## **Department of Mechanical Engg.** INDIAN INSTITUTE OF TECHNOLOGY, DELHI



# **MEL110 GRAPHIC SCIENCE**

Notes on

**Autodesk Inventor Professional Software** 

### Contents

- 1.What are the basic features of an Autodesk Inventor Professional Software? (Page:3) <u>1.</u> <u>What are the basic features of an Autodesk</u> <u>Inventor Pro...</u>
- 2.How is it useful in MEL110 course? (Page:4)
- 3.How can I explore existing solid models? (Page:5)
- 4.How can I create solid models-I? (Page:14 21)
- 5.How can I create a solid model using multiple extrusion? (Page 22-26)

**1. What are the basic features of an Autodesk Inventor Professional Software?** 

- It is a 3D mechanical engineering, design, visualization, and simulation software
- Autodesk Inventor is a parametric and feature-based solid modeling tool. It allows you to convert the basic 2D sketch into a solid model using very simple modeling options.
- It creates digital prototyping as opposed to physical prototyping (*which is costly and time consuming*) by integrating 2D AutoCAD drawings and 3D data into a single digital model
- It can quickly and easily create stunning renderings, animations, and presentations that improve communication
- It can easily generate and share production-ready drawings for manufacturing teams
- The automatic updating feature allows easy changes in models
- It has a simulation environment that allows motion simulation, static and modal finite element analysis (FEA) of parts, assemblies, and load-bearing frames

#### 2. How is it useful in MEL110 course?

- It helps in understanding the underlying concepts of graphic science much faster and in a better manner by a beginner
- It helps in improving visualization of the solid model and understanding its drawing
- A solid model of the object is what is required to be created in the first step
- and rest of the process like creating a standard drawing in either first or third angle projection method with different views (like orthographic, isometric, auxiliary, sectional, perspective) is completely automated
- The process of adding dimensions (with all standard notations) to the drawing is also automated
- It gives flexibility in editing the solid model to visualize the effect of the changes

 Open Autodesk Inventor by double clicking its shortcut on desktop or by selecting it from program list



• The initial screen will be displayed, as shown below

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#### 3. How to explore existing solid models? Go to Libraries>Documents>Autodesk>Inventor 2011>tutorial files>PivotBracket.ipt (or any other part file) and click open



 Go to Libraries>Documents>Autodesk>Inventor 2011>tutorial files>PivotBracket.ipt (or any other part file) and click open



7

•This screen will appear

This is MODEL TAB. It provides all
modeling tools that are used to convert sketch into feature



information about the model

•The **Quick Access Bar** is placed here, for easy access and use of very frequently used commands



•The commands are invoked from the tabs in the **Ribbon**. The **Ribbon** is a long bar below the **Quick Access Toolbar**.

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•View cube: It is displayed at the top right hand corner in the active area.



The tools on the **Navigation Bar** helps to control the view and orientation of components in the drawing window

•View Cube is used to switch between standard and trimetric views of the model •Click the 'Home button' to return to a user-defined base view •Click the cube corners to snap the 3D model to trimetric views •Click the faces to view orthographic views



•Navigation Bar is used to zoom, pan and rotate (orbit) the 3D model

• the model can also be explored using the VIEW TAB



To change the appearance of the solid model

- First create a new sketch and then use modeling operations to create solid model
- Click the 'projects' command located in the launch panel and select the 'default' project and click 'done'

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Create a new part file: In the 'Get Started' tab, go to new > Metric > standard (mm).ipt and click OK.

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This will open the sketch mode with a default file name '**Part1**' (you can later save this part with a different name. It will have an extension .ipt)

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If the sketch mode is not activated, then select sketch 1 from the browser window which is located on the left side below the main tab.

Select a working plane (XY, XZ or YZ plane) under 'origins' tab from the browser window to create a 2D sketch or use the default working plane.

Use the 2D 'sketch' commands to create the following sketch using lines.



Press 'esc' button on the keyboard to exit out of any command.
Now Dimension the sketch using 'dimension' command located in the 'constrain' command.



Select the 'Finish sketch' command to exit the sketcher after completing the sketch and 'close' the active file.

A new window will appear with Model tab activated

Now use the "Extrude" command to create a solid model of the sketch



A window will appear, enter the extrusion distance (by default its showing 10mm), and click **OK** 



This extrusion appears in the browser window and solid model is created.



•Now use the view cube and navigation bar to explore this solid model

To create a cylinder at the top of a hexagonal prism

- Open a new part file and create sketch of a hexagon
- click finish sketch



• In the Model tab click on the **Extrusion** command and extrude the hexagon to the required height and click **OK** 



•Select the top face of hexahedral by double clicking that face.

• Click the option Create Sketch to activate the sketcher window



Create a sketch of a circle of required diameter and click on finish sketch
In the Model tab click on the Extrusion command and extrude the hexahedron to the required height and click OK. The required solid model is complete.



•If the extrude direction is reversed, a hole will be created in the hexahedron



•Create a sketch of a right angled triangle, finish sketch





•Use **Revolve** command and select the triangle as profile

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•Select vertical side as Axis and Extents as full (360 degrees) click OK.









•Use **Revolve** command and select the rectangle as profile



•Select vertical side as **Axis** and **Extents** as full (360 degrees) click **OK**.



•A cylinder is thus created.

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•Create a sketch of a semicircle.



•Create a line which is to be used as axis about which to revolve the sketch, finish sketch



•Use Revolve command and select the semi-circle as profile


### 8. How to create a solid models-IV?

•Select the vertical line as Axis and Extents as full (360 degrees) click OK



### 9. How to create a solid model-V?

• Open a new part file and create a sketch of the base of the pyramid, click **Done** and **finish sketch** 

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#### 9. How to create a solid model-V?

• Use "Extrude" command and set extrusion distance equal to height of pyramid.



### 9. How to create a solid model-V?

• Now use **"More"** command in Extrusion and by trial and error set the taper angle so that pyramid is created

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#### 9. How to create a solid model -V?

continue trial and error to set the taper angle so that pyramid is created, click OK



### 9. How to create a solid model -V?

#### Pyramid is thus created



### 9. How to create a solid model -V?

Alternatively, create a point on offset plane and use "Loft" command to make pyramid



#### **10. How to add Datum Feature to a sketch/solid model?**

•Datum features are used during the construction of other features.

•Working planes, axes and curves are some of the common datum features.

- •Datum features do not change the properties of the model
- •In the Model tab, go to Work Features panel

•Select Plane command to define a work plane using feature like vertices, edges, faces,



44

### **10. How to add Datum Feature to a sketch/solid model?**

Some examples of Work planes



#### Work plane offset from face

Select:	A planar face. Click the edge of the face and drag in the direction of the offset. Enter a value in the edit box to specify the offset distance.
Result:	Creates a work plane parallel to the selected face at the specified offset distance.
3-point wo	rk plane
Select:	Any three points (endpoints, intersections, midpoints, work points).
Result:	Positive X axis is directed from first point to second point. Positive Y axis is perpendicular to the positive X axis through the third point.
Work plane	through two coplanar edges
Select:	Two coplanar edges.
Result:	The positive X axis is oriented along the first selected edge.

Adding Working planes

•Use of ribs or stiffeners is one of the most common ways of adding stiffness to localized areas of a structure.

•Ribs are usually attached to the surface of an existing structure by casting, welding, gluing or bolting.



- •Create a solid model as shown below.
- Click Free Orbit and rotate the object as shown below
- Click Done/Esc to come out of orbit command



Create a new **working plane** by activating the **Plane** command from the Model Tab. Select the option **Mid plane between two parallel planes** in Plane sub menu



This creates a new working plane (mid plane) between two parallel planes



- •Click at the corner of the plane and right click Create Sketch
- Create a line between two corners of walls in this working plane as shown below



•Use **Coincident Constraint** for coinciding the edges of the walls with the end points of the lines. Click on **Finish Sketch** Command and exit the sketcher window.



•Activate the **Rib** Command. Enter the rib thickness, select the direction by moving the cursor in the mid plane and **click OK** 



•This creates the required rib

To hide the mid plane, click the Work plane in the browser window and uncheck the Visibility option



• In the 'Get Started' tab, go to new > Metric > ANSI(mm).idw and click OK



#### A new drawing sheet will open with a default name Drawing1



Now activate the **Base** command to insert the solid model whose orthographic projections is required.

Select proper scaling factor, orientation and style (*with or without hidden lines*) of the drawing and click **OK**.



You will be prompt to save the part file if not done earlier. Save the part file.



The front view of the solid model will be created. Now use **Projected** command to create the necessary views (top view, side view etc) and click **create** 





The front view, top view and side view of the solid model will be created. Now use **Annotate** command to add dimensions to the orthographic projections



Click on **Dimension** command to add dimensions to the orthographic projections.

Add dimension to the orthographic projections by clicking the line/feature (that need to be dimensioned) and then dragging the mouse to a suitable distance to place the dimension line

Click right mouse button and select 'Done'

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The complete orthographic projections of the solid model are thus created



•Open the drawing file



Click 'Section' and draw a cutting plane line passing through the centre of the top view by selecting the mid point of the right-most vertical edge



•Draw the cutting plane through selected point, right click and click **continue** and then click **OK**.



### **13. How to create Sectional Views of the solid model?** This window will then open, click **OK** and the sectional view will be created

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After the sectional view is created, add center lines, axes etc to the view



### 14. How to create auxiliary views of the solid model?

Create a solid model as shown below and create its orthographic projections



### 14. How to create auxiliary views of the solid model?

•Now in **'Place Views'** tab, click **'Auxiliary'** and select the surface, perpendicular to which the auxiliary view is to be drawn.



### 14. How to create auxiliary views of the solid model?

 Click the side on which the auxiliary view is to be drawn and the auxiliary view will be created.



- A vertical cylinder of 60 mm diameter and 100 mm length is completely penetrated by a horizontal cylinder (40 mm diameter and 100 mm length)
- Axes of both cylinders bisect each other
- Create the orthographic projections of intersecting solids and obtain the curves of intersection



# **Procedure:**

Create the solid models of two cylinders using the "Revolve" command

In the Sketch mode, draw the two rectangles in such a way that, after revolution of these rectangles, the required cylinders are obtained as per the given dimensions



Select geometry to dimension

Create the vertical cylinder by selecting the 'profile' and by selecting the left side vertical edge as an 'axis' in 'Revolution' tab and click 'OK' to complete the solid.



Select Profile

### One cylinder is thus created


# For drawing the other cylinder, click revolution1 (in the browser window) then right click Sketch1/ Share Sketch



15. How to create curves of intersection of two solids? Create the horizontal cylinder by selecting this 'profile' and by selecting the upper horizontal edge of the rectangle as an 'axis' in 'Revolution' tab and click OK to complete the solid.



## 15. How to create curves of intersection of two solids? The two cylinders are thus created, intersecting with each other



## In "Get started" tab, open a new ANSI.idw file and create orthographic views of the solid model showing the curves of intersection. Add important dimensions



Select model or sketch geometry

A square prism of base 50 mm and height 80mm is resting on one of its base edges inclined at 40 degrees to VP. A horizontal cylinder of diameter 30 mm and length 90mm, having its axis parallel to both HP and VP, intersects the prism bisecting its axis. Create the solid model and obtain curves of intersection



Create square of 50 mm side, with one of its edge inclined at 40 degrees to horizontal reference line



78

#### Create the prism using Extrude command and make X-Z plane visible



Create a plane parallel to X-Z plane through one of the corners away from X-Z



### A working plane is created, select it to create a sketch



## Create a circle at required position on this working plane.



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## Extrude the circle asymmetrically on both sides of the plane to create the penetrating cylinder



## 15. How to create curves of intersection of two solids? Solid model is created, uncheck the visibility of the working plane



**Open a new ANSI(mm).idw file and create the orthographic projections of the** intersecting solids to obtain the curves of intersection

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Rectangle ABCD of given dimensions inclined at an angle of 45° with the vertical plane (plane 1)
Distance between line AB and point E is given

Measure length of line ED and DC
Measure angle DEC using Inspect tool



- •Create line AB of given dimensions on plane xy
- •Create vertical line, perpendicular to line AB

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87

16. How to use inspect tool to measure dimensions of a 3D sketch?
•Create a working plane 2 inclined at 45° with the vertical plane (plane xy)



•Project geometry on this working plane (plane 2)



•Create rectangle ABCD of given dimensions on this working plane (plane 2) and finish the sketch











•Click on Inspect 🔗 - 🔂 -Color Autodesk Inventor Pr menu to activate PRO MO Tools Manage View Environments Get Started Inspect it H H Use Distance Section Distance Angle Area Region Loop Zebra Draft Surface Curvature Δ Properties command to Measure Analysis measure length × 23 Model 👻 2 Measure Distance of line FD and F åå,  $\overline{\mathbf{Y}}$ F EC Part3 🗄 📂 Origin TZ Plane 🗇 XZ Plane 🗖 XY Plane X Axis Y Axis Z Axis Center Point 🖉 Sketch 1 Work Plane 1 🖉 Sketch2 🙏 3D Sketch 1 🙆 End of Part С



•Use Angle command to measure angle between line ED and EC



•Select lines ED and EC



The angle between the two lines will be indicated <sup>98</sup>