

Laboratory 0

A Tutorial of AutoCAD Functions

This is an optional laboratory aimed at providing a brief review of various AutoCAD functions. The laboratory was originally used in ENGR150 at UVic. The tutorial covers most of the fundamental functions of AutoCAD for modeling a mechanical component. A procedure for creating the part model, shown in Figure 1, is listed in this section.

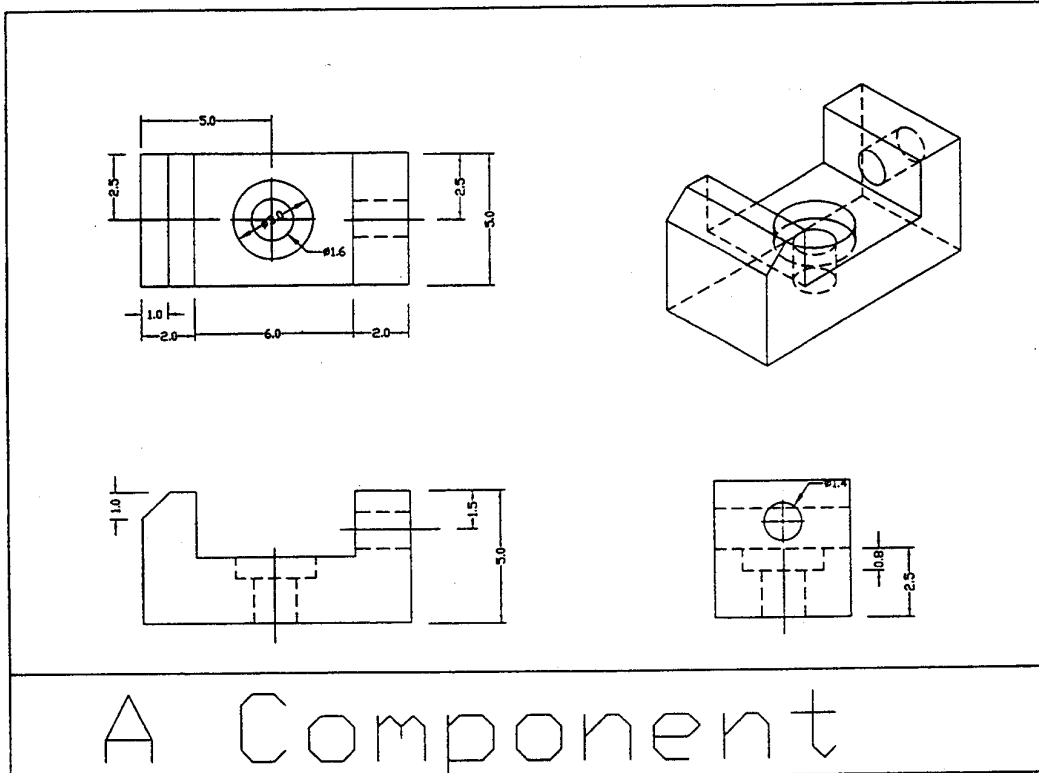


Figure 1 A Mechanical Component

The commands, messages, and input data are listed at the left side of the instructions, while explanations of these commands are given at the right side of the instructions. The messages of AutoCAD are described using Arial Narrow font. Actions to execute the commands are given in *italic* font. The text commands and data entered by the users using keyboard are described using **bold** font. Explanations are described with underlines. Some instructions do not provide complete data due to their simpleness.

1. Starting AutoCAD system

You may run either AutoCAD R12 on the UNIX, or AutoCAD R13 on the Windows NT workstations.

2. Modeling the base part of the component

Command: *Choose* Draw, Polyline

Draw a 2-D Polyline

From point: **0,0**

Current line-width is 0.0000

Arc/Close/Halfwidth/Length/Undo/Width/<Endpoint of line>: **10,0**

Arc/Close/Halfwidth/Length/Undo/Width/<Endpoint of line>: **10,5**

Arc/Close/Halfwidth/Length/Undo/Width/<Endpoint of line>: **8,5**

Arc/Close/Halfwidth/Length/Undo/Width/<Endpoint of line>: **8,2.5**

Arc/Close/Halfwidth/Length/Undo/Width/<Endpoint of line>: **2,2.5**

Arc/Close/Halfwidth/Length/Undo/Width/<Endpoint of line>: **2,5**

Arc/Close/Halfwidth/Length/Undo/Width/<Endpoint of line>: **1,5**

Arc/Close/Halfwidth/Length/Undo/Width/<Endpoint of line>: **0,4**

Arc/Close/Halfwidth/Length/Undo/Width/<Endpoint of line>: **close**

Command: *Choose* Draw, Solids, Extrude

Select objects: *Select the polyline*

Select objects:

Path/<Height of Extrusion>: **5**

Extrusion taper angle <0>: **0**

Command: *Choose* View, 3D Viewpoint Presets, SW Isometric

Command: *Choose* View, 3D Viewpoint Presets, Top

Command: *Choose* View, Pan, Point

Displacement: **0,0**

Second point: **1,1**

Create a closed 2-D section

Extrude the 2-D polyline

Display the isometric view

Return to the top view

Move the object to the center

3. Creating the stepped hole

Command: *Choose* Draw, Polyline

From point: **5, -0.5**

Current line-width is 0.0000

Arc/Close/Halfwidth/Length/Undo/Width/<Endpoint of line>: **4.2, -0.5**

Arc/Close/Halfwidth/Length/Undo/Width/<Endpoint of line>: **4.2,1.7**

Arc/Close/Halfwidth/Length/Undo/Width/<Endpoint of line>: **3.5,1.7**

Arc/Close/Halfwidth/Length/Undo/Width/<Endpoint of line>: **3.5,3**

Arc/Close/Halfwidth/Length/Undo/Width/<Endpoint of line>: **5,3**

Arc/Close/Halfwidth/Length/Undo/Width/<Endpoint of line>: **close**

Command: *Choose* Draw, Solids, Revolve

Select objects: *Select the polyline*

Select objects:

Axis of revolution - Object/X/Y/<Start point of axis>: **5,5**

Draw a Polyline

Revolve the 2-D polyline

<End point of axis>: **5,3**

Angle of revolution full circle>:

Command: *Choose* View, 3D Viewpoint Presets, Back

Change viewpoint

Command: *Choose* Modify, Move

Move the object to correct position

Select objects: *Select the circle*

Select objects:

Base point or displacement: **0,0,0**

Second point of displacement: **0,0,2.5**

Command: *Choose* Construct, Subtract

Subtract object to create a stepped hole

Select objects: *Select the base*

Select objects:

Select solids and regions to subtract...

Select objects: *Select the stepped hole*

Select objects:

4. **Creating the straight hole**

Command: *Choose* View, 3D Viewpoint Presets, Right

Change viewpoint

Command: *Choose* View, Preset UCS ...

Change User Coordinate System

Select right icon and click OK

Command: *Choose* Draw, Circle, Center & Radius

Draw a circle

3P/2P/TTR/<Center point>: **3.5,2.5,10.5**

Diameter/<Radius>: **0.7**

Command: *Choose* Draw, Solids, Extrude

Extrude the circle

Select objects: *Select the circle*

Select objects:

Path/<Height of Extrusion>: **-3**

Extrusion taper angle <0>: **0**

Command: *Choose* Construct, Subtract

Subtract the object to create a hole

Select objects: *Select the base*

Select objects:

Select solids and regions to subtract...

Select objects: *Select the straight hole*

Select objects:

5. Rotating the created model

Command: *Choose* Modify, Rotate

Select objects: *Select the component*

Select objects:

Base point: **0,0**

<Rotation angle>/Reference: **90**

Command: *Choose* View, Preset UCS...

Select world icon and click **OK**

Command: *Choose* View, 3D Viewpoint Presets, SW Isometric

Rotate the component

Return to World Coordinate System

Display the isometric view

6. Generating 3-D images

Command: *Choose* Tools, Hide

Command: *Choose* Tools, Shade, 256 Color

Command: *Choose* Tools, Render, Render

Click Render Scene *button*

Command: **regain**

Display image without hidden lines

Display shading image

Display rendering image

Display wireframe image

7. Creating multiple viewports

Command: *Choose* Data, Layers...

Enter a layer name **BORDER** *and click* New *button*

Select layer BORDER *on the list and click* Current *button*

Click **OK** *button*

Command: *Choose* View, Paper Space

Already in paper space.

Command: **mview**

ON/OFF/Hideplot/Fit/2/3/4/Restore/<First Point>: **4**

Fit/<First Point>: **1,1**

Second point: **12,8**

Regenerating drawing.

Command: *Choose* View, Floating Model Space

Select the top left viewport

Command: *Choose* View, 3D Viewpoint Presets, Top

Command: *Choose* View, Zoom, Out

Select the bottom left viewport

Create a layer for drawing borders

Change to paper space

Create multiple viewports

Enter the number of viewports

Change to floating model space

Select the top view

Change viewport

Reduce the size of display

Select the front view

Command: *Choose View*, 3D Viewpoint Presets, Front

Command: *Choose View*, Zoom, Out

Select the bottom right viewport

Command: *Choose View*, 3D Viewpoint Presets, Right

Command: *Choose View*, Zoom, Out

Command: *Choose Data*, Linetype...

Click Load... button

Select HIDDEN2 on the list and click OK button

*Click **OK** button*

Select the top view

Command: **solprof**

Initializing...

Select objects: *Select the component*

Select objects:

Display hidden profile lines on separate layer? <Y>:

Project profile lines onto a plane? <Y>:

Delete tangential edges? <Y>:

One solid selected.

Using the same method to create profile lines for the other three viewports

8. Defining dimensions on the top view

Select the top view

Command: *Choose Data*, Layers...

Create three new layers called DIMTOP, DIMFRONT, and DIMRIGHT, and make DIMTOP the current layer

Select layer 0 and click OFF button

Select a layer with a name started as PH-

and click Set Ltype... button

Select HIDDEN2 and click OK button

In the same way, change the linetype of the other three layers

with the names started as PH-to HIDDEN2

Select DIMFRONT layer and click Cur VP: Frz button

Change viewpoint

Select the right view

Change viewpoint

Load HIDDEN2 linetype

Generate the profile lines of 3 views

Select the top view

Create 3 avers for model dimensions

Turn wireframe model of the part off
Change hidden lines to dashed lines

Turn off the layer in this view

Select DIMRIGHT layer and click Cur VP: Frz button
Click OK button
Command: Choose View, Preset UCS...
Select CURRENT VIEW icon and click OK
Command: Choose Data, Linetype...
Click Load... button
Select CENTER2 on the list and click OK button
Click OK button
Command: Choose Draw, Line
Draw the centerline of the straight hole
Command: Choose Modify, Properties...
Select objects: Select the created line
Select objects:
Select Linetype... button
Select CENTER2 and Click OK button
Click OK button
Command: Choose Data, Dimension Style...
Click Geometry... button
Define Arrowheads size as 0.3, choose Line button of
Center item, and click OK button
Click Format... button
In the Text item, unselect button of Inside Horizontal
and click OK button
Click Annotation... button
Click Units button
In the Precision item, select 0.0 and click OK button
In the Text item, select Style as Standard and height as 0.3,
and click OK button
Click Save button
Click Radial button in Family item, and click Format... button
In Fit item, select Leader; and click OK button
Click Save button
Click Diameter button in Family item, and click Format... button

Turn off the layer in this view

Set UCS the same as in this view

Load CENTER2 linetype

Draw a centerline

Change the linetype of the centerline

Set dimension styles

In Fit item, select Leader; and click OK button

Click Save button and click OK button

command: *Choose* Draw, Dimensioning, Center Mark

Select arc or circle: *Select the circle of the stepped hole*

command: *Choose* Draw, Dimensioning, Radial, Diameter

Select arc or circle: *Select the big circle*

Dimension line location (Text/Angle): *Select a location*

command: *Choose* Draw, Dimensioning, Radial, Diameter

Select arc or circle: *Select the small circle*

Dimension line location (Text/Angle): *Select a location*

command: *Choose* Draw, Dimensioning, Linear

First extension line origin or RETURN to select: *Select
the first extension line origin*

Second extension line origin: *Select the second extension
line origin*

Dimension line location (Text/Angle/Horizontal/ Vertical/Rotated):
Select the text location

*In the same way, define all the linear dimensions on the top
view*

command: *Choose* Options, UCS, Icon

9. Defining dimensions on the front view

Select the front view

command: *Choose* Data, Layers...

Select DIMFRONT *and make it the current layer*

Select DIMTOP *layer and click* Cur VP: Frz *button* *Select*
DIMRIGHT *layer and click* Cur VP: Frz *button* *Click* OK
button

command: *Choose* View, Preset UCS...

Select CURRENT VIEW *icon and click* OK

command: *Choose* Draw, Line

Draw centerline of the straight hole

command: *Choose* Modify, Properties...

Select objects: *Select the created line*

Select objects:

Draw center mark for a circle

Draw a diameter dimension

Draw a diameter dimension

Draw a linear dimension

Turn off UCS icon

Select the front view

Change current layer and turn off other
two dimension layers

Set UCS the same as in this view

Draw a centerline

Change the linetype of the centerline

Select Linetype... button

Select CENTER2 and Click OK button

Click OK button

In the same way, draw a centerline for the stepped hole

Define the three linear dimensions on the front view

command: *Choose Options, UCS, Icon*

Turn off UCS icon

10. Defining dimensions on the right view

Select the right view

command: *Choose Data, Layers...*

Select DIMRIGHT and make it the current layer

Select DIMTOP layer and click Cur VP: Frz button

Select DIMFRONT layer and click Cur VP: Frz button

Click OK button

command: *Choose View, Preset UCSS ...*

Select CURRENT VIEW icon and click OK

Draw a centerline for the stepped hole

Define the diameter of the straight hole and two linear dimensions on right view

command: *Choose Options, UCS, Icon*

Select the right view

Change current layer and turn off other two dimension layers

Set UCS the same as in this view

Turn off UCS icon

11. Accessing to isometric view

Select the isometric view

command: *Choose Data, Layers...*

Create a layer called TITLE and make it the current layer

Select DIMTOP layer and click Cur VP: Frz button

Select DIMFRONT layer and click Cur VP: Frz button

Select DIMRIGHT layer and click Cur VP: Frz button

Select BORDER layer and click OFF button

Click OK button

command: *Choose Options, UCS, Icon*

Select the isometric view

Create a layer for title box
Turn off the three dimension layers

Turn off the BORDER layer

Turn off UCS icon

12. Drawing title box

command: *Choose View, Paper Space*

command: *Choose Options, UCS, Icon*

Change to paper space

Turn off UCS icon

command: *Choose* Draw, Polygon, Rectangle

Draw the box

command: *Choose* Draw, Line

Draw a line below the views

command: *Choose* Draw, Text, Dynamic Text

Justify/Style/<Start point>: *Select the first point*

Height <0.2000>: **0.7**

Rotation angle <0>:

Text: **A Component**

Text:

Draw the title box

Enter title of the component

13. Saving AutoCAD file

command: *Choose* File, Save

Enter a file name called component.dwg and click OK button

Save the model to a file

14. Plotting AutoCAD drafting

command: *Choose* File, Print

Click OK button to create a plotting file named component.plt

Plot a PostScript (PS) file

15. Exiting from AutoCAD environment

command: *Choose* File, Exit

Quit AutoCAD system

16. Printing out PostScript file

% lpr -P1w component.plt

Print the PS file using laser printer