Namal University, Mianwali

Department of Electrical Engineering

EE-254– Engineering Drawing

Lab – 9

3D Modelling Basics

Student Name	Institute ID	Marks
Fahim Ur Rehman Shah	NIM-BSEE-2021-24	

Date: 6 / 13 / 2023

Instructor: Engr. Rizwan Shabbir

Document History

Rev.	Date	Comment	Author
1.0	02-2022	Initial Draft	MB
1.1	06-2023	Revision	RS

Course Learning Outcomes

CLO-3: Reproduce 2-D and 3-D sketches using AutoCAD by applying engineering drawing principles.
CLO-4: Present Auto CAD designs effectively through design documentation and reports.

Equipment

- Software
  - AutoCAD 2016 Educational Version

Instructions

The following instructions are to be followed while performing in the labs.

- The manual must be thoroughly read before starting the lab.

- The theoretical concepts related to the lab and experiments must be revised.

- All attempts shall be made to complete the lab during the lab session.
- Any attempt to plagiarize from any source will be reported to the disciplinary committee for further action, so keep the work original.

- Carefully use the laboratory equipment.

Objectives

- To get hands on experience of designing 3D Modeling in AutoCAD software.

Background Information

In AutoCAD, you can create three types of 3D models: surfaces, solids, and meshes. Solids are used to create 3D models of engineering components and assemblies; surfaces are used to create complex shapes such as plastic parts and meshes are used for games and movies. Solids are three-dimensional models of actual objects that possess physical properties such as mass properties, center of gravity, surface area, moments of inertia, and so on. Surfaces are construction features without any thickness. They do not possess any physical properties. Meshes are like solids without mass and volume properties. In this lab, you will learn the basics of 3D modeling such as creating, navigating, and visualizing solid models.

3D Modeling Workspaces in AutoCAD

There are two workspaces of 3D modeling: 3D Basics and 3D Modeling. The 3D Basics workspace has commonly used tools, whereas the 3D Modeling workspace includes all the tools required for creating 3D models.

The Box tool

The Box tool is used to create boxes having six rectangular or square faces. It is the most used tool as many 3D objects are made of boxes.

- Click **Home** > **Create** > **Box** on the ribbon or type **BOX** in the command line; the message, "**Specify the first corner**" appears in the command line.
- Pick an arbitrary point in the graphics window; the message, "Specify the other corner" appears in the command line.
- Ensure that the **Dynamic Input** icon is active on the status bar. You will notice the two value boxes to specify the length and width of the box.
- Type **100** in the length box and press the **TAB** key.
- Type **70** in the width box and press **ENTER**.
- Move the pointer upward, type **60** as height and press **ENTER**; the box will be created as shown.
- Click **Zoom** > **Zoom All** on the **Navigation Bar**.

The Extrude tool

The **Extrude** tool is used to add a third dimension (height) to an existing 2D shape. If you extrude a closed shape such as circle and closed polylines, a solid is created. If you extrude an open sketch such as lines and arcs, a surface is created.

- Click **Home** > **View** > **3D Navigation** > **Front** on the ribbon; the front view will become parallel to thescreen.
- Click **Home** > **Draw** > **Polyline** on the ribbon and create the sketch as shown below.

3
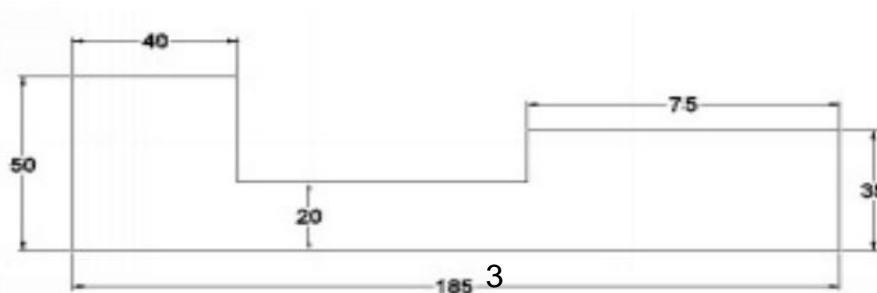
- Select **SE Isometric** from the **In-canvas controls**; the view orientation will be changed Southeast isometric.
- Click **Home** > **Modeling** > **Extrude**.
- Select the polyline sketch and press Enter.
- Move the pointer toward right.
- Type **100** in the command line or **Dynamic Input** box and press **ENTER**; the polyline sketch will be extruded.

The Union tool

The Union tool joins two or more solids together into a single solid. For example, when you try to select the complete model, its individual objects are selected. But, after performing the Union operation, all the solid objects are combined and act as one object.

- To perform the **Union operation**, click **Solid** > **Boolean** > **Union** on the ribbon.
- Click the left mouse button and create a selection window across the model; all the objects of the model will be selected.
- Press ENTER; all the solid objects of the model will be combined.
- Now, when you select an individual object, the complete model will be selected.

The Presspull tool

The Presspull tool is used to create and modify solid models with greater ease and speed. It can be used to accomplish two types of operations: extruding closed 2D shapes and adding or removing material from a solid object based on whether you "pull" or "push" the extrusion.

Subtract Tool

The Subtract tool is used to subtract one or more solid objects from another object.

- Create two concentric circles of 240- and 100-mm diameter.
- Use the **Presspull** tool and extrude up to 250 mm distance.
- Click **Solid > Boolean > Subtract** on the ribbon to remove the inner circle.
- Set the view orientation to Right and create a cylinder of 100 mm diameter and 50 mm length.
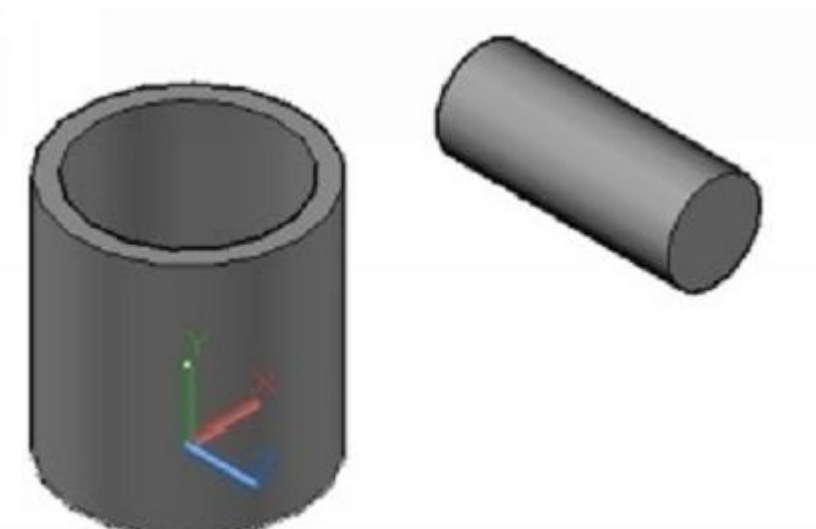
- Change the view orientation to SE Isometric.
- Expand the Modify panel and click the Align button.

- Select the horizontal cylinder and press ENTER; the message, "Specify first source point:" appears in the command line.
- Press and hold the SHIFT key. Right-click and select the Center option.
- Select the center point of the front face of the horizontal cylinder; the message, "Specify first destination point:" appears in the command line.
- Press and hold the SHIFT key. Right-click and select the Quadrant option.
- Select the quadrant point of the outer circle on the top face of the hollow cylinder.
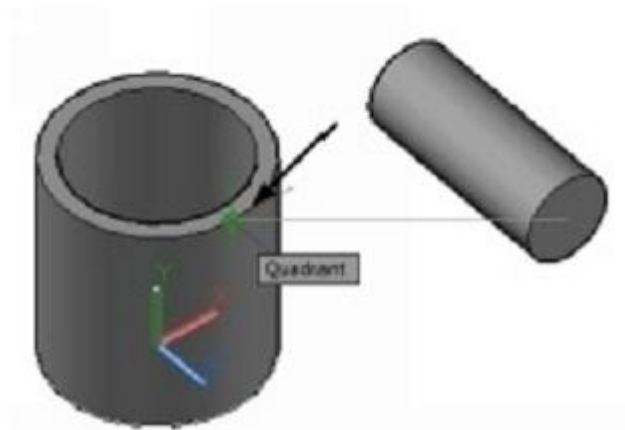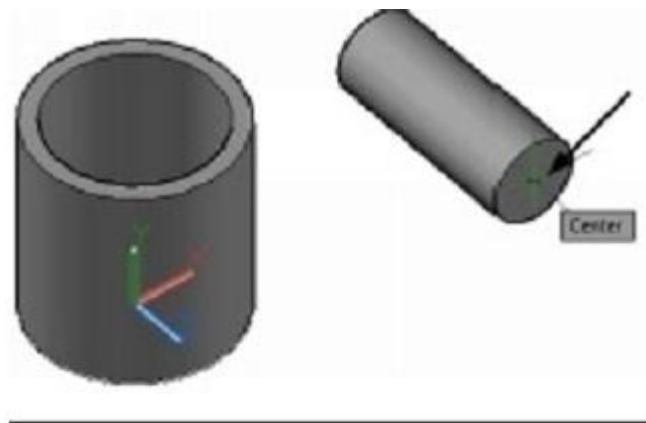
- Press **ENTER; the horizontal** cylinder will be aligned with hollow cylinder.

- Click **Solid > Boolean > Subtract** on the ribbon; the message, "Select solids, surfaces, and regions to subtract from" appears above the command line.
- Select the hollow cylinder and press ENTER; the message, "Select solids, surfaces, and regions to subtract" appears above the command line.
- Select the horizontal cylinder and press ENTER; it will be subtracted from the hollow cylinder as shown below.
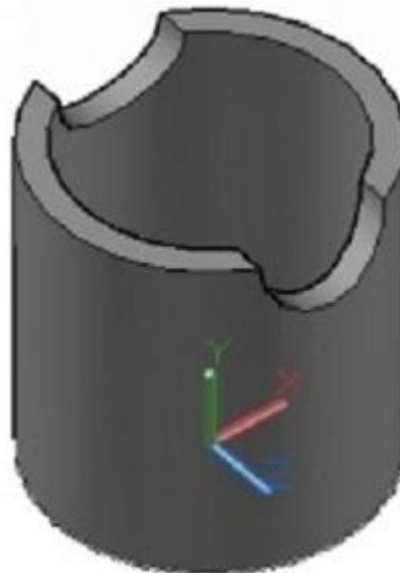
The Loft tool
Using the **Loft** tool, you can create a solid or surface by selecting a series of cross sections. The selected cross sections will define the shape of the lofted solid.

- Create three circles as shown below. The diameters and center point locations are given in the table.

Circle center points (Absolute Coordinates)	Circle Diameters
0,0,0	Ø 50
0,0,70	Ø 100
0,0,140	Ø 50

- Click **Home** > **Create** > **Loft** on the ribbon or type **LOFT** in the command line and press **ENTER**.
- Select the cross-sections one by one; the preview of the lofted solid appears.
- Press ENTER to accept the selection; the message, "Enter an option [Guides/Path/Cross sections only/Settings]:" appears in the command line.
- Select the **Settings** option from the command line; the **Loft Settings** dialog appears. In this dialog, the **Smooth Fit** option creates a smooth connection between the cross-sections. If you select the **Ruled** option, the lofted solid or surface has sharp edges.

The **Normal to** option creates a solid or surface normal to the cross-section. You can select the loft solid or surface to be normal to **All cross sections** or **Start Cross Section** or **End Cross Section** or **Start and End Cross Sections.**
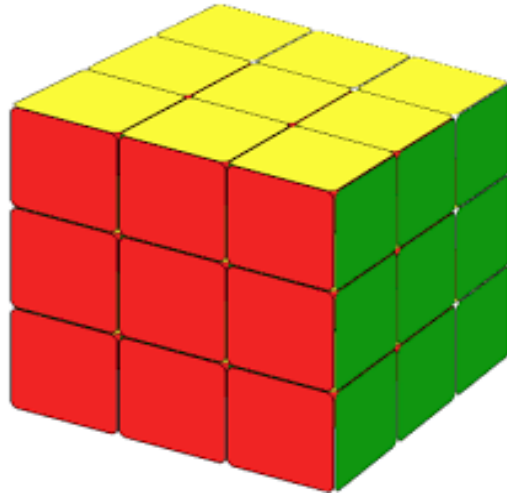
The **Draft angles** option defines the draft angle and magnitude at start and end cross sections. The draft angle is the beginning direction of the loft surface. If you set the draft angle to 90 degrees, the loft surface starts vertically from the cross section and the 0-draft angle starts loft surface horizontally. The Magnitude is the relative distance up to which the loft surface will follow the draft angle before it bends.
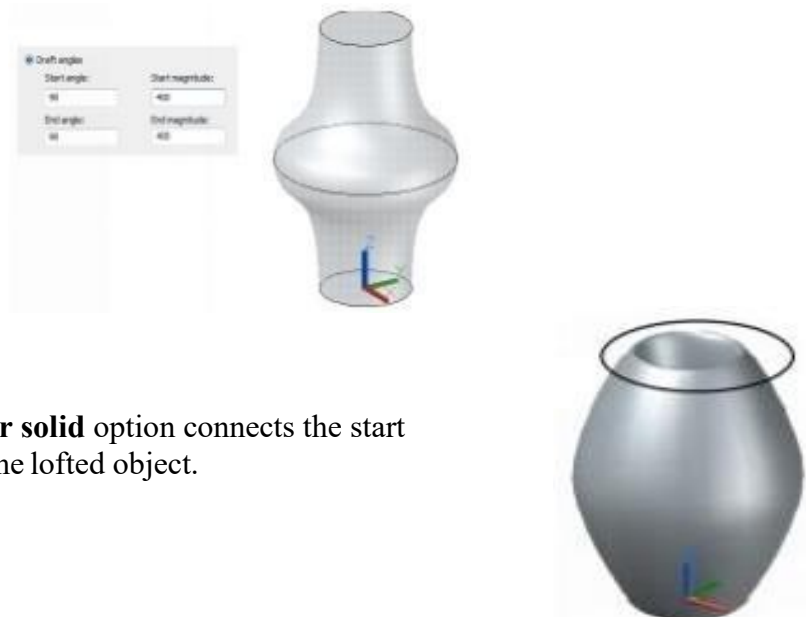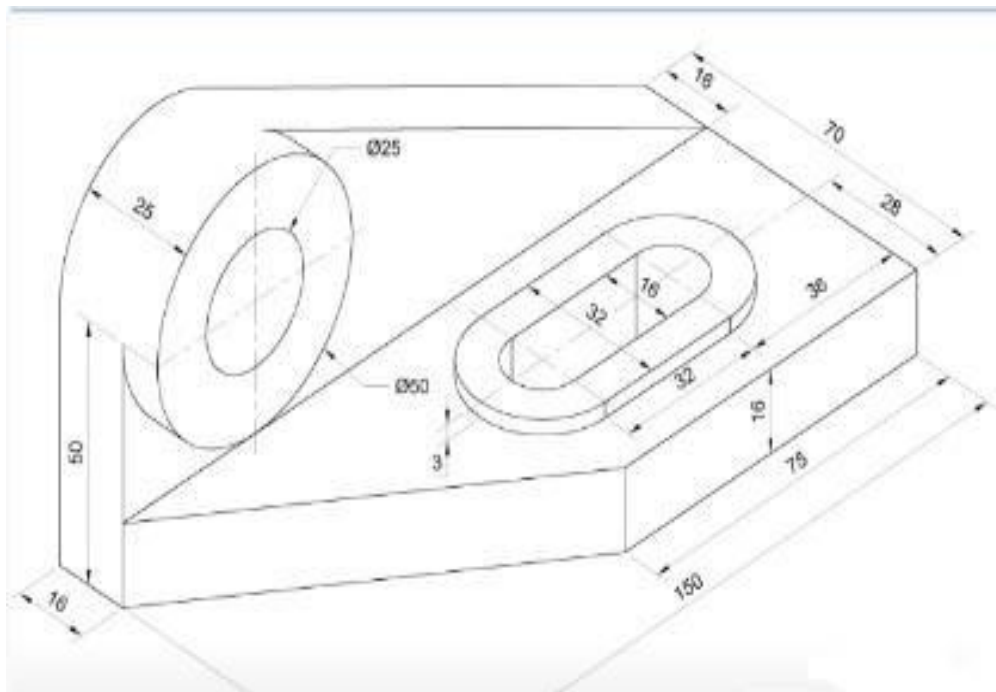

The **Close surface or solid** option connects the start and end section of the lofted object.

Tasks

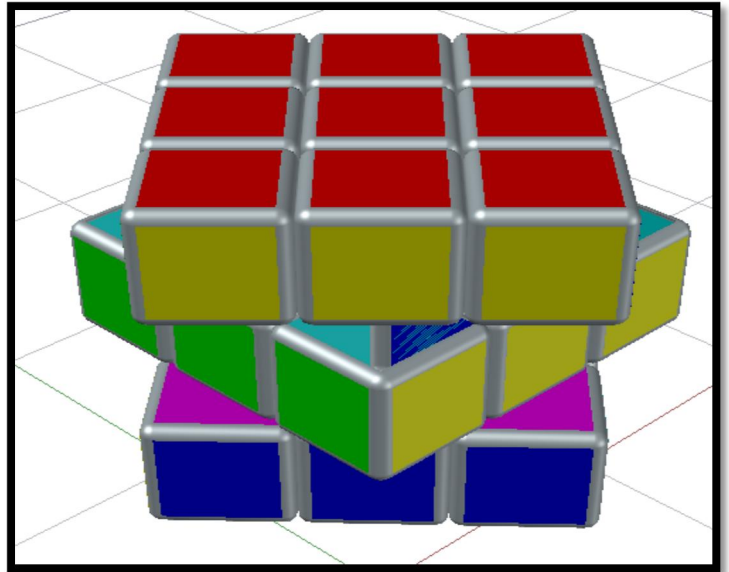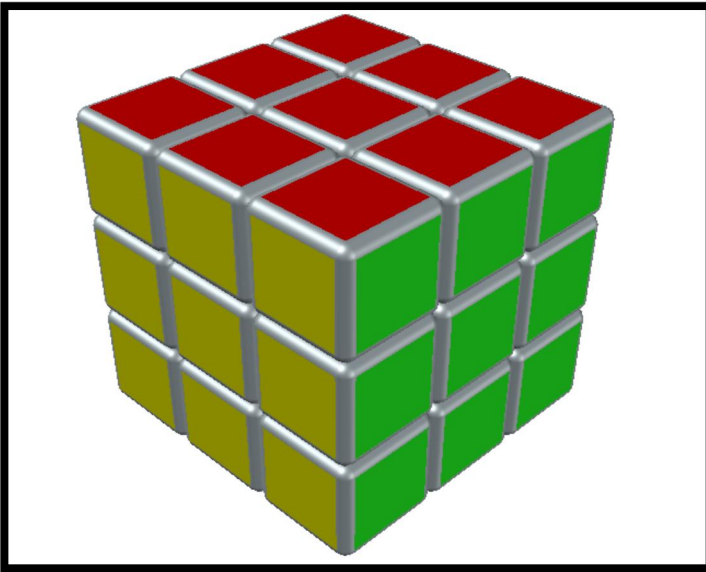Create Rubik's Cube 3*3 with dimension of one box (L5, W5, H5)
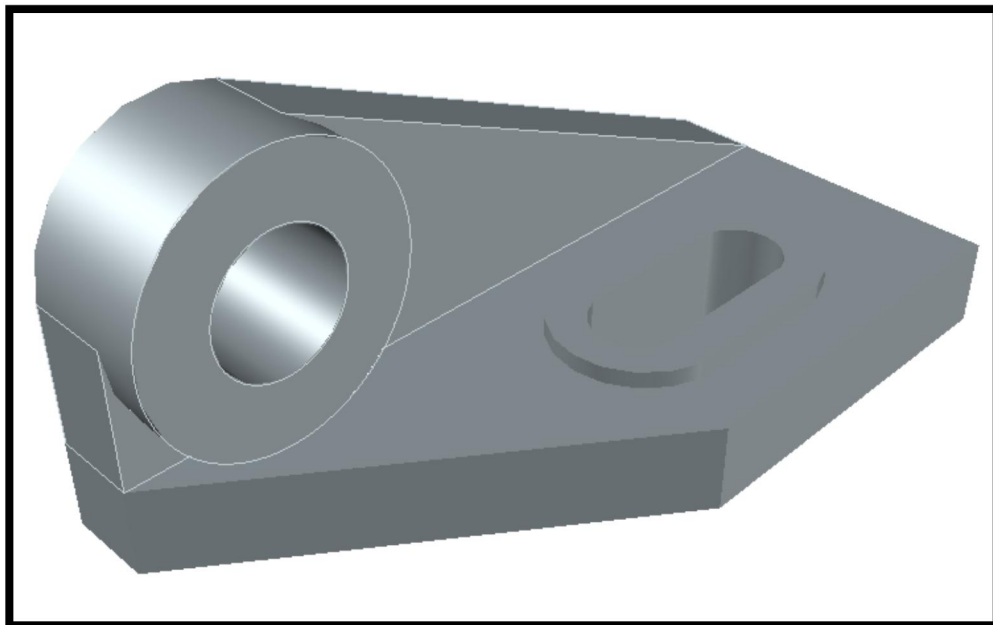

Draw the drawing as shown below.

Commands :

Opening an AutoCAD 2018 format file.	Current wire frame density: ISOLINES=4, Closed profiles
AutoCAD menu utilities loaded.	creation mode = Solid
AutoCAD Electrical menu utilities loaded.*Cancel*	Select objects to extrude or [MOde]: 1 found
Command:	Select objects to extrude or [MOde]:
Autodesk DWG. This file is a TrustedDWG last saved by an Autodesk	Specify height of extrusion or [Direction/Path/Taper
application or Autodesk licensed application.	angle/Expression] <5.0000>: 5
Command: _.erase 1 found	Command:
Command: *Cancel*	Command: FILLETEDGE
Command: _qsave	Radius = 0.5000
Command: _rectang	Select an edge or [Chain/Loop/Radius]:
Specify first corner point or	Select an edge or [Chain/Loop/Radius]:
[Chamfer/Elevation/Fillet/Thickness/Width]:	Select an edge or [Chain/Loop/Radius]:
Resuming RECTANG command.	Select an edge or [Chain/Loop/Radius]:
Specify first corner point or	Select an edge or [Chain/Loop/Radius]:
[Chamfer/Elevation/Fillet/Thickness/Width]:	Select an edge or [Chain/Loop/Radius]:
Specify other corner point or [Area/Dimensions/Rotation]: d	Select an edge or [Chain/Loop/Radius]:
Specify length for rectangles <10.0000>: 5	Select an edge or [Chain/Loop/Radius]:
Specify width for rectangles <10.0000>: 5	Select an edge or [Chain/Loop/Radius]:
Specify other corner point or [Area/Dimensions/Rotation]:	Select an edge or [Chain/Loop/Radius]:
Point or option keyword required.	Resuming FILLETEDGE command.
Specify other corner point or [Area/Dimensions/Rotation]:	Select an edge or [Chain/Loop/Radius]:
Command: EXT	Resuming FILLETEDGE command.
EXTRUDE	Select an edge or [Chain/Loop/Radius]:
Select an edge or [Chain/Loop/Radius]:	Select an edge or [Chain/Loop/Radius]:
Select an edge or [Chain/Loop/Radius]:	Select an edge or [Chain/Loop/Radius]:
Resuming FILLETEDGE command.	Select an edge or [Chain/Loop/Radius]:
Select an edge or [Chain/Loop/Radius]:	Resuming FILLETEDGE command.
Select an edge or [Chain/Loop/Radius]:	Select an edge or [Chain/Loop/Radius]:
Resuming FILLETEDGE command.	Resuming FILLETEDGE command.
Select an edge or [Chain/Loop/Radius]:	Select an edge or [Chain/Loop/Radius]:
Resuming FILLETEDGE command.	Resuming FILLETEDGE command.
Select an edge or [Chain/Loop/Radius]:	Select an edge or [Chain/Loop/Radius]: r
Resuming FILLETEDGE command.	Command:
	Command:

```
Enter fillet radius or [Expression] <0.5000>: 0.5
Select an edge or [Chain/Loop/Radius]:
12 edge(s) selected for fillet.
Press Enter to accept the fillet or [Radius]:
Command: *Cancel*
Command: EXPLODE
Select objects: Specify opposite corner: 1 found
Select objects:
Command: *Cancel*
Command: *Cancel*
Command: UNION
Select objects: Specify opposite corner: 26 found
Select objects:
Command: *Cancel*
Command: ARRAY
Select objects: Specify opposite corner: 7 found
Select objects:
Enter array type [Rectangular/PAth/POlar] <Rectangular>:
Type = Rectangular  Associative = Yes
Select grip to edit array or [ASsociative/Base
point/COUnt/Spacing/COLumns/Rows/Levels/eXit]<eXit>:
Select grip to edit array or [ASsociative/Base
point/COUnt/Spacing/COLumns/Rows/Levels/eXit]<eXit>:
Select grip to edit array or [ASsociative/Base
point/COUnt/Spacing/COLumns/Rows/Levels/eXit]<eXit>:
Select grip to edit array or [ASsociative/Base
point/COUnt/Spacing/COLumns/Rows/Levels/eXit]<eXit>:
Select grip to edit array or [ASsociative/Base
point/COUnt/Spacing/COLumns/Rows/Levels/eXit]<eXit>:
Select grip to edit array or [ASsociative/Base
point/COUnt/Spacing/COLumns/Rows/Levels/eXit]<eXit>:
Command:
Command:
Command: _qsave
```

Task 2 :

10

Commands :

Select source object or multiple objects to join at once:
Specify opposite corner: 5 found
Select objects to join:
4 objects joined into 1 spline, 1 object discarded from the operation
Command:
Press ESC or ENTER to exit, or right-click to display shortcut-menu.
Command:
Command: *Cancel*
Command: *Cancel*
Command:
Command:
Command:
Command: _copyclip 1 found
Command:
Command:
Command: _pasteclip Specify insertion point:
Command:
Press ESC or ENTER to exit, or right-click to display shortcut-menu.
Command:
Command: Specify opposite corner or [Fence/WPolygon/CPolygon]:
Command: m
MOVE 1 found
Specify base point or [Displacement] <Displacement>:
Specify second point or <use first point as displacement>:
Command: *Cancel*
Command:
Press ESC or ENTER to exit, or right-click to display shortcut-menu.
Command: _u '3DORBIT GROUP
Command: _u MOVE
Command:
Command: *Cancel*
Command:
Command: *Cancel*
Command: m
MOVE
Select objects: 1 found
Select objects:
Specify base point or [Displacement] <Displacement>:
Cancel
Command: *Cancel*
Command: m
MOVE
Select objects: 1 found
Select objects:
Specify base point or [Displacement] <Displacement>:
Specify second point or <use first point as displacement>:
Command:
Press ESC or ENTER to exit, or right-click to display

Specify stretch point or [Base point/Copy/Undo/eXit]:*Cancel*
Command: *Cancel*
Command:
Press ESC or ENTER to exit, or right-click to display shortcut-menu.
Command: *Cancel*
Command:
Command: _.erase 1 found
Command: *Cancel*
Command:
Command: *Cancel*
Command:
Command: EXT
EXTRUDE
Current wire frame density: ISOLINES=4, Closed profiles creation mode = Solid
1 found
Specify height of extrusion or [Direction/Path/Taper angle/Expression] <16.0000>: 16
Command:
Press ESC or ENTER to exit, or right-click to display shortcut-menu.
Command:
Command: EXTRUDE
Current wire frame density: ISOLINES=4, Closed profiles creation mode = Solid
1 found
Specify height of extrusion or [Direction/Path/Taper angle/Expression] <16.0000>:
Command: *Cancel*
Command: _u EXTRUDE
Command:
Command: EXT
EXTRUDE
Current wire frame density: ISOLINES=4, Closed profiles creation mode = Solid
1 found
Specify height of extrusion or [Direction/Path/Taper angle/Expression] <19.0000>: 22
Command:
Press ESC or ENTER to exit, or right-click to display shortcut-menu.
Command: SUBTRACT
Select solids, surfaces, and regions to subtract from ..
Select objects: 1 found
Select objects:
Select solids, surfaces, and regions to subtract ..
Select objects: 1 found
Select objects:
Command:
Press ESC or ENTER to exit, or right-click to display shortcut-menu.
Command:
Specify stretch point or [Base point/Copy/Undo/eXit]:*Cancel*
Command: *Cancel*
Command:
Press ESC or ENTER to exit, or right-click to display shortcut-menu.

shortcut-menu.
Command:
Command:
Command: *Cancel*
Command:
Command:
** STRETCH **
Command: *Cancel*
Command: '3DORBIT
Press ESC or ENTER to exit, or right-click to display
shortcut-menu.
Command: SUBTRACT
Select solids, surfaces, and regions to subtract from ..
Select objects: 1 found
Select objects:
Select solids, surfaces, and regions to subtract ..
Select objects: *Cancel*
Command: _u SUBTRACT
Command: _u '3DORBIT
Command: _u '3DORBIT GROUP
Command: _u SUBTRACT
Command:
Press ESC or ENTER to exit, or right-click to display
shortcut-menu.
Command: SUBTRACT
Select solids, surfaces, and regions to subtract from ..
Select objects: 1 found
Select objects:
Select solids, surfaces, and regions to subtract ..
Select objects: 1 found
Select objects:
Command: *Cancel*
Command:
Press ESC or ENTER to exit, or right-click to display
shortcut-menu.
Command:
Command:
Press ESC or ENTER to exit, or right-click to display
shortcut-menu.
Command: Specify opposite corner or
[Fence/WPolygon/CPolygon]: *Cancel*
Command: *Cancel*
Command:
Press ESC or ENTER to exit, or right-click to display
shortcut-menu.
Command:
Command:
Press ESC or ENTER to exit, or right-click to display
shortcut-menu.
Command: *Cancel*
Command: Specify corner of window, enter a scale factor
(nX or nXP), or [All/Extents/Window/Previous/Object]
<real time>:
Command: _u 'ZOOM GROUP
Command: _u '3DORBIT GROUP
Command: _u '3DORBIT GROUP
Command: _u '3DORBIT GROUP
Command: _u EXPLODE

Command: *Cancel*
Command:
Command: _.erase 1 found
Command: *Cancel*
Command:
Command: *Cancel*
Command:
Command: EXT
EXTRUDE
Current wire frame density: ISOLINES=4, Closed profiles creation
mode = Solid
1 found
Specify height of extrusion or [Direction/Path/Taper
angle/Expression] <16.0000>: 16
Command:
Press ESC or ENTER to exit, or right-click to display shortcut-menu.
Command:
Command: EXTRUDE
Current wire frame density: ISOLINES=4, Closed profiles creation
mode = Solid
1 found
Specify height of extrusion or [Direction/Path/Taper
angle/Expression] <16.0000>:
Command: *Cancel*
Command: _u EXTRUDE
Command:
Command: EXT
EXTRUDE
Current wire frame density: ISOLINES=4, Closed profiles creation
mode = Solid
1 found
Specify height of extrusion or [Direction/Path/Taper
angle/Expression] <19.0000>: 22
Command:
Press ESC or ENTER to exit, or right-click to display shortcut-menu.
Command: SUBTRACT
Select solids, surfaces, and regions to subtract from ..
Select objects: 1 found
Select objects:
Select solids, surfaces, and regions to subtract ..
Select objects: 1 found
Select objects:
Command:
Press ESC or ENTER to exit, or right-click to display shortcut-menu.
Command:
Command: _u '3DORBIT GROUP
Command: _u SUBTRACT
Command: *Cancel*
Command:
Press ESC or ENTER to exit, or right-click to display shortcut-menu.
Command:
Command: *Cancel*
Command:
Command: EXPLODE
1 found
Command:
Command: _.erase 1 found
Command: _u ERASE

Command: _u '3DORBIT GROUP	Command: *Cancel*
x	Command: *Cancel*
Select cross sections in lofting order or [POint/Join	Command: <Switching to: Layout1>
multiple edges/MOde]: Specify opposite corner:	Regenerating layout.
Selected profile cannot be a multi-face surface or region.	Command: Specify opposite corner or [Fence/WPolygon/CPolygon]:
The selected entities are not valid.	*Cancel*
Selected cross section is not a valid type.	Command: _.MSPACE
Selected profile cannot be a multi-face surface or region.	Command: Specify corner of window, enter a scale factor (nX or
The selected entities are not valid.	nXP), or [All/Extents/Window/Previous/Object] <real time>:
Cross-section curves must be all open or all closed.	Command:
Cross-section curves must be all open or all closed.	Command:
Cross-section curves must be all open or all closed.	Command: _.PSPACE
Cannot create a solid from open cross sections.	Command: Specify opposite corner or [Fence/WPolygon/CPolygon]:
Cross-section curves must be all open or all closed.	*Cancel*
Cannot create a solid from open cross sections.16 found	Command: _.MSPACE
7 were filtered out	Command:
Select cross sections in lofting order or [POint/Join	Command: _.PSPACE
multiple edges/MOde]:	Command: Specify opposite corner or [Fence/WPolygon/CPolygon]:
9 cross sections selected	Command: Specify opposite corner or [Fence/WPolygon/CPolygon]:
Enter an option [Guides/Path/Cross sections	*Cancel*
only/Settings/COntinuity/Bulge magnitude] <Cross	Command: _.MSPACE
sections only>:	Command: <Grid off>
The selected entities are not valid.	Command:
Command:	Command:
LOFT	Command: _PLOT Effective plotting area: 210.00 wide by 297.00
Current wire frame density: ISOLINES=4, Closed profiles	high
creation mode = Solid	Effective plotting area: 155.96 wide by 205.53 high
Select cross sections in lofting order or [POint/Join	Plotting viewport 1.
multiple edges/MOde]:	Command: _.PSPACE
At least two profiles must be selected.	Command:
Command:	Press ESC or ENTER to exit, or right-click to display shortcut menu.
LOFT	Command: Specify opposite corner or [Fence/WPolygon/CPolygon]:
Current wire frame density: ISOLINES=4, Closed profiles	*Cancel*
creation mode = Solid	Command: _.MSPACE
Select cross sections in lofting order or [POint/Join	Command:
multiple edges/MOde]:	Command: _.PSPACE
At least two profiles must be selected.	Command:
At least two profiles must be selected.	Command:
Command: *Cancel*	Command: _qsave
Command: Specify opposite corner or	
[Fence/WPolygon/CPolygon]:	
Command: JOIN	
16 found	
0 objects joined, 15 objects discarded from the operation	
Command:	
Press ESC or ENTER to exit, or right-click to display	
shortcut-menu.	
Command: Specify corner of window, enter a scale factor	
(nX or nXP), or [All/Extents/Window/Previous/Object]	
<real time>:	
Command:	
Command: Specify opposite corner or	
[Fence/WPolygon/CPolygon]: *Cancel*	
Command: *Cancel*	
Command:	
Press ESC or ENTER to exit, or right-click to display	
shortcut-menu.	

Engineering Drawing Lab 09

Method of Evaluation: Lab Report and in-lab marking by instructors.
Measured Learning Outcomes
CLO-3: Reproduce 2-D and 3-D sketches using AutoCAD by applying engineering drawing principles.
CLO-4: Present Auto CAD designs effectively through design documentation and reports.

	Excellent (10)	Good (9 to 7)	Satisfactory (6 to 4)	Unsatisfactory (3 to 1)	Poor (0)	Marks Obtained
Assignment (CLO4)	Required document filled-in neatly with meaningful answers to all questions, proper grammar and punctuations with proper conclusion drawn	Required document filled-in neatly with meaningful answers to most questions and proper conclusions drawn with some grammar mistakes	Some correct/meaningful answers and conclusions with some irrelevant ones. Some parts of the document not neat or some grammar mistakes.	Answers not understandable/ not relevant to questions. Conclusions not based on results. Illegible writing with no proper grammar/punctuation	Report/Hand out Not submitted	
Task Completion (CLO3)	All Tasks were completedsuccessfully in the time ofthe lab	Most of the tasks were completed in the given time of the lab	Some of the tasks were completed in the given lab time	Very few tasks were performed and completed in given lab time	Lab tasks weren't performed at all	
Total						