STUDENT GUIDE

Student Notes:

CATIA V5 Training
Foils

Mechanical Design V5R19 Update

Version 5 Release 19
August 2008

EDU_CAT_EN_MD2_UF_V5R19

Student Notes:

About this course

Objectives of the course

Upon completion of this course you will be able to use the new and enhanced tools of the Mechanical Design Workbenches in V5R19 Release.

Targeted audience

Mechanical Designers

Prerequisites

Students attending this course should have knowledge of
CATIA Mechanical Design V5R18

1 day

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Student Notes:

Sketcher Updates

You will learn about the following new and enhanced functionalities of the CATIA Sketcher workbench.

- **What's New in CATIA Sketcher Workbench**
- **New command: Upgrade**
- **New command: Explode**
- **Enhancement in Positioning As Result With Link Sketches**

Student Notes:

What's New in CATIA Sketcher Workbench

The list of enhanced functionalities are given below:

New Functionalities:

- **Upgrading the Features:**
  - The new ‘Upgrade’ contextual command is available on the Sketch features which allow you to access the ‘Best so far’ level of the software compatible with sketch features data.
- **Exploding Sketch:**
  - The new ‘Explode’ contextual command allows you to convert ‘As Result With Link’ pasted sketch (non-editable Sketch), into an editable sketch.

Enhanced Functionality:

- **Positioning As Result With Link sketches:**
  - This capability allows you to position the sketch obtained by ‘As Result With Link’ paste option. You can position the sketch by specifying the directions and position of origin in the Sketch Positioning dialog box.
  - You can also restore the link in position with the copied feature using the new option ‘Positioned as reference’ which is available in the Type field.

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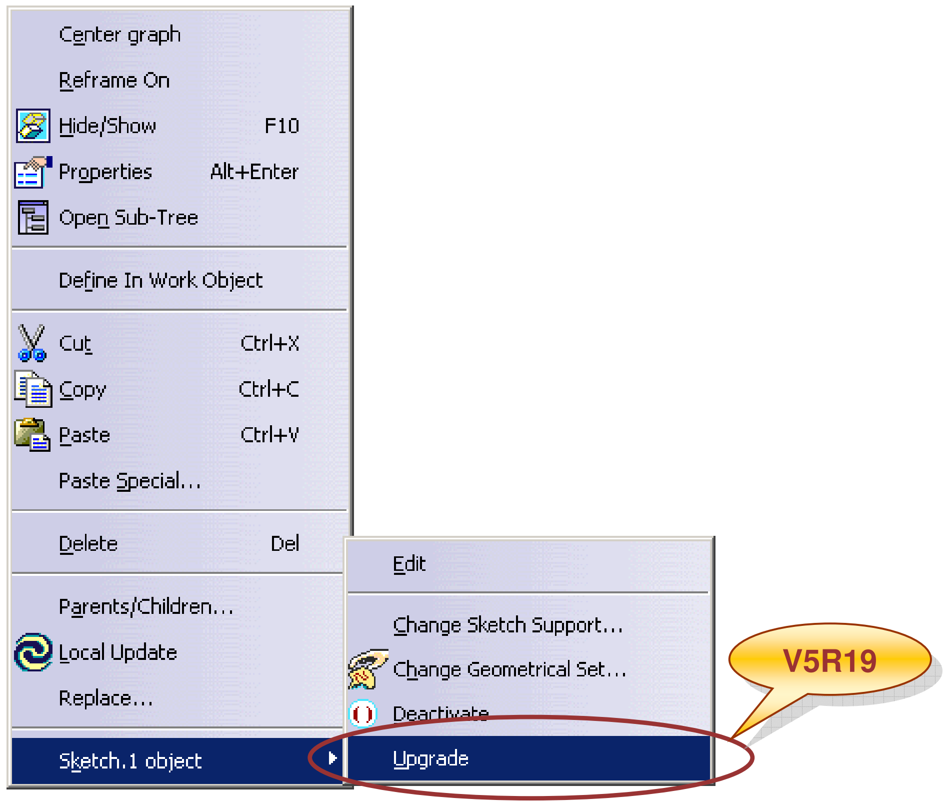
Student Notes:

New Command: Upgrade

You will learn how to Upgrade the sketch features.



Student Notes:

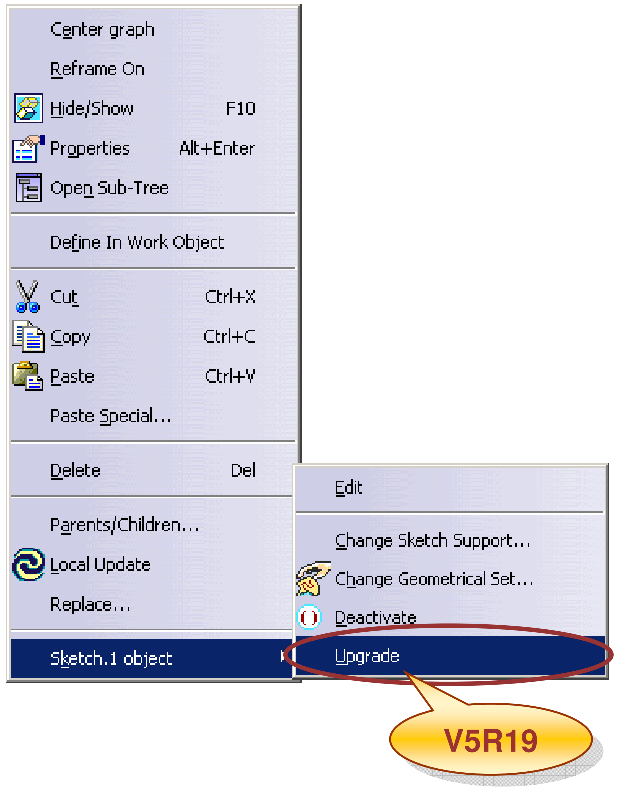
About the New Upgrade Command

The new Upgrade command will modify all the versioning data in order to put in place the latest version compatible with the data contained in the sketch. It is available through the contextual menu on the sketch feature.

Upgrade capability on the sketch features allows:

- **End-users to take advantage of all the versioned corrections between the original level and current level, without recreating the feature.**
- **End-users to use the last optimized algorithms (performance improvements) and also access the new capabilities/behaviors.**
- **Administrators to upgrade the old template data.**

This capability is also applicable for part features.



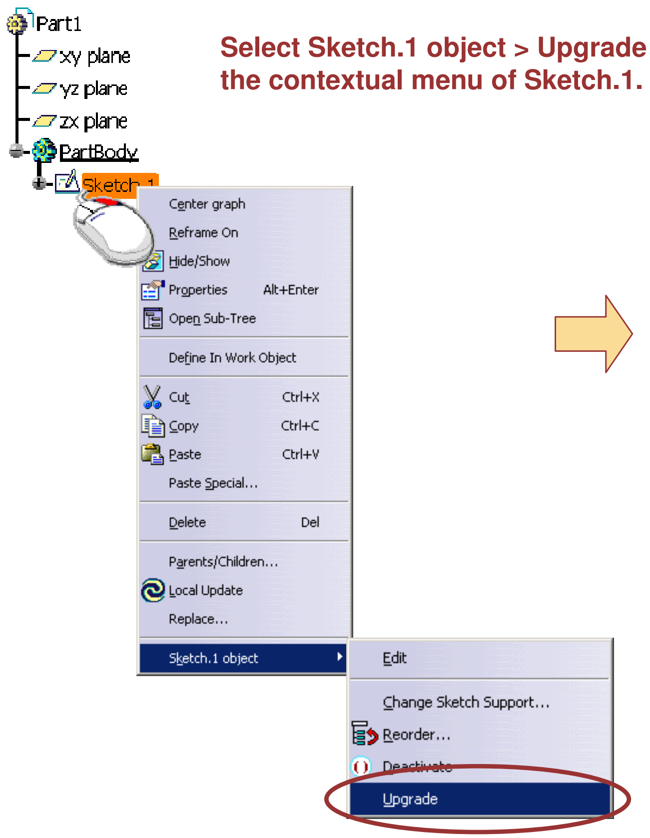
It is necessary to update the feature and its subsequent features after Upgrade command. The orientation of the element may be modified after Upgrade.

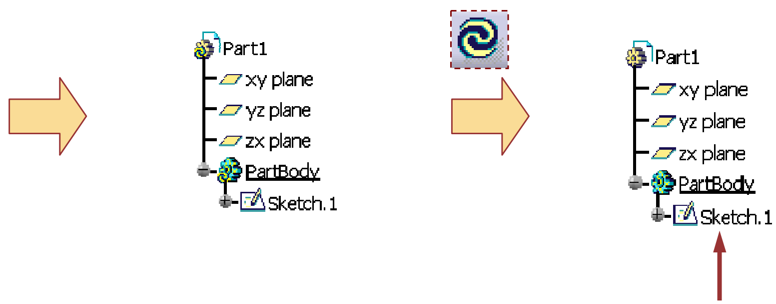
Student Notes:

How to Upgrade a Sketch

You can upgrade a sketch feature by using the contextual command Upgrade.

Select Sketch.1 object > Upgrade from the contextual menu of Sketch.1.



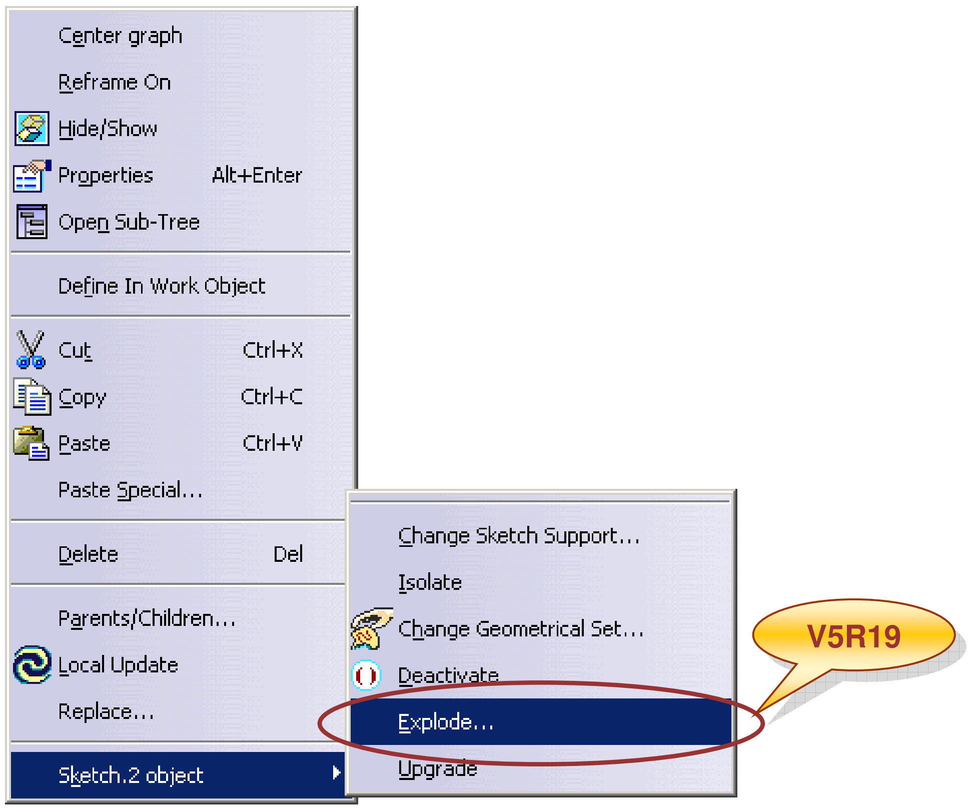


Upgrade operation will modify all the versioning information of Sketch.1 in order to put in place the ‘Best so far’ version that is compatible with the existing Sketch data.

Student Notes:

New Command: Explode

You will learn how to Explode the Sketch feature.

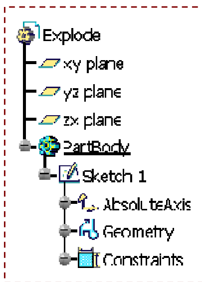


Student Notes:

Why to Explode

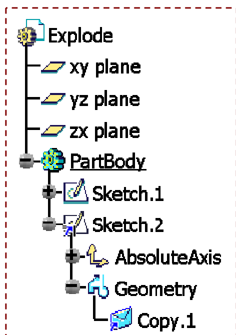
A sketch obtained by copying and pasting with As Result With Link option is a copy of its reference sketch feature.

Exploding the sketch removes the Copy feature from the specification tree and converts every wireframe geometry associated to the datum feature to a standard 2D geometry feature.

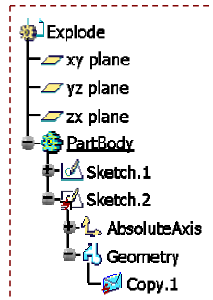


Original Sketch

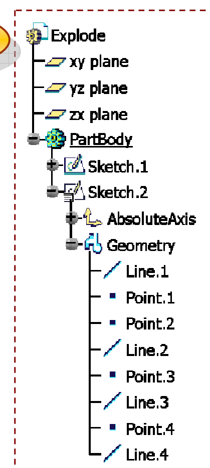
Copy/Paste
As Result with Link



Associativity is maintained between the reference geometry and the copy.
The copied sketch is non-editable.



Sketch feature is non-associative, and the isolated sketch is non-editable.



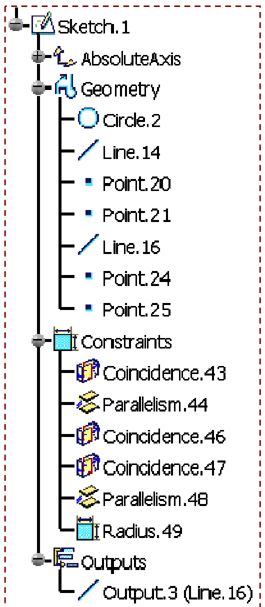
You cannot edit sketch obtained by copying and pasting with As Result With Link option till V5R18.
The new 'Explode' command helps you to avoid this problem.

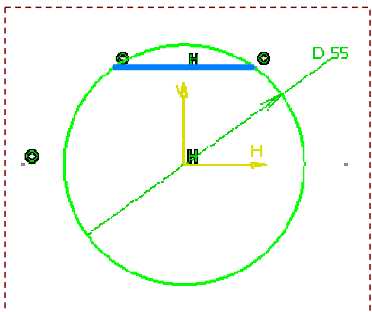
After this 'Explode' operation, there is no more associativity between the 'Exploded' Sketch and its previous reference Sketch. The exploded sketch is editable.

Student Notes:

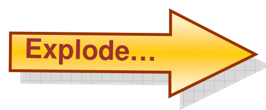
More About Explode

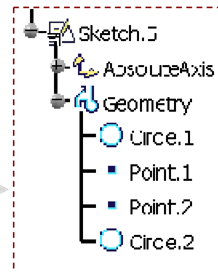
- **The copied sketch does not take into account the following data of its reference sketch while exploding:**
  - **Construction, Axis geometries, and Use-edges (projections, intersections etc)**
  - **Constraints and Dimensions**
  - **Geometry on Output feature**

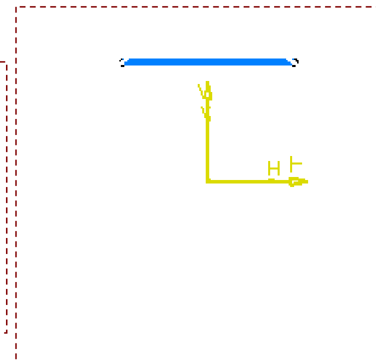




Copy/Paste
As Result with Link







You can see Axis geometry, Constraints and Output features are not retrieved after explode.

- **Generic naming which identifies each geometry will not be kept when performing the explode operation.**
- **The order of the geometry in the specification tree after explode operation is different from the order in the reference sketch.**

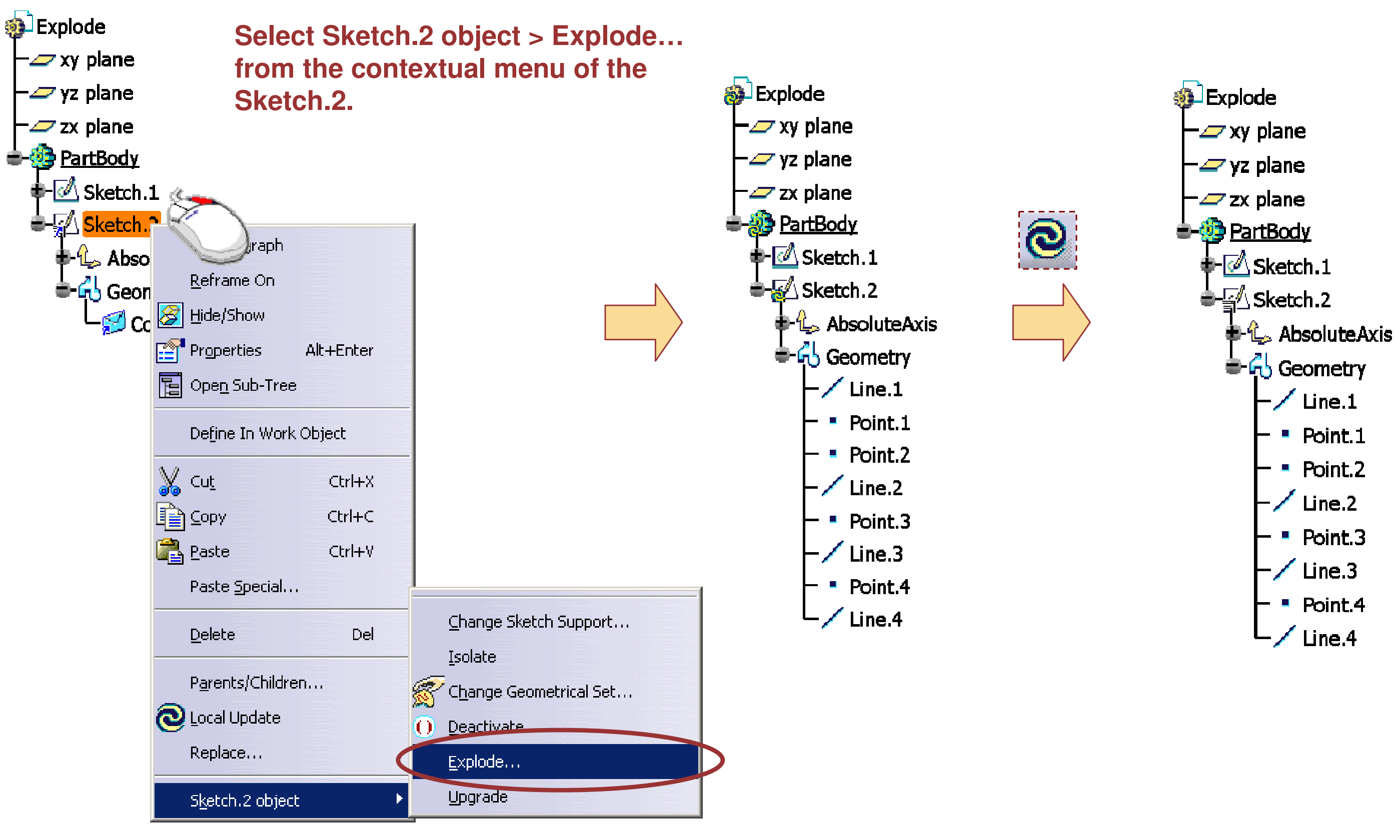
After explode, it is necessary to update the sketch feature.

Student Notes:

How to Explode a Sketch

You can explode the sketch feature by using the contextual command Explode...

Select Sketch.2 object > Explode… from the contextual menu of the Sketch.2.



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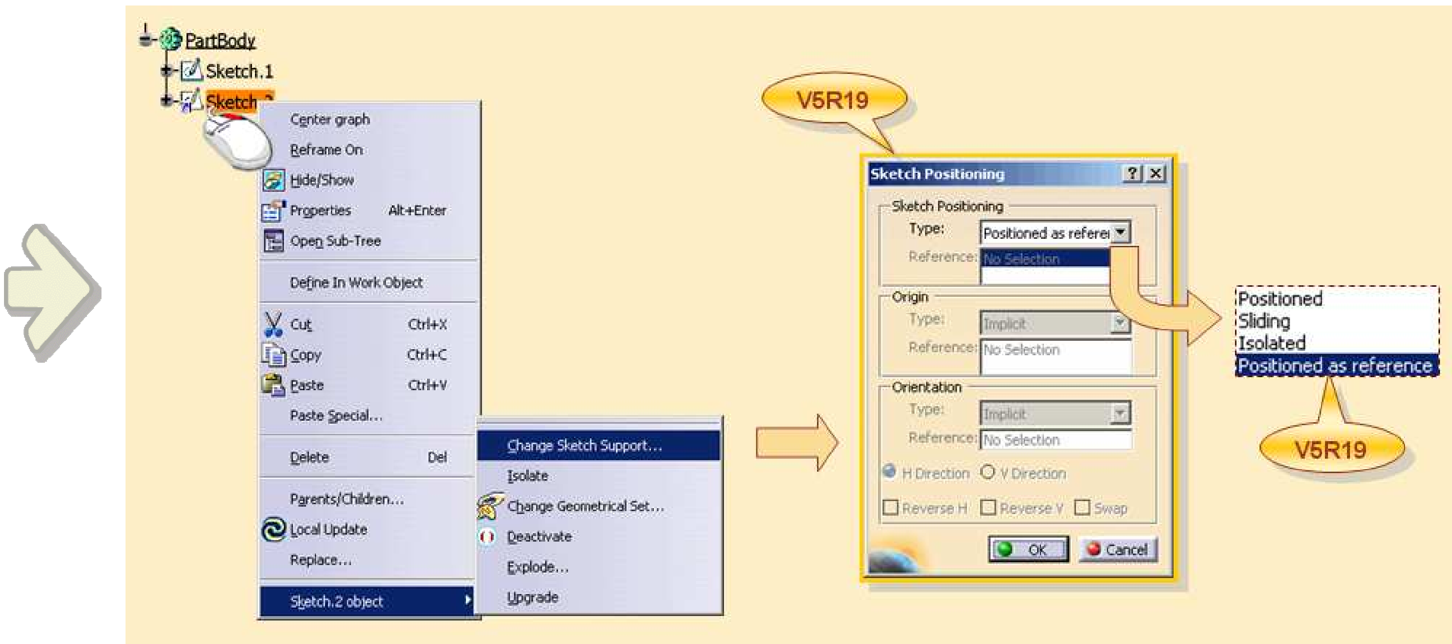
Student Notes:

Enhancement in Positioning As Result With Link Sketches

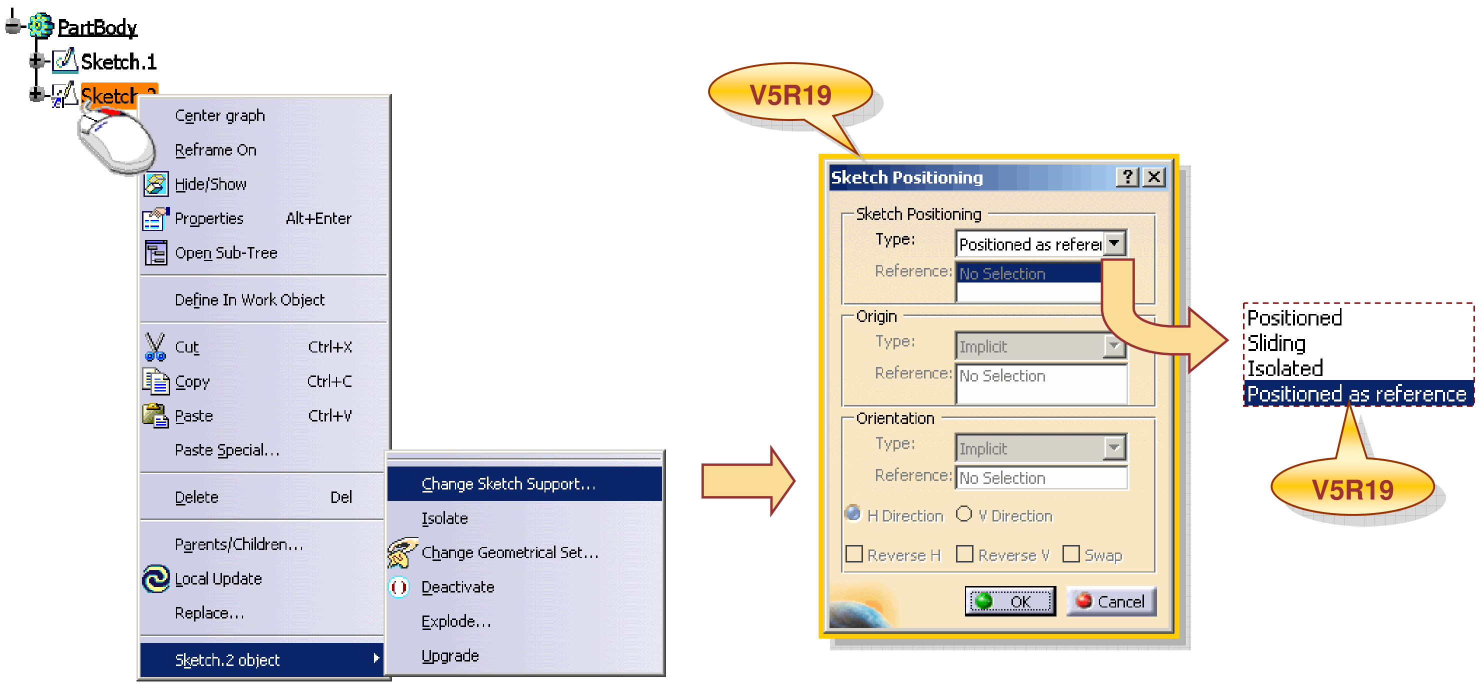
You will learn how to position As Result With Link Sketches.



Student Notes:

About Enhancement in Positioning As Result With Link Sketches

You can position the sketch (obtained by copying and pasting) using the 'As Result With Link' option. You will be able to use the 'Sketch Positioning' dialog box to position 'As Result With Link' pasted sketches.



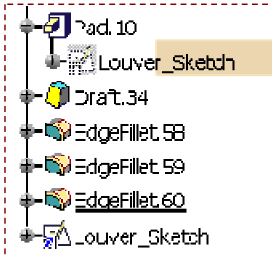
A new option 'Positioned as reference' is available for positioning the sketch. This enhancement will allow you to restore the positional link with the copied feature. When you select the Positioned as reference type, all the other fields are disabled and set to the default value.

Student Notes:

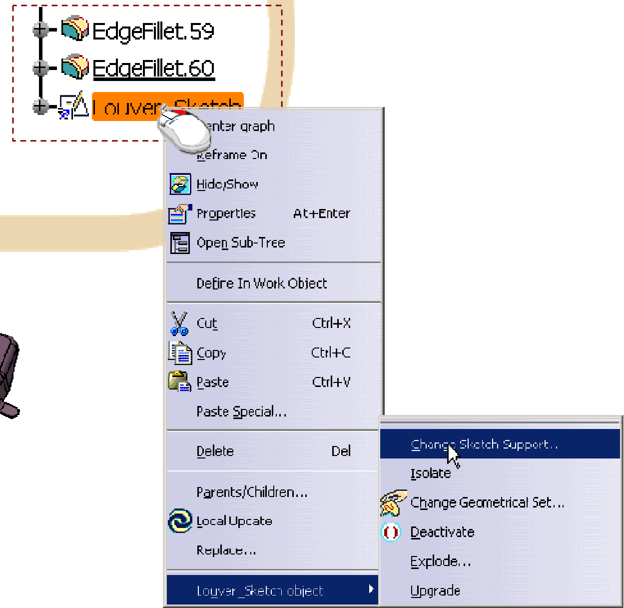
How to Position ‘As Result With Link Sketch’ (1/2)

Let us learn how to use position As Result With Link Sketch to create louvers.

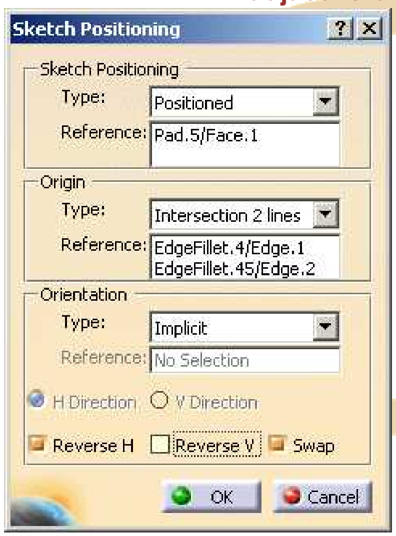
1 Copy the Louver_Sketch and paste it with ‘As Result With Link’ option.



2 To change the position of the copied sketch, select Change Sketch Support from the contextual menu.



3 Use the Positioned Type to position the sketch on the adjacent face.

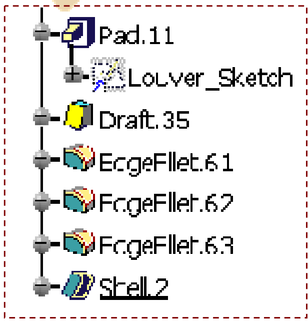


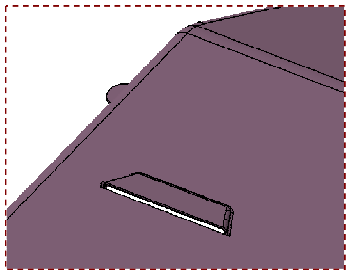
Student Notes:

How to Position 'As Result With Link Sketch' (2/2)

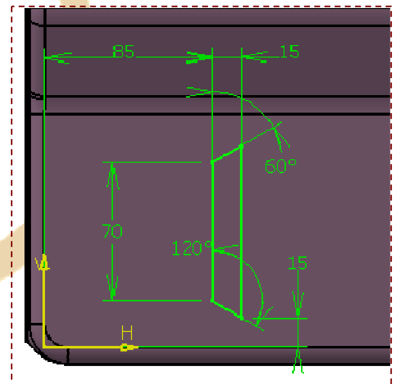
Let us learn how to use position As Result With Link Sketch to create louvers.

4 Use the copied positioned sketch to create a Louver.

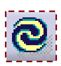


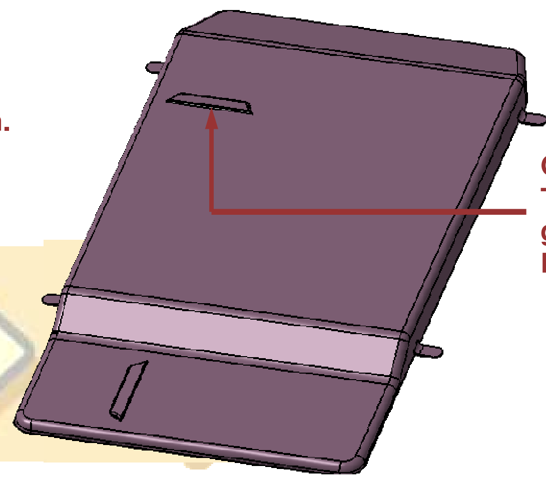


5 Change the dimensions of the original Louver_Sketch. Change the width of Louver.



6 Update the Sketch.





Observation:
The copied Louver_Sketch gets updated as per original Louver_Sketch.

Student Notes:

Part Design Updates

You will learn about the following new and enhanced functionalities of the CATIA Part Design workbench:

- What's New in CATIA Part Design Workbench
- Enhancement in the Edge Fillet
- Recap Exercise: Edge Fillet with Blend Corner
- Enhancement in the Hole Definition
- Other Enhancements

Student Notes:

What's New in CATIA Part Design Workbench

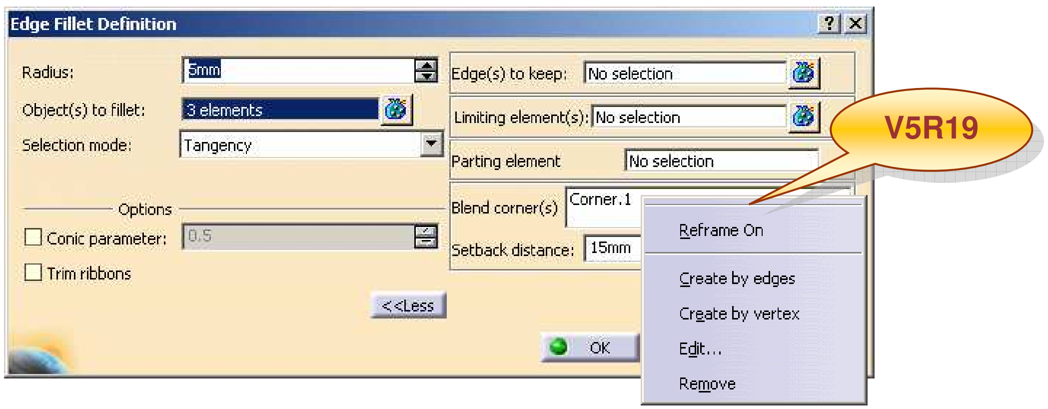
The list of enhanced functionalities in CATIA Part Design V5R19 is given below:

- **Enhancement in the Edge Fillet:**
  - **This contextual menu is added to create and manage the ‘blend corner' capability of the edge fillet. Now it is easier to edit the edge fillet containing blend corners.**
- **Enhancement in the Hole Definition:**
  - **This highlight adds capability to select the anchor point of the tapered hole. Two check boxes are provided to select the anchor point of the hole.**
- **Upgrading the Part Design Features:**
  - **This new contextual command is available on the part design features which allow you to access the current release level of CATIA and its data.**
- **Displaying and Editing Parameters in the specification tree :**
  - **This enhancement provides a display of features parameters under its node in the tree. This is very convenient to quickly edit the values without opening the dialog box.**

Student Notes:

Enhancement in the Edge Fillet

You will learn about the new options available for managing the blend corners in the Edge Fillet.



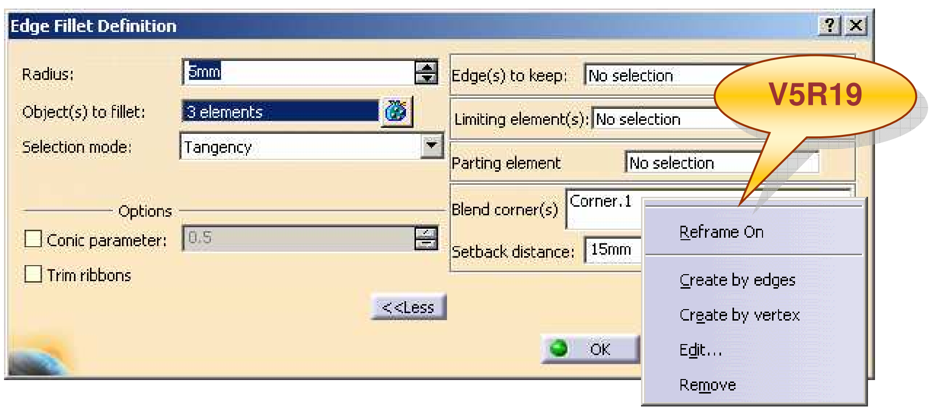
Student Notes:

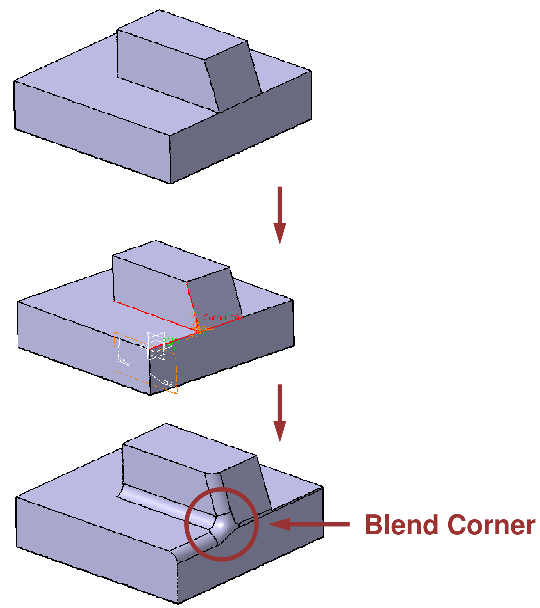
About Enhancement in the Edge Fillet (1/3)

While applying the fillets to the sharp edges, the corners resulting from the operation are not always satisfactory. The goal of this enhancement is to easily create and edit the 'blend corners' of the edge fillet.

A contextual menu has been added in the blend corner field to create, edit and manage the blend corners of the edge fillet.

In V5R18, when you change the edges to be filleted, then the blend corners must also be redefined, even though they are not impacted by the modifications performed in the edge fillet. There is no way to keep the definition of the existing blend corners.



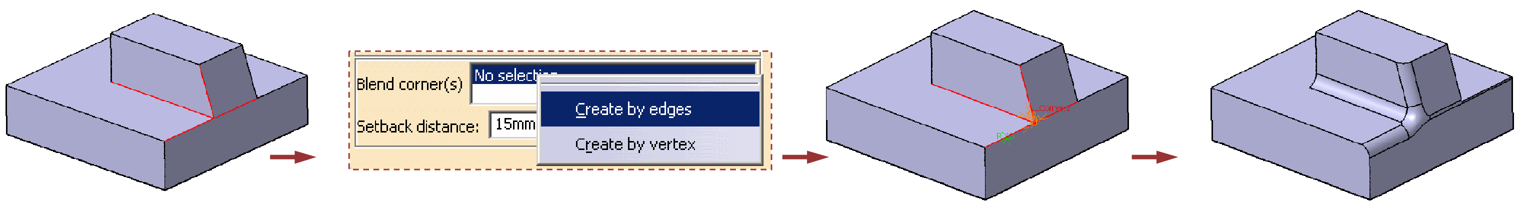


Student Notes:

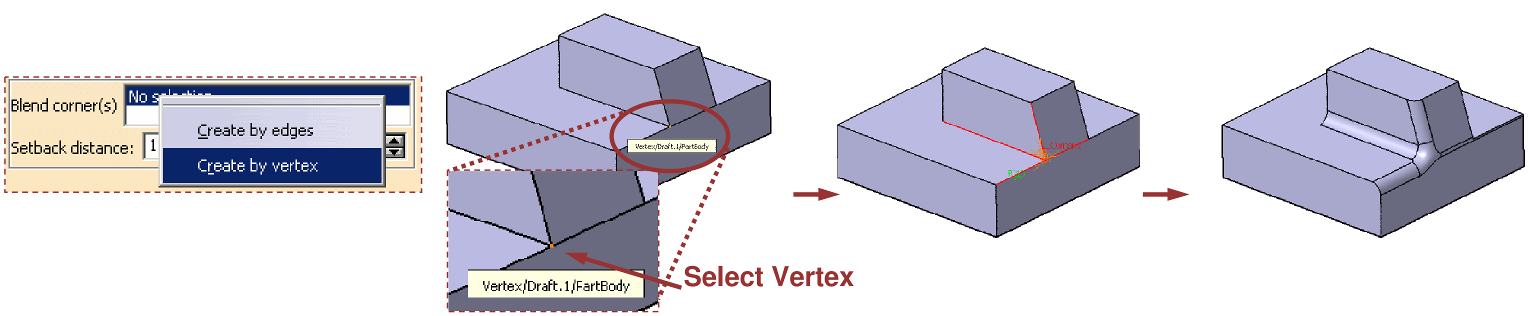
About Enhancement in the Edge Fillet (2/3)

The enhancement provides the following options in the Blend corner's contextual menu to create and edit an edge fillet containing at least one blend corner :

- Reframe On: It reframes the model at the selected corner.
- Create by edges: It automatically adds the corresponding blend corner with default setback distance. The selected and propagated edges are added to the list of objects. This option does not remove the existing blend corners.



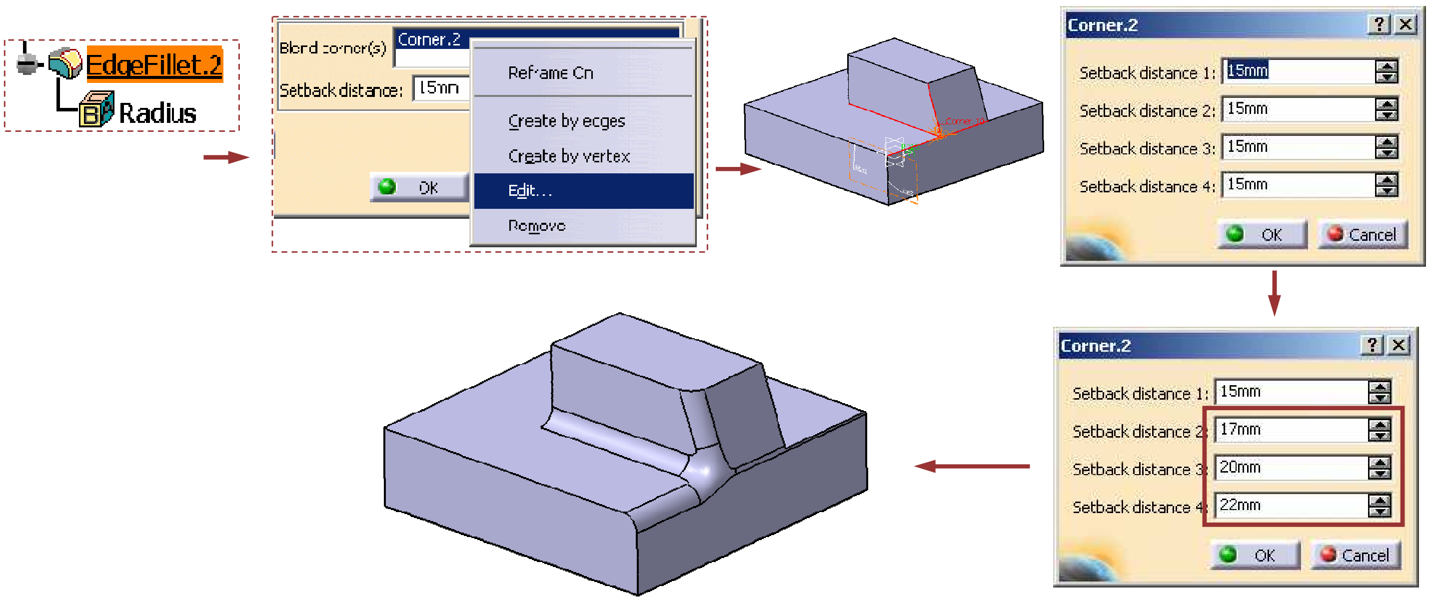
- Create by vertex: It automatically adds the corresponding blend corner with default setback distance. The concurrent edges of the selected vertex are added to the list of object(s) to fillet if these edges are new ones or if they are not part of the propagations. This option is available even if some corners are already defined.



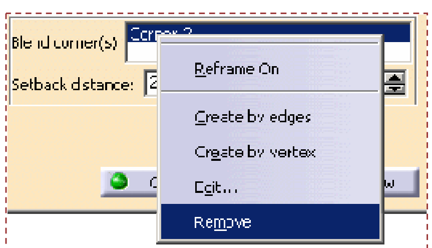
Student Notes:

About Enhancement in the Edge Fillet (3/3)

- **Edit: This option displays a list of editable setback values. However, an error message is displayed if you edit multiple blend corners.**



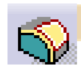
- **Remove.**



Student Notes:

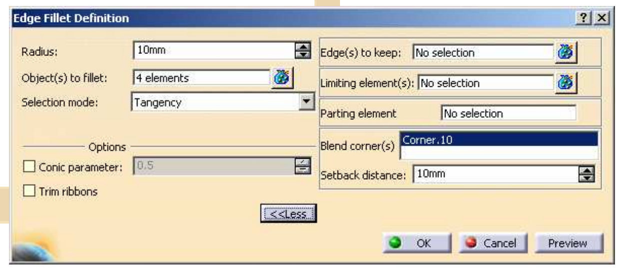
How to Create a Blend Corner Using Create by Edges

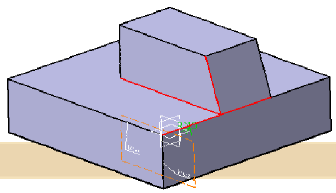
You will perform the following steps to create a Blend Corner using Create by edges option



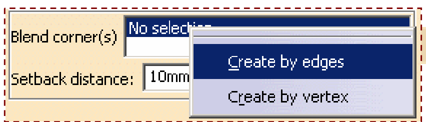
1 Click on the Edge Fillet icon.

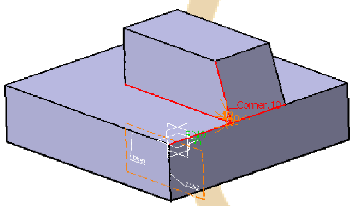
2 Select the edges to be filleted.



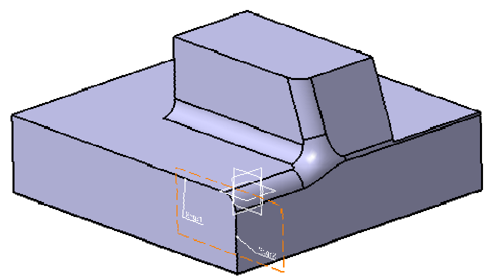


3 Right-click the Blend corner(s) field and select Create by edges.





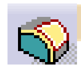
4 Enter the Setback distance as 15mm and click OK to create the Edge fillet.



Student Notes:

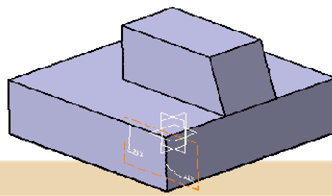
How to Create a Blend Corner Using Create by Vertex

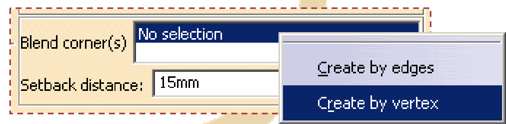
You will perform the following steps to create a Blend Corner using Create by vertex option



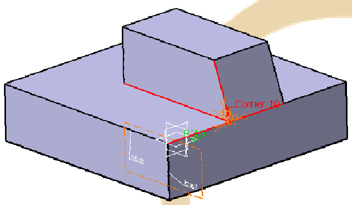
1. Click on the Edge Fillet icon.

2. Right-click the Blend corner(s) field and select Create by vertex.

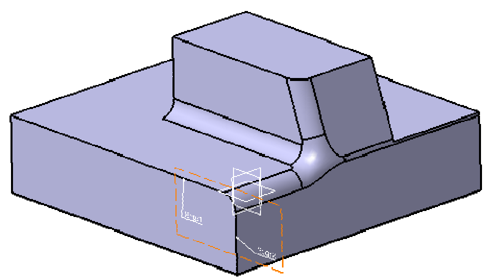




3. Select the vertex.
(The concurrent edges will be automatically selected.)



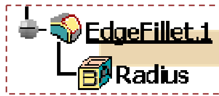
4. Enter the Setback distance and click OK to create the Edge fillet.



Student Notes:

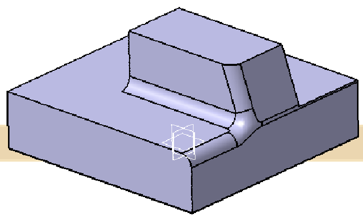
How to Edit a Blend Corner in the Edge Fillet

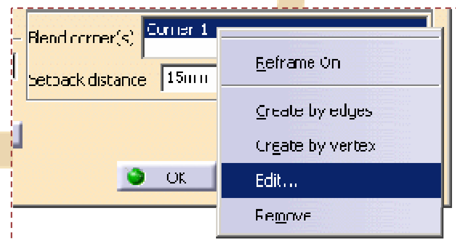
You will perform the following steps to edit a Blend Corner in the edge fillet



1 Double-Click the Edge Fillet in the specification tree.

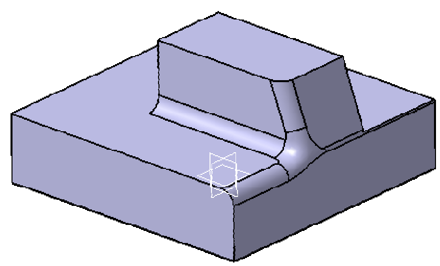
2 Right-click the Corner.1 in the Blend corner(s) field and select Edit.





3 Enter the Setback distance values and click OK.

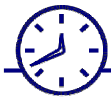
4 Click OK to create the Edge fillet.



Student Notes:

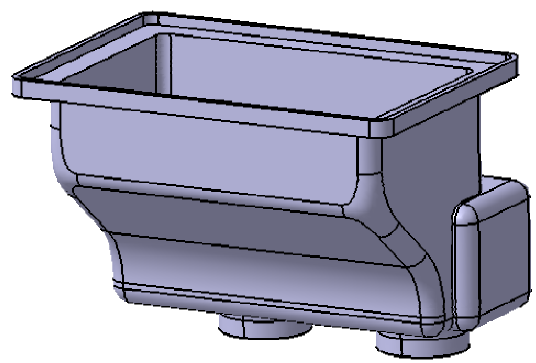
Edge Fillet with Blend Corner

Recap Exercise



In this exercise you will create:

- **Edge Fillet with Blend Corner**
- **Blend Corner with Setback distance**
- **Edge fillet with different setback distance values**

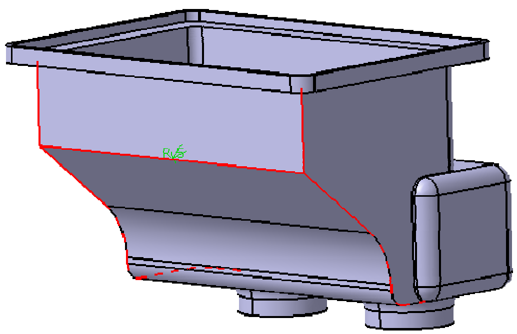


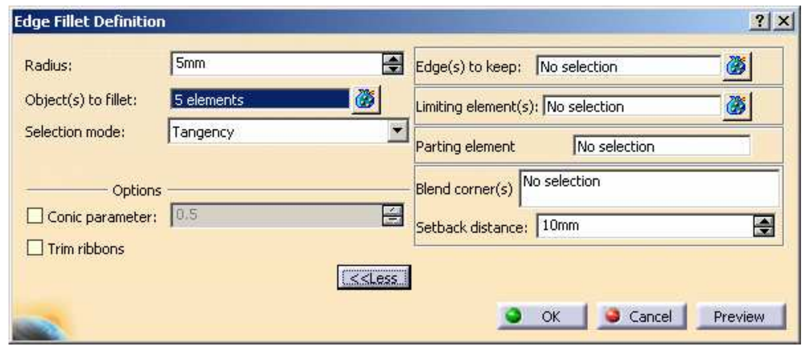
Student Notes:

Do It Yourself (1/3)

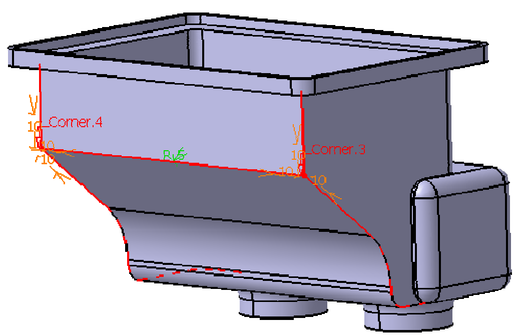
Part used: UMD19_BlendCorner_5edges_Start.CATPart

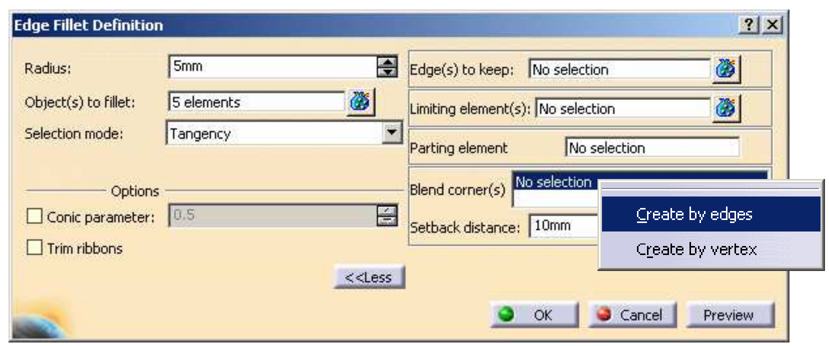
- **Select the five edges of the solid.**





- **Right-click the Blend corner field and select 'Create by edges'.**

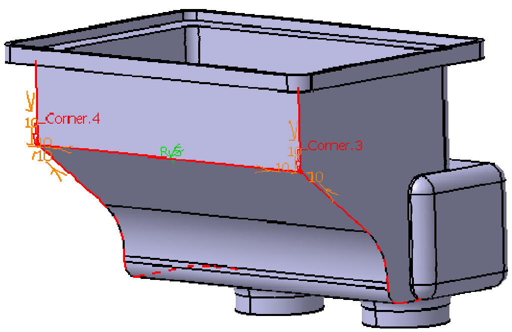


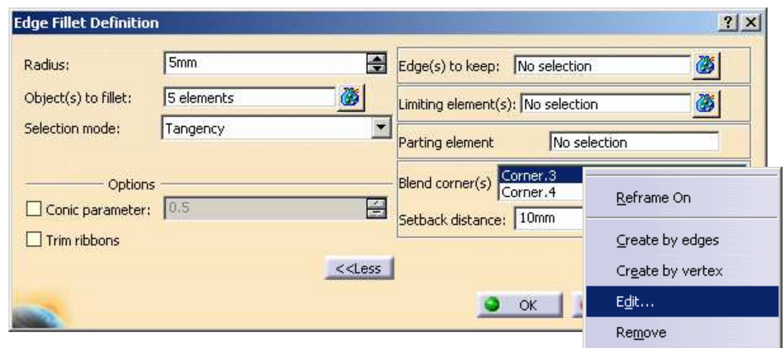


Student Notes:

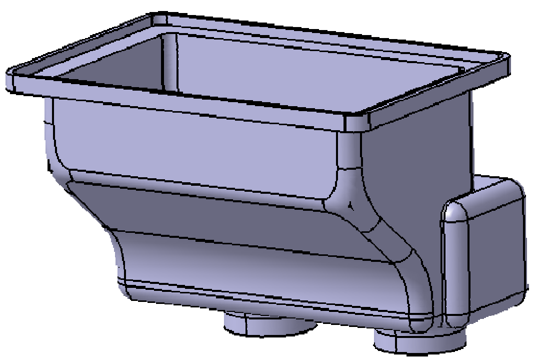
Do It Yourself (2/3)

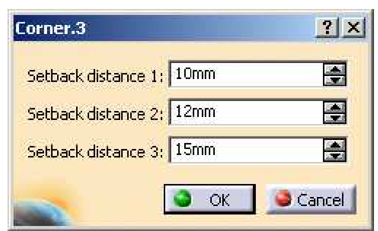
- **Right-Click the newly created Corner.9 in the Blend corner field and select Edit…**

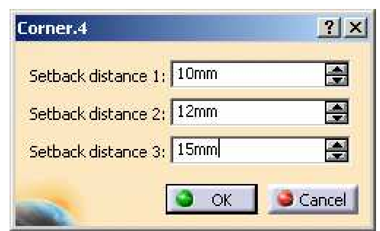




- **Enter the different values of Setback distance and click OK to create the fillet.**



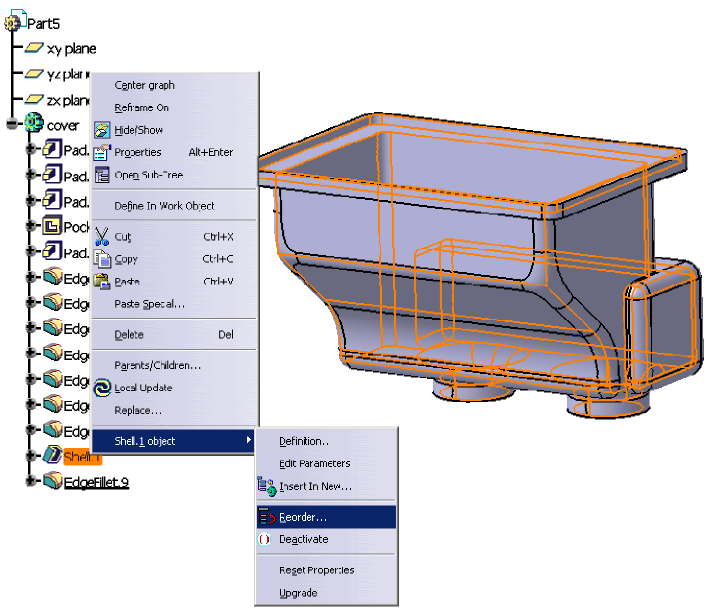


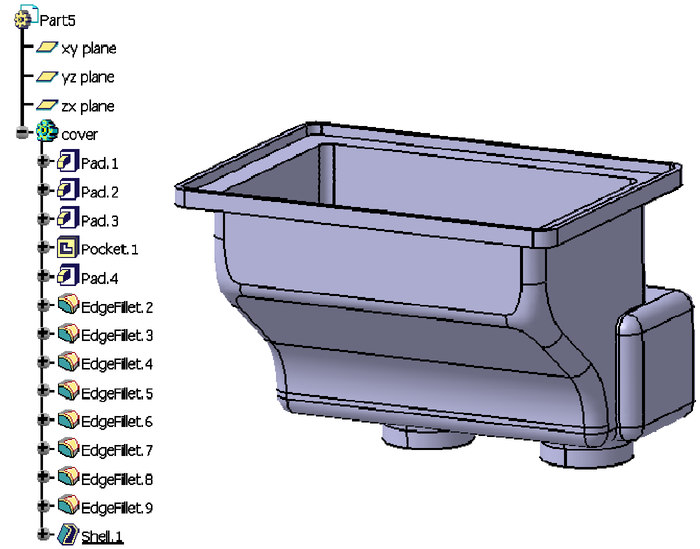


Student Notes:

Do It Yourself (3/3)

- **Right-Click the Shell.1 and reorder it to the end of the specification tree.**





Observation:

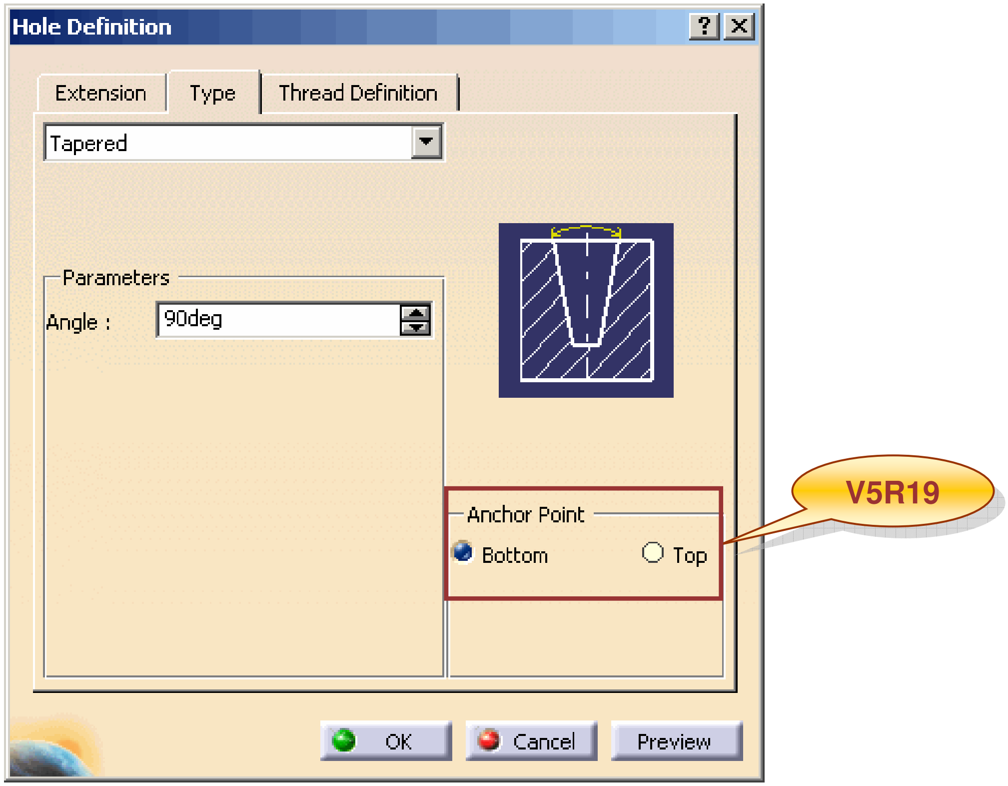
You can see that now it is easier to edit the fillets with blend corners.

End Part: UMD19_BlendCorner_5edges_End.CATPart

Student Notes:

Enhancement in the Hole Definition

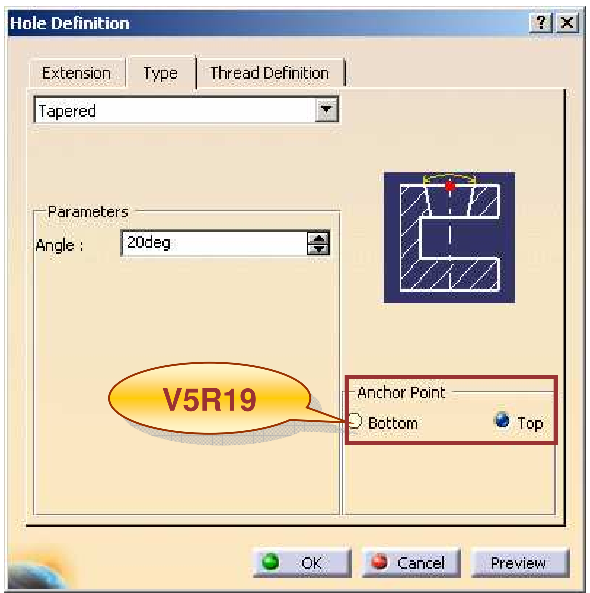
You will learn about the anchor point in the tapered hole.

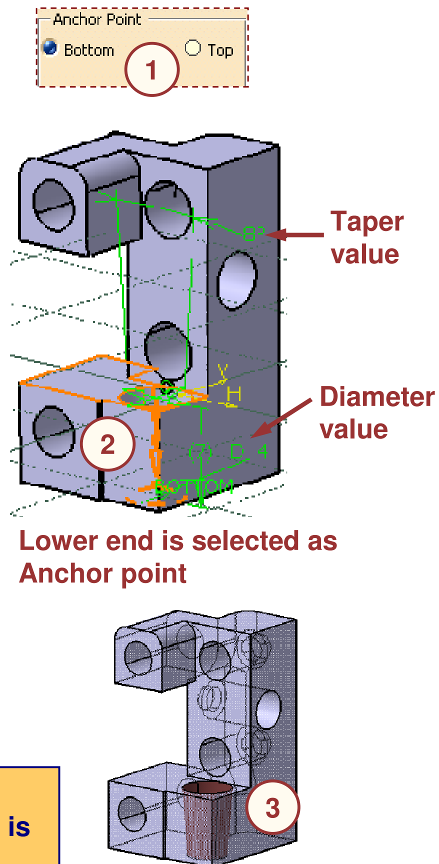


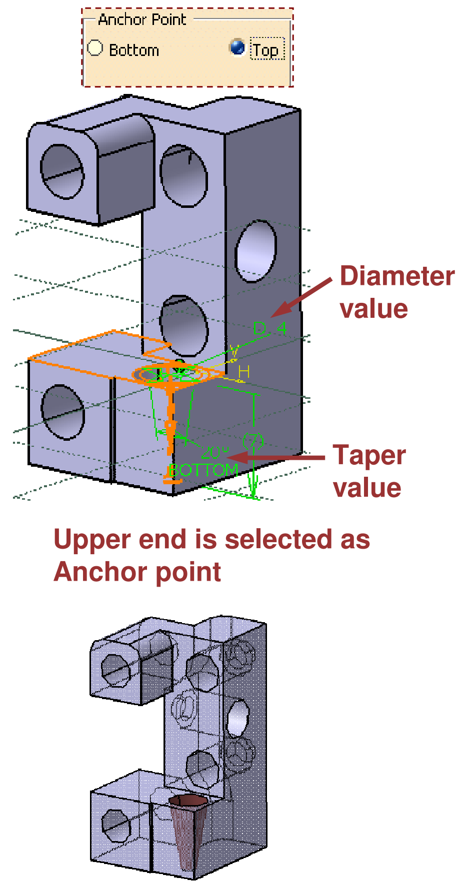
Student Notes:

About Enhancement in the Hole Definition (1/2)

You can specify the diameter value for the ‘Tapered’ hole at the level of the selected face. You can specify the diameter value for the hole's upper or lower end by selecting Top or Bottom in the Anchor Point selection.





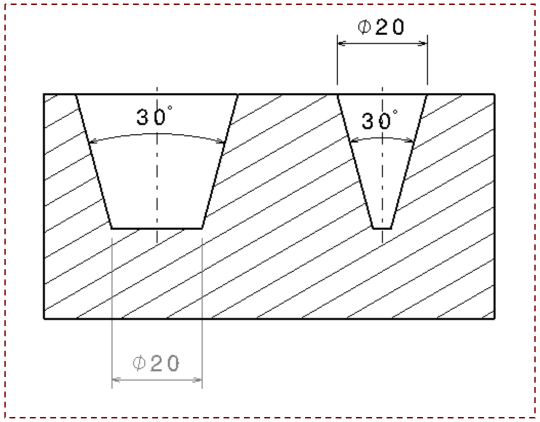


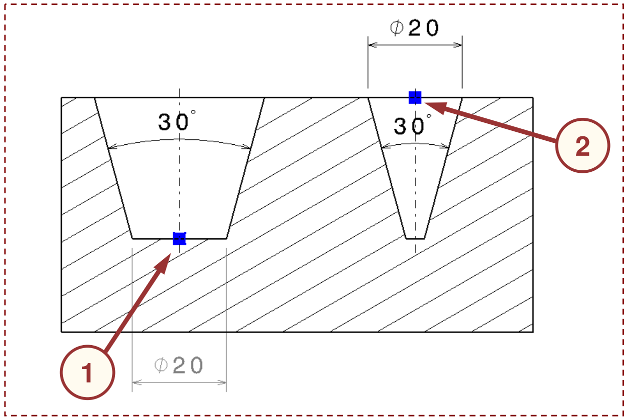
If the bottom diameter leads to an invalid profile, the main diameter is kept and the hole is computed like a simple hole.

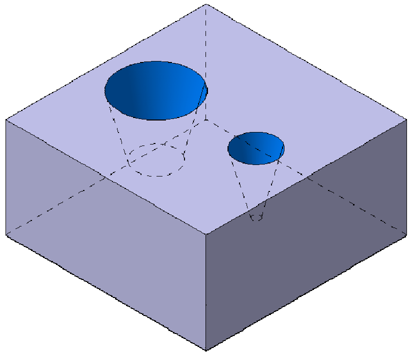
Student Notes:

About Enhancement in the Hole Definition (2/2)

In the example discussed in the previous slide, the design intent is to have holes as per the following drawing. This design intent is now quickly achieved in V5R19 using the new enhancement of Anchor point setting.







1 Anchor Point: Bottom

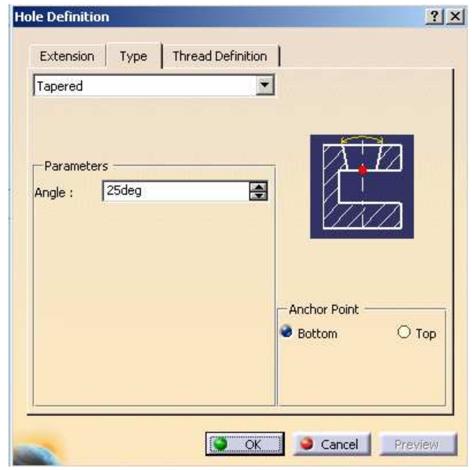
2 Anchor Point: Top

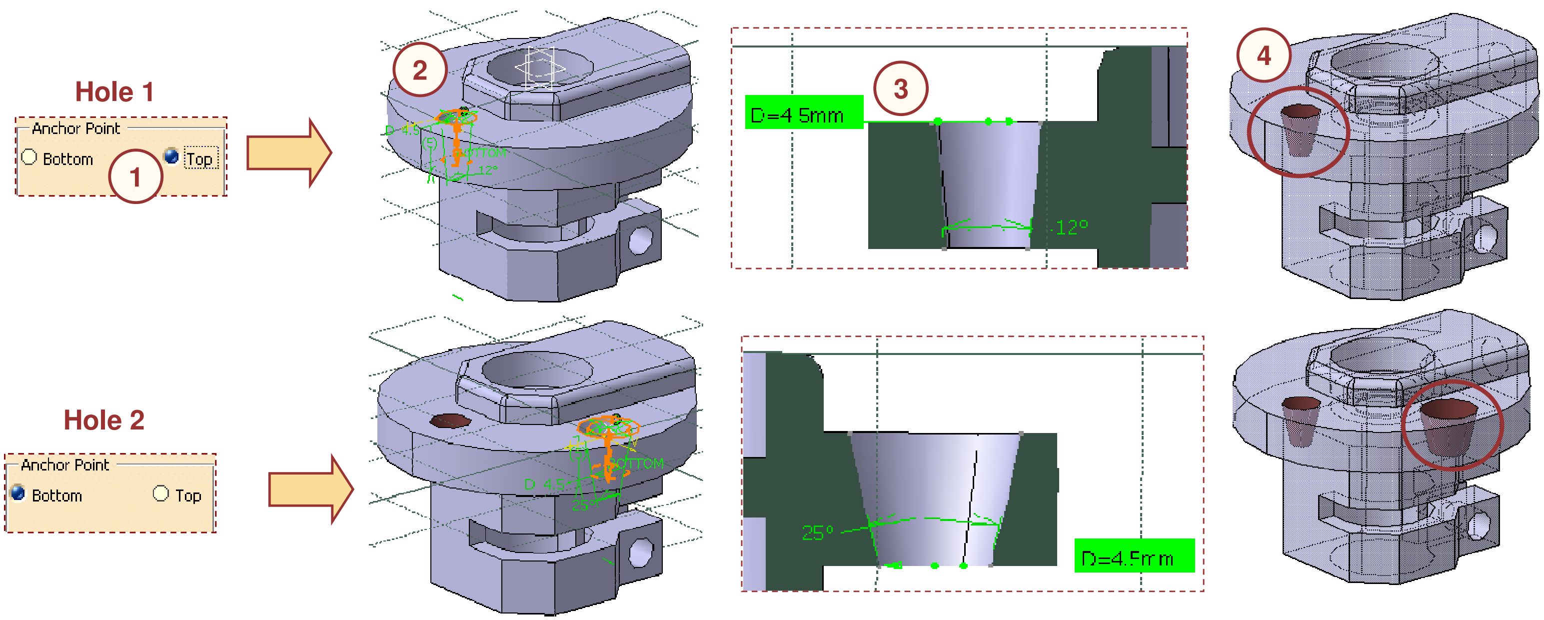
Student Notes:

Do It Yourself

Part used: UMD19_Hole_Definition.CATPart

- Create the hole
  - Diameter = 4.5mm; Depth = Up to next
  - Tapered type with 12º angle
  - Anchor point as ‘Top’
- Create the hole
  - Diameter = 4.5mm; Depth = Up to next
  - Tapered type with 25º angle
  - Anchor point as ‘Bottom’

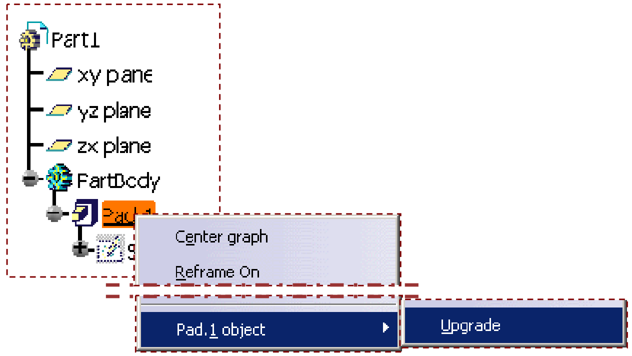




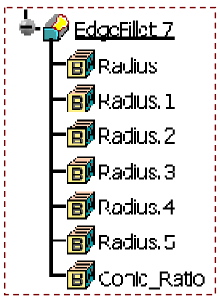
Student Notes:

Other Enhancements

- **Upgrading the Part Design Features:**
  - **This new ‘Upgrade’ contextual command available in the part design features allows you to access the current release level of CATIA and its data.**
  - **It enables you to take advantage of all the versioned corrections between the original level and the current level of CATIA, without recreating the feature.**



- **Displaying and Editing Parameters in the specification tree:**
  - **It displays the parameters under features node in the tree. This is very convenient to quickly edit the values without opening the dialog box.**
  - **It helps you to see which parameters are bound by knowledge and check the parameter value.**
  - **It improves your productivity when modifying an input value for a Part Design feature.**



Student Notes:

Assembly Design Updates

You will learn about the following new and enhanced functionalities of the CATIA Product Design workbench:

- What's New in CATIA Assembly Design Workbench
- New Command: Associativity
- New Command: Add to Associated Part
- Recap Exercise: Associativity and Add to Associated Part
- Enhancement in the Constraint Creation

Student Notes:

What's New in CATIA Assembly Design Workbench

The list of enhanced functionalities in CATIA Assembly Design V5R19 is given below.

- **New Command: Associativity**

  This capability allows you to modify the CATPart geometry in the assembly context (without modifying the reference) to create a new CATPart. The new part is instantiated in the assembly. It contains a copy of the chosen geometry, which is obtained by the Copy/Paste As Result With Link operation. If required, you can also select customized components of the active assembly.

- **New command: Add to Associated Part**

  This capability allows you to create a body obtained by Copy As Result With Link of a chosen geometry. The created body will be added to the selected CATPart's PartBody.

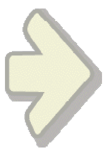
- **Redundancy check during constraint creation:**

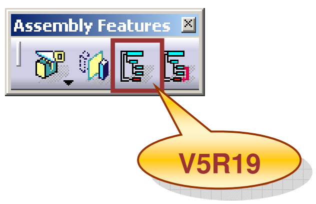
  This enhancement provides you with an option in CATIA Settings to enable/disable the redundancy check performed during constraint creation.

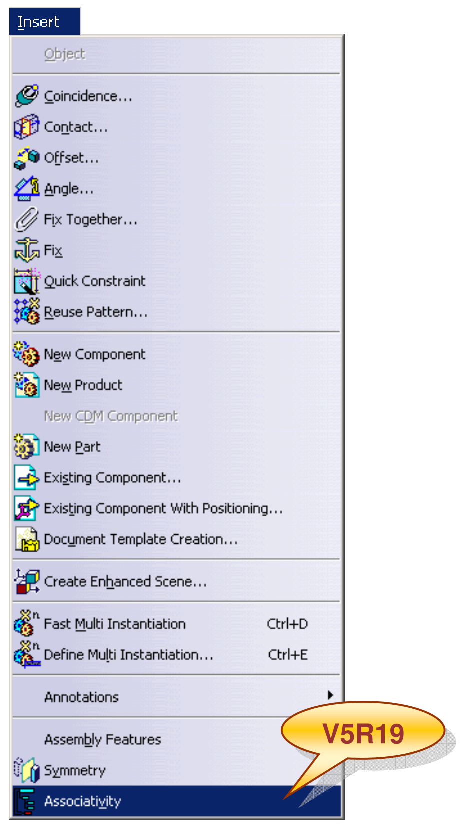
Student Notes:

New Command: Associativity

You will learn how to create and modify an associated CATPart of the CATProduct without modifying the reference CATParts.





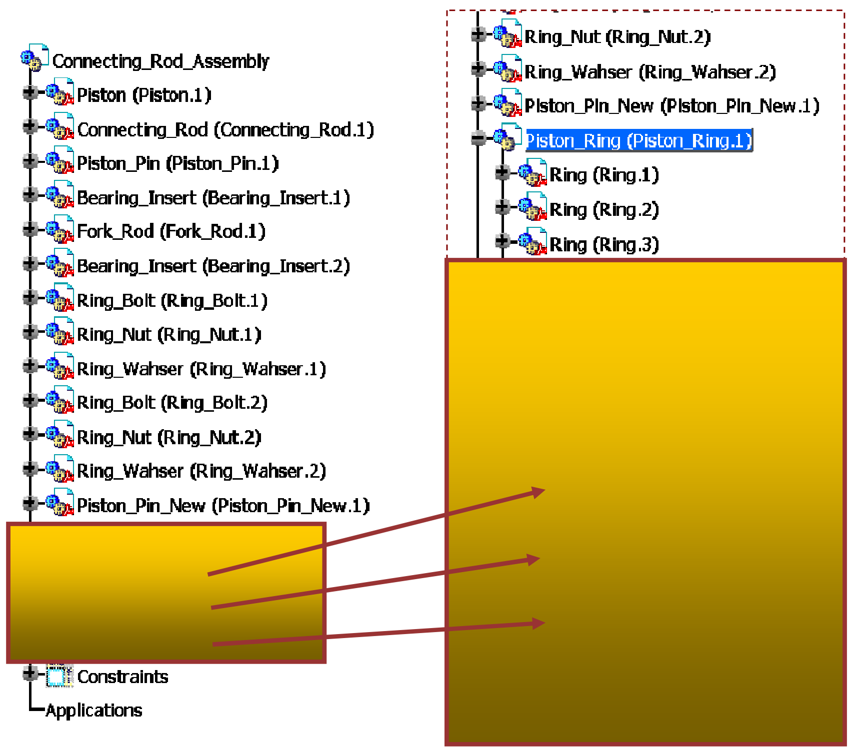


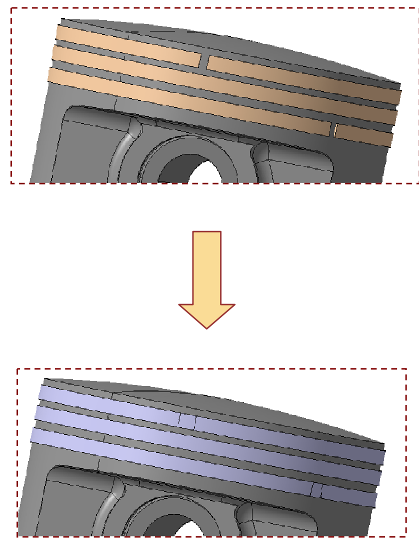
STUDENT GUIDE

Student Notes:

What is an Associativity

The 'Associativity'command allows you to modify a CATPart geometry in the assembly context (without modifying the CATPart reference). A new instantiated CATPart is created using the Copy / Paste As Result With Link option, and an 'Associated Part' node containing the copied geometries is added under the active assembly. The copied geometries are associative to their reference CATParts.





PartBodies of Rings are instantiated with default PartBody color.

Student Notes:

More About Associativity (1/2)

You can use the 'Associativity' command for CATPart with multiple instances, by which a separate PartBody for each instance will be created. However these PartBodies will have only one CATPart as reference. If you make any change in the reference CATPart, it will be reflected in all the instances.

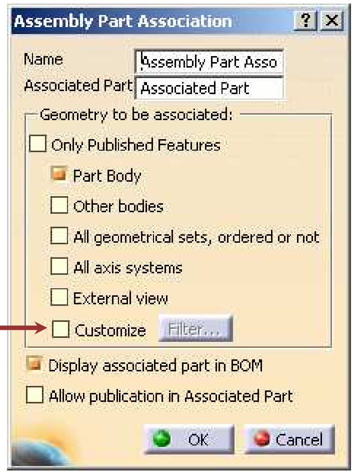
You can restore the deleted 'Associated Part' node by updating the assembly.

For creating an Associated Part using 'Associativity', the instances and PartBodies must be in the active state.

While creating an Associated Part, you can select the following features of the geometry to be associated:

- Part Body
- Other bodies
- All Geometrical Sets and Ordered Geometrical Sets
- All axis systems
- External view

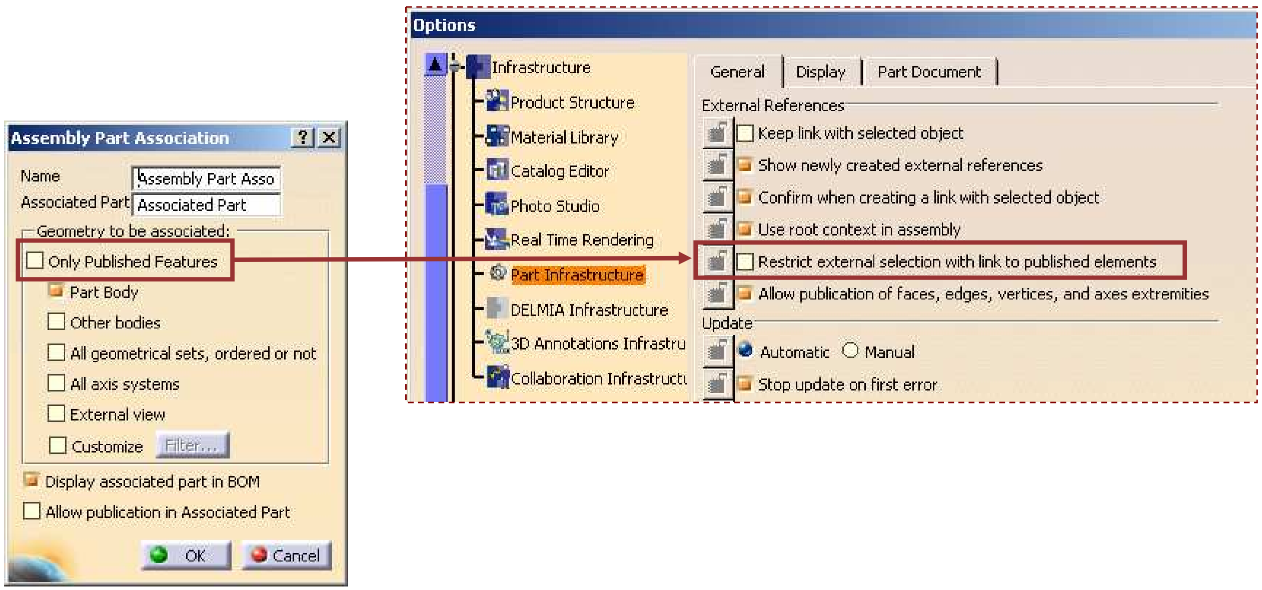
You can use Customize option to precisely select the bodies to copy for each selected part.



Student Notes:

More About Associativity (2/2)

When you select the 'Restrict external selection with link to published elements' check box in Tools > Options, it automatically checks the 'Only published Features' checkbox and deactivates it in the Assembly Part Association dialog box.

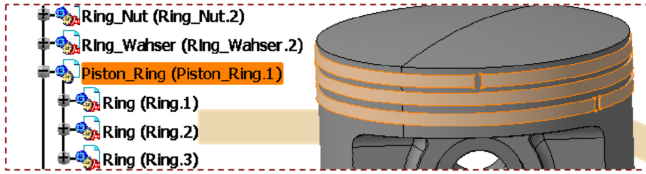


- **'Allow publication in Associated Part' check box, if checked then the publication from the reference part will be copied to the associated part.**

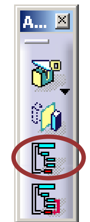
Student Notes:

How to Create an Associative Part from a CATProduct (1/2)

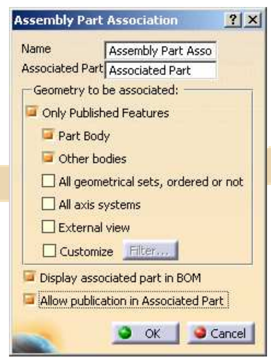
1 Double-click the sub-assembly to activate it.



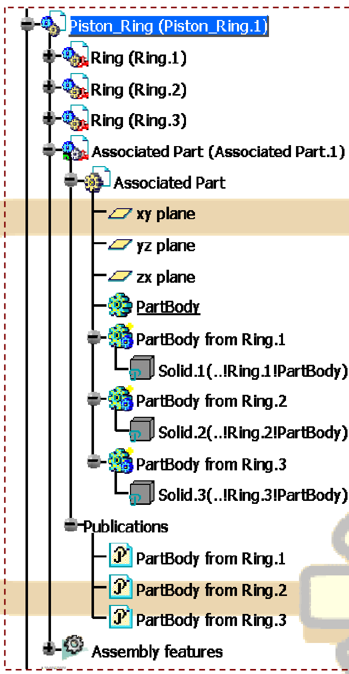
2 Select the Associativity icon.

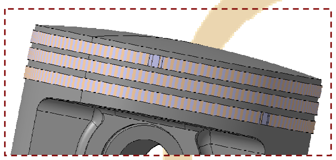


3 Select the options as per the dialog box and click OK.

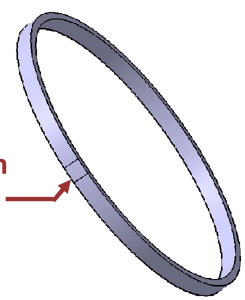


4 A new Associated part will be created with the publications under the product node.





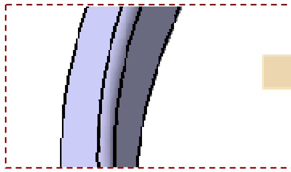
5 Open the Associated Part in the New window and fill the open end of each ring with Pad.



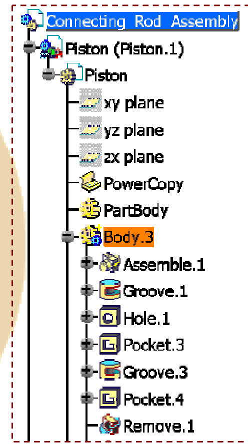
Student Notes:

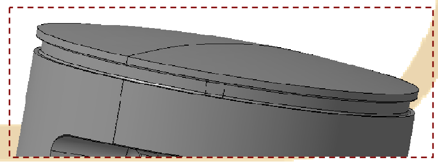
How to Create an Associative Part from a CATProduct (2/2)

6 Modify the associated part by providing a fillet of 0.5mm radius to the inner corners and save it.

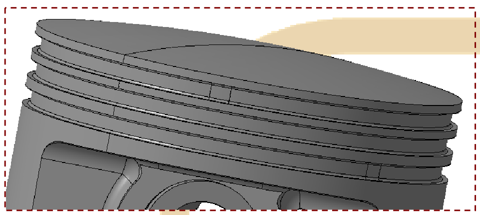


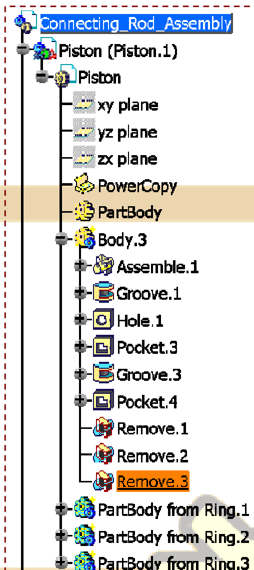
7 Double-click the Body from which the Associated part is to be removed. Remove the Associated PartBody from the Main Body. Make sure that Keep link with selected object option is active.



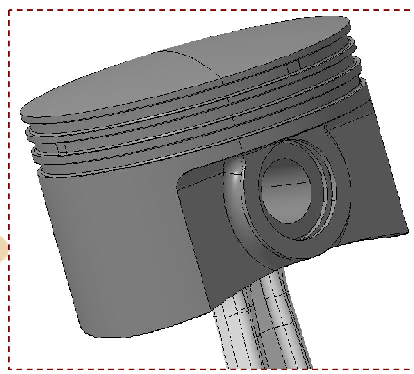


8 Similarly remove other associated PartBodies from the Main Body.





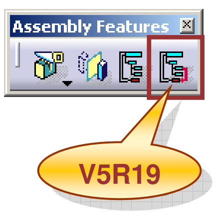
9 Now after hiding the Associated Part and Rings, you will see that the grooves are created on the Piston.

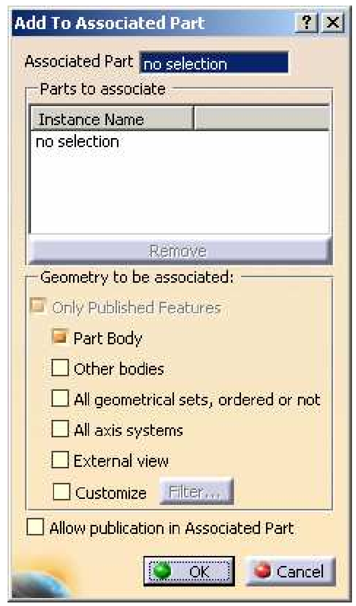


Student Notes:

New Command: Add to Associated Part

You will learn how to create and modify an associative CATPart without modifying its reference CATPart.



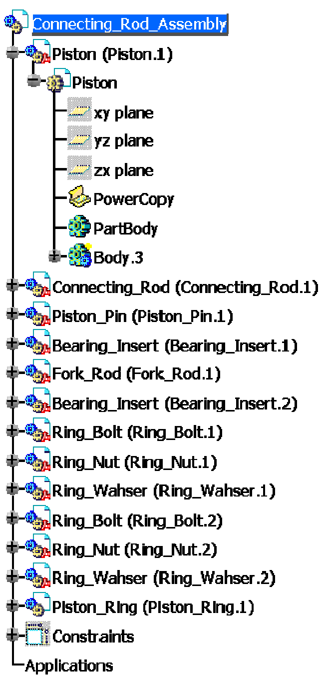


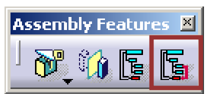
Student Notes:

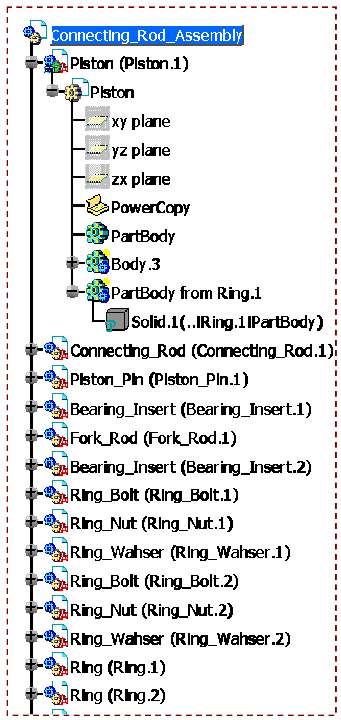
About ‘Add to Associated Part’

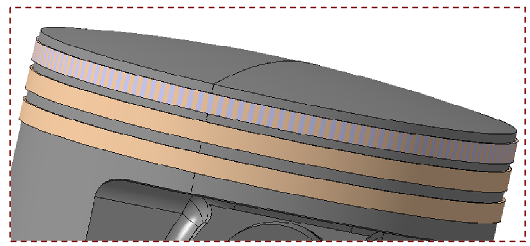
You can modify the CATPart geometry in the assembly context using the ‘Add Associated Part’ command. You can add geometries from existing/new source parts in the product into the associated Part. The geometries will be associative with their reference CATParts.

In the case of existing source parts, if the bodies/geometries of these parts already exist in the associated part, then you cannot add them to it again.







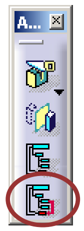


In this example using ‘Add to Associated Part’, geometries from Ring.1 are copied and added to the Piston.

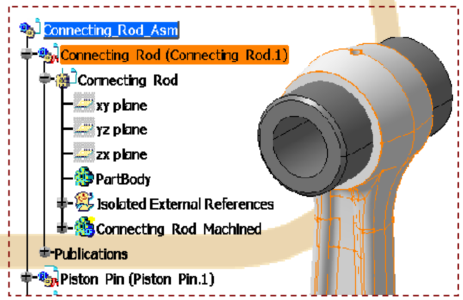
Student Notes:

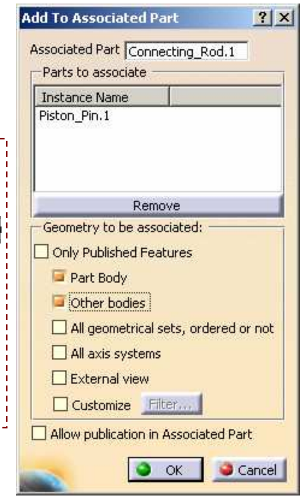
How to Add an Associated Part (1/2)

1 Click the 'Add to Associated Part' icon.

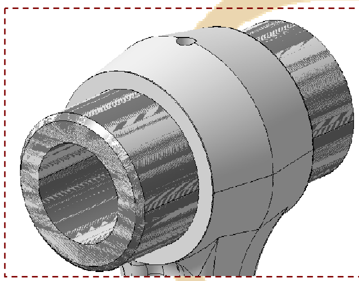


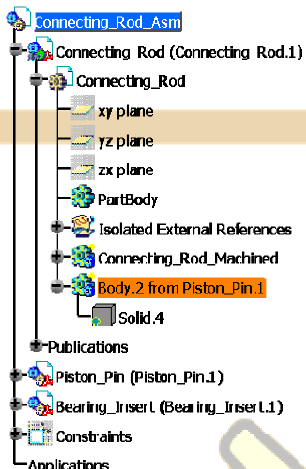
2 Select the Associated Part (The part in which other part is to be added) and the Part to be associated (which is added to the part) and select other options.

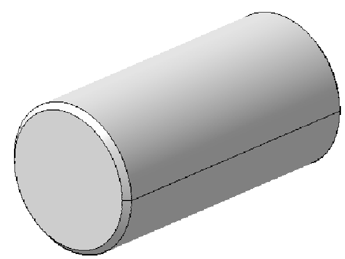




3 Double-click Piston_Pin.1 Body created under the Associated Part.







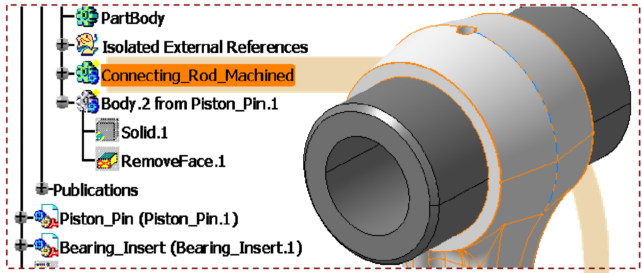
4 Now edit the associated Piston_Pin Body by filling the hole in it.

Student Notes:

How to Add an Associated Part (2/2)

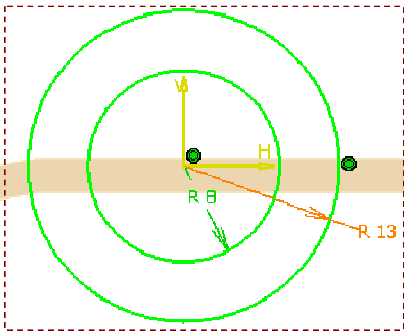
5

Now Double-click Connecting_Rod_Machined and remove 'Body.2 from Piston_Pin.1' from it.



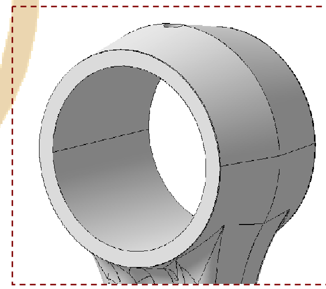
7

Open Piston_Pin in new window and change the radius of ring from 12.5mm to 13mm



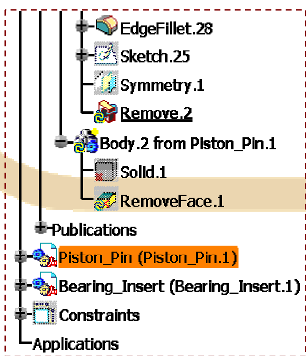
6

Hide the Piston Pin and you will see hole as per size of the Piston_Pin.



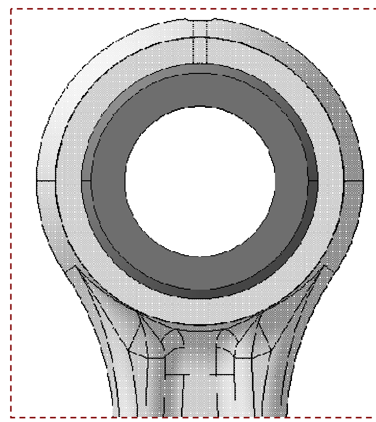
8

Update the assembly.



9

After updating you will see that the hole gets modified as per the Pin radius.



Student Notes:

Associativity and Add to Associated Part

Recap Exercise

20 min

In this step you will create pockets on the Cellphone cover for placement of the Buttons using Associativity and Add to Associated Part command. This enables you to use buttons for pocket creation. You can maintain associativity between the Buttons and their locations without modifying them in the assembly.

By the end of this exercise you will be able to

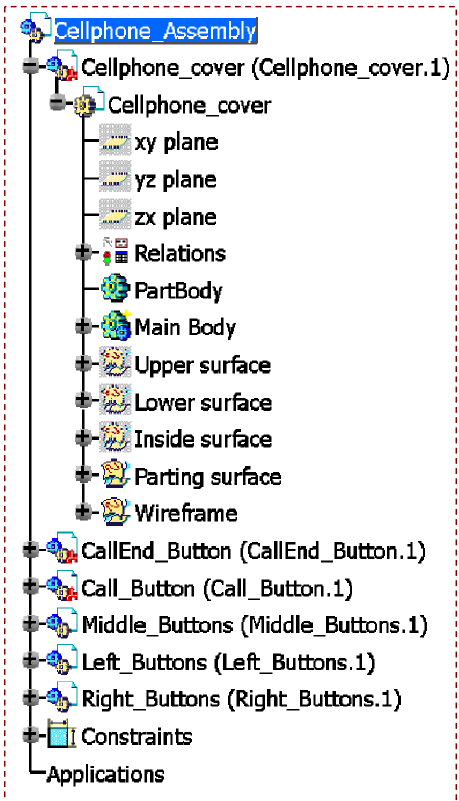
- Create the Associativity feature.
- Add a CATPart to the Associated Part.

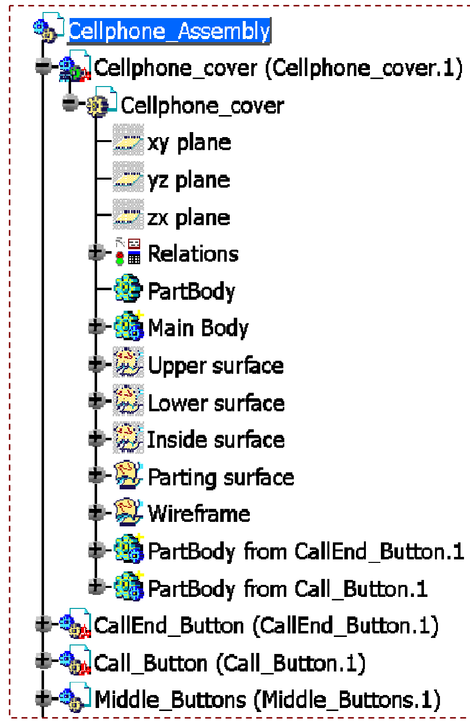
Student Notes:

Do It Yourself (1/4)

Part used: Cellphone_Assembly.CATProduct

- **Create the pockets in the mobile cover using the Blue Buttons present in the assembly**
- **Using the assembly feature Add to Associated Part add CallEnd_Button and Call_Button into the Cellphone_cover.**

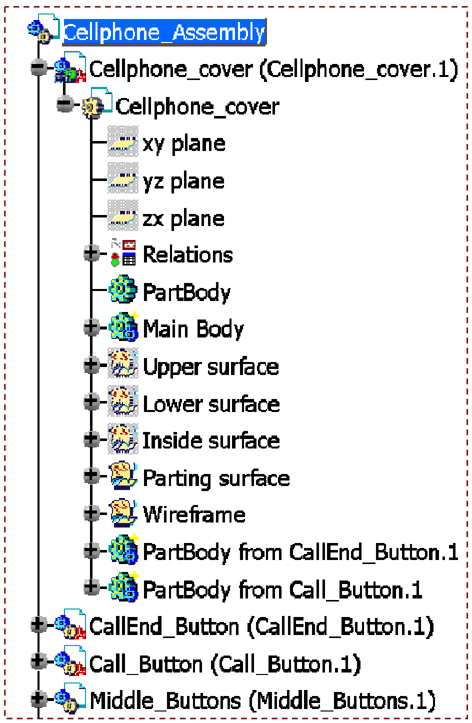


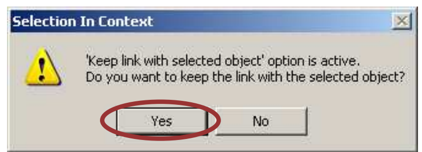


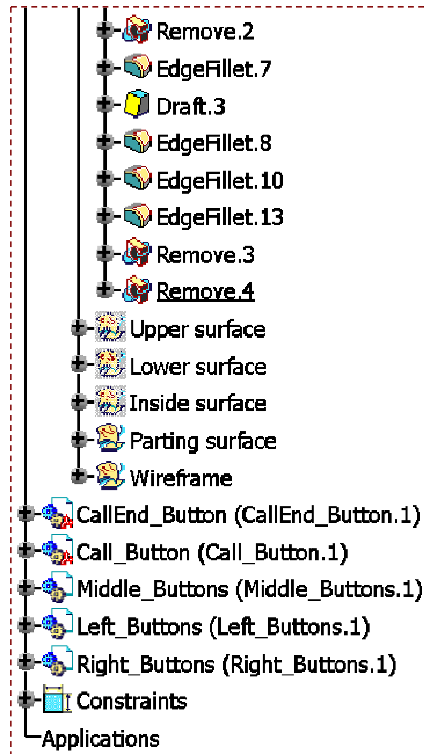
Student Notes:

Do It Yourself (2/4)

- **Remove the copied bodies from the Cellphone_Cover using the Boolean operation. Make sure that ‘Keep link with selected object’ option is checked.**



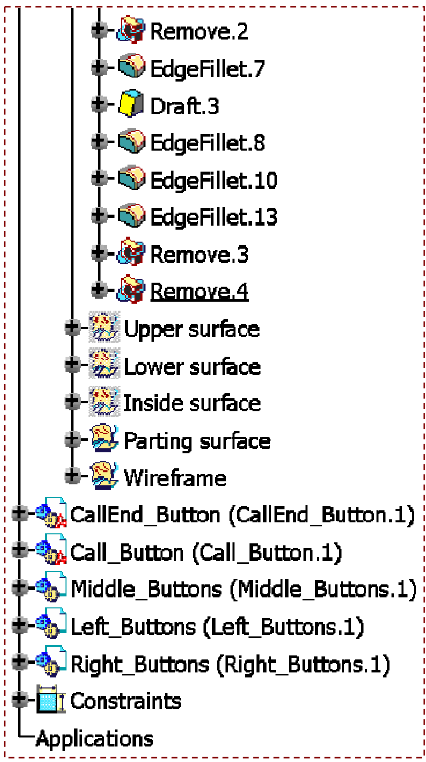


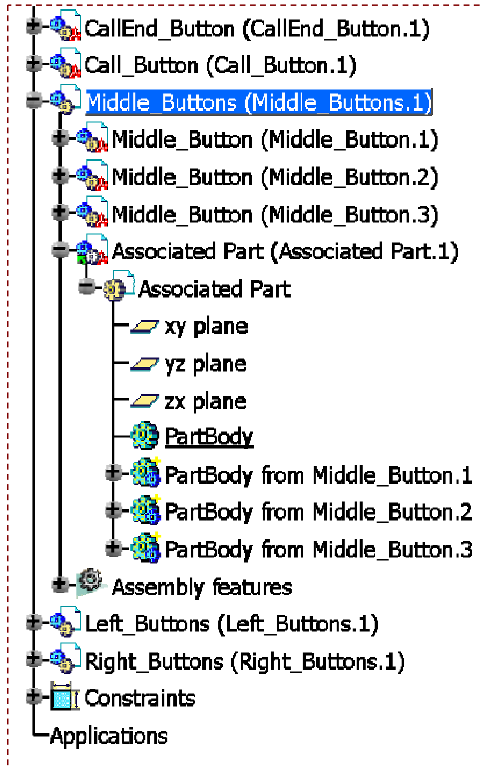


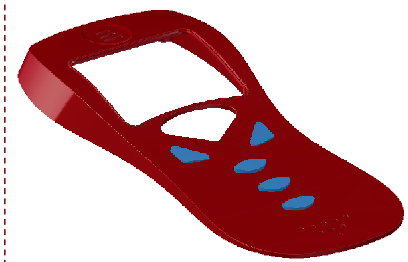
Student Notes:

Do It Yourself (3/4)

- **Using the assembly feature Associativity, create three PartBodies of the Middle_Button under the Middle_Button Product node.**



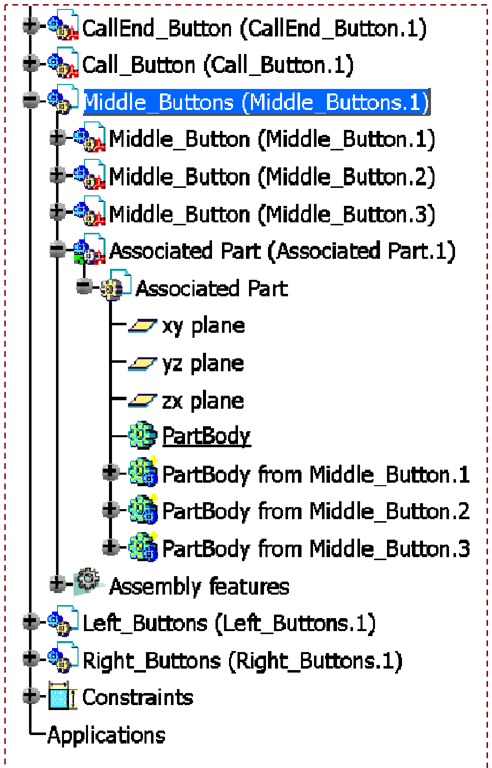


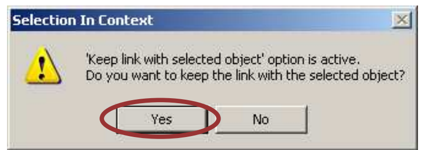


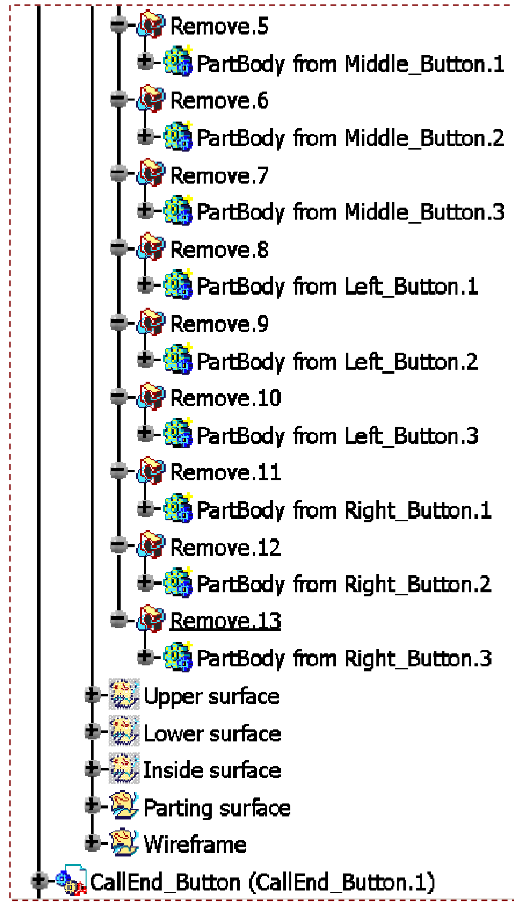
Student Notes:

Do It Yourself (4/4)

- **Remove these three PartBodies of Middle_Button from the Cellphone_Cover using the Boolean operation. Make sure that 'Keep link with selected object' option is checked. Repeat the same procedure for creating pockets of Left_Button and Right_Button.**







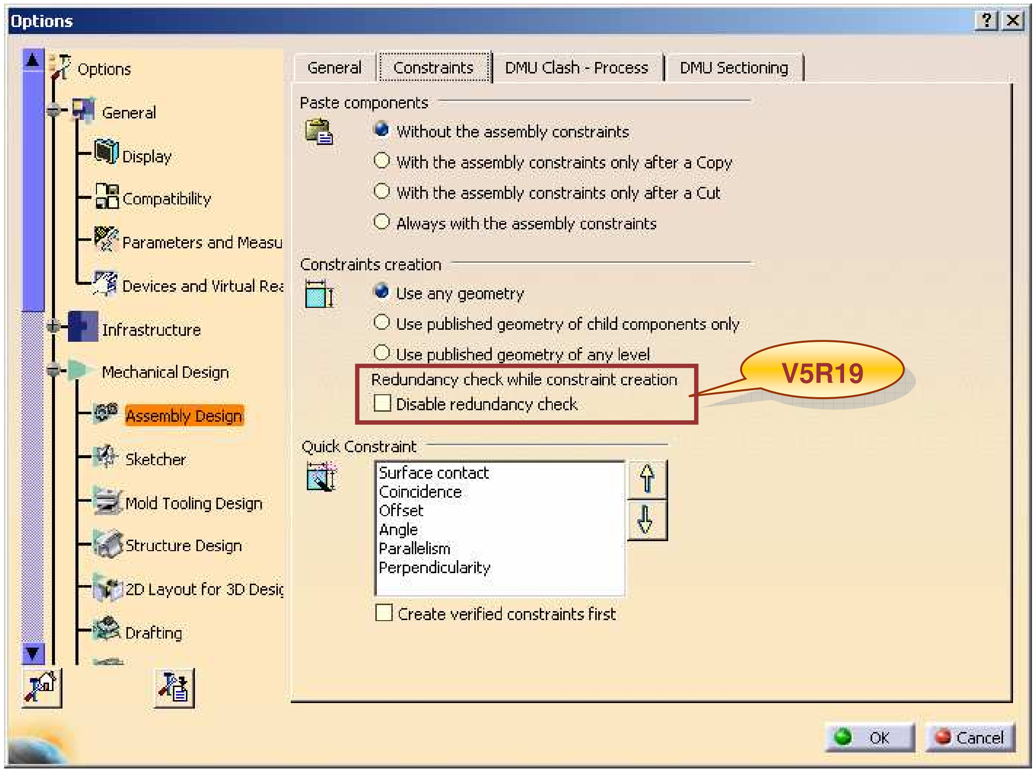
End Part: Cellphone_Assembly_Completed.CATProduct

STUDENT GUIDE

Student Notes:

Enhancement in the Constraint Creation

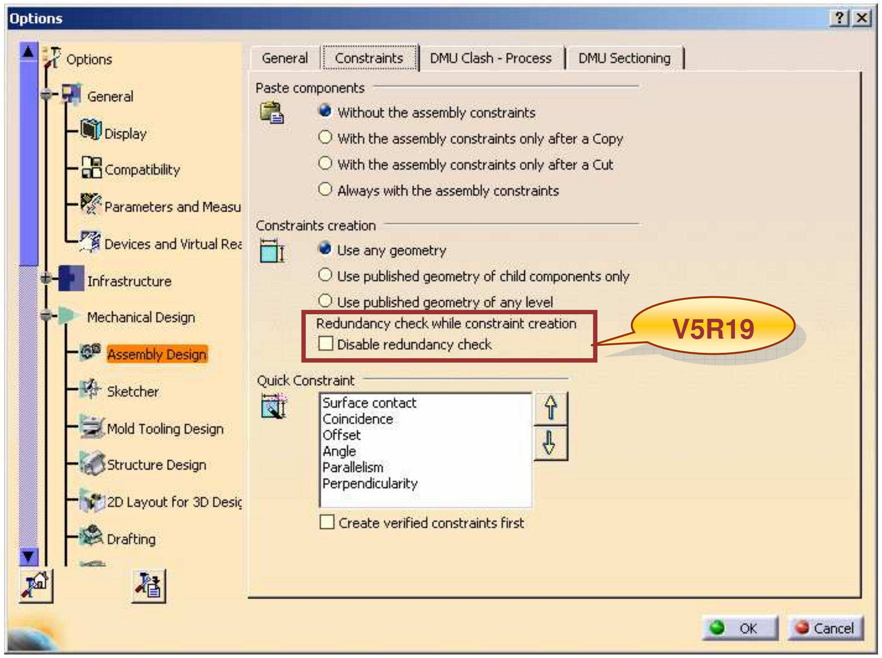
You will learn about new option Redundancy Constraint Check in constraint creation.



Student Notes:

About Redundancy Constraint Check

In CATIA while you create the constraints the Redundancy constraint check is performed. This check affects directly the amount of time required for constraint creation. This happens particularly when you work with huge assemblies with hundreds of parts loaded together with many constraints.



Now with this new enhancement a setting has been provided at Tools > Options > Mechanical Design > Assembly Design > Constraints with which you can enable or disable the redundancy check while the constraints are created. Hence You can avoid the “redundancy checks” .

Student Notes:

Generative Shape Design Updates

You will learn about the following enhanced functionalities in the Generative Shape Design workbench:

- What's New in Generative Shape Design Workbench
- Enhancement in the Sweep
- Enhancement in Points and Planes Repetition
- Enhancement in the Planes Between

Student Notes:

What's New in CATIA Generative Shape Design Workbench

The list of enhanced functionalities is given below:

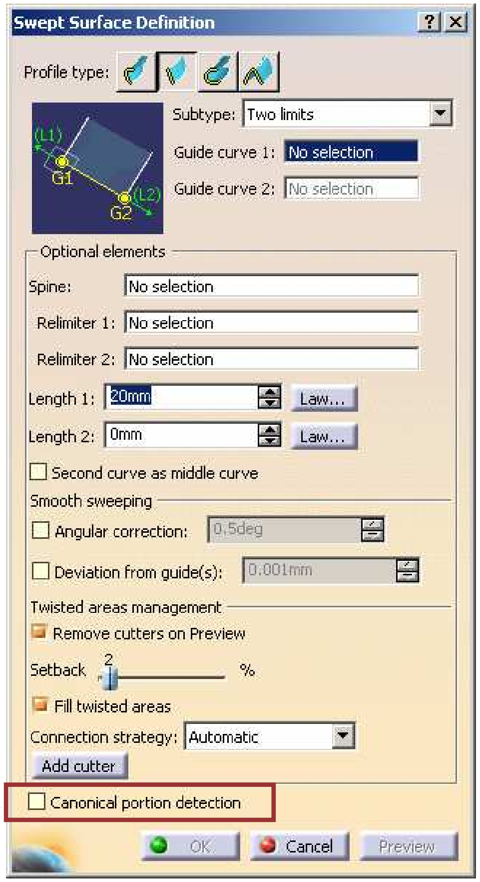
- Enhancement in the Sweep
  - This highlight will enable you to allow canonical surface detection in sweeps like linear, circular or conical.
- Enhancement in the Points and Planes Repetition
  - This highlight will enable you to select any type of point (e.g. point created on the curve with datum mode, intersect, extremum, projection, and the points which are not lying on the curve) as a reference point to create multiple points in the Points and Planes repetition command.
  - A Preview button is added in the dialog box to view the result before exiting the dialog box.
- Enhancement in the Planes Between
  - A Preview button is added in the dialog box to view the result before exiting the dialog box.

Student Notes:

Enhancement in the Sweep

You will learn about the canonical portion detection in the sweep command.

V5R19



Student Notes:

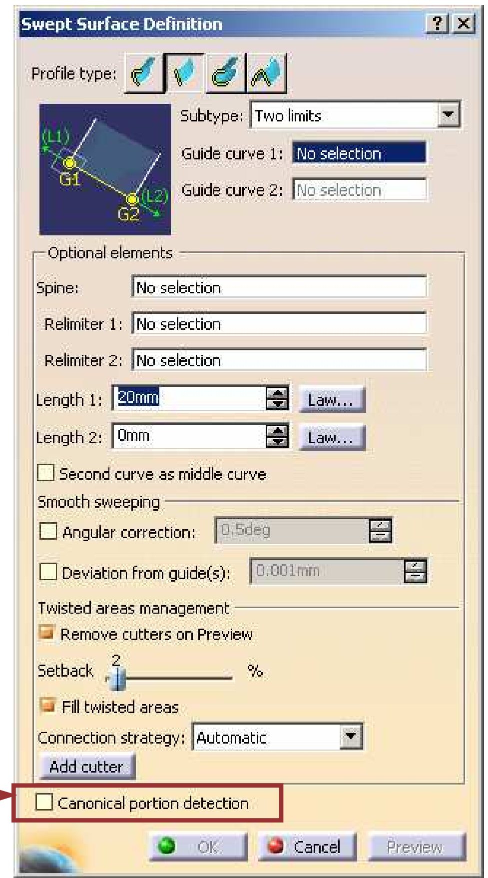
About Canonical Portion Detection in Sweep (1/2)

A check button called 'Canonical portion detection' is added in the sweep dialog box (for the Line, Circle and Conic Sweep).

This highlight will provide you with an option where you can allow the canonical surface detection of sweeps like linear, circular, or conical.

Canonical surface: It is a surface, which is defined in a simplest form, without loosing its original geometry. Sphere and cylinder are examples of canonical surface.

In V5R18, for 'Explicit Sweep', canonical surface detection is always ON, by default.

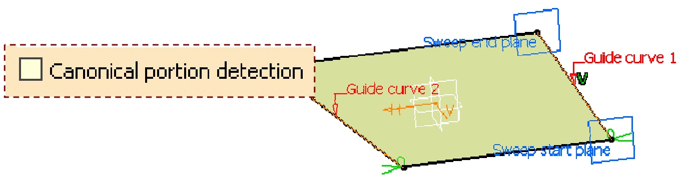


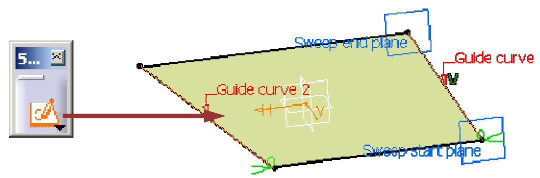
Student Notes:

About Canonical Portion Detection in Sweep (2/2)

Now you will see the use of this option.

If you create a sweep surface without selecting the Canonical portion detection option, then you will not be able to use the surface as a sketch support.

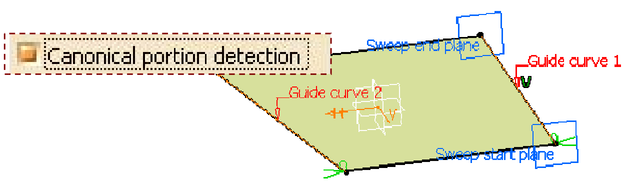


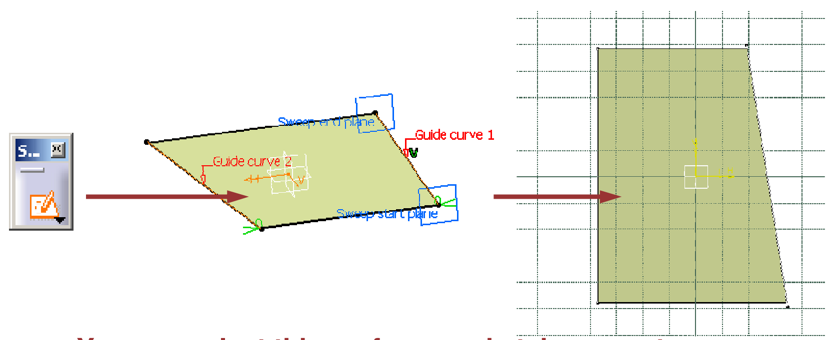


Sweep surface created without selecting the option

You can not select this surface as sketch support

If you check the Canonical portion detection option, you can use the surface as a sketch support.





Surface created using the option

You can select this surface as sketch support

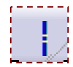
Student Notes:

Do It Yourself (1/2)

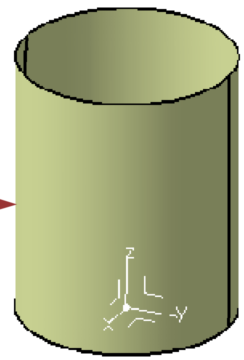
Data to be used: Canonical_Portion_Detection_Sweep. CATPart

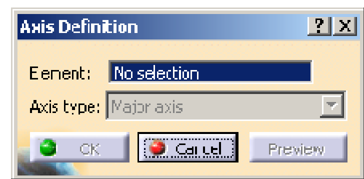
In this exercise, you will detect whether the surface is canonical or not.

- **Click the Axis Icon.**
- **Select Sweep.1 to compute whether it is a canonical surface or not.**
- **Get an error of invalid element.**
- **Cancel the command.**



Sweep.1





It shows that the surface is not canonical.

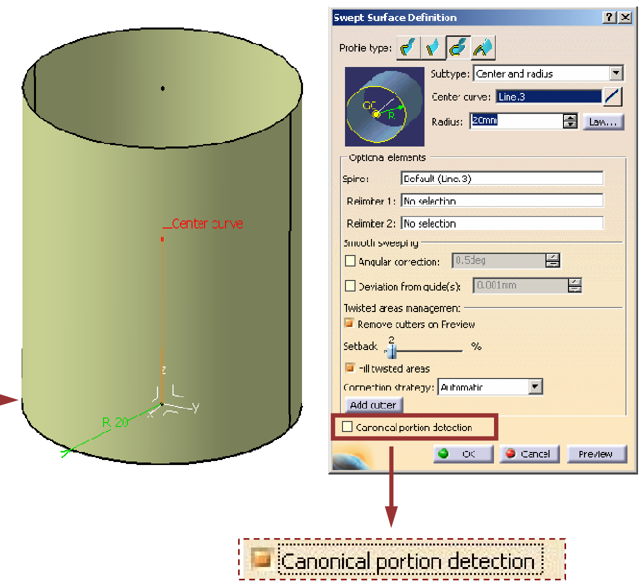
Student Notes:

Do It Yourself (2/2)

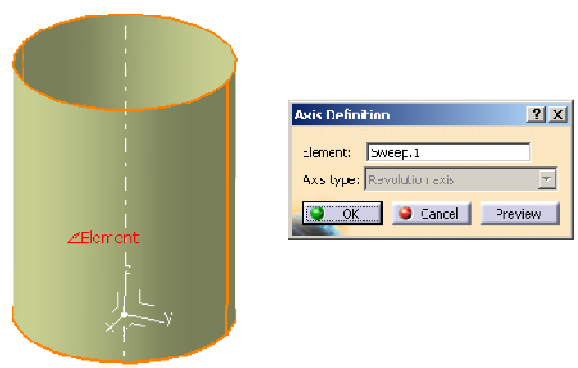
In this exercise, you will use the enhancement.

- **Edit the Sweep.1and turn on (or check) the ‘Canonical portion detection’ button.**
- **Create the axis by selecting the sweep surface.**

Sweep.1



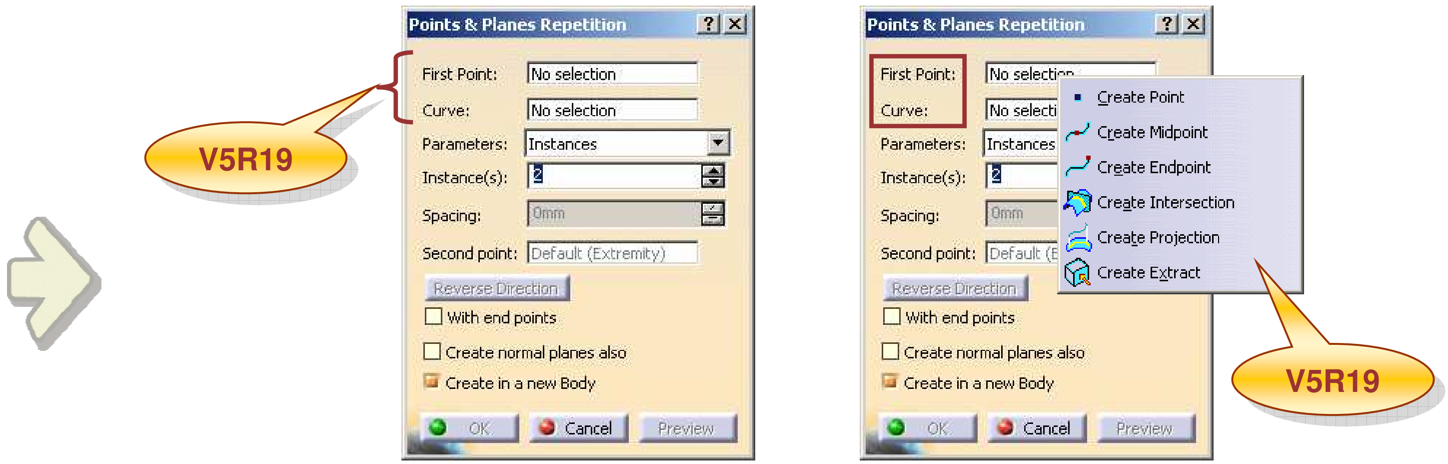
It shows that the surface is canonical.



Student Notes:

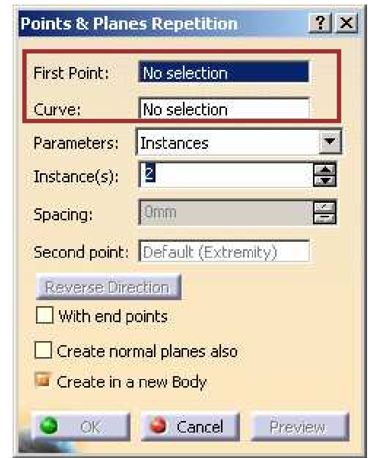
Enhancement in the Points and Planes Repetition

You will learn about the enhancement in the Points and Planes Repetition command.



About Enhancement in the Points and Planes Repetition (1/2)

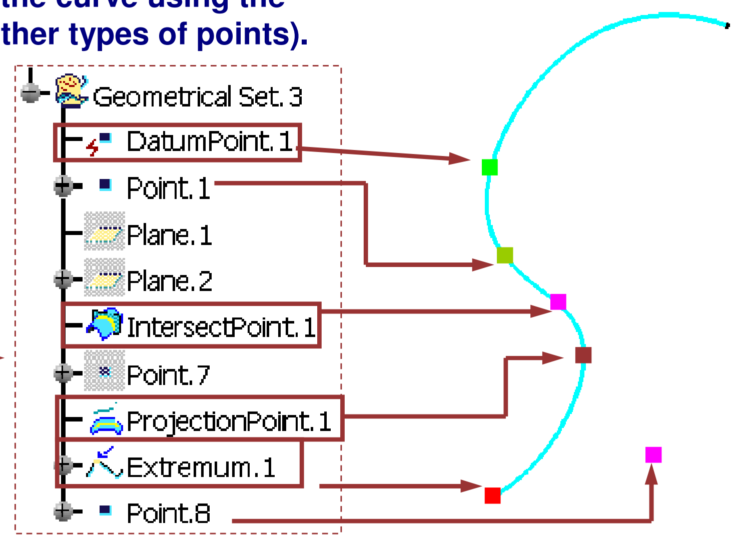
New input fields 'First Point' and 'Curve' with contextual menu are added in the Points and Planes Repetition command, to specify the first reference point and the support curve on which multiple points will be created.



This highlight adds capability of selecting any type of point (e.g. point created on the curve with datum mode, intersect, extremum, projection, and the points which are not lying on the curve) as a reference point to create multiple points in the Points and Planes repetition command.

In R18 you are able to select points created on the curve using the command Point with type 'On Curve' (but not other types of points).

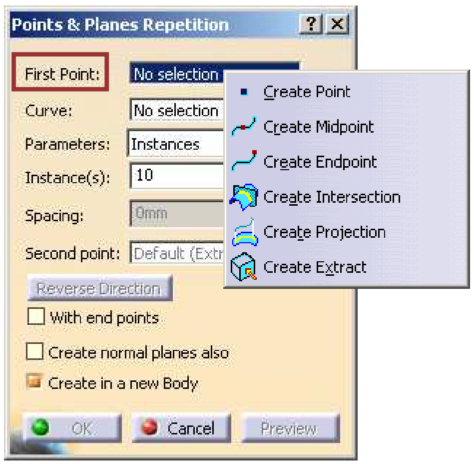
In R19, you can select all these types of points as a First point.

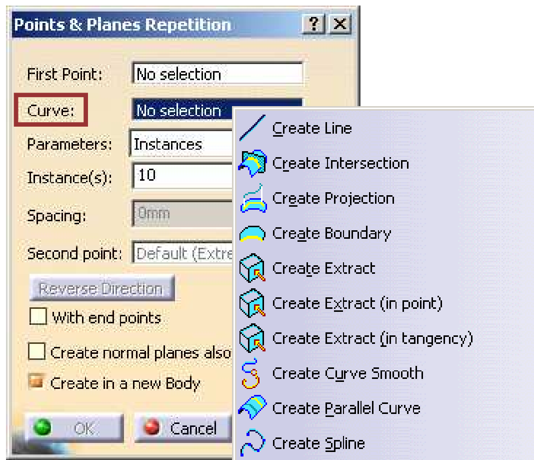


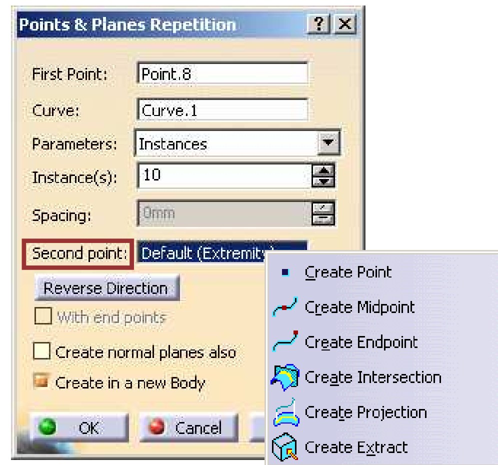
Student Notes:

About Enhancement in the Points and Planes Repetition (2/2)

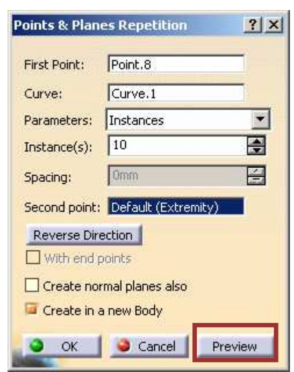
Contextual menu is provided for the first input in the 'First Point' field, second input in the 'Curve' field, and third input in the 'Second Point' field. Contextually created inputs will be aggregated under the first point which was created using this functionality.







A Preview button is added in the dialog box to view the result before exiting the dialog box.



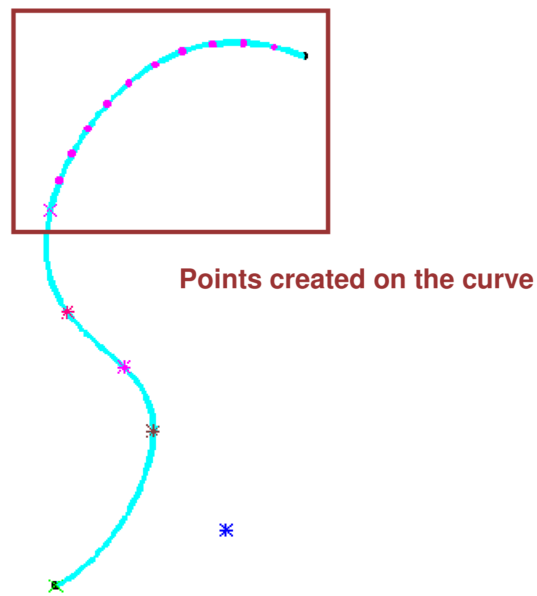
Student Notes:

Do It Yourself (1/3)

Data to be used: UMD_R19_Points_Planes_repetition.CATPart

You will create points on the curve using different types of reference points.

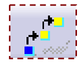
- **Click the Points and Planes Repetition icon.**
- **Select Datumpoint.1 as the First Point and Curve.1 as Curve.**
- **Set the instance value as 10 and see the result.**
- **Repeat the above steps for different points.**



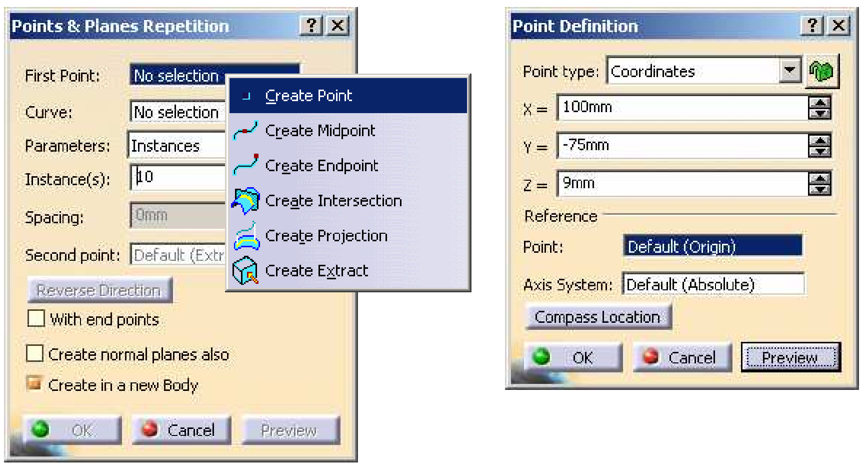
Student Notes:

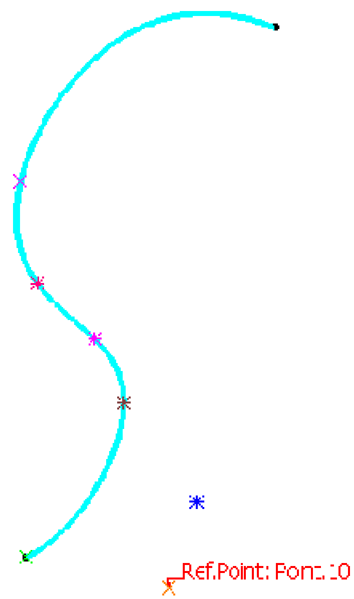
Do It Yourself (2/3)

- **Use the data from the previous slide.**
- **Click the Points and Planes Repetition icon.**



- **Right-click the 'First Point' field. Choose Create Point from the contextual menu and create a first reference point using the Coordinates type.**

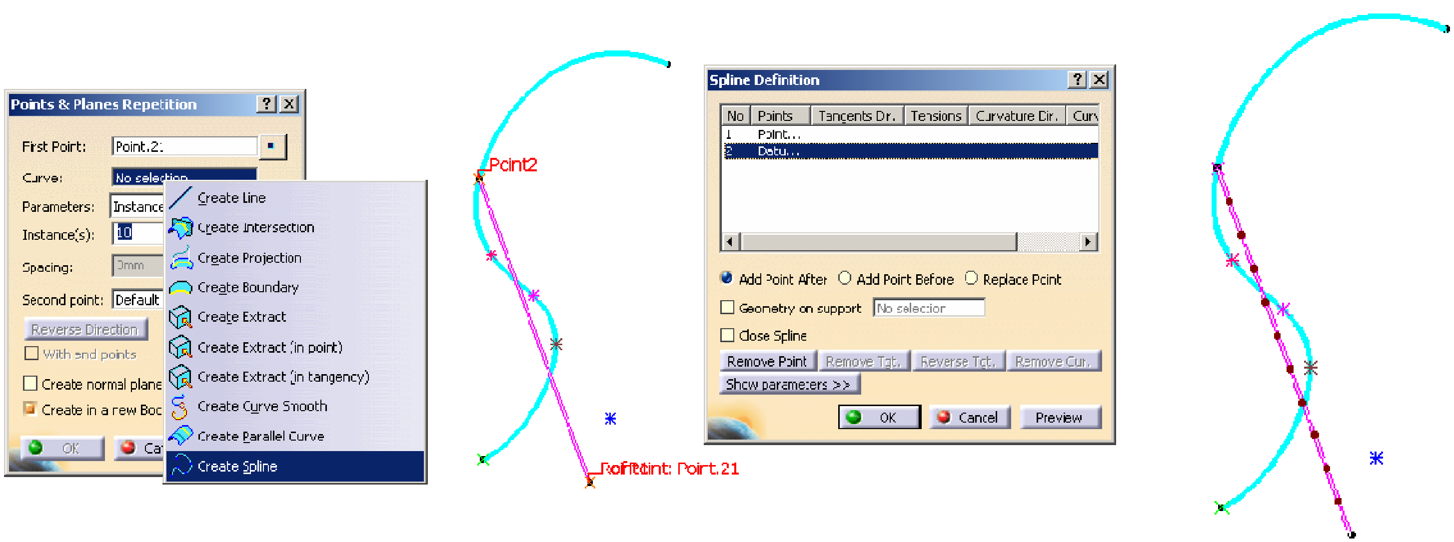




Student Notes:

Do It Yourself (3/3)

- **Right-click on the ‘Curve’ field. Choose Create Spline from the contextual menu and create Spline.1 using the contextually created point in the previous step, Point.1 and DatumPoint.1.**
- **Set the instance value as 10 and see the result.**



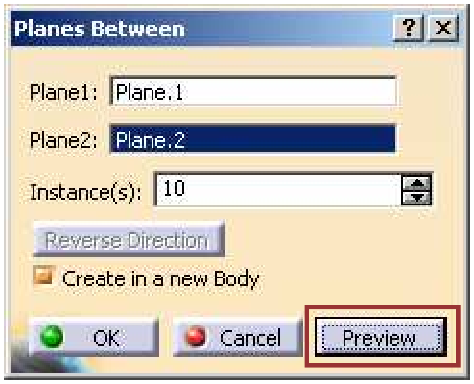
Student Notes:

Enhancement in the Planes Between

The highlight provides you the capability to preview the output for the Planes Between command, before clicking the OK button.

This highlight enables you to confirm the number of planes to be created. If the output obtained is contrary to the expectations, you can change the inputs and then create the plane again.

The Preview button is required for confirming the expected output and to have ease in performing the modifications.



STUDENT GUIDE

Student Notes:

Drafting Updates

You will learn about the following new and enhanced functionalities of the CATIA Drafting workbench:

- What's New in CATIA Drafting Workbench
- New Command: Advanced Bill of Material
- Enhancement in Rigid position Links
- Enhancement in Broken Constraints Visualization
- Enhancement in 3D Clipping

Student Notes:

What's New in CATIA Drafting Workbench

The list of enhanced functionalities is given below.

- **New Command: Advanced Bill of Material (BOM):**

  This highlight enhances capability and productivity of the BOM by enabling you to perform variety of tasks such as customization of BOM, editing of BOM table etc.

- **Enhancements in Positional Links:**

  With this highlight it is now possible to create a rigid positional link. The rigid character of the positional link makes the move relatively to the reference impossible.

- **Enhancement in Constraints:**

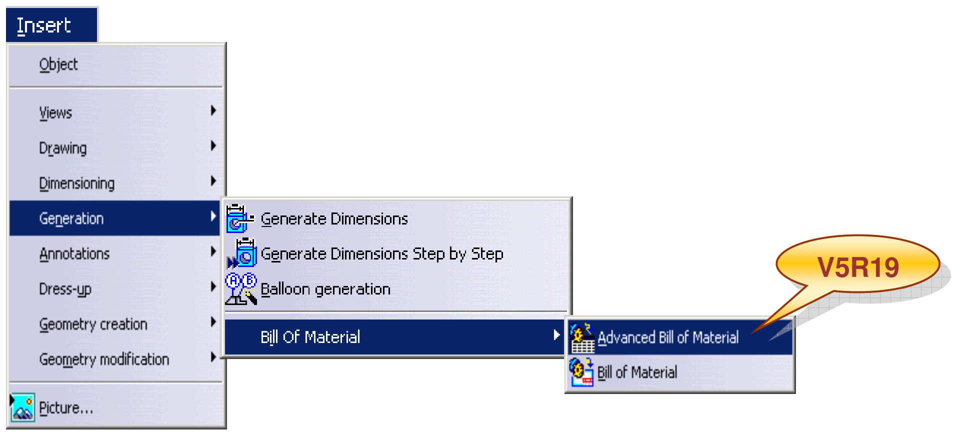
  This highlight enables you to see the broken constraints as well as normal (non broken) constraints. You will be able to visualize the broken constraints and be able to quickly edit those thus enhancing the productivity.

- **Enhancement in Clipping:**

  In addition to the existing methods of defining the 3D clipping object (manipulators being moved manually, spinners, and edit boxes), you will be able to select a point or a line (in 2D and 3D view) or a planar face / plane / edge in 3D view in order to define the position of the current manipulator.

New Command: Advanced Bill of Material

You will learn how to insert the Advanced Bill of Material



Student Notes:

Student Notes:

Why Advanced Bill of Material

Bill of material is a critical part of the design and manufacture of any product.

A Bill of material can define products as they are designed, manufactured, ordered, or maintained. A Bill of Material (BOM) lists all the items that go into a finished good or subassembly. The Bill of material is the base part of any production process. If you don't have a well defined list of ingredients you can't make the part. Of course in the real world a BOM is much more than a listing of the parts which make up the items we manufacture.

The Advanced BOM provides you the following advantages:

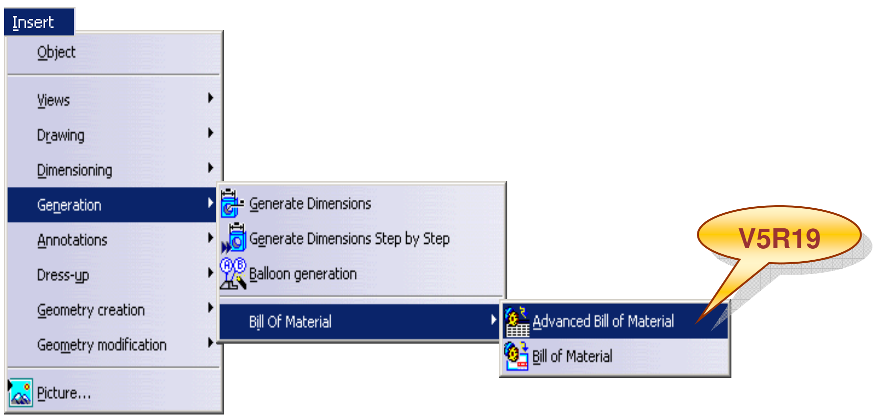
- You can insert the BOM in a particular view or sheet as separate sheets are created
- You can insert BOM without opening the assembly file.
- You can modify the contents of the table using the properties of the table.
- You can customize the BOM table.
- You can invert or split rows at the time of BOM insertion.

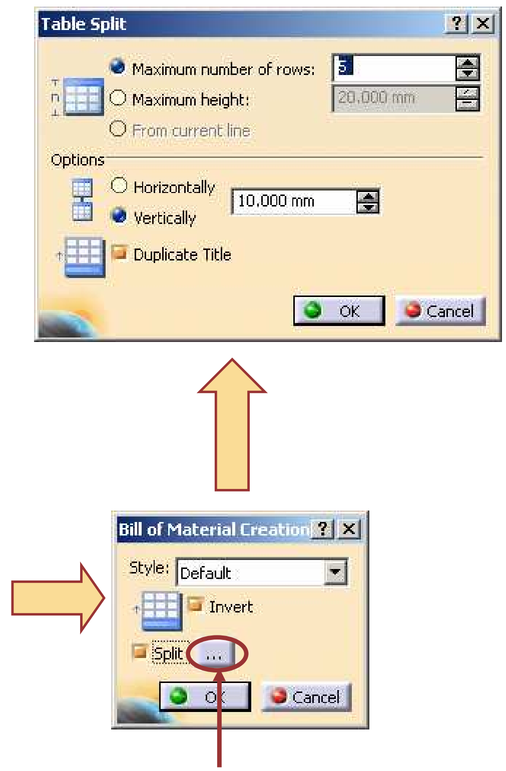
You can use the Advanced Bill of Material to modify the contents of the Bill of Material and also add additional contents in it such as Material, Definition, Source, Weight etc.

Student Notes:

More About Advanced Bill of Material (1/3)

A new command, Advanced Bill of Material is added. The Bill of Material command is also available in the same menu. You can opt for any one of the two commands.





In the Bill of Material creation dialog box you have two options:

- **Invert: Select this option if you want the table to be read from bottom to top (i.e. column headers will be at the bottom and numbering order will be reversed).**
- **Split: Select this option to split the BOM table into several tables.**

After clicking on the icon you will see the Table split dialog box.

Student Notes:

More About Advanced Bill of Material (2/3)

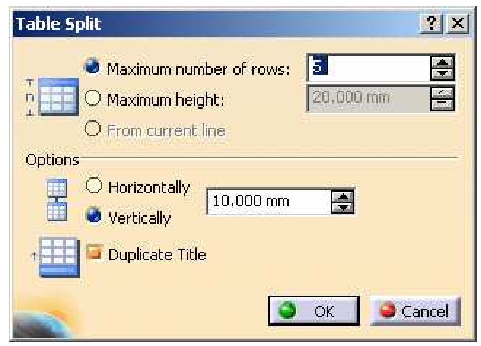
For splitting the table there are some options provided such as,

- **Maximum number of rows: You can specify the maximum number of rows that each table should contain after splitting, irrespective of the total number of rows.**
- **Maximum height: You can specify the maximum height of each table.**
- **Horizontally: You can provide option of splitting horizontally.**
- **Vertically: You can split the table vertically.**
- **Duplicate Title: You can also repeat the Title in each table.**

1	Jig	CATPart	-	-
1	Drill Diam 8	CATPart	Super Drill	-
1	Housing Right	CATPart	-	-
1	Housing Left	CATPart	-	-
Quantity	Part Number	Type	Nomenclature	Revision

1	User Finger Envelop	CATPart	-	-
4	Screw	CATPart	-	-
1	TopSwitch	CATPart	-	-
1	Transmission	CATPart	-	-
Quantity	Part Number	Type	Nomenclature	Revision

1	Round Motor	CATPart	MEGA POWER	-
1	BackSwitch	CATPart	-	-
1	VSR Switch	CATPart	-	-
1	Battery	CATPart	BAT-XX-1209	-
Quantity	Part Number	Type	Nomenclature	Revision



This is what we can get with Advanced Bill of Material using the enhanced functionalities.

You can split the BOM table after generating it.

Student Notes:

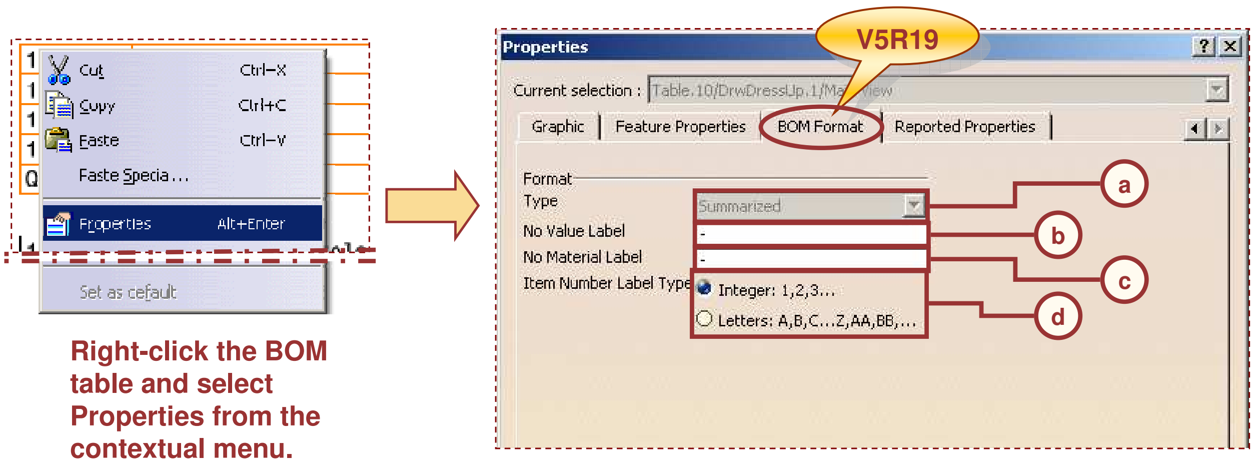
More About Advanced Bill of Material (3/3)

- You can insert the Bill of Material to the single part also.
- You can edit the BOM table by right-clicking on the table and select Table.X object > Edit BOM from the contextual menu.
- You can manually add columns. The modifications made in them are updated and saved.
- The Bill of Material update status is not taken into account in the drawing update status. So even if the Bill of Material is not up-to-date, the Update icon is not activated. You must use force update command to update the Bill of Material.

Student Notes:

Customizing the Bill of Material (1/2)

You can customize the Bill of Material suitable to your requirement of detailing, designing, and manufacturing.



Right-click the BOM table and select Properties from the contextual menu.

In the Properties dialog box, the BOM Format tab has the following options:

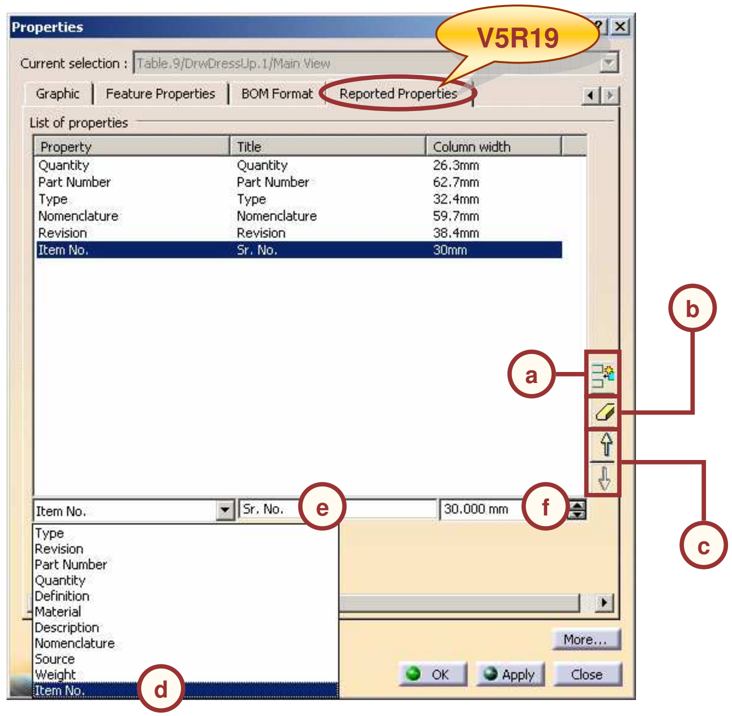
a. **Type: The administrator can change the settings in Tools > Standards > Drafting > [standard name] > Styles > Bill of Material. This option is unavailable by default.**
b. **No Value Label: You can set a default label when no value is set in the appropriate property field. The default label is "-".**
c. **No Material Label: You can set a default label that will be displayed when no material is specified for the part.**
d. **Item Number Label Type: You can label the components in an assembly alphabetically or numerically by selecting the Integer or the Letters option.**

Student Notes:

Customizing the Bill of Material (2/2)

You can add different Titles for one property suitable to your Bill of Material.

Right-click BOM table and select Properties from the contextual menu.



In the Properties dialog box, Reported Properties tab has the following options:

a. **New: To add an extra column to the table.**

b. **Delete: To delete the unwanted columns from the table.**

c. **Move up and Move down arrows: To rearrange the order of columns from left to right in the table.**

d. **Drop down list: You can add new properties to the column from the drop down list.**

e. **Title: You can edit the Title of the added property.**

f. **Column width: You can specify the width of the column that is added.**

Student Notes:

Editing the Bill of Material

You can edit the BOM table which has been inserted in the assembly Drawing.

12	Jig	-	1	CATPart	-	-	0kg
11	Drill Diam 8	Super Drill	1	CATPart	-	Steel	0.024kg
10	Housing Right	-	1	CATPart	-	-	0kg
9	Housing Left	-	1	CATPart	-	-	0kg
Sr. No.	Part Number	Nomenclature	Quantity	Type	Revision	Material	Weight

12	Jig	-	1	CATPart	-	-	0kg
11	-	-	-	-	-	-	0kg
10	Housing Right	-	1	CATPart	-	-	0kg
9	Housing Left	-	1	CATPart	-	-	0kg
Sr. No.	Part Number	Nomenclature	Quantity	Type	Revision	Material	Weight

Effect of removing the component Drill_Diam_8 from the assembly.

12	Jig	-	1	CATPart	-	-	0kg
11	Drill Diam 8	Super Drill	1	CATPart	-	Steel	0.024kg
10	Housing Right	-	1	CATPart	-	-	0kg
9	Housing Left	-	1	CATPart	-	-	0kg
Sr. No.	Part Number	Nomenclature	Quantity	Type	Revision	Material	Weight

1	Part1	CATPart	-	-
Quantity	Part Number	Type	Nomenclature	Revision

1	Jig	CATPart	-	-
1	Drill Diam 8	CATPart	Super Drill	-
1	Housing Right	CATPart	-	-
1	Housing Left	CATPart	-	-
Quantity	Part Number	Type	Nomenclature	Revision

Effect of adding the new part “Part 1” to the assembly.

12	Jig	-	1	CATPart	-	-	0kg
11	Drill Diam 8	Super Drill	1	CATPart	-	Steel	0.024kg
10	Housing Right	-	1	CATPart	-	-	0kg
9	Housing Left	-	1	CATPart	-	-	0kg
Sr. No.	Part Number	Nomenclature	Quantity	Type	Revision	Material	Weight

12	Jig	-	1	CATPart	-	-	0kg
11	Drill Diam 9	Super Drill	1	CATPart	-	Steel	0.034kg
10	Housing Right	-	1	CATPart	-	-	0kg
9	Housing Left	-	1	CATPart	-	-	0kg
Sr. No.	Part Number	Nomenclature	Quantity	Type	Revision	Material	Weight

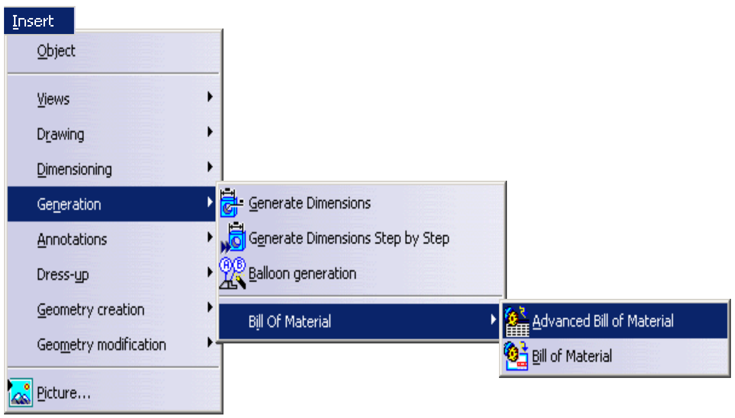
Effect of replacing the Drill_Diam 8 by Drill_Diam 9.

Student Notes:

Do It Yourself (1/2)

Part used: Drill_Machine.CATDrawing

- Select Insert > Bill of Material > Advanced Bill of Material.
- Select the options for inverting and splitting the BOM table.



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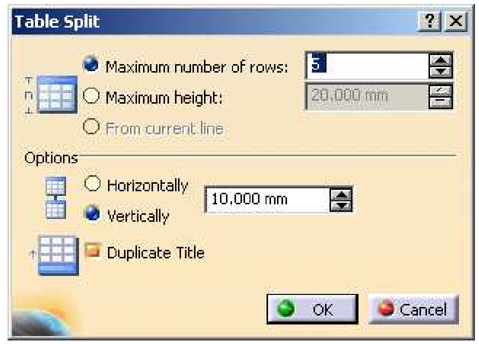
Student Notes:

Do It Yourself (2/2)

- **Set the values in the Table split dialog box.**
- **Click OK.**



- **Select the view or product and click the location of placement of the BOM table.**
- **Finally you will get the format of the required table.**

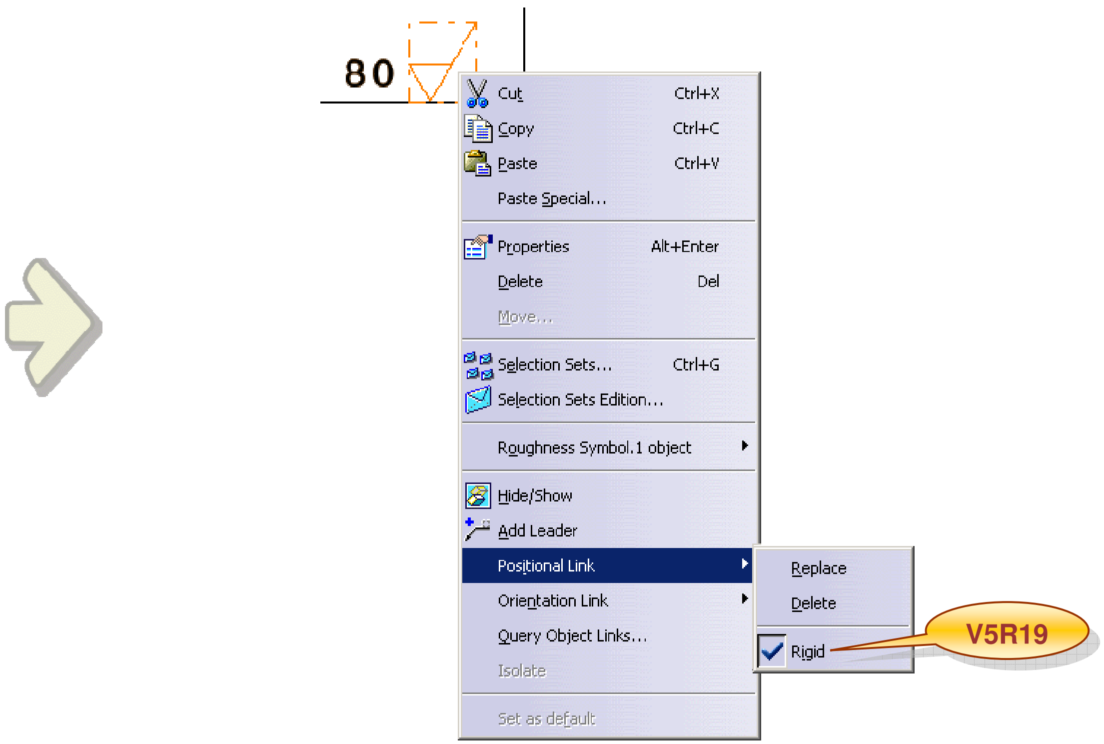
1	Jig	CATPart	-	-
1	Drill Diam 9	CATPart	Super Drill	-
1	Housing Right	CATPart	-	-
1	Housing Left	CATPart	-	-
1	User Finger Envelop	CATPart	-	-
4	Screw	CATPart	-	-
1	TopSwitch	CATPart	-	-
1	Transmission	CATPart	-	-
1	Round Motor	CATPart	MEGA POWER	-
1	BackSwitch	CATPart	-	-
1	VSR Switch	CATPart	-	-
1	Battery	CATPart	BAT-XX-1209	-
Quantity	Part Number	Type	Nomenclature	Revision

End Part: Drill_Machine_BOM.CATDrawing

Student Notes:

Enhancement in Rigid Positional Link for Annotations

You will learn how to create the rigid position link for Annotations.

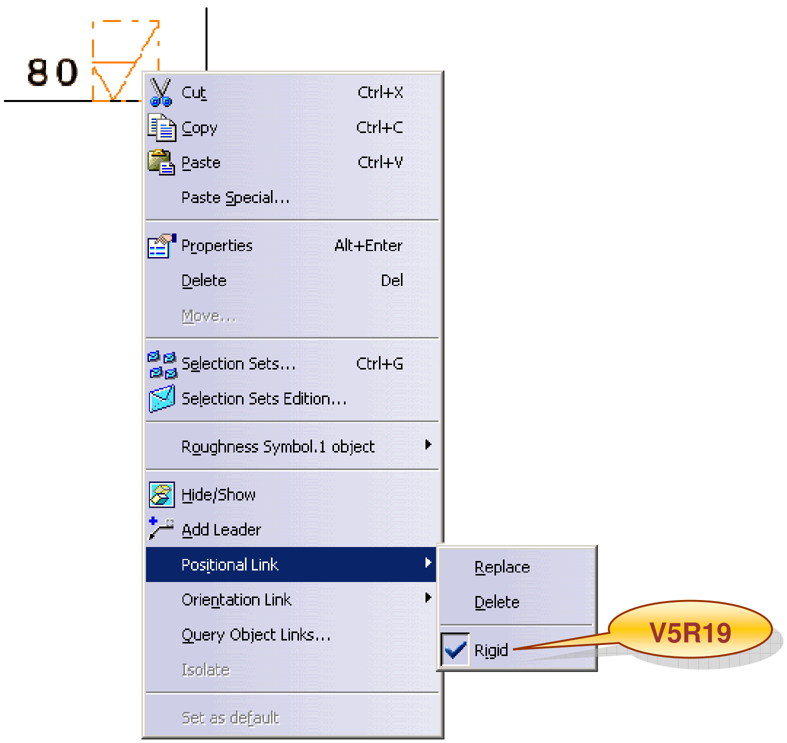


Student Notes:

About the Enhancement in Rigid position Link for Annotations

A new option called Rigid is added under the Positional Link command. If this option is selected the relative position between the annotation and its reference cannot be changed.

You can fix annotations such as Text, Roughness Symbol, Balloon, Datum Feature, Datum Target, Welding Symbol, Table, and Geometrical Tolerances with respect to their references.

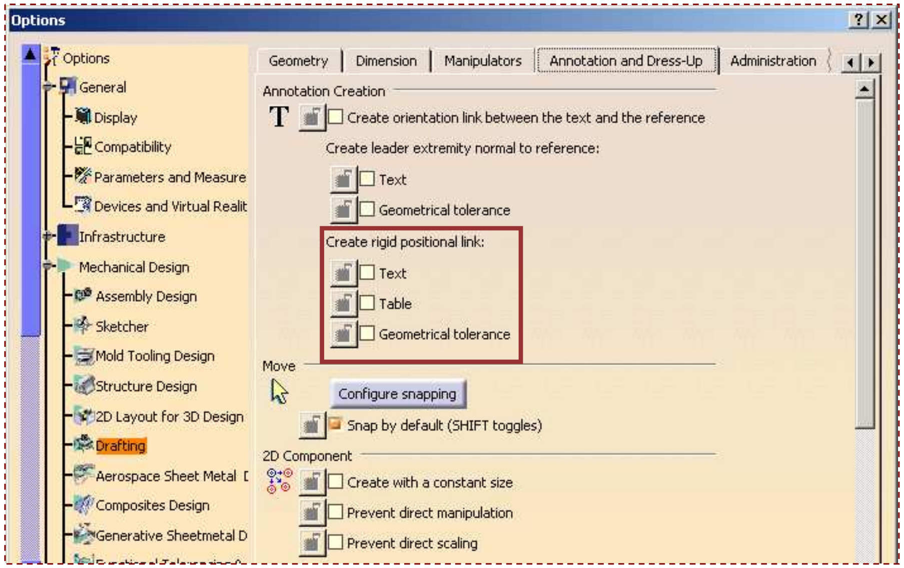


If required, you can later change the rigid character of a positional link by clearing the check mark.

Student Notes:

More About Rigid Position Link for Annotations

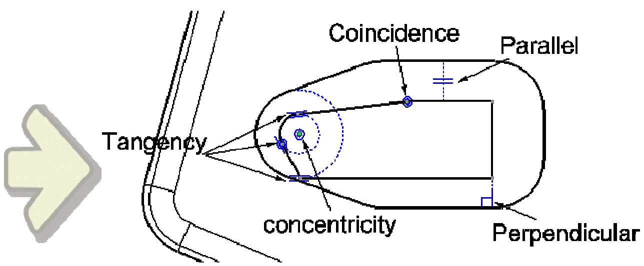
- You can have control on the creation of a rigid positional link by default. The setting Create rigid positional link in Tools > Options > Mechanical Design > Drafting > Annotation and Dress-up is provided.

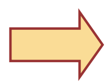


Student Notes:

Enhancement in Visualization of Broken Constraints

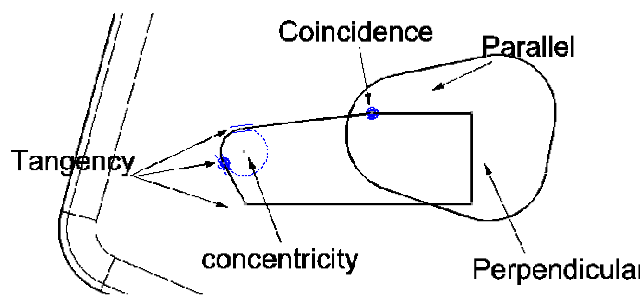
You will learn about the enhancement in visualization of broken constraints

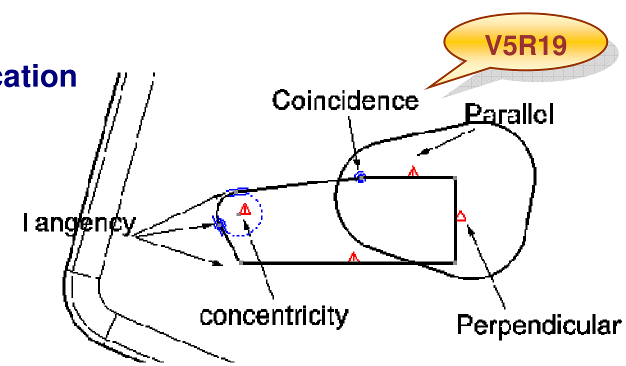




After modification and update

CATIA V5R18





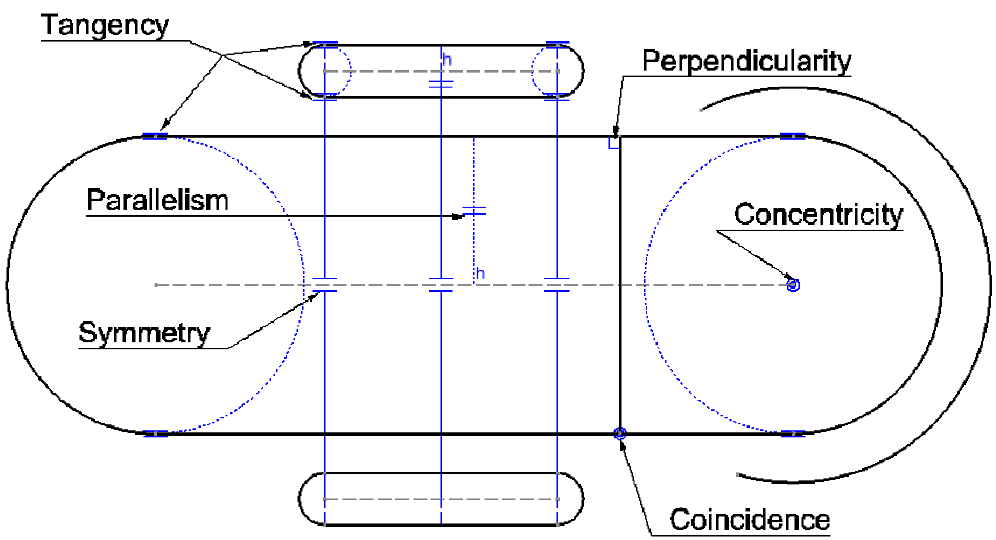
After modification in the reference profile used to sketch, the broken constraints are highlighted with a red triangle.

Student Notes:

About the Enhancement in Broken Constraints Visualization

Now the broken constraints are visualized at the same time as the normal (non-broken) constraints. You will be able to see them and to select them.

Types of Constraints:
There are six types of constraints that can be broken such as Parallelism, Perpendicularity, Coincidence, Concentricity, Tangency, and Symmetry.

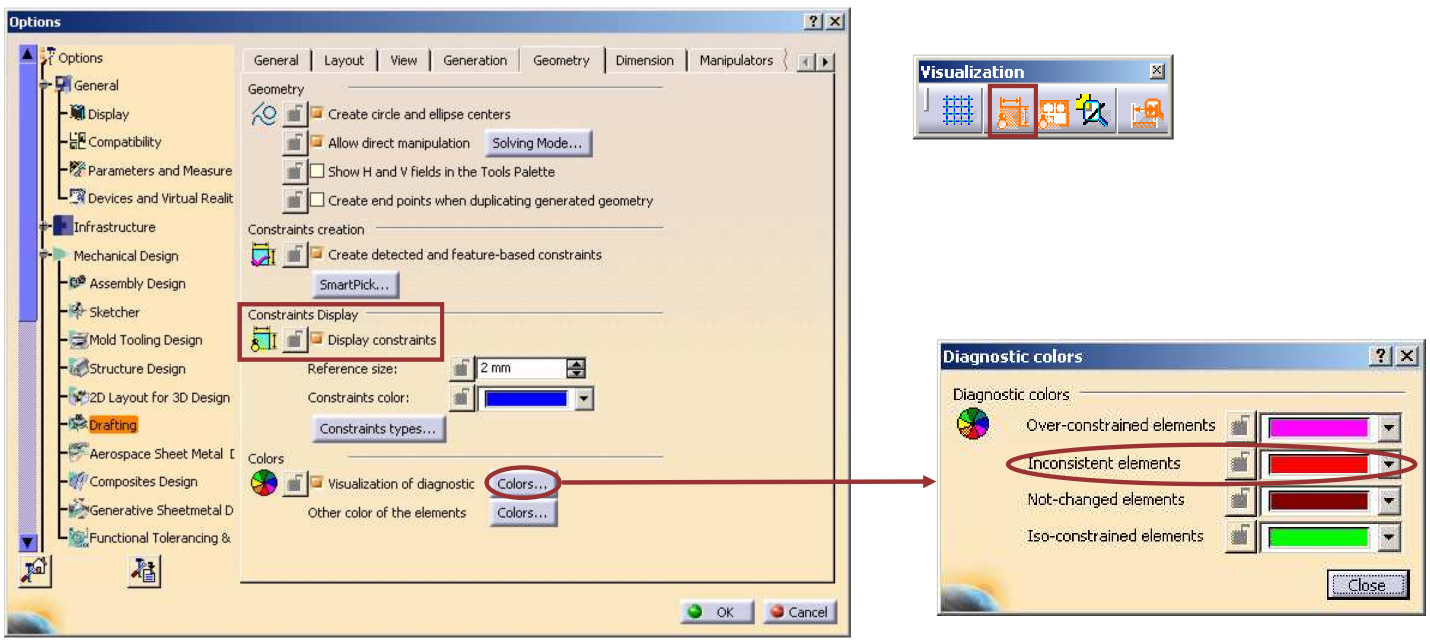


Previously the only way to access and delete the broken constraints was to use the Edit/Search command. You had to find out all constraints and retrieve those which were not valid.
You can interactively access the broken constraints, see them, and delete them if needed.

Student Notes:

More About the Broken Constraints Visualization

- **For visualizing the broken constraints you need to check the following settings to confirm that the broken (or non-broken) constraints are not hidden.**
  - **Go to Menu Tools > Options > Mechanical Design > Drafting > Geometry and check the Display constraints option.**
  - **Check in the Visualization toolbar and confirm that the Show Constraints option is active.**



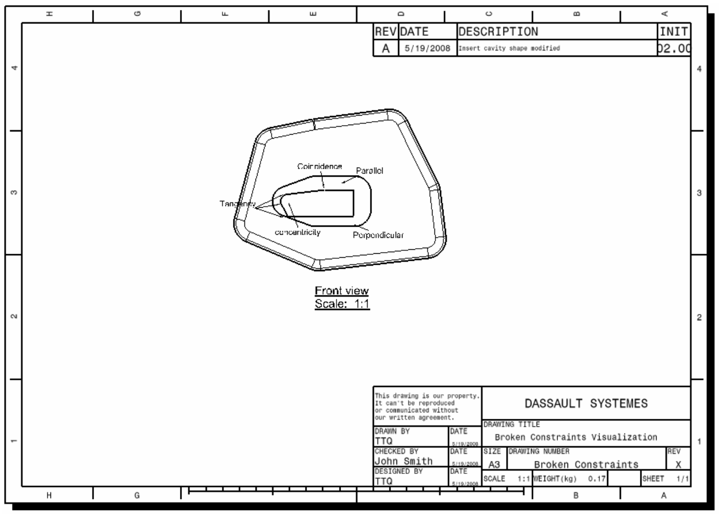
- **The colors of the broken constraints are those of the inconsistent elements. This can be changed in Tools > Options > Drafting > Geometry.**

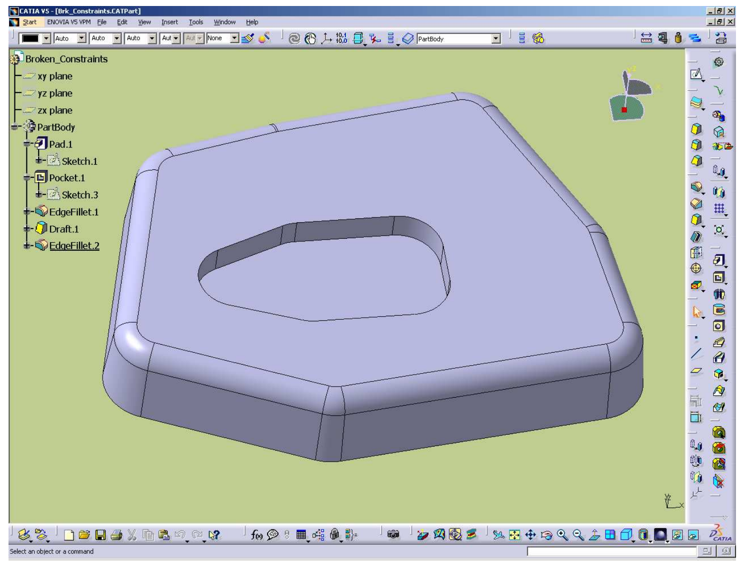
Student Notes:

Do It Yourself (1/2)

Part used: Brk_Constraints.CATDrawing

- Open the drawing and visualize the constraints.
- Open the Part linked to the drawing.





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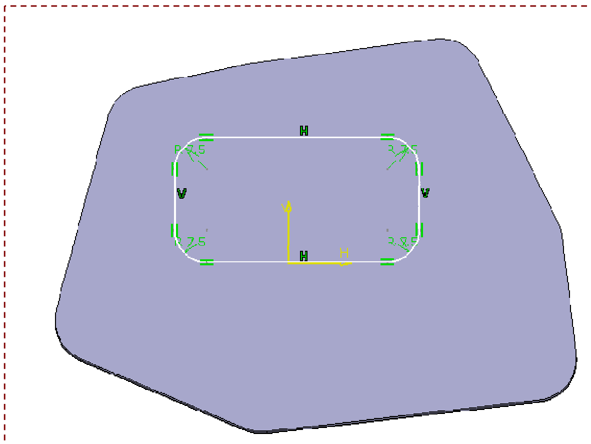
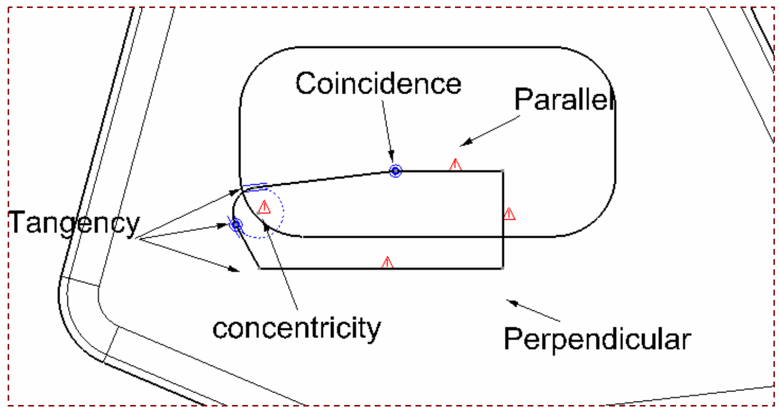
Student Notes:

Do It Yourself (2/2)

- Change the sketch profile of the pocket and save it.
- Open and update the Drawing.
- Broken constraints will be visualized.

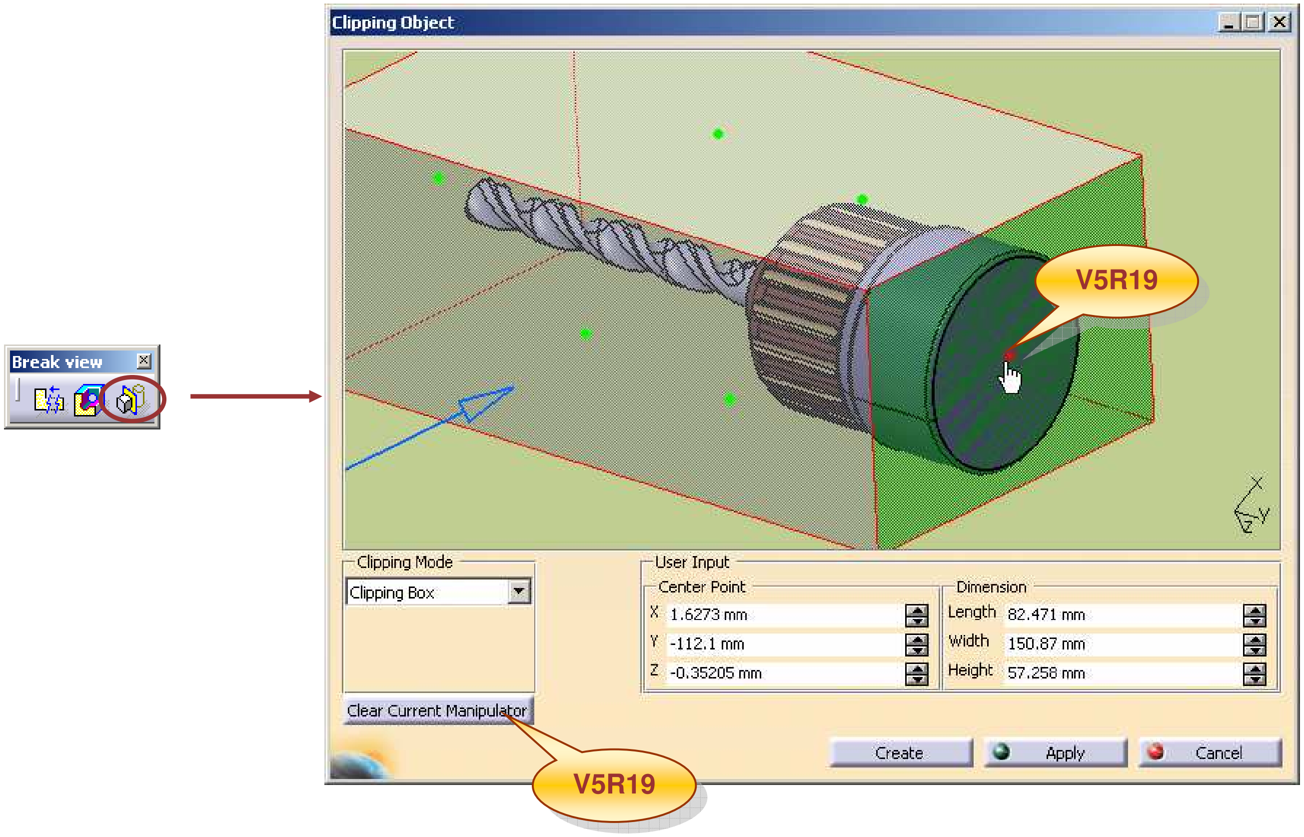




Student Notes:

Enhancement of 3D Clipping Box

You will learn how to use 3D Clipping box to generate a view.

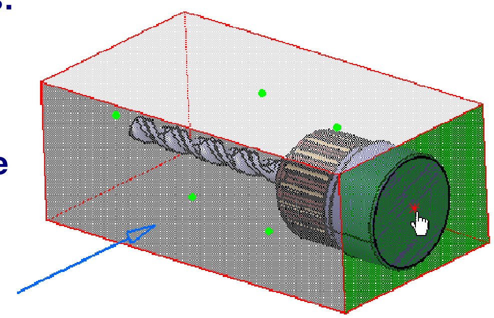


Student Notes:

About the Enhancement of 3D Clipping Box

In addition to the existing methods of defining the 3D clipping object (manipulators being moved manually, spinners and edit boxes), you will be able to select a point or a line in 2D and 3D view or a planar face / plane / edge in 3D in order to define the position of the current manipulator.

- The following geometries can be selected if the plane containing the geometry is parallel to the current manipulator plane:
  - Point, Line (they must be generatively created) in 2D.
  - Point, Line, Edge, Planar face, Plane in 3D, FTA Plane in 3D and the geometry outside the 3D clipping box.
- Some of the important things regarding 3D Clipping
  - The 3D Clipping object is not available for the following views:
    Section cut views, Aligned section views, Unfolded views, Views from 3D, Advanced front view with DMU box, CGR, Raster and Approximate views.
  - Any geometry inside the Clipping Object dialog box cannot be selected.
  - 2D view selection is not supported.
  - 3D clipping object is not associated with the 3D element.
  - Only one clipping object can be added per view.
  - Cut in section view has no effect on a 3D Clipping.
  - After the 3D clipping is created it can be modified or removed using the options provided in the Contextual menu.



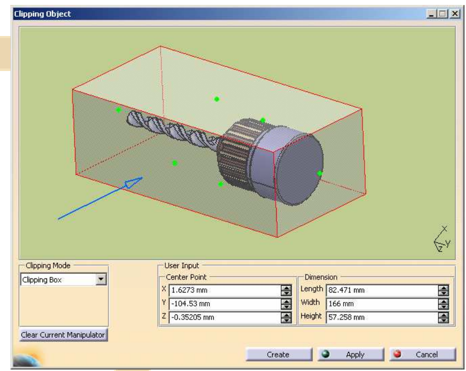
Student Notes:

How to Add/Modify 3D Clipping

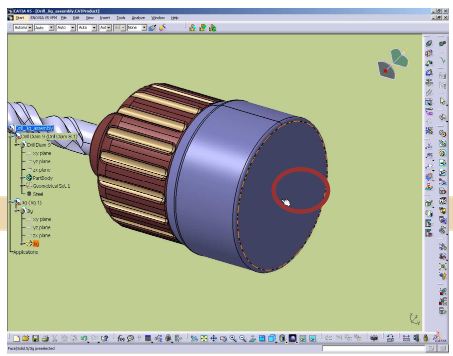
This new enhancement enables you to generate more precise and accurate 3D clippings. It also improves the productivity by reducing the time for manipulation of the plane.

1 Select the view to create 3D clipping and click Add 3D Clipping icon.

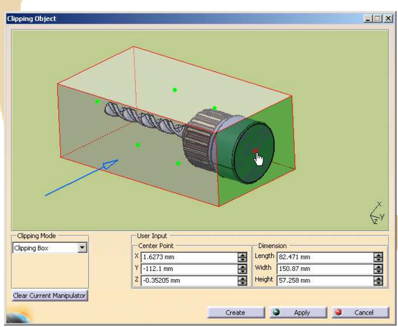
2 Double-click the manipulator to set it as current.



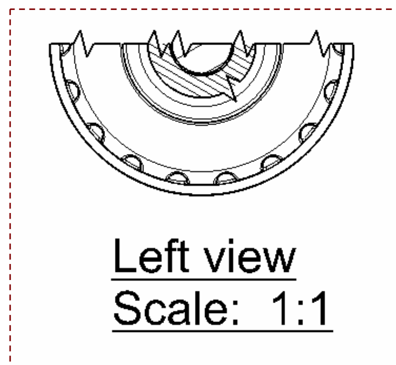
3 Select the face from the 3D product/part.



4 Manipulator plane will be aligned to the face.



5 Repeat this procedure for other planes and click Create. Clipping view will be generated.

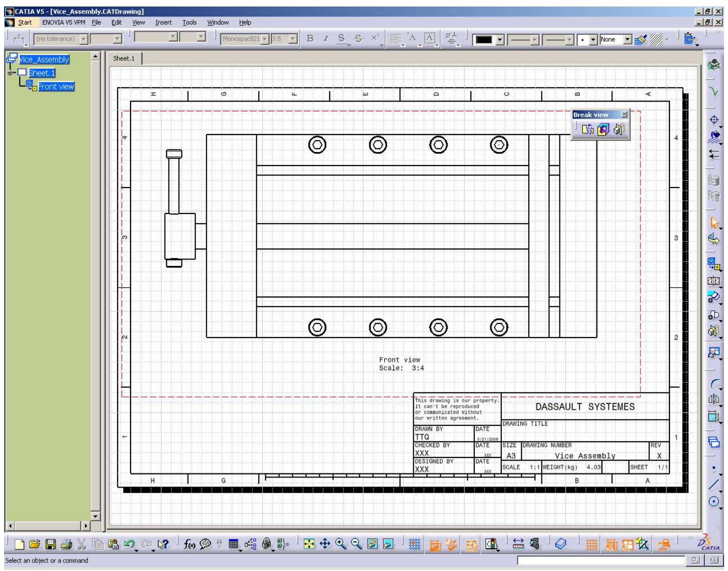


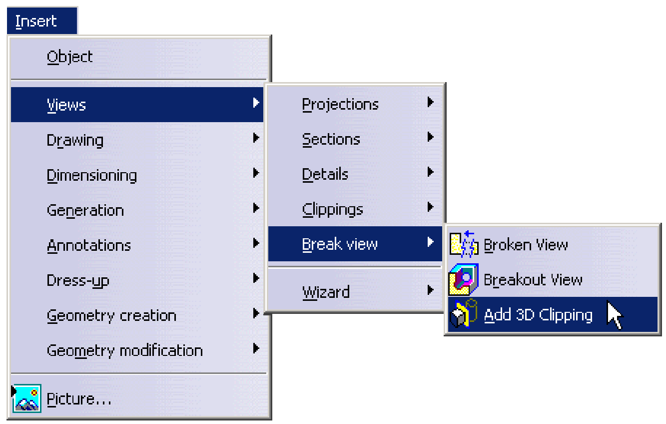
Student Notes:

Do It Yourself (1/5)

Part used: Vice_Assembly.CATDrawing

- Open Vice_Assembly.CATDrawing.
- Open ViceAssembly.CATProduct linked to the drawing.
- Select the Front View and then from the menu select Add 3D Clipping.

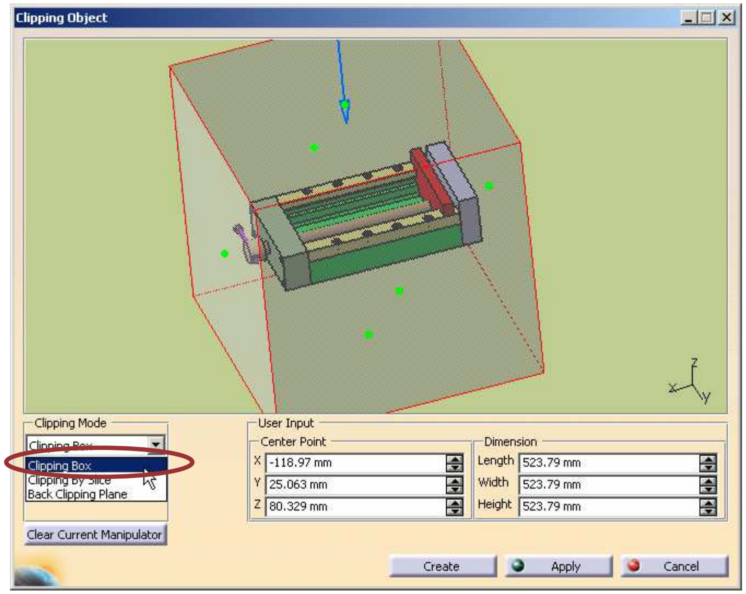


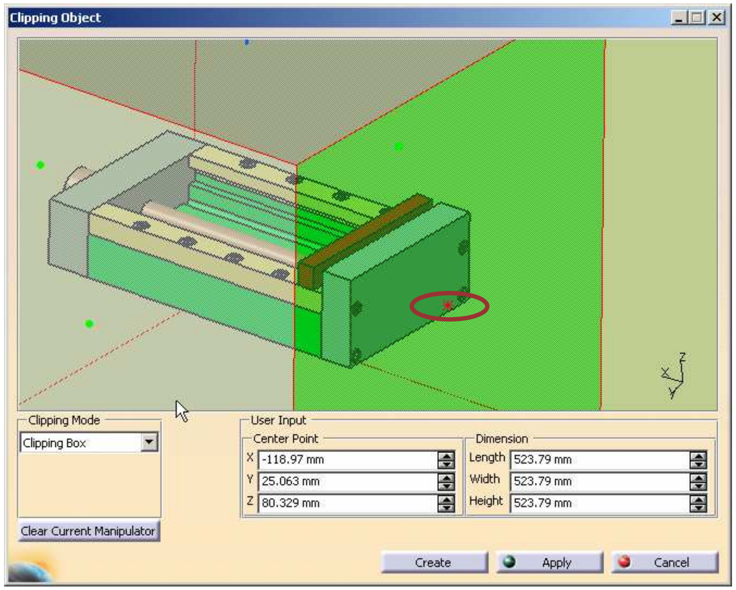


Student Notes:

Do It Yourself (2/5)

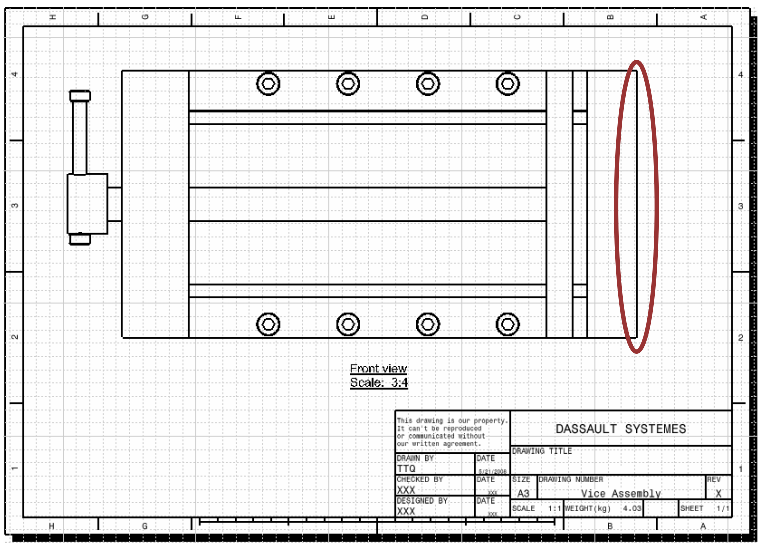
- **From the Clipping Mode select the Clipping Box.**
- **Double-click green manipulator to set it as current.**

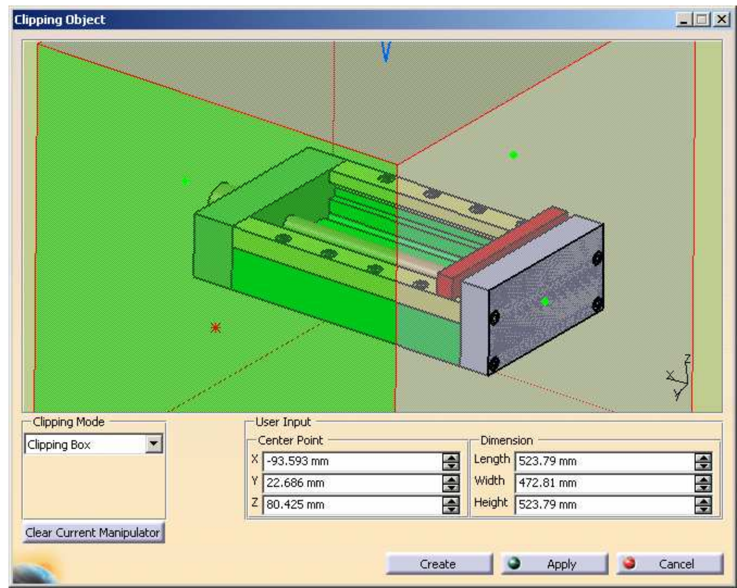




Do It Yourself (3/5)

- **Select the edge from the Drawing, the Clipping Box face will be aligned to the face corresponding to the selected edge.**
- **Double-click the other green manipulator to set it as current.**



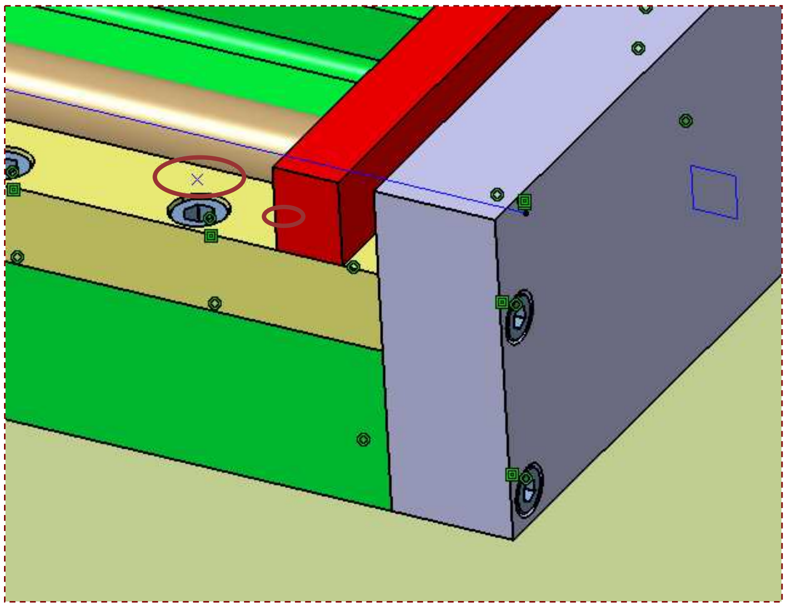


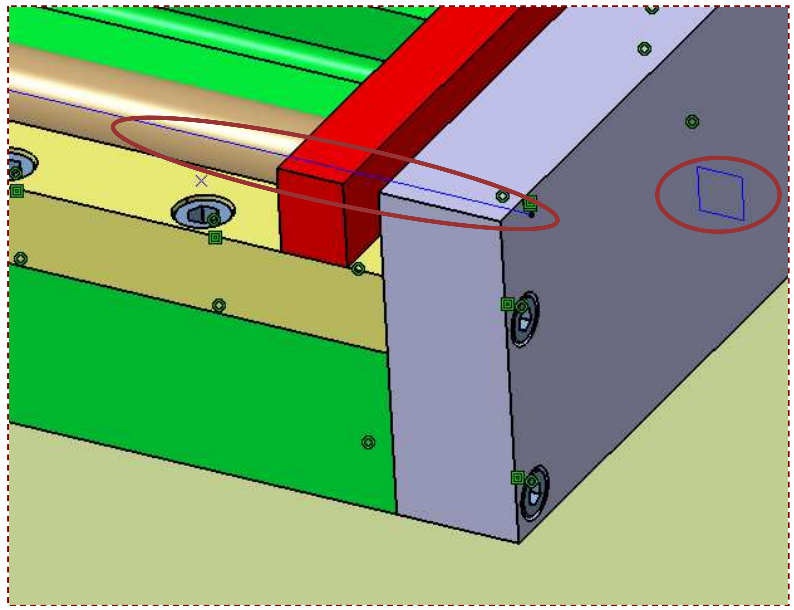
Student Notes:

Student Notes:

Do It Yourself (4/5)

- Select the point from the product.
- Similarly, for top and opposite side manipulators select blue line and blue plane as shown.

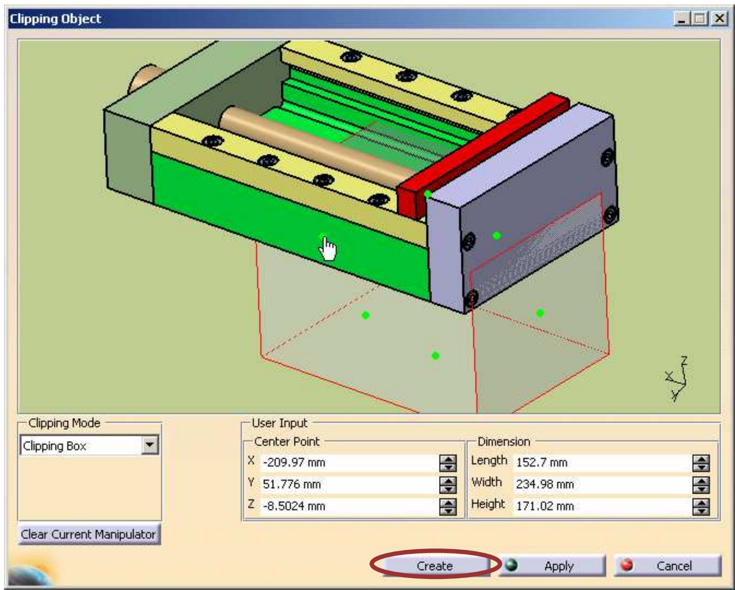


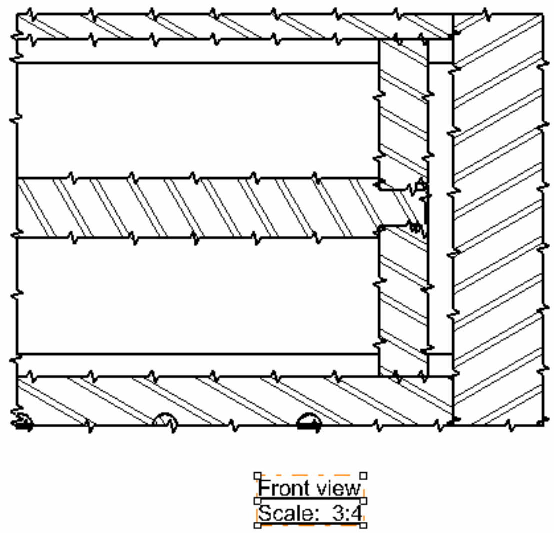


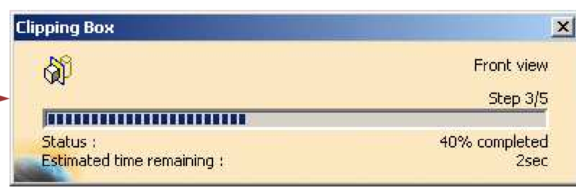
Student Notes:

Do It Yourself (5/5)

- **Drag the other manipulators and position the box.**
- **Click Create.**
- **Clipping view will be created.**







Part used: Vice_Assembly_Completed.CATDrawing

Sum up

In this course you have seen the major enhancements in the CATIA Mechanical Design solutions domain for V5R19.

- **Part Design Enhancements**
- **Product Design Enhancements**
- **Wireframe and Surface Design Enhancements**
- **Drafting Enhancements**

Student Notes: