CATIA for Design and Engineering
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This tutorial will demonstrate the creation of extruded features within CATIA. An extruded feature is defined as a profile swept a straight direction a user-defined distance. CATIA’s Pad command is used to create positive spaced extrusions (mass), while the Pocket command is used to create negative space or cut features.

The following topics will be covered within this tutorial:

- Setting CATIA options
- Viewing toolbars
- Sketching geometry
- Using the Profile tool to create sketch geometry
- Creating dimensional constraints
- Creating geometric constraints
- Creating a Pad feature
- Creating a Pocket feature
STARTING CATIA AND SETTING OPTIONS

The first section of this tutorial will show steps for starting a new part file, setting modeling units, and viewing toolbars.

Step 1:  Start CATIA

Step 2:  Select FILE >> NEW (or select the New icon on the toolbar)
Step 3: Select PART on the New dialog box’s list of available work benches then pick OK.

NOTE: If you receive an additional New Part dialog box, check the Do not show this dialog at startup option then select OK.

Step 4: Select TOOLS >> OPTIONS on the menu bar to open the Options dialog box.

The Options dialog box is used to customize a variety of CATIA’s workbench options.

Step 5: Under GENERAL, select the PARAMETERS AND MEASURE option (Figure 2).

Step 6: Select the Units tab then set your unit’s length value to inch (in) (Figure 2).

Step 7: On the Options dialog box under MECHANICAL DESIGN, select SKETCHER.

Step 8: Under the Sketcher tab, ensure that the Grid options Display and Snap-To-Point are NOT selected (Figure 3).

Step 9: Ensure that the Constraint option Creates the Geometric Constraints is selected.

Step 10: Select OK on the Options dialog box.

Figure 4 Toolbar Options
PART DESIGN WORKBENCH
TOOLBAR OPTIONS

CATIA has a variety of toolbars that can be displayed are closed depending on the needs of the designer. Within this tutorial, you will be utilizing toolbars that relate directly to the sketching and modeling of extruded features. Toolbars can be dragged and repositioned around the perimeter of the work screen. Some toolbars may actually be hidden at the bottom of the right column.

Step 1: Select VIEW >> TOOLBAR on the menu bar
Step 2: Ensure that the Part workbench toolbars selected in Figure 4 are check

SKETCHING THE BASE PROTRUSION

This section of the tutorial will detail how to create the sketch shown in Figure 5. You will utilize the following toolbars:

Step 1: On the Sketcher Toolbar, select the Sketcher Icon
Step 2: On the Specification Tree (Figure 6) pick the XY PLANE

You will sketch the first profile for the base protrusion on the XY Plane of the default reference planes. You can also pick suitable sketching planes directly within the graphics window.

Once you select a sketching surface, your sketch plane will orient parallel to your work screen.

Step 3: Select VIEW >> TOOLBARS and ensure that the Sketch Tools toolbar is selected
Step 4:  \[\text{Select the PROFILE tool on the Profile Toolbar}\]

The Profile option allows you to create a continuous chain of line or circular entities. To cancel the profile command, double-pick your last vertex point or pick the Select icon.

**Modeling Strategy**

CATIA’s sketching environment allows for the capturing of product design intent through a variety of geometric and dimensional constraints. While sketching, attempt to capture any geometric constraints dynamically. The following is a partial list of available geometric constraints:

- Coincidence – Aligns a vertex with another vertex or with a line or axis
- Tangent – Aligns an Arc or Circle tangent with a line or circular entity
- Vertical – Snaps a Line entity vertical within your sketching environment
- Horizontal – Snaps a Line entity horizontal within your sketching environment
- Parallelism – Defines a line entity parallel to another line entity
- Perpendicular – Defines a line entity perpendicular to another line entity
- Concentricity – Defines a circular entity concentric with another circular entity.

Step 5:  \[\text{Using the Profile tool, sketch the three line entities shown in Figure 7}\]

Perform the following sub-steps while completing this task:

1. Ensure that your first vertex is coincident with the Y-Axis of the default coordinate system. Your mouse cursor should snap to the Y-Axis dynamically while sketching.
2. Ensure that your first line entity snaps horizontal.
3. Ensure that your second line entity snaps vertical.
4. Ensure that your third line entity snaps horizontal.
5. Ensure that your last vertex is coincident with the Y-Axis of the default coordinate system.
6. Double-Pick your last vertex or select the Pick icon to terminate the Profile command.
Modeling Strategy

If you don’t capture the geometric constraints as defined in Step 5, you can explicitly create geometric constraints with the Constraints Definition dialog box option. Using your Control-Key, pre-select any required geometry entities (e.g. Axis, Lines, Vertices, Center-Point, etc) then select the Constraints Defined in Dialog Box option on the Constraints toolbar. Valid geometric constraints will be available for definition.

Step 6: Select the THREE-POINT ARC WITH LIMITS option

The Three-Point Arc With Limits option requires the picking of the two ends of an arc, followed by the dragging of the arc’s radius.

This option is located on the sketcher workbench’s Circle toolbar. Many toolbar options are hidden behind the last selected icon on its respective toolbar. You can expand a compact toolbar by picking the down-arrow in the bottom right-hand corner of one of its icons.

Step 7: On the work screen, pick the two arc endpoints as shown in Figure 8

Ensure that you dynamically create coincidence constraints at each arc endpoint

Step 8: For the third point of the arc, drag to create the tangent constraint shown in Figure 9

Drag the limit of the arc until a tangent constraint is created at the arc endpoints. If a tangent constraint was not created dynamically while defining the arc entity, explicitly create one with the Constraints dialog box.
Figure 9 Tangent Constraint

Figure 10 Line Dimensional Constraints

Figure 11 Linear Distance Dimension
Step 9: Use the Select tool to pick the line entity shown in Figure 10

Step 10: Select the CONSTRAINT tool

The Constraint tool is used to create dimensional constraints. The first dimension will define the width of the feature. Within this tutorial you will create this dimension by defining the length of the selected line. Another option would be defining the distance between the two parallel horizontal and vertical lines. Within CATIA’s sketching work bench, any parallel lines picked for dimensioning will create a linear dimension. Any non-parallel lines will create an angular dimension.

Step 11: Place the dimension as shown in Figure 10

Step 12: Create a linear dimensional constraint between the vertical line and V-Axis as shown in Figure 11

Pick the vertical line then pick the V-Axis of the default coordinate system. Place your dimension as shown.

Step 13: Pick the SELECT icon

Notice how most of your sketch entities are white. White represents an unconstrained or partially constrained entity. Next, you will use your cursor to drag the sketch, revealing any unconstrained degrees of freedom. Your current sketch should have a linear degree of freedom along the Y-Axis.

Step 14: On the sketch, use your cursor to drag the bottom most horizontal line.

Drag to resemble Figure 12. Note, you may have one or two degrees of freedom to drag. Next, you will create a coincident constraint between the center vertex of the arc entity and the center point of the default coordinate system.

Step 15: Using your Control-Key, pre-select the center vertex of your arc and the center point of the default coordinate system (Figure 13)

Figure 12 Dragged Sketch
Step 16: Select the CONSTRAINTS-DEFINED IN DIALOG BOX icon

Step 17: Select the COINCIDENCE constraint then select OK

Your sketch should be Green, the default color of a fully defined sketch.

Step 18: Create a Circle entity as with the circle’s center vertex coincident with the center of the default coordinate system

Step 19: Select the Constraints tool

Step 20: Create a diameter dimension for the circle entity (Figure 14)

Step 21: Use the SELECT option to modify the dimension values to match Figure 14

A dimensions value can be modified by double picking the value on the work screen with your cursor. Modify the following dimensions:

- Horizontal Linear Dimension = 4.00
- Vertical Linear Dimension = 3.00
- Diameter Dimension = 1.50
Step 22: When your sketch is complete, exit the Sketcher workbench

Step 23: Save your part file (File >> Save)

CATIA part files are given a *.CatPart file extension.

Step 24: Your model should resemble Figure 15

Notice on your screen how the Specification Tree tracks features and entities, including sketches. Sketches can be modified by double-picking the sketch node on the tree.

CREATING A PAD FEATURE

This section of the tutorial will create a Pad feature by extruding the previously created sketch. You will utilize the following toolbar within this tutorial:

Step 1: Pick the Sketch.1 node on the Specification Tree

You can also pick the sketch on the work screen. A sketch must be selected for the Pad command is available.

Step 2: Select the PAD icon on the Sketch-Based Features toolbar

Step 3: On the Pad Definition dialog box, select DIMENSION as the First Limit Type (Figure 16)

The Dimension type will extrude the feature a user defined distance. Other types include Up To Last, Up To Next, Up to Plane, and Up to Surface.
CREATING EXTRUDED FEATURES

Step 4: Enter 1.50 as the First Limit Length value

On the dialog box, notice that Sketch.1 is listed under the Profile/Surface Selection box. This is the selected sketch for the extruded pad. If needed, you can edit the sketch by selecting the Sketch icon.

Step 5: Select the PREVIEW option and observe your model on the work screen.

Step 6: Select OK on the dialog box

Step 7: Save your part

Step 8: Observe the part feature on the Specification Tree

Your model should look similar to Figure 17.

Step 9: Use the following dynamic viewing capabilities to view your model

- **Dynamic Pan Middle** - Mouse Button
- **Dynamic Rotation Middle** - Mouse Button & Right-Mouse Button
- **Dynamic Zoom Middle** - Mouse Button & Right-Mouse Button, then release Right-Mouse Button

**Modeling Strategy**

Amongst other capabilities, the Specification Tree is used to track part features. Pan and Zoom viewing options can be used to reposition and resize your tree. Selecting the tree’s primary truck will activate the viewing tools for the tree. Selecting the truck a second time will deactivate focus on the specification tree. The View >> Command List.. >> Swap Tree/View option can also be used to switch the view focus.
SKETCHING A POCKET FEATURE

This section of the tutorial will detail how to create the sketch for the first Pocket feature (Figure 18). You will create this sketch on the ZX PLANE. When complete, the next section of this tutorial will have you extrude this sketch with the Pocket tool to create a two directional cut feature.

Step 1:  Select the SKETCHER tool

Step 2:  As shown in Figure 19, use the Specification Tree to pick the ZX-Plane

The sketch for the pocket feature will be created on the ZX-Plane. In the next section of this tutorial the sketch profile will be extruded two directions through the extents of the part. You can also pick sketching planes directly on the work screen.

Step 3:  Select the PROFILE tool on the Profile Toolbar

Step 4:  Sketch the first line horizontal and the second line vertical (Figure 20)

Locate the ends of the profile off the existing part as shown in the figure. To end the profile command, double-pick the last endpoint or select the Selection icon.

Step 5:  Using the control-key, pre-select the end vertex of the first line and the vertical part edge (Figure 21)

Next, you will create a coincidence constraint between these two entities.

Step 6:  Select the CONSTRAINT DEFINITION icon on the Constraint toolbar

Step 7:  Select the COINCIDENCE constraint option then select OK

Notice the “O” character adjacent to the line endpoint. This symbol represents a coincidence constraint.

Step 8:  Create a Coincidence constraint for the other profile endpoint and its respective part edge

Step 9:  Select the Constraint icon

![Figure 18 Sketch for Pocket Feature](image1.png)

![Figure 19 Sketching Plane Selection](image2.png)
Step 10: Create the two Dimensions shown in Figure 22

Step 11: Modify each dimension’s value

Using the Selection tool, double-pick each dimension value and modify according to the following values:

- Vertical Dimension = 0.75
- Horizontal Dimension = 2.50

Step 12: Select the ISOMETRIC VIEW icon on the View toolbar

The View toolbar is usually docked at the bottom of the geometry area. Observe your sketch from a three-dimensional vantage point.

Step 13: Select EXIT WORKBENCH icon to exit the sketching environment

Step 14: Save your part
CREATING A POCKET FEATURE

This section of the tutorial will create a Pocket feature that is extruded both directions from the sketching plane.

Step 1: On the Specification Tree, pick the last Sketch (Sketch.2)

Note: This sketch may already be selected

Step 2: Select the POCKET tool on the Sketch-Based Features toolbar

Step 3: On the Pocket dialog box, select the MORE option (Figure 23)

By default, a Pocket or Pad feature is extruded in one direction from the sketching plane. The Second Limit group of options on the Pocket definition dialog box is used to define a second direction. For this Pocket feature, you will utilize an Up-To-Last Limit Type for both directions.

Step 4: For the First Limit, select the UP-TO-LAST type

The Up-To-Last depth option is similar to the Through All option found in many CAD systems. It will extrude a feature completely through a part.

Step 5: For the Second Limit, select the UP-TO-LAST type

Step 6: On the Pocket Definition dialog box, select the SKETCHER icon

You can edit the definition of a sketch directly from a Sketch-Based Feature’s dialog box.

Step 7: Modify the horizontal dimension to have a value of 2.25

Step 8: Exit the workbench to return to the Pocket Definition dialog box

Step 9: Select OK on the Pocket Definition dialog box

Step 10: Save your part

Your part should resemble Figure 24.
CREATING A SECOND POCKET FEATURE

This section of the tutorial will create the Pocket feature shown in Figure 25.

Step 1: 
Pick the planar surface defined by the last pocket feature (Figure 26)

Your next feature will be sketched on the planar surface shown in the figure.

Step 2: 
Select the SKETCHER icon

Step 3: 
Select the PROFILE icon on the Profile toolbar

Step 4: 
Sketch the single horizontal line shown in Figure 27 (don’t terminate the profile command after the second endpoint)

Step 5: 
Select the TANGENT ARC icon on the Sketch Tool dialog box

The Tangent Arc option is used to create a tangent arc dynamically within a continuous profile.

Step 6: 
Sketch the Tangent Arc and the horizontal line as shown in Figure 28
First Endpoint

Second Endpoint
(Don't terminate Profile after Picking this point)

Figure 27 Profile Tool

Figure 28 Profile Geometric Elements

Figure 29 Coincidence Constraints
Step 7: Create the two Coincidence constraints shown in Figure 29

Elements to be constrained must be pre-picked before selecting the Constraints-Definition option. Separately, pre-select a line endpoint and its respective part edge. Use the Constraint Definition dialog box to create the Coincidence constraint.

Step 8: Create the three linear dimensional constraints shown in Figure 30

Step 9: Modify the three dimension’s values

Modify each vertical dimension to have a value of 1.0 and modify the horizontal dimension to have a value of 1.50.

Step 10: Exit the Sketcher Workbench

Step 11: Select the POCKET tool on the Sketch Based Features toolbar

If necessary, pick Sketch.3 on the Specification Tree.

Step 12: On the Pocket Definition dialog box, select the UP-TO-LAST Limit Type (Figure 31)
Step 13: Use the REVERSE DIRECTION option on the dialog box to define a cut direction that extrudes through the part (Figure 32)

Step 14: Select OK on the Pocket Definition dialog box

Step 15: Save your part document

Figure 32 Extrude Direction
MODELING PROBLEMS

Figure 33 Problem One

Figure 34 Problem Two