STUDENT GUIDE

Student Notes:

Complex Assembly Design

In this lesson, you will learn what skeleton model and published geometry are and how they are used to control external references in assemblies.

Lesson Content:

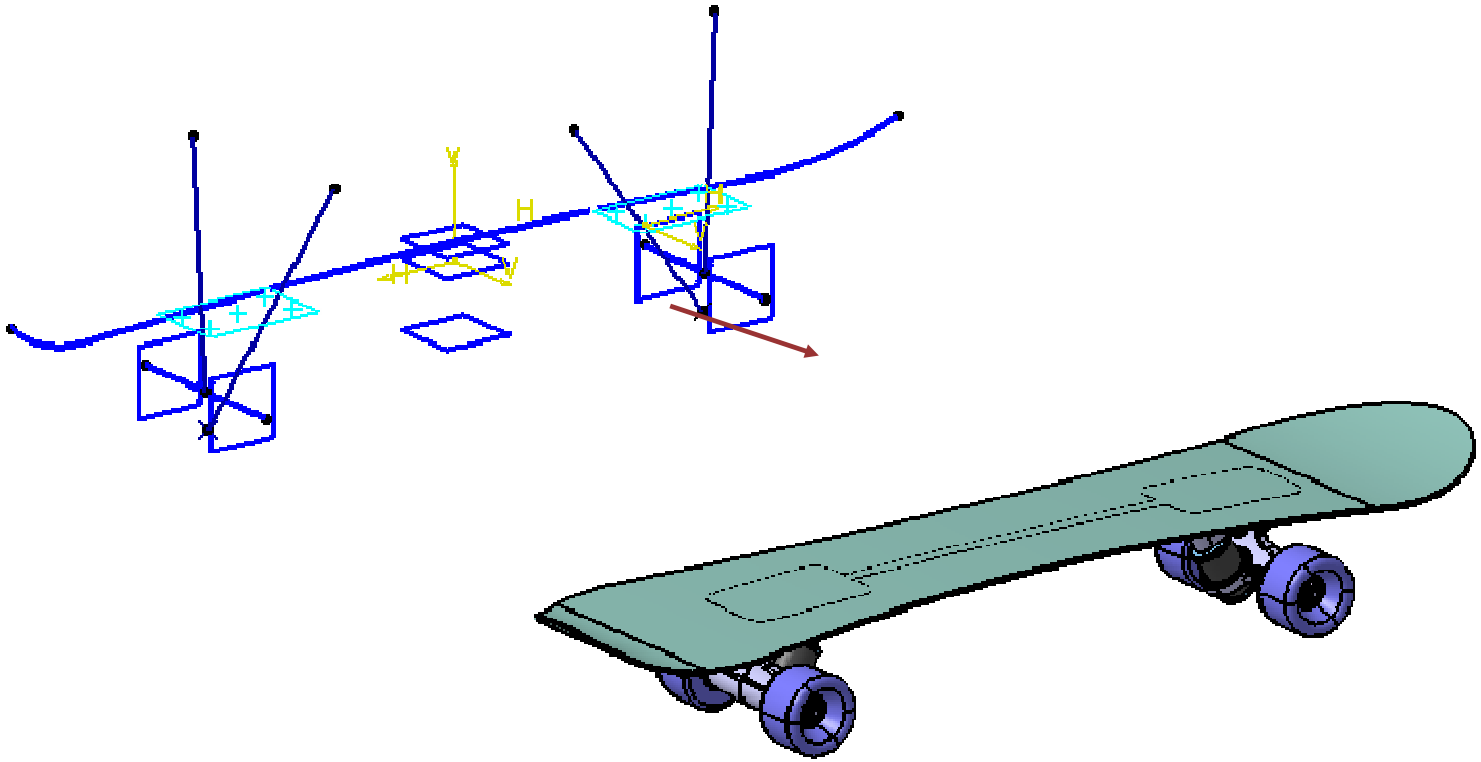
- Case Study: Complex Assembly Design
- Design Intent
- Stages in the Process
- Create a Skeleton Model
- Create the Published Elements
- Use the Published Elements

Duration: *Approximately 4 Hours*

Student Notes:

Case Study: Complex Assembly Design

The case study for this lesson is a skateboard assembly as shown below. The focus of this case study is the design of the support component. This support uses references from a skeleton model to control its overall size and location in the assembly. Publications are used to control the external references created between product components.

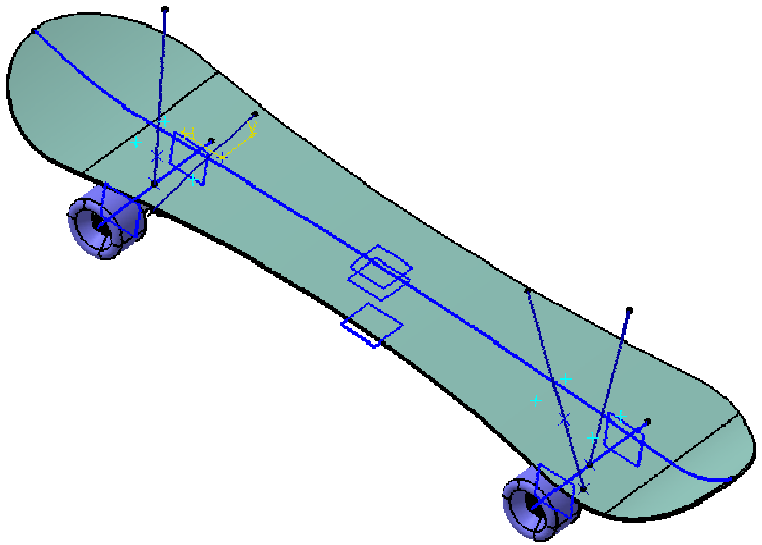


Student Notes:

Design Intent

The skateboard assembly must meet the following design intent requirements:

- ✓ Component locations must be controlled from a centralized location.
  - Using the skeleton method, component locations are controlled by referencing geometry in a skeleton model.
- ✓ Support geometry must update in all components.
  - Reference geometry and parameters created in the skeleton model can be linked to the necessary features. When the dimensions are changed in the skeleton model, they will update in all linked components.
- ✓ References must be strictly controlled.
  - Using published geometry, only published elements are allowed for selection when creating external references and assembly constraints.

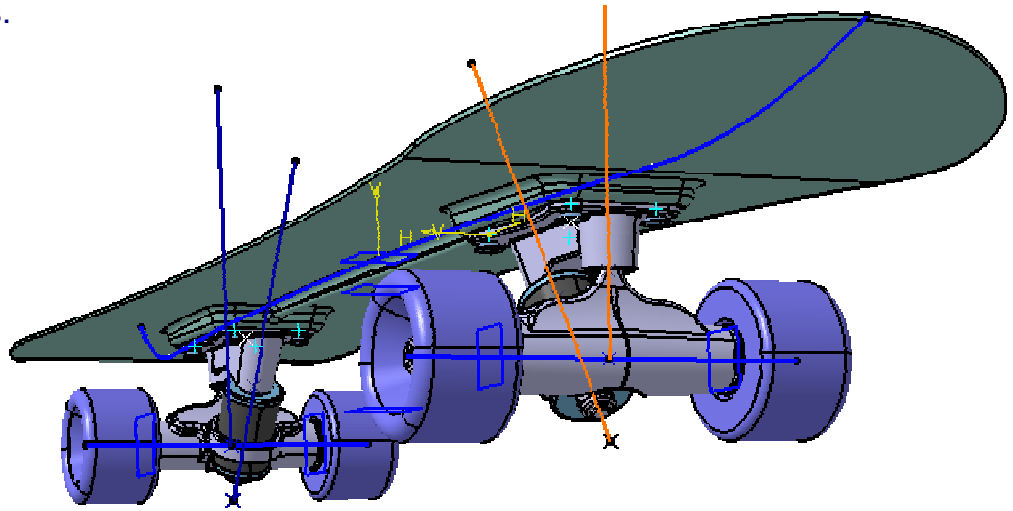


Student Notes:

Stages in the Process

Use the following steps to create the skateboard assembly:

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



Student Notes:

Create the Skeleton Model

In this section, you will learn what a skeleton model is and how to create one.

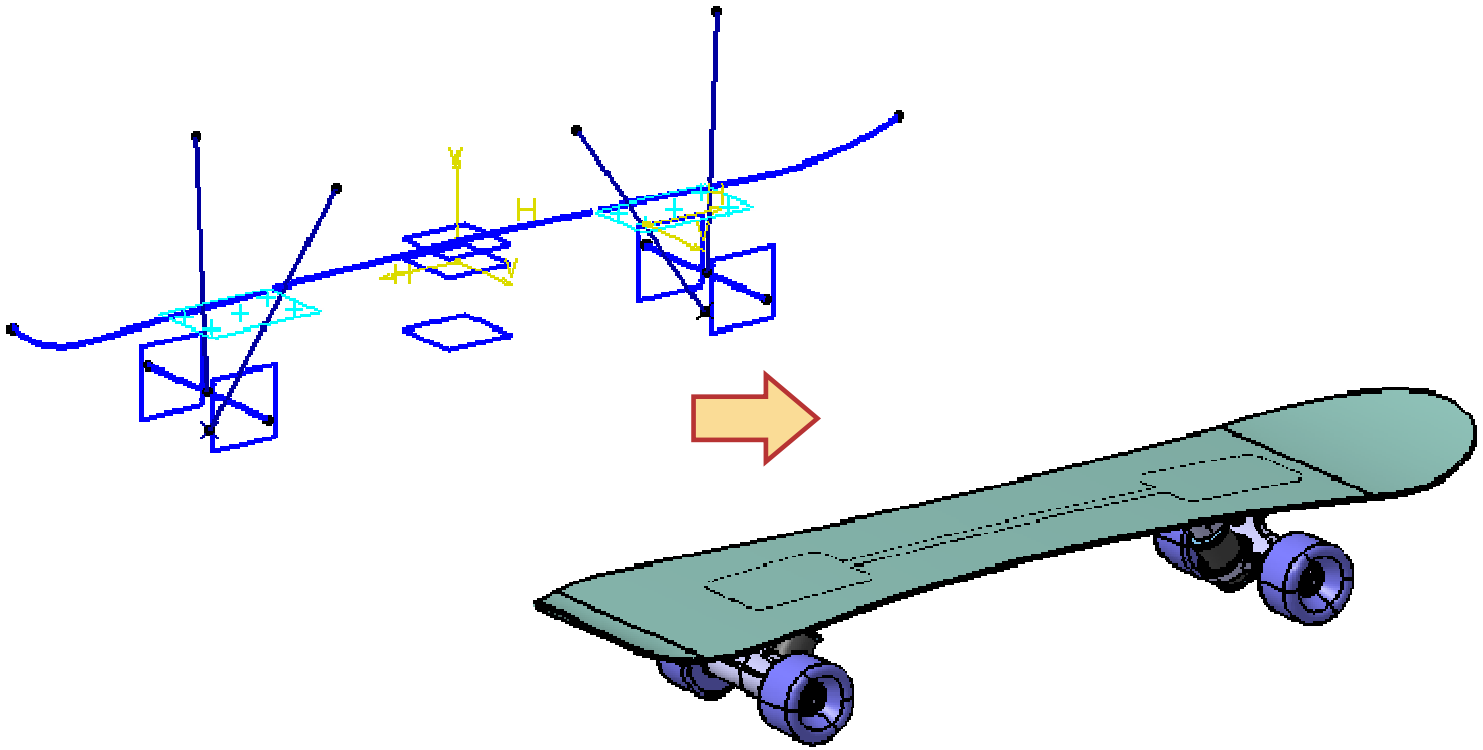
Use the following steps :

1. **Create the skeleton model.**
2. Create the published elements
3. Use the published elements

Student Notes:

What is the Skeleton Method? (1/2)

The skeleton method is a top down design approach. Using the skeleton method you can create and reuse the information stored in a single part, called the skeleton, to define the underlying design framework of individual components and assemblies.



Student Notes:

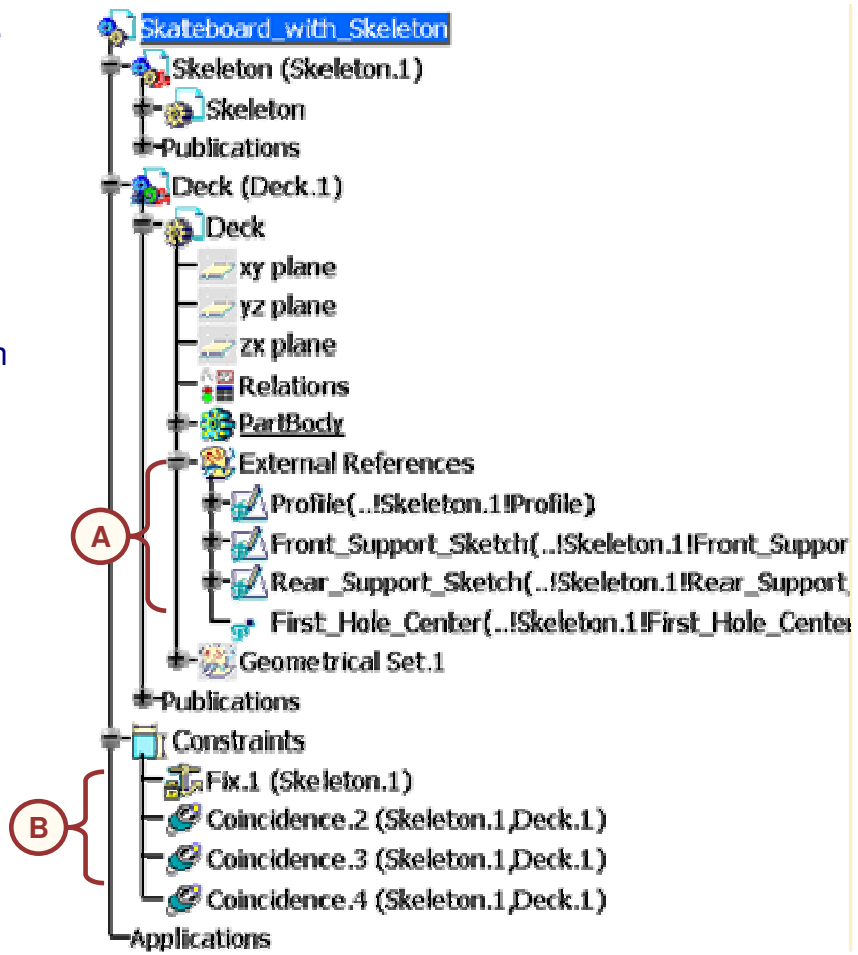
What is the Skeleton Method? (2/2)

Geometrical elements such as curves, axis, points, planes, and surfaces are stored in the skeleton.

These are used either to:

A. Design the other components of the product by creating external references pointing to the skeleton.

B. Position constraints between the skeleton and other components of the product.



Student Notes:

Why use Skeleton Method? (1/2)

The skeleton method offers some of the following advantages to the designers:

A. Specification-driven design:
- All important information is stored in the skeleton model. Space constraints are clearly defined within the skeleton to help allocate space for the components within the assembly.

B. Design changes:
- The skeleton method helps manage high-level design changes and propagate them throughout the assembly. Modifications to design information in the skeleton model propagates to all the relative individual components and sub-assemblies. This provides you more control over changes in design.

C. 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. Changes to the design can be made in the skeleton and all models will update to reflect these modifications. As the components are not linked to each other, the deletion of a component within an assembly will not impact the others.

STUDENT GUIDE

Student Notes:

Why use Skeleton Method? (2/2)

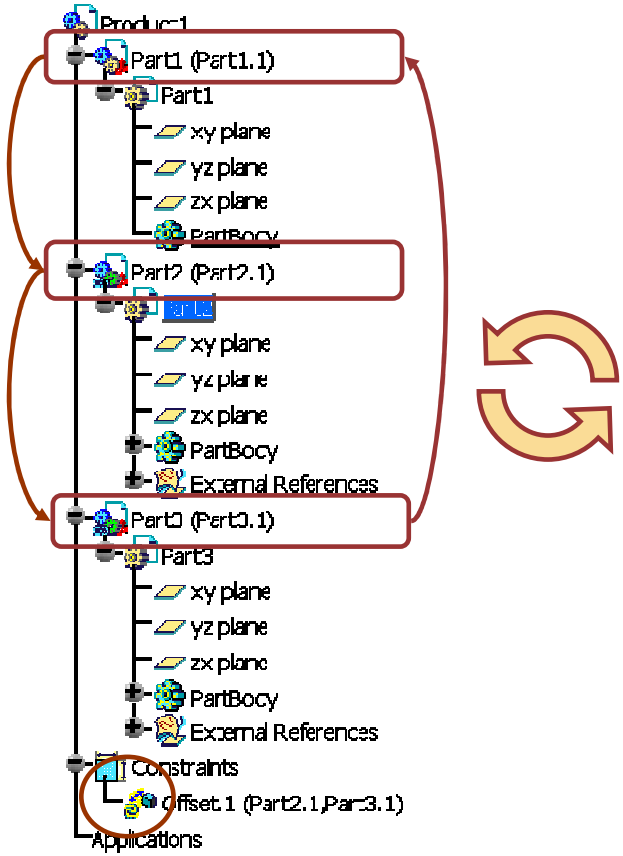
The skeleton method offers some of the following advantages to the designers (continued):

D. Avoid update loops:

- When you use the skeleton method, all are external references point to the skeleton part and you will have to avoid update loops.
- All links are unidirectional, the skeleton model is used as an external reference for other components, but the skeleton model does not use external reference within the assembly to define its geometry.

In this example, the Offset constraint cannot update because the contextual links and the positioning constraint interfere. An update loop is created and the system cannot resolve it. This situation occurs when the skeleton is not used.

The skeleton method can avoid this problem.



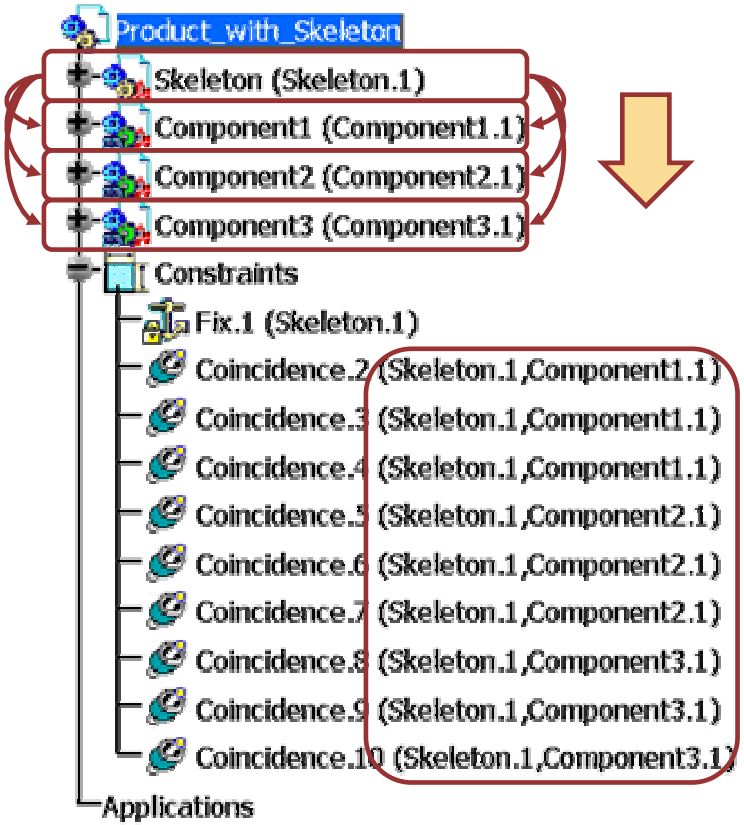
Student Notes:

How is the Skeleton Method Implemented?

When using the skeleton method, contextual and positioning links only point to the skeleton part. This ensures the links do not interfere.

Moreover, you can delete one contextual part, “Component2” for example, without any impact on the others.

Notice the direction of information is always downwards (i.e., top down), from the skeleton model to the other components.



Student Notes:

What Does a Skeleton Model Contain?

A skeleton model contains the elements that will drive the main dimensions and positions of the components of the assembly. The skeleton can also help you to define space allocations. Below, are some example of elements that can be used to define a skeleton:

Element	Example of Contextual use	Example of Positioning use
Point	• To locate the center of a hole • As a limit in a sketch	• To position the origin of components in an assembly
Line	• As the axis for shafts, grooves, and revolutions.	• To apply an axis coincidence constraint (cylindrical parts)
Plane	• To limit the depth of a feature (such as a pad) • As a sketch support	• To apply a planes coincidence constraint
Surface	• To limit the depth of a feature (such as a pad) • As a sketch support	• To apply a planes coincidence constraint
Sketch	• To reuse the same profile in several parts • To create a user defined pattern	
Solid	• As an allocation volume • As a base feature	
Curve	• As a guide for features such as ribs and sweeps	
Axis	• As a multi-direction reference system	
User Parameter	•To control dimensions in the assembly	• Control the constraint value of an offset or an angle

Student Notes:

Skeleton Example (1/4)

When creating an assembly using the skeleton method, wireframe and surface geometry created in the skeleton model is referenced by the other components in the assembly.

The following example will explore the elements used to construct the skateboard assembly. The skateboard has the following components:

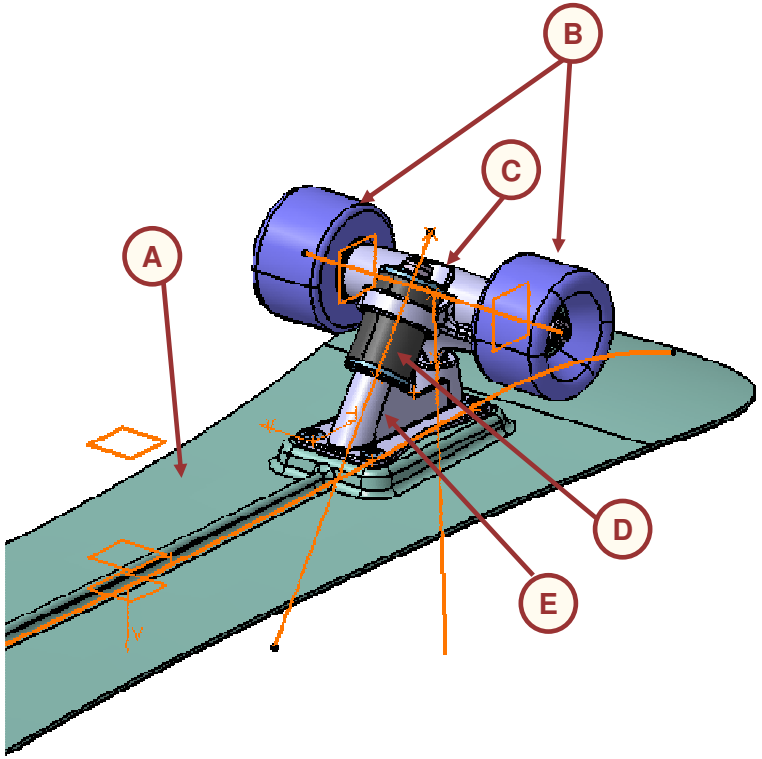
A. Deck

B. Two wheels

C. Axle tree

D. Shock Absorbers

E. Support



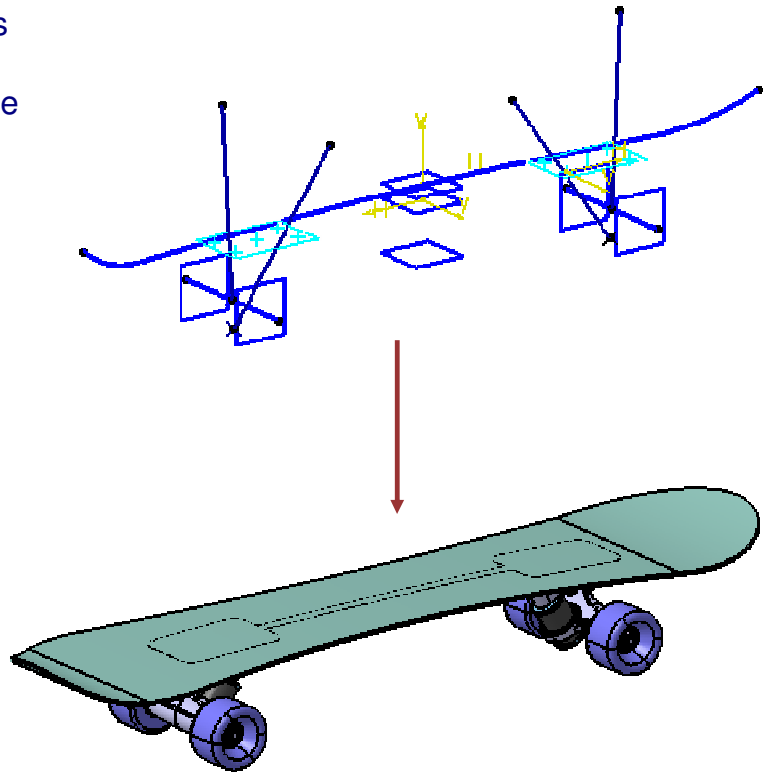
Student Notes:

Skeleton Example (2/4)

A skeleton model is constructed that will represent the position and overall dimensions of the skateboard assembly. All critical positions and dimensions are controlled inside this model.

In this example, lines, points, sketches, and planes are created to control the model.

Next, we will look at how these elements will control the model.



Student Notes:

Skeleton Example (3/4)

The following describes some of the elements used from the skeleton to constrain the assembly:

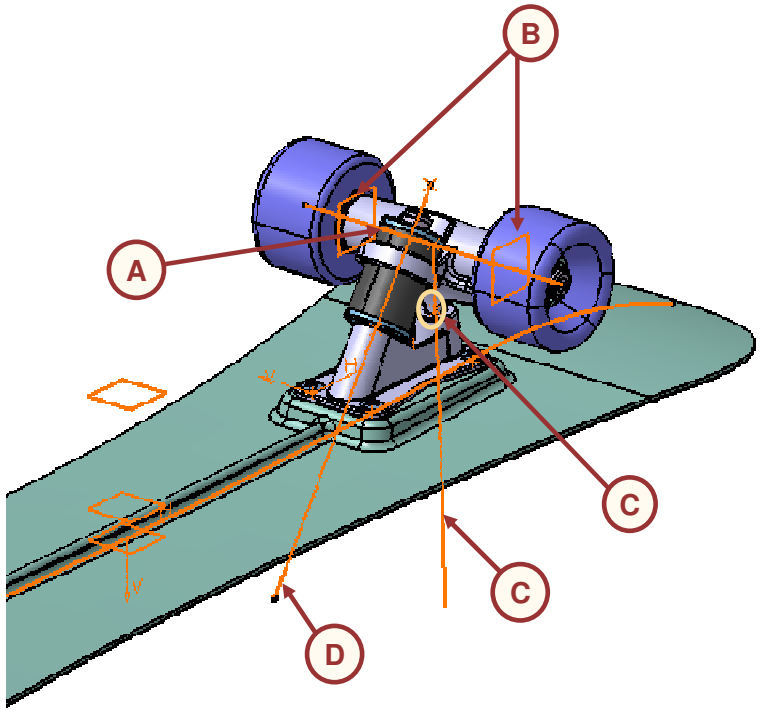
A. A line is created to define the location of both wheel axis. It is also used to design the axle tree model.

B. Two planes are created to locate the wheels. These planes are also used as limiting elements for the axle tree.

C. A point is used to define the intersection between the support and the tree axle.

D. A line is used to define axis location while designing both the axle tree and the support.

E. A line is used to create an axis coincident constraint to position the shock absorber. It is also used to ensure proper design of the axle tree and the support.



Student Notes:

Skeleton Example (4/4)

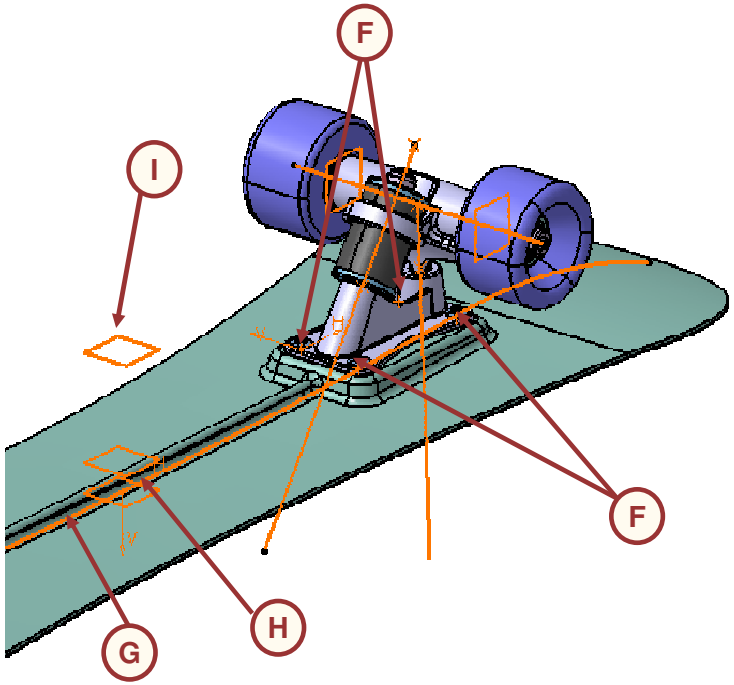
The following describes some of the elements used from the skeleton to constrain the assembly (continued):

F. A sketch containing four points is copied into the deck and the support to correctly locate the fixation holes. It is also used to constrain the bolts.

G. A sketch defining the side profile of the deck is copied into Deck component and used for feature creation.

H. A plane is used to position the support and design the deck.

I. A plane is created that is offset from the line used to define the wheel axis. The deck is positioned using this plane to ensure that the correct distance from the wheels to the deck is maintained.



Student Notes:

How is a Skeleton Created?

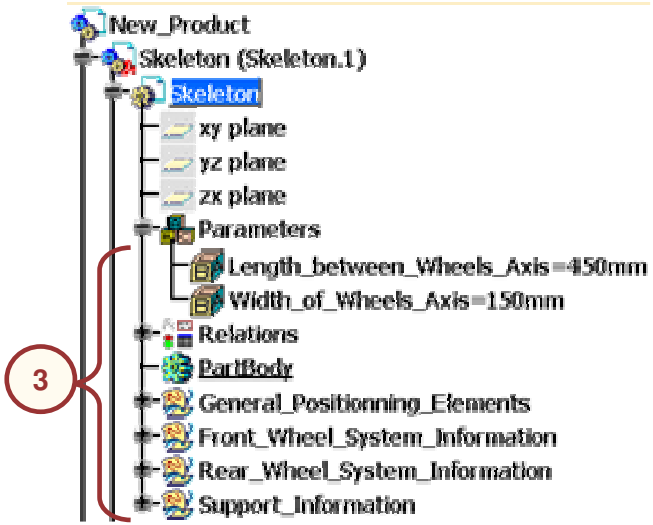
Use the following general steps to create a skeleton model:

1. Create a new part file inside the assembly. The skeleton must be the first component assembled into the product.

2. Position the skeleton model into the assembly using the Fix constraint.

3. Create the reference geometry and user parameters necessary to define the design intent of the assembly.

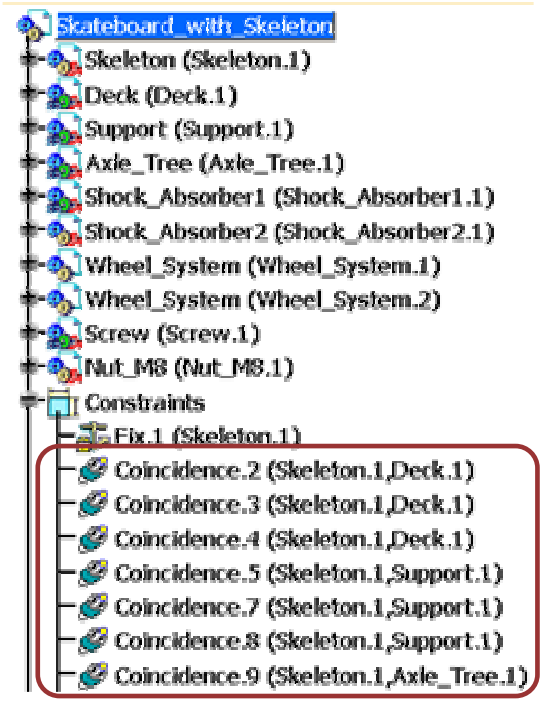
4. Design the assembly components using the skeleton model as reference.

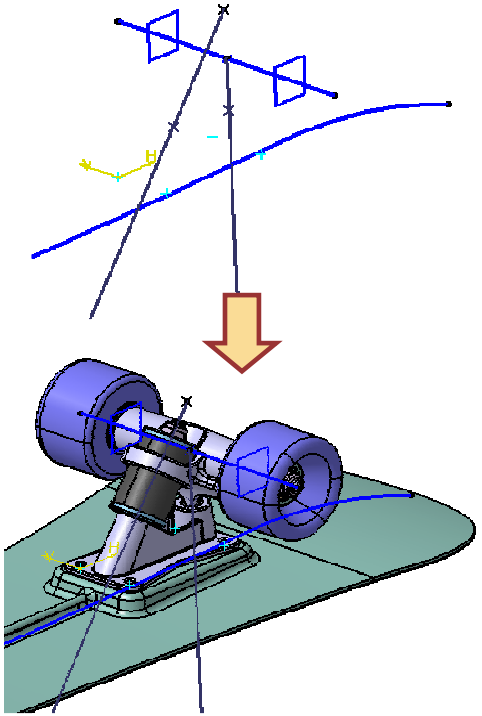


Student Notes:

Constraints and the Skeleton Model

To properly use the skeleton method, models are constrained using only the skeleton model as reference for positioning. Geometrical elements within the skeleton model (such as points, curves, planes, and axis) are used as constraint references for the assembly components.





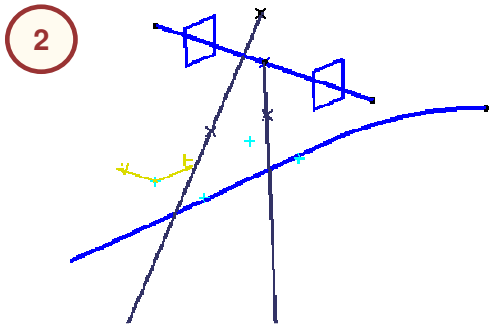
Student Notes:

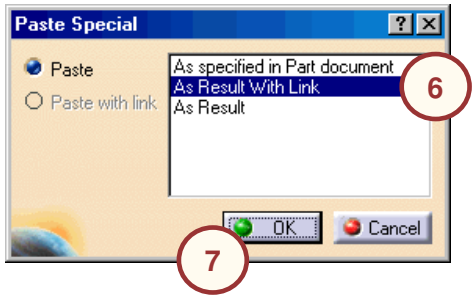
Using Skeleton to Design the Components of a Product (1/2)

Often, more than one designer will work on an assembly. Models are opened on their own and changes in the assembly need to be circulated to all parts. With skeleton models, elements such as overall parameters and important user parameters can be copied from the skeleton into the necessary components.

Use the following steps to copy geometrical information into a component:

1. Activate the skeleton component.
2. Create the necessary wireframe and surface geometry in the skeleton model.
3. Copy the elements needed in the components.
4. Activate the components.
5. Right-click on the target part and from the contextual menu click **Paste Special**.
6. Select **As Result with Link** from the Paste Special dialog box.
7. Click **OK**.



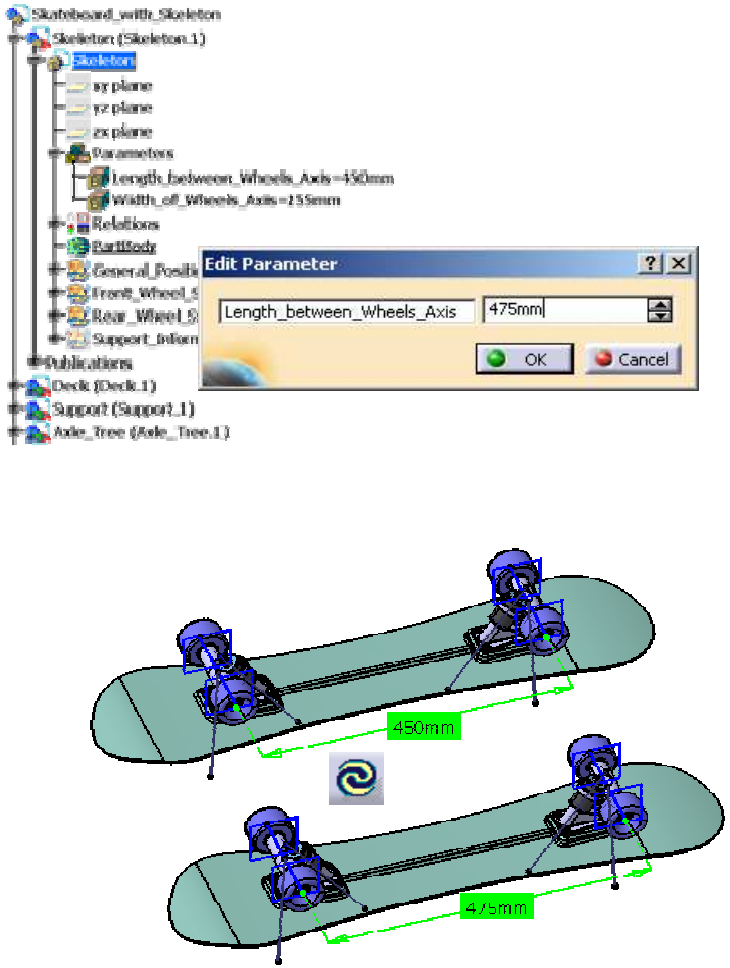


Student Notes:

Using Skeleton to Design the Components of a Product (2/2)

By associatively copying information from the skeleton model into the components, the designers can be sure the elements they are using to construct their component are up to date. Any change to the overall design are made in the skeleton model and, because of the links, are automatically propagated to the components.

For example, a user-defined parameter is created in the skeleton model to control the distance between the front and rear axes of a skateboard assembly. This parameter is then used to drive wireframe geometry in the skeleton model. The wireframe geometry is referenced while constraining and designing the skateboard model. If the value of the user parameter is modified the axes position will update accordingly. This will propagate through the entire assembly and the new position and design of the deck will be updated.

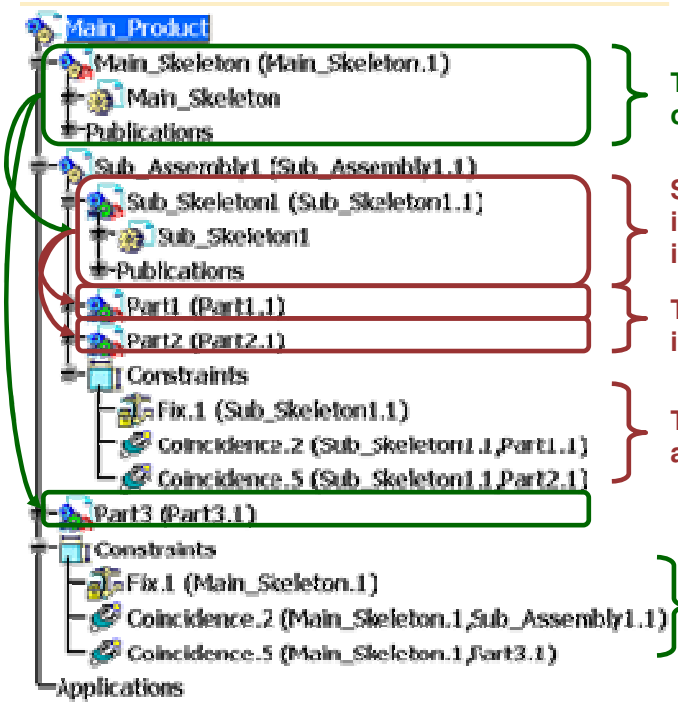


Student Notes:

Reusing Skeleton in Sub-Assemblies

It is possible to use the skeleton method in a product which contains sub-assemblies.
In this case, you create a sub-skeleton for each of the sub-assemblies that require additional information to drive it. All necessary information from the main skeleton is copied into the sub-skeletons using the Paste Special option **As Result with link**. Additional information only relevant to the particular sub-assembly is then added.

Top-down direction of the information



The main skeleton contains main information of the product.

Sub-skeleton which contains copied with link information from the main skeleton + added information.

The parts of the sub-assembly are reusing information from the sub-skeleton.

The sub-skeleton is fixed in its sub-assembly and the other parts are constrained to it.

The sub-assembly is constrained to the main assembly with the help of positioning constraints between sub-skeleton and main-skeleton. It is also possible to fix the sub-skeleton in the main assembly.

Student Notes:

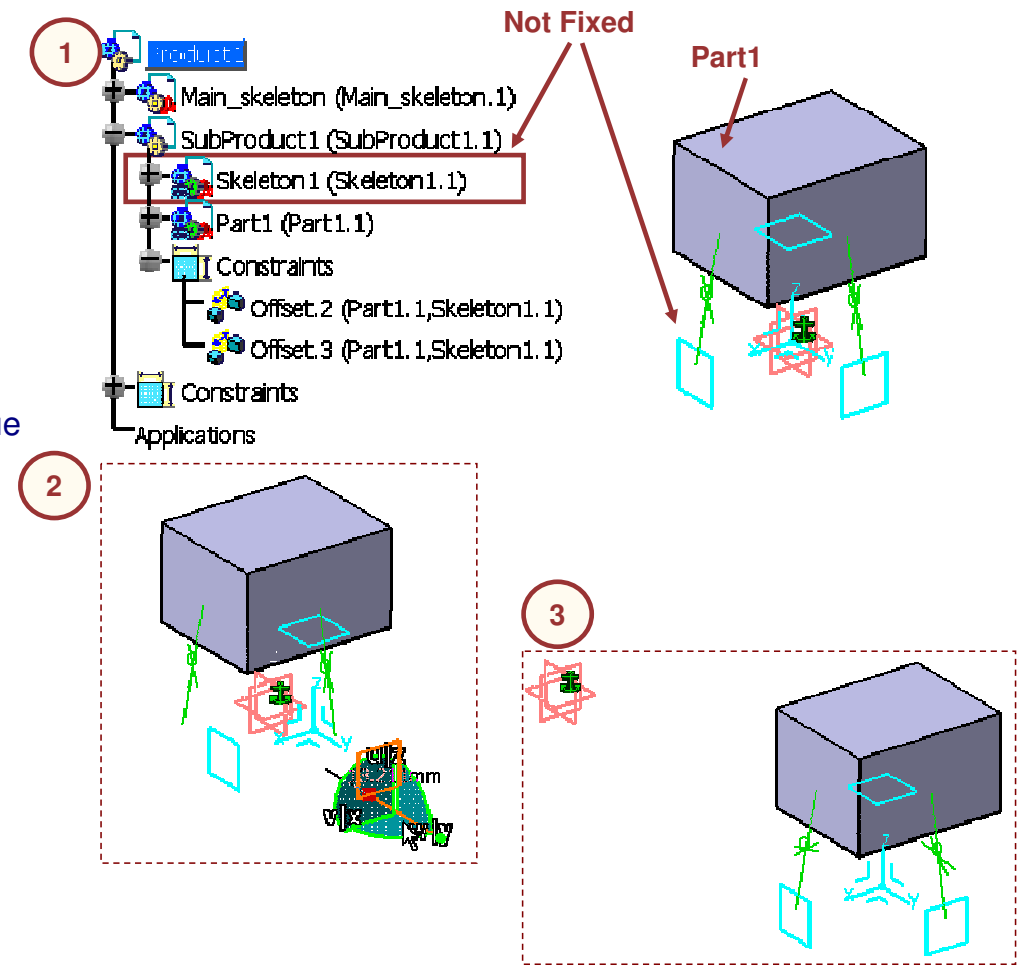
Recommendation for Skeleton Model

It is recommended to fix the skeleton part inside an assembly.

By fixing it the degrees of freedom of a skeleton part becomes zero. This guarantees the positioning of the parts depending on the skeleton.

In the example shown:

1. SubProduct.1 has skeleton.1 (blue elements), which is not fixed and position of Part1 depends on Skeleton1.
2. Manipulate the position of skeleton.1.
3. Position of Part1 changes in the space.



Student Notes:

Exercise: Skeleton Model Use

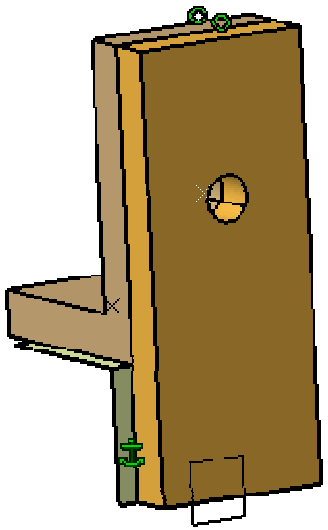
Recap Exercise

20 min

In this exercise, you will create an assembly using the skeleton method. You will use the tools learned in this lesson to assemble the skeleton model into the assembly, then constrain the components to it. You will use reference geometry copied from the skeleton to design a hole in each component. Finally, you will test the links by modifying the reference geometry in the skeleton to ensure the component geometry updates properly. Detailed instructions for this exercise are provided.

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

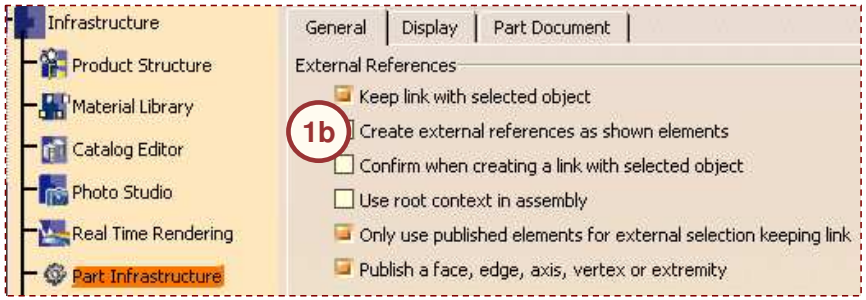
- Create a product using the skeleton method
- Create geometry by referencing the skeleton model

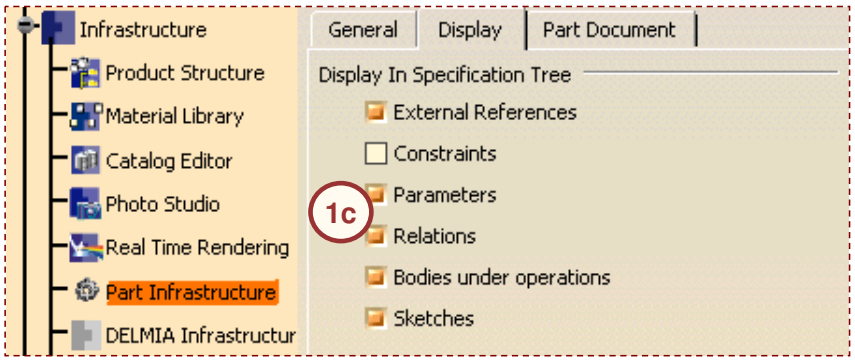


Student Notes:

Do it Yourself (1/12)

1. **Set options.**
   - Set options to display parameters and relations in the specification tree and to keep a link to external references.
     a. Click **Tools > Options > Infrastructure > Part Infrastructure**.
     b. Activate the **Keep link with selected object** option from the General tab.
     c. Activate the **Parameters** and the **Relations** options from the Display tab



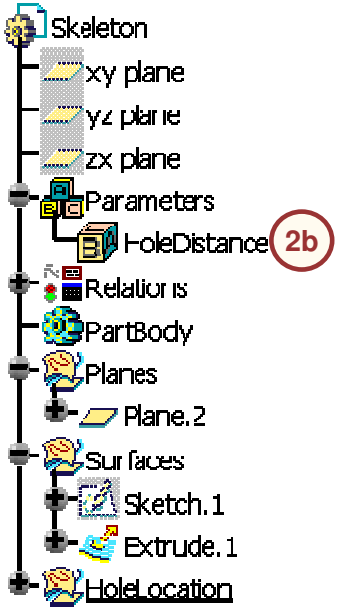


Student Notes:

Do it Yourself (2/12)

2. Open and Observe the Part

- **Open Skeleton.CATPart.** This part already has a point, a plane, and a surface created for you.
  a. Notice that the wireframe elements and surface features have been created in a separate geometrical sets.
  b. Expand the **Parameters** node of the specification tree. The Skeleton model has a user-defined parameter called **HoleDistance**.

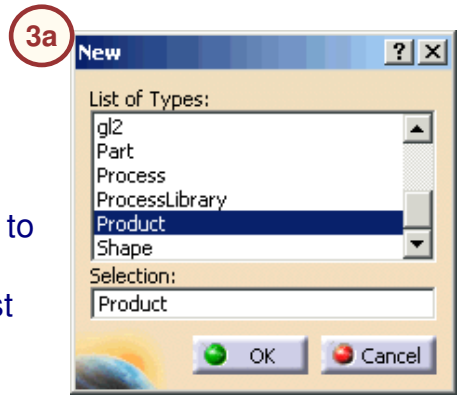


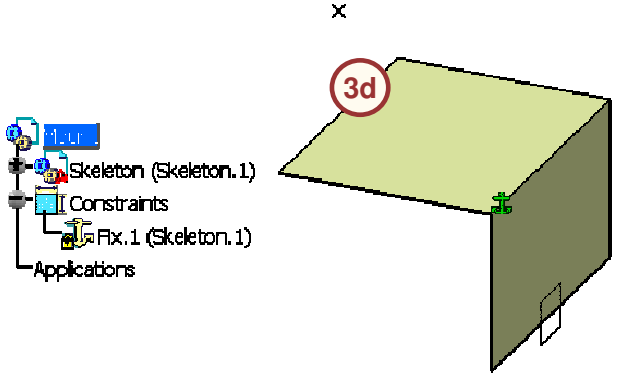
Student Notes:

Do it Yourself (3/12)

3. Create a product file.

- Create a new **Product** file to house the skeleton model. Use the skeleton model to build the assembly components. The skeleton model should always be the first assembled component.
- Constrain the Skeleton Model.
  a. Click **File > New > Product**.
  b. Name the product with the name **Mount.**
  c. Right-click on Mount and select **Components> Existing Component** from the contextual menu.
  d. Select **Skeleton.CATPart** as the part to assemble.
  e. Select the **Fix** Component icon to apply a Fix constraint.
  f. Save the product.



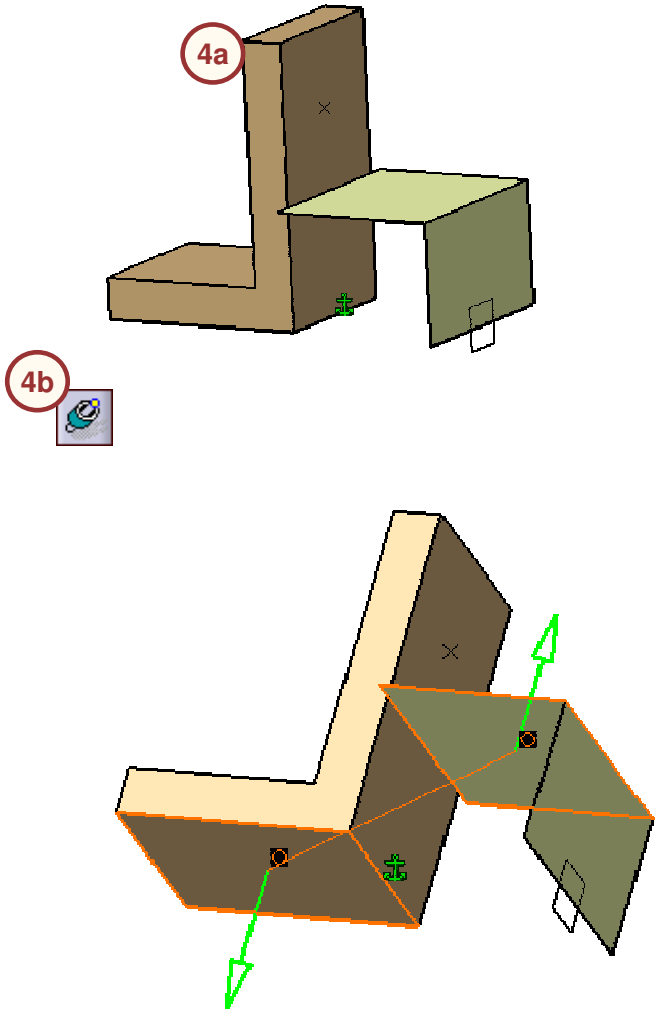


Student Notes:

Do it Yourself (4/12)

4. Insert a Component, Assemble and Constrain it.

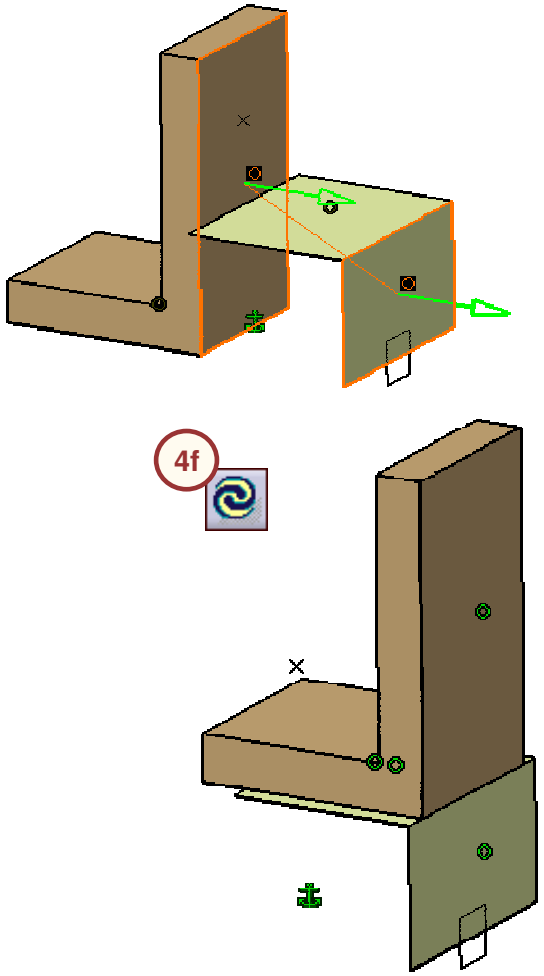
- Assemble the first solid component into the product.
- Constrain a component.
- Update the product.
  a. Use the right mouse button to get the contextual menu select **LBracket.CATPart** as the part to assemble.
  b. Select the **Coincidence** Constraint icon to apply a coincident constraint.
  c. Apply the coincidence between the surface feature of the skeleton and the bottom face of the LBracket.



Do it Yourself (5/12)

4. Insert a Component, Assemble and Constrain it. (Continued)

- Assemble the first solid component into the product.
- Constrain a component.
- Update the product.
  - e. Apply another coincident constraint. Apply a **Coincident** constraint between the surface feature of the skeleton and the face of the LBracket.
  - f. Select the **Update All** icon to update the assembly constraints. The updated assembly appears as shown.



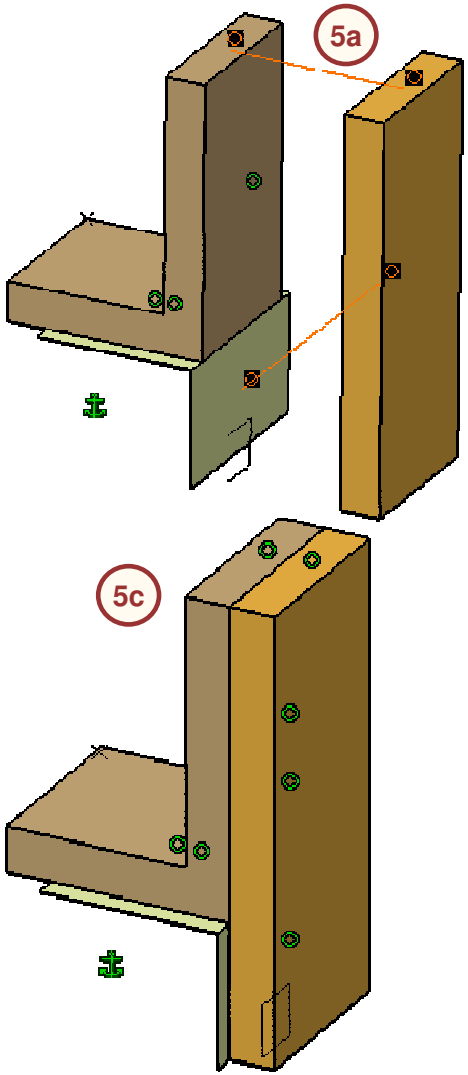
Student Notes:

Student Notes:

Do it Yourself (6/12)

5. Insert another Component, Assemble and Constrain it.

- Assemble a second solid component into the product. Constrain it only to the skeleton model.
- Update the product.
- Save the assembly.
  a. Assemble the **IBracket.CATPart.**
  b. Use **coincident** constraints to position the Ibracket part as shown.
  c. Update the assembly to view the components in their correct locations. The updated assembly appears as shown below.
  d. Save the Mount assembly. Close the Mount file.

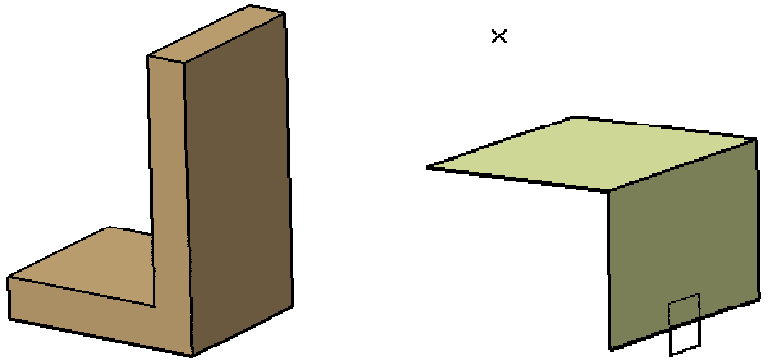


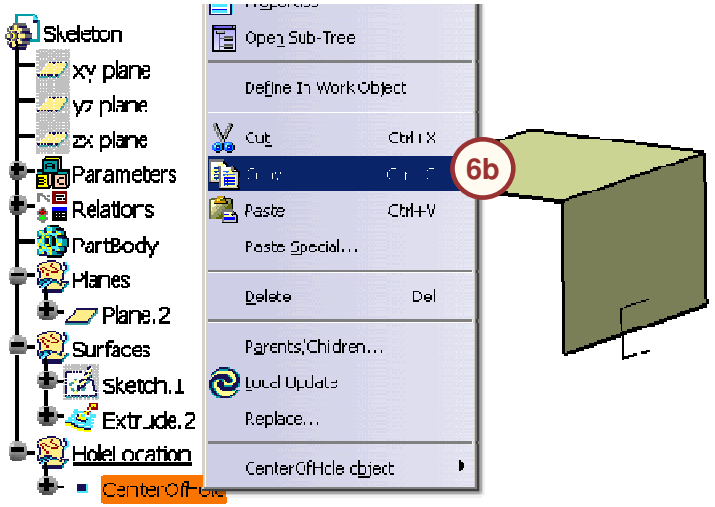
Student Notes:

Do it Yourself (7/12)

6. Create Geometry referencing to the skeleton model.

- Open two part files.
- Copy geometry from the skeleton and paste it into Lbracket part.
  a. Open **Skeleton.CATPart**. Open **LBracket.CATPart**.
  b. Right click on CenterOfHole point and select **Copy** from the contextual menu to copy the geometry.

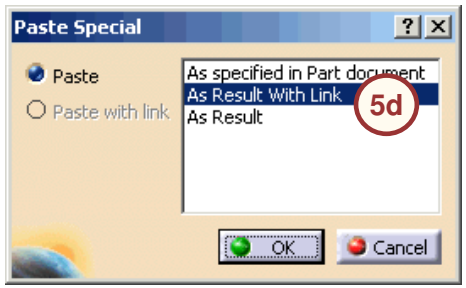


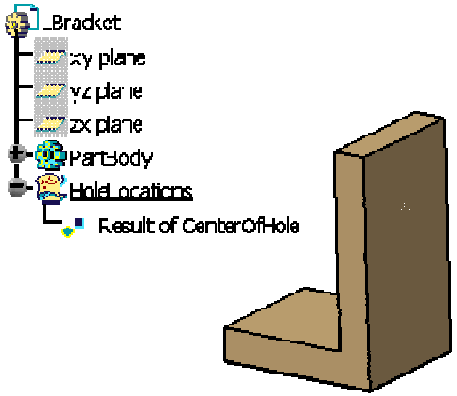


Do it Yourself (8/12)

6. Create Geometry referencing to the skeleton model. (Continued)

- Open two part files.
- Copy geometry from the skeleton and paste it into Lbracket part.
  a. Activate the LBracket window.
  b. Select HoleLocations geometrical set.
  c. Use the right mouse button pop-up menu to **Paste Special**.
  d. Select the **As Result With Link** paste special option.
  e. The pasted point will maintain a positional link with the Skeleton.





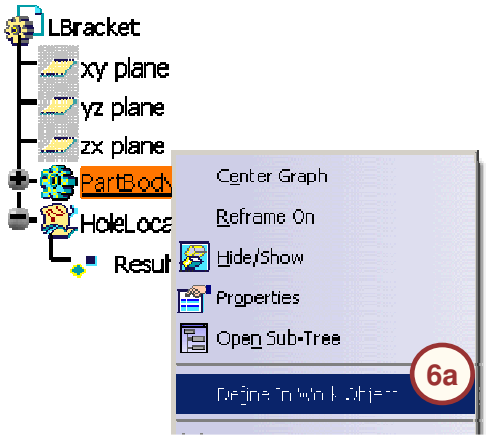
Student Notes:

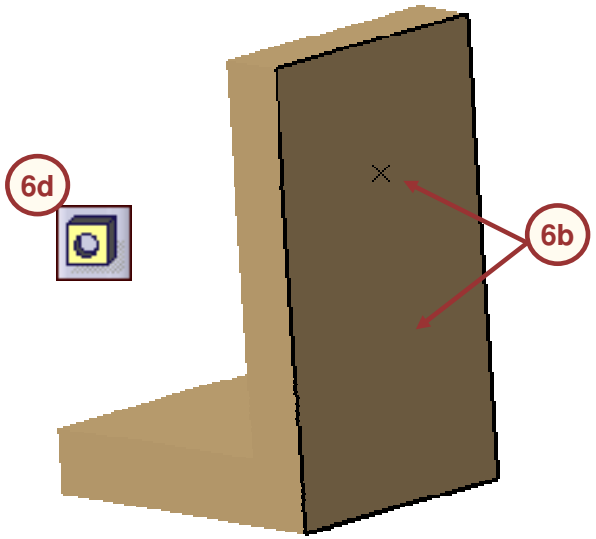
Student Notes:

Do it Yourself (9/12)

6. Create Geometry referencing to the skeleton model. (Continued)

- Create a Hole
  - a. Define the Part Body to be the work object. Use the right mouse button contextual menu to define the Part Body to be the Work Object.
  - b. Create a hole using the copied point as the center reference.
  - c. Pre-select the face and the pasted point.
  - d. Select the **Hole** icon.



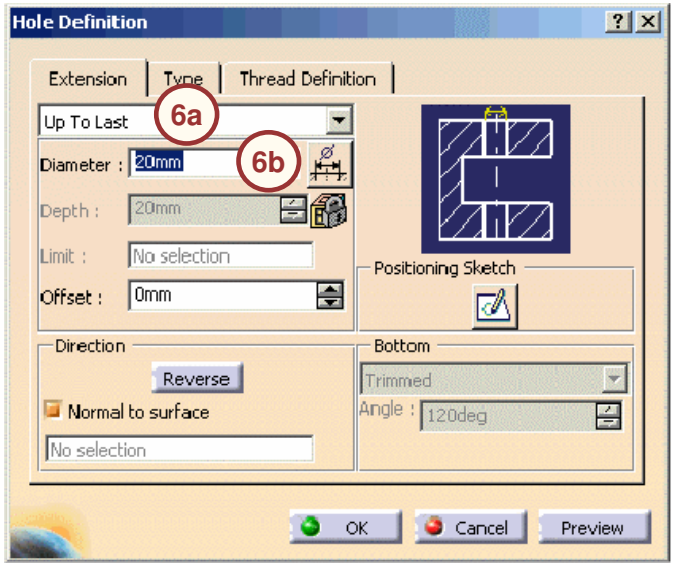


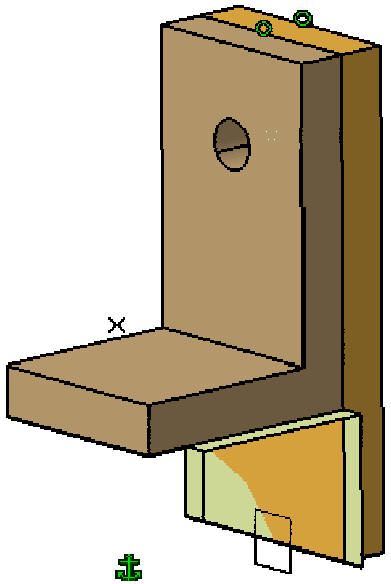
Student Notes:

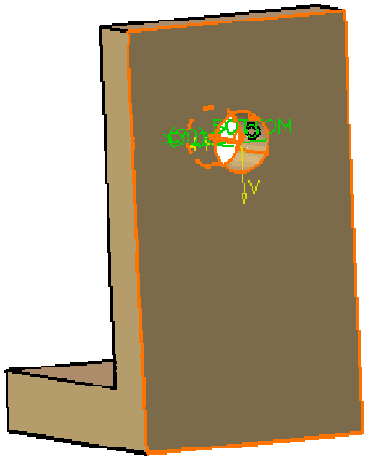
Do it Yourself (10/12)

6. Create Geometry referencing to the skeleton model. (Continued)

- Create Hole
  a. Specify **Up To Last**.
  b. Enter **[20mm]** diameter.





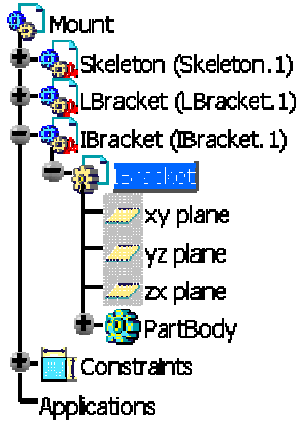


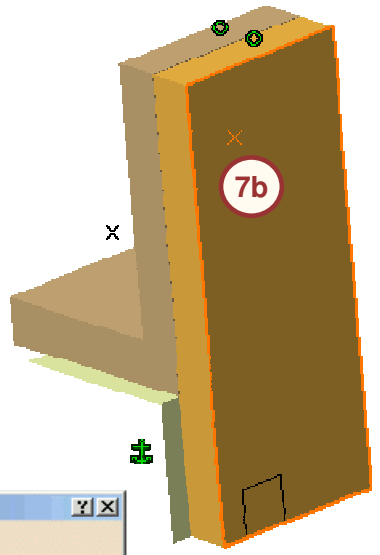
Student Notes:

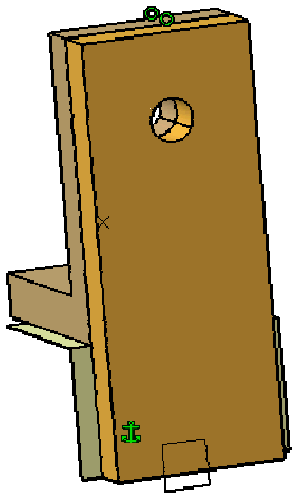
Do it Yourself (11/12)

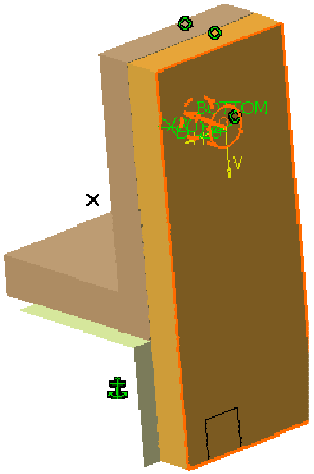
7. Create Geometry in context with the skeleton model.

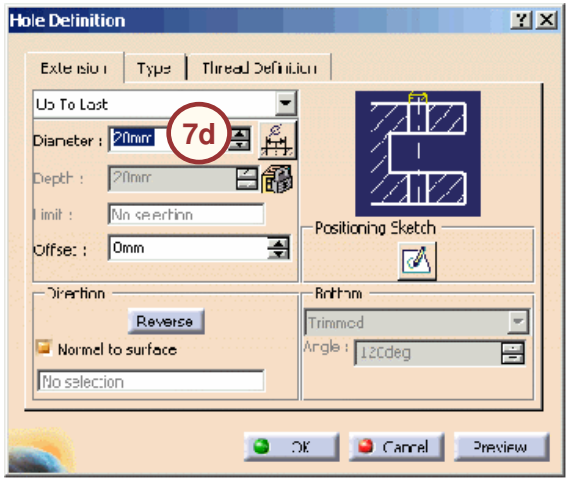
- Open Mount.Catpart
- Create a Hole in Context
  a. Create a hole using external references. Double-click IBracket to activate the Part Design workbench.
  b. Select the face and Point as references for a hole feature.
  c. Select the Hole Icon.
  d. Enter diameter =20mm and depth as Up To Last.









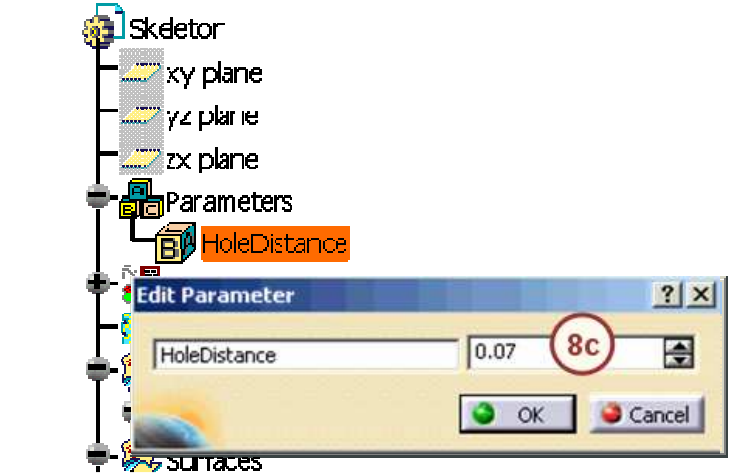


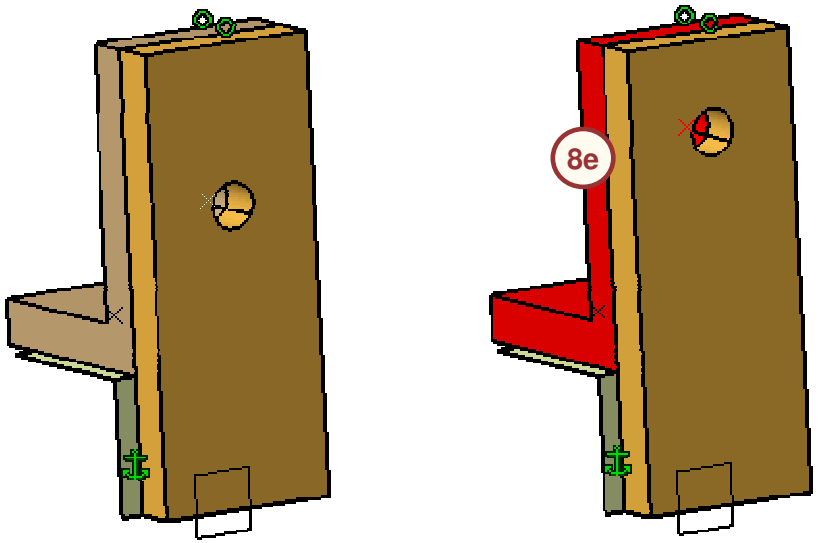
Student Notes:

Do it Yourself (12/12)

8. Change Parameter Values in the Contextual Part

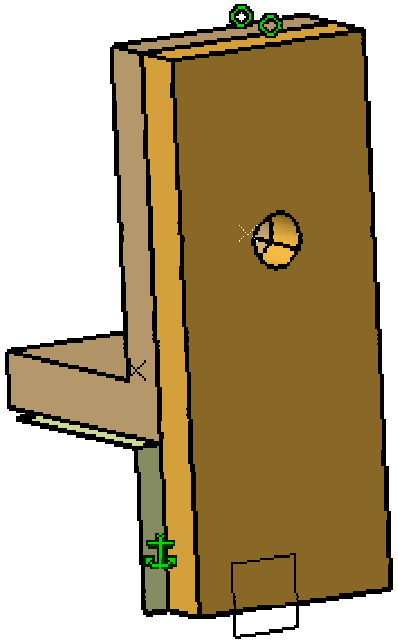
- Modify the HoleDistance parameter. This parameter controls the location of the point that was used to locate the center of both holes.
- Update the assembly file.
  a. Open the **Skeleton.CATPart** window. The Skeleton model appears as shown.
  b. Double-click the **HoleDistance** parameter.
  c. Enter [0.07] as the new value for the HoleDistance parameter.
  d. Activate the **Mount.CATProduct** window. The LBracket part is out of date. The assembly file must be updated.
  e. Update the assembly, the changes to the skeleton appear in the updated assembly.





Exercise: Skeleton Model Use Recap

- ✓ Create a product using the skeleton method
- ✓ Create geometry by referencing the skeleton model



Student Notes:

Student Notes:

Exercise: Skeleton Parameter Use

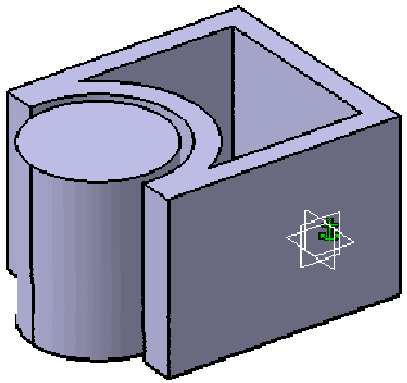
Recap Exercise

20 min

In this exercise, you will create parameters in skeleton model to drive component geometry. You will use the tools learnt in this lesson to create a product using the skeleton method. The component geometry will be driven by elements referenced from the skeleton model. Detailed instructions are provided for the new topics present in this exercise.

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

- **Create a product using the skeleton method**
- **Reference geometry and parameters from the skeleton to drive component geometry**

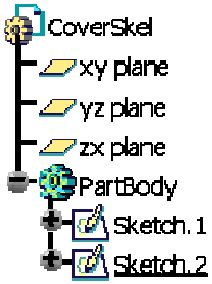


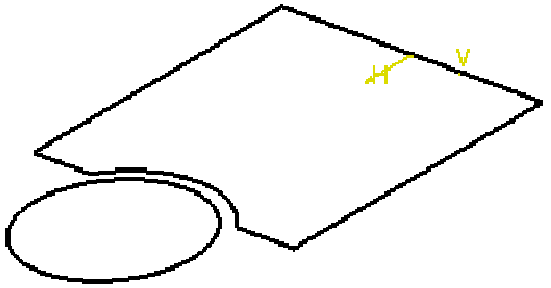
Student Notes:

Do it Yourself (1/13)

1. Open a part file.

a. Open CoverSkel.CATPart. This part already has two sketches created for you.





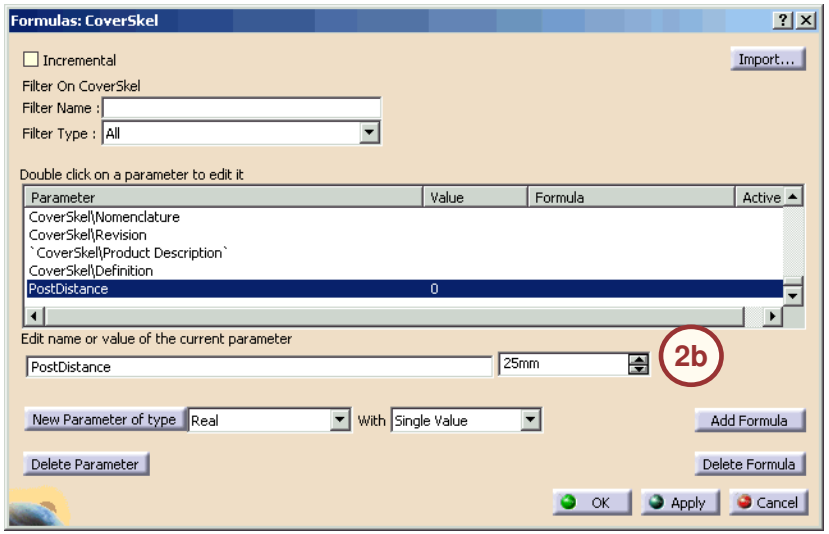
2. Create a parameter.

- Create user-defined parameters to control the sketch dimensions

a. Select the **Formula** icon to create a parameter.

2a 

b. Create a parameter named **PostDistance** and give it a value of [25mm].

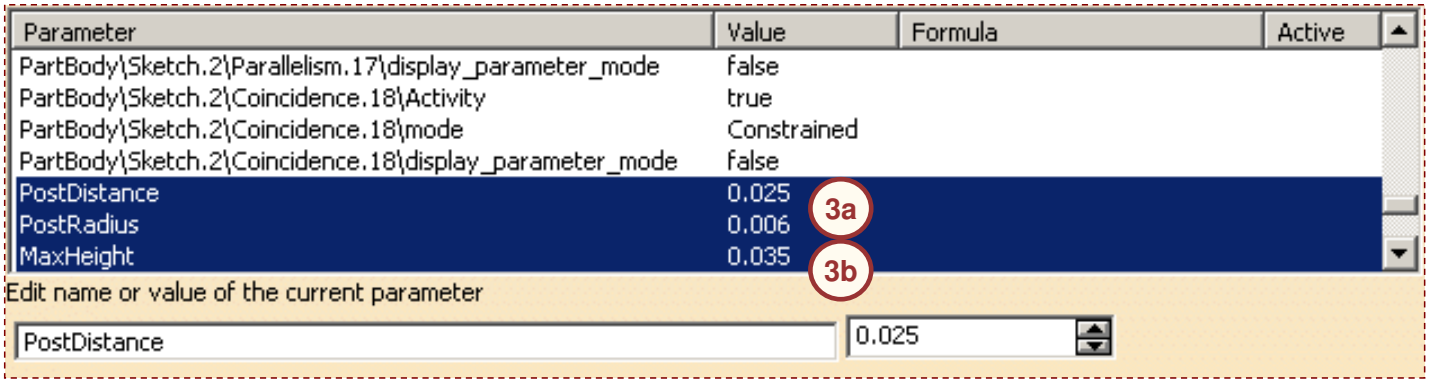


Student Notes:

Do it Yourself (2/13)

3. Create parameters.

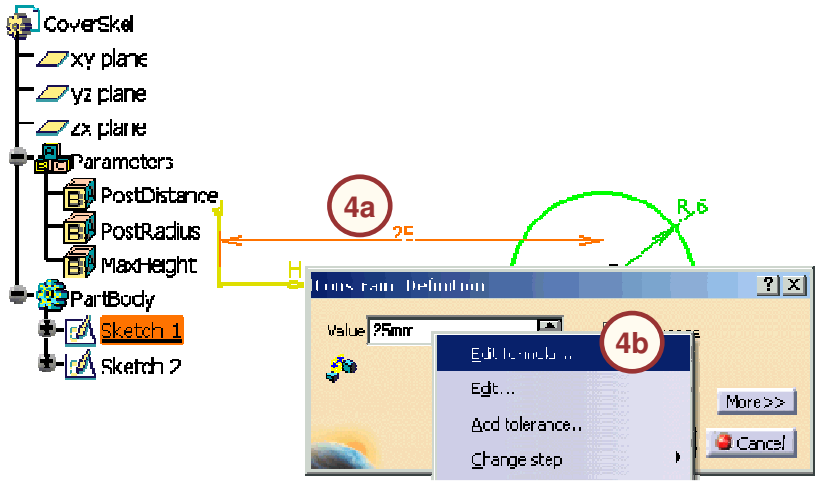
- Create two more parameters.
  a. PostRadius = 6mm.
  b. MaxHeight = 35mm.

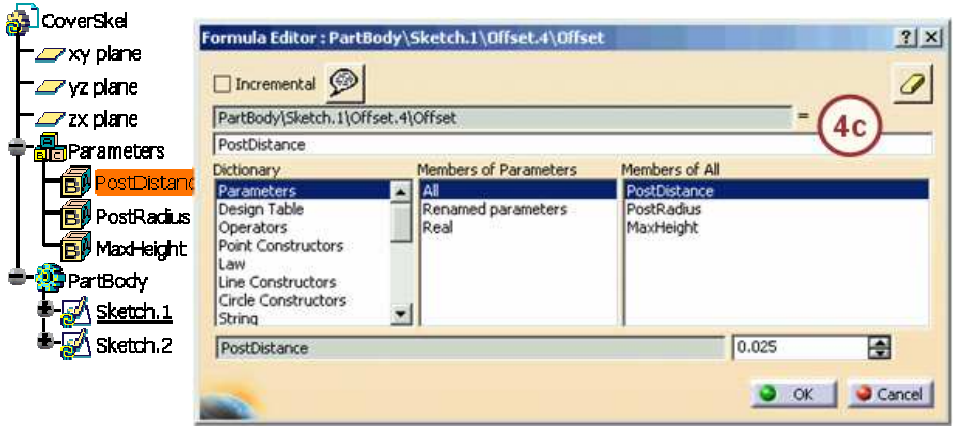


Do it Yourself (3/13)

4. Drive the sketch with parameters.

- Edit the sketch dimensions and equate them to the user-defined parameters.
  a. Edit Sketch.1 and modify the dimension equal to 25.
  b. From the contextual menu click **Edit formula**.
  c. Equate the dimension to the PostDistance parameter.





Student Notes:

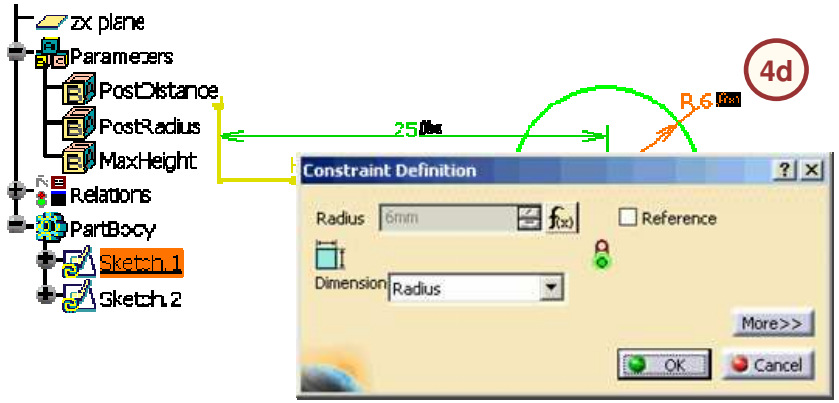
Student Notes:

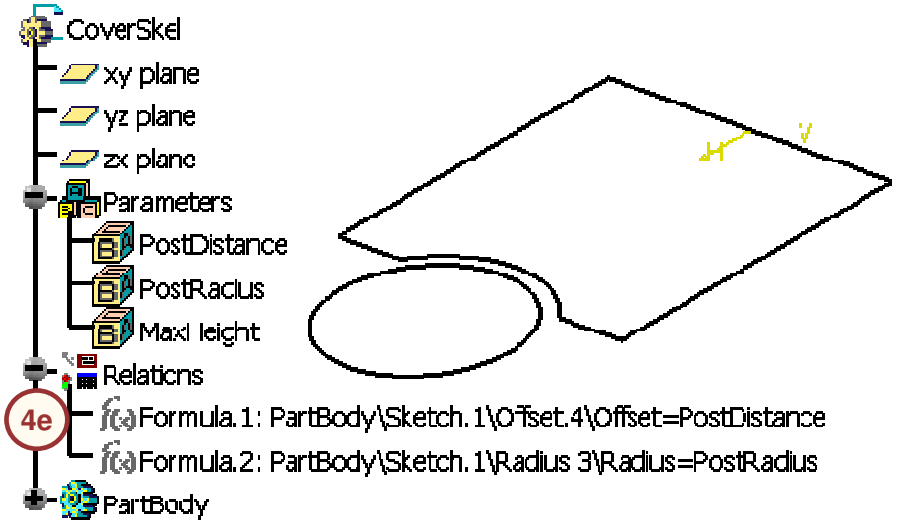
Do it Yourself (4/13)

4. Drive the sketch with parameters, (continued).

d. Edit the dimension equal to a radius of 6. Drive this dimension with the PostRadius dimension.

e. Expand the Relations node of the specification tree to view the resulting relations.



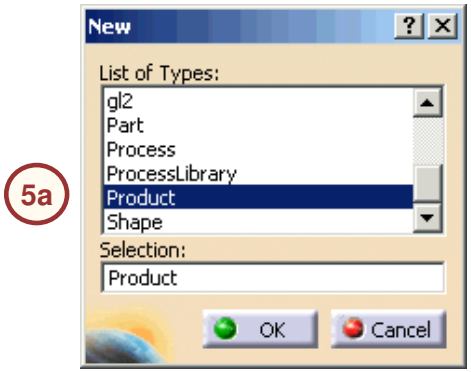


Student Notes:

Do it Yourself (5/13)

5. Create a product file.

- Create a new product file and assemble the skeleton model.
  a. Create a new product file.
  b. Rename the product to Cover and save the file.
  c. Assemble CoverSkel.CATPart and apply a Fix constraint.

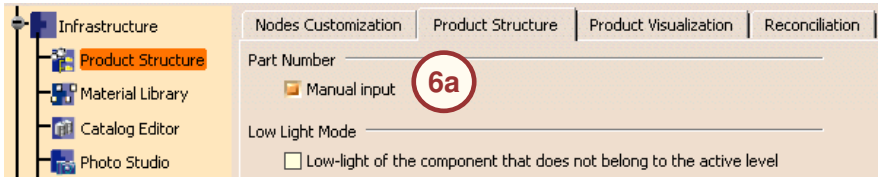


Student Notes:

Do it Yourself (6/13)

6. Create a part file.

- Change option settings and create a new component in the assembly.
  a. Activate the **Manual input** option under **Infrastructure > Product Structure > Product Structure** tab.
  b. Create a new part while in the Assembly Design workbench.
  c. Enter [Post] for the name of the new part.



Student Notes:

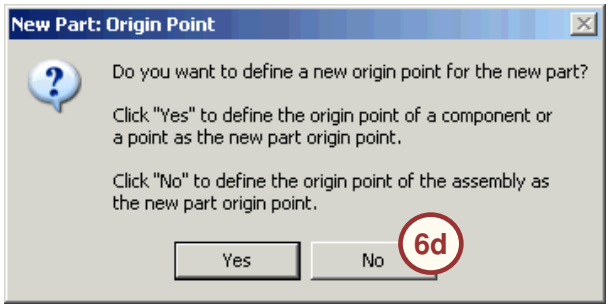
Do it Yourself (7/13)

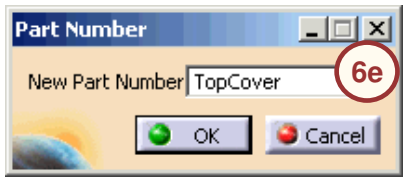
6. Create a part file (continued).

d. Select No from the New Part: Origin dialog box.

e. Create another part, enter TopCover as the name for the part file.

f. View the specification tree.



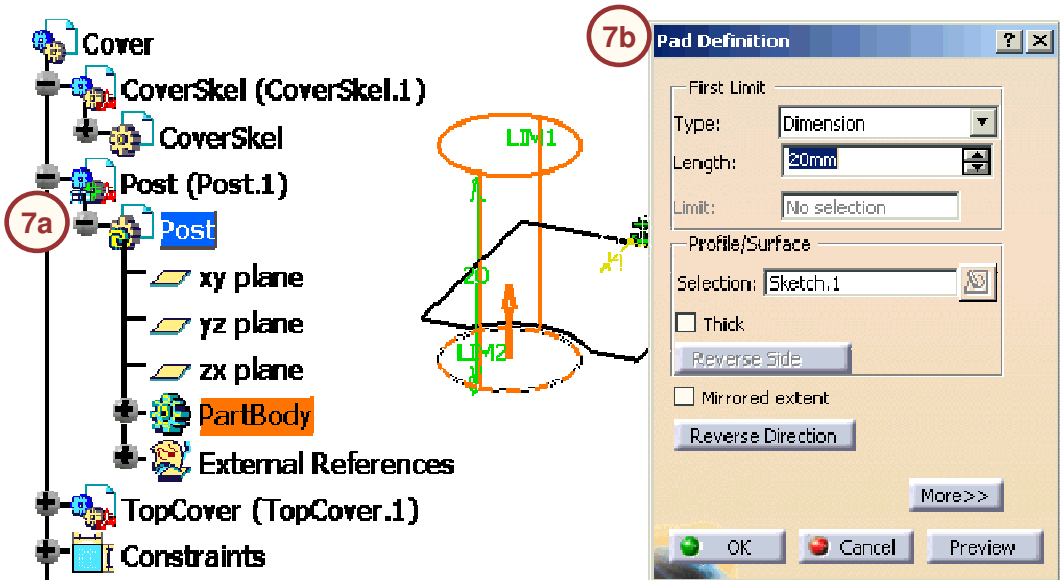


Student Notes:

Do it Yourself (8/13)

7. Create a solid feature.

- Create a pad for the Post part using a sketch from the skeleton.
  a. Activate the Post part.
  b. Create a pad from Sketch.1 of the CoverSkel part.

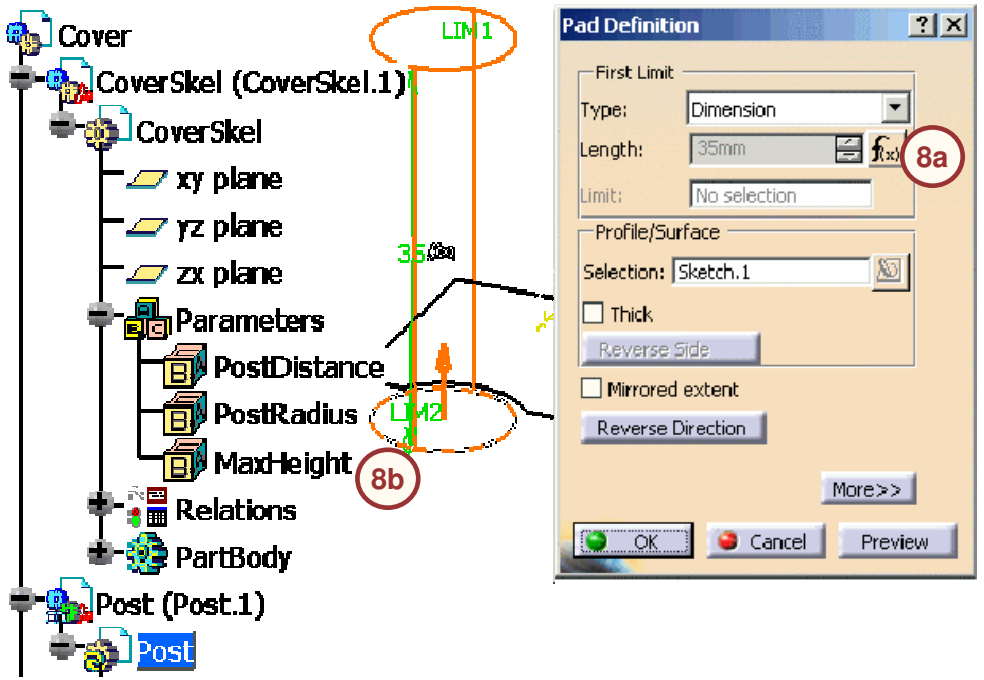


Student Notes:

Do it Yourself (9/13)

8. Use skeleton parameters.

- Use a skeleton parameter to drive the length of a pad.
  a. Edit the formula of the Length parameter.
  b. Select MaxHeight parameter from the skeleton.

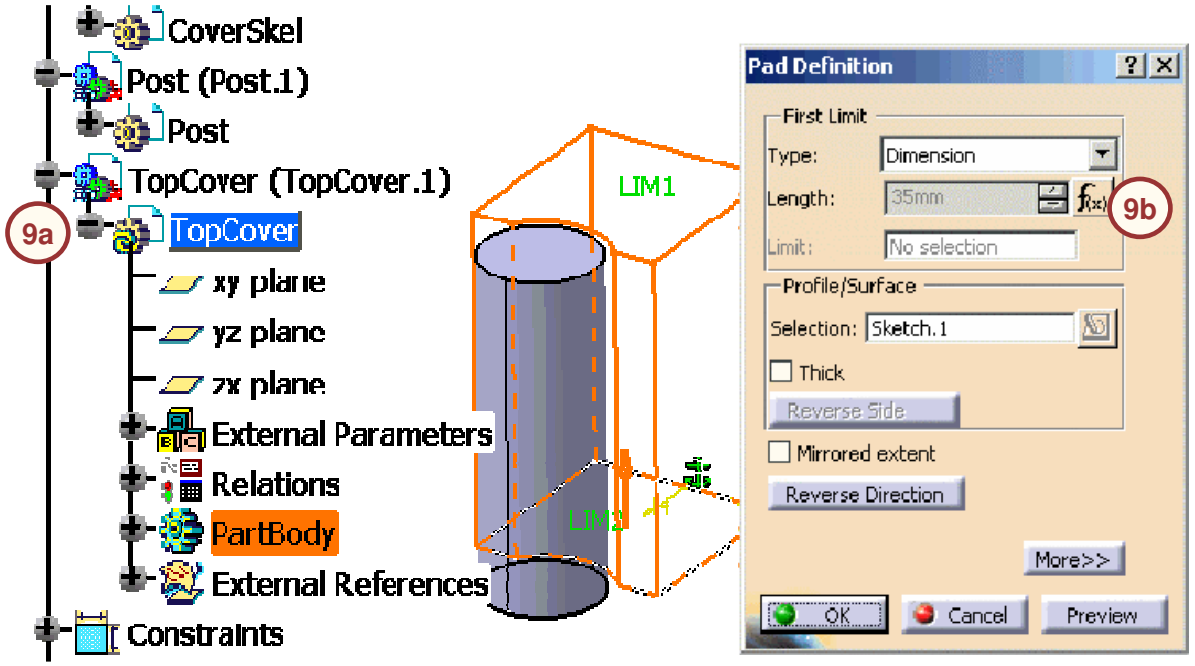


Student Notes:

Do it Yourself (10/13)

9. Create part geometry in context.

- Use a sketch and a parameter from the skeleton.
  a. Activate TopCover part.
  b. Use Sketch.2 and MaxHeight parameter from the skeleton model to define a pad feature.

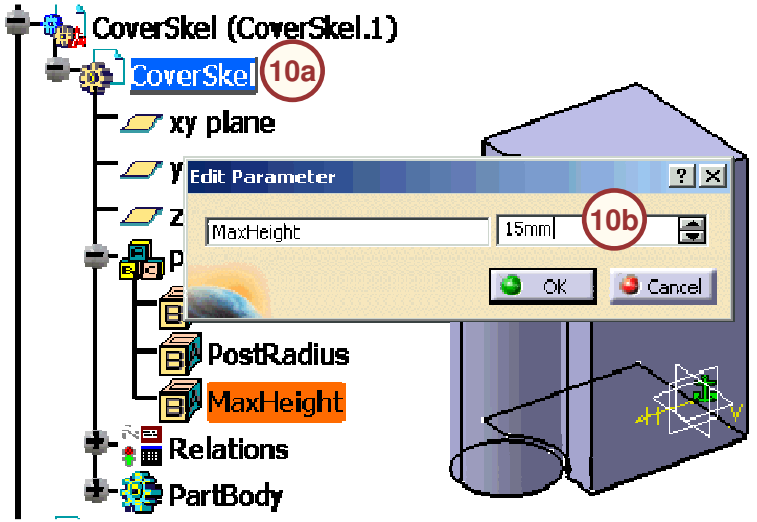


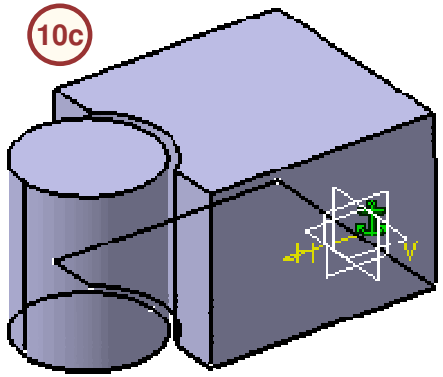
Student Notes:

Do it Yourself (11/13)

10.Change a skeleton parameter.

- Drive a part level change using the skeleton.
  a. Activate the CoverSkel part.
  b. Edit the MaxHeight parameter to [15].
  c. Activate the assembly and update. The two part files are driven by a skeleton parameter.





Do it Yourself (12/13)

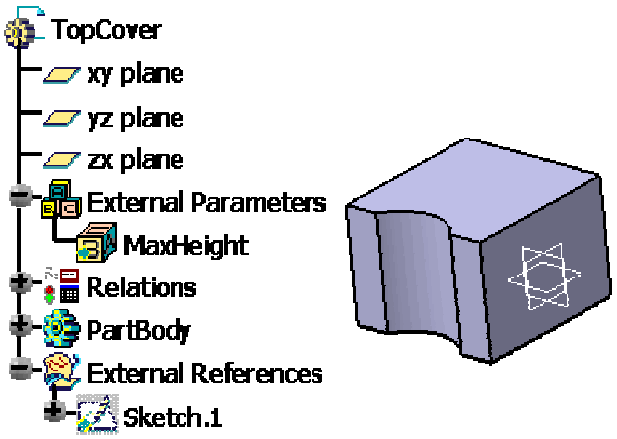
11.Add features to a part file.

a. Open TopCover part in a separate window.

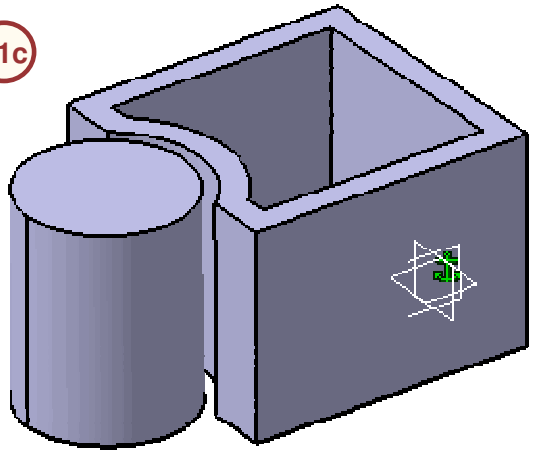
b. Shell the part to 2mm removing the top surface.

c. Activate the Cover.CATProduct file to view the change.

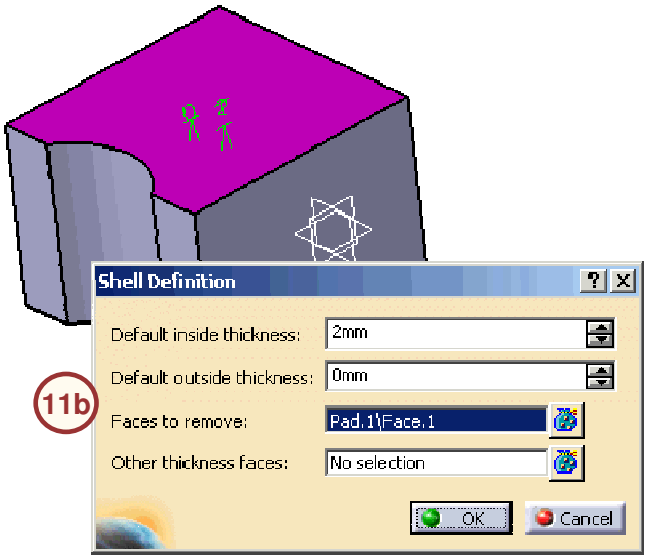
11a



11c



11b



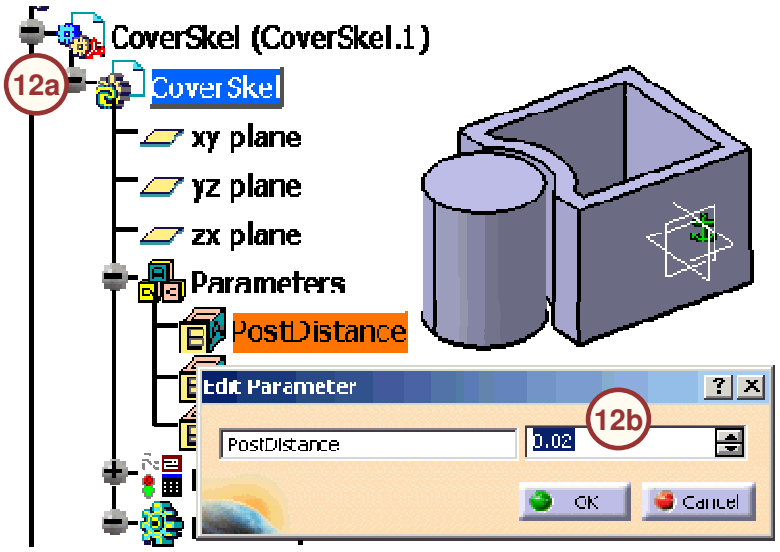
Student Notes:

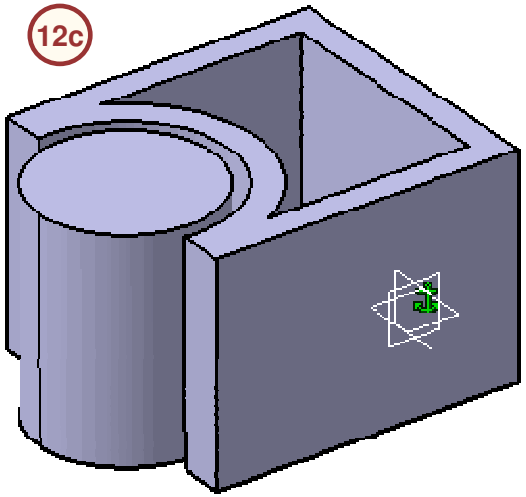
Student Notes:

Do it Yourself (13/13)

12.Drive part geometry from a skeleton.

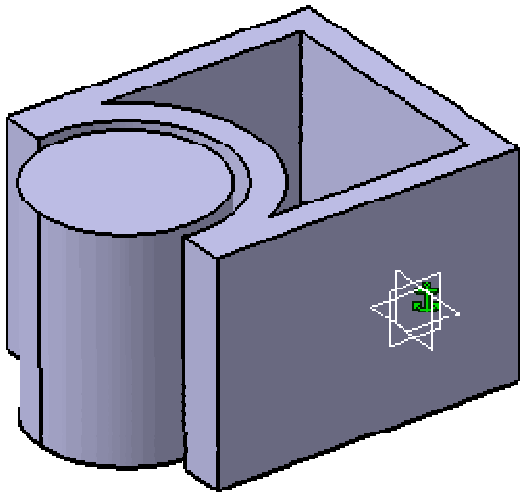
- Changes made to the skeleton affect referenced part geometry.
  a. Activate the CoverSkel part.
  b. Edit the PostDistance parameter to [20mm].
  c. Update the assembly to see the changes.





Exercise: Skeleton Parameter Use Recap

- ✓ Create a product using the skeleton method
- ✓ Reference geometry and parameters from the skeleton to drive component geometry



Student Notes:

Student Notes:

Exercise: Skeleton and Design in Context

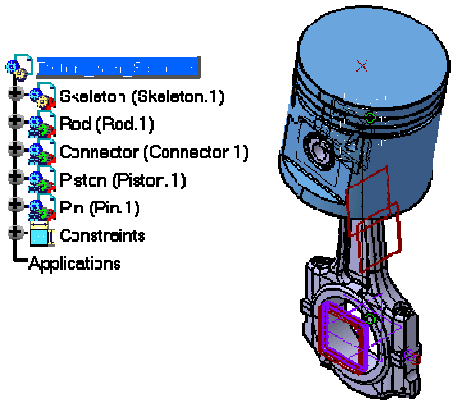
Recap Exercise

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:

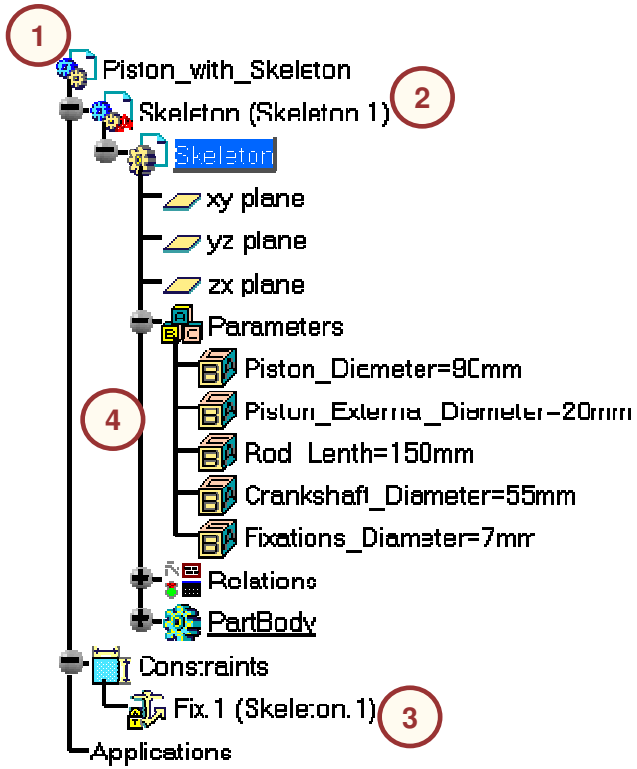
- Use the skeleton method
- Design a part in context using the skeleton model for external references
- Constrain an assembly using a skeleton model



Student Notes:

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.

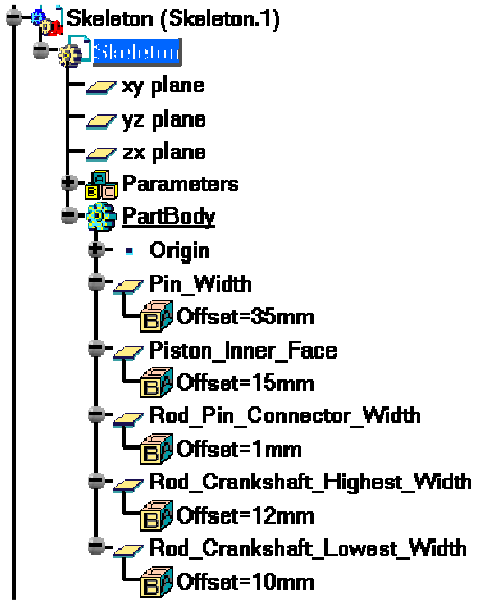


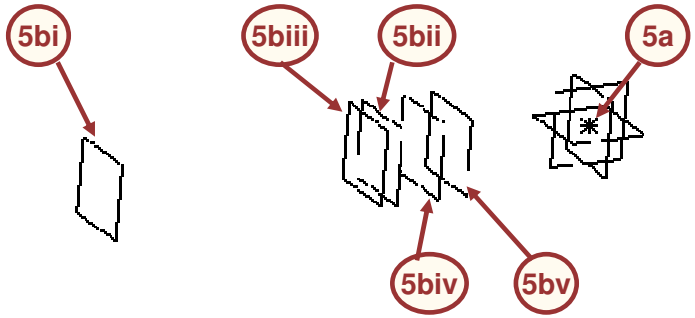
Student Notes:

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.



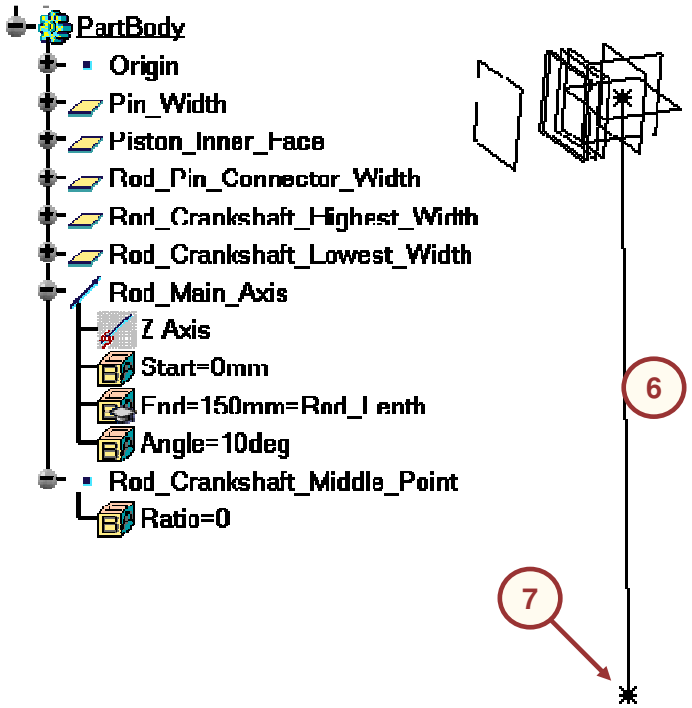


Student Notes:

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].

Student Notes:

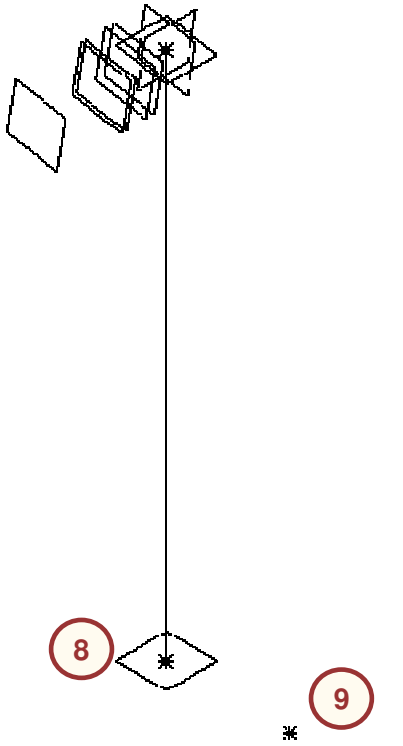
Do it Yourself (4/14)

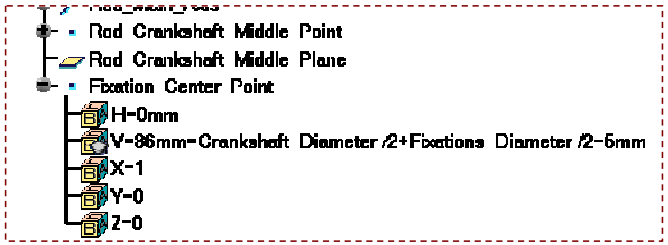
8. Create a plane.

- 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]



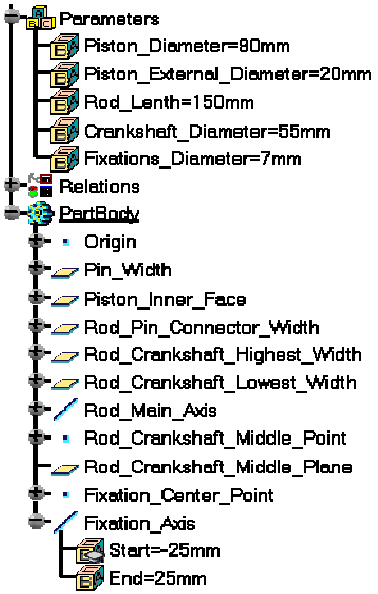


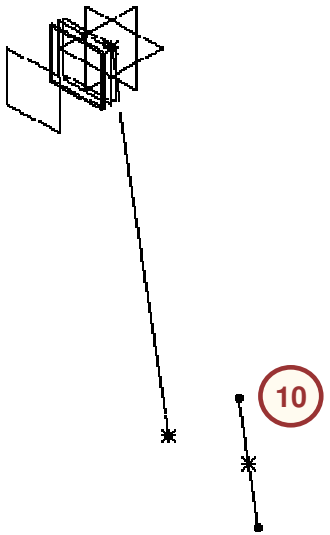
Student Notes:

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





Student Notes:

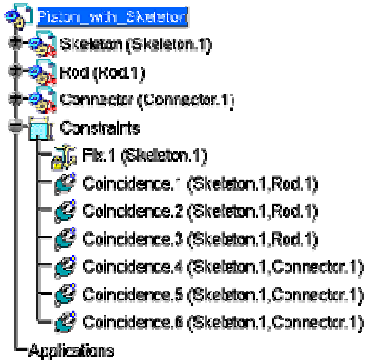
Do it Yourself (6/14)

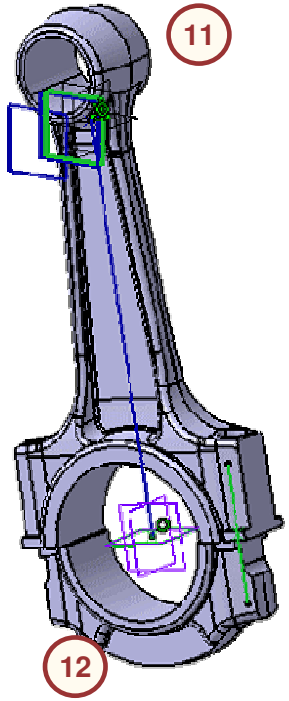
11. Insert an existing component.

- 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.



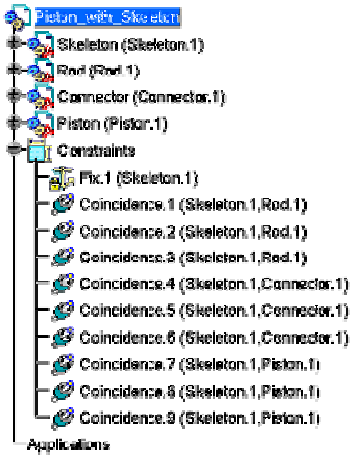


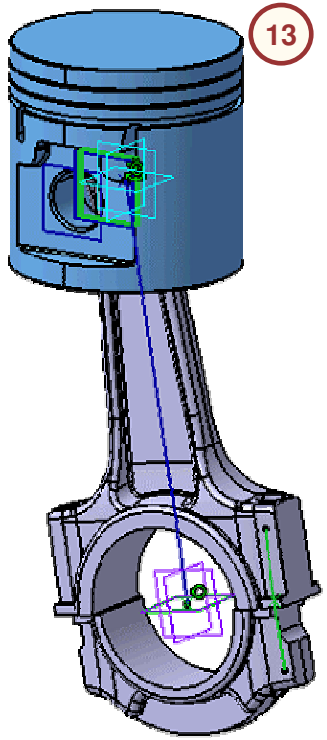
Student Notes:

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.



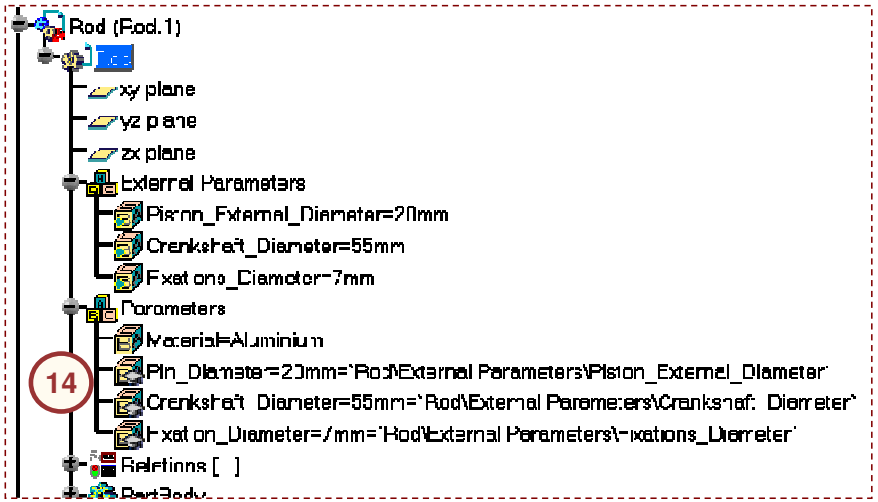


Student Notes:

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.

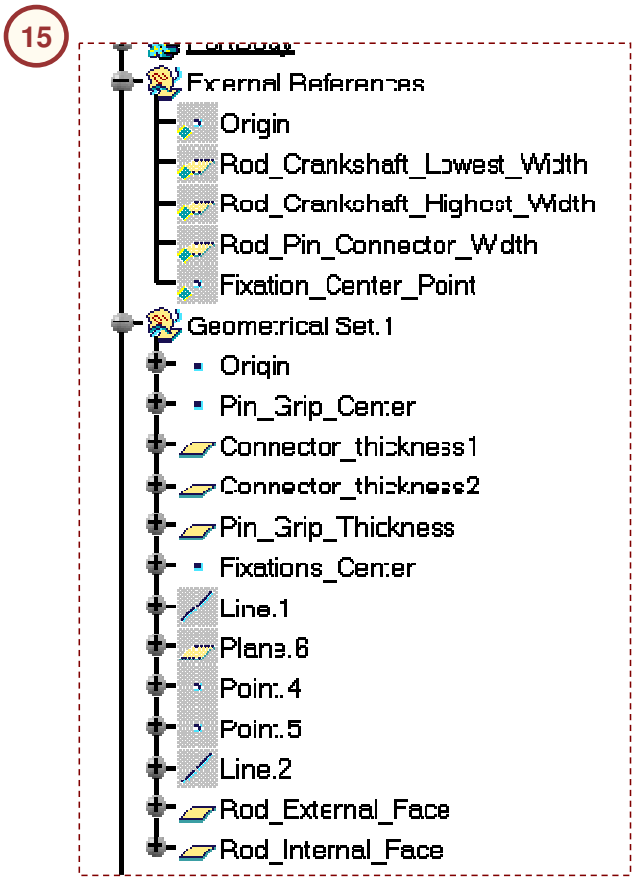


Student Notes:

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.



Student Notes:

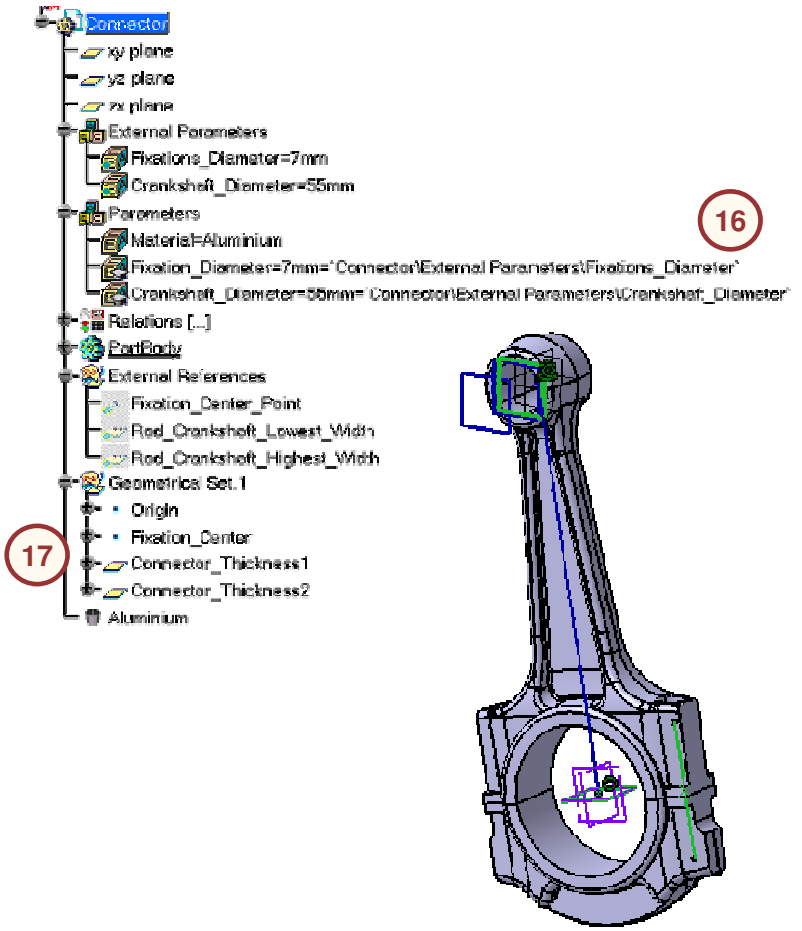
Do it Yourself (10/14)

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.



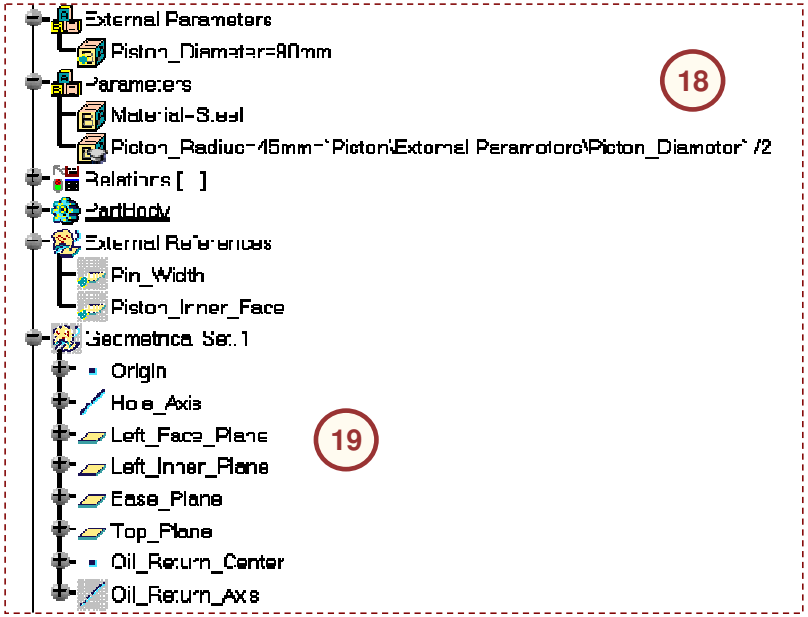
Do it Yourself (11/14)

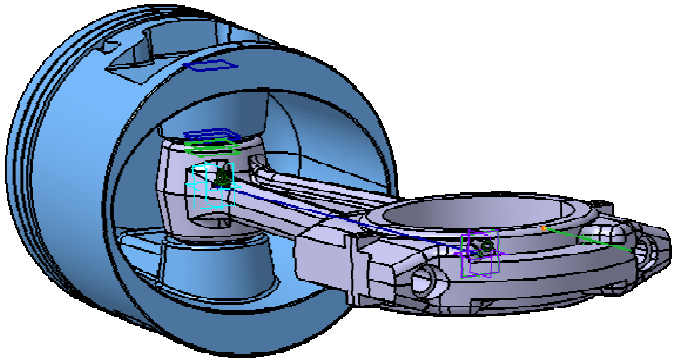
18. Link the piston parameters.

- 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.





Student Notes:

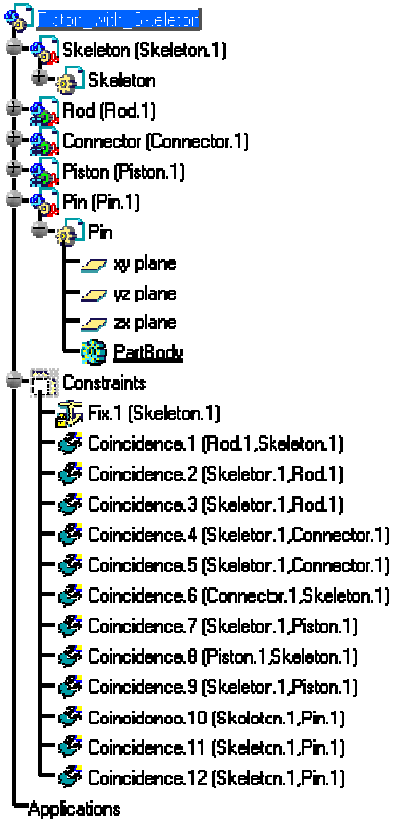
Do it Yourself (12/14)

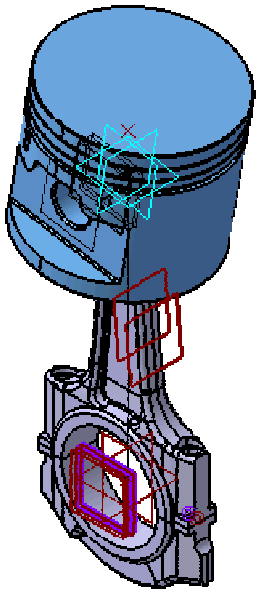
20. Create a new part.

- 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.





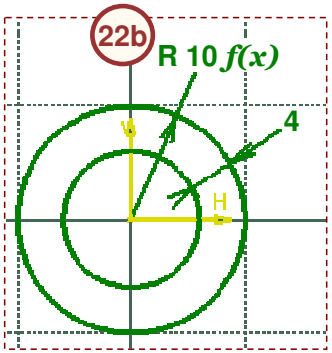
Student Notes:

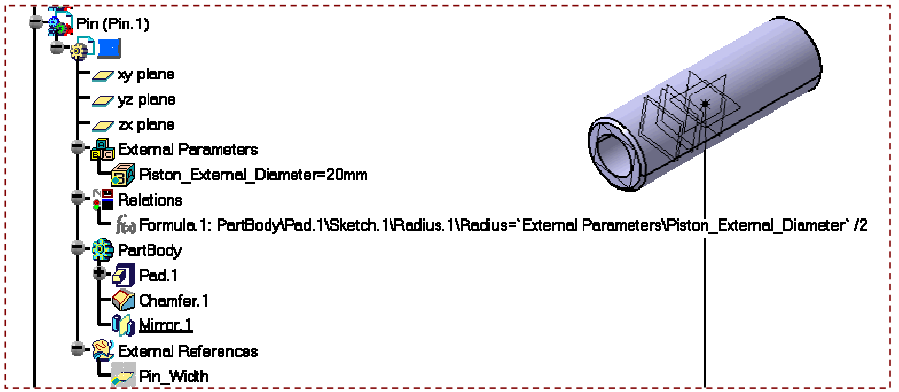
Student Notes:

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.





Student Notes:

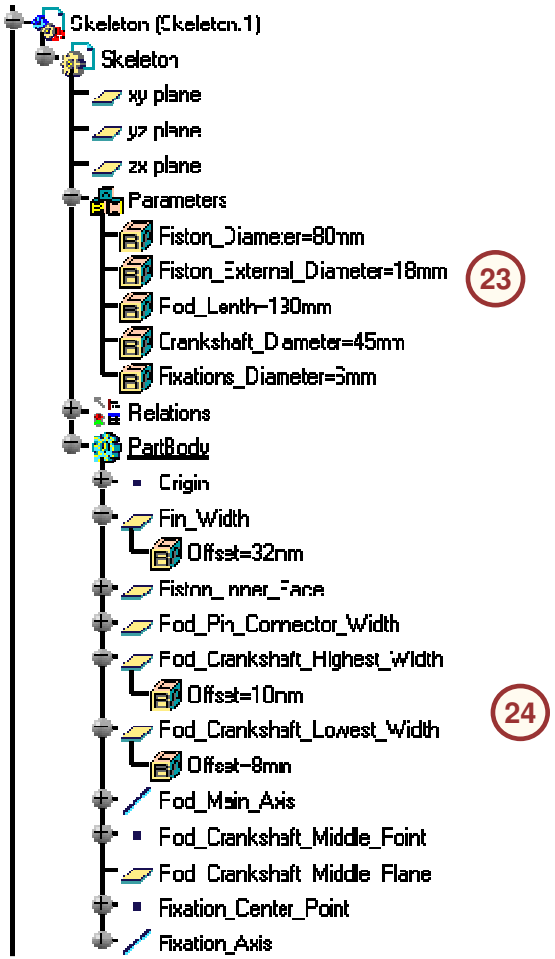
Do it Yourself (14/14)

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.

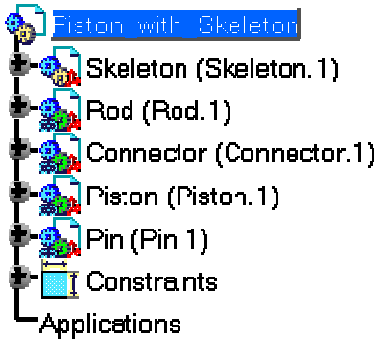
- Activate the top-level assembly and update it. Notice the changes made to the skeleton propagate through the entire assembly.

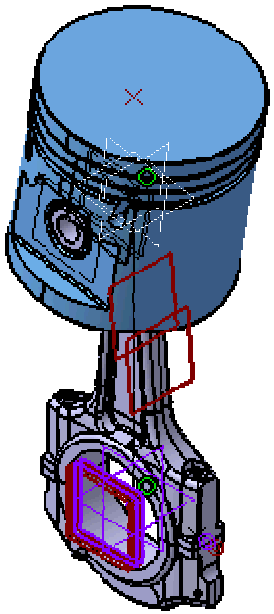


Student Notes:

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





Student Notes:

Create the Published Elements

In this section, you will learn what published geometry is and how to create it.

Use the following steps :

1. Create the skeleton model.
2. **Create the published elements**
3. Use the published elements

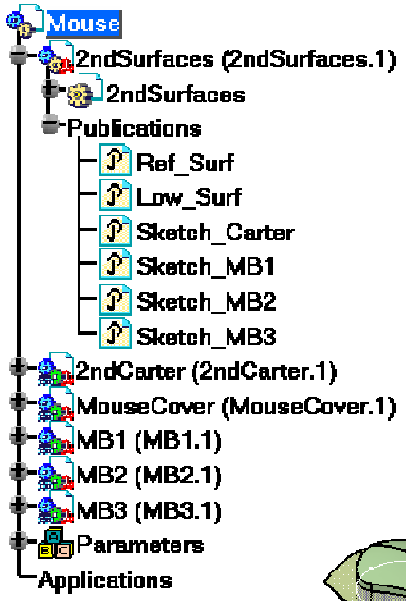
Student Notes:

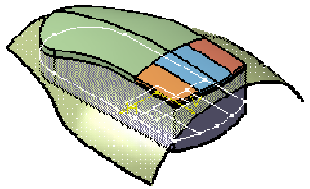
Introduction to Publishing Geometry

Publishing geometrical elements is the process of making geometrical features available to different users.

Although not essential, publishing geometry and parameters in a skeleton model is suggested to help control the external references created.

Publishing elements are not just used when applying the skeleton method. Consider using published elements anytime you want to control external references.





STUDENT GUIDE

Student Notes:

Why Publish Geometry?

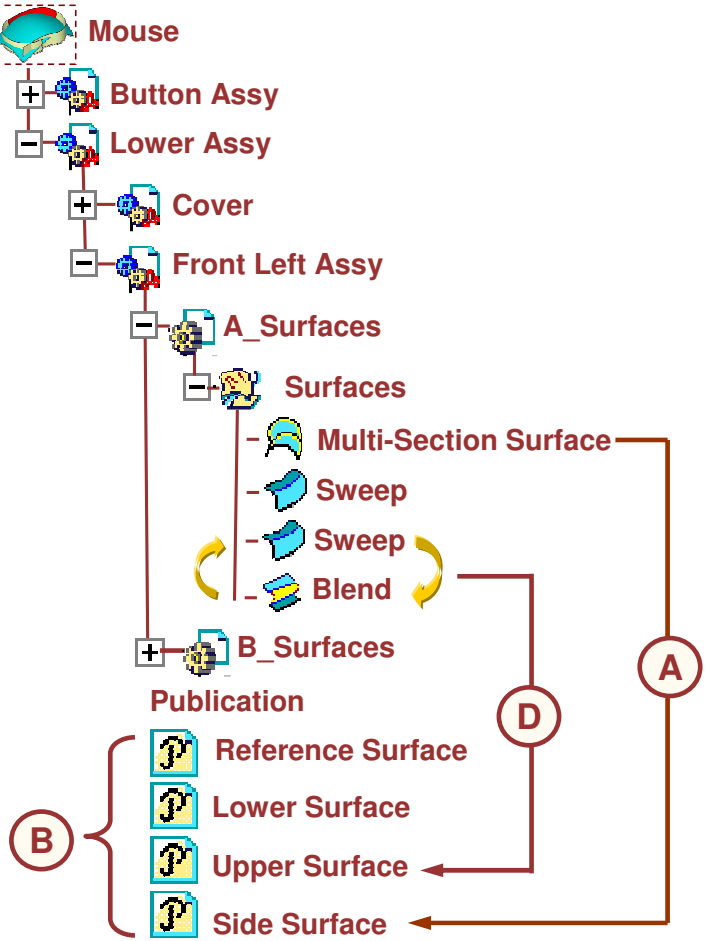
Publishing geometry has many benefits such as:

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 only select as external reference only the published elements.

D. Ease replacement of one feature of the part with another. Published elements that have same name in the source part and the child part are automatically reconnected, as you would have to reconnect them all one by one if they are not published.



Student Notes:

What Kind of Geometry Can be Published?

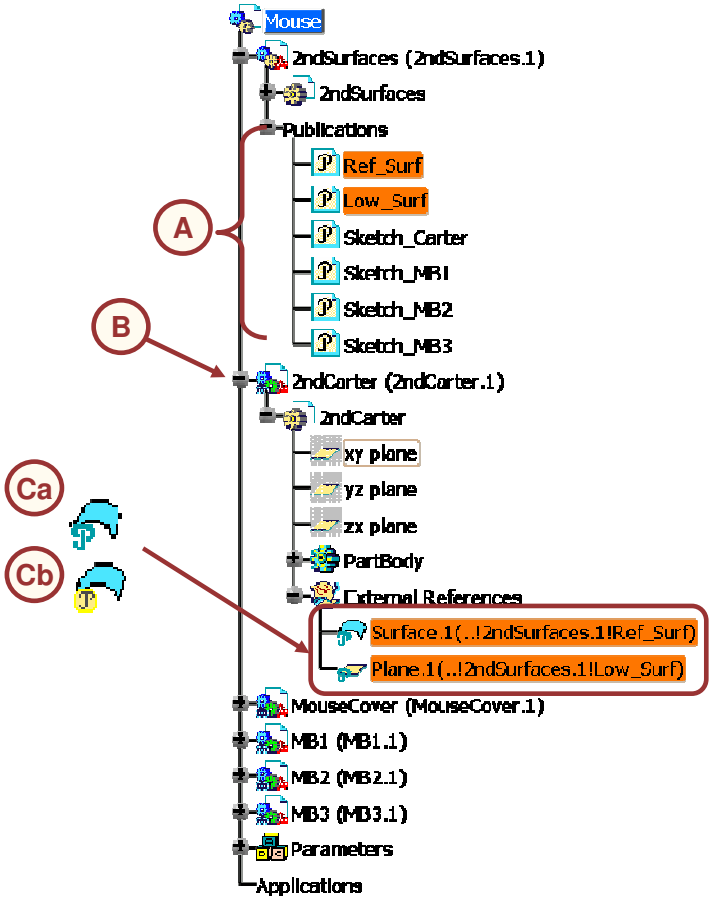
Many types of geometries can be published :

- Wireframe features (points, lines, curves, planes)
- Whole sketches
- Bodies (PartBody, other body)
- Part design features (pad, pocket, hole etc.)
- Generative Shape Design features (extrudes surfaces, offsets, joins etc.)
- Free Style Design features (planar patches, curves etc.)
- Sub elements of all geometrical elements (faces, edges, vertices etc.)

Published Elements in the Tree

Published elements can be identified in the specification tree.

A. The tree displays names of published elements under the components Publication node.
B. The green gear on a component icon indicates that the component has been designed using external references.
C. When a published element is used, it is denoted in the external references node.
   a. Elements that are updated are denoted by the letter P in a cyan color.
   b. Published elements that are not synchronized are denoted by a P in a yellow circle.



Student Notes:

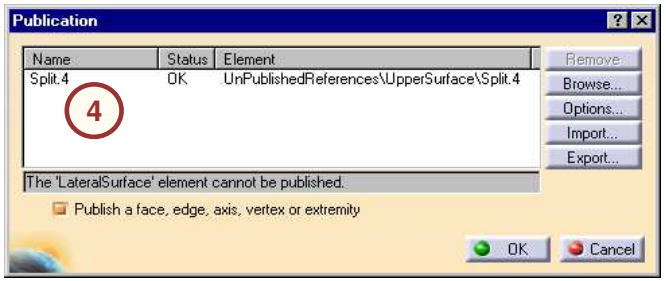
STUDENT GUIDE

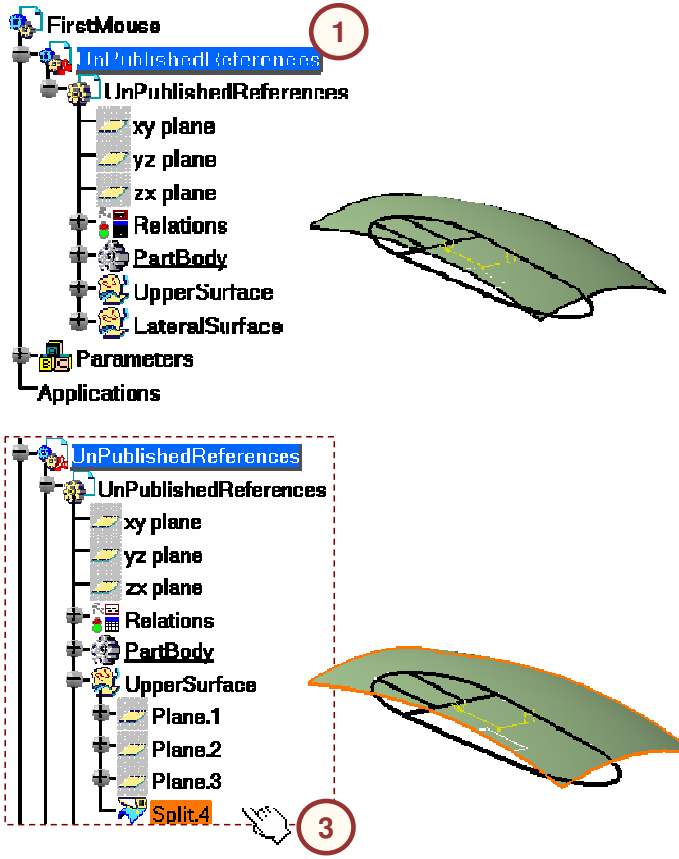
Student Notes:

Publishing Geometry (1/2)

Use the following steps to publish geometry:

1. Activate the components that contains the geometry to be published.
2. Click **Tools > Publications**.
3. Select the geometrical element to publish.
4. The selected geometry is added to the publication window.



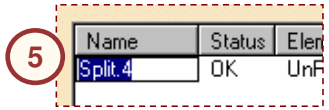


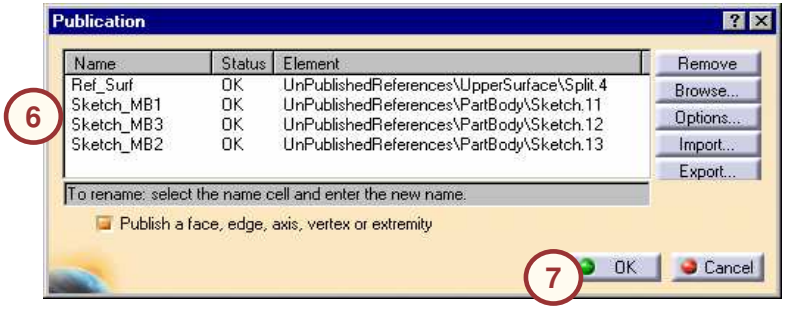
Student Notes:

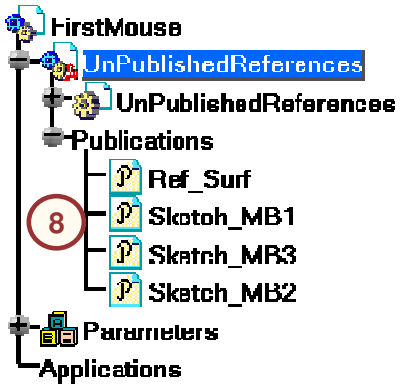
Publishing Geometry (2/2)

Use the following steps to publish geometry (continued):

5. To rename the published elements, select in the row with the element to rename and activate it. Select in the field again to edit the name.
6. Repeat step 3 to 5 to publish other elements.
7. Click on **OK** to validate.
8. The published elements are displayed under the publication node of the components.





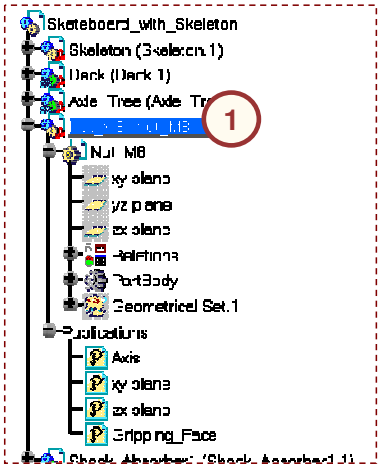


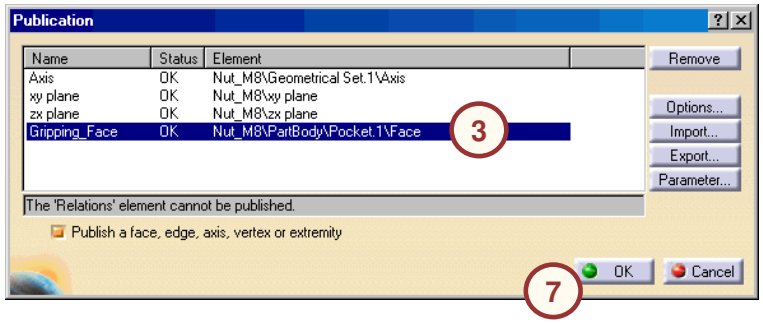
Student Notes:

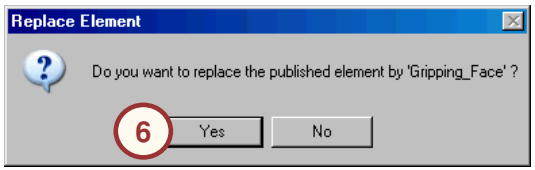
Changing a Published Element

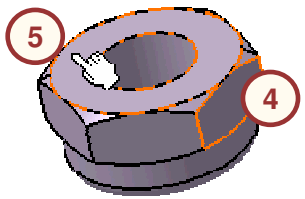
If necessary, the geometrical element referenced by a publication can be replaced using the following steps:

1. Activate the component containing the published geometry to be replaced.
2. Click **Tools > Publications**.
3. Select the publication to replace.
4. The current geometrical element highlights on the model
5. Select the replacing geometrical element.
6. Click **Yes** from the Replace Element dialog box.
7. Click **OK** to close the publications dialog box.









Student Notes:

Publishing Parameters

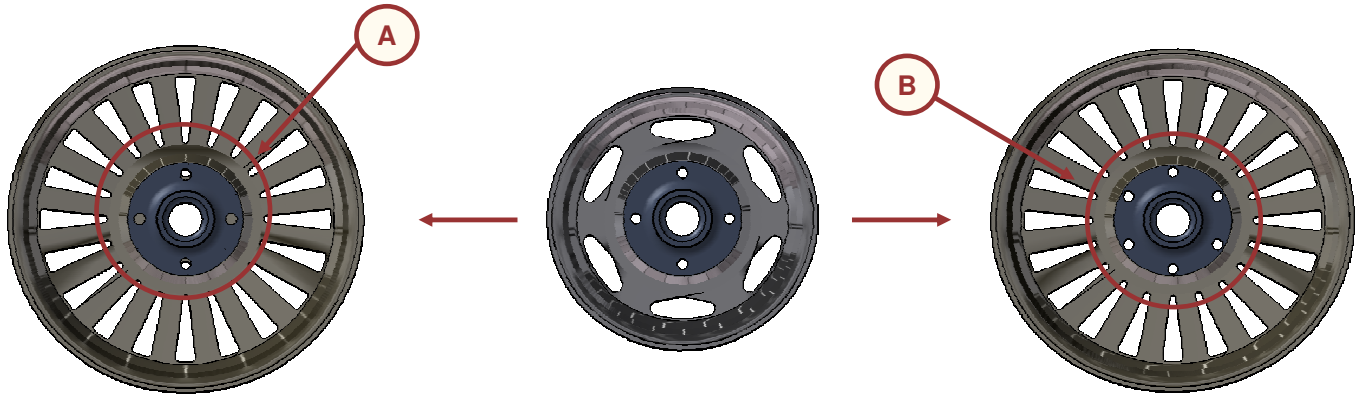
Publication of parameters is useful while replacing a component in an assembly that contains parameters used to drive other components (i.e., exported parameters).

If the exported parameters are published and the parameters of the replacing component are published under the same names, they will inherit the control of the exported parameters.

Otherwise the parameters of the replaced component will keep the control.

For example, the number of holes and pattern diameter of the rim are reused in the hub.

A. If the parameters are not published, the hub will continue to sue the parameters for the original rim and will not update.
B. If the parameters are published and the rim is replaced with a bigger one, the parameters update to the new values.

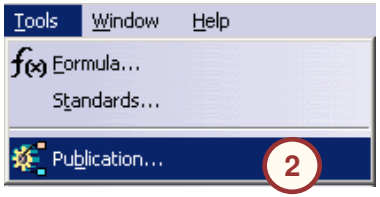


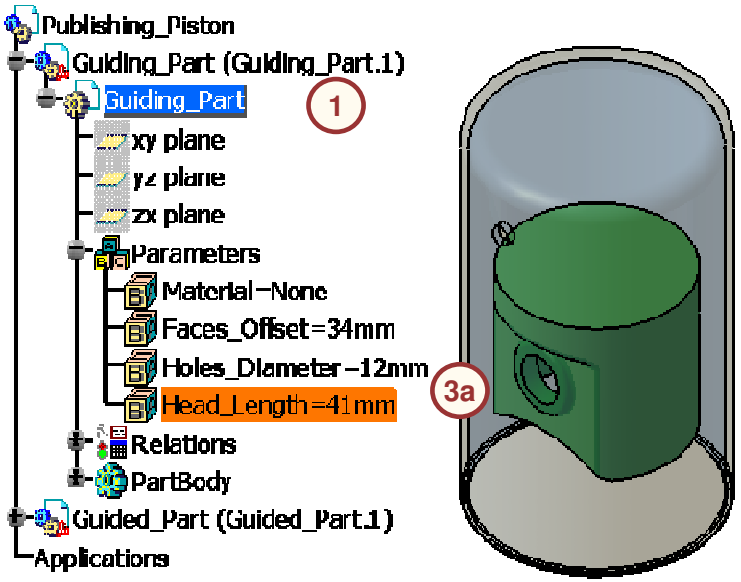
Student Notes:

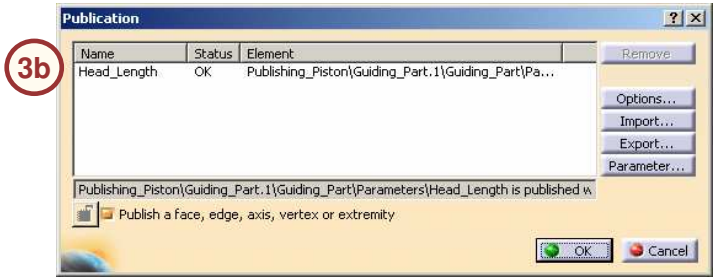
Publishing a Parameter (1/3)

Use the following steps to publish a parameter:

1. Activate the component containing the parameter(s) to be published.
2. Click **Tools > Publication.**
3. For a user defined parameter:
   a. Click on the parameter from the specification tree
   b. The user parameter appears in the list of published elements.





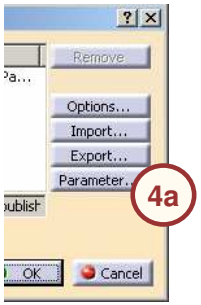


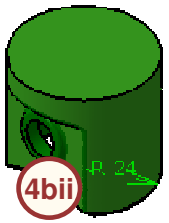
Student Notes:

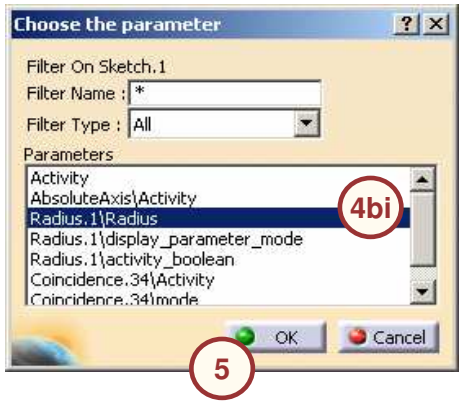
Publishing a Parameter (2/3)

Use the following steps to publish a parameter (continued):

4. To publish an intrinsic parameter:
   a. Select Parameter from the Publications dialog box.
   b. Select the parameter
      i. From the **Choose the parameter** dialog box.
      ii. By selecting the appropriate geometry to display the parameter in the model. Select the parameter to highlight it in the **Choose the Parameter** dialog box.
5. Click **OK** to add the parameter to the list.





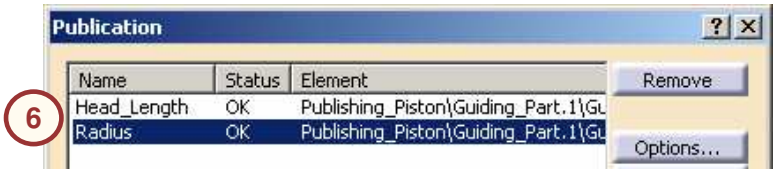


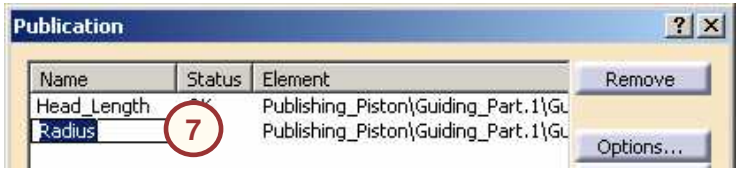
Student Notes:

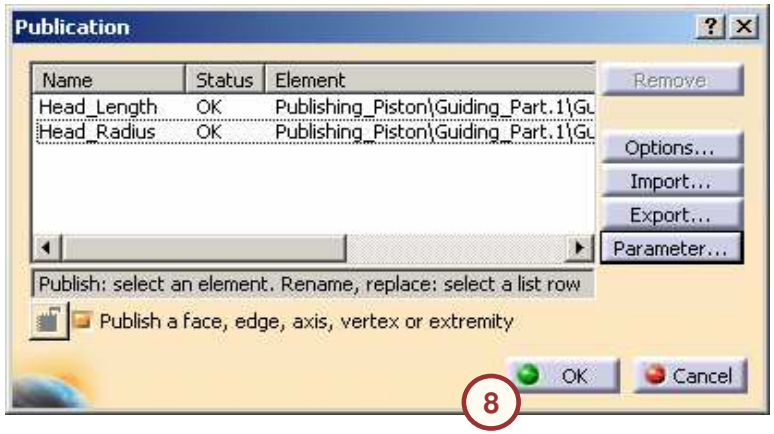
Publishing a Parameter (3/3)

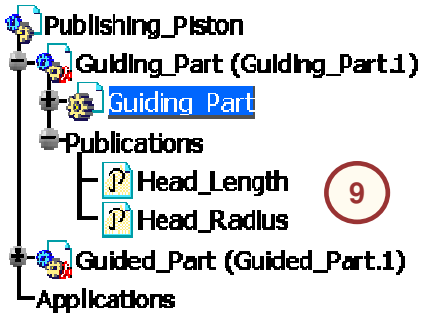
Use the following steps to publish a parameter (continued):

6. The published parameters appear in the list with a default publication name.
7. To rename the publication select the publication to highlight it and select it again to edit the name.
8. Click **OK** to validate the publication.
9. The newly published parameters appear under the publications of the component.









STUDENT GUIDE

Student Notes:

Use the Published Elements

In this section, you will learn how to use published elements.

Use the following steps :

1. Create the skeleton model.
2. Create the published elements
3. **Use the published elements**

Student Notes:

When Can You Use Published Geometry?

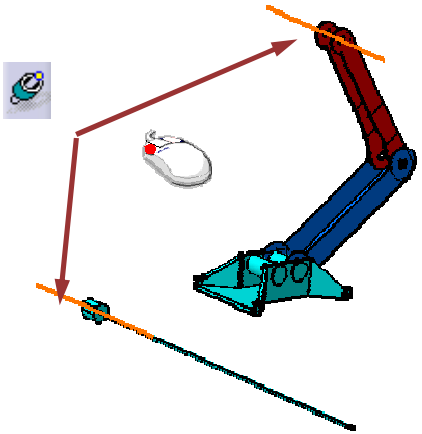
Published geometry can be used to control external references when:

A. Constraining an assembly.

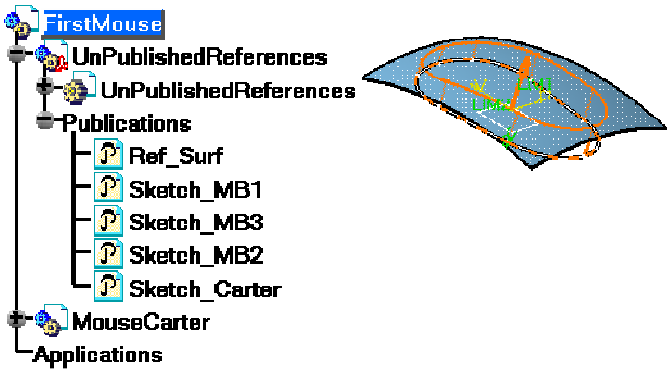
B. Designing in context.

It is particularly useful when replacing components with assembly constraints or that have been designed in context.

A



B

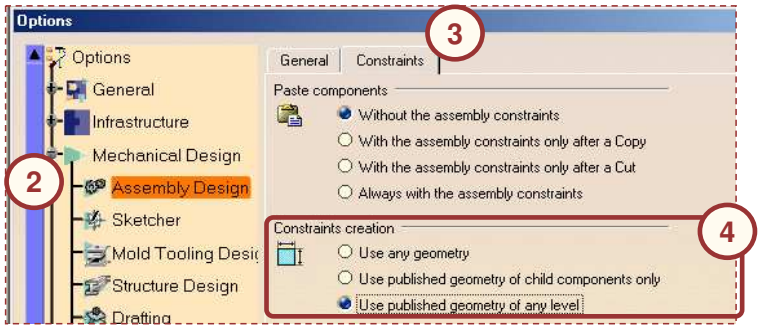


Student Notes:

User Setting: Use Published Geometry to Constrain (1/3)

Use the following steps to configure CATIA to accept only published geometry when constraining an assembly:

1. Click **Tools > Options**.
2. From the Option dialog box select **Mechanical Design > Assembly Design**.
3. Choose the Constraints tab.
4. Three options for constraints creation are available.
   a. Selecting **Use any geometry**, the default option, lets you select any geometry within the assembly for constraining references.
   b. Select the **Use published geometry of child components only** to only allow constraint reference belonging to child components.
   c. Select **Use published geometry of any level** to use any published geometry when constraining.



Student Notes:

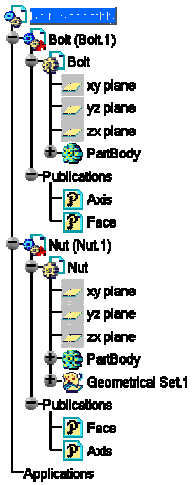
User Setting: Use Published Geometry to Constrain (2/3)

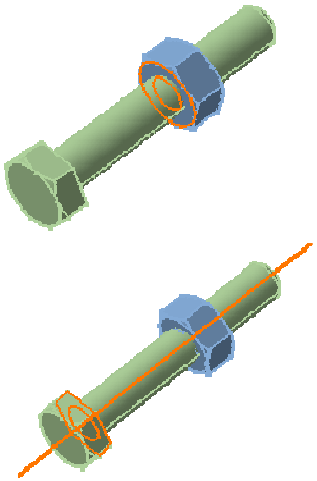
To understand the difference between the Use published geometry of child components only and the Use published geometry of any level, consider the following example

A. A bolt assembly is created consisting of a bolt and a nut. The bolt and nut axes and the faces shown are published inside bolt assembly.

B. The assembly is then inserted into another assembly containing two planes. The face of the nut need to be constrained to the face of the plane.

A





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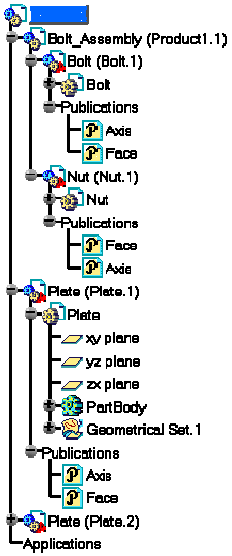
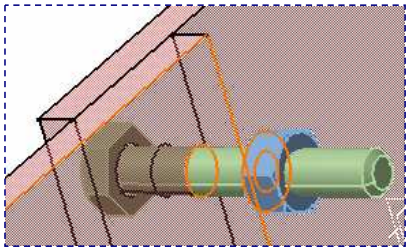




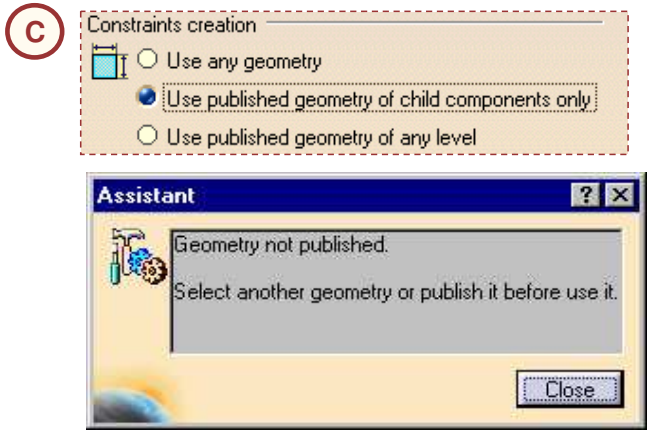
Student Notes:

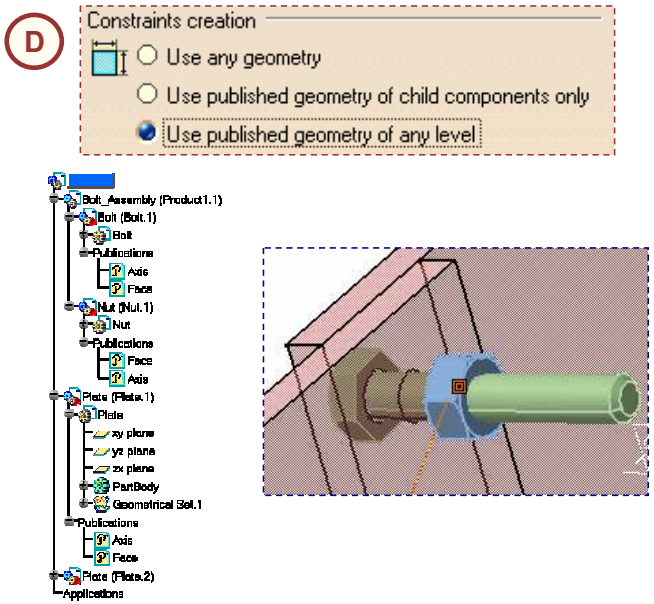
User Setting: Use Published Geometry to Constrain (3/3)

To better understand the difference between the User published geometry of child components only and the Use published geometry of any level, consider the following example (continued):

C. If the option **Use published geometry of child components only** was selected, the constraint will not be allowed. The published geometry would need to be created at the Bolt assembly level to be considered a child component.

D. If the option **Use published geometry at any level** was select the constraint can be created using the published geometry from the sub-level.





Student Notes:

User Setting: Only Use Published Geometry for External References

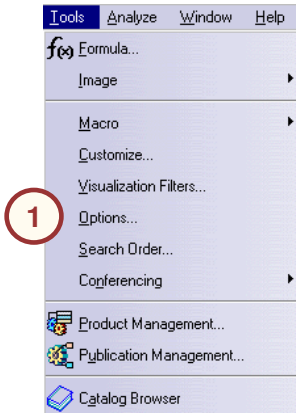
Use the following steps to configure CATIA to only allow published elements to be selected when creating geometry in context:

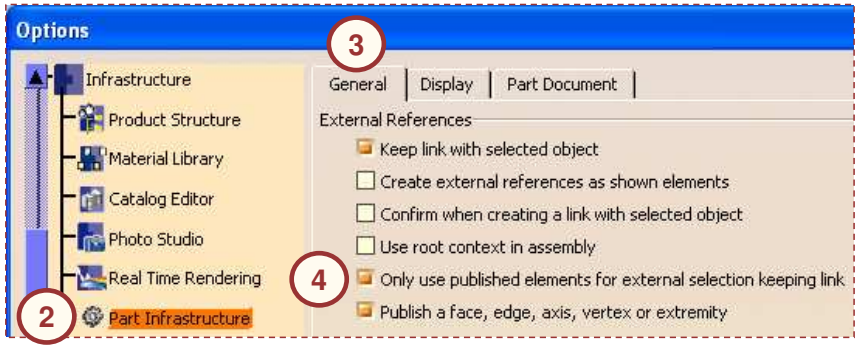
1. Click **Tools > Options**.

2. From the options dialog box select **Infrastructure > Part Infrastructure**.

3. Choose the General tab.

4. Select the **Only use published elements for external selection keeping link** option.





Using Published Geometry in Contextual Design (1/2)

Use the following steps to use published geometry as an external reference to design associative parts in context of the assembly:

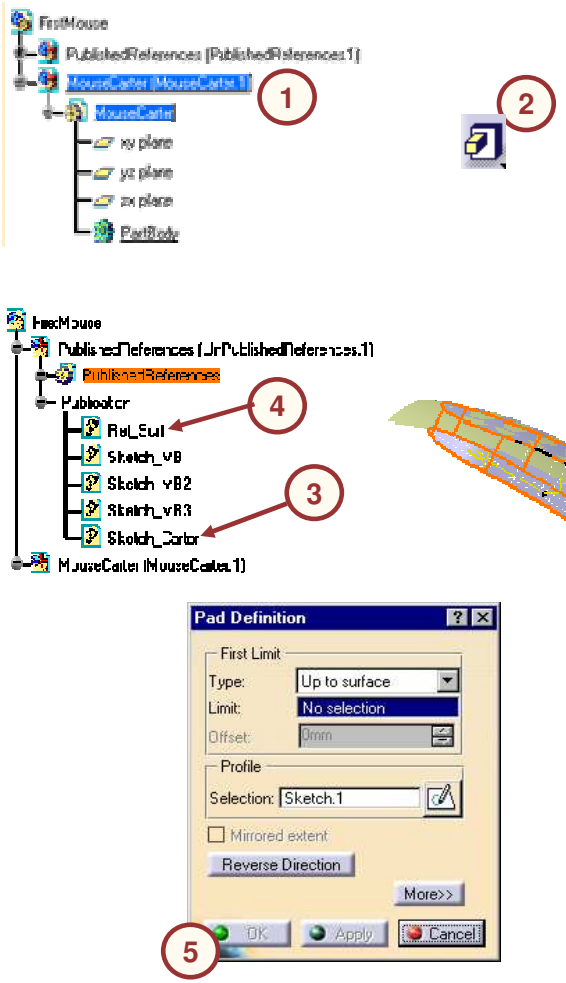
1. Activate the part.

2. Select the feature tool. In this example, a pad is created.

3. Select the profile sketch. In this example a published sketch from a different component is selected.

4. Specify the limits. In this example, the pad is limited **Up to surface**. The limiting surface selected is a published surface from another component.

5. Click **OK** to create the feature.



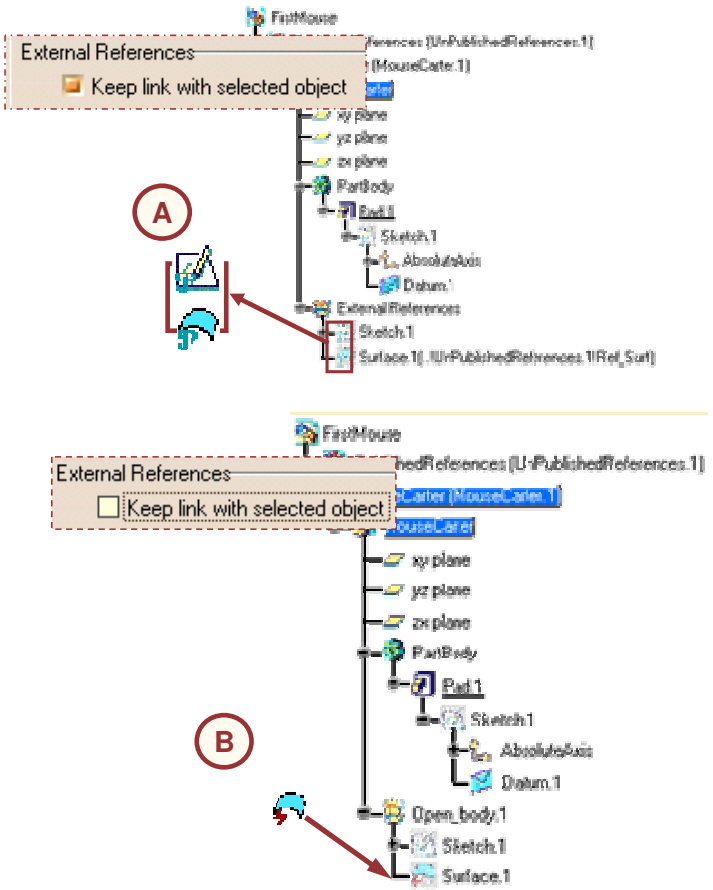
Student Notes:

Student Notes:

Using Published Geometry in Contextual Design (2/2)

Use the following steps to use published geometry as an external reference to design associative parts in context of the assembly (continued):

6. The published geometry will appear in one of two spots in the specification tree, depending on the external reference option:
   A. If the **Keep link with selected object** option is selected, the published geometry appears under the External Reference node. Published geometry is denoted by the letter P in its specification tree icon.
   B. If the **Keep link with selected object** option is cleared, the published geometry will appear in a geometrical set. The specification tree icon for the element will have a red lightening symbol, indicating that the element is non-associative. That is, when changes occur to the original element it will not propagate.



Student Notes:

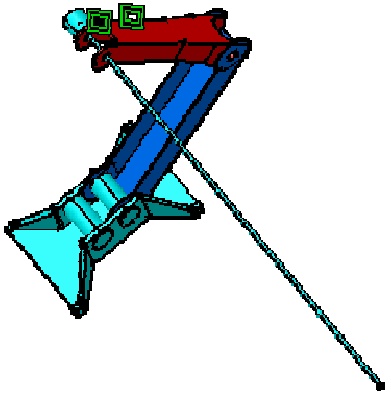
What is Replacing Published Component?

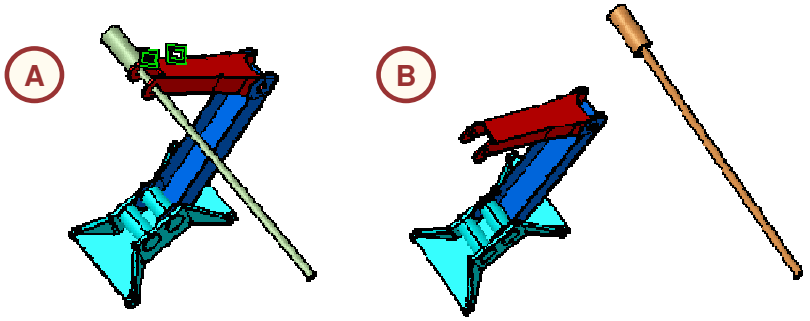
Published geometry is useful when replacing a component that is involved in a constraint or driving other contextual components.

For example, the rod component is replaced in the assembly shown:

A. If the geometry is published, the constraints will be preserved.

B. Without published geometry, the constraints will need to be reconnected.





Student Notes:

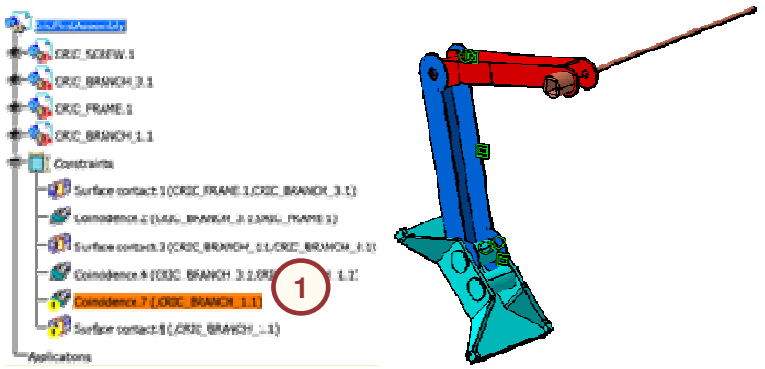
Reconnecting a Constraint (1/2)

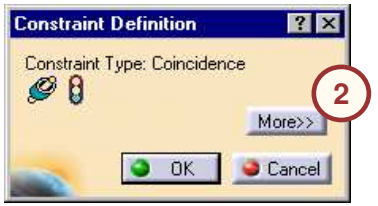
A constraint can become unresolved after the replacement of a component or if the constraint was connected to a wrong geometric element.

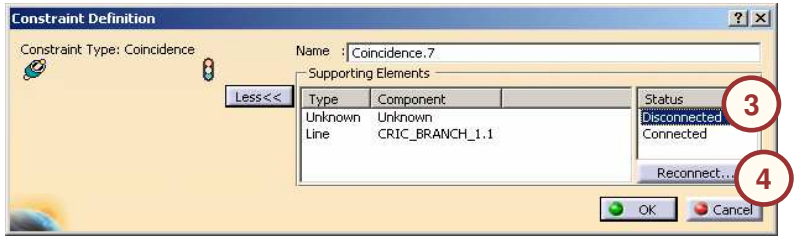
For example, if the rod in the previous example is replaced, and no publications are used, the constraints using elements from the Rod component will fail. The constraints will require reconnection to the new rod.

Use the following steps to change the geometric reference for a constraint:

1. Double-click the constraint to edit.
2. Select More to expand the dialog box.
3. Select the geometric element to reconnect from the Supporting Elements window. In this example, the disconnected constraint is selected.
4. Select Reconnect.





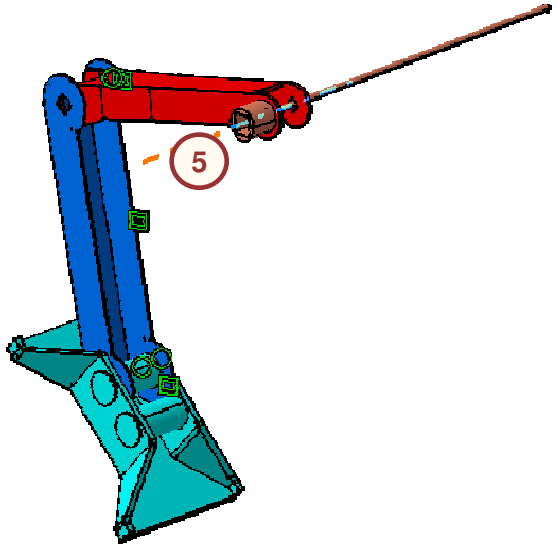


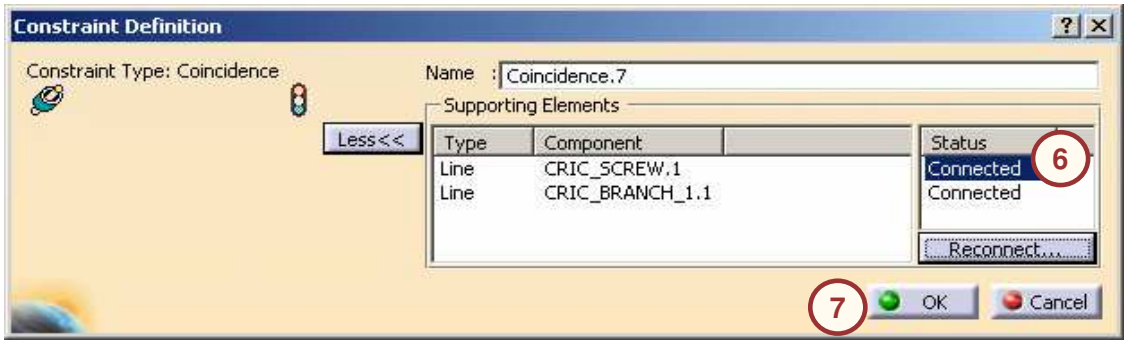
Student Notes:

Reconnecting a Constraint (2/2)

Use the following steps to change the geometric reference for a constraint (continued):

5. Select the replacing geometric element. In this example, the axis of the rod is selected.
6. The new element is listed in the Supporting Elements window.
7. Click **OK** to update the constraint.





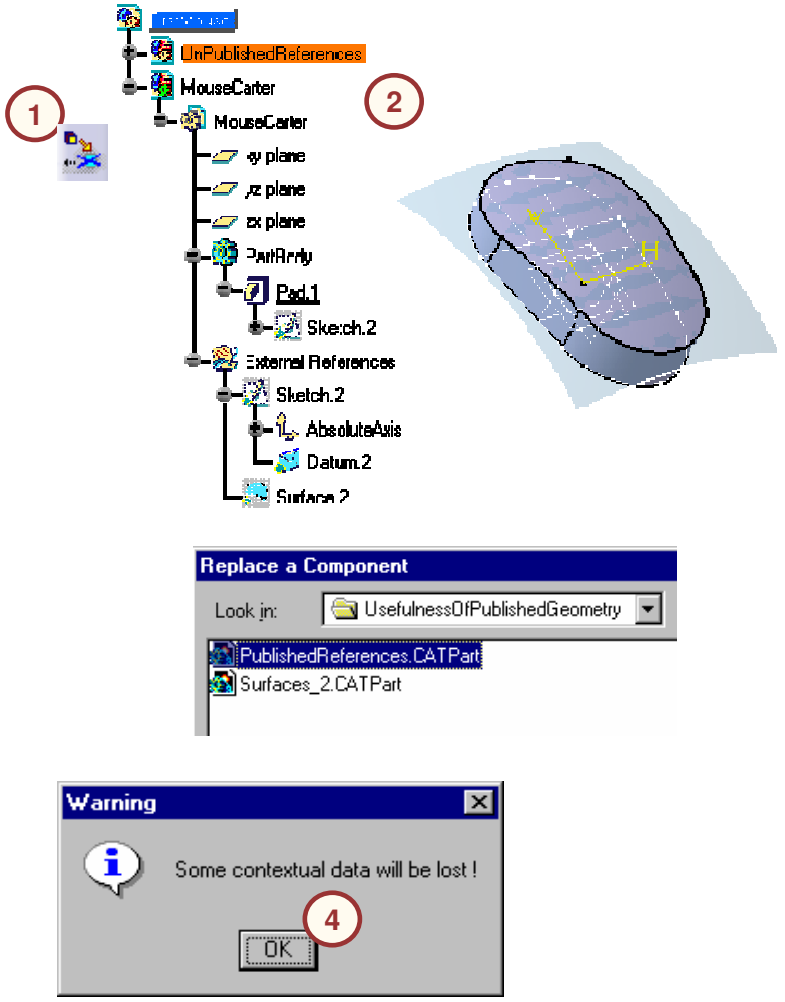
Student Notes:

Replacement Of a Non-Published Driving Component (1/3)

When you replace a component that is used as a reference for other contextual components, the driven components need to be reconnected to the new driving geometry.

Use the following steps to replace a non-published component that is referenced by other components:

1. Select the Replace Component icon.
2. Select the component to be replaced. In this example the Unpublished References component is selected.
3. Select the replacing component.
4. A warning dialog box appears indicating that contextual data will be lost. Click **OK**.

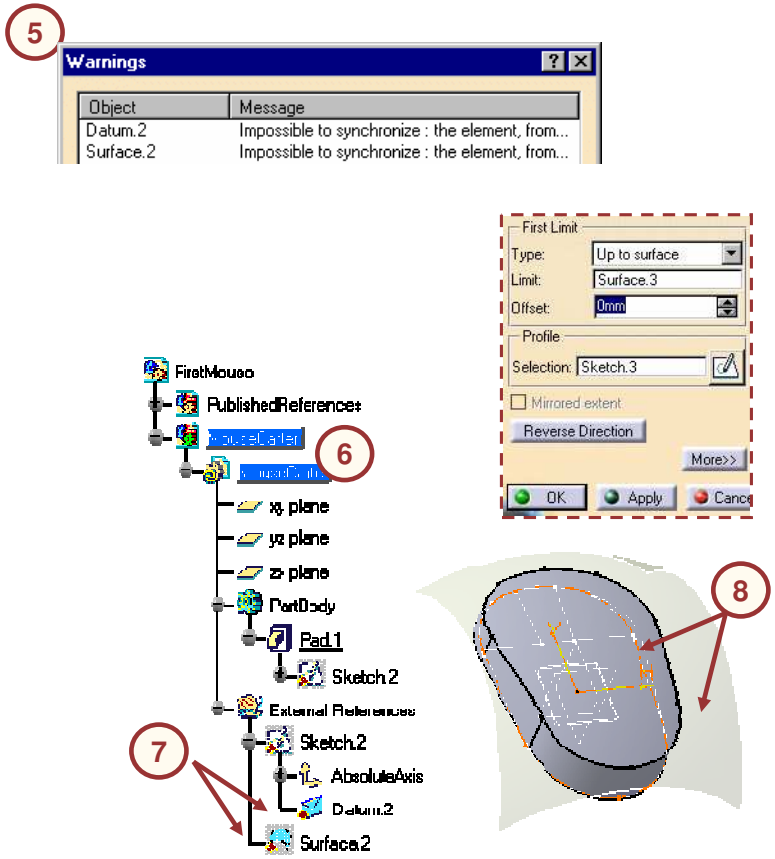


Student Notes:

Replacement Of a Non-Published Driving Component (2/3)

Use the following steps to replace a non-published component that is referenced by other components (continued):

5. A second warning dialog box appears indicating the geometry that is no longer synchronized because their references are lost.
6. Activate the failing part.
7. Notice that the external reference has a red circle on its specification tree icon. This indicates that the element is no longer synchronized. It is not synchronized because the referencing element has been removed from the assembly.
8. Edit the feature that is missing its references. In this example, the Profile and the limiting element both need to be re-defined. Select new references for both missing elements.



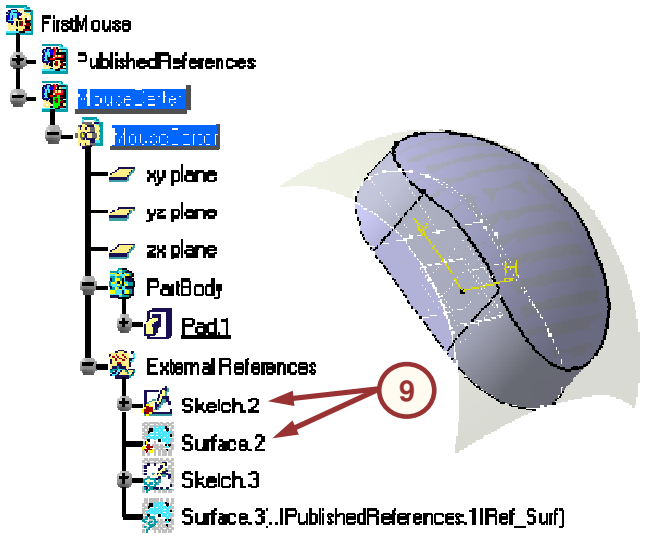
STUDENT GUIDE

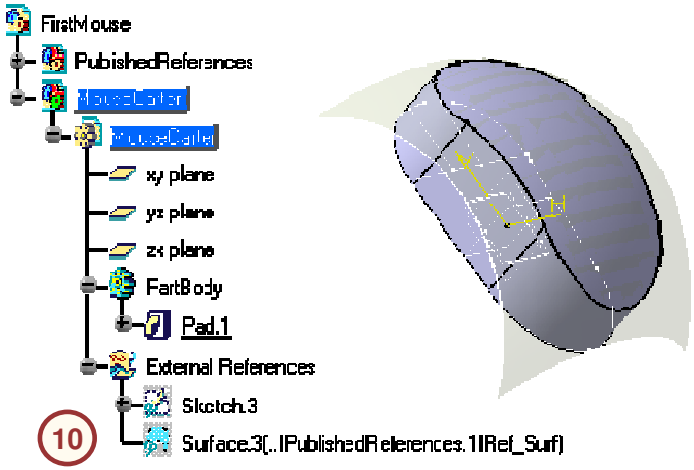
Student Notes:

Replacement Of a Non-Published Driving Component (3/3)

Use the following steps to replace a non-published component that is referenced by other components (continued):

9. Once the geometry has been reconnected, delete the invalid references.
10. The contextual part now references only geometry of the replacing component.





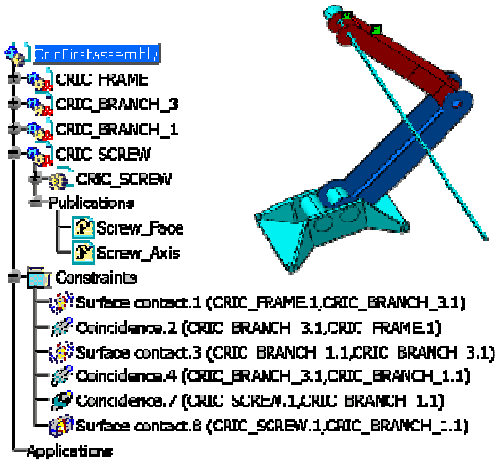
Published Geometry and Assembly Constraints (1/2)

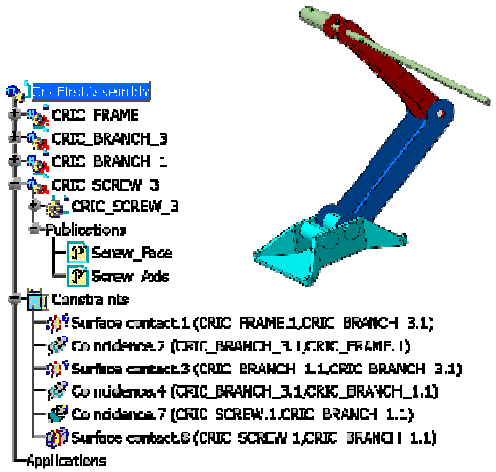
Student Notes:

When replacing a component with published elements, the constraints are automatically re-connected. This makes replacing components quick and simple.

Use the following steps to replace a component with published geometry:

1. Ensure that the replacing published elements are named the same as the published elements to be replaced. For example, two constraints in the assembly are connected to published elements of “Cric_Screw” component. Cric_Screw” has been replaced with “Cric_Screw_3” which has the same published geometry named identically.

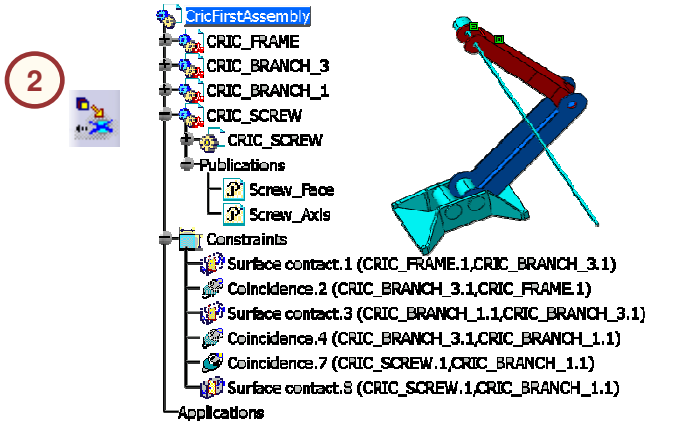


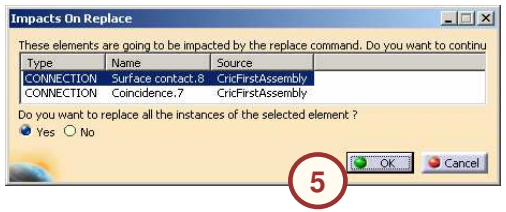


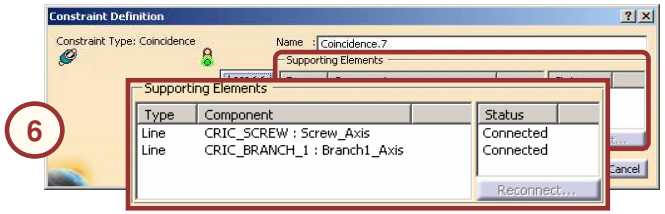
Published Geometry and Assembly Constraints (2/2)

Use the following steps to replace a component with published geometry (continued):

2. Select the Replace component icon.
3. Select the component to be replaced.
4. Select the replacing component.
5. Click **OK** on the Impacts on Replace dialog box.
6. The constraints are automatically re-connected to the published geometry in the new component. (i.e., the Screw_Axis publication in the original component is replaced with the Screw_Axis publication in the replacing component).







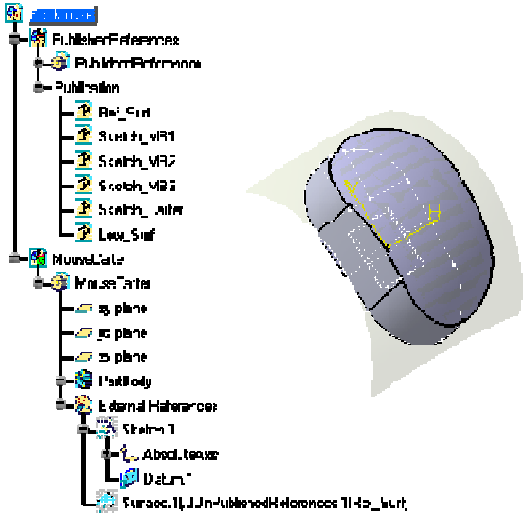
Student Notes:

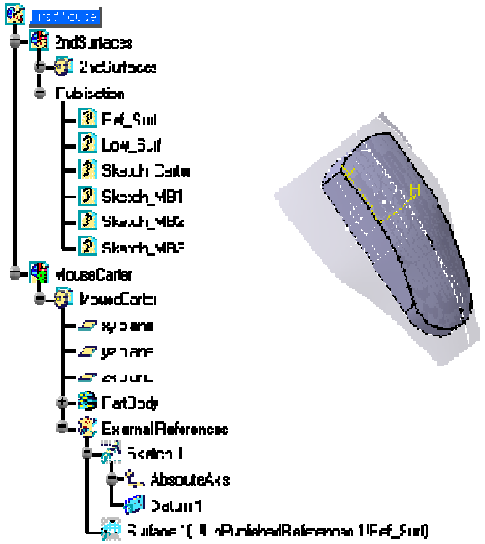
Student Notes:

Published Geometry and Contextual Design

When you replace a component with published elements, the links to contextual components are automatically reconnected.

With published elements there is no need to re-connect the removed external references, they are automatically replaced with the corresponding published element from the replacing component.





Student Notes:

Exercise: Publication

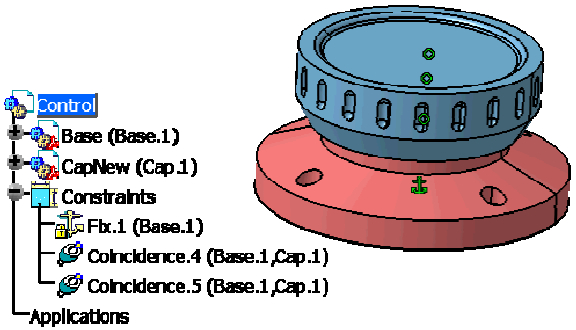
Recap Exercise

20 min

In this exercise, you will open two existing parts. You will publish the elements in the parts and use them to position these parts in an assembly. You will then replace one of the components using published elements to ensure references are not lost. Detailed instructions are provided for this exercise.

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

- Publish elements
- Use published elements to position components in an assembly
- Replace components

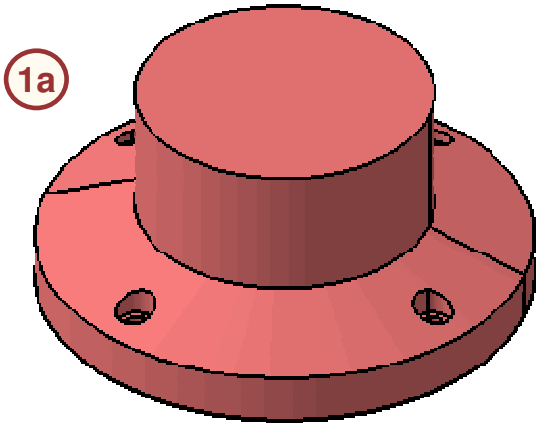


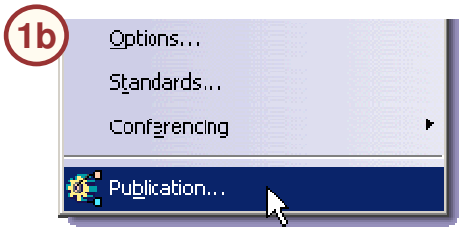
Student Notes:

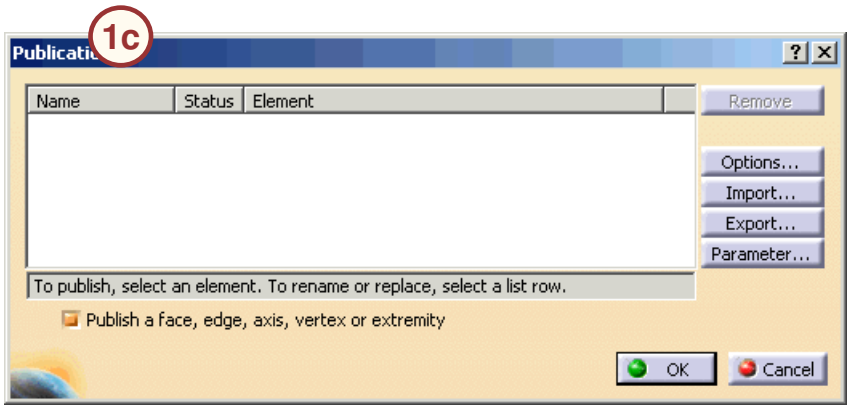
Do it Yourself (1/12)

1. Open a part file.

- Open a part file and activate publications.
  a. Open Base.CATPart.
  b. Click **Tools > Publication**.
  c. The Publication dialog box appears.







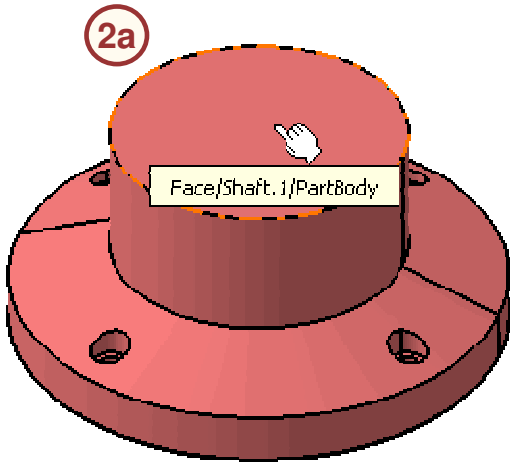
Student Notes:

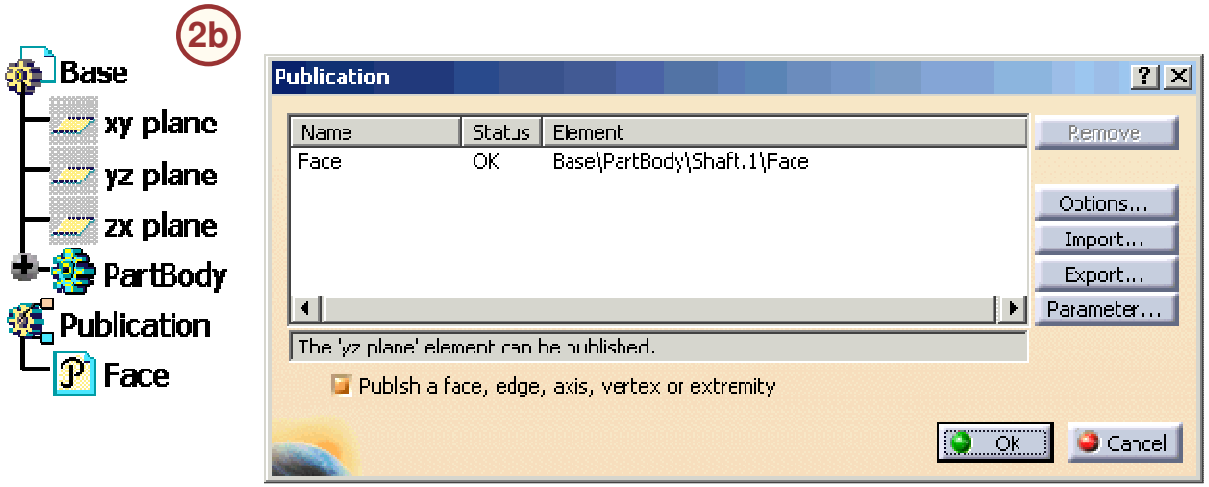
Do it Yourself (2/12)

2. Publish a face.

a. Select the top face of the base part.

b. The face is listed in the dialog box and the publication is shown in the specification tree.



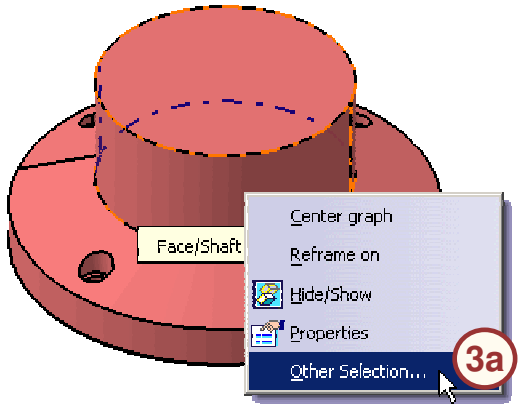


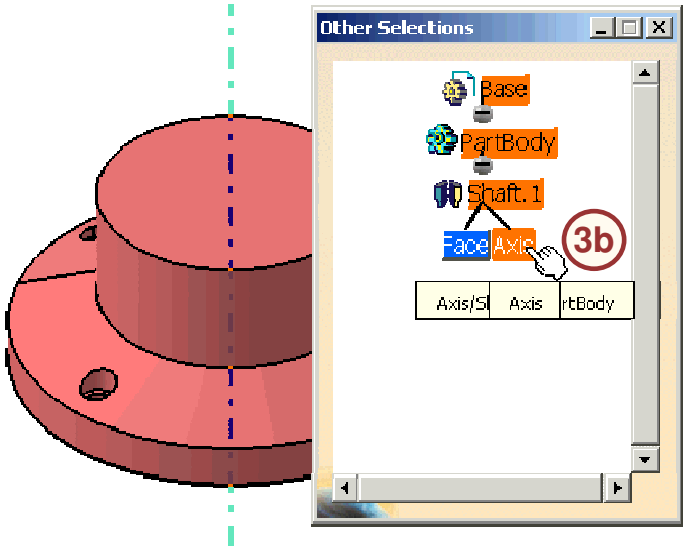
Student Notes:

Do it Yourself (3/12)

3. Publish an implicit axis.

- Use Other Selection to select an implicit axis.
  - a. Hold the cursor over the revolved face, press the right mouse button and click **Other Selection** from the contextual menu.
  - b. Select **Axis** from the Other Selections tree.



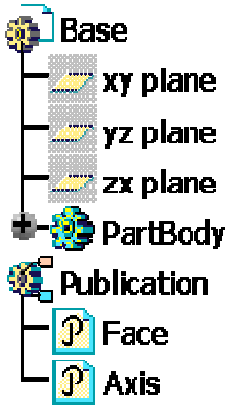


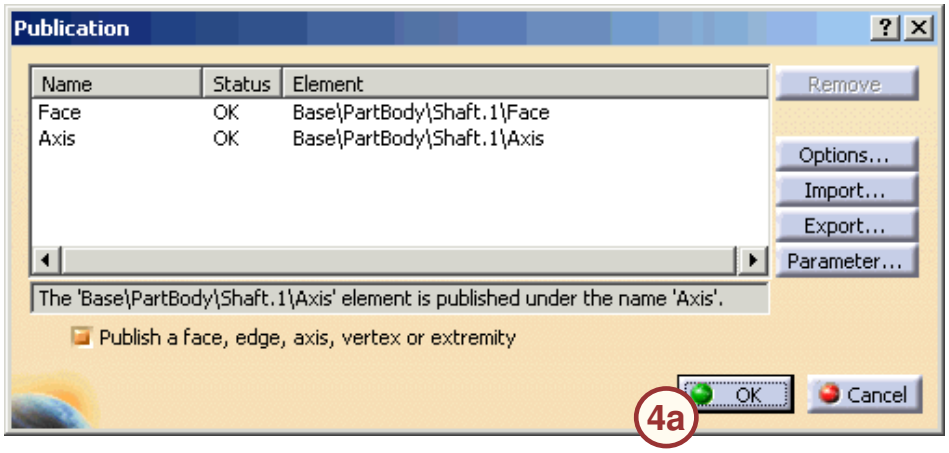
Student Notes:

Do it Yourself (4/12)

4. Publish the elements.

- The published elements appear in the Publication dialog box and in the specification tree.
  - a. Select **OK** to close the Publication dialog box.
  - b. Save the part.



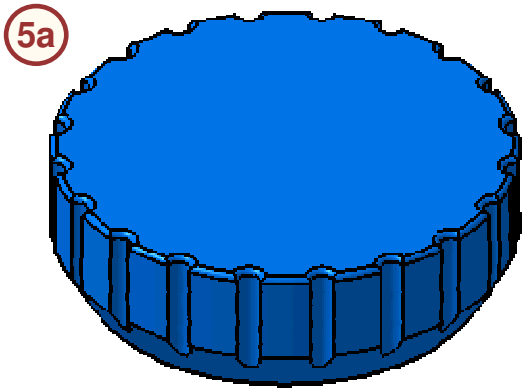


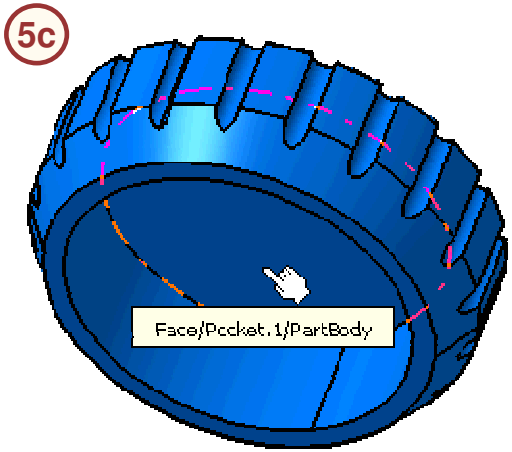
Student Notes:

Do it Yourself (5/12)

5. Open a part file.

- Open a part file and activate publications.
  a. Open Cap.CATPart.
  b. Click **Tools > Publication**.
  c. Publish the face shown.





Student Notes:

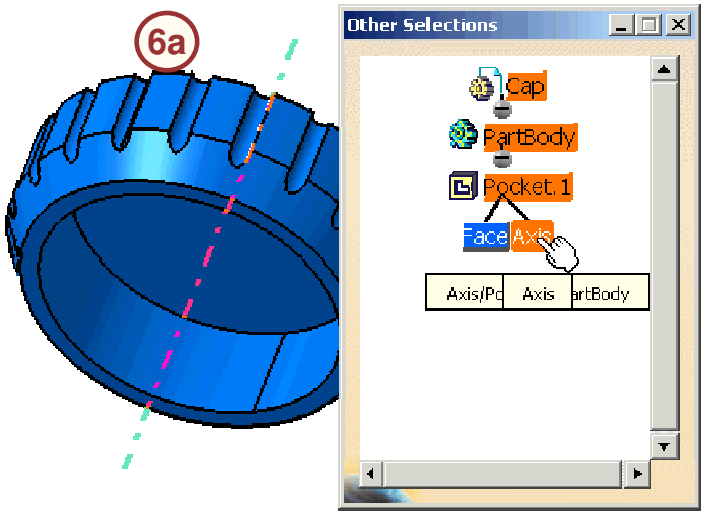
Do it Yourself (6/12)

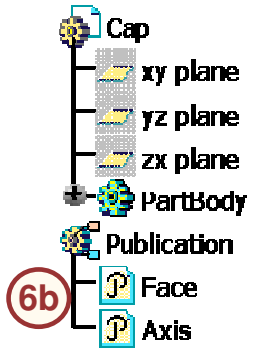
6. Publish an implicit axis.

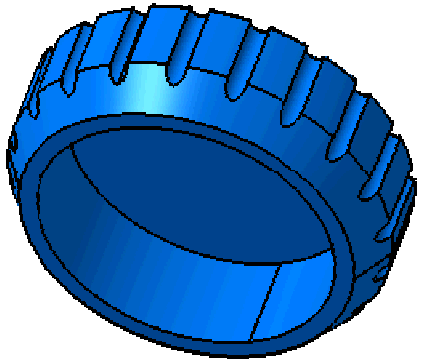
a. Select the axis using Other Selections.

b. The implicit axis is now published.

c. Save the part.





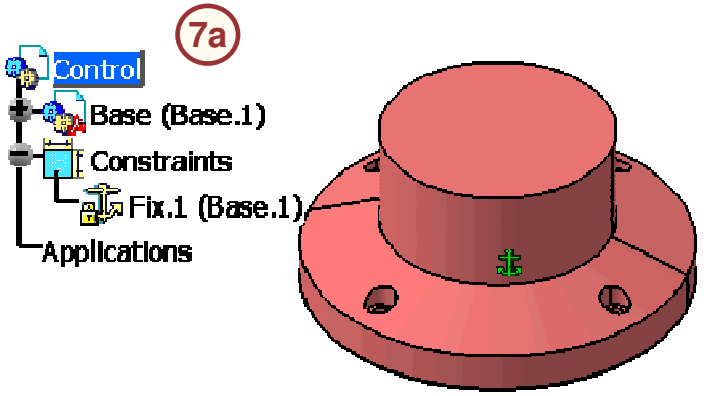


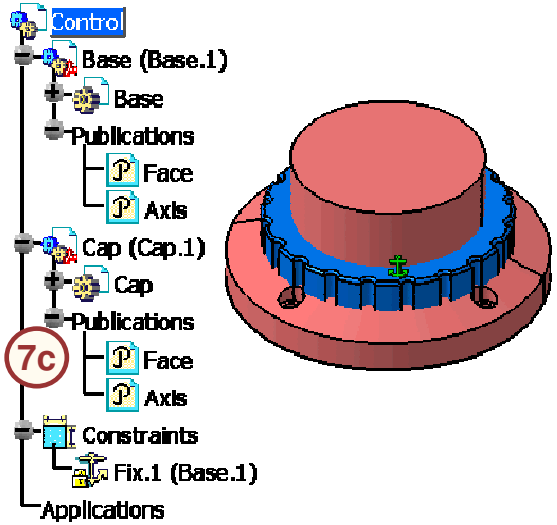
Student Notes:

Do it Yourself (7/12)

7. Open a product file.

- Assemble a part into an existing product file.
  - a. Open Control.CATProduct.
  - b. Insert existing Cap.CATPart.
  - c. Expand the specification tree to view published elements.



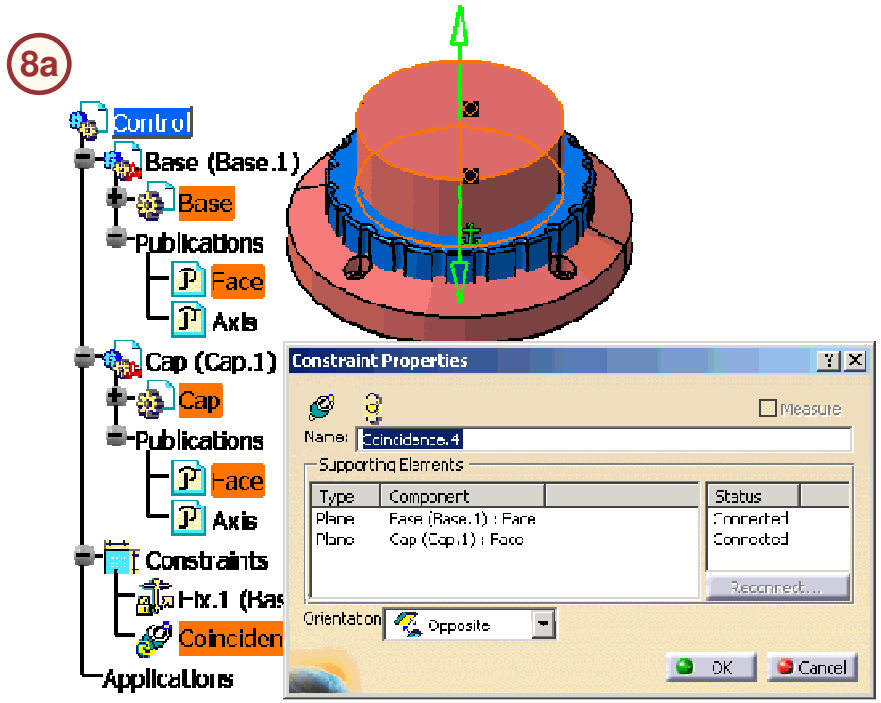


Student Notes:

Do it Yourself (8/12)

8. Apply constraints.

- Apply constraints between published elements.
  - a. Apply a **coincident** constraint between the two published face publications.
    (select them from the specification tree).

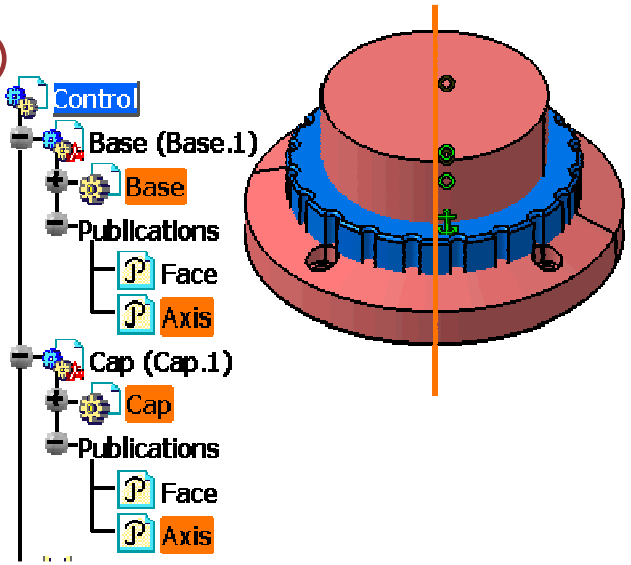


Do it Yourself (9/12)

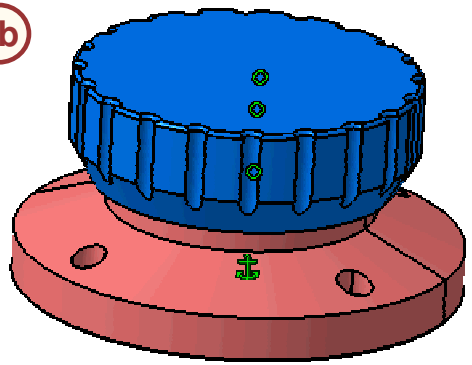
9. Apply constraints.

- Apply constraints between published elements.
  - a. Apply a **coincident** constraint between the two published Axis publications.
    (select them from the specification tree).
  - **b. Update** the assembly.
  - c. Save the assembly file

9a



9b



Student Notes:

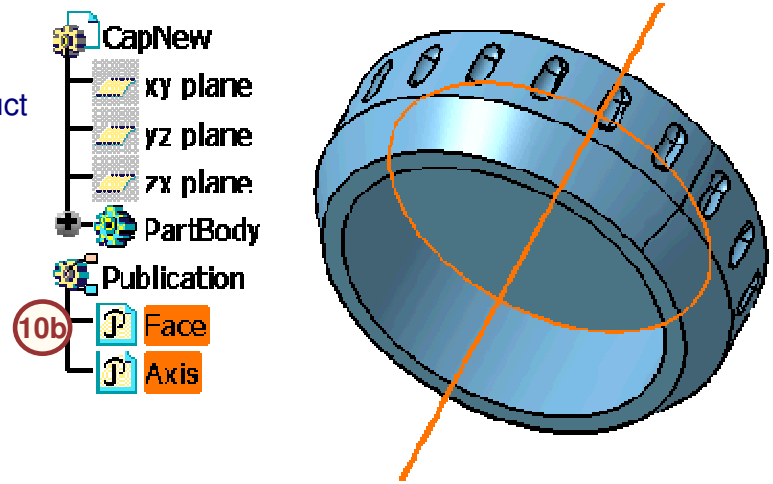
Do it Yourself (10/12)

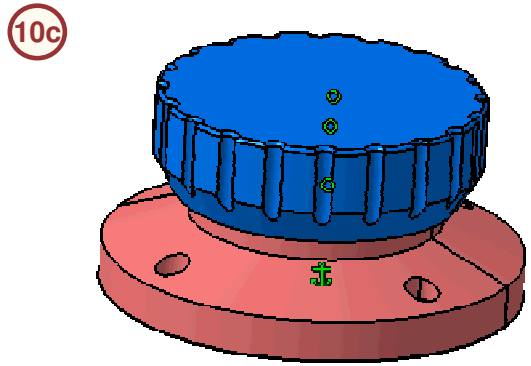
10. Open a part file.

a. Open CapNew.CATPart.

b. View the published elements.

c. Activate the Control.CATProduct window.





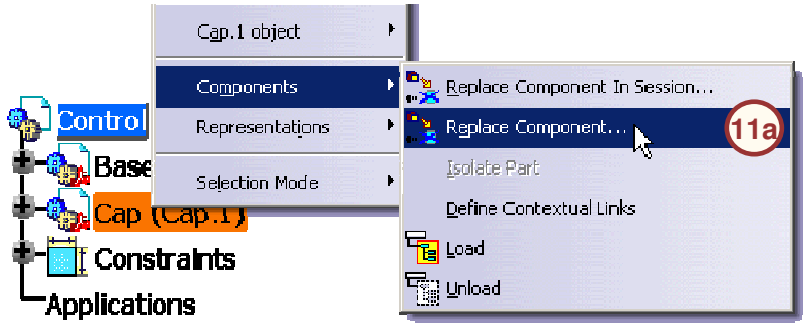
Student Notes:

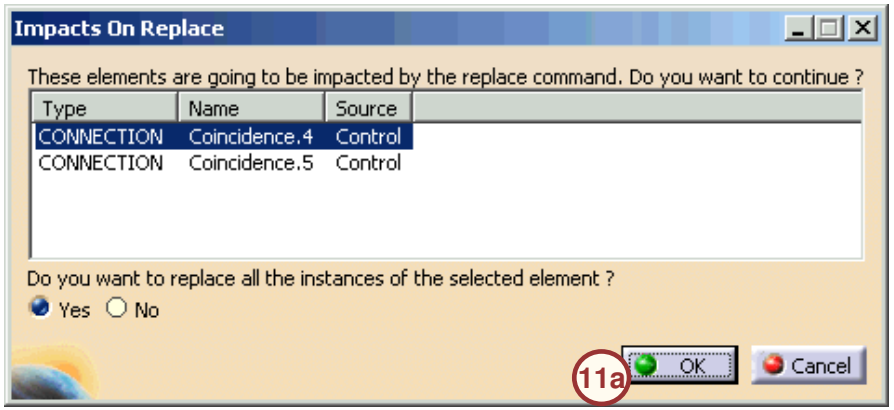
Student Notes:

Do it Yourself (11/12)

11. Replace a component.

- Replace the cap with a different one.
  - a. Use the right mouse button pop-up menu to replace cap component with the CapNew part.
  - b. Accept the Impacts On Replace.



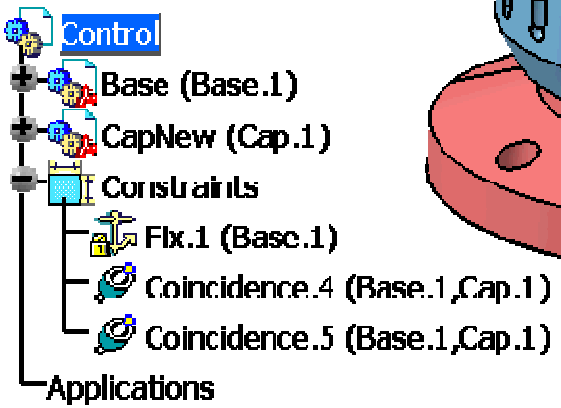


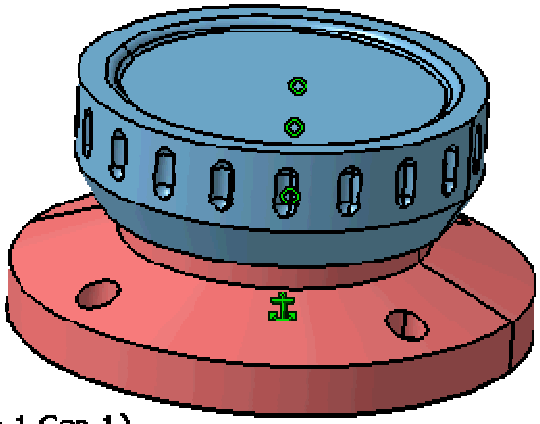
Do it Yourself (12/12)

12. View the replacement.

- The assembly uses a different cap part.
  - a. The constraints are still valid due to the published elements.
  - b. Save the product file and close all files.

12a



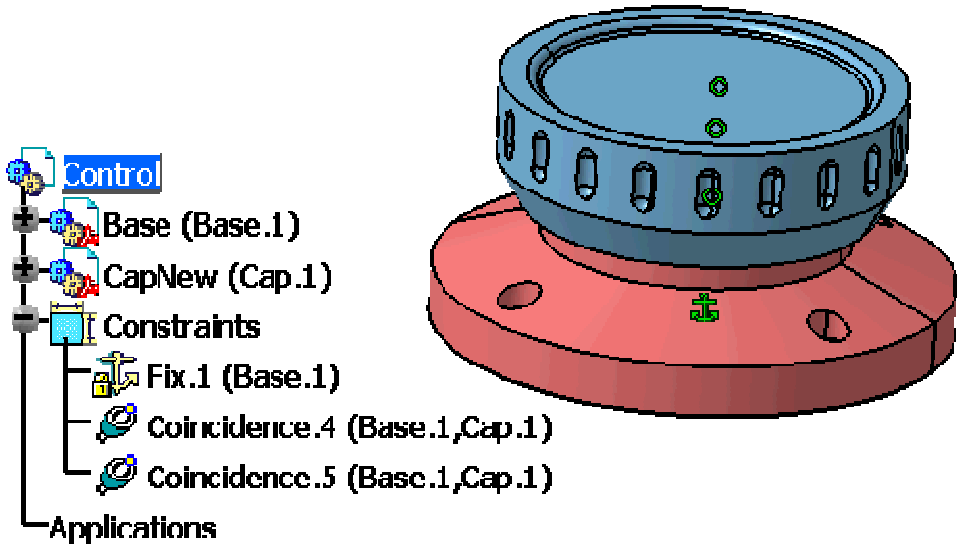


Student Notes:

Student Notes:

Exercise: Publication Recap

- ✓ Create published elements
- ✓ Use published elements to position components in an assembly
- ✓ Replace components



Student Notes:

Exercise: Parameter Publication

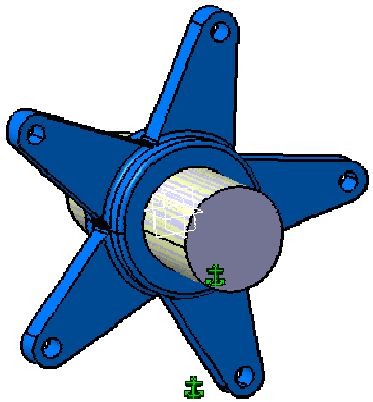
Recap Exercise

30 min

In this exercise, you will open an existing skeleton model that contains surface geometry and user parameters. You will publish these elements and use them to create two part files. Detailed instructions for the new topics are provided for this exercise.

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

- **Publish geometry**
- **Publish parameters**
- **Use published elements**

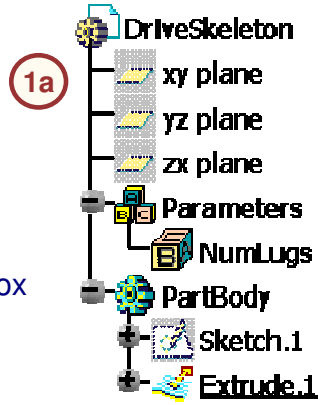


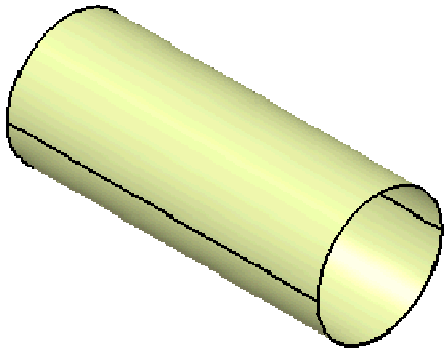
Student Notes:

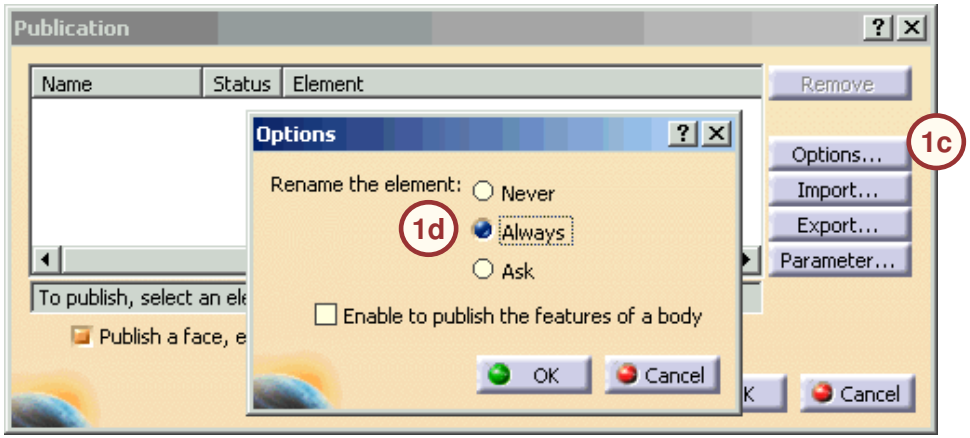
Do it Yourself (1/9)

1. Open part file.

- Open a part file and configure the publications naming option.
  a. Open DriveSkeleton.CATPart.
  b. Click **Tools > Publication**.
  c. Select **Options.**
  d. Select **Always**.
  e. Select **OK** to the Options dialog box.
  f. Leave the Publications dialog box open.





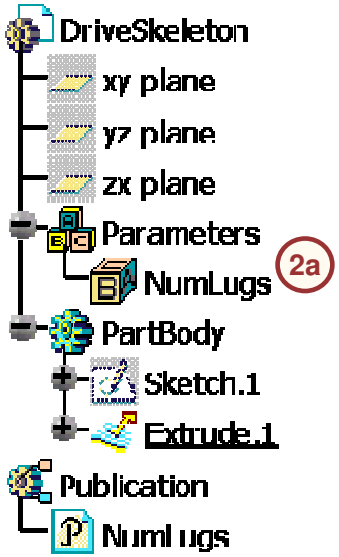


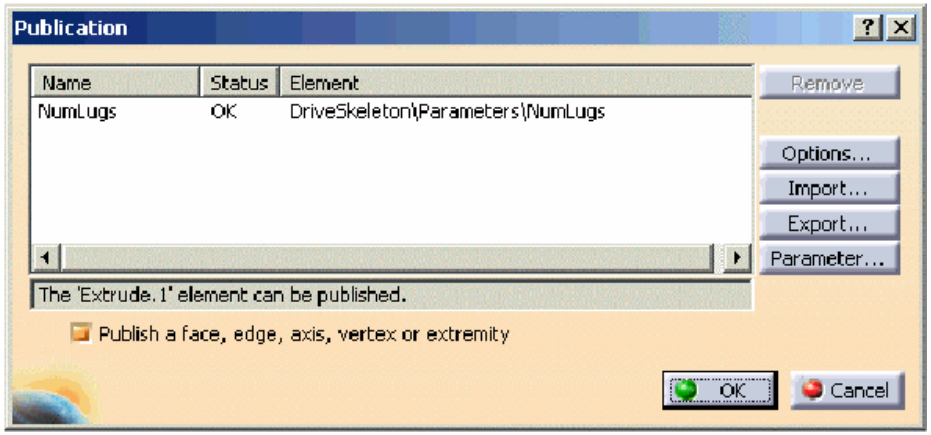
Student Notes:

Do it Yourself (2/9)

2. Publish a parameter.

a. Publish the NumLugs parameter.



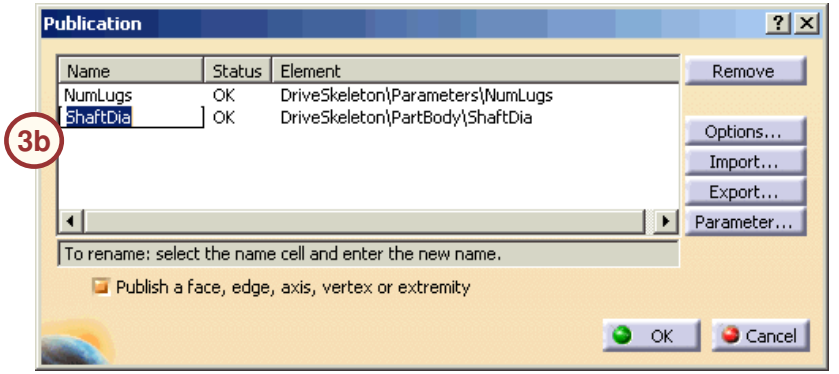


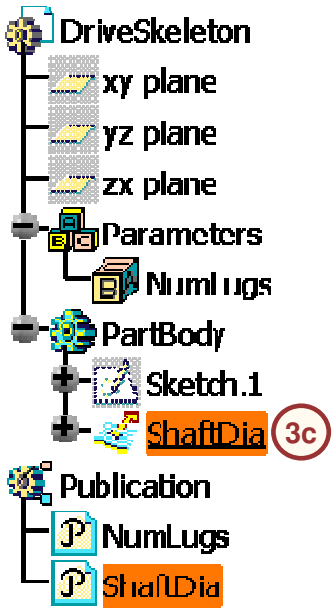
Student Notes:

Do it Yourself (3/9)

3. Publish a surface.

- Publish a surface feature and rename it.
  a. Publish Extrude.1 surface.
  b. Rename the surface to ShaftDia.
  c. Due to the naming option that was set, the Extrude.1 surface is renamed to the same name as the name of the publication.



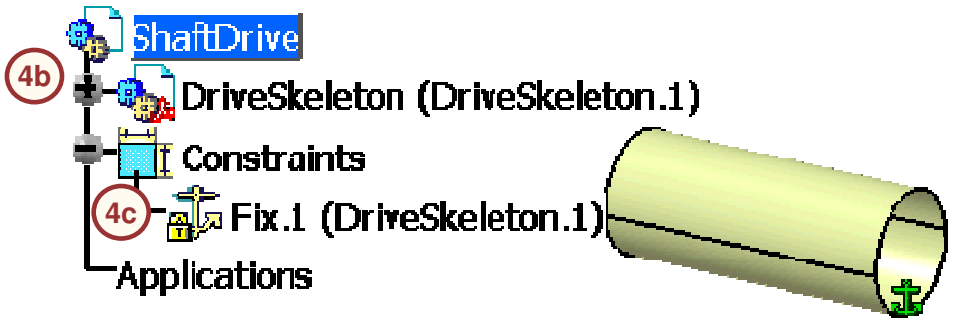


Student Notes:

Do it Yourself (4/9)

4. Create a product file.

- Create a product file and assemble a skeleton model.
  a. Create a new product file named ShaftDrive.
  b. Assemble DriveSkeleton.CATPart as the first component.
  c. Apply a **Fix** constraint to the skeleton.

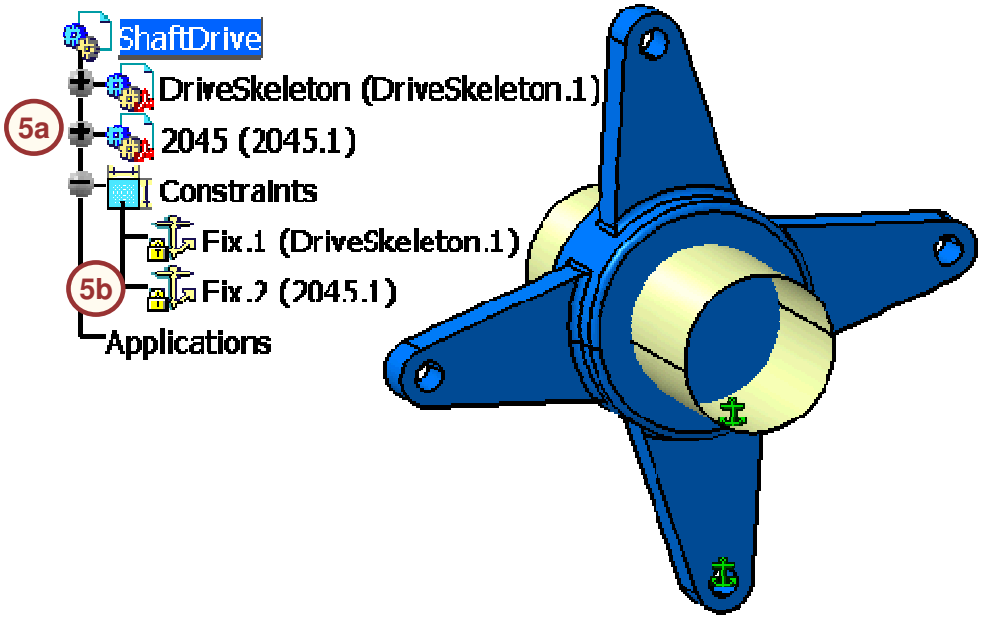


Student Notes:

Do it Yourself (5/9)

5. Assemble a part file.

- Assemble a part and constrain it.
  a. Assemble Carrier.CATPart.
  b. Apply a Fix constraint.

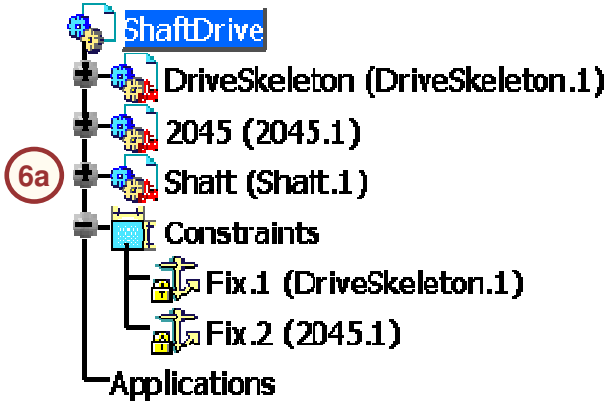


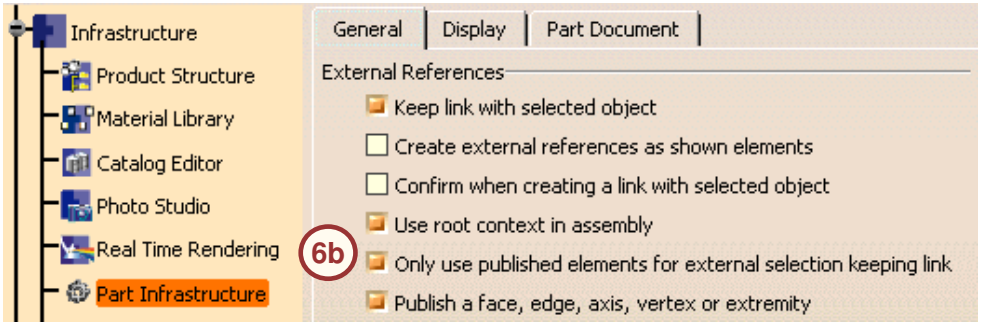
Student Notes:

Do it Yourself (6/9)

6. Create a new part.

- You will create a new part file and change settings.
  a. Create a new part file while in the Assembly Design workbench and name the part file Shaft.
  b. Change the Part Infrastructure options to **Only use published elements for external selection keeping link**.

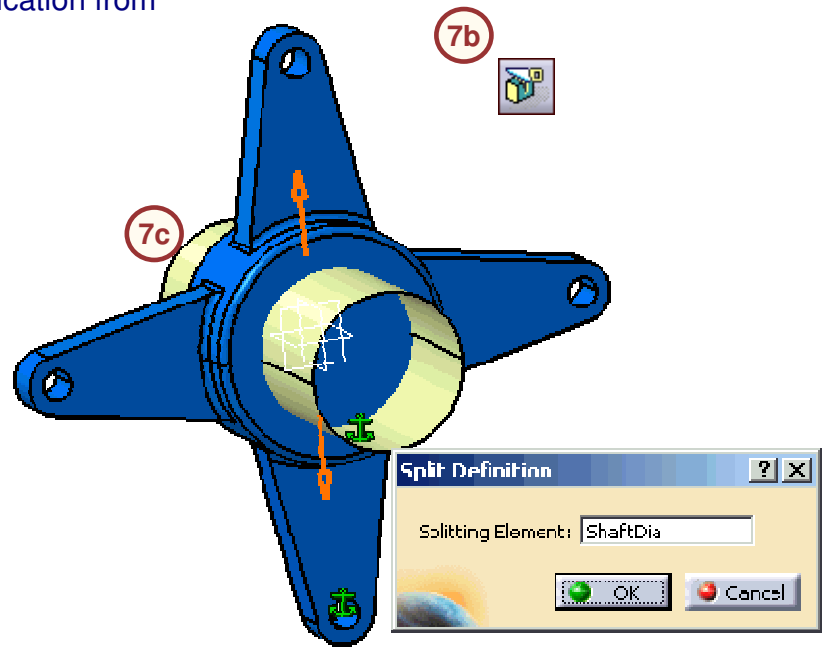




Do it Yourself (7/9)

7. Design in context.

- Use the ShaftDia surface to split the carrier part. Create the feature in context, using a publication.
  a. Activate part number 2045.
  b. Select the **Split** icon.
  c. Select the ShaftDia publication from the DriveSkeleton.



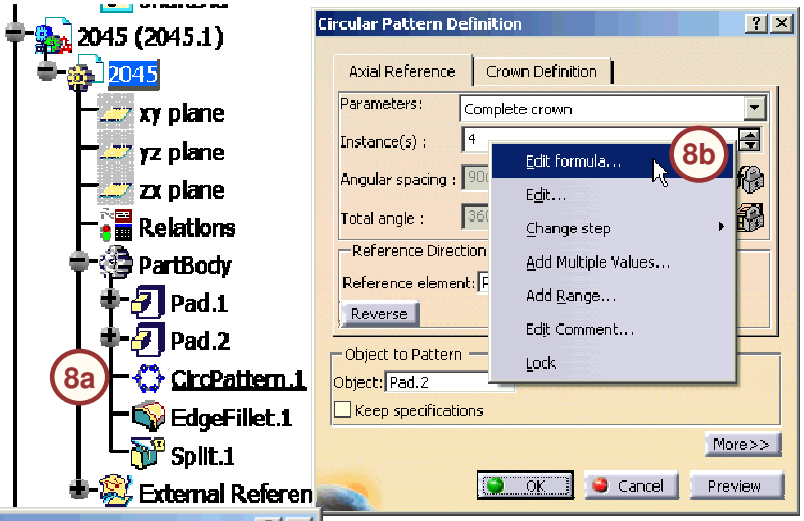
Student Notes:

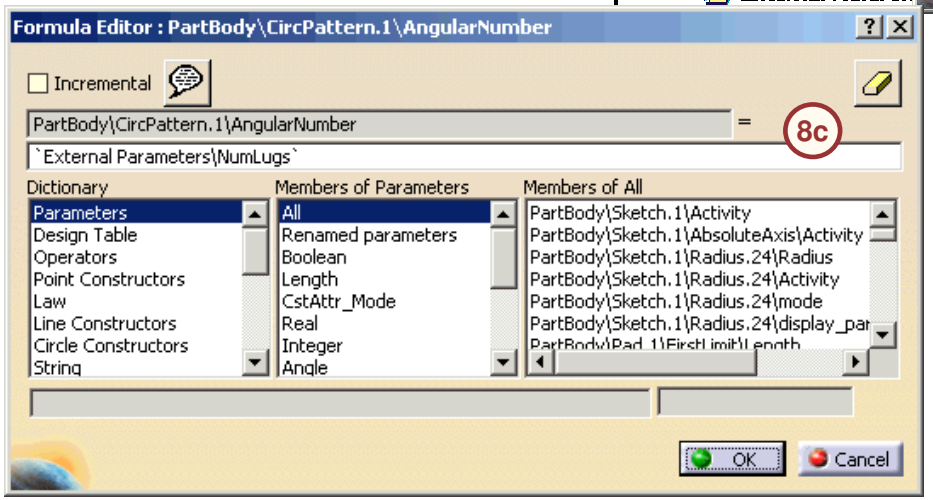
Student Notes:

Do it Yourself (8/9)

8. Use a published parameter.

- You will use a published parameter to control the number of pattern instances.
  a. Edit CircPattern.1.
  b. From the Instance(s) field contextual menu click Edit formula.
  c. Equate the instances value to the NumLugs publication.



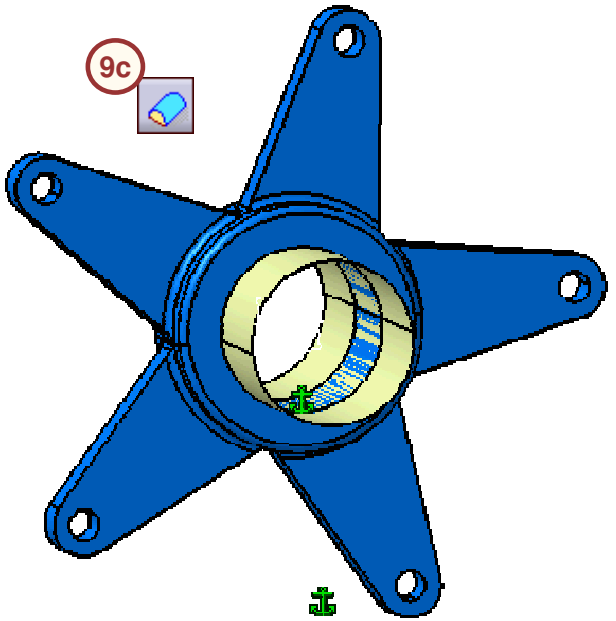


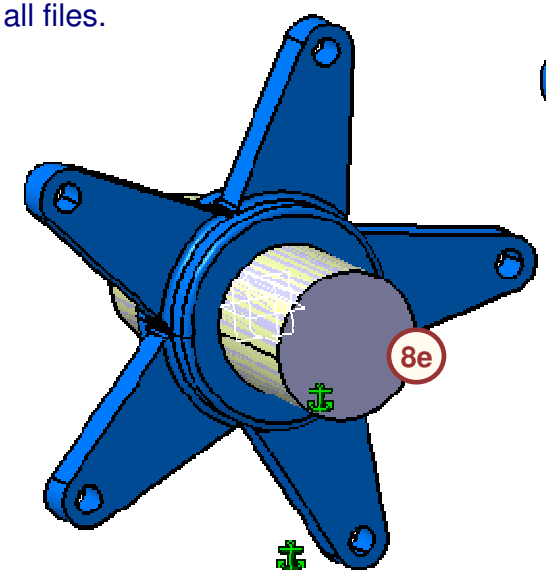
Student Notes:

Do it Yourself (9/9)

9. Use a publication to create a part.

- See the results of referencing published elements of a skeleton.
  a. The 2045 part updates to appear as shown.
  b. Activate Shaft part.
  c. Select the Close Surface icon to create a Close feature.
  d. Select ShaftDia publication from the skeleton.
  e. The Shaft part appears as shown below.
  f. Save and close all files.

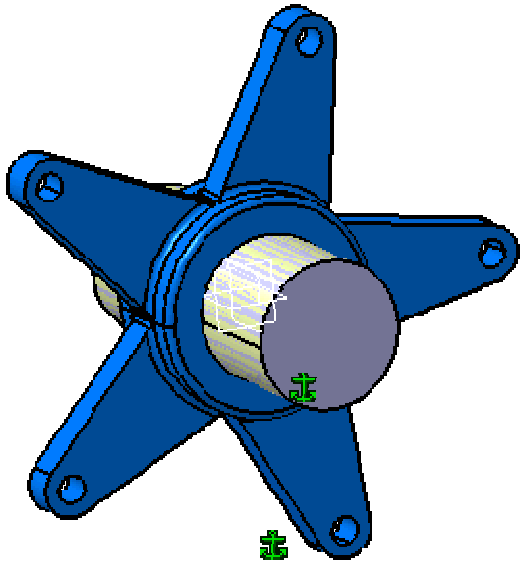




Student Notes:

Exercise: Parameter Publication Recap

- ✓ Publish geometry
- ✓ Publish parameters
- ✓ Use published elements



STUDENT GUIDE

Student Notes:

Exercise: Publication

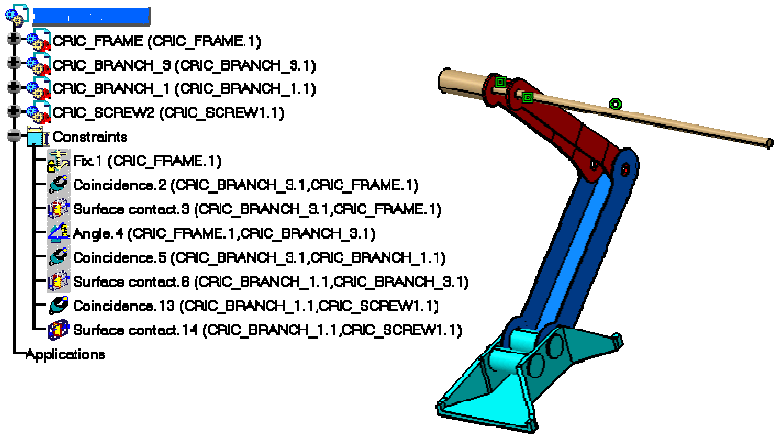
Recap Exercise

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.

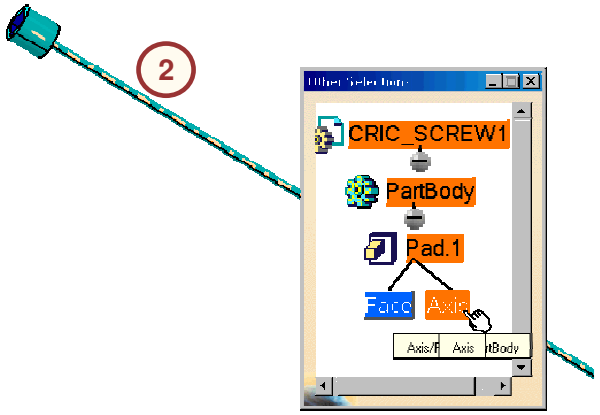
By the end of this exercise you will be able to:

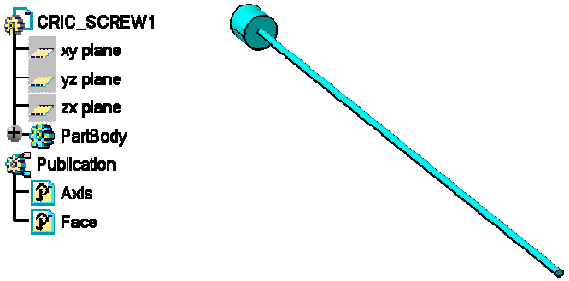
- Create published elements
- Replace components

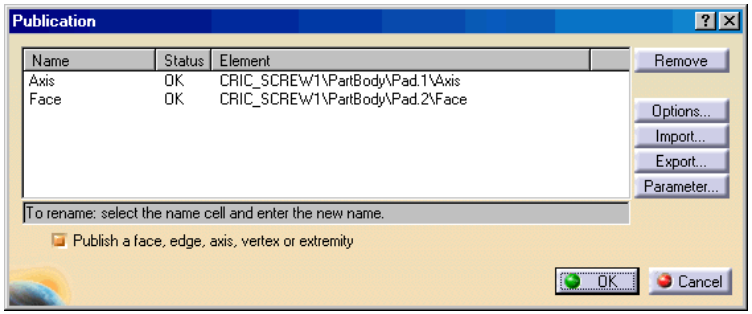


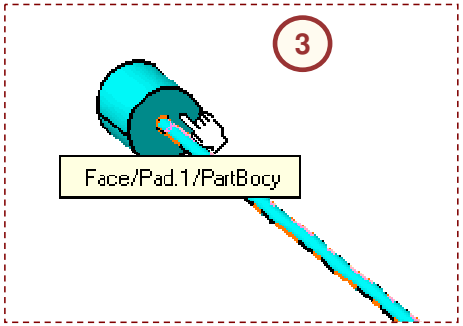
Do it Yourself (1/4)

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







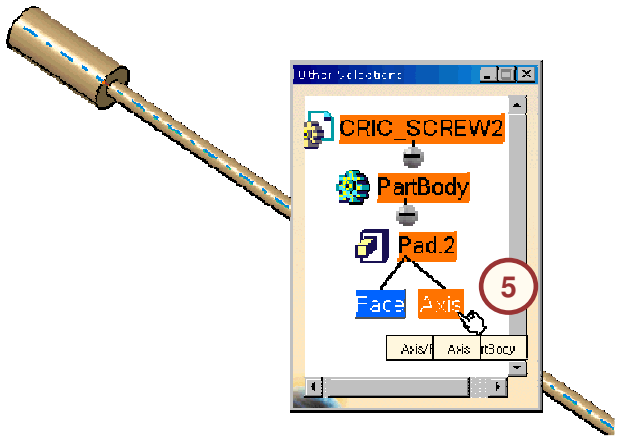


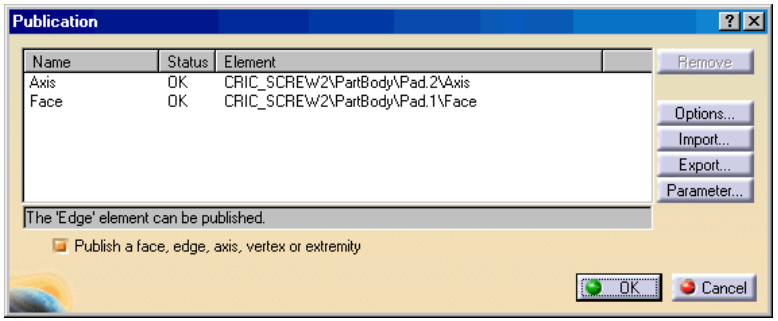
Student Notes:

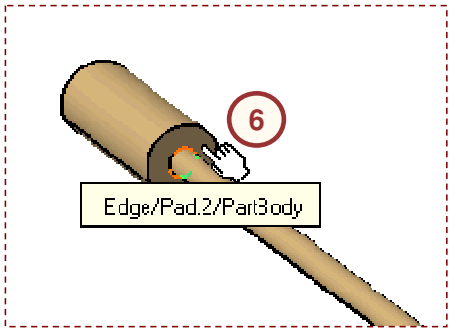
Student Notes:

Do it Yourself (2/4)

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



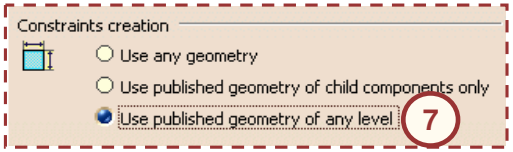




Student Notes:

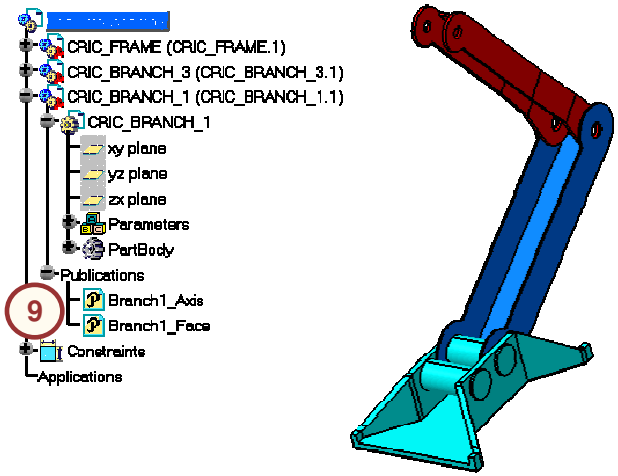
Do it Yourself (3/4)

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.

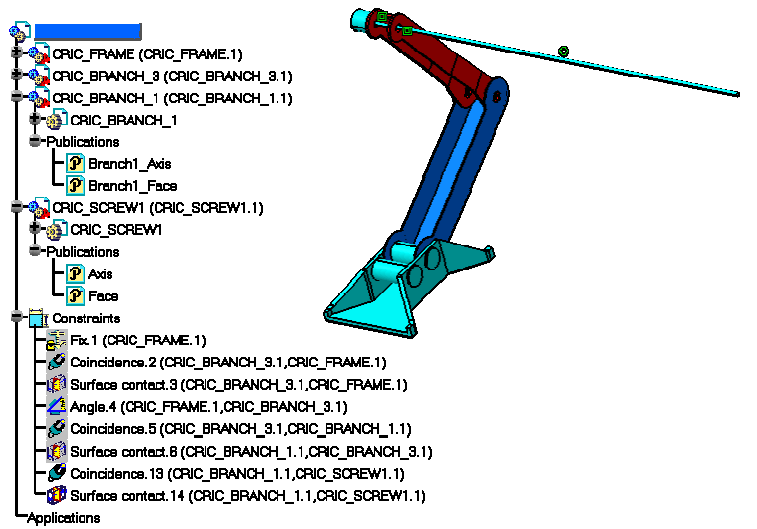


Student Notes:

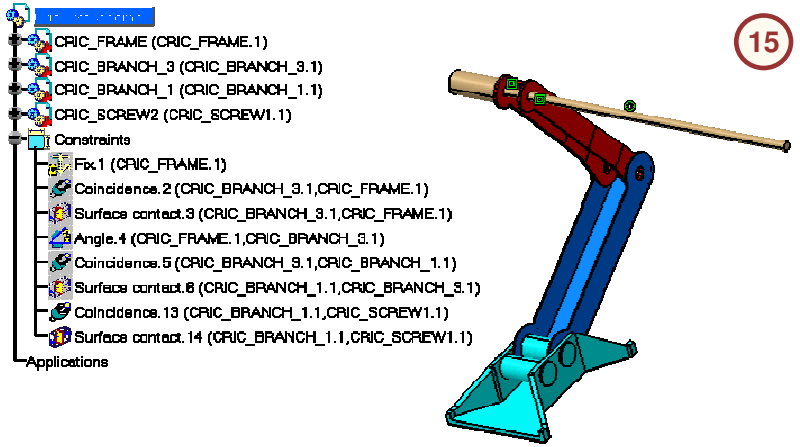
Do it Yourself (4/4)

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

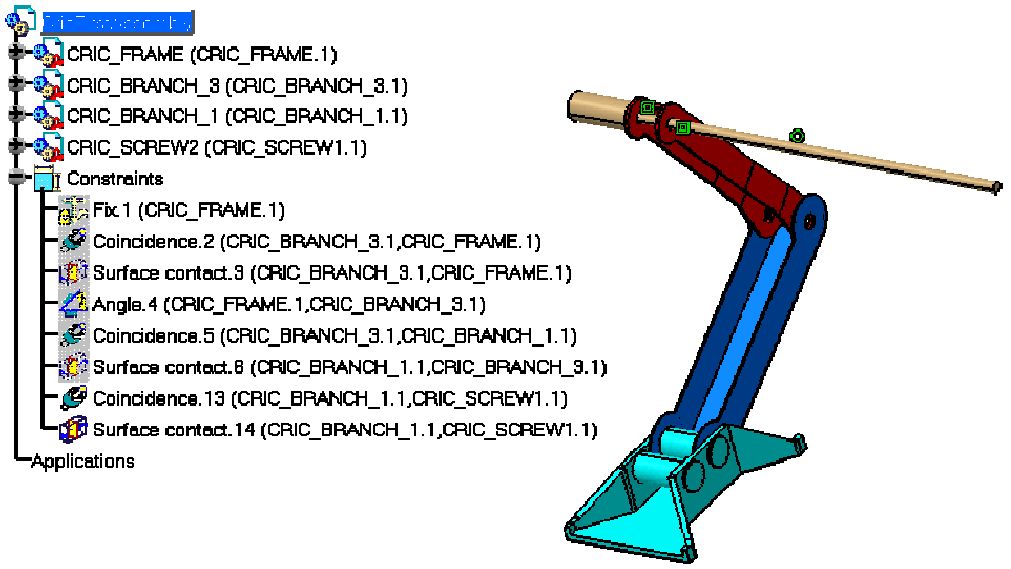


15



Exercise: Publication Recap

- ✓ Create publications
- ✓ Replace components



Student Notes:

STUDENT GUIDE

Student Notes:

Case Study: Complex Assembly Design

Recap Exercise

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

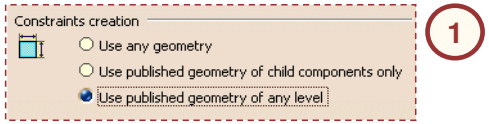
- ✓ Component locations must be controlled from a centralized location.
- ✓ Support geometry must be defined contextually.
- ✓ References must be strictly controlled.

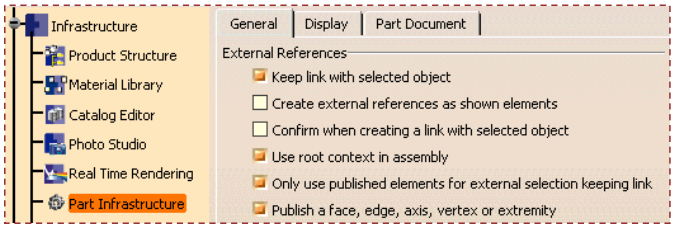
Using the techniques you have learnt in this and previous lessons, create the model with only high-level instruction.

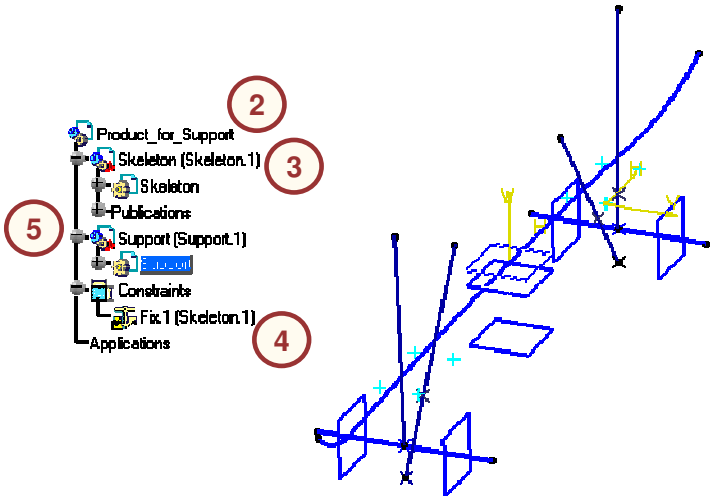
Do It Yourself: Skateboard (1/17)

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.**





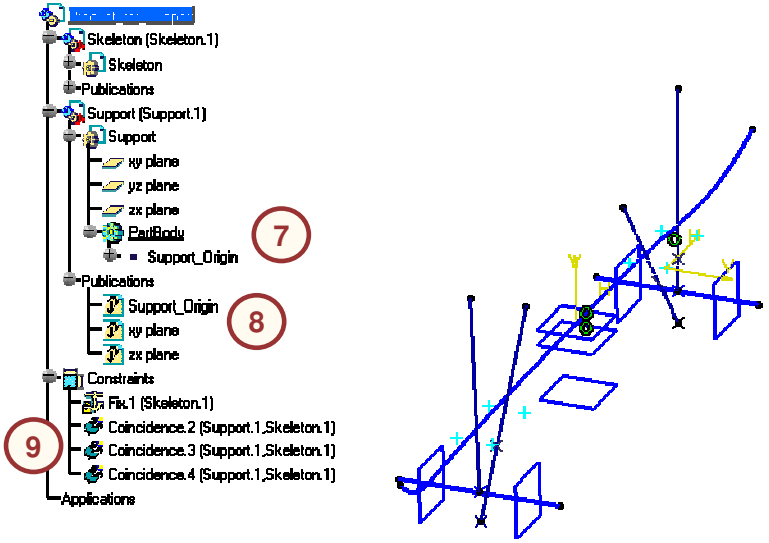


Student Notes:

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.



Student Notes:

Student Notes:

Do It Yourself: Skateboard (3/17)

You must complete the following tasks (continued):

10. Insert a body.

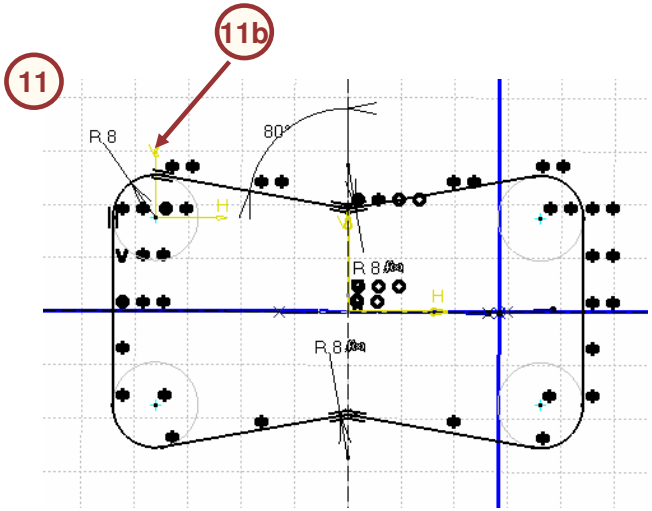
- Insert a new body into the Support component, name the body [Base].

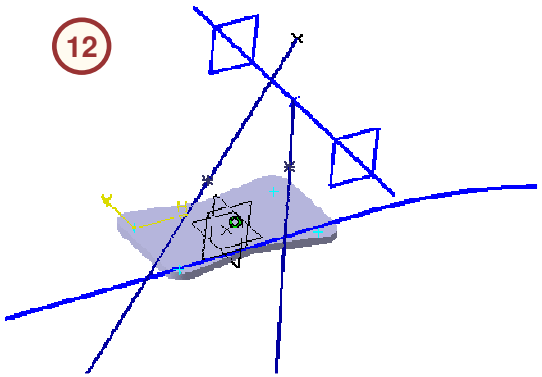
11. Create a sketch.

- 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.





Student Notes:

Do It Yourself: Skateboard (4/17)

You must complete the following tasks (continued):

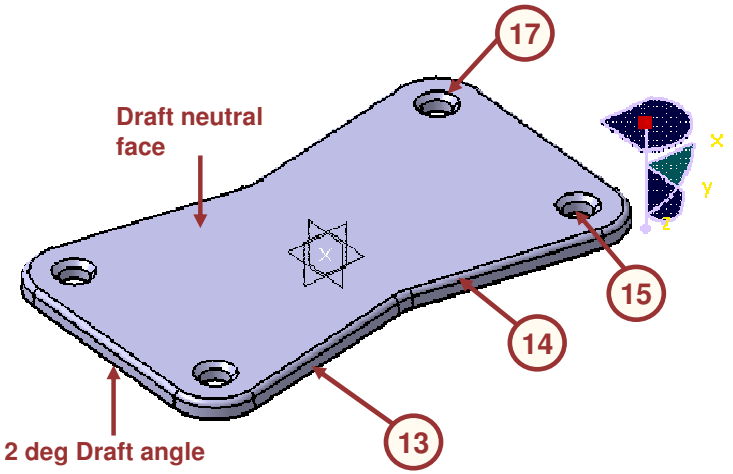
13. **Apply draft.**
    - Create a [2deg] draft on the sides of the pad. Use the top surface as the neutral plane.

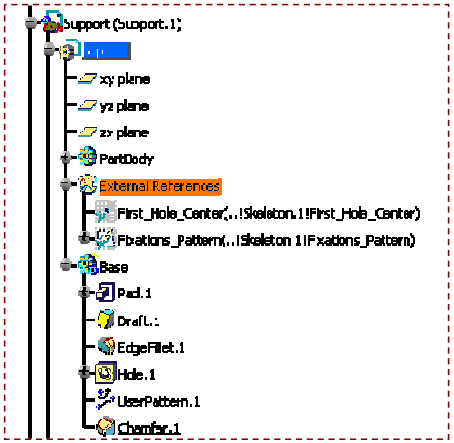
14. **Create a [1mm] edge fillet on the upper face of the pad.**

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)

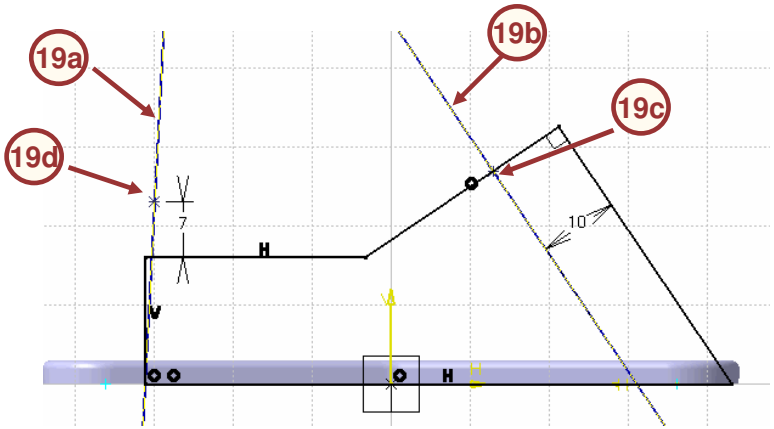
You must complete the following tasks (continued):

18. Create a new body

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

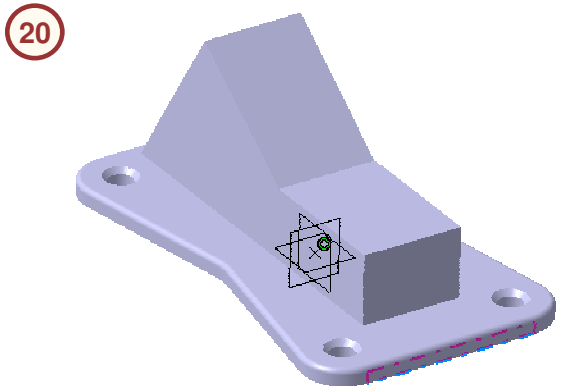
19. Create the sketch.

- 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].



Student Notes:

Student Notes:

Do It Yourself: Skateboard (6/17)

You must complete the following tasks (continued):

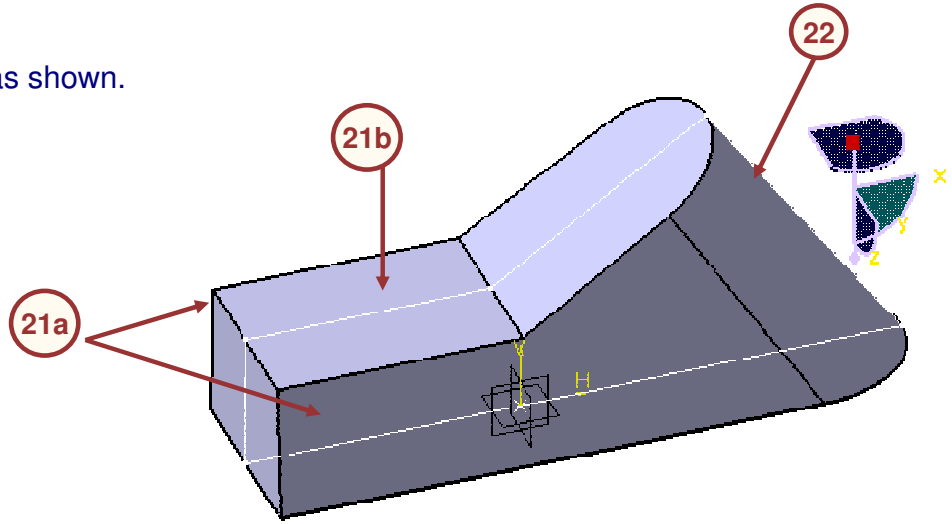
21. Apply draft.

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.



Student Notes:

Do It Yourself: Skateboard (7/17)

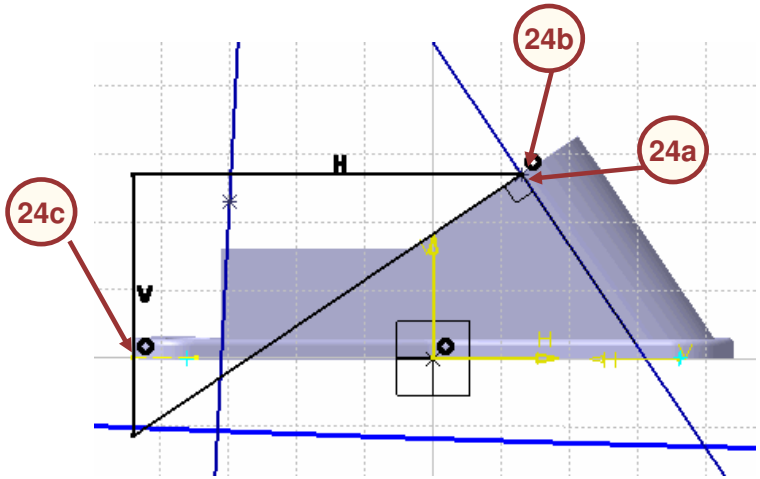
You must complete the following tasks (continued):

23. Create a new body.

- Create a new body in the Support component.

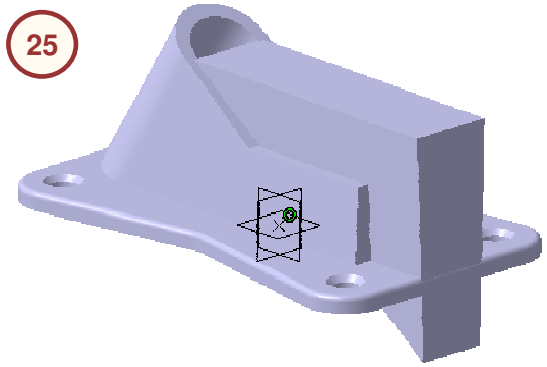
24. Create the sketch.

- 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.

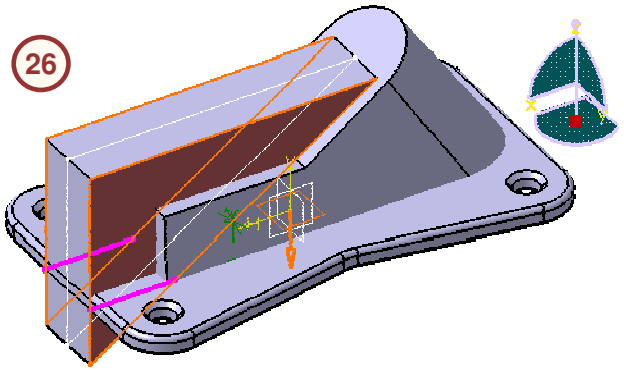
- Add draft of [3deg] to both sides of the pad. Use the XY plane as the neutral plane.

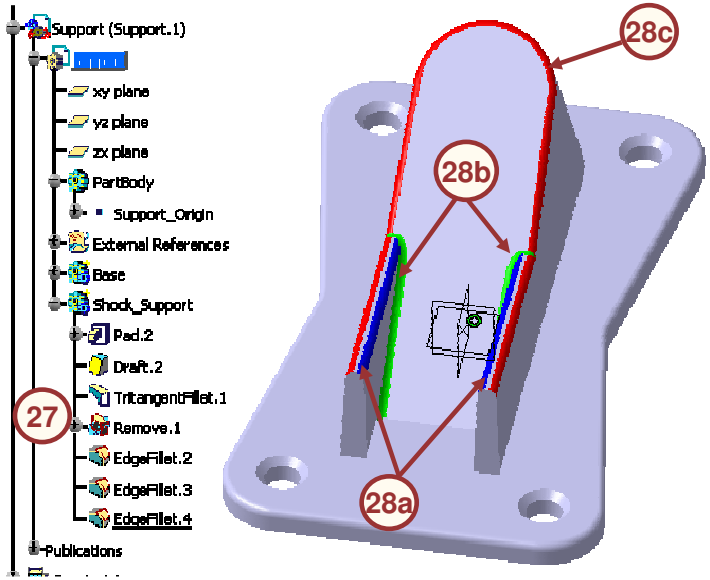
27. Remove the body.

- Remove the new body from the Shock_Support body.

28. Create fillets.

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





Student Notes:

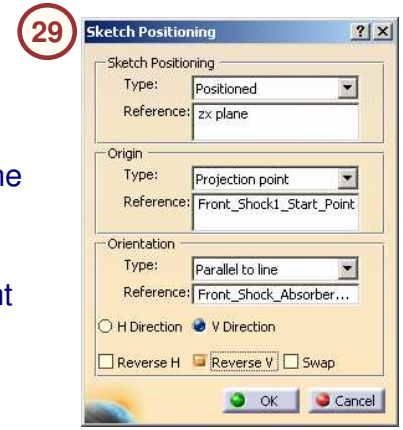
Student Notes:

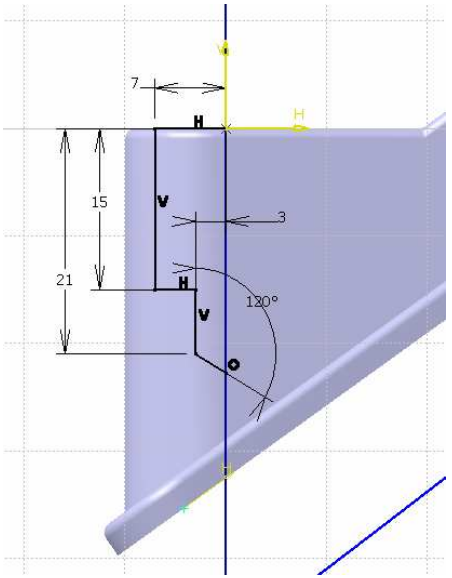
Do It Yourself: Skateboard (9/17)

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.



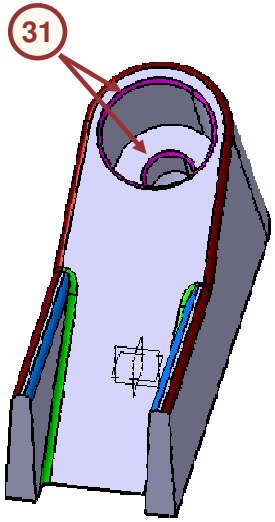


30. Create a groove feature.

- Create a groove feature using the sketch.

31. Add a chamfer.

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



Student Notes:

Do It Yourself: Skateboard (10/17)

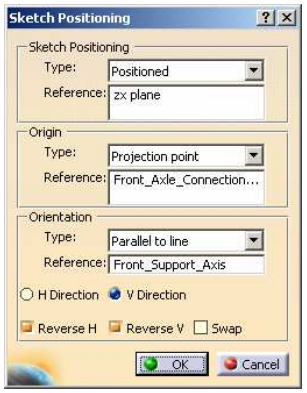
You must complete the following tasks (continued):

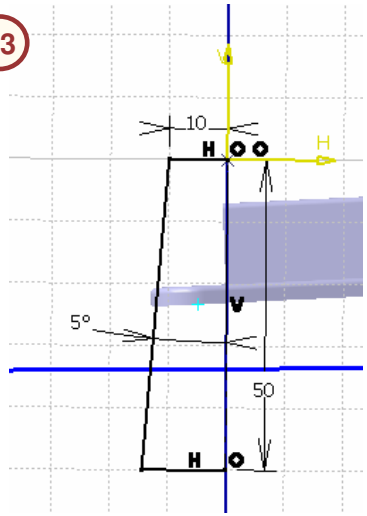
32. Create a new body.

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

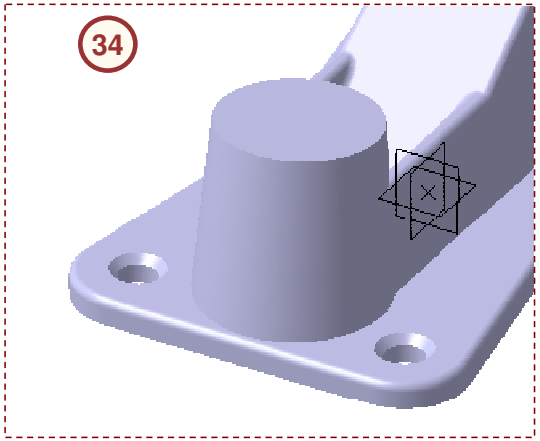
33. Create a shaft.

- 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.



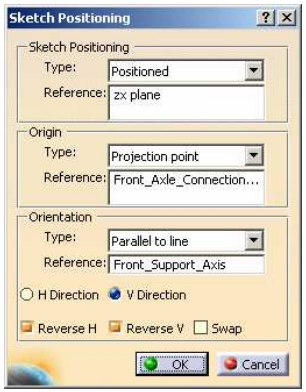
Student Notes:

Do It Yourself: Skateboard (11/17)

You must complete the following tasks (continued):

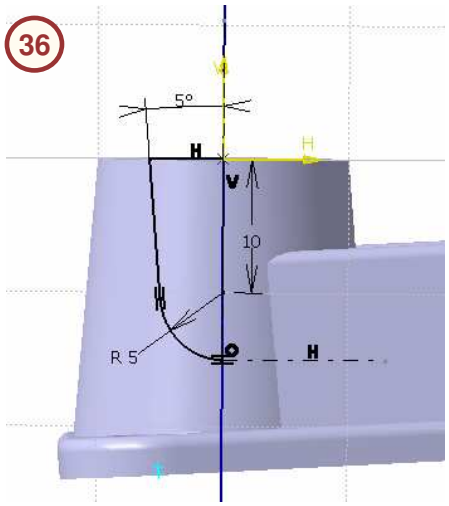
35. Create a hole.

- 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.



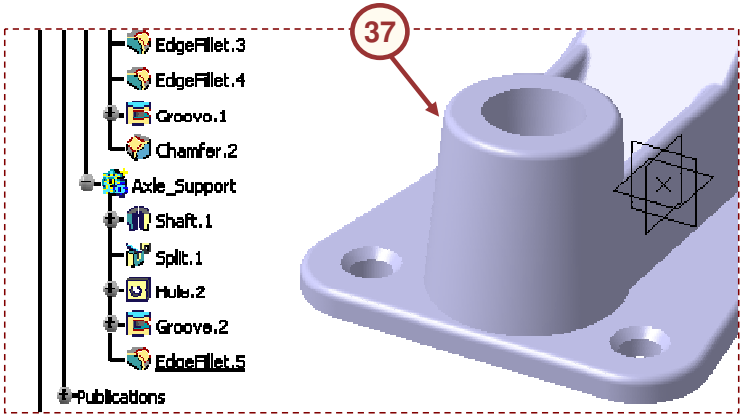
36. Create a groove.

- 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.



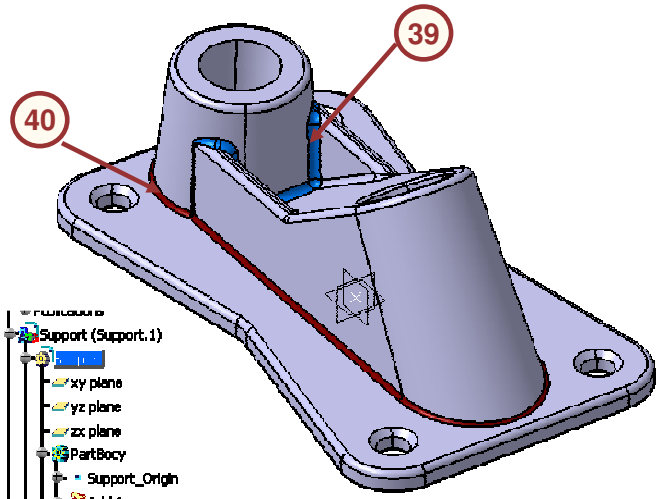
Do It Yourself: Skateboard (12/17)

You must complete the following tasks (continued):

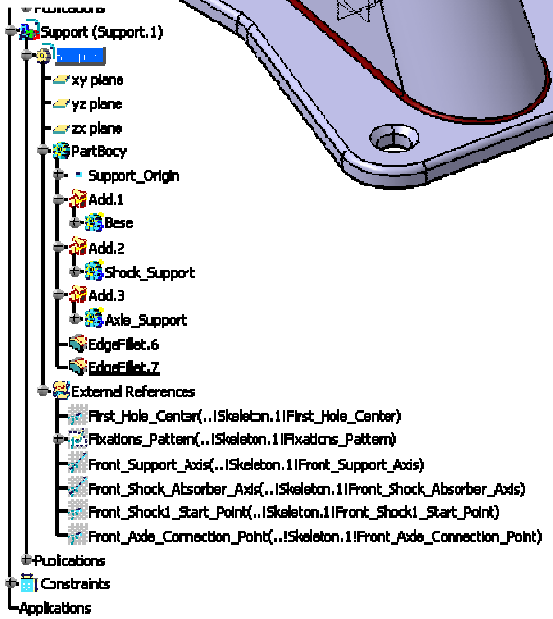
38. **Add the three previously designed bodies in the PartBody.**

39. **Create an edge fillet.**
    - Create a [1mm] edge fillet.

40. **Create an edge fillet.**
    - Create a [0.7mm] edge fillet.



38



Student Notes:

Student Notes:

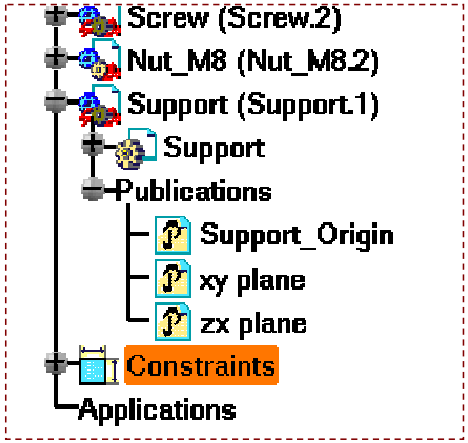
Do It Yourself: Skateboard (13/17)

You must complete the following tasks (continued):

41. Open Skateboard_with_Skeleton.CATProduct.

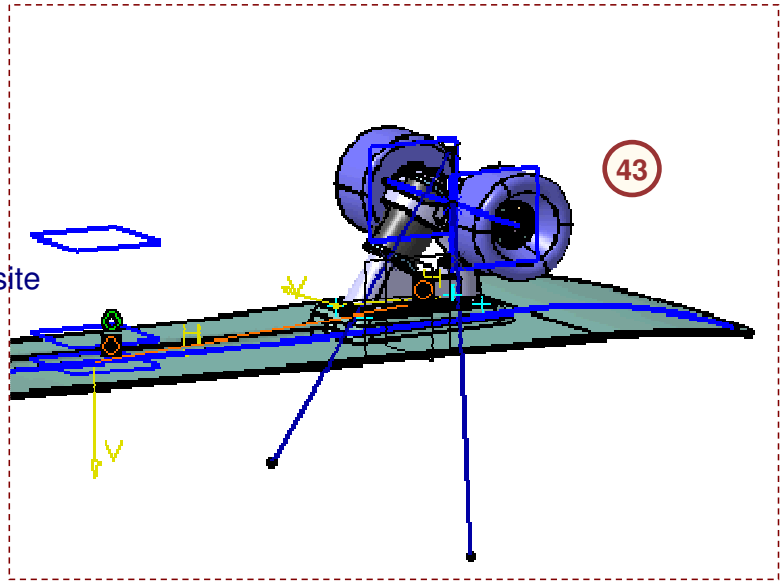
42. Insert Support.CATPart.

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



43. Position the support component.

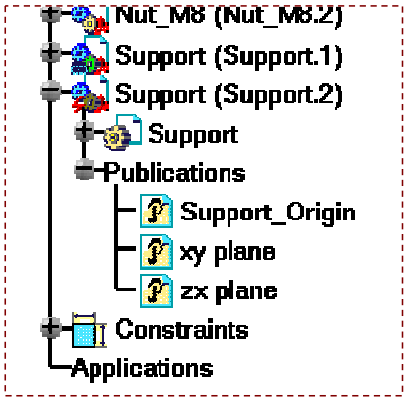
- Use published elements from the support and the skeleton to constrain the support.
  a. Support_Origin with Front_Support_Middle_Point.
  b. XY plane with Support_Plane, opposite direction.
  c. ZX plane with ZX plane, same orientation.



Student Notes:

Do It Yourself: Skateboard (14/17)

You must complete the following tasks (continued):

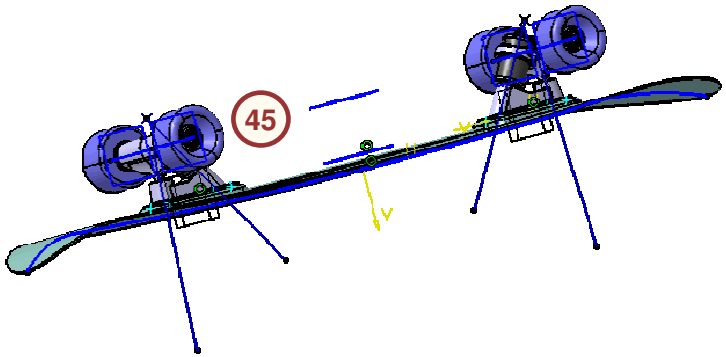


44. Validate links.

- 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.



Student Notes:

Do It Yourself: Skateboard (15/17)

You must complete the following tasks (continued):

46. Insert screw component.

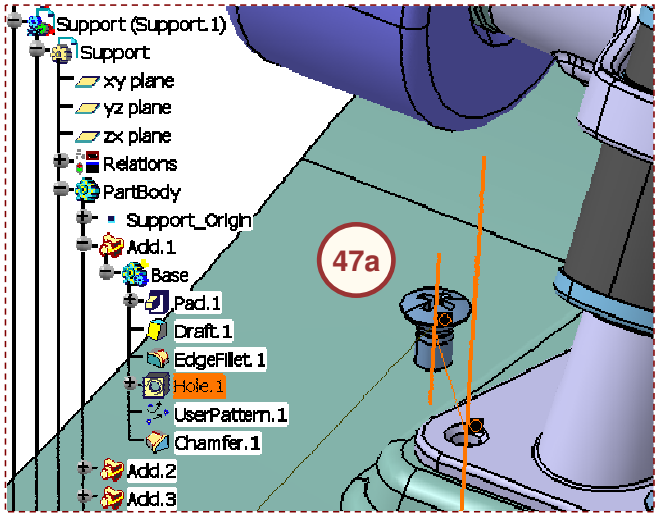
- Insert ISO_SCREW_M6x10-Z.CATPart.

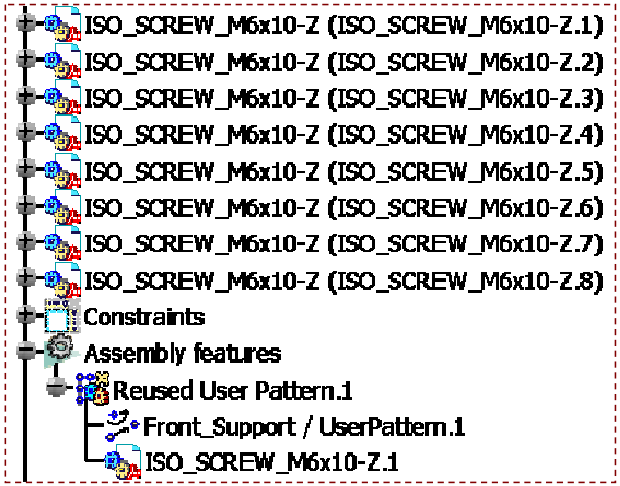
47. Position the screw.

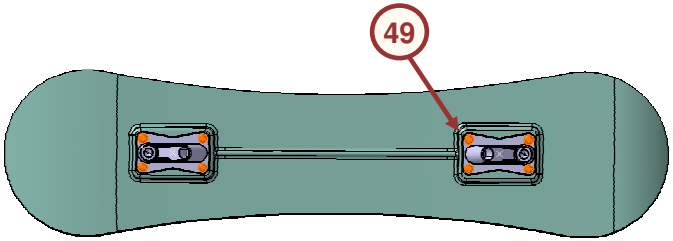
- Position the screw by creating the following constraints between the published elements:
  a. Coincidence constraint between the Hole of the Support.1 and the axis of the Screw.
  b. Offset of [-3mm] between Support_Plane in the skeleton and Mating_Plane in the screw, opposite orientation.

48. Instantiate the screw.

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







Student Notes:

Do It Yourself: Skateboard (16/17)

You must complete the following tasks (continued):

49. Edit the skeleton.

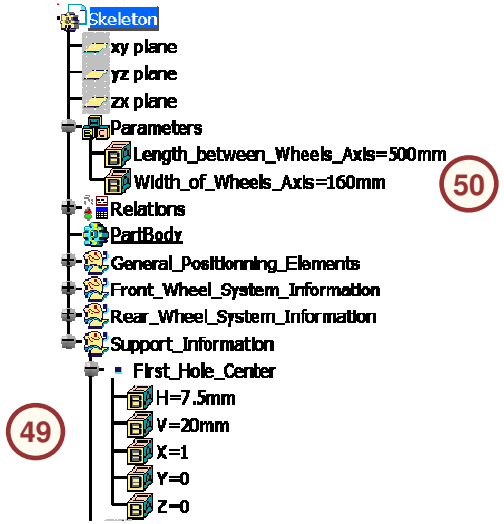
- Change the coordinates of First_Hole_Center point to:
  a. H = 7.5mm.
  b. V = 20mm.
- Notice that all the points of the fixation pattern are recalculated accordingly.

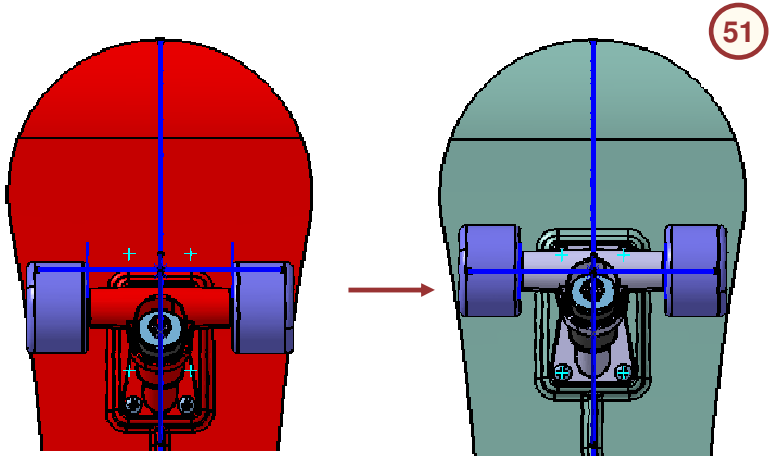
50. Edit the skeleton.

- 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.

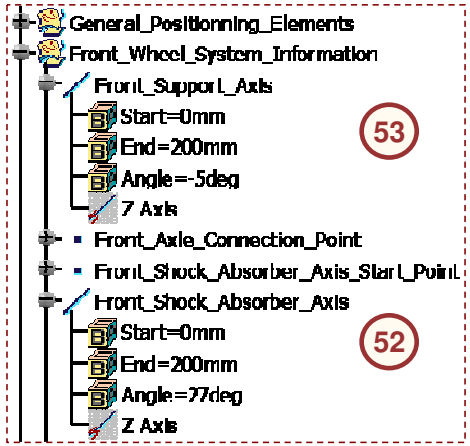
- Change the angle of Front_Shook_Absorber_Axis from [34deg] to [27deg].

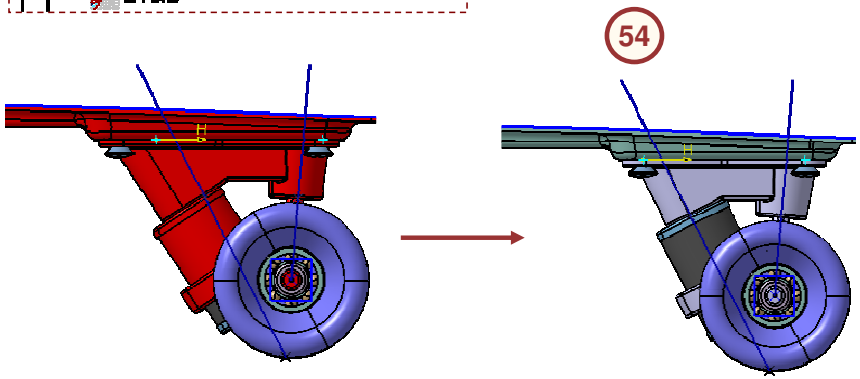
53. Edit the skeleton.

- 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.



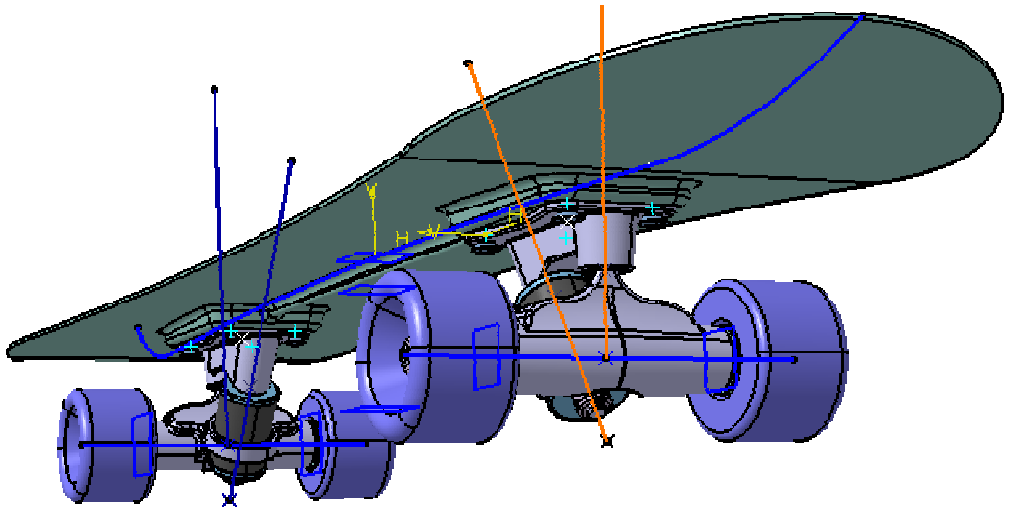


Student Notes:

Student Notes:

Case Study: Complex Assembly Design Recap

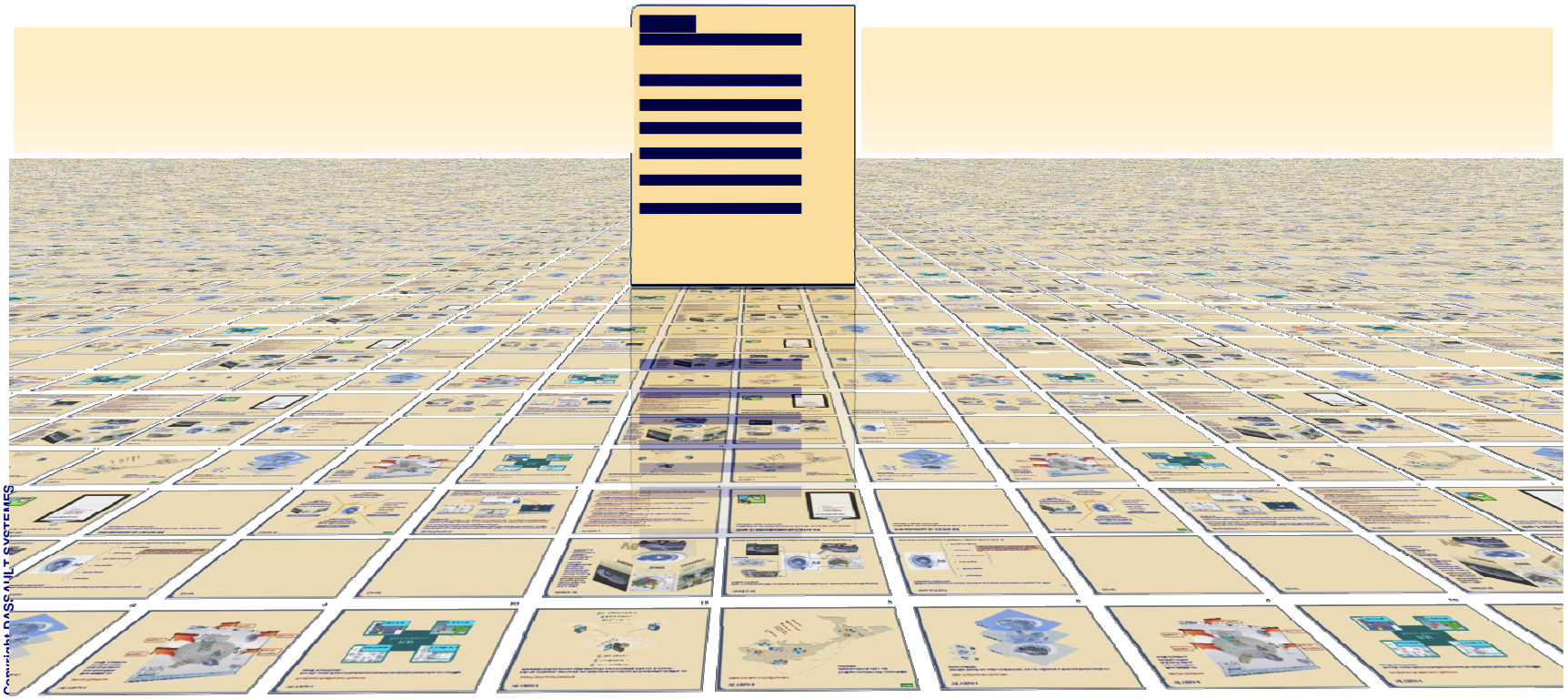
- ✓ Design in context using the skeleton method
- ✓ Publish geometry
- ✓ Position geometry using the skeleton method
- ✓ Propagate design changes



Student Notes:

To Sum Up

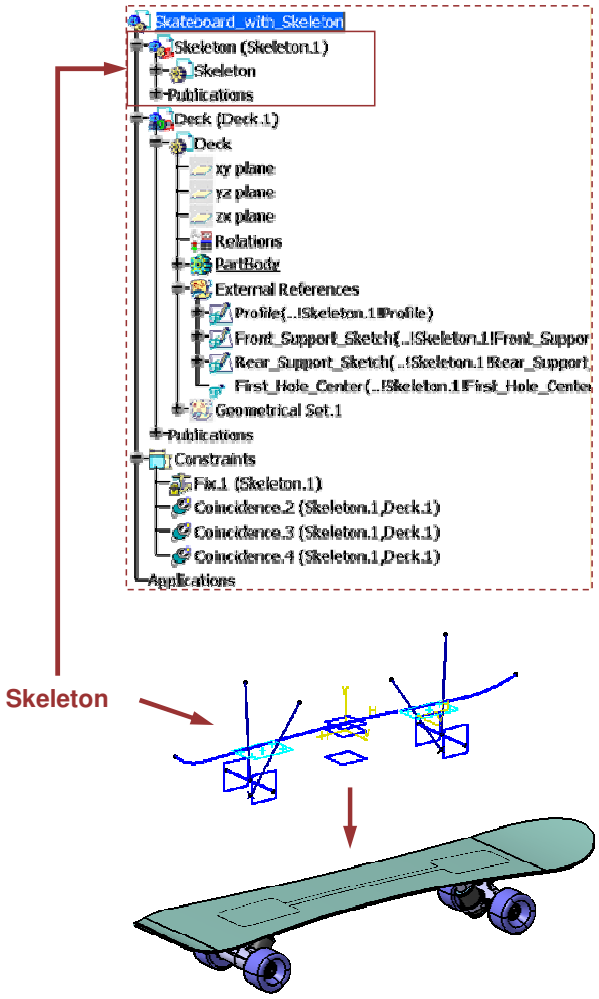
In the following slides you will find a summary of the topics covered in this lesson.



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.

Student Notes:

Student Notes:

Create Published Geometry

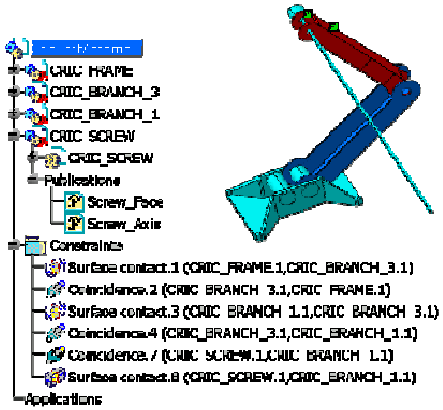
Publishing geometrical elements is the process by which geometrical features are made available to different users. It is most applicable to the geometry and parameters of a skeleton model. It has the following benefits:

A. Label geometry to give it a name that can be easily recognized (particularly in the case of publishing edges, faces, etc.).

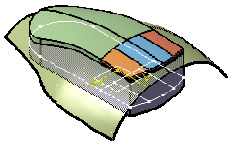
B. To make particular geometry easier to access from the specification tree.

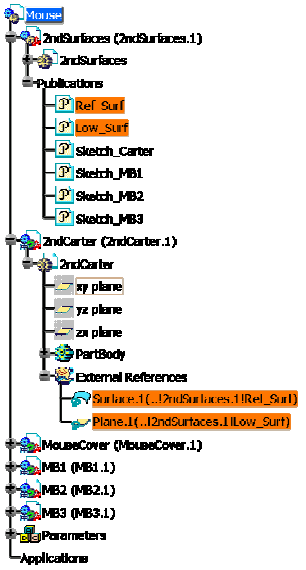
C. Control external references. An option is available that lets you to select as external reference only (For the published elements).

D. Easy replacement of one feature of the part with another.



Use Publications for creating assembly constraints





Use Publications in contextual design

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.

Student Notes:

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.