

# AUTOCAD-II

## Lectures Notes

Government Polytechnic, Bhubaneswar

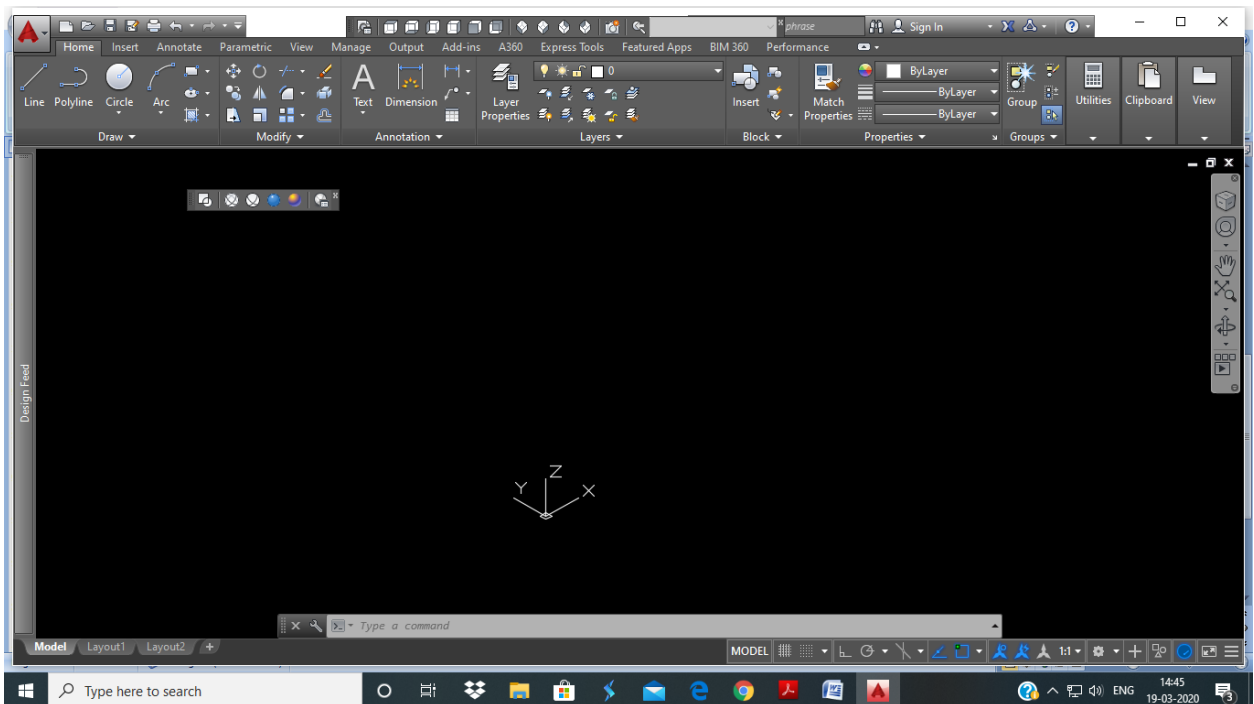
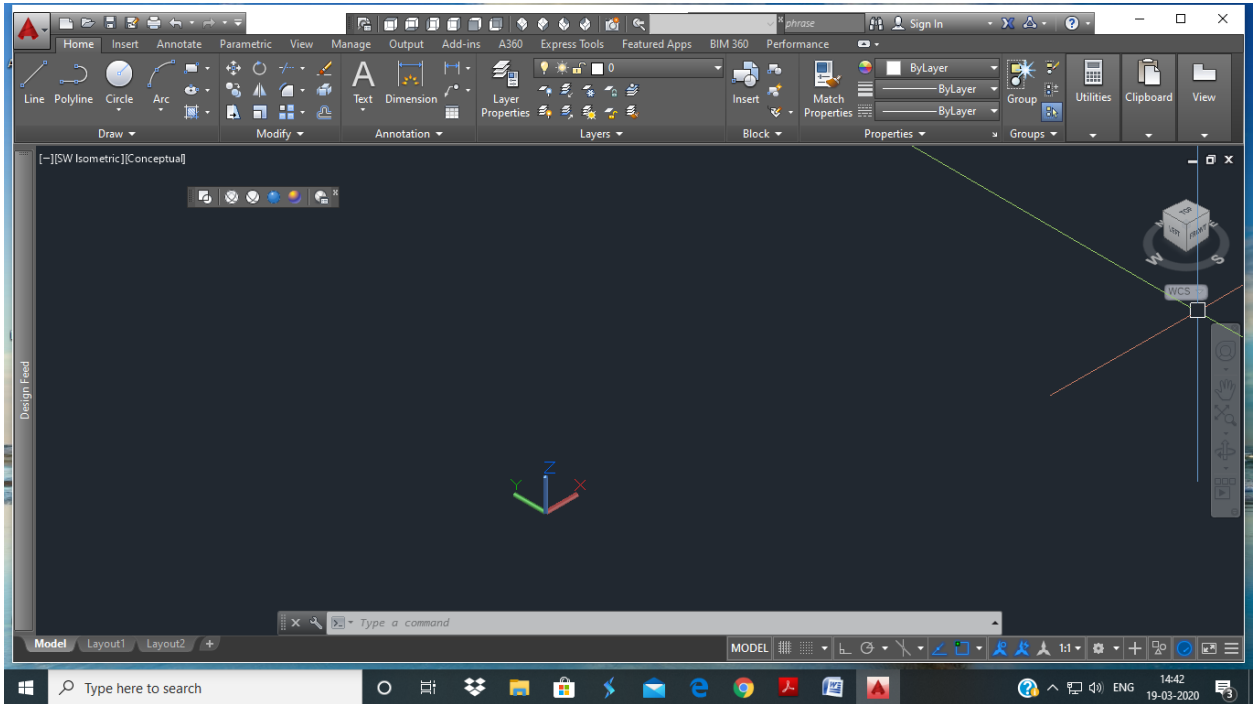
Diploma in Architecture Assistantship | IV Semester

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**COORDINATE SYSTEM.**

When you create a 2D drawing you create drawing in XY plane only. As you create Geometry in 3D you will need to reorient the coordinate system to create and modify the geometry.

In the following figure, the icon's show the direction of XY&Z axis of the Cartesian coordinate system based is shown in its shaded mode & wire frame mode.



**CHANGING THE COORDINATE SYSTEM.**

The process of creating a 3D model can be made a lot simpler when you, generate or define different aspect of your design in a flat plane and the plane you need to drawing in usually not in the same location or orientation as the WCS, you need to be able to define your coordinate system. The UCS you define then allows you to create, the geometry you need in the appropriate location & orientation.

**STEPS:-**

Command line:-UCS



**OPTIONS FOR DEFINING A UCS.**

Following the typical work flow command option, you either reposition the origin of the co-ordinate system while keeping its XYZ axis alignment or you completely re-orient and reposition the co-ordinate system based on three points in space.

You can define the UCS based on the same option as follows.

**WORLD:**



Used to set the co-ordinate system back to WCS.

**NAMED UCS:-**



Use to display the dialog box .Save the UCS and active a saved UCS.

**PREVIOUS:-**



Use to step the co-ordinate system back to previous UCS.



Use to align the UCS to any of the Face of a solid.



Use to align the UCS to any selected Object.



Use to align the UCS

**View**  
Aligns the XY plane of the user coordinate system with the screen

Aligns the XY plane of the UCS to a plane perpendicular to your viewing direction. The origin point remains unchanged, but the X and Y axes become horizontal and vertical.

**Origin**

Defines a new user coordinate system by shifting the origin point

Shifts the user coordinate system origin point (0,0,0) to facilitate entering absolute coordinates, creating ordinate dimensions, or setting the work plane in 3D.



**Z Axis Vector**

Aligns the user coordinate system to a specified positive Z axis

The UCS origin is moved to the first point and its positive Z axis passes through the second point.



**3 Point**

Defines a new user coordinate system using three points

The three points specify an origin point, a point on the positive X axis, and a point on the positive XY plane.

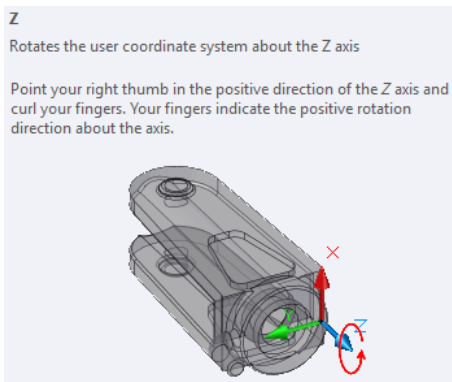
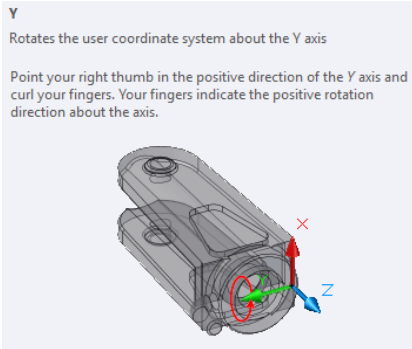


**X**

Rotates the user coordinate system about the X axis

Point your right thumb in the positive direction of the X axis and curl your fingers. Your fingers indicate the positive rotation direction about the axis.





Applies the current UCS setting to a selected viewport or all the viewport.

### TYPES OF 3D MODELS.

The extent of the benefits of a 3D Model depends on which of the three model types you created those three types of models are.

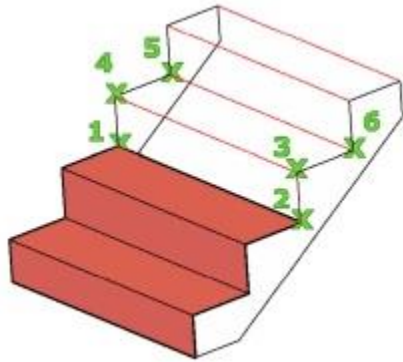
- Wire frame Model.
- Surface Model.
- Solid Model.

- **Wire frame Model:-**

The most basic form for 3D model representation, you draw lines , arcs, & circles in 3D space to represent the edges of your design. Here we define only the edges of 3D object created, e.g. creating a cube using 12 nos of match sticks. It contains only edges no surface or faces.

- **Surface Model:-**

A higher level of model re-presentation since it not only defines the edges of the design but also on outer skin to the model. Surface models can add clarity to the display of a design by hiding all geometry that resides behind of a surface. a WIRE FRAME MODEL can be converted to a SURFACE MODEL by adding surfaces e.g. a wireframe model of a cube can be converted in to a hollow cube by adding six planes.



Adding surface to a wire frame model.

● **Solid Model:-**

This model type defines the volume, outer surface & edges of your design within a single object solid models represents aspects of a design & thus are computerised representational of 3D model you can computerised solid models together to create even more computer solid models.

**ORBITING YOUR 3D MODEL:-**

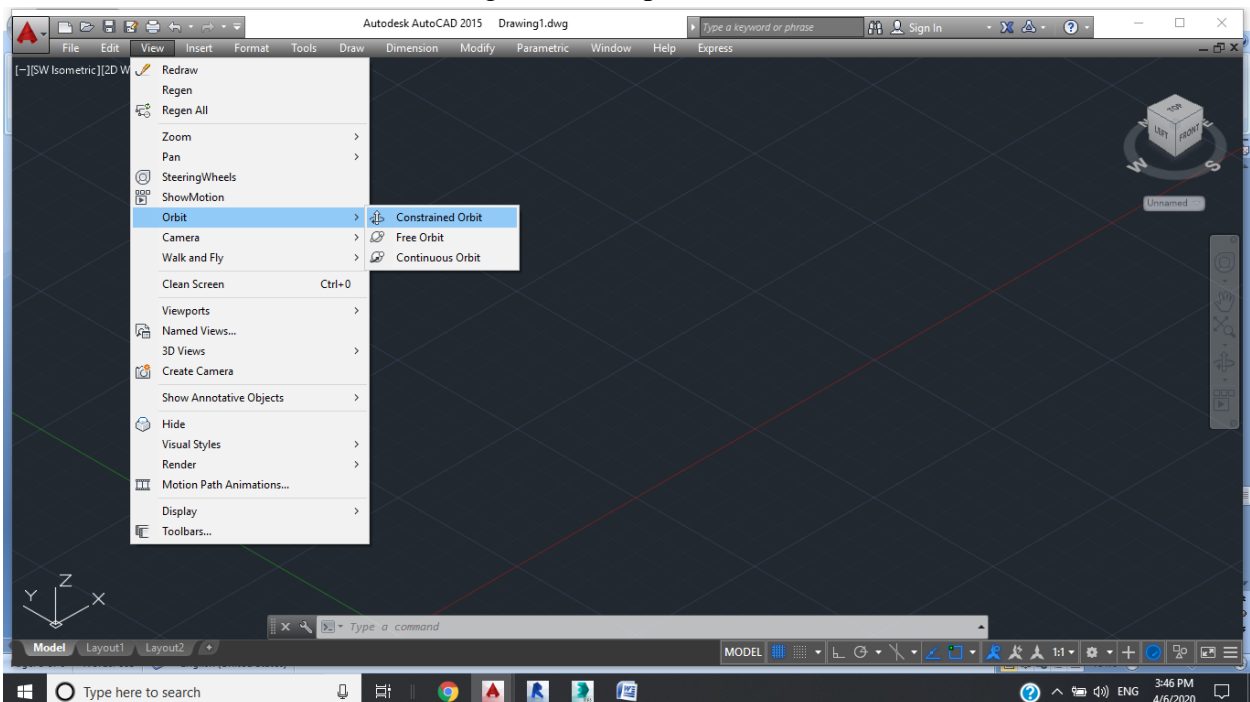
Orbiting your viewing point around your model allows you to see different aspects & details of your design with **cons orbit** command you can freely rotate your view around your model if no geometry is selected at the start of the command pivots the view about the center point of a bounding box of all the geometry. If you select geometry before executing the command then the view orbits around the center of the selected geometry.

Command :- 3D Orbit , 3DO

Menu→View→Orbit→Constrained Orbit

Toolbar→Orbit

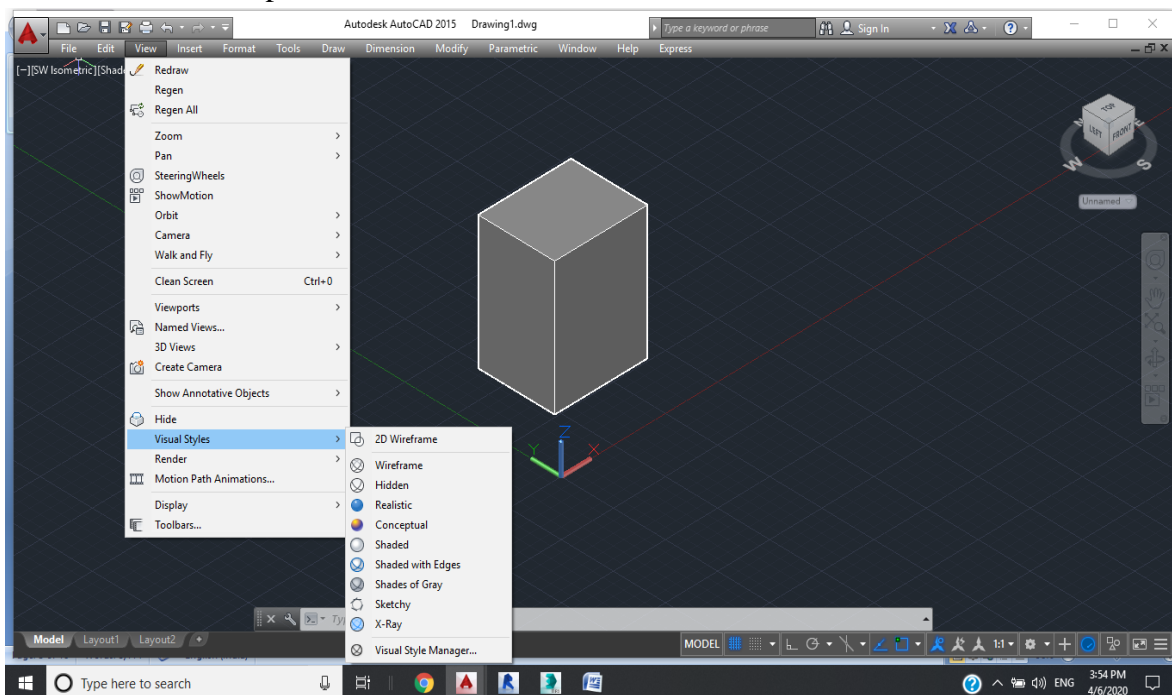
Dash Board→3D Navigate control panel



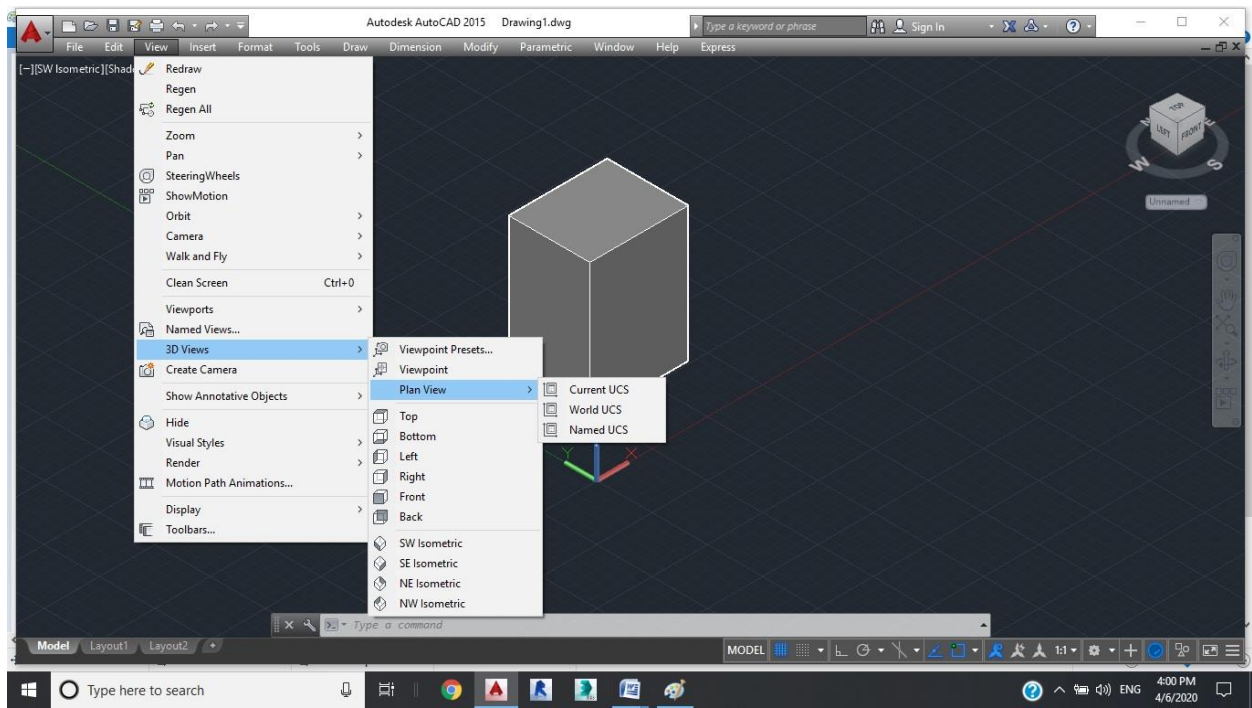
## CHANGING MODEL DISPLAY:-

When selecting a display mode of your 3D surface or solid model, you have five preset display modes you can select to as visual styles. The five print display modes are

- 2D Wireframe
- Wireframe
- Glidden
- Realstic
- Conceptual



## 3D VIEWS.



It shows plan view i.e in WCS/current UCS/Named UCS, SIX Orthographics view(top/bottom/left/right/Front/back and four Isometric views(SW Isometric /SE Isometric/NE Isometric/NW Isometric Views.

## SETTING VIEW POINT DISPLAY:-

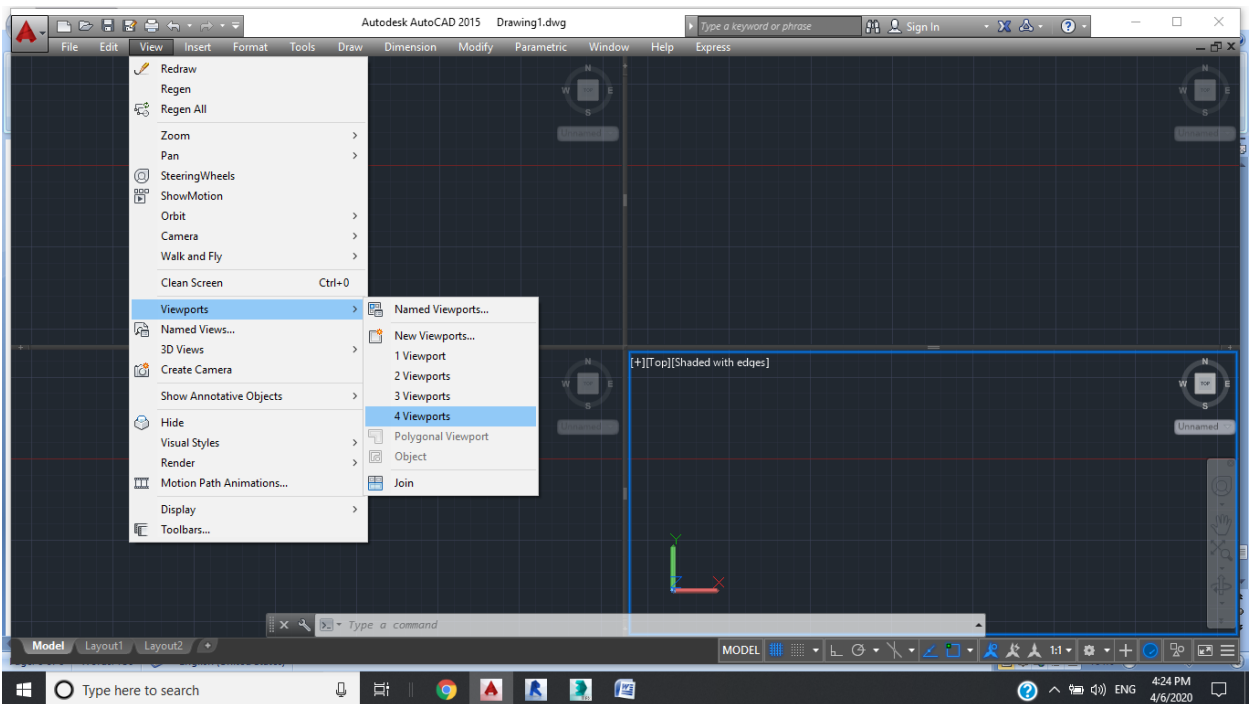
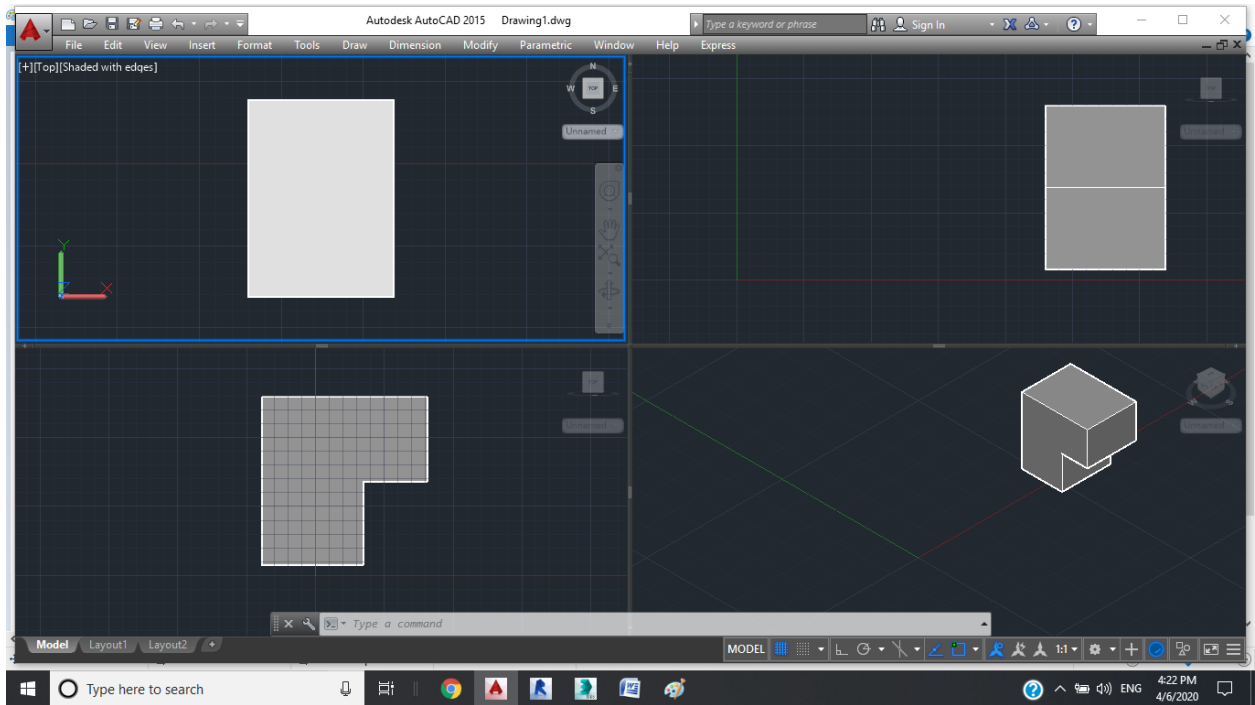
When working in 3D you can increase your productivity by changing the number of view parts displayed within the drawing area , By creating a appropriate number of view parts , you can view model from multiple directions at the same time .

Command :-View port, Vports

Menu→View→View ports→New View Ports

Toolbar→View ports.





**VIEW PORTS DIALOGUE BOX:-**

For typical workflow you first display the viewports dialogue box & then configure the number of viewports the display the view orientation & the display style.

If you make a number of changes, enter a name so that when you click ok, this view ports configuration is added to the list of named viewports. To apply a saved viewports tab double click a configuration name & click ok.

## **SOLID MODEL:-**

Solid models are predefined geometric provided to you have, seven basic solids you can design with a box sphere, cylinder, cone, wedge & pyramid.

To create these shape you only need to supply a creation location & actual size , only you have created a solid , you have information own to you such as its volume & other mass properties, when you have created a solid you have information to you have created more than one solids you can create more complex model by combining them into a single model you can also subtract the volume at one model from another.

## **CREATING A SOLID BOX:-**

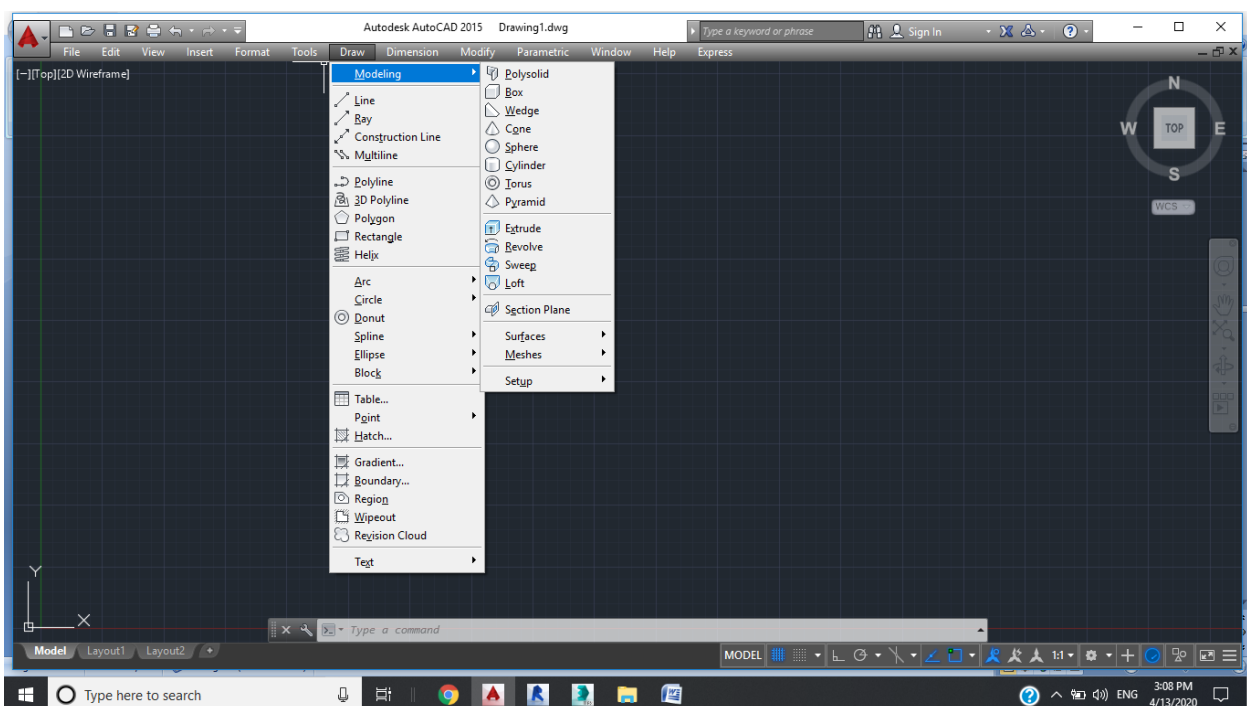
You use the box command to create rectangle or wheel shaped solids.

Command : Box

Menu→Draw→Modelling→Box

Toolbar→Modelling

Dash board→3D Make ctrl panel



**COMMAND OPTION:-**

**Center:-** Use to define the location of the solids geometric center prior to specifying its size.

**Cube:-** Use to create a box with all of its edges equal to a single specified value instead of specifying the three separate.

**Length:-** Use this option to create the base rectangular shape so its edge are not parallel to the X & Y axis of the current Vcs.

**CREATING A SOLID SPHERE:-**

Use the sphere command to create circular solid.

Command: Sphere

Menu→Draw→Modeling→Sphere

Toolbar→Modeling

Dash Board. 3D make ctrl panel

Use to define the size of the circular cross section by specifying three points that reside in the same co-ordinate.

**CREATING A SOLID CYLINDER:-**

- **3D:-** Use to define the base circular shape by having its circular edge pass through three points in space.
- **2D:-** Use to define the base diameter of the circular base using two opposite points on its outer edge.
- **ITr:-** Use when you need the circular base to be tangent to two different edges & a specific radius.
- **Elliptical:-** Use when you want the base shape of the cylinder to be an ellipse inside of a circle
- **Axis & Point:-** Use to specify the top cylinder height so its center axis extends from its base center point to the selected axis end point.

**CREATING SOLID CONE:-**

Use the cone command to create a triangular shaped with curved slides that transition in shape from the base to the top . The default shape is circular but you can also create an elliptical base . The cone then transition from its base to a point at the top or to a shape smaller or larger than its base .

Command: Cone

Menu→Draw→Modelling→Cone

Toolbar→Modelling

Dashboard – 3D make ctrl panel

### **COMMAND OPTION:-**

- **3D:-** Use to define the base circular shape by having its circular edge pass through the points in shape space.
- **2D:-** Use to define the diameter of line circular base using to opposite points on its outer.
- **ITr:-** Use when you need the circular base of tangent to two different you want to be the base.
- **Elliptical:-** Use when you want to be the base shape of the cone to be an ellipse instead of a circle.
- **Axis end point:-** Use to specify the top center point of the cone. This sets the cone height & radius of the cone so its center axis extends from its base center point to the select axis end point.

### **CREATING A SOLID WEDGE:-**

Use the wedge command to create a triangular solid with three rectangular faces. When you create a wedge you end up with a shape that appears to be half of box that is split diagonally from one edge to another. The high side of wedge is the side opposite the second point specified. When creating the base rectangular shape.

Command: Wedge

Menu→Draw→Modelling→Wedge

Toolbar→Modelling

### **CREATING A SOLID TORUS:-**

Use the torus command to create circular tube with its final shape resembling by circle inner tube you create a torous by defining the size & position of two circular shape.

Command:Torus

Menu→Draw→Modelling→Torus

Toolbar→Modelling

**Procedure:-**

The following steps give an overview of creating a solid torus.

- Start the torus command.
- Specifying the start position, orientation, & size of the torus . Do this by radius or diameter a select one of the sub option & respond its requirement.
- Specify the radius or diameter for the solid part of the solid torus.

**CREATING A SOLID PYRAMID:-**

Use the pyramid command to create a solid that has a polygonal base with flat side that transition in shape from the base to the top. The pyramid can transition from its base shape to a single point or to a shape smaller or larger than its base by default , The polygon base has four sides but you can change that number based on your requirements.

Command: Pyramid

Menu→Draw→Modelling→Pyramid

Toolbar→Modelling

**COMMAND OPTION:-**

- **Edge:-** Use to specify the length of a flat segment of the polygon base when picking the points to define the base size of the segment you also set the position & orientation of the base polygon shape.
- **Sides:-** Use to change the shape of the pyramid by changing the number of sides from the default of 4 to any value greater than & less than 33.

**CIRCUMSCRIBE/INSCRIBE:-**

Use to change which outer point you will define when specifying the size of the base polygon shape with circumscribe you size the polygon from the center point to the end point of a polygon segment.

**CREATING A MODEL USING POLYSOLID:-**

Create multiple segment solids using polysolid command . Multiple segment solids are 3D solids that consists of a rectangular profile & a single line path the rectangular profile follows when create a solid using polysolid , you first set the width & the height value for the

rectangle. Then you draw the path of line & arcs like drawing a polyline or select existing of a geometry to define the path.

Command: Polysolid

Menu→Draw→Modelling→Polysolid

Toolbar→Modelling

### **COMMAND OPTION:-**

- **Height:-** Use to set the distance from the plane path is being drawn on the top of the rectangular profile
- **Width:-** Use to set the width of the rectangular path.
- **Justify:-** Use to align the rectangular profile relative to the path being drawn options arc left, center & right justification.
- **Object:-** Use to make of existing planner geometry as a path for the rectangular profile.
- **Arc:-** Use to create arc instead of straight path segment.
- **Line:-** Use to switch back to creating straight path segments after selecting the arc suboption.
- **Close:-** Use to have the last segment automatic connect to start point of the first segment.

### **PRESSPULL:-**

Using the presspull command , you create a solid model by processing a planar bounded area. You can create multisided models from profile areas very quickly using are after the result of different operation or the intersection objects. This command only requires a closed boundary.

Command line : Presspull or hold ctrl+ alt

Toolbar→Modelling

### **REVOLVE COMMAND:-**

Use the revolve command to create a circular 3D model. When you revolved a profile you span the profile around a defined axis. The amount of revolution can be a full 360<sup>0</sup> or any start & stop angle within 360<sup>0</sup>. You create a solid model, if you select a closed loop object , if you select a open loop geometry or a closed loop composed of separate objects then you create a surface model.

Command: Revolve

Menu→Draw→Modelling→Revolve

Toolbar→Modelling

### **COMMAND OPTION:-**

- **Object:-** Use the revolve to select the profile geometry around a line segment.
- **X/Y/Z:-** Use to revolve the selected geometry around its corresponding axis in the current coordinate system alignment.
- **Start angle:-** Use to have the profile start revolving & creating a model at a position other than the plane on which it resides.

### **MODELS FROM 2D PROFILE:-**

**PLANNER SURFACE:-** Use the planner surface command create surface in a flat plane, you either draw a rectangle surface in flat plane or convert a closed loop planner surface with closed loop planner objects . You can convert circles, ellipse, polylines & spling.

Command: Plane Surface

Menu→Draw→Modelling→Plane Surface

Toolbar→Modelling

**EXTRUDE:-** Use the extrude command to create 3D modelling from geometry representing a 2D profile of that model when you extrude a profile a model is created a specified distance& direction between the original planner profile & a projection of that profile, you can create a single model if you see a single closed loop object, if you select open loop geometry a closed loop composed of separate objects. You create a surface model.

Command line: Extrude

Menu→Draw→Modelling→Extrude

Toolbar→Modelling

### **COMMAND OPTION:-**

- **Direction:-** Use to specify a linear extrusion direction of the distance other than perpendicular to the plane of the cross section geometry. The face at the end of the extrusion is parallel to the plane where the cross section geometry results.
- **Path:-** Use to extrude the cross section geometry along other extrude will follow a parallel path starting at the cross section will be perpendicular to the path & not parallel to its original plane

- **Taper angle:-** Use to have the model get narrower or wider as it extrudes away of the cross section geometry so specify a positive or negative angle to get smaller & larger extrudes.

### **CREATING A HELIX:-**

Helix command is use as a path for a swept or lofted model.

Command line: Helix

Menu→Draw→Modelling→Helix

Toolbar→Helix

### **COMMAND OPTION:-**

- **Axis end point:-** Use to specify the top center point of the helix. The values sets the helix height, it also re-orient the helix so its center axis extends from its base center point to the selected axis end point.
- **Turns:-** Use to set the number of revolution for the helix
- **Turn height:-** Use to set a positive distance between each turn
- **Twist:-** Use to set the rotation of the helix to clockwise or counter clockwise

### **SWEEP COMMAND:-**

Use the sweep command to create a model has a more free form or compound shape , that a shape, that isn't linear & doesn't have a single axis it can revolve around, you create swept model by having profile geometry follow the path defined by another piece of geometry.

Command line: Sweep

Menu→Draw→Modelling→Sweep

Toolbar→Modelling

### **COMMAND OPTION:-**

- **Allignment:-** Use to have to the profile geometry maintain its current angle at the start point of the path instead of its being rotated.so it is perpendicular to the start point of the path.
- **Base point:-** Use to select a point on the profile geometry other than its center, that you want to have match up with the path.
- **Scale:-** Use to have the profile be a space factor larger or smaller than the original profile geometry



- **Twist:-** Use to rotate the profile geometry or specified number of degrees as travels for the start to the end of the path.

### **LOFT COMMAND:-**

Use the loft command to create models of free shape. This can be a model that changes from one shape to another , when you create a lofted model you select multiple cross section & the model transition inside shape & from one cross section to another.

Command line: loft

Menu→Draw→Modelling→loft

Toolbar→Modelling

### **COMMAND OPTION:-**

- **Guides:-** Use to control the shape & way the model transition from one profile to another.

### **CREATING COMPOSITE SOLIDS:-**

Creating solids using union , use the union command to combine two or more solid models into a single composite model. During the operation of command you select all the solids you want to join into a single composite solid.

Command line: union

Menu→Modify→ solid editing→union

Toolbar→solid editing

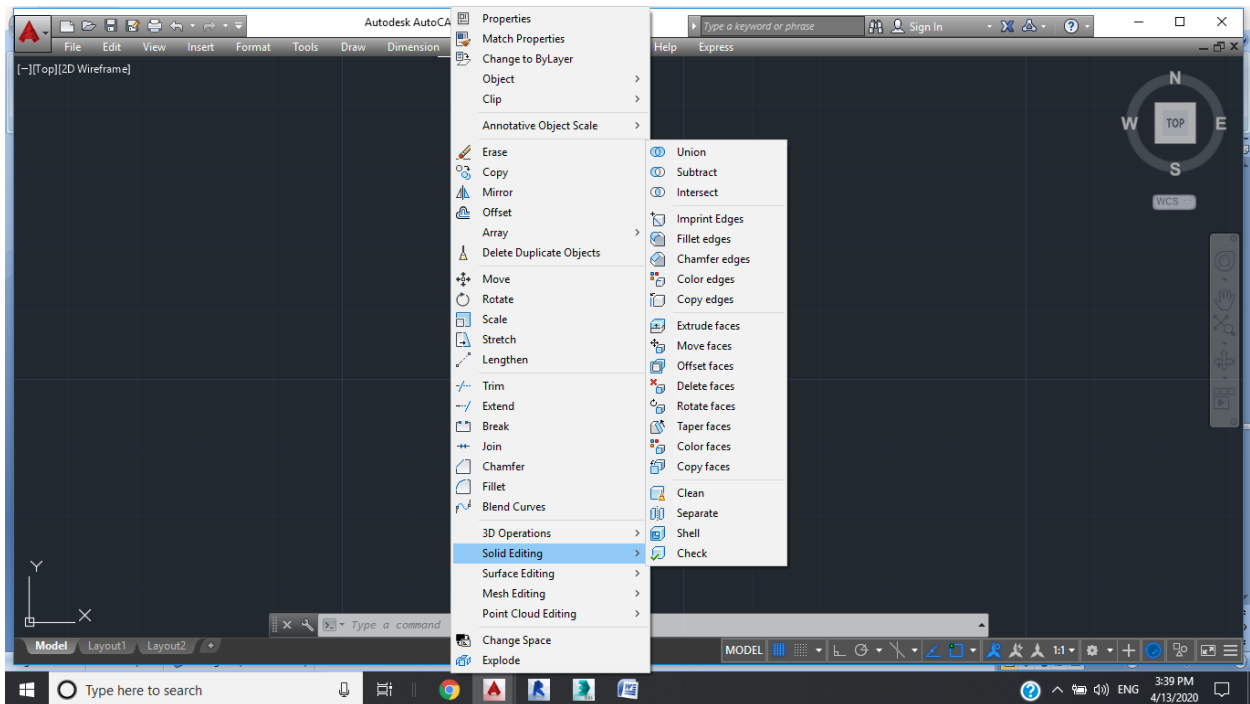
### **CREATING SOLIDS USING SUBSTRACT:-**

Use the substract command to remove matrix from a solid based on the volume of other intersect solids. During the operation of the command you define two separate solid models are the one that you want to keep the second set of solids are one that you want to substract from first set.

Command line: substract

Menu→Modify→ solid editing→substract

Toolbar→solid editing



### **CREATING SOLIDS USING INTERSECT:-**

Using the intersect command to create a single more model from common space shared by two or more solids, when prompted to selected objects you can window select of the solid models or select them individually.

Command line: intersect

Menu→Modify→ solid editing→intersect

Toolbar→solid editing

### **EXTRUDING FACES:-**

Using extruding faces method, you can easily adjust the size of a solid model based on faces that are the results of combining multiple solids.

Command line: solid edit→face→extrude

Menu→Modify→ solid editing→extrude

Toolbar→solid editing

**COMMAND OPTIONS:-**

- **Path:-** Use to extrude a face in the direction & distance of a selected object this option sweeps the selected face along the selected path, you don't have the option to enter or taper angle when selecting a path

**MOVING FACES:-**

When you move face on a solid model you either locate the selected faces to the new location on the model . if you select all the faces that represent a destination aspect of your solid model you locate those faces

Command line: solid edit→face→move

Menu→Modify→ solid editing→move

Toolbar→solid editing

**Procedure:-**

The following steps are given & overview of move the faces of solid model

- Start the move face command
- Select the faces on the solid model you wish to move
- Specify the base point for the move
- Specify the second point to set the direction & distance of the move

**OFFSETTING FACES:-**

When you offset faces on a solid model, you either add material or remove it from the solid model. While being parallel to the selected faces. This is especially useful when you want to increase or decrease the diameter of a cylindrical face.

Command line: solid edit→face→offset

Menu→Modify→ solid editing→offset

Toolbar→solid editing

**Procedure:-**

The following steps given an overview of offsetting the faces of a solid model

- Start offset faces command
- Select the faces on the solid model you wish offset

- Specify the offset distance as a positive or negative volume

### **DELETING FACES:-**

Create & refine your solid model, you may decide that you no longer want to various aspect of the design e.g. you may have rounded edge or added material & you can use delete faces to remove those faces , the faces on the remaining solid model must be able to creat closed object .

Command line: solid edit→face→delete

Menu→Modify→ solid editing→delete

Toolbar→solid editing

### **Procedure:-**

The following steps are given an overview delecting face from a solid model.

- Start the delete face command .
- Select the faces on the solid model you wish to delete .
- If prompted to erase this solids history , click on continue & delete the faces . click canel maintain the solids history different method

### **ROTATING FACES:-**

When you rotate faces on a solid model . You either re-orient the selected faces to a new location on the model you by spining them around an axis or add. Material to or remove it from the solid model if you select all faces represent a distance aspect of your solid model, you relocated & reorient those faces.

Command line: solid edit→face→rotate

Menu→Modify→ solid editing→rotate face

Toolbar→solid editing

### **Procedure:-**

The following steps give an overview of rotating faces on a solid model

- Start the rotate faces command
- Select the faces on the solid model you wish to rotate

- Specify the first point on the axis you want to revolve around or select one of the options & specify the axis of revolution as per the options requirements
- Specify the second point on the axis to revolve around with also establishing to positive direction for the axis.
- Enter a value for the rotation angle

### **TAPERING FACES:-**

When you taper faces on a solid model, you slape the faces at an angle reletive to the position of the base point & direction you specify. Adding taper a design is especially beneficial when designning a point that will be cast & then require being removed forms or model. You can also use tapering to quickly a greater complex to your design.

Command line: solid edit→face→taper

Menu→Modify→ solid editing→taper face

Toolbar→solid editing

### **Procedure:-**

The following steps give an overview of tapering faces on a solid model

- Start the taper faces command
- Select the faces on the solid model you wish to taper
- Specify the base point from which the tapering will being to add or remove material
- Specify a second point to define a vector from the taper in measured
- Enter a positive or negative taper angle

### **CREATING A SHELL:-**

When you shell a solid model , you create a new solid model with a constant wall thickness, this constant wall thickness is defined by the parallel offsets of all exciting face on the solid model prior to exciting the cell. If you specify a positive offset value, the new faces are value toward the inside of the model. Specifying a negative.

### **SEPARATE SOLIDS:-**

Use the separate command to create two or more solid models from a single defined model exposed of two or more disjoined volume. A disjoined volume means the single solid model has solid mases of material that aren't connected. This can occur after doing a boplean union or substract.

Command line: solid edit→body→separate

Menu→Modify→ solid editing→separate

Toolbar→solid editing

### **Procedure:-**

The following steps give an overview of separating faces on a solid model with disjointed volumes into multiple solid models

- Start the separate command
- Select the solid model you want to separate, if prompt to erase the solids history. Click or to continue

### **CHANGING EDGE COLOUR:-**

When you are working on a design that has interface & edges & you want to see all aspect of the model, you may find it beneficial to display in 3D wire frame. That way you can see all edges on model unobstruct, by changing the colour of different edges , you can make it easy to view & intersect a wire frame display while as emphasizing the internal edge you are interested in.

Command line: solid edit→edge→colour

Menu→Modify→ solid editing→colour edge

Toolbar→solid editing

### **Procedure:-**

The following steps give an overview of changing that colour of edges on a solid model

- Start the color edge command & select a solid
- Select the edges you want to assign a new colour
- In the select colour dialogue box , select the colour or set by layer to have the edge inherit the colour of layer on which the solid model resides

### **COPYING FACES:-**

Use the copy faces command to copy selected faces of a solid model. The copied objects you create either regions or surfaces depending on the contour of the solid face you select. A planar solid face creates a region & all other faces create surfaces.

Copying is especially useful when you want to create a new solid model that matches some aspect of a current solid model. Since the copied face is an exact copy of the selected face, its size perfectly matches the current size of the selected solid model.

Command line: solid edit→face→copy

Menu→Modify→ solid editing→copy faces

Toolbar→solid editing

### **Procedure:-**

The following steps give an overview of creating a region & surface by copying faces from solid models

- Start the copy face command
- Select the faces on the solid model you want to copy
- Specify the base point
- Specify the point where you want to position the copied faces

### **COPYING EDGES:-**

Use the copy edges command to copy selects edges of a solid model. The copied objects you create are either lines , arcs, circles or splines.

Copying edges of a solid model can save your time in the creation of profiles , paths or guides to use in the creation of additional models

Command line: solid edit→edge→copy

Menu→Modify→ solid editing→copy edge

Toolbar→solid editing

### **Procedure:-**

The following steps give an overview of creating arcs & splines by copying edges from solid models

- Start the copy edge command
- Select the edge on the solid model you want to copy
- Specify the base point
- Specify the point where you want to position the copied edges

### **CHECKING SOLID MODEL:-**

You check your 3D models to ensure they are valid shape manager solids. Shape manager is the 3D geometric modelling engine from Auto desk that creates & manipulates 3D models.

When you check a solid arc manually, the test window display either this object is a valid shape manager solid or this object isn't a valid manager solid.

Command line: solid edit→body→check

Menu→Modify→ solid editing→check

Toolbar→solid editing

### **Procedure:-**

The following steps give an overview of checking a solid models to ensure it is a valid shape manager solid models

- Start the check command
- Select the solid model you want to check
- In the text window, review the return information for the selected model.

### **SHELLING SOLID MODEL:-**

Value offset the faces towards the outside . These by increasing the over all size of the model.

Command line: solid edit→body→Shell

Menu→Modify→ solid editing→shell

Toolbar→solid editing

### **Procedure:-**

The following steps give an overview of shelling a solid model

- Start the shell command
- Select the solid model you want to shell. If prompt to erase this solids history, click or to continue
- Select any face on the solid you don't want to shell in effect, removing it from the model when the shell is created.
- Enter the shell thickness

### **IMPRINTING EDGES:-**

When you imprint a solid model, you either score the face with the geometry or split the face the solid model into multiple faces for the geometry to imprint on the face. It must be co-planar with the solid model face.

Command line: solid edit→body→imprint



Menu→Modify→ solid editing→imprint edge

Toolbar→solid editing

### **Procedure:-**

The following steps give an overview of imprinting geometry into the faces of a solid model.

- Start the imprint command
- Select on object to imprint on a face. If prompted to erase this solid history, click or to continue
- When prompted to delete the source object, select yes to delete the object you are imprinting on the solid click no to keep it.
- Continue to select objects to imprint on to faces of the solid are reply to the delete sources object prompt.
- Press enter to complete the command & Imprint the faces of the solid model.

### **CHANGING FACE COLOUR:-**

You typically change the face colour on a model to enhance the display, emphasize a specific area or aspect of the model.

Command line: solid edit→face→colour

Menu→Modify→ solid editing→colour face

Toolbar→solid editing

### **Procedure:-**

The following steps give an overview change the colour of faces on a solid model

- Start the colour face command
- Select the face you assign a new colour to
- In the select colour dialog box, select the colour or select by layer to have the faces in merit the colour of the layer on which the solid model resides.

### **CHANGING EDGE COLOUR:-**

You typically change the edge colour a magic to emphasize a specific area or aspect of the model to return into displaying the colour of the layer on which the solid model resides

### **MOVING IN 3D:-**

Use the 3D move command to relocate objects in 3D space. During the operation of the command, you select the object to move & then select a plane by which to orient the move grip

tool. If an existing plane isn't selected, the move grip tool defaults to the XY-Plane of the current Ucs.

Command line: 3D move, 3M

Menu→Modify→ 3D operation→3D move

Toolbar→modelling

### **Procedure:-**

The following steps give an overview of moving objects in 3D space

- Start the 3D move command
- Select the objects to be move
- Select a plane by which to orient the move grip tool or specify a base point
- Select the 3D axis or plane required on the 3D move grip tool
- Enter the displacement

### **ALIGNING IN 3D:-**

Use the 3D align command to align. An object based on the location & the orientation of another object this is especially useful in situation where the exact angle of either object is difficult to obtained or where multiple rotations of an object would be required to properly orient the object.

Command line: 3D align, 3AL

Menu→Modify→ 3D operation→3D Align

Toolbar→modelling

### **Procedure:-**

The following steps give an overview of aligning objects in 3D space

- Start the 3D aligning command
- Select the objects to be aligned
- Specify the base point on the object to be align
- Specify the second point on the object to be aligning
- Specify the third point on the object to be aligning
- Specify the first destination point
- Specify the second destination point
- Specify the third destination point

## **ROTATING IN 3D:-**

Use the 3D rotate command to re-orient objects in 3D space. During the operation of the command, you select the object to rotate & specify the axis to rotate about. You can rotate about the X,Y or Z axis of the current Ucs or you can define a new orientation by playing the rotate grip tool on the face of a solid model

Command line: 3D rotate→ 3R

Menu→Modify→ 3D operation→3D Rotate

Toolbar→modify

## **Procedure:-**

The following steps give an overview of rotating objects in 3D space

- Start the 3D rotate command
- Select the objects to rotate
- However the rotate grip tool over the face of a solid model to orient by or select in the drawing area to use the current Ucs
- Select the band of the axis to rotate about
- Enter the rotation angle.

