part design

مثالی از ایجاد مدل حجمی در فضای

در این مثال گام به گام مراحل ایجاد مدل شکل زیر ذکر گردیده است بنا برای همراه با توضیحات

اگری شده اقدام به اجرای دستورات مربوطه کنید.

How to deal with a mechanical part design, starting « from scratch »
First, you are going to open the Part Design Workbench.

1. Select the “Start” menu.

2. Select the “Part Design” workbench. Maximize window size.

The Part Design Workbench is open, with its dedicated toolbar.
Create a Pad Feature

- You are now going to choose the plane on which you will draw your first sketch.

- In Tools / Options, set the Part / Sketcher values to 20 for Graduations and 100mm for Primary spacing.

1. Select the XY plane.

2. Select the “Sketcher” icon.

The XY plane comes into the screen plane. A grid is displayed.
You are now going to sketch a circle that you will use later to create a pad feature.

1. Select the "Circle" icon.
2. Select the origin point as circle center.
3. Select about 6 grid squares from the circle center.
4. Select the "Constraint" icon.
5. Select the dimension position inside or out of the circle.
You are going to modify the circle diameter.

1. **Double-click** on the circle diameter dimension value.

   The “**Constraint Edition**” dialog box is displayed.

2. Key in **48mm** as diameter value.

3. Select **OK** to confirm the modification.
Create a Pad Feature

- You are going to leave the Sketcher, and create a Pad feature.

1. Select the “Exit” icon.

2. Select the “Pad” icon.
   - The “Pad Definition” dialog box is displayed.

3. Key in 41mm as Pad length.

4. Select OK to confirm.
Design a Motor Piston

Create two symmetric cut-outs

You are now going to create a second part body. This body will be later duplicated to perform 2 cut-outs on the first Pad feature.

1. Select the "Insert" menu.
2. Select "Body"

The features that will be created in this new Part body will be stored in a separate feature editor branch.
Design a Motor Piston

Create two symmetric cut-outs

- You are going to select the plane on which you will create the next sketch.

1. Select the ZX plane.
2. Select the "Sketcher" icon.

The ZX plane comes into the screen plane. The grid is displayed again.
Create two symmetric cut-outs

You are going to sketch a new profile.

1. Select the “Profile” icon.

2. Sketch the profile as indicated.
   - The starting point may be any of the lines ends.
   - The profile is automatically closed when it loops back on the starting point.
Create two symmetric cut-outs

- You are going to define the right profile elements dimensions.

1. Double-click on the “Constraint” icon.

   A double-click on an icon “locks” the command.

2. Place the cursor on the lower line as indicated, and press the mouse button 3 to access the contextual menu.

3. Select the “Fix” option.

   An anchor symbol shows that the line position is fixed.
Create two symmetric cut-outs

- You are going to create length and distance dimensions.

- (1) Select the indicated vertical line.
- (2) Select on the left side of the sketch to define the dimension location.
- (3) Repeat the same operation for the lower fixed horizontal line.
- (4) Select the left vertical line, and the yellow “V” vertical axis, and place the dimension under the last one.
Create two symmetric cut-outs

- You are going to create an angle dimension, and adjust its value.

1. Select the 2 indicated lines.
2. Select on the sketch left side to locate the dimension.
3. Double-click on the dimension value, and set it to 15deg in the “Constraint Edition”.
4. Select OK to confirm.
Create two symmetric cut-outs

- You are going to modify the sketch dimensions.

1

(1) Repeat the last operation on the other dimensions to specify the indicated values.
(2) Select the “Exit” icon.
Create two symmetric cut-outs

- You are now going to create a new Pad feature.

1. (1) Select the "Pad" icon.
2. (2) Select the "Mirrored extent" in the "Pad Definition" dialog box, and Select OK to confirm.

This second Pad is still independent from the first one.
Create two symmetric cut-outs

You are going to make a symmetric copy of the Pad feature you just created.

1. Select the "Mirror" icon.
2. Select the YZ plane.
3. Select OK in the "Mirror Definition" dialog box to confirm.
Create two symmetric cut-outs

You are now going to remove the mirror operation result from the first Pad feature.

1. Place the cursor on "Body.1" in the Feature Editor, and press the mouse button 3 to access the contextual menu.

2. Go down to the last line "Body.1 object", and choose the "Remove" option.
Create a double boss

You are going to sketch a profile that you will use to create a double boss.

1. (1) Select the YZ plane, and then the “Sketcher” icon.

2. (2) Select the “Circle” icon, and create a circle as indicated.

3. (3) Select the “Constraint” icon, and create a diameter dimension on the circle.

4. (4) Double-Click on the dimension value to set it to 18mm in the “Constraint Edition” dialog box, and select OK to confirm.
Create a double boss

(1) Select the "Constraint" icon and create a distance dimension between the circle center and the yellow "H" horizontal axis.

(2,3,4) Double-Click on this dimension value to set it to 20mm in the "Constraint Edition" dialog box, and select OK to confirm.

(5) Select the "Exit" icon.
Create a double boss

You are going to create a new Pad Feature.

1. Select the ‘Pad’ icon.
2. (2,3,4) Select the “Mirrored extent” in the “Pad Definition” dialog box, and Select OK to confirm.
You are going to sketch a profile that you will use to create a cut-out.

(1) Select the indicated part face.
(2) Select the “Sketcher” icon.
(3) Select the “Three Point Arc” icon.
(4) Select the 3 indicated A, B, C points to create the arc.
(5) Select the “Exit” icon.
Perform a cut-out

You are now going to make the cut-out itself.

1. Select the "Pocket" icon.
2. Select the indicated arrow to invert its direction.

The highlighted area of the part will be removed.

3, 4. In the menu available under the indicated arrow, choose the "Up to next" option, and select OK to confirm.
Create a Pocket Feature

You are going to sketch a profile that you will use to create a pocket.

1. Select the indicated part face.
2. Select the "Sketcher" icon.
3. Select the "Profile" icon.
4. Sketch the profile as indicated.

Use the "Three point arc" icon to switch from line to circle arc. This icon is located on the lower icon bar of CATIA V5.
Create a Pocket Feature

- You are now going to specify the sketch dimensions.

1. Double-click on the “Constraint” icon.
2. Create a radius dimension on each arc.
3. Select the top circular edge and circle arc as indicated. Using mouse button 3, access the contextual menu and select the “Concentricity” option.
4. Repeat the same operation between the lower circular edge and circle arc.
(1) **Double-Click** on the top radius dimension, to set it to **20mm**.

(2) **Double-Click** on the lower radius dimension.

(3) In the **"Constraint Edition"** dialog box, place the cursor in the radius value field. Using mouse button 3, access the contextual menu and select the "**Edit formula**" option.
(1) Select the top radius dimension, and **OK** to confirm in the "**Formula Editor**" and "**Constraint Edition**" dialog boxes.

*This dimension is now linked to the other one by a simple formula.*
Create a Pocket Feature

1. Create the 2 distance dimensions between the “V” vertical axis and the 2 profile vertical lines, as indicated.

2. Repeat the formula creation between these 2 dimensions.

3. Set the leading dimension of this relation to 14mm.

4. Select the “Exit” icon.
Create a Pocket Feature

You are now going to make the Pocket Feature, using the profile you just created.

1. (1) Select the "Pocket" icon
2. (2) Set the depth to 37 mm in the "Pocket Definition" dialog box.
3. (3) Select OK to confirm.
You are going to create a through-hole, where the piston axis will take place.

1. Select the “Hole” icon.
2. Select the indicated circular face.
3. Selecting the indicated arrow button in the “Hole Definition” dialog box, choose the ‘Up to Last’ hole option. Set the hole radius to 12 mm. Select OK to confirm.

The centered through-hole is created
Add rounding fillets

1. You are now going to create rounding fillets.

2. (1) Select the 4 indicated edges using the keyboard "Ctrl" key.

   The "Ctrl" key allows elements multi-selection.

3. (2) Select the "Edge Fillet" icon.

4. (3) Set the radius value to 3mm in the "Edge fillet Definition" dialog box.

(4) Select OK to confirm.
(1) Select the pocket bottom face.

(2) Select the "Edge Fillet" icon.

(3) Set the radius value to 5mm in the "Edge fillet Definition" dialog box.

(4) Select OK to confirm.
Add rounding fillets

1. Repeat the same operation on the 2 indicated edges (Radius 2mm).

2. Repeat the same operation on the 2 other indicated edges (Radius 2mm).

Fillets are propagated on chained tangent edges.
(1) Select the 2 indicated small cylindrical faces, pressing the keyboard CTRL key, and repeat the Fillets creation operation (Radius 1mm).

Congratulations! You went through the whole Piston Design phase.
At any time you can modify the Part graphic properties.

1. Place the cursor on the “PartBody” line in the Features Editor.
2. Using mouse button 3, access the contextual menu and choose the “Properties” option.
3. In the “Properties” dialog box, select the “Graphic” option, and choose a new fill colour.