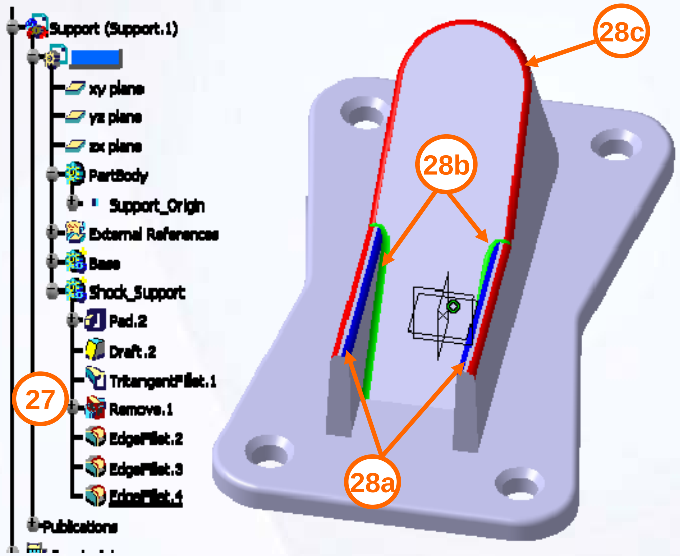


27. Remove the body.

- Remove the new body from the Shock_Support body.

28. Create fillets.

- Create three [1mm] edge fillets in the order shown.



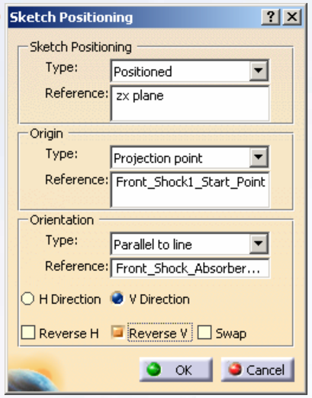
Do It Yourself: Skateboard (9/17)

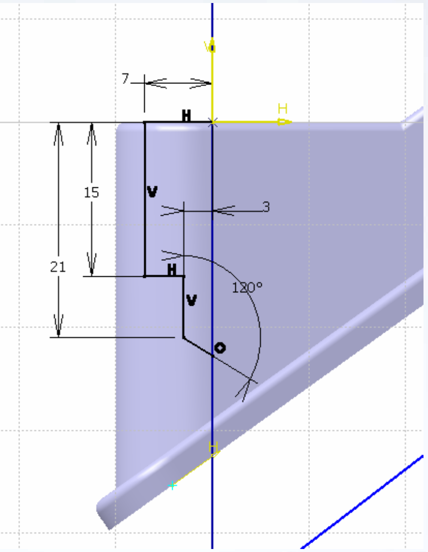
You must complete the following tasks (continued):

29. Create a sketch.

- Create a sketch with absolute axis definition on the ZX plane position the sketch as shown using the following references:
  - a. Origin: Front_Shock1_Start_Point
  - b. V Direction: Front_Shock_Absorber_Axis.

29





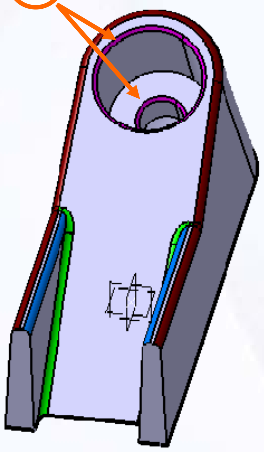
30. Create a groove feature.

- Create a groove feature using the sketch.

31. Add a chamfer.

- Add a [0.5mm/45deg] chamfer to the edges of the groove.

31



Do It Yourself: Skateboard (10/17)

You must complete the following tasks (continued):

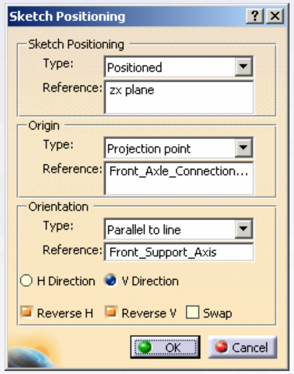
32. Create a new body.

- Create a new body in the Support component called Axle_Support.

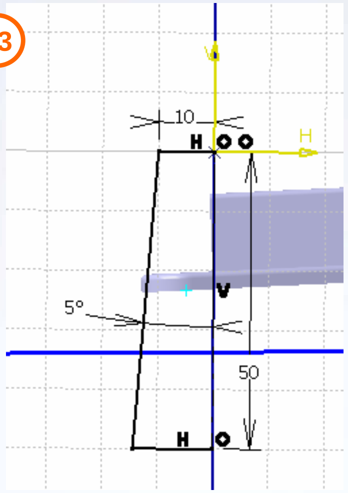
33. Create a shaft.

- Create the sketch shown.
- Use the following references to position it:
  a. Positioned on the ZX plane
  b. Origin: Front_Axle_Connection_Point
  c. V-direction: Front_Support_Axis

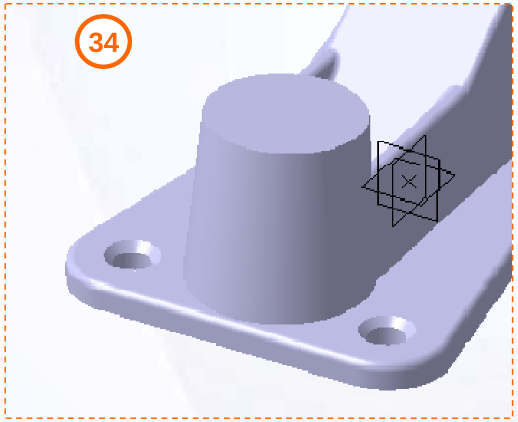
34. Split the body with the XY plane.



33



34



Do It Yourself: Skateboard (11/17)

You must complete the following tasks (continued):

35. Create a hole.

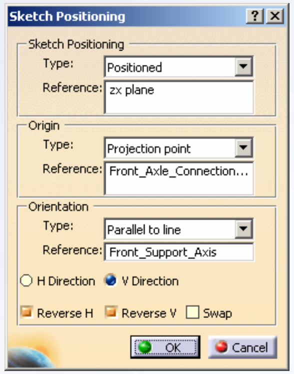
- Create a [2mm] hole using **Up to Last** option, centered on the Front_Axle_Connection_Point and in the direction of the Front_Support_Axis.

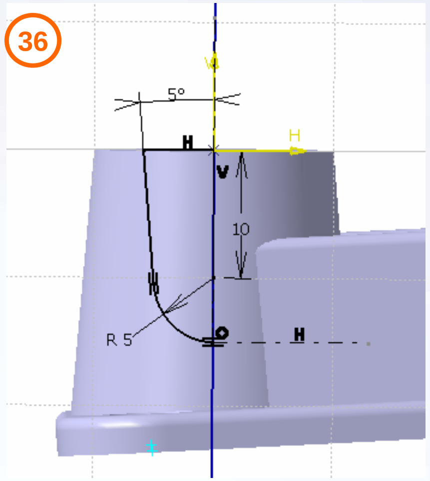
36. Create a groove.

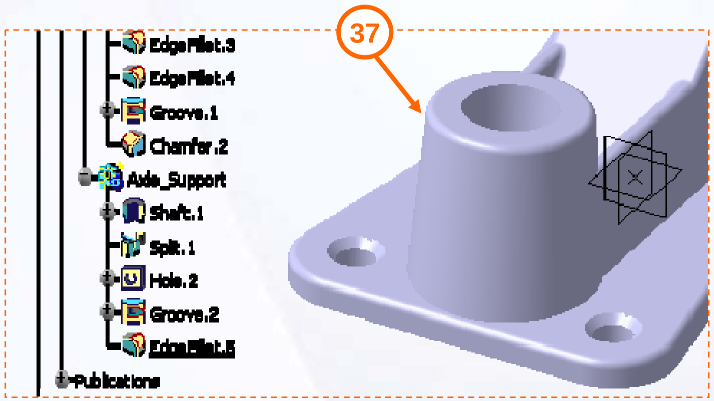
- Create a groove feature using the sketch shown. Position the same as the last sketch:
  - a. Positioned: ZX plane
  - b. Origin: Front_Axle_Connection_Point
  - c. V-Direction: Front_Support_Axis

37. Create a fillet.

- Apply a [1mm] edge fillet to the top of the shaft.







Do It Yourself: Skateboard (12/17)

You must complete the following tasks (continued):

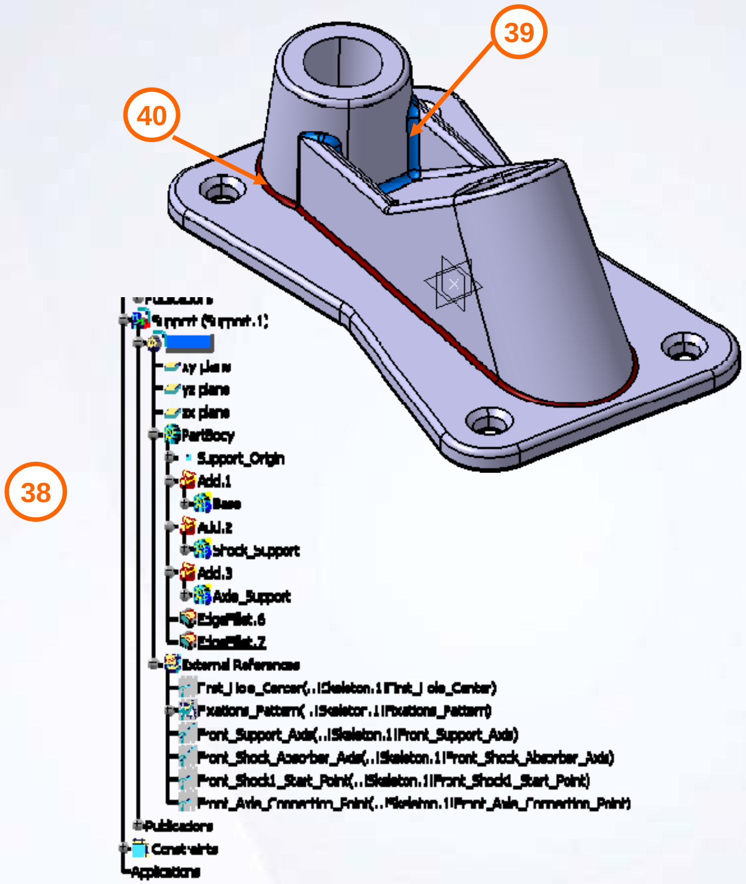
38. Add the three previously designed bodies in the PartBody.

39. Create an edge fillet.

- Create a [1mm] edge fillet.

40. Create an edge fillet.

- Create a [0.7mm] edge fillet.



Do It Yourself: Skateboard (13/17)

You must complete the following tasks (continued):

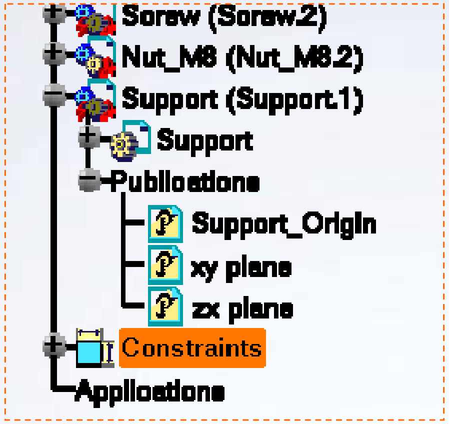
41. Open Skateboard_with_Skeleton.CATProduct.

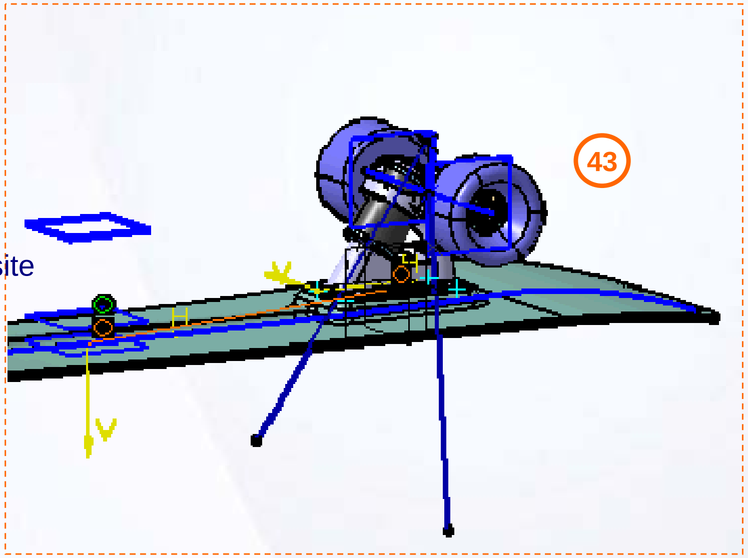
42. Insert Support.CATPart.

- If you did not complete the Support.CATPart from the previous steps, insert Support_Complete.CATPart instead.

43. Position the support component.

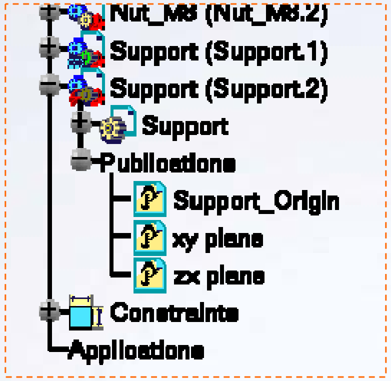
- Use published elements from the support and the skeleton to constrain the support.
  - a. Support_Origin with Front_Support_Middle_Point.
  - b. XY plane with Support_Plane, opposite direction.
  - c. ZX plane with ZX plane, same orientation.





Do It Yourself: Skateboard (14/17)

You must complete the following tasks (continued):

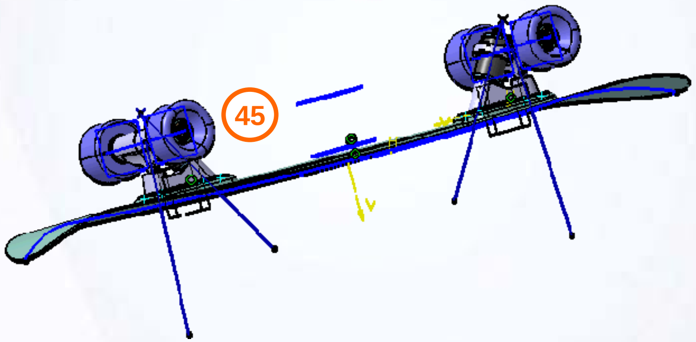


44. Validate links.

- From the support component's contextual menu, click Components > Define Contextual links. Ensure all links are in connected status and validate by selecting OK.

45. Insert a new instance.

- Insert a new instance of the Support part and position it at the rear of the skateboard using the following coincidence constraints:
  a. Support_Origin with Rear_Support_Middle_Point.
  b. XY plane with Support_Plane, opposite direction.
  c. ZX plane with ZX plane, opposite direction.



Do It Yourself: Skateboard (15/17)

You must complete the following tasks (continued):

46. Insert screw component.

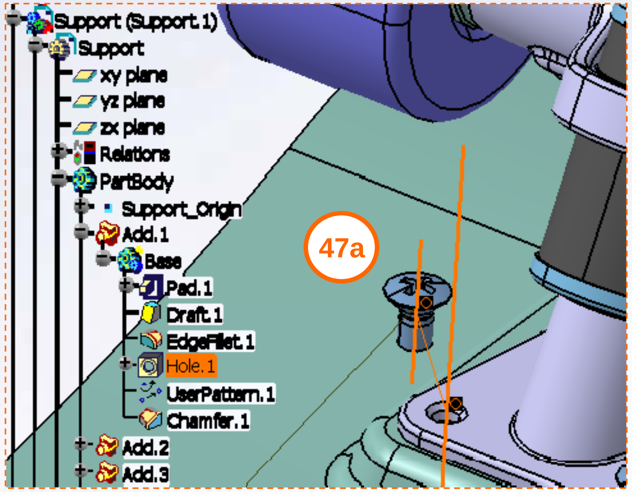
- Insert ISO_SCREW_M6x10-Z.CATPart.

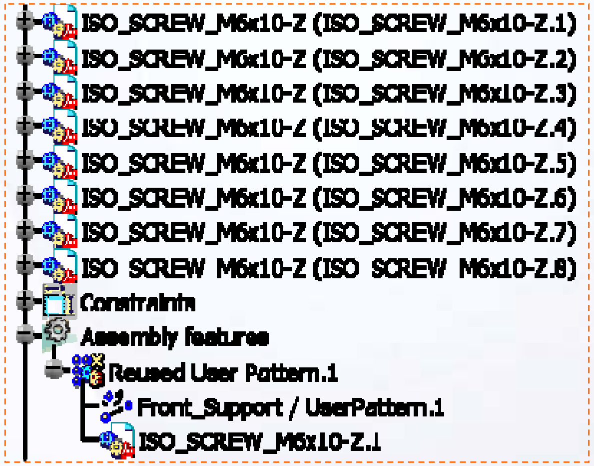
47. Position the screw.

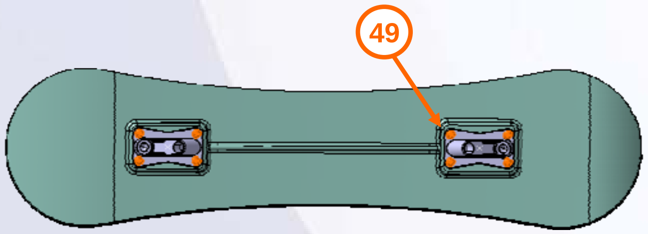
- Position the screw by creating the following constraints between the published elements:
  - a. Coincidence constraint between the Hole of the Support.1 and the axis of the Screw.
  - b. Offset of [-3mm] between Support_Plane in the skeleton and Mating_Plane in the screw, opposite orientation.

48. Instantiate the screw.

- Instantiate the screw by reusing the user pattern created in Support component during the base conception.







Do It Yourself: Skateboard (16/17)

You must complete the following tasks (continued):

49. Edit the skeleton.

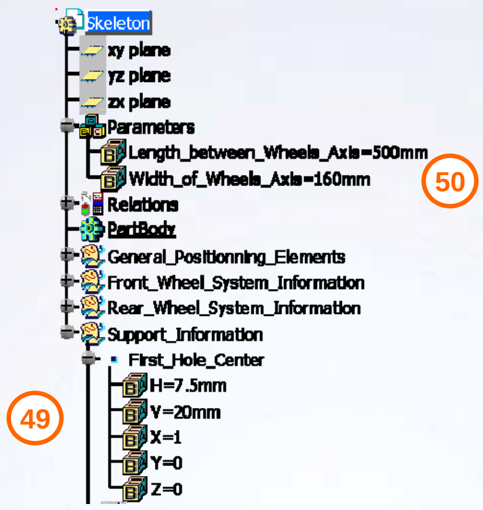
- Change the coordinates of First_Hole_Center point to:
  - a. H = 7.5mm.
  - b. V = 20mm.
- Notice that all the points of the fixation pattern are recalculated accordingly.

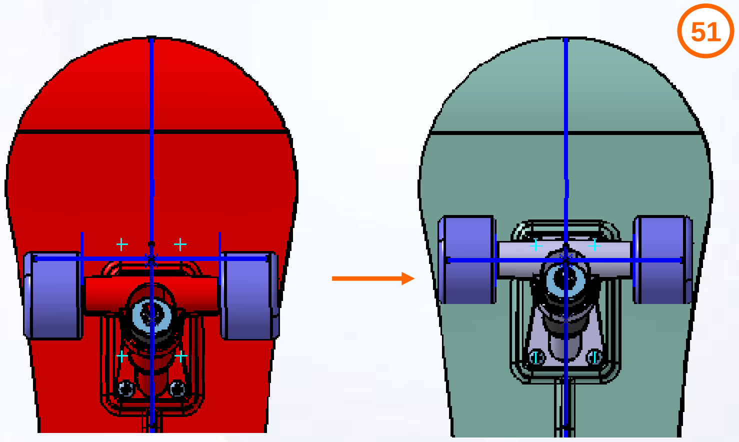
50. Edit the skeleton.

- Change the value of Length_between_Wheel_Axis to [500mm].
- The wheel axes and fixation center is moved.

51. Update the assembly.

- Update the assembly and notice the support geometry has been recalculated and the screws have been repositioned.





Do It Yourself: Skateboard (17/17)

You must complete the following tasks (continued):

52. Edit the skeleton.

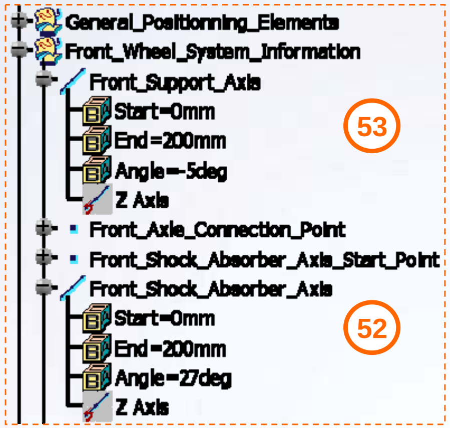
- Change the angle of Front_Shook_Absorber_Axis from [34deg] to [27deg].

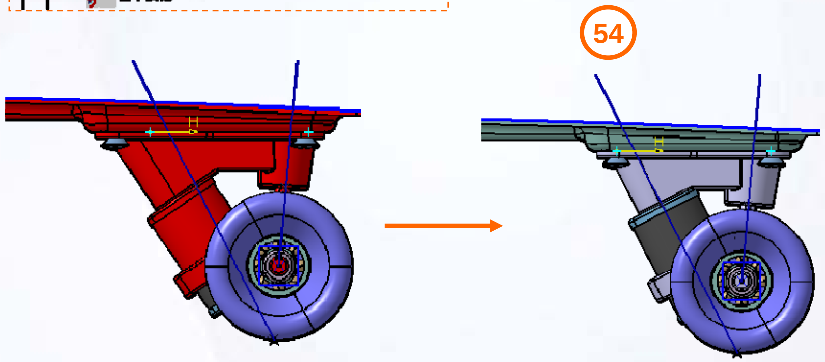
53. Edit the skeleton.

- Change the angle of Front_Support_axis from [-3deg] to [-5deg].

54. Update the assembly.

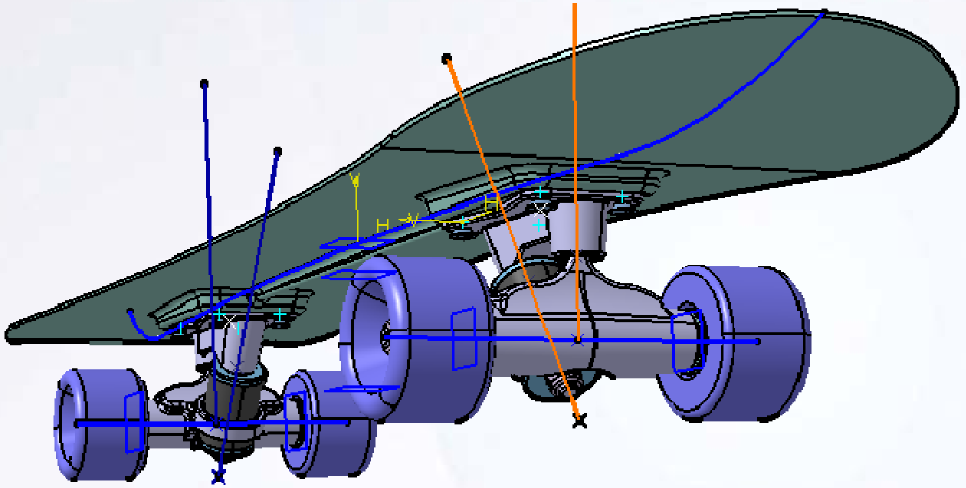
- Update the assembly and notice that the support's geometry has been recalculated again. Notice that the screws have been repositioned to meet the new design requirements.





Case Study: Complex Assembly Design Recap

- ✓ **Design in context using the skeleton method**
- ✓ **Publish geometry**
- ✓ **Position geometry using the skeleton method**
- ✓ **Propagate design changes**



Master Project

Lifting Truck

4 Hours

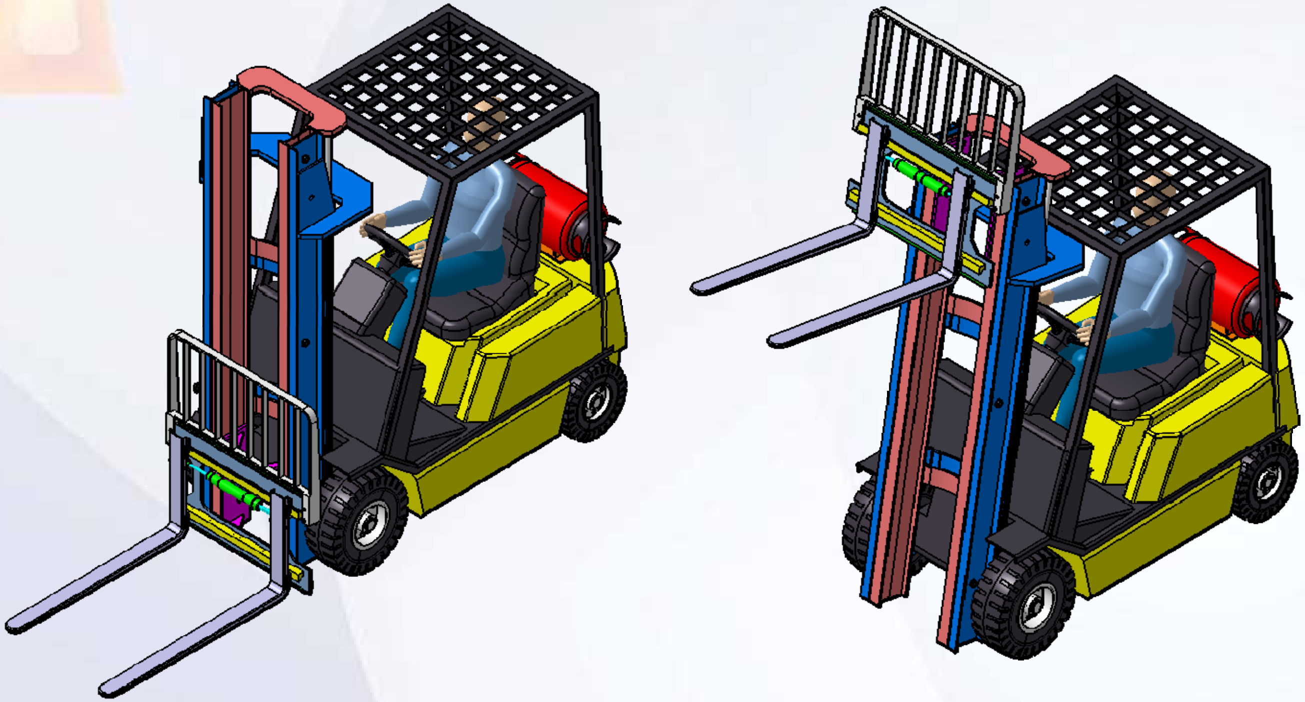
The lifting truck has double poles with a carriage in which geometries and positions are guided by skeleton. The two forks are to be added for lifting / handling material. The forks are to be designed in the context of a skeleton so that they can be easily modified in future. A pneumatic jack will drive the distance between the two forks. You are provided with V4 data of the pneumatic jack.

By the end of this Project you will be able to:

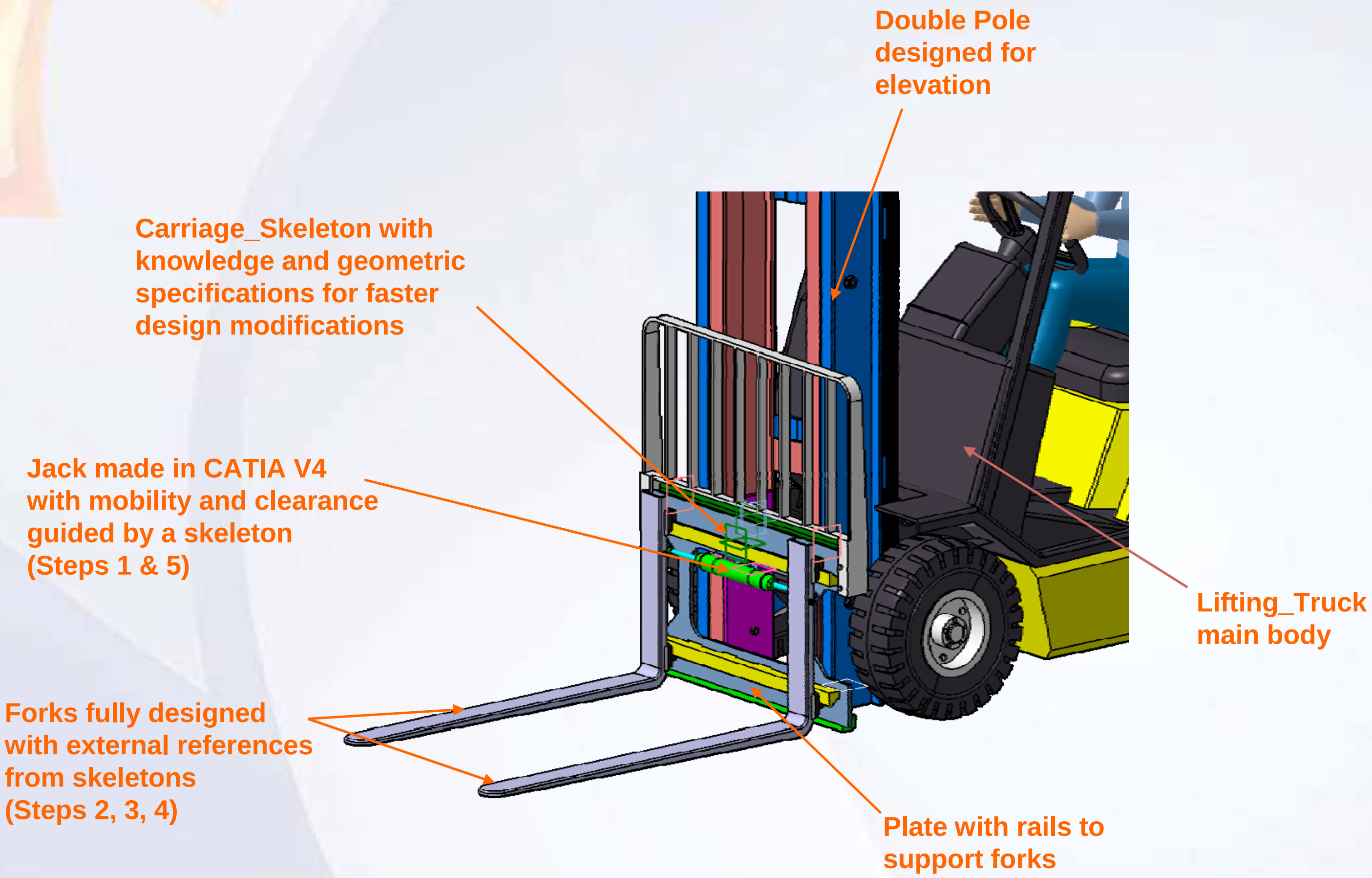
- Reuse V4 data and manage assembly constraints.
- Modify rigid component into flexible to add mechanical constraints.
- Create knowledge parameters and 3D specifications in skeleton structures, and use external references.
- Define a skeleton, then design parts in context, and modify parameters to watch automatic geometry updates.
- Analyze the assembly to check for clearance, and modify geometry by editing skeleton specifications.

Master Project: Lifting Truck

During this project, you will follow sequential steps from inserting specifications to mechanical analysis through the design of components in assembly context.



Master Project: Overview



Master Project : Reuse the Existing V4 Jack

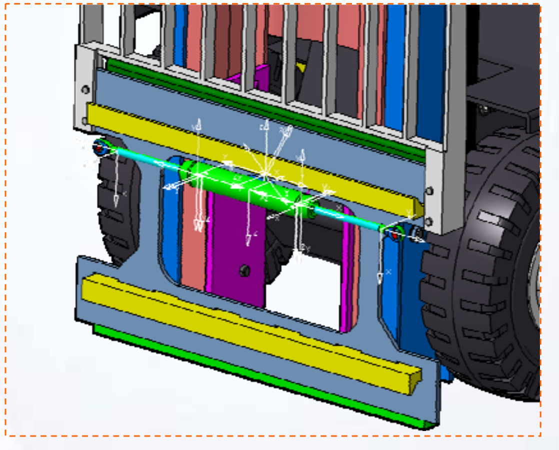
Lifting Truck

30 min

The objective of this step is to assemble the jack and to define parameters in the parent skeleton that will control the mobility of the jack. High-level instruction for this exercise is provided.

By the end of this step, you will be able to:

- **Load a V4 model of the jack and save it as CATProduct.**
- **Insert this jack into the correct sub-assembly of Lifting Truck.**
- **Constrain components of the jack.**
- **Assemble the jack onto the support plate.**
- **Instantiate bolts.**
- **Add geometric specifications driven parameters in the parent skeleton.**
- **Make the jack flexible.**

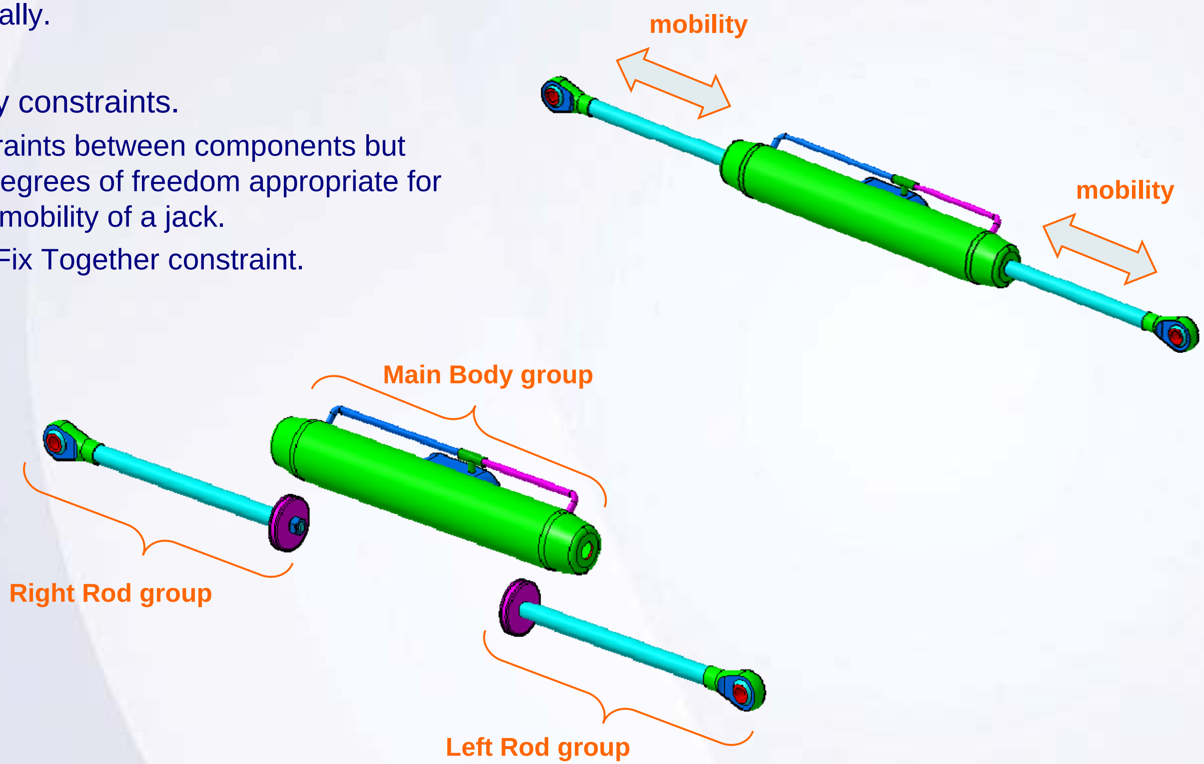


Master Project: Reuse the Existing V4 Jack (1/4)

Here is a list of required tasks to guide you:

1. Open the V4 assembly.
   - Open Fork_Jack_V4_Step1.session.
   - Save it locally.

2. Add assembly constraints.
   - Add constraints between components but keep the degrees of freedom appropriate for the actual mobility of a jack.
   - Avoid the Fix Together constraint.

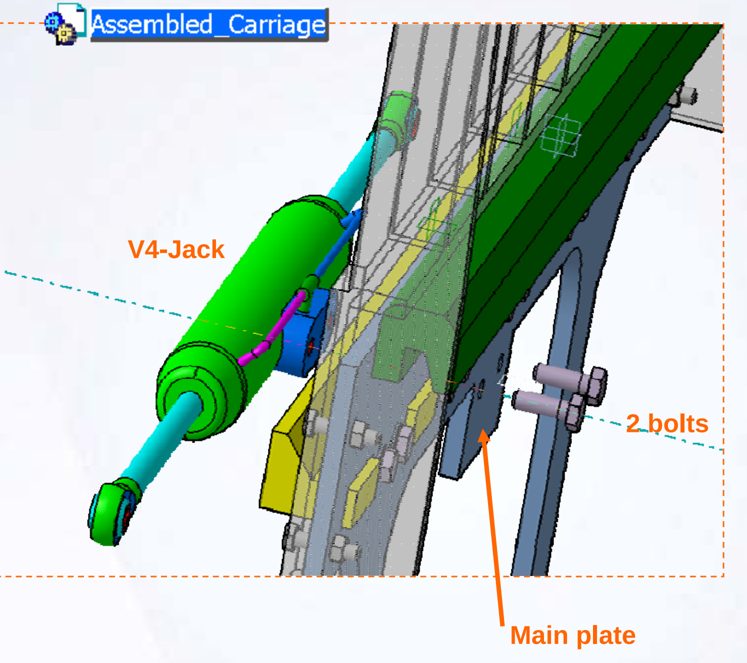


Master Project: Reuse the Existing V4 Jack (2/4)

Here is a list of required tasks to guide you (continued):

3. Open the top-level assembly.
   - Open Lifting_Truck_Step1.CATProduct.
   - Save it locally.

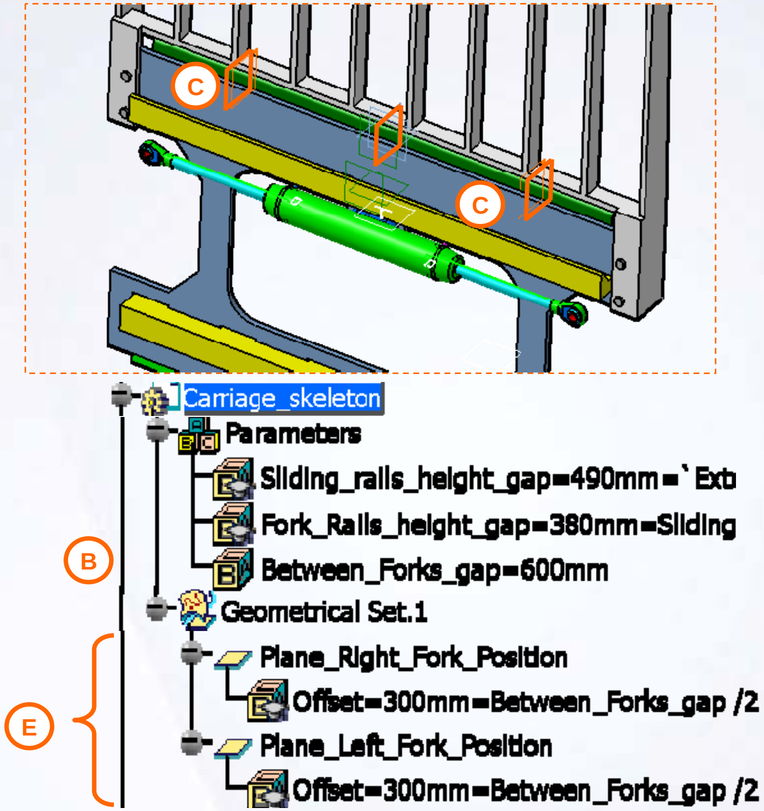
4. Insert the jack component.
   - Insert Fork_Jack_V4 into Assembled_Carriage sub-assembly.
   - Add constraints using planes of Carriage_Skeleton, and existing holes of the main plate.
   - Instantiate two bolts and constrain them. Use catalog bolt M8x30 or use the provided parts.



Master Project: Reuse the Existing V4 Jack (3/4)

Here is a list of required tasks to guide you (continued):

5. Add specifications to the parent skeleton that will be used to control the mobility of the jack.
   - A. Edit Carriage_Skeleton, which is in the Assembled_Carriage.CATProduct sub-assembly.
   - B. Add a parameter called between_forks_gap.
   - C. Add two planes offset from the ZX plane to define the forks' positions along Y-axis.
   - D. Publish these new elements.
   - E. Drive the plane offsets by adding formulas between the parameter and planes offset values.



Master Project: Reuse the Existing V4 Jack (4/4)

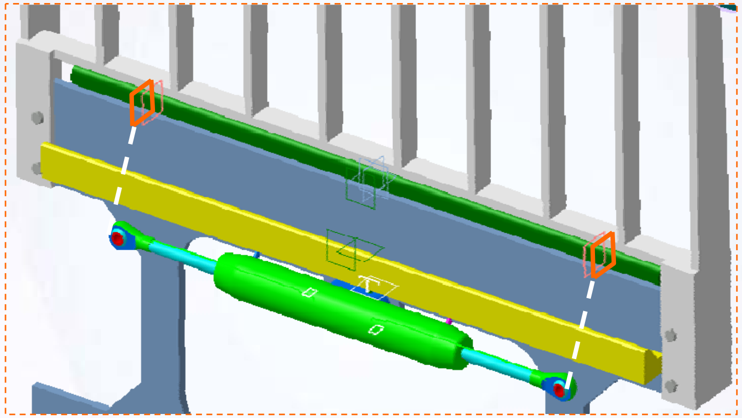
Here is a list of required tasks to guide you (continued):

6. Constrain the rods of the jack to the parent skeleton using a flexible sub-assembly.
   - Edit Assembled_Carriage.CATProduct.
   - Make V4_Jack flexible.
   - Add constraints between the rods' extremity axes and the new planes of Carriage_Skeleton.

7. Check the assembly by modifying the parameter.
   - Edit Carriage_Skeleton.CATPart.
   - Change the value of between_fork_gap from 600mm to 650mm.
   - Edit Assembled_Carriage.CATProduct and update. Both the rods should follow the modification.

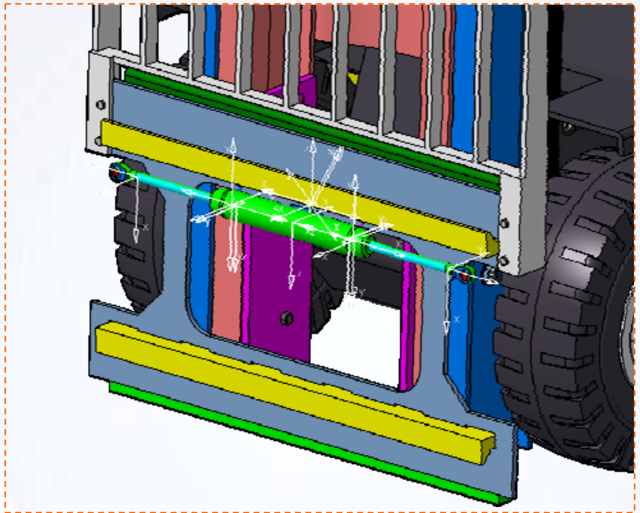
8. Add range values to the parameter.
   - Edit Carriage_Skeleton.CATPart.
   - Add range value to the between_fork_gap parameter (min = 525mm ; max = 750mm).

9. Save all.



Master Project: Reuse the Existing V4 Jack Recap

- ✓ Load a V4 model of the jack and save it as CATProduct
- ✓ Insert this jack into the correct sub-assembly of Lifting_Truck
- ✓ Constrain components of the jack
- ✓ Assemble the jack onto the support plate
- ✓ Instantiate bolts
- ✓ Add geometric specifications driven parameters in the parent skeleton
- ✓ Make the jack flexible



Master Project: Preparing for Design in Context

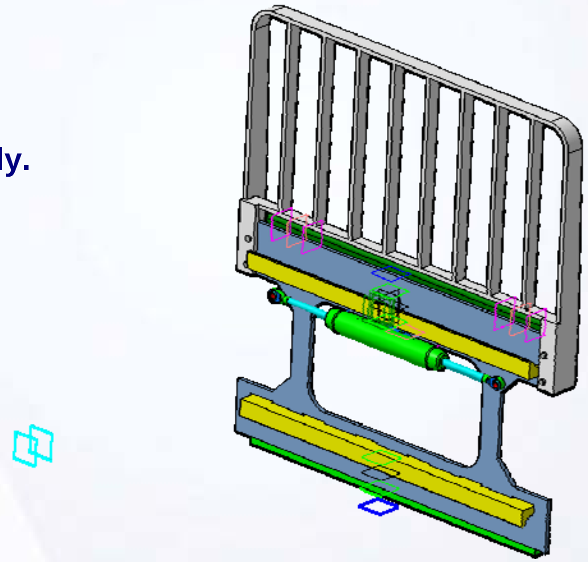
Lifting Truck

60 min

The objective of this step is to create a new skeleton to prepare the design of the forks in the following steps. High-level instructions for this exercise are provided.

By the end of this step, you will be able to:

- **Insert a skeleton in a new product in the carriage assembly.**
- **Constrain the new skeleton to the parent skeleton.**
- **Import the carriage assembly specifications from its skeleton into the new skeleton.**
- **Create knowledge parameters in the new skeleton.**
- **Add 3D geometry in the new skeleton using knowledge formulas.**

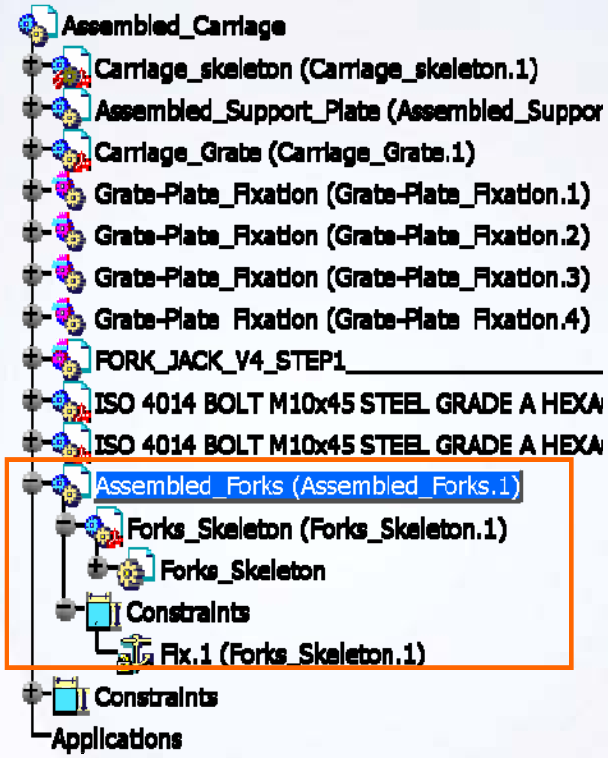


Master Project: Preparing for Design in Context (1/8)

Continue with the models used in step 1. If you did not complete step 1, use Lifting_Truck_Step2.CATProduct.

Here is a list of required tasks to guide you:

1. Insert a new sub-component in preparation for design-in-context of the forks.
   - Load Assembled_Carriage.CATProduct.
   - Insert a new product with a fixed new part (the skeleton) as shown in the specification tree.
   - Publish the XY, YZ, ZX planes of the skeleton.
   - Save all locally.

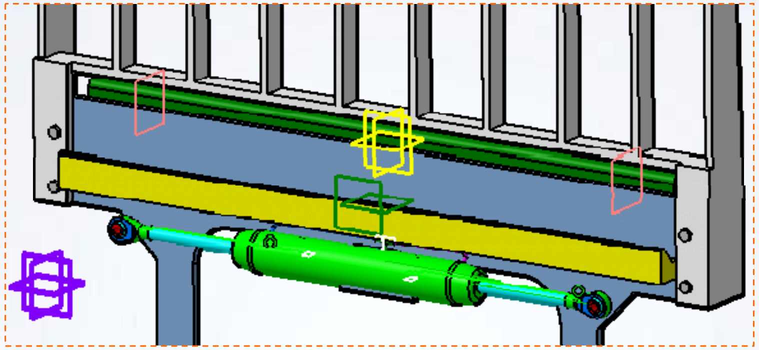


Master Project: Preparing for Design in Context (2/8)

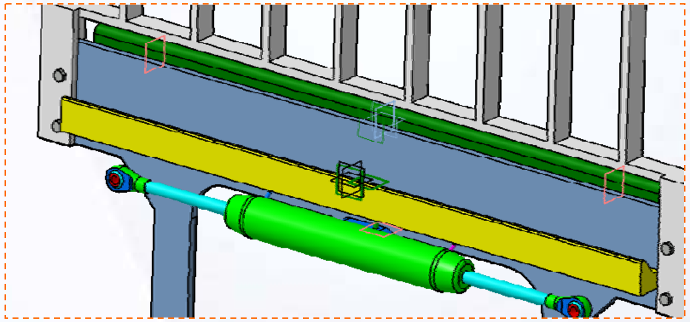
Here is a list of required tasks to guide you (continued):

2. You need to place the forks skeleton on the carriage skeleton before using external references for the design in context. Constrain the fork skeleton's origin with respect to the Top Rail contact planes.
   - Edit Assembled_Carriage.CATProduct.
   - Add constraints between the forks skeleton and Carriage_Skeleton as shown. The forks skeleton will be superimposed onto the parent skeleton.
   - Save all.

Before constraints:



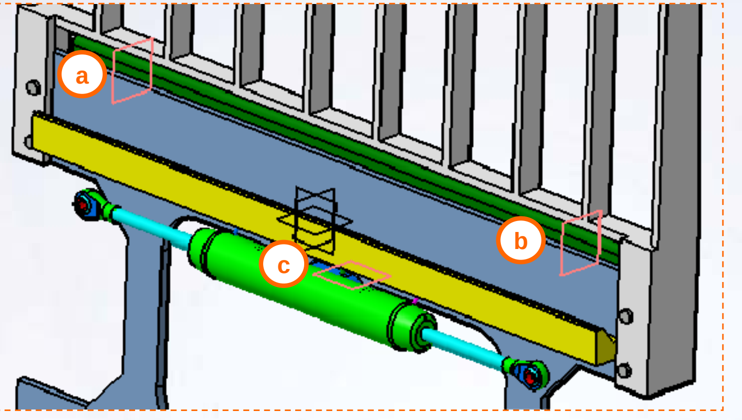
After constraints:



Master Project: Preparing for Design in Context (3/8)

Here is a list of required tasks to guide you (continued):

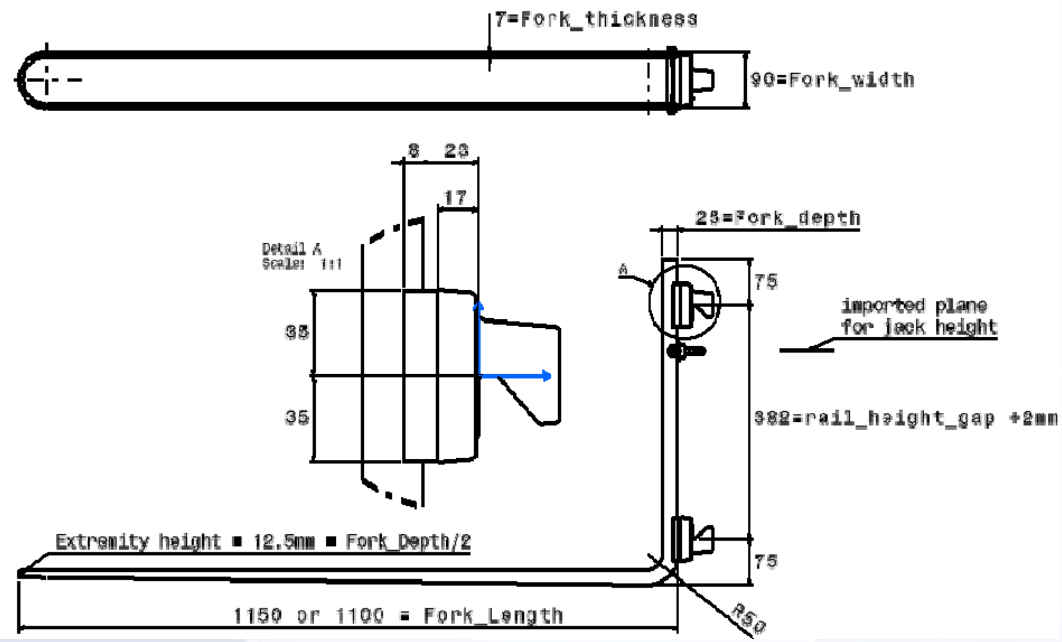
3. Import the external specifications into the forks skeleton.
   - Copy the following specifications of Carriage_Skeleton:
     a. Plane_Right_Fork_Position
     b. Plane_Left_Fork_Position
     c. Plane_Fork_Jack_Height_Position
     d. Fork_Rail_height_gap parameter
   - Paste them as result with link into the forks skeleton.
   - Publish these external features in the forks skeleton.
   - Save all.



Master Project: Preparing for Design in Context (4/8)

To help, here is a list of required tasks (continued):

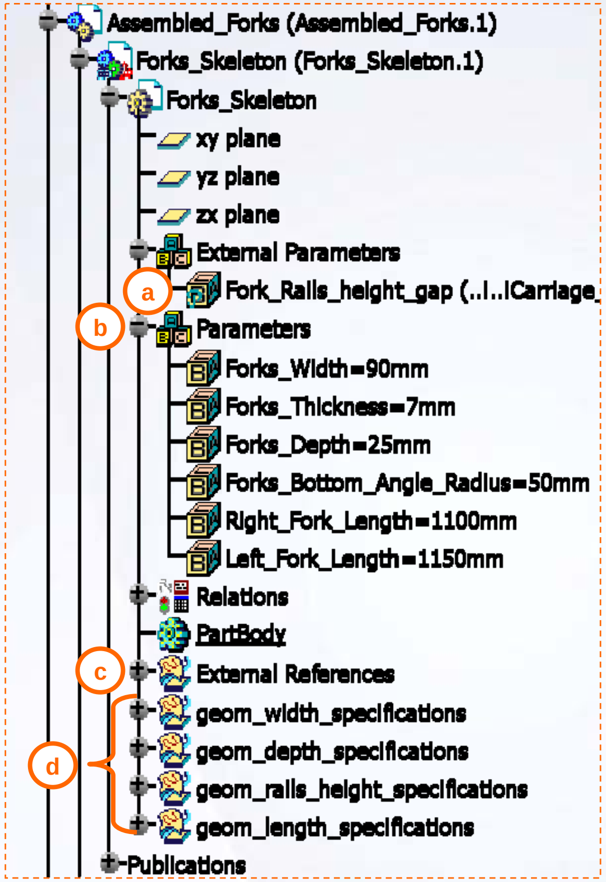
4. Identify the parameters that must be inserted in the forks skeleton.
   - Study the dimensions of the assembly. All dimensions are in mm.
   - Identify the required parameters that you will insert into the forks skeleton. You will create the required 3D geometry driven by those parameters.



Master Project: Preparing for Design in Context (5/8)

To help, here is a list of required tasks (continued):

5. Create the parameters in the forks skeleton.
   - Create the following tree structure in the forks skeleton.
     a. External parameters contain the imported parameter from Carriage_Skeleton.
     b. Parameters contain all the parameters you have to create. Publish them.
     c. External references contain the geometric specifications imported from Carriage_Skeleton.
     d. You can create new geometrical sets in order to sort the geometric specifications you are about to create.
     e. Do not forget to publish all the specifications you will create.



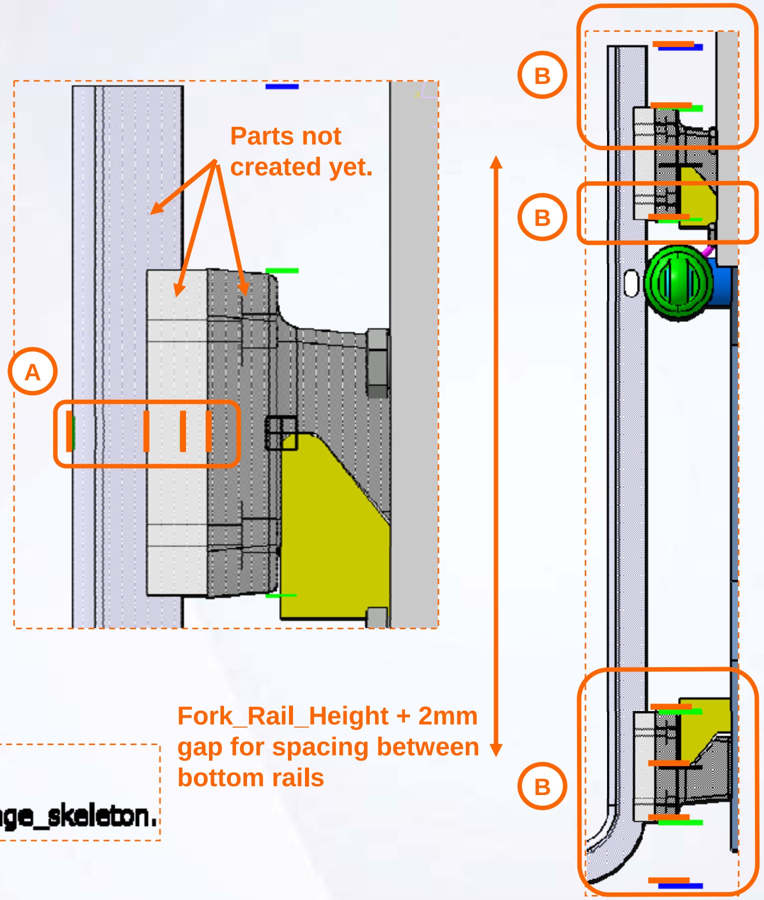
Master Project: Preparing for Design in Context (6/8)

To help, here is a list of required tasks (continued):

6. Create the forks skeleton's 3D geometry.
   A. In the geom_depth_specifications geometric set, create four planes using the previously given 2D drawing and depth parameter.
   B. In the geom_rails_height_specifications geometric set, create seven planes using the previously given 2D drawing and the Rail_Height external parameter.

Forks_Depth=25mm

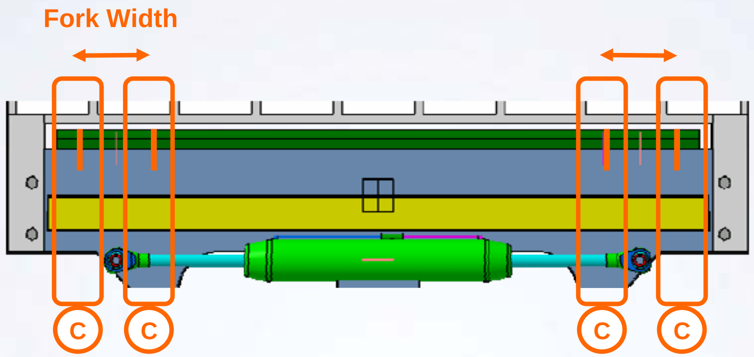
External Parameters
Fork_Rails_height_gap (..!..!Carriage_skeleton.

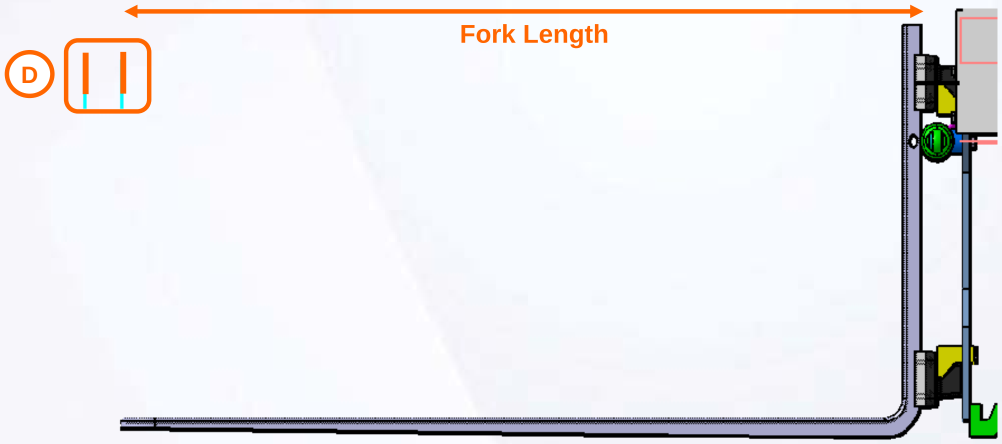


Master Project: Preparing for Design in Context (7/8)

To help, here is a list of required tasks (continued):

7. Create the forks skeleton's 3D geometry (continued).
   - C. In the geom_width_specifications geometric set, create four planes using the previously given 2D drawing, width parameter, and imported planes.
   - D. In the geom_length_specifications geometric set, create two planes using the previously given 2D drawing and parameters.

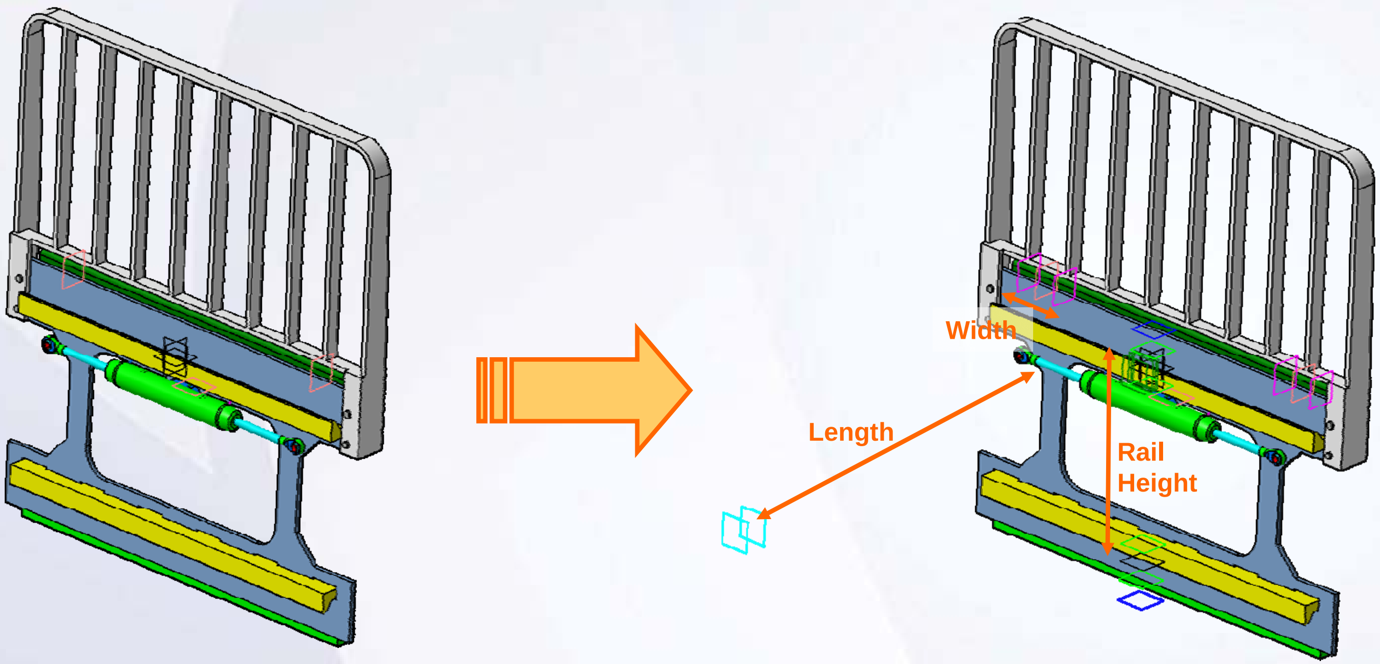




Master Project: Preparing for Design in Context (8/8)

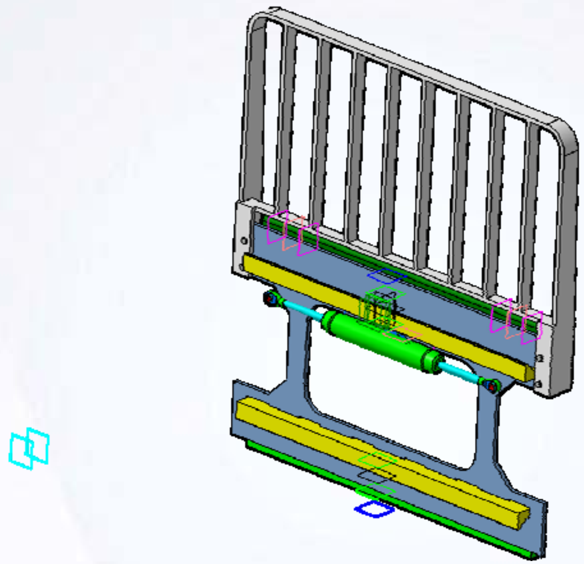
To help, here is a list of required tasks (continued):

8. Create the forks skeleton's 3D geometry (continued).
    - Upon completion of this step you should get this result.



Master Project: Preparing for Design in Context Recap

- ✓ Insert a skeleton in a new product in the carriage assembly
- ✓ Constrain the new skeleton to the parent skeleton
- ✓ Import the carriage assembly specifications from its skeleton into the new skeleton
- ✓ Create knowledge parameters in the new skeleton
- ✓ Add 3D geometry in the new skeleton using knowledge formulas



Master Project: Design of Right Fork Assembly

Lifting Truck

90 min

The objective of this step is to design the right fork assembly by using specifications of the forks skeleton created in previous step. High-level instruction for this exercise is provided.

By the end of this step, you will be able to:

- **Design new parts in the context of the skeleton.**
- **Insert additional components.**

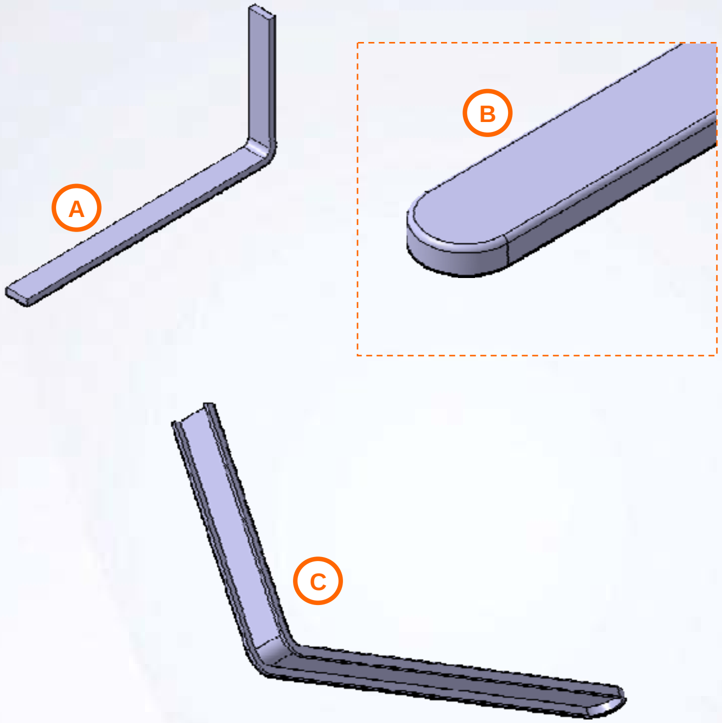
Master Project: Design of Right Fork Assembly (1/6)

Continue with the models used in step 2. If you did not complete step 2, use Lifting_Truck_Step3.CATProduct.

Here is a list of required tasks to guide you:

1. Create a new part in Assembled_Forks.CATProduct.
   A. Insert a new CATPart called Right_Fork.
   B. Publish its planes.
   C. Constrain it onto the forks skeleton to place it on the right side of fork assembly.

2. Use the Part Design workbench to design its shape with respect to the skeleton.
   A. Use a rib with two sketches to create the profile.
   B. Add a tritangent fillet and edge fillets.
   C. Apply a shell with a parameterized thickness.



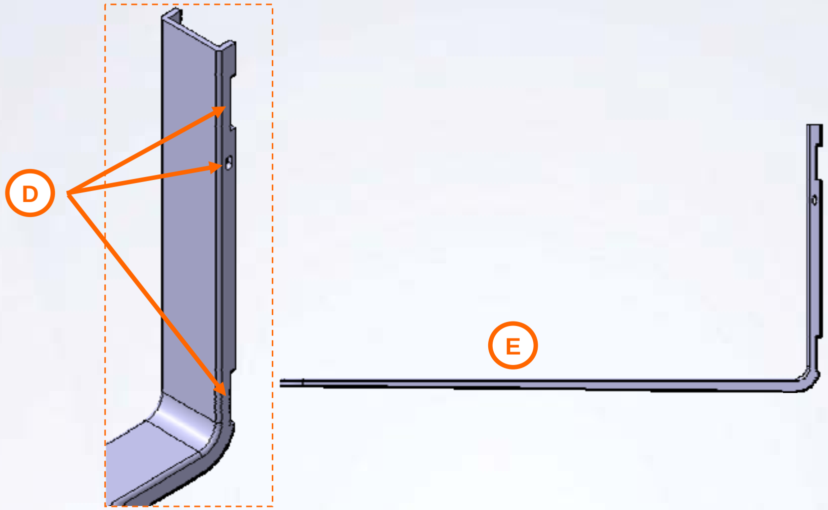
Master Project: Design of Right Fork Assembly (2/6)

Here is a list of required tasks to guide you (continued):

D. Add pockets for placing Fork_Plates and Fork_Mobile_Axis.

E. Remove material to obtain the bottom slope.

F. Use the Part Design workbench to design its shape with respect to the skeleton.

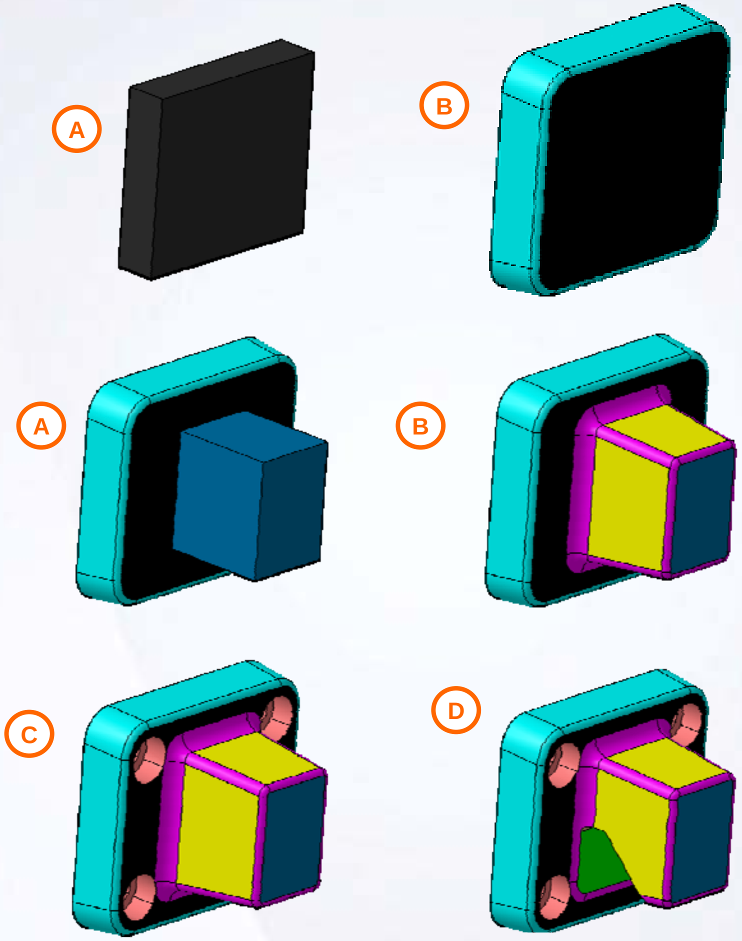


Master Project: Design of Right Fork Assembly (3/6)

Here is a list of required tasks to guide you (continued):

3. Design the molded hook up part. Assume the design will not be associative with the product specification, and therefore does not require any external references.
    - Insert a new CATPart called Hook_Up.CATPart in Assembled_Forks.CATProduct.
    - Publish its planes.
    - Constrain it onto the skeleton to place it on the right side.

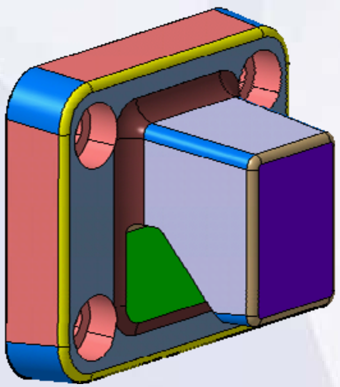
4. Use the Part Design workbench to design its shape.
    A. Create pads.
    B. Create drafts and fillets.
    C. Create counterbored holes.
    D. Create a pocket.

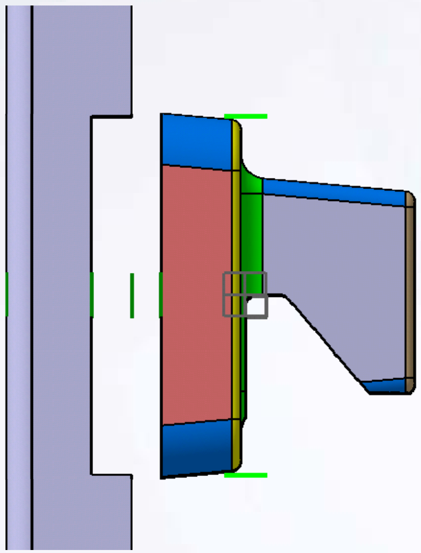


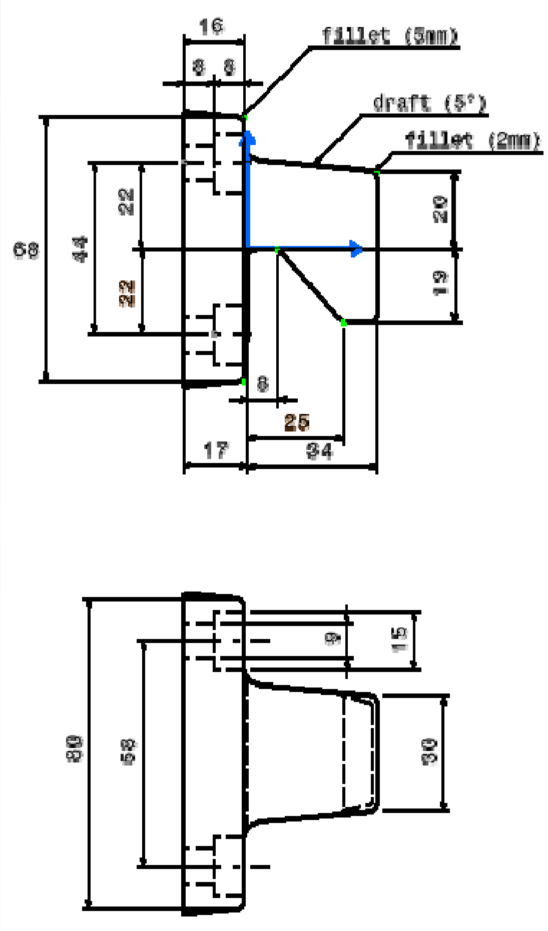
Master Project: Design of Right Fork Assembly (4/6)

Here is a list of required tasks to guide you (continued):

4. Use the Part Design workbench to design its shape (continued).





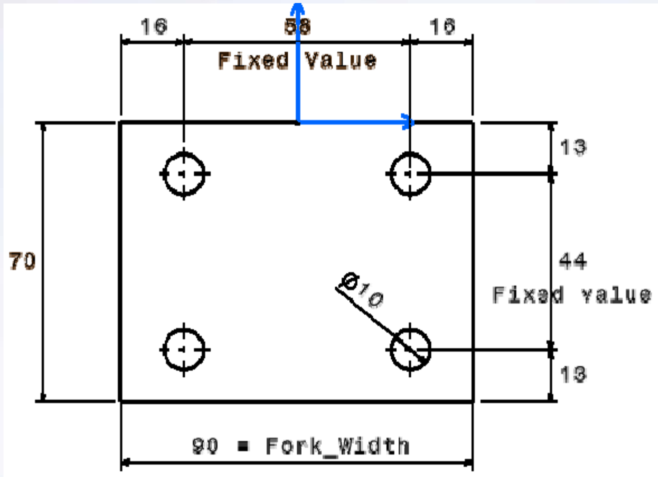


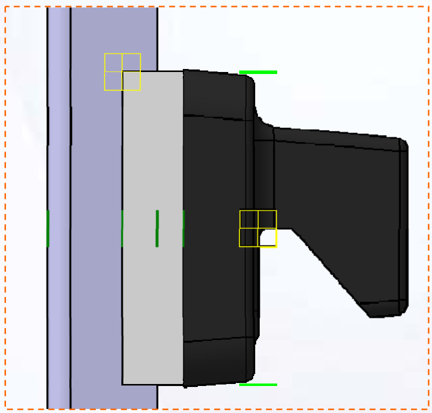
Master Project: Design of Right Fork Assembly (5/6)

Here is a list of required tasks to guide you (continued):

5. Design the the fork plate in the context of the skeleton created in Step 2.
   - Insert a new CATPart called Fork_Plate.CATPart in Assembled_Forks.CATProduct.
   - Publish its planes.
   - Constrain it onto the skeleton to place it on the right side.

6. Use the Part Design workbench to design its shape with respect to the skeleton.
   - Create a pad.
   - Create holes. The position of the holes should not follow any parameter as those in Hook_up holes do not.



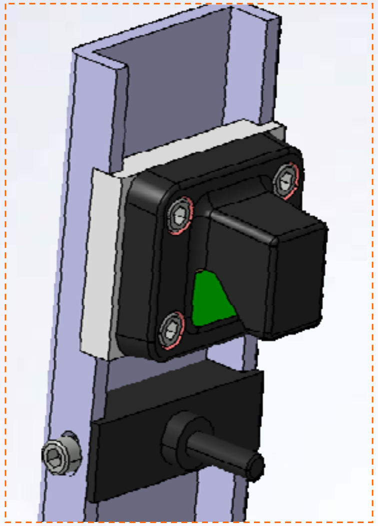


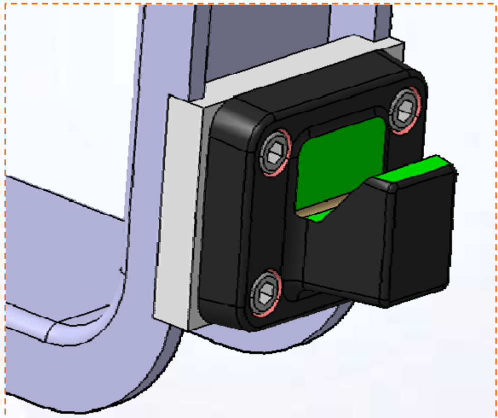
Master Project: Design of Right Fork Assembly (6/6)

Here is a list of required tasks to guide you (continued):

7. Assemble the remaining components.
   A. Insert a screw (M8x25 CHC from catalog, or use the provided one) and add constrain it to the existing parts.
   B. Repeat this for the four screws.
   C. Load Fork_Mobile_Axis.
   D. Save it locally.
   E. Drag and drop it into Assembled_Fork.
   F. Add constraints with respect to the skeleton.
   G. Add two screws.
   H. Copy/Paste Fork_Plate, Hook_Up, and the four screws.
   I. Move these components and add constraints to place them onto the bottom notch of Right_Fork.

8. Save all.





Master Project: Design of Right Fork Assembly Recap

- ✓ Design new parts in the context of the skeleton
- ✓ Insert additional components

Master Project: Design of Left Fork Assembly

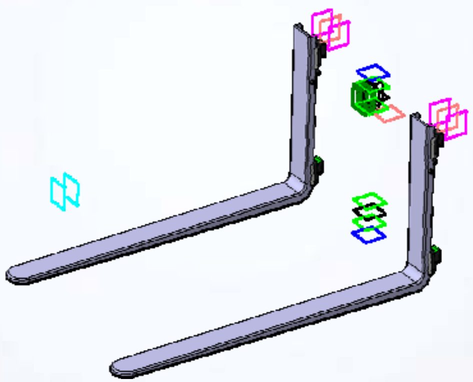
Lifting Truck

45 min

The objective of this step is to design the left fork using the PowerCopy tool from the right fork. You will then instantiate other components and constrain them in the assembly. High-level instruction for this exercise is provided.

By the end of this step, you will be able to:

- **Insert a new empty part and constrain it onto the skeleton.**
- **Create the right fork's geometry PowerCopy.**
- **Use right fork's design PowerCopy in this new part by changing some geometrical references.**
- **Instantiate all components needed for the assembly.**
- **Modify Skeleton's parameters and notice automatic geometry modifications.**



Master Project: Design of Left Fork Assembly (1/3)

Continue with the models used in step 3. If you did not complete step 3, use Lifting_Truck_Step4.CATProduct.

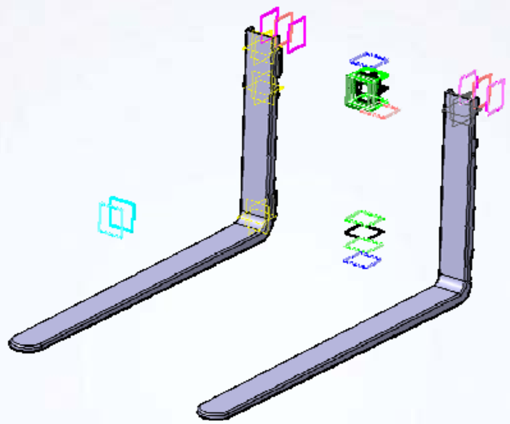
Here is a list of required tasks to guide you:

1. Create a new part in Assembled_Forks.
   - Insert a new CATPart called Left_Fork.
   - Publish its planes and constrain it to the forks skeleton on the left side.

2. Create a PowerCopy of Right_Fork.
   - Edit Right_Fork and PowerCopy all elements of PartBody (features and sketches) and Relations (formulas).
   - Save the PowerCopy in a new local catalog.

3. Use the PowerCopy to design the Left_Fork.
   - Edit Left_Fork and use the catalog browser to instantiate the PowerCopy.
   - Replace the external references for the width and length definitions. Select the same specifications for all other external references from the skeleton.
   - Validate. Left_Fork is now created.

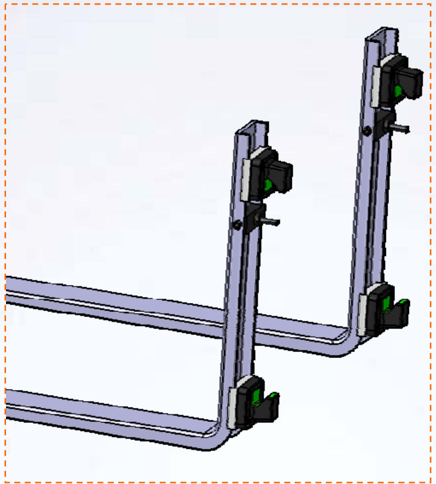
4. Save all.

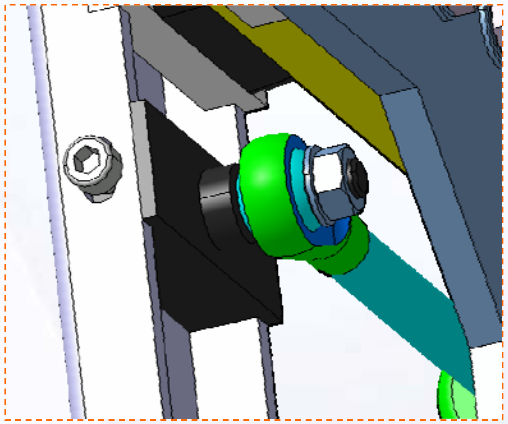


Master Project: Design of Left Fork Assembly (2/3)

Here is a list of required tasks to guide you (continued):

5. Instantiate the elements required for mechanical assembly. Left_Fork must be exactly like Right_Fork.
   - Instantiate the following in Assembled_Forks.CATProduct:
     - 2 Hook_Up
     - 2 Fork_Plate
     - 1 Fork_Mobile_Axis
     - 6 Screws
   - Add constraints.
   - Save all.
   - Instantiate the following in Assembled_Carriage.CATProduct:
     - 2 washers (M10), are already existing in the local files
     - 2 nuts (M10)
   - Add constraints.
   - Save all.

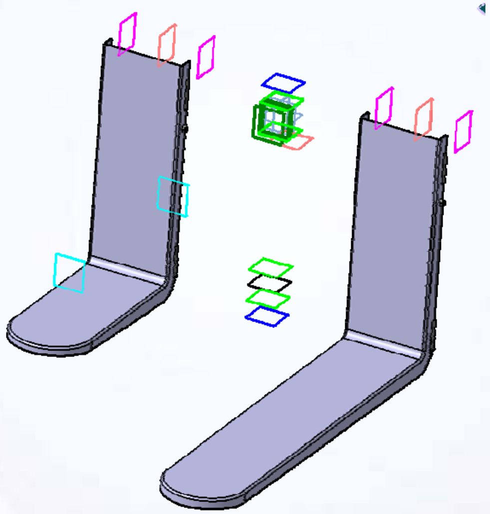




Master Project: Design of Left Fork Assembly (3/3)

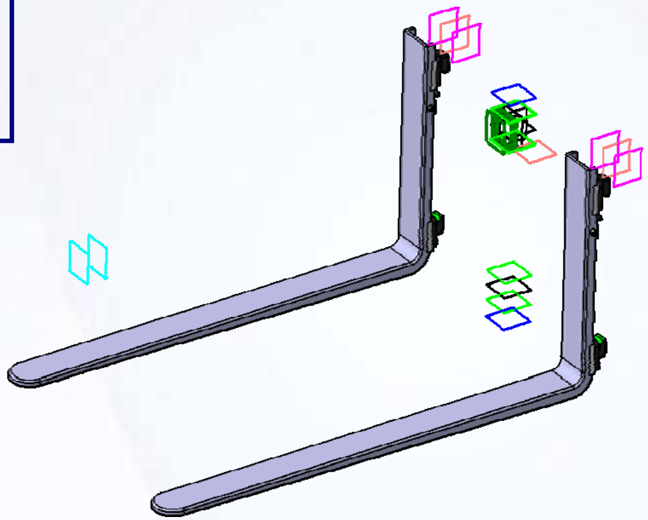
Here is a list of required tasks to guide you (continued):

6. Modify the parameters in the skeleton to check if the geometry is properly driven by the wireframe reference elements.
   - Note that Fork_Depth must not exceed Fork_Bottom_Angle_Radius.
   - Note that Fork_Bottom_Angle_Radius must not exceed 50mm as bottom rail position is not dependant of this parameter.
   - Do not save those last modifications.



Master Project: Design of Left Fork Assembly Recap

- ✓ Insert a new empty part and constrain it onto the skeleton
- ✓ Create the right fork's geometry PowerCopy
- ✓ Use right fork's design PowerCopy in this new part by changing some geometrical references
- ✓ Instantiate all components needed for the assembly
- ✓ Modify Skeleton's parameters and notice automatic geometry modifications



Master Project: Jack Clearance Check

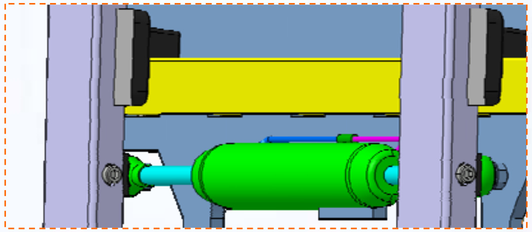
Lifting Truck

15 min

The objective of this step is to make sure that the 10mm clearance between "V4 Jack" and "top rail" is respected. High-level instruction for this exercise is provided.

By the end of this step, you will be able to:

- **Analyze the clearance between the jack and the fork guide (i.e., the supporting top rail).**
- **Perform a modification to the correct skeleton specification to achieve the clearance.**
- **Restart the analysis to check the impact of the design change.**

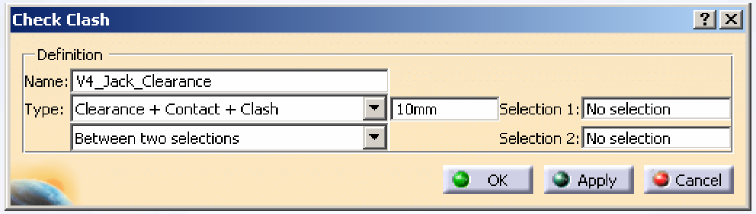


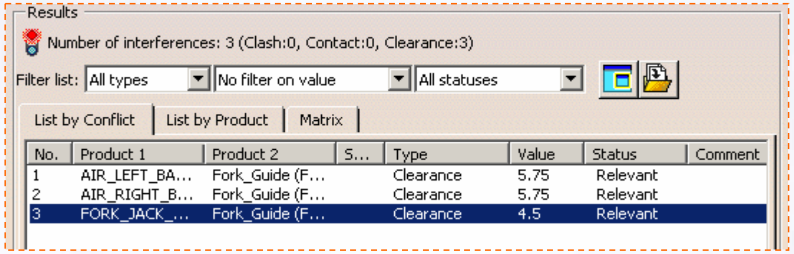
Master Project: Jack Clearance Check (1/3)

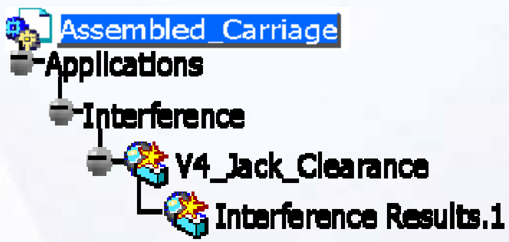
Continue with the models used in Step 4. If you did not complete Step 4, use Lifting_Truck_Step5.CATProduct.

Here is a list of required tasks to guide you:

1. Create a clearance analysis in Assembled_Carriage.CATProduct.
   - Create a clearance analysis as shown using Fork_Guide (Fork_Guide.1) and Fork_Jack_V4.CATProduct (Fork_Jack_V4_STEP1________SESSION) as the selections.
   - Apply it. You have three results for which distance values are less than the required clearance.
   - Select one of them. A new window shows a 3D preview of concerned sub-elements of the selections.
   - Select the interference with the lowest spacing value.
   - Click OK. The spacing value appears in the 3D view. The analysis appears in the tree under the Applications branch.



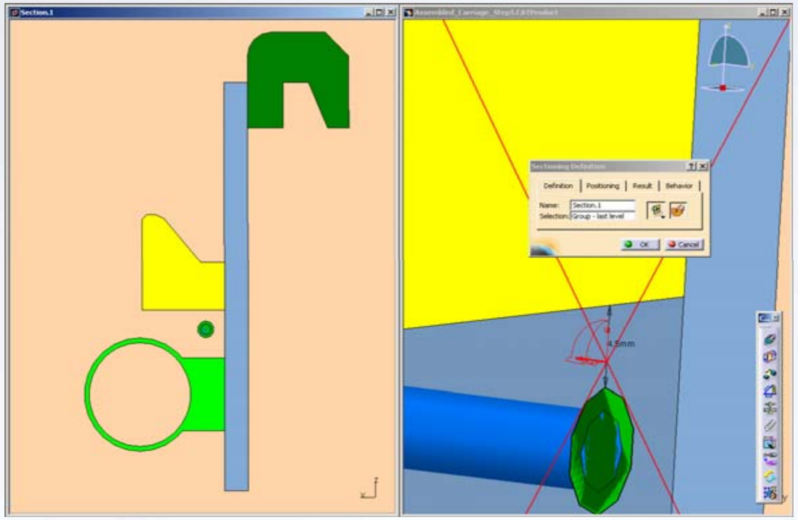


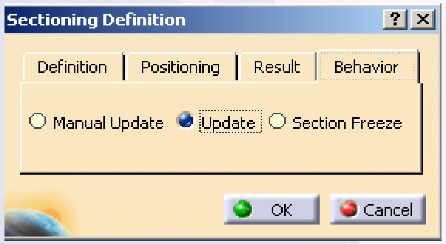


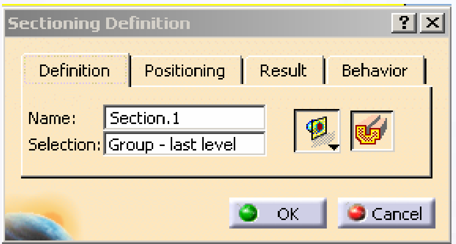
Master Project: Jack Clearance Check (2/3)

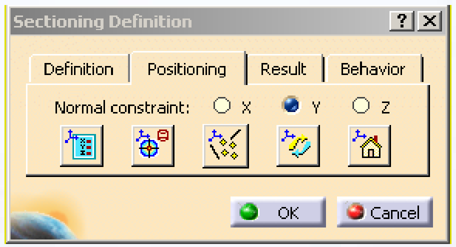
Here is a list of required tasks to guide you (continued):

2. Use a sectioning view to gain a better view of the clearance, and of the geometric modification you will make.
   - Create a sectioning view in In Assembled_Carriage.CATProduct.
   - Cut the 3D view along section view.
   - Place section view on the gap dimension.





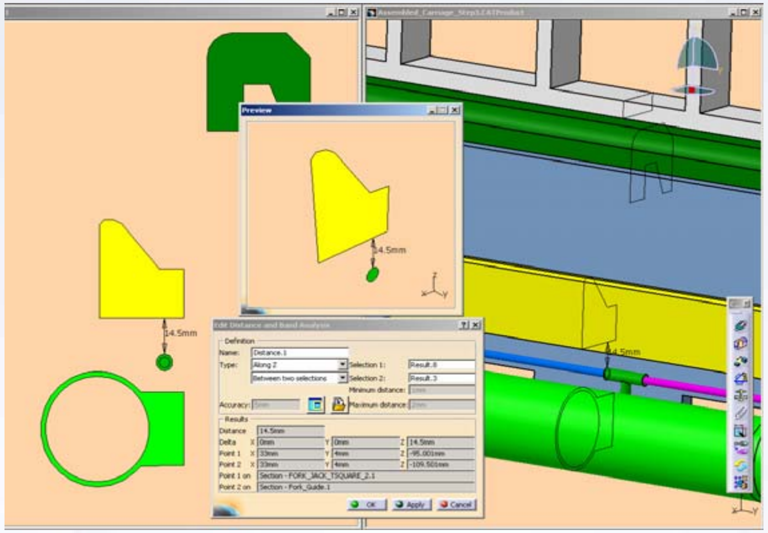


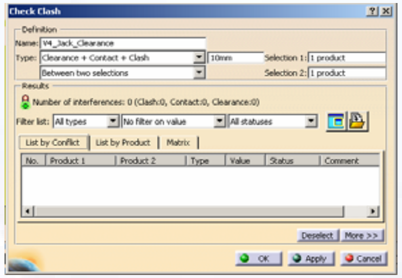


Master Project: Jack Clearance Check (3/3)

Here is a list of required tasks to guide you (continued):

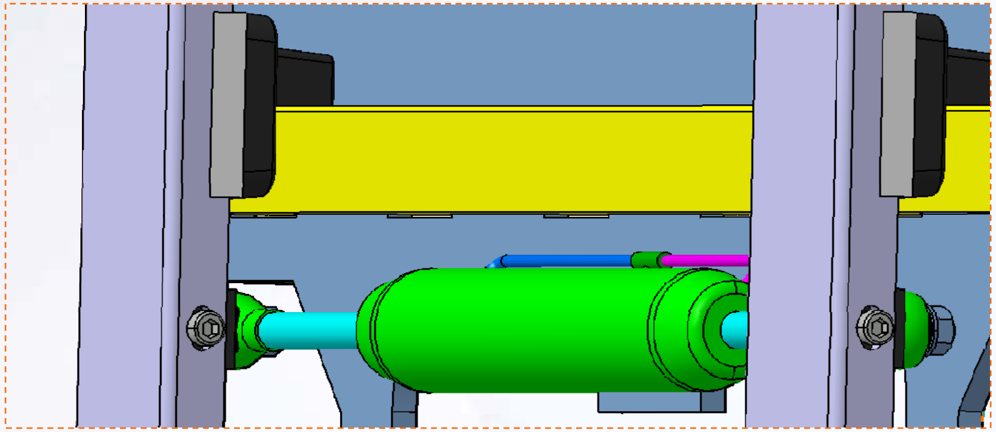
3. Modify one specification in the skeleton to correct the clearance around V4_Jack.
   - In Assembled_Carriage.CATProduct, edit Carriage_Skeleton and increase the offset value of Plane_fork_Jack_height_position by 10mm.
   - Update the assembly.
   - Notice that the following elements have been automatically modified.
     - Fork support plate
     - Right fork & left fork
     - Jack height position
     - Bolts constrained with those elements
     - The section view is also updated
   - Edit the clearance analysis to re-apply with the same parameters. No interference must be detected now. You can use a Distance & Band Analysis to measure the new distance.
4. Save All





Master Project: Jack Clearance Check Recap

- ✓ Analyze the clearance between the jack and the fork guide (i.e., the supporting top rail)
- ✓ Perform a modification to the correct skeleton specification to achieve the clearance
- ✓ Restart the analysis to check the impact of the design change



Shortcuts

F1	Link to on-line documentation	**Ctrl** + several selections	Multiple selection
Shift F1	Contextual help for an icon	**Shift** + 2 selections	Selection of all elements between and including the 2 selected elements
Shift F2	Overview of the specification tree		
F3	Hide/Show the specification tree		
Ctrl + Tab	Change CATIA V5 window	**Alt** F8	Macros
Ctrl N	New file	**Alt** F11	Visual Basic editor
Ctrl O	Open file	**Alt** + Enter	Properties
Ctrl S	Save file	**Alt** + MB1	Pre-selection Navigator
Ctrl P	Print	**Ctrl** F11	Pre-selection Navigator
Ctrl Z	Undo	**Up/Down** or **Left/Right** arrow	Pre-selection Navigator
Ctrl Y	Redo	**Shift** + MB2	Local zoom and change of viewpoint
Ctrl C	Copy		
Ctrl V	Paste	**Shift** + manipulation with compass	Displacement respecting constraints
Ctrl X	Cut		
Ctrl U	Update		
Ctrl F	Find		

Glossary

Activate: It is an operation of making a component active by double-clicking on it.

Add: It is a Boolean operation in which features in another body are added to the current body.

Affected Part: This is a part which is affected by the Boolean operation and will undergo change in its geometry as a result of this operation.

Affinity: It is an operation in which an element is transformed by applying X, Y, Z affinity ratios with respect to a reference axis system.

Assemble: It is a Boolean operation in which a union will be performed between the two bodies.

Assembly feature: It is a feature created by assembly Boolean operations (split, hole, pocket).

Associativity: It is a term which represents interdependent relationships between entities.

Bill of material: It is a list of data about the components contained in the active component.

Boolean operation: It is an operation in which two or more bodies are added, removed, split, trimmed.

Cache: It is an option by which you can enable the visualization mode wherein light CGR models will be loaded while opening the assembly in CATIA.

Center Curve: It is a tangency continuous curve along which a rib or a slot feature is computed.

CGR: It is a CATIA Graphical Representation (CGR) file type.

Child feature: It is a feature originating from an original feature (parent feature).

Closing Points: It is the end point of the profile Component.

Concurrent engineering: It is a practice in which the design of various parts in an assembly is done concurrently by various designers and final assembly is created after integrating these parts.

Glossary

Construction geometry: It is a set of geometric elements used in the sketch profile as construction elements and are useful in constraining / creating main sketch profile.

Coupling: It is an option in Multi-section Solid definition. The coupling decides the shape of the multi-section solid computed between the two sections.

Define in work object: It is making the current feature active.

Degree of freedom: It is a numerical value indicating the number of ways a part can move in the 3D space.

Design mode: It is a mode in which the geometrical representations of the assembly and its parts are loaded and accessible for review and modification.

Design Table: It is a table containing various configurations of design created by a specifying a set of parameters for each design.

DMU: It is an abbreviation for Digital Mock-Up.

Draft: It is a feature provided with a face with an angle and a pulling direction.

Driving part: It is a part whose value is determined by an external parameter and is driven by this external parameter.

Driving property: It is a property whose value is determined by an external parameter and is driven by this external parameter.

Exploded state: It is a state where the assembly parts are in an exploded condition.

Extrapolate: It is an operation in which an element is extended a specified amount while respecting tangency or curvature conditions. Typically a surface boundary can be selected for in order to extrapolate the surface a specified length.

Extrude: It is a surface created by extruding a 2D/3D Profile within the defined extrusion limits.

Glossary

Feature: It is a component of a part.

Fillet: It is a curved surface of a constant or variable radius that is tangent to and joins two surfaces. Together these three surfaces form either an inner or outer corner.

Healing: It is a process of repairing small gaps between two adjacent surfaces.

Hybrid Design: It is a design approach where Wireframe and Surface Geometry and Part Design features can be created in the same body.

IGES: It is a file format named for Initial Graphics Exchange Specifications.

Instantiate: It is an operation of creating a new instance of a component or its features from same or different documents.

Join: It is a wireframe or surface created by joining adjacent wireframe or surface elements.

Load: It is an operation in which the document (CATPart, CATProduct, XLS) is loaded in CATIA.

Multi-Body Method: It is an approach where the design of a complex part is split into separate bodies and then the final geometry is obtained by performing Boolean operations on the bodies.

Multi-model link: It is a link referring to another part.

Multi-section Surface: It is a surface that is obtained by sweeping two or more planar section curves along a spine, which may be automatically computed or user-defined.

Multi-sections Solid: It is a solid created by computing a solid through the profiles in multiple planes.

Non-associativity: It is a term which represents no interdependent relationship between entities.

Offset surface: It is a surface that is obtained by offsetting an existing surface a specified distance.

Glossary

Parent feature: It is a feature which is hierarchically higher in the order of relationships with other features.

PartBody: It is a default body created in a CATPart Power copyis a feature created in a part which contains input parameters and features which can be duplicated in another part quickly.

Published Geometry: It is a geometry which is exposed and available to external users for selection during contextual design or applying constraints in an assembly.

Reflect line: It is the neutral line from which the draft faces will be generated in case of Reflect line drafts.

Representation: It is a term which describes the geometrical shape of the part.

Revolve: It is a surface of revolution created by revolving a planar profile about an axis.

Rotate: It is an operation in which an element is rotated by a specified angle about the given axis.

Scan: It is a process of reviewing the design of the model feature by feature.

Selective load: It is an operation in which the assembly is loaded in CATIA in stages.

Skeleton: It is a part which contains the wireframe geometry, specifying the design of the assembly.

Sketch: It is a set of geometric elements/profile created in the Sketcher workbench.

Solid Combine: It is a solid created by intersection of two or more extruded profiles.

Spine: It is a curve which normal planes are used to position a profile when creating a surface (lofted or swept surface for example). The profile does not necessarily intersect with this spine.

Split: It is a feature created by cutting existing elements with other geometric elements (planes, surfaces, etc.).

Glossary

Swept Surface: It is a surface obtained by sweeping a profile in planes normal to a spine curve while taking other user-defined parameters (such as guide curves and reference elements) into account.

Symmetry: It is an operation in which an element is transformed by means of a mirror symmetry with respect to a reference plane, line or point.

Synchronize: It is an operation of updating the link / assembly /document so that design changes done in parent document are propagated.

Translate: It is an operation in which an element is displaced along the specified direction by a specified distance.

Trim: It is a an operation in which wireframe or surface are cut mutually.

V4 model: It is a CATIA V4 model.

Vertex: It is a point where more three or more edges meet.

Visualization mode: It is a mode in which the lighter CGR models are loaded in CATIA for visualization purpose. The design features of the parts are not seen in this mode.

VRML: It is a file format named Virtual Reality Modeling Language.

Wireframe element: It is and element such as a point, line or curve that can be used to represent the outline of a 3D object.