CATIA V5 Design Fundamentals

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CATIA V5 Design Fundamentals
A Step by Step Guide

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Preface

This textbook explains how to create solid models, assemblies and drawings using CATIA V5. CATIA is a three dimensional CAD/CAM/CAE software developed by Dassault Systèms, France. This textbook is based on CATIA V5 Release 21. Users of earlier releases can use this book with minor modifications. We provide files for exercises via our website. All files are in Release 19 so readers can open the files using later releases of CATIA V5.

It is assumed that readers of this textbook have no prior experience in using CATIA V5 for modeling 3D parts. This textbook is suitable for anyone interested in learning 3D modeling using CATIA V5.

Each chapter deals with the major functions of creating 3D features using simple examples and step by step self-paced exercises. Additional drawings of 3D parts are provided at the end of each chapter for further self exercises. The final exercises are expected to be completed by readers who have fully understood the content and completed the exercises in each chapter.

Topics covered in this textbook

- Chapter 1: Basic component of CATIA V5 software, options and mouse operation.
- Chapter 2: Basic step by step modeling process of CATIA V5.
- Chapter 3 through 6: Creating sketches and sketch based features.
- Chapter 7: Usage of reference elements to create complex 3D geometry.
- Chapter 8: Dress-up features such as fillet, chamfer, draft and shell.
- Chapter 9: Modification of 3D parts to take advantage of parametric modeling concepts.
- Chapter 10: Creating complex 3D parts by creating multiple bodies and applying boolean operations.
- Chapter 11: Copying or moving geometrical bodies.
- Chapter 12 and 13: Constructing assembly structures and creating or modifying 3D parts in the context of assembly.
- Chapter 14 and 15: Creating drawings for parts or assemblies.
- Chapter 16: Advanced functions in creating a solid part such as a rib, stiffener and multisections solid.

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Chapter 8 Dress-Up Features

Chapter 12 Assembly Design I (Bottom-Up Assembly)

Chapter 14 Creating Drawing Views

Chapter 15 Dimension, Annotation and Assembly Drawing

Chapter 16 Sketch Based Features - Advanced

chapter

6 Sketch Based Features II (Shaft, Groove and Hole)

After completing this chapter you will understand

- how to create Shaft and Groove features.
- how to create a hole using the Hole command

6.1 Shaft

Using the **Shaft** command in the **Sketch-Based Features** toolbar, you can revolve a profile or surface to create a 3D geometry. The 3D geometry created with the **Shaft** command is added to the existing geometry.

Procedure

① Create a sketch.
② Choose the Shaft button from the Sketch-Based Features toolbar.
③ Select the profile or surface.
④ Select an axis of revolution.
⑤ Input the limit angle of the revolution and press OK.

If you press the button designated by A in Fig 6-1, you can invoke the **Sketcher** workbench and modify the sketch that you selected as the profile. If you did not create a sketch in advance, you can create a new sketch while running sketch based features such as **Pocket**, **Pad** and **Shaft** etc. by pressing the **Sketcher** button in the dialog box and specifying the sketch plane. After creating the sketch, you can exit the **Sketcher** as usual and the sketch is selected as the profile.

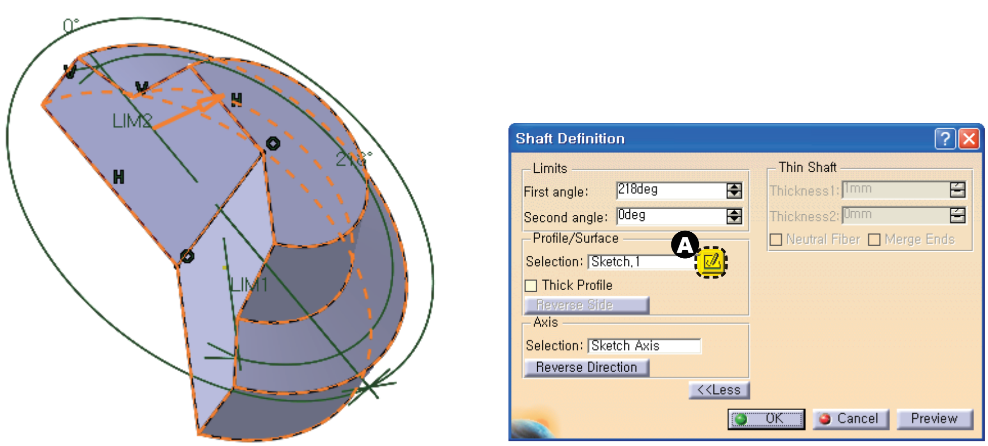


Fig 6-1 Creating a Shaft Feature

6.1.1 Characteristics of Profile and Axis of Shaft

When you define the profile and axis for the **Shaft**, you have to bear in mind the following characteristics.

① You can use the **Axis** line, **Reference Element** or **Standard Element** as an axis of the **Shaft**.
② You can create a solid body by revolving a closed profile.
③ When you are using an open profile as the profile of the first **Shaft** feature, you have to choose the **Thick** option to create a solid body (except for the case of no.⑥ below).
④ You cannot use an intersecting profile for a single **Shaft** feature under any circumstances.
⑤ You can select as many profile as you want unless they are intersecting, provided they are all closed.

⑥ For cases where the end points of an open profile are on the axis, you can create a solid body with an open profile.
(Refer to Fig 6-2.)
⑦ The axis cannot intersect the profile. (Refer to Fig 6-3.)
⑧ Values in the **First angle** and **Second angle** should not exceed 360°.

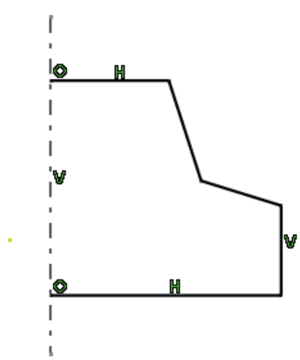


Fig 6-2 Open Profile

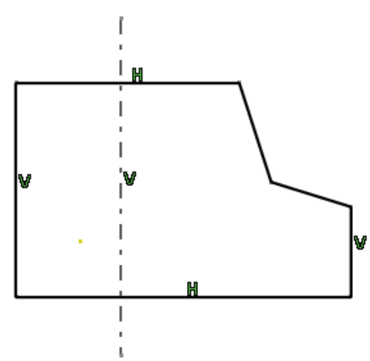


Fig 6-3 Axis Intersecting the Profile

6.2 Groove

Using the **Groove** command in the **Sketch-Based Features** toolbar, you can revolve a profile and remove material from the existing solid body. The **Groove** icon is activated only when at least one solid body exists. Therefore, you cannot create a **Groove** as the first feature.

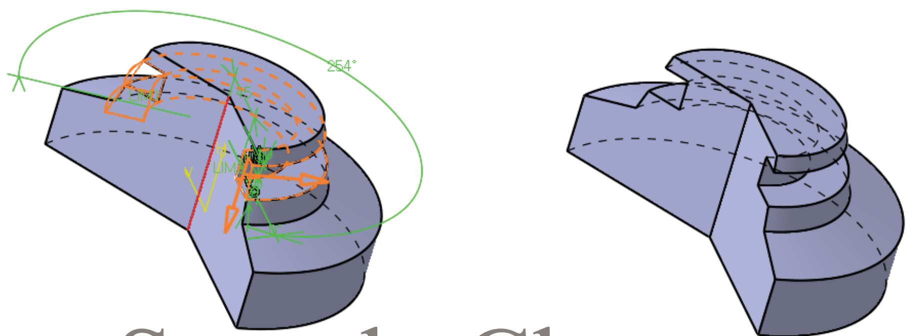


Fig 6-4 Groove Feature

Exercise 01 Creating a Groove Feature

ch06_001.CATPart

Let's create a Groove feature after creating a sketch.

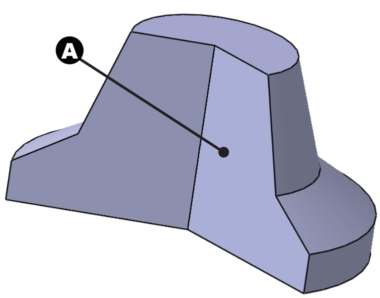


Fig 6-5 Sketch Plane

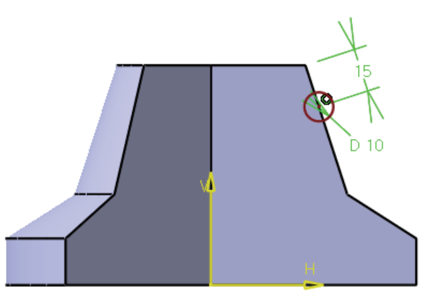


Fig 6-6 Sketch

Step 1: Sketch

1. Open the given part file. (ch06_001.CATPart)

2. Invoke the **Sketcher** workbench by specifying the plane **A** in Fig 6-5 as the sketch plane.

3. Create a circle as shown in Fig 6-6 and iso-constrain it.

4. Exit the **Sketcher**.

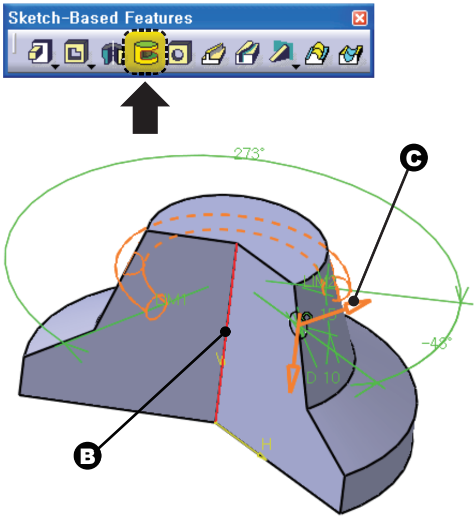


Fig 6-7 Selecting Sketch and Axis

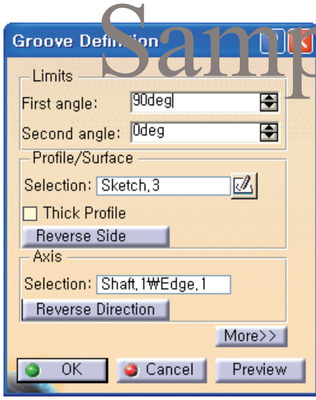


Fig 6-8 Limit Values

Step 2: Groove

1. Choose the **Groove** button from the **Sketch-Based Features** toolbar.

2. Select the sketch created in Step 1.

3. Select the axis of revolution (B in Fig 6-7).

4. Press the **Reverse Direction** button in the dialog box, if required, such that the direction of the arrow appears as shown in Fig 6-8 C.

5. Input the limit values as shown in Fig 6-8. The preview is updated.

6. Press the **OK** button.

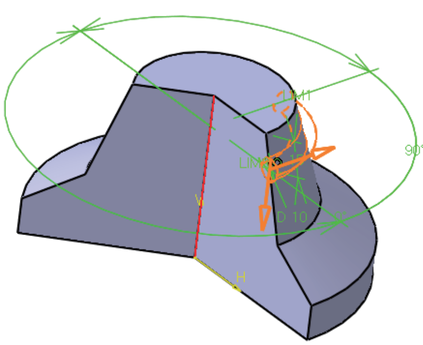


Fig 6-9 Preview

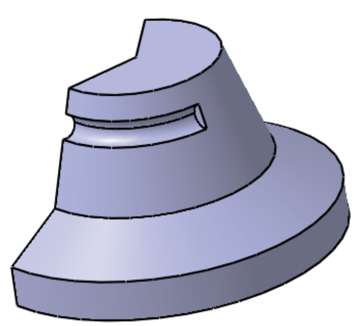


Fig 6-10 Groove Feature

END of Exercise

6.3 Hole Command

Using the **Hole** command in the **Sketch-Based Features** toolbar, you can easily create standard holes that are frequently used in mechanical parts.

You can create five types of standard holes in CATIA V5.

① Simple Hole
② Tapered Hole
③ Counterbored Hole
④ Countersunk Hole
⑤ Counterdrilled Hole

Fig 6-11 shows the section of each standard hole.

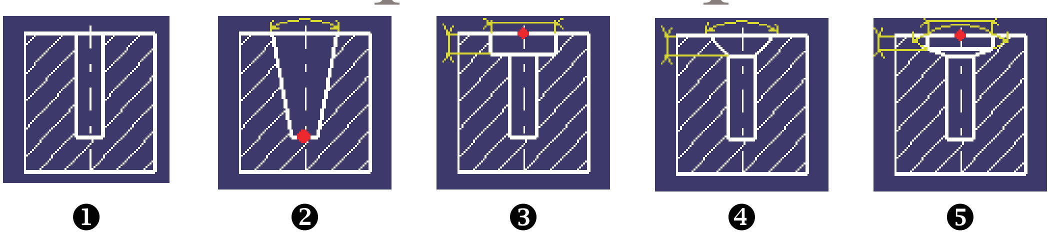


Fig 6-11 Section of Standard Holes

Remember that you can create holes by using the **Pocket** or **Groove** commands if you do not know how to use the **Hole** command.

6.3.1 Creating a General Hole

You can create a general hole according to the following procedure.

① Press the **Hole** button in the **Sketch-Based Features** toolbar.
② Select a plane onto which to create a hole.
③ Set the **Extension**, **Type** and **Thread Definition** in the **Hole Definition** dialog box.
④ Invoke the **Sketcher** by pressing the **Positioning Sketch** button and define the location of the hole center.
⑤ Press the **OK** button.

If you press the **Positioning Sketch** button, you can define the location of the hole center by using the sketch constraint. You may modify the location of the hole later by accessing the sketch feature in the Spec Tree.

Note that you can create only one hole in a single operation. If you have to create a hole recursively, you can copy or pattern the hole using the commands in the **Patterns** toolbar.

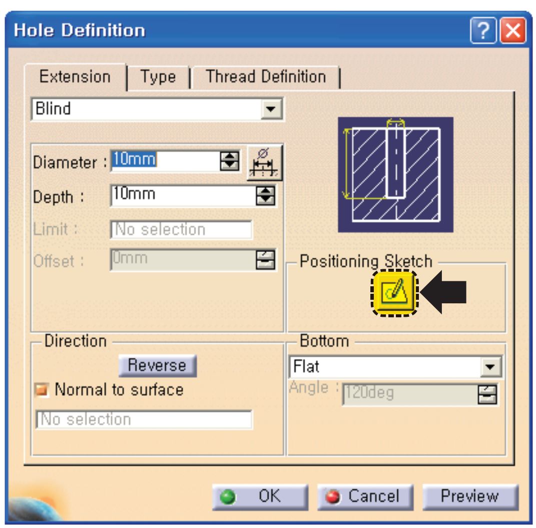
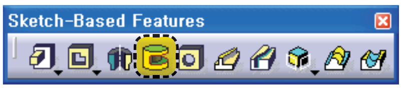


Fig 6-12 Hole Definition Dialog Box

Exercise 02 Creating a General Hole

ch06_002.CATPart



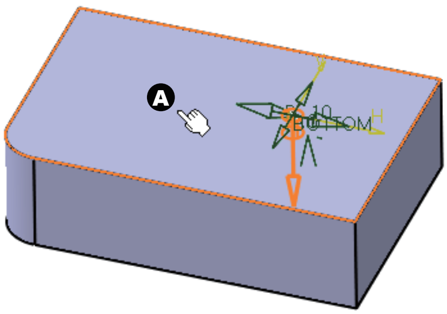


Fig 6-13 Plane for Hole

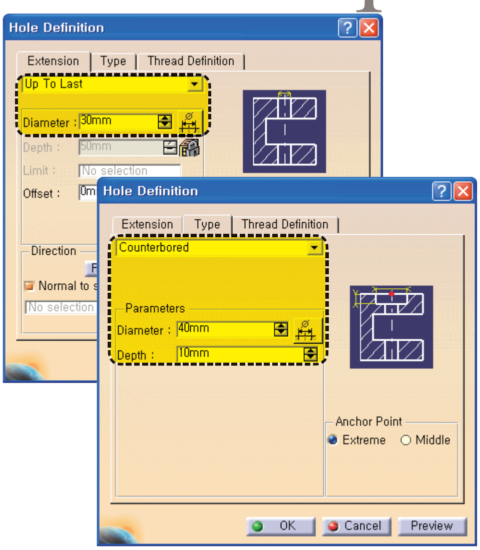


Fig 6-14 Options

1. Open the given part file. (ch06_002.CATPart)

2. Choose the **Hole** button from the **Sketch-Based Features** toolbar.

Read the message in the status bar.

3. Select the face **A** shown in Fig 6-13.

4. Set the options in the **Extension** and **Type** tab in the **Hole Definition** dialog box as shown in Fig 6-14 and press **OK**.

A counter bored hole is created at an arbitrary location as shown in Fig 6-15.

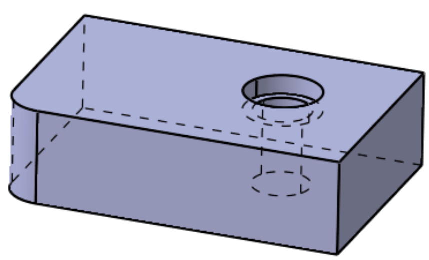


Fig 6-15 Counterbored Hole Created

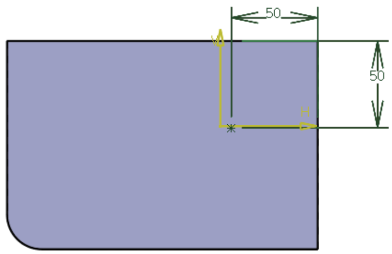


Fig 6-16 Define the Location

5. Double click the **Sketch** feature under the **Hole** feature in the Spec Tree. The **Sketcher** workbench is invoked.

6. Press the **Constraint** button and define the location of the point as shown in Fig 6-16.

7. Exit the **Sketcher**.

The location of the hole is updated.

END of Exercise

Anchor Point

The **Anchor Point** option in the **Type** tab of **Hole Definition** dialog box defines the location of the base point in the hole section.

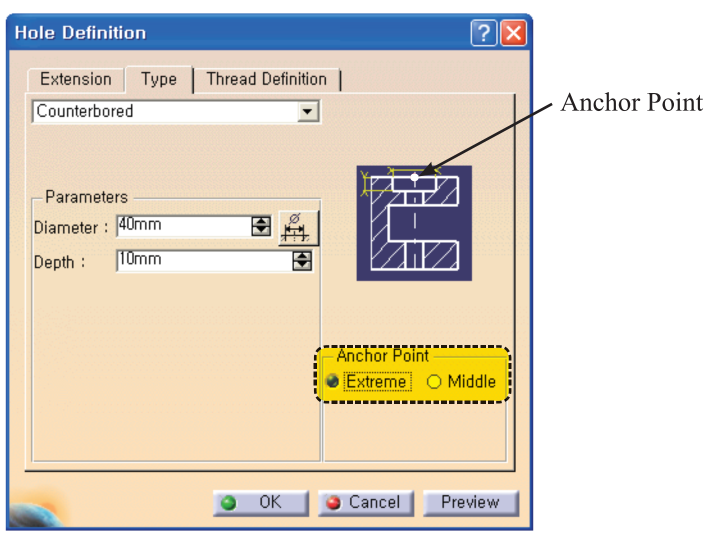


Fig 6-17 Anchor Point Option

6.3.2 Creating a Hole at an Existing Center

You can create a hole at the center of an existing circle or arc according to the following procedure.

① Press the Hole button.
② Select a circle or an arc whose center will be coincident with the center of the hole.
③ Select a plane.
④ Set the Extension, Type and Thread Definition in the Hole Definition dialog box.
⑤ Press the OK button.

Exercise 03 Creating a Hole at an Existing Center

ch06_003.CATPart

Open the part file ch06_003.CATPart and create a simple hole whose center is coincident with the center of an existing arc. The number in Fig 6-18 corresponds to the procedure explained above.

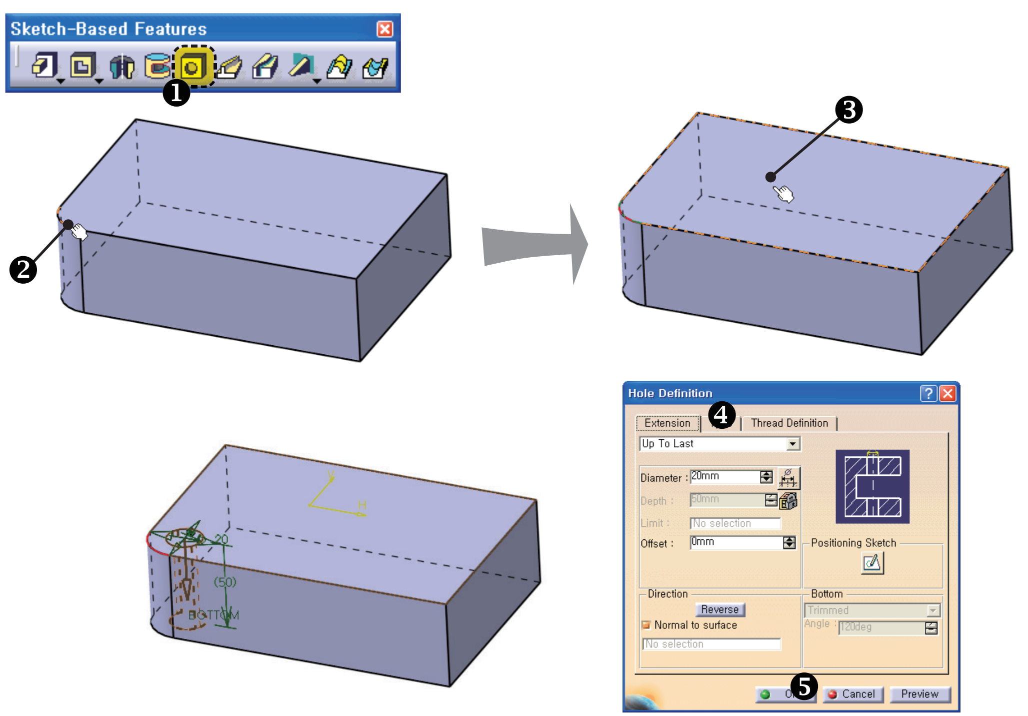


Fig 6-18 Creating a Simple Hole

END of Exercise

6.3.3 Creating a Hole at a Distance from Existing Edges

You can create a hole at a distance from the existing linear edge(s) according to the following procedure.

① Press the Hole button.
② Select one or two linear edge(s).
③ Select a plane.
④ Set the Extension, Type and **Thread Definition** in the **Hole Definition** dialog box.
⑤ Press the **OK** button.

ch06_003.CATPart **Creating a Hole at a Distance from Existing Edges** ***Exercise 04***

Using the part file in Exercise 03, create a countersunk hole at a location measured from the existing edges.

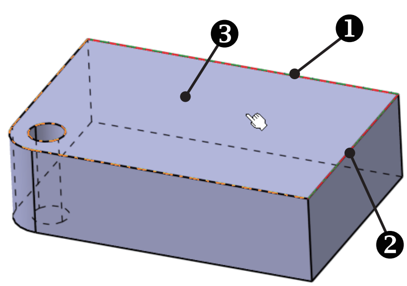


Fig 6-19 Selecting Edges and Plane

1. Press the **Hole** button and select the edges ❶, ❷ and plane ❸ in order as shown in Fig 6-19.

2. Double click the dimension Ⓐ and Ⓑ shown in Fig 6-20 and modify each value to 40mm.

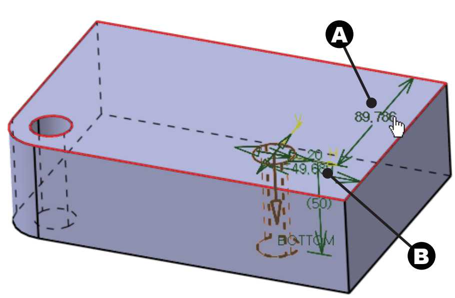


Fig 6-20 Modifying the Dimensions

Message in the Status Bar

Note that the status bar message after pressing the Hole button is as follows.

"Select a face or a plane. Optionally select a point or line first to position the sketch."

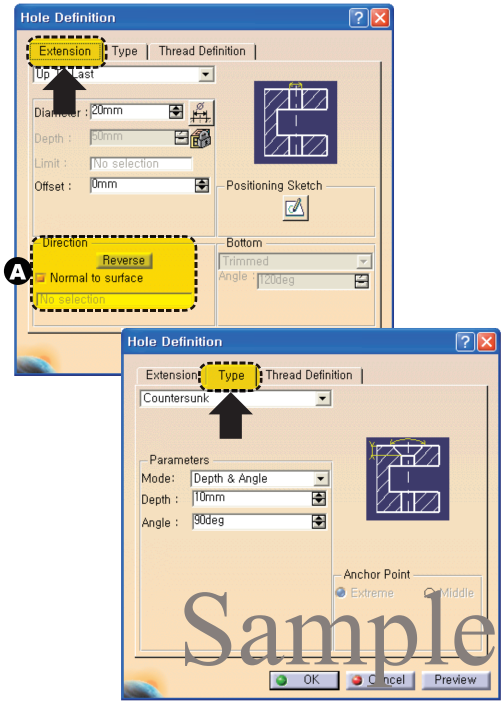


Fig 6-21 Type and Extension Option

3. Set the **Extension** and **Type** option in the dialog box as shown in Fig 6-21.

Fig 6-22 shows the countersunk hole created at a specified location from the existing edges.

Direction of the Hole

With the **Direction** option designated by **A** in Fig 6-21, you can specify the direction of the hole not normal to the sketch plane.

Creating Points in Advance

You can create points in advance at which to create holes. If you want to create holes at several points, you have to run the **Hole** command for each hole.

When you are creating several holes with the same section and options but at different locations, you may create a hole first and use the **User Pattern** command which will be explained further in Chapter 11.

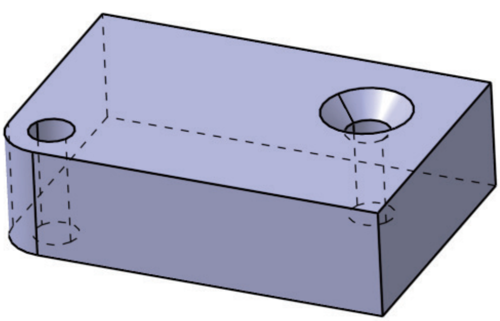


Fig 6-22 Countersunk Hole Created

END of Exercise

Let's create the **Shaft** and **Groove** feature according to the following steps.

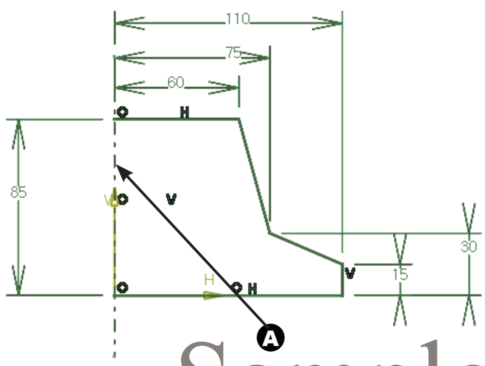


Fig 6-23 The First Sketch

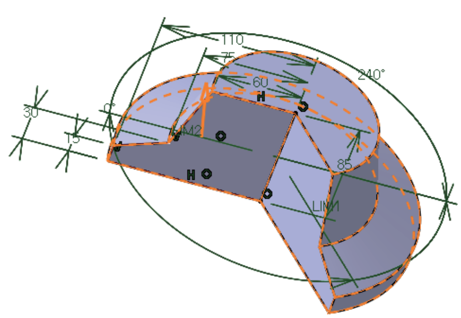


Fig 6-24 Preview of the Shaft Feature

Step 1: Create a Shaft

1. Create a sketch on the xy plane as shown in Fig 6-23. The axis **A** is created with the **Axis** command in the **Profile** toolbar.

2. Exit the **Sketcher** and press the **Shaft** button.

3. Select the sketch created above as the profile.

4. Input 240 deg in the **First angle** input box and press the **Preview** button. The preview is shown in Fig 6-24.

5. Press the **OK** button to create the first **Shaft** feature.

Step 2: Second Sketch

1. Choose the **Positioned Sketch** button from the **Sketcher** toolbar and select the plane **B** shown in Fig 6-25. Align the H and V axis as it appears in the figure.

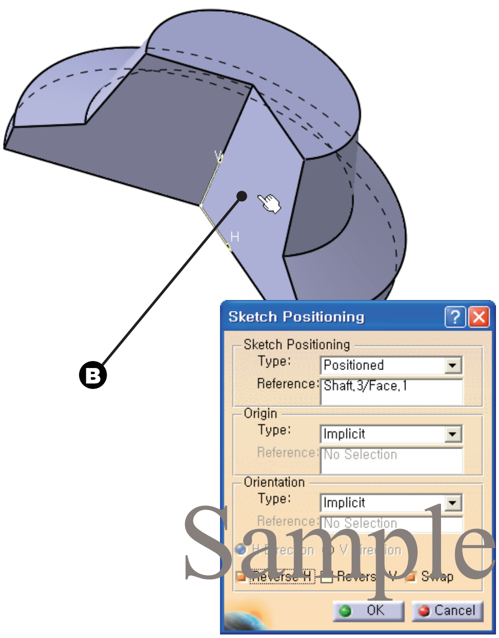


Fig 6-25 The Second Sketch Plane

2. Use the **Project 3D Elements** command in the **Operation** toolbar to project the curves C and D shown in Fig 6-26 on the sketch plane.

3. Offset the projected curve 10mm inside the geometry as shown in fig 6-27 E.

4. Create an additional line located 35mm above the bottom edge as shown in Fig 6-27 F.

5. Convert the projected curve C and D shown in Fig 6-26 as the construction elements.

6. Trim out the unnecessary portion of the curves using the **Quick Trim** command.

7. Exit the **Sketcher**.

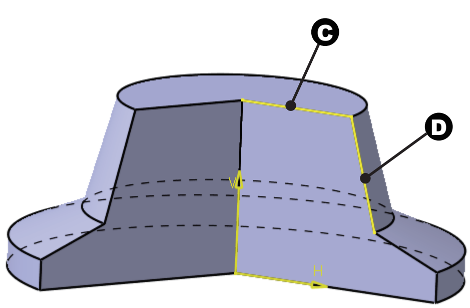


Fig 6-26 Project

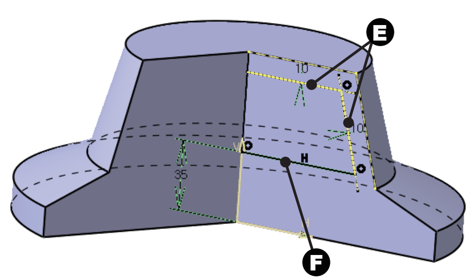


Fig 6-27 The Second Sketch Completed

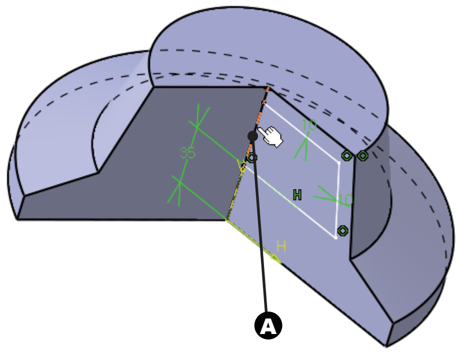


Fig 6-28 Axis

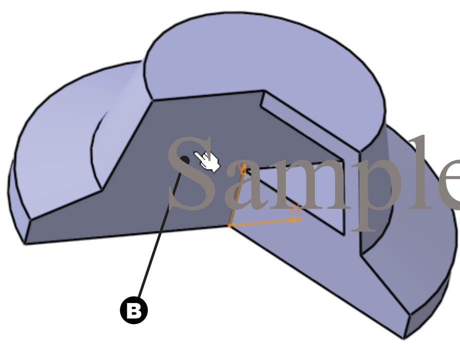


Fig 6-29 The Third Sketch Plane

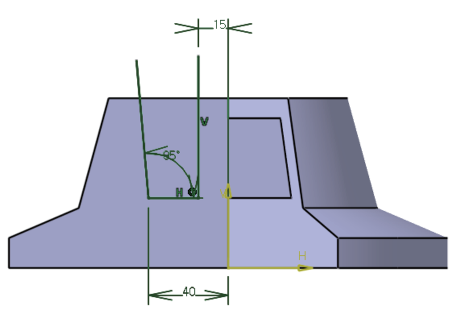


Fig 6-30 The Third Sketch

Step 3: Create a Groove

1. Choose the **Groove** button from the **Sketch-Based Features** toolbar and select the sketch that was created in Step 2.

The status bar message prompts you to select an axis.

2. Select the edge A shown in Fig 6-28 as the axis.

3. Input 60 deg in the **First angle** input box and press **OK**.

Step 4: The Third Sketch

1. Choose the **Positioned Sketch** button from the **Sketcher** toolbar and select the plane B shown in Fig 6-29 as the sketch plane.

2. Create an open sketch as shown in Fig 6-30.

3. Exit the **Sketcher** workbench.

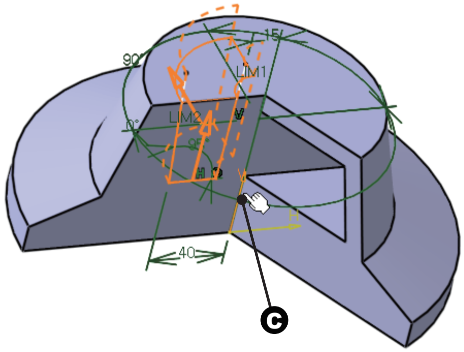


Fig 6-31 Preview of the Groove

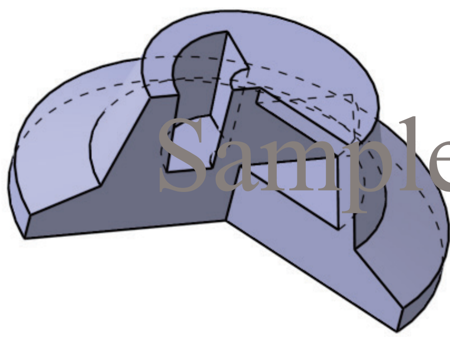


Fig 6-32 Completed Model

Step 5: Create a Groove

1. Choose the **Groove** button from the **Sketch-Based Features** toolbar and select the sketch that is created in Step 4.

2. Select the edge C in Fig 6-31 as the axis and set the **Reverse side** and/or **Reverse direction** options so that the arrows point as shown in Fig 6-31. You can reverse the direction by clicking the arrow head in the preview.

3. Input the angle to 90 deg.

4. Press the **OK** button.

Fig 6-32 shows the completed model.

END of Exercise

Exercise 06 **Spherical Guide Block**

Create a spherical guide block referring to the drawing in Fig 6-33.

Requirements

① The diameter of the four holes specified by A are linked such that if you modify one, all other holes are updated.

② The **Pocket** feature designated by B in Fig 6-33 has to remove the sphere completely regardless of the radius of the sphere.

③ The **Pad** feature designated by C has to end on the spherical surface.

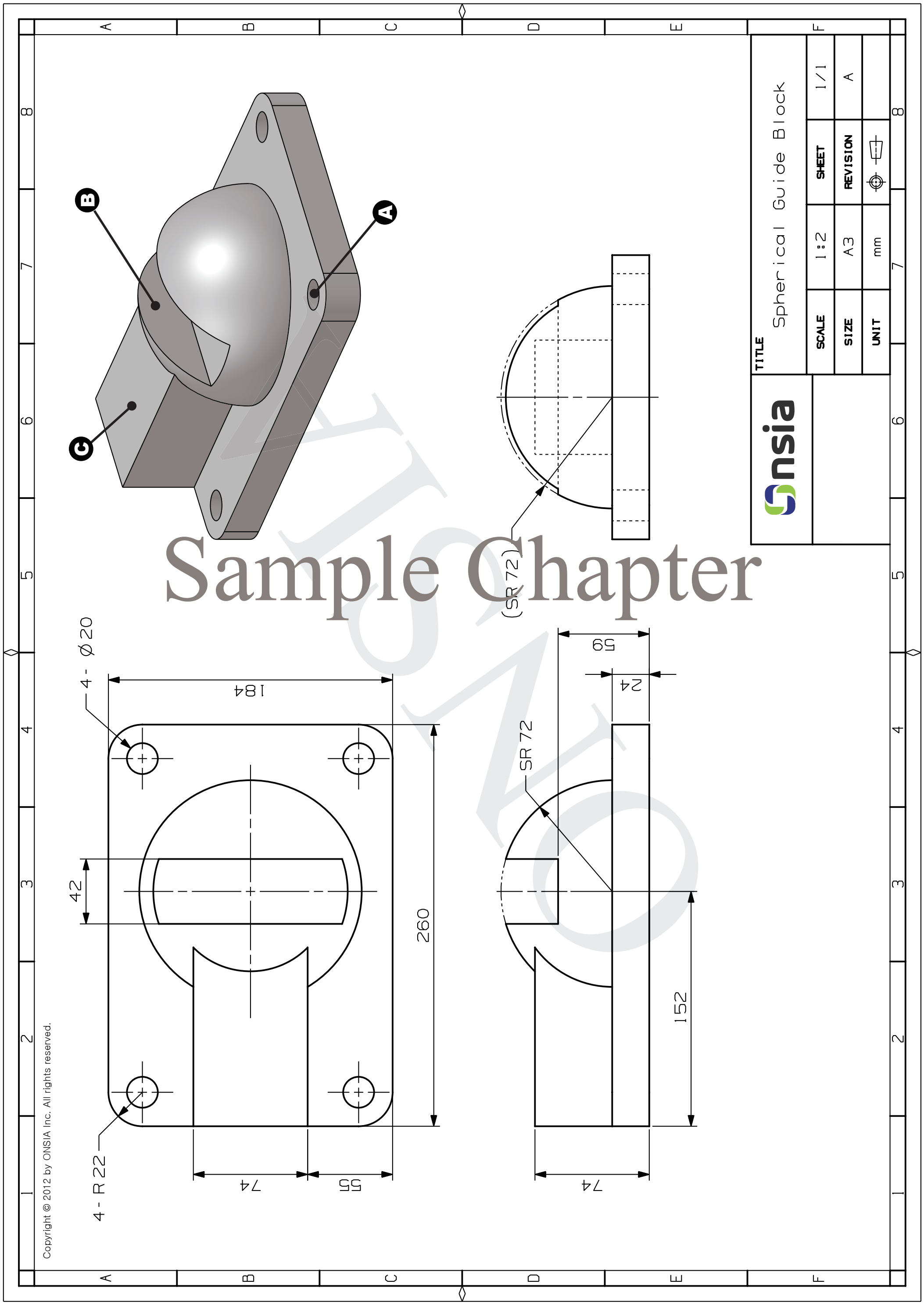


Fig 6-33 Spherical Guide Block

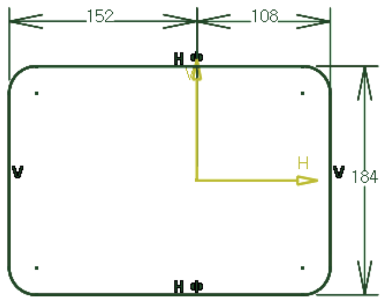


Fig 6-34 Base Sketch

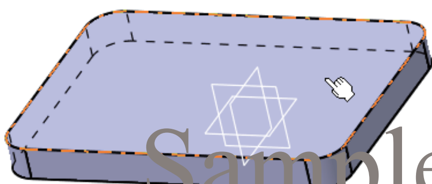


Fig 6-35 Sketch Plane

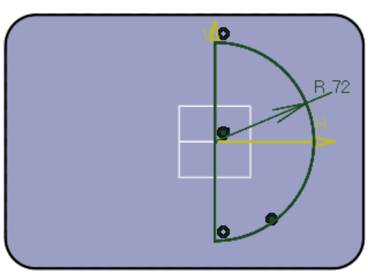


Fig 6-36 Half Circle

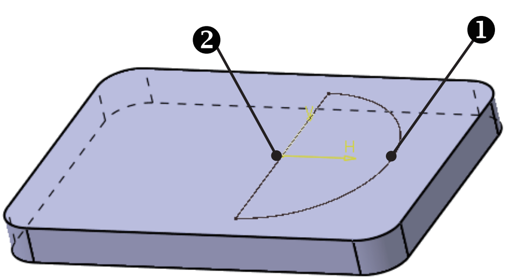


Fig 6-37 Selecting Profile and Axis

Step 1: Bottom Plate

1. Create a sketch for the bottom plate on the xy plane as shown in Fig 6-34.

2. Pad it by 24mm along the Z axis.

Step 2: Creating a Hemisphere

1. Define a sketch on the upper plane of the bottom plate as shown in Fig 6-35.

2. Create a half circle as shown in Fig 6-36 and iso-constrain it. The line passing through the center of the circle will be used as the axis. You may create the line with the **Axis** command because the end points of the half circle are on the axis.

3. Press the **Shaft** button.

4. Select the sketch designated by ❶ in Fig 6-37 as the profile. The **Selection** area of the **Axis** option is activated and the status bar message prompts you to select an axis.

5. Select the line ❷ shown in Fig 6-37 as the axis.

6. Input 180 deg in the **First angle** input area and press **OK**.

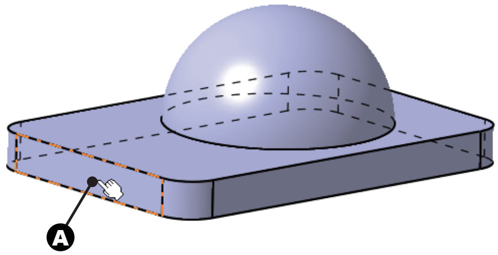
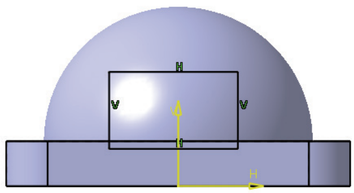


Fig 6-38 Sketch Plane



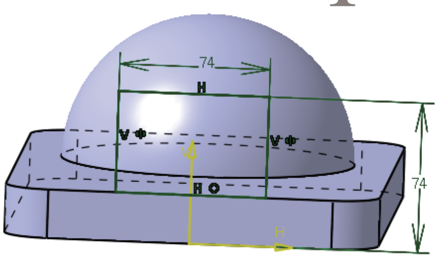


Fig 6-39 Rectangle

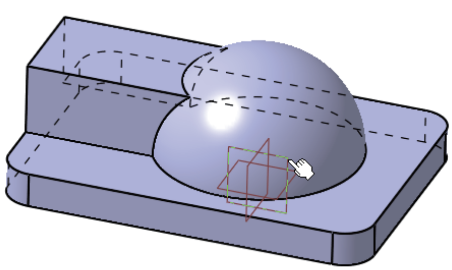


Fig 6-40 Sketch Plane

Step 3: Pad Feature* C *in Fig 6-33

1. Press the **Sketch** button and select the plane A shown in fig 6-38 as the sketch plane. Be careful not to select the other side of the base plate because the hemisphere is one-sided.

2. Create the sketch as shown in Fig 6-39.

3. Exit the sketcher and press the **Pad** button.

4. Select the sketch in Fig 6-39 as the profile.

5. Select the type of the **First limit** as **Up to surface**.

6. Select the surface of the hemisphere.

7. Confirm the preview and press **OK**.

Step 4: Pocket Feature* B *in Fig 6-33

1. Press the **Sketch** button and select the xz plane as the sketch plane as shown in Fig 6-40. You can select the zx plane in the Spec Tree.

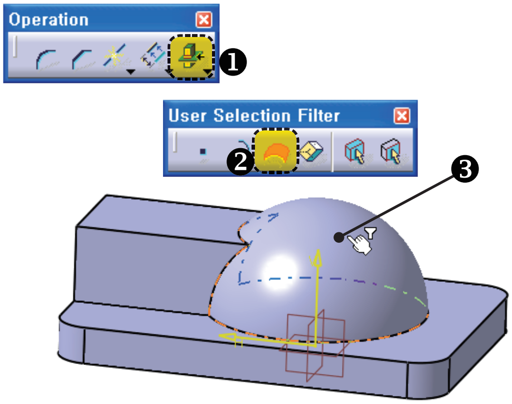


Fig 6-41 Selecting the Surface

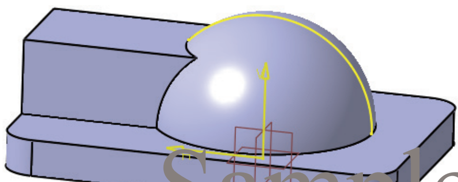


Fig 6-42 Intersection Curve

2. Choose the **Intersect 3D Elements** button (❶ in Fig 6-41) from the **Operation** toolbar.

3. Click the **Surface Filter** button (❷ in Fig 6-41) from the **User Selection Filter** to turn it on.

4. Select the spherical surface ❸ shown in Fig 6-41.

A yellow intersection curve is created as shown in Fig 6-42.

5. Click the **Surface Filter** button from the **User Selection Filter** to turn it off.

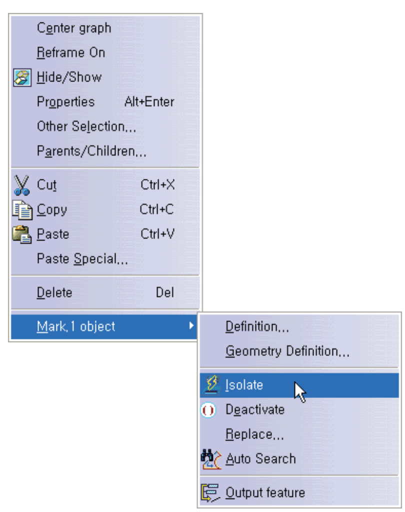
Yellow Intersection Curve

If the yellow intersection curve is not obtained, check the option as follows.

① Choose **Tools** > **Options** from the menu bar.
② The "**Keep link with selected object**" option in the **General** tab of **Infrastructure** > **Part Infrastructure** has to be checked.

The yellow curve implies that the curve is linked to another geometric element. If the source element is modified, the linked element is updated.

If you want to break the link, place the mouse cursor on the curve and press MB3. The **Isolate** menu is available in the pop-up menu of the linked object.



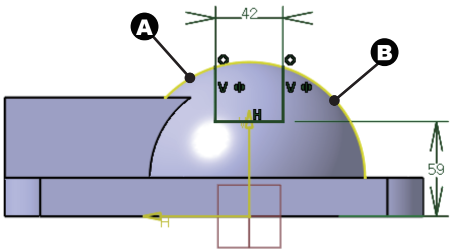


Fig 6-43 Sketch

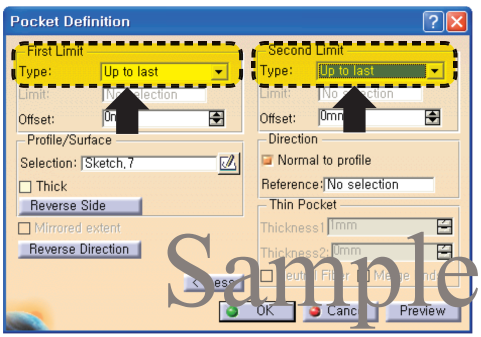


Fig 6-44 Limits Options

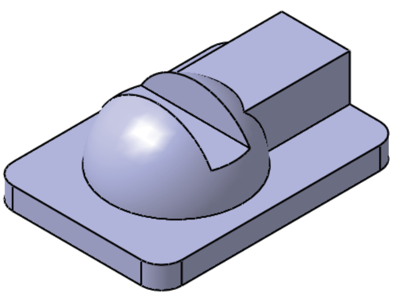


Fig 6-45 Pocket Feature

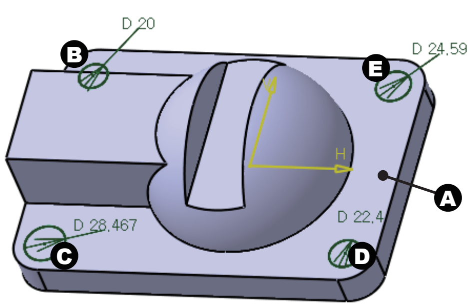


Fig 6-46 Sketch for Holes

6. Align the sketch plane by pressing the **Normal View** button in the **View** toolbar. Note that the normal view is reversed if you click the button one more time.

7. Create a sketch as shown in Fig 6-43. Trim out the portion A and B of the intersection curve.

8. Exit the **Sketcher**.

9. Press the **Pocket** button.

10. Press the **More** button in the **Pocket Definition** dialog box. Set the **First limit** and the **Second limit** as shown in Fig 6-44 and press **OK**.

This is to fulfill the second requirement of this exercise.

Fig 6-45 shows the model at this point.

Step 5: Pocket Feature A in Fig 6-33

1. Define the sketch plane on A in Fig 6-46 and create 4 circles as shown.

2. Apply the **Concentricity** constraint between the circular edge and the circle.

3. Create the diametral dimension for each circle.

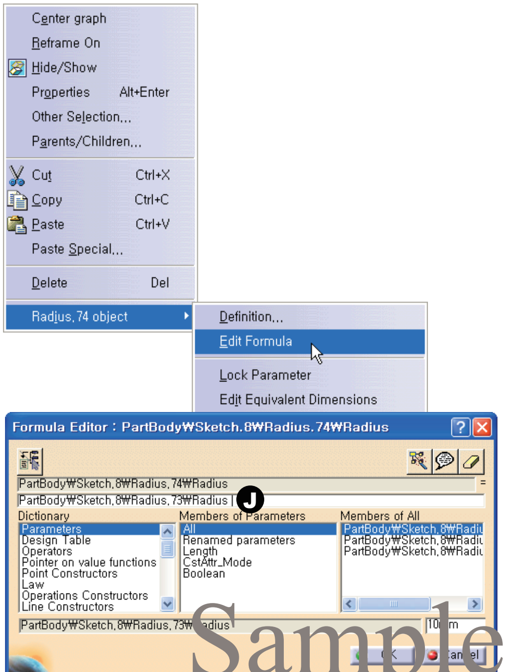


Fig 6-47 Applying Formula

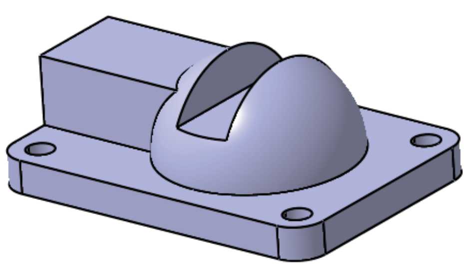


Fig 6-48 Completed Model

4. Modify the dimension designated by **B** in Fig 6-46 as 20mm

5. Press MB3 on the dimension **C** in Fig 6-46.

6. Select **Edit Formula** from the pop-up menu of the object.

7. Select the dimension **B** in Fig 6-45. The selected dimension variable appears in the input area **J** in Fig 6-47.

8. Press **OK**. The dimension **C** in Fig 6-46 is modified to 20 and the function symbol f(x) is appended.

9. Link the dimensions **D**, **E** and **F** to the dimension **B** in the same way.

11. Exit the **Sketcher** and create holes using the **Pocket** command.

Fig 6-48 shows the completed model.

END of Exercise

Verifying the Requirement ②

① Double click the sketch of the hemisphere.
② Modify the radius to 90mm.
③ Exit the **Sketcher**.

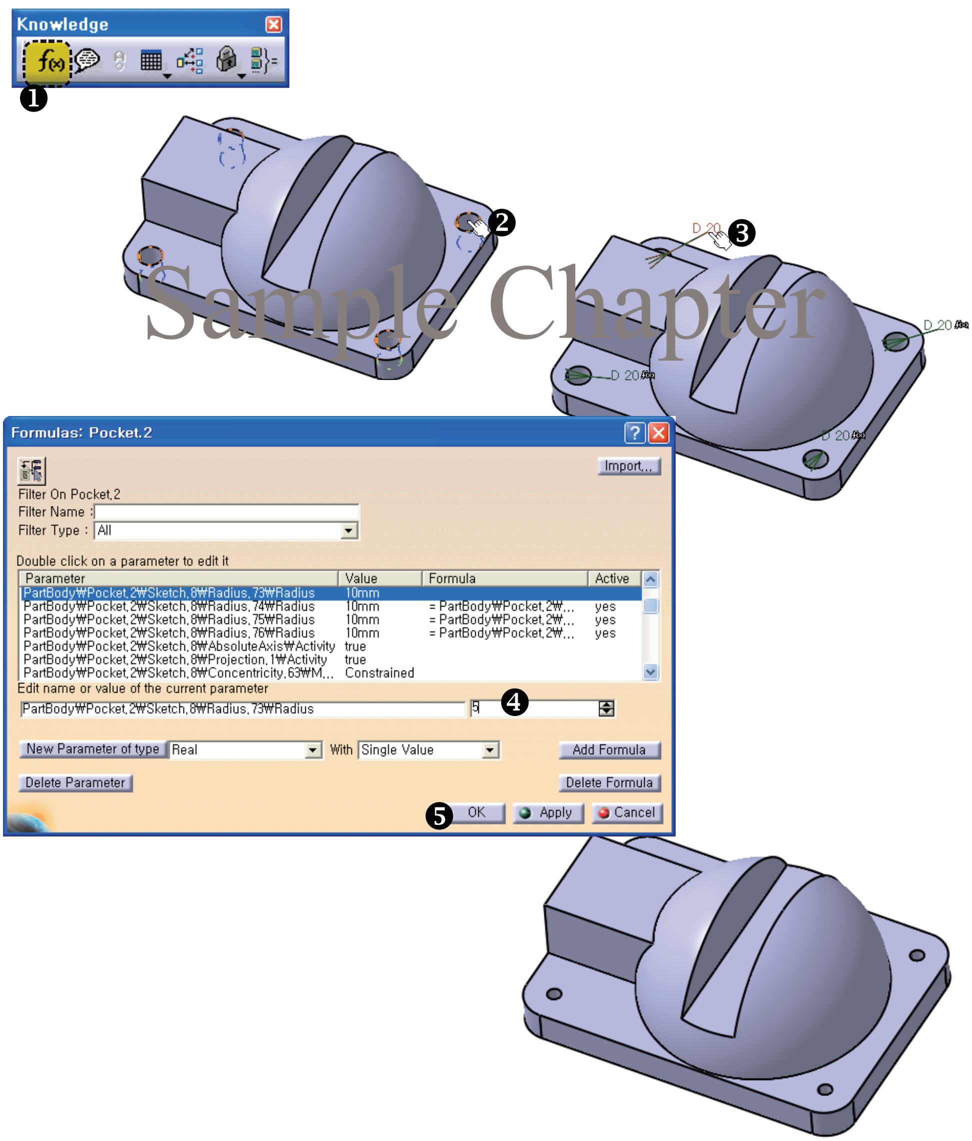
The **Shaft** feature **B** in Fig 6-33 is created successfully because we have created an intersection curve with the spherical surface.

Verifying the Requirement ③

① Method 1: Modify the sketch dimension ❸ in Fig 6-46.

② Method 2: Use the **Formula** icon in the **Knowledge** toolbar.

The following figure shows the procedure to modify the dimension value using the **Formula** icon. Input a new value in the input box ❹ and press **OK**.



Exercise 07 Guide Bracket

Create a guide bracket referring to the drawing in Fig 6-49.

General Modeling Procedure (Refer to p.51)

① Create the features that add material.
② Create the features that remove material.
③ Complete the model by applying detailed features.

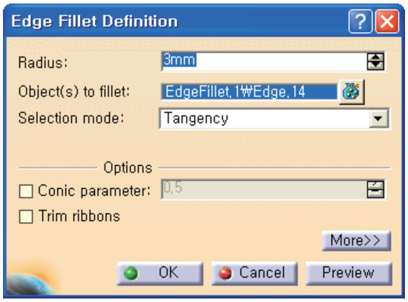
! Fillet

A fillet makes sharp edges rounded.

A convex fillet is a measure to make an exterior look aesthetic. If you manufacture a part from a mold, convex fillets are naturally generated because the mold cavity is machined with a rounded drill. If you want to generate a sharp edge with a mold, you have to finish the rounded toolpath with a flat end mill.

A concave fillet is used to relieve stress concentration to avoid fracture of a part.

The **Fillet** feature designated by Ⓐ in Fig 6-49 can be created by using the **Edge Fillet** command available in the **Dress-Up Features** toolbar.



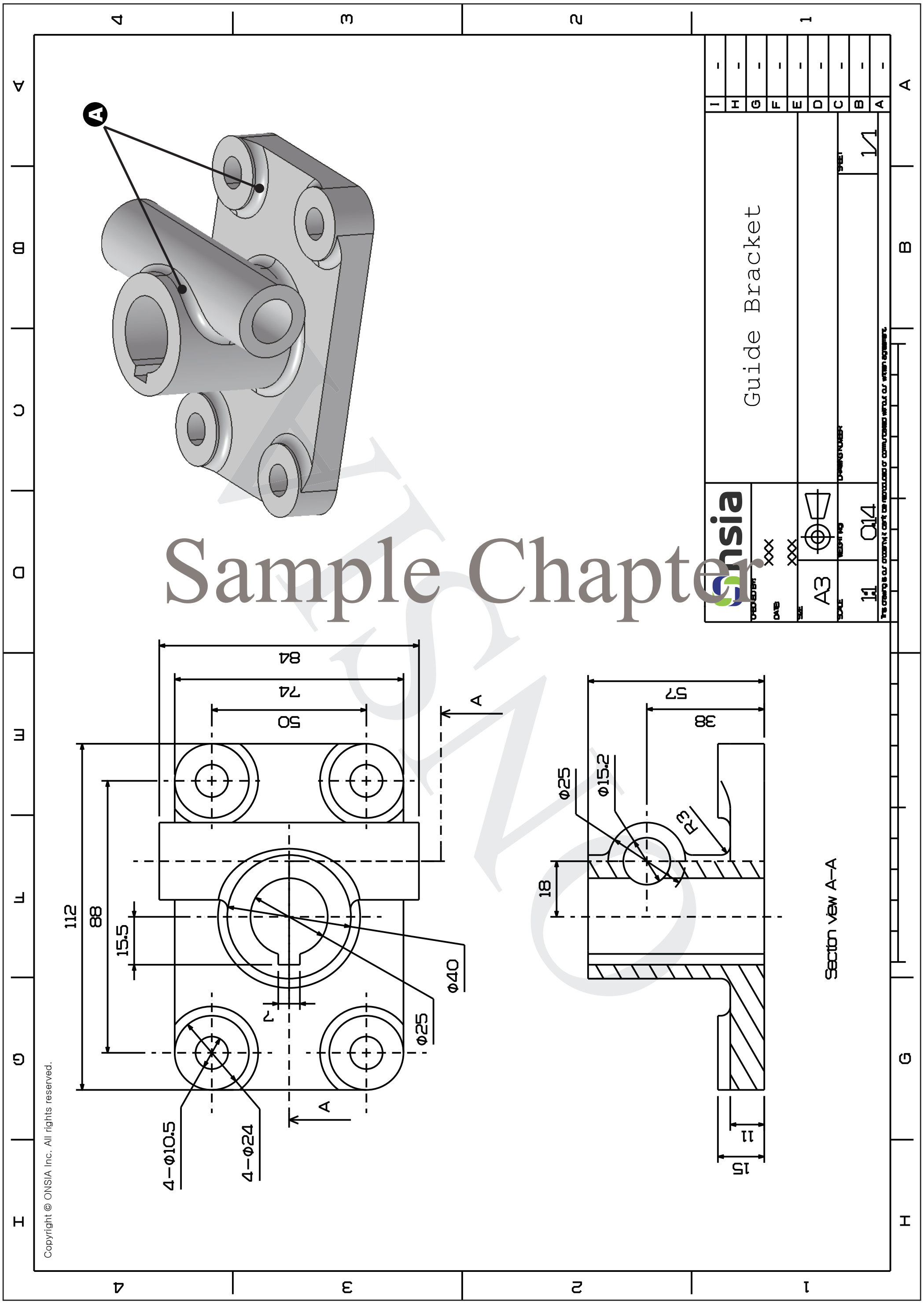


Fig 6-49 Guide Bracket

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