

Engineering High School Outreach Program

## Session # 6

# Creating Engineering Drawings in CAD

**Agenda**

- Extracting 3 Orthographic Views from a 3D Isometric Model.
- Creating an Engineering Drawing with a Title Block.

**Reference**

Inventor® R10, Chapter 8 pages 1-4, 7-13, 19-22

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## Why Make Drawings When We Have the Part File?

- Some automated computer aided manufacturing can directly use the “virtual” CAD model to create a “real” physical part.
- However, in most cases for traditional manufacturing, we need a dimensioned drawing.
- Engineers and technicians do not want to measure each feature of a part to know the size or location.

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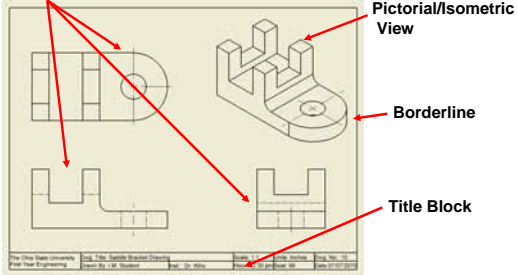
## 3D CAD Model of Saddle Bracket



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## Typical Engineering Drawing



**Orthographic Views**

**Pictorial/Isometric View**

**Borderline**

**Title Block**

**NOTE:** Dimensions, which are an important part of an engineering drawing, have been purposely left out for this assignment.

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## From Model to Drawing with Orthographic Views

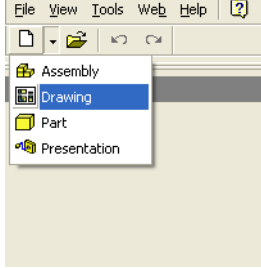
1. Open a model file containing a part - in this case the “Saddle Bracket” part file provided to you
2. Open a new drawing (XYZ.idw) file
3. Create a base view (normally the front view)
4. Project the other views needed to describe the object
5. Add the information for the title block

**NOTE:** We will learn how to add dimensions to the drawing later (next semester).

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## Starting a New Drawing



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## Screen Layout After Opening a New Drawing File

Drawing Views Panel

C-size sheet (22" x 17")

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## Changing to the Standard OSU Sheet

In the Model History area:

1. Click on the plus in front of Drawing Resources
2. Then the plus in front of Sheet Formats
3. Double click on A-Size-OSU Sheet

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## Changing to the Standard OSU Sheet

The OSU sheet (A-size – 8 1/2" x 11") And title block appear

Right click on Sheet:1 and select Delete to remove the C-size Sheet

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## Now You're Set !

Drawing Views Panel

A-size sheet (8 1/2" x 11")

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## Drawing Views Panel & Browser

The two functions that we will use in the drawing.

Default borderline and title block

Drawing Views Panel Browser for Drawing

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## First make sure that one part file is open! Start the 2D Drawing Using *Base View*

These steps are in Chapter 8, pages 10-21 of the Inventor book for a different part than we're using

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## The "Drawing View" Dialog Box

Open part file

Browse through the directory to locate file

Scale

Note: Default Is Front

**NOTE: The preview of Base/Front View is hidden behind the Dialog Box. Move cursor down to view the Base View**

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## Front View (*Base View*)

1. Move the cursor to the bottom left corner of the drawing sheet.
2. Left click to place the Base view.
3. Front view is added without any dimensions
4. The view can be dragged around for appropriate placement.

The Ohio State University  
First Year Engineering

Dwg. Title  
Drawn By  
Insp.

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## Insert Other 2D Views Using *Projected View*

File Edit View Insert Format T

Drawing Views Panel

Base View...

Projected View

Auxiliary View

Section View

Detail View

Broken View

Break Out View

New Sheet

Draft View...

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## Projecting Top View from Base View

Preview of Top View projected from Front View

Left click to Position view

Right click to Create View

**NOTE: You can position all the views you want and create them all at one time – just don't forget to right click and "Create" or you'll lose them!**

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## Top, Right Side and Pictorial View Created Using *Projected View*

Top View

Pictorial View

Front View

Right Side View

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## Modifying Views

- The *scale* and other options can be modified by right clicking near a view to bring back the Drawing View menu
- Views can be moved by placing the cursor near the view where a "move" icon appears, then clicking and dragging the view
- The front view can be moved in both directions; other principal views will remain aligned with the front view
- The "Drawing Annotation Panel" allows us to add Centerlines, Dimensions and Text to the views.

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## Functions Available in the Drawing Annotation Panel

We will use some of these tools as this assignment and other assignments are completed

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## Centerlines

Right click on the triangle of option under "Hole/Thread Notes"

NOTE: This option can change to any of those shown below depending on which of them was used last.

For this assignment, we will only use:

- "Center Mark"
- "Centerline Bisector"

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## Center Mark

produces centerlines for a circle or ellipse.

1. Left click on the center of the inner circle or ellipse to add center mark.
2. To extend the ends of centerlines, simply drag the green dots outside the edge of the outer circle or ellipse.

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## Centerline Bisector

produces centerlines for the side view of a hole.

1. Left click on the hidden lines representing the one side of the hole.
2. Left click on the second hidden line to add the centerline.

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## Editing the Title Block

1. Right click on "A-Horizontal" in the Model History Window
2. Select Edit Definition
3. Put your cursor on text in the title block
4. Right click and select Edit
5. Add your information to each block

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## Completed Drawing

Centerlines have been added to all views  
Title block must be Completed

NOTE: Dimensions have NOT been added yet. We will add them after learning the rules of good dimensioning next semester.

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## Review – From Model to Drawing

1. In general, you create a model or part first
2. Then you set up a sheet to draw on and insert a borderline and title block
3. A base view (normally the front view) is selected and inserted on the sheet
4. From the base view the other views are projected
5. Centerlines and Title Block information are added to complete the drawing

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## Today's Assignment

- The file for the "Saddle Bracket" part is provided – you do not need to create the 3D model.
- Create the Engineering Drawing from the "Saddle Bracket" part as shown in slide #24.
- Complete the information in the Title Block, save the drawing and print it.
- **Save/Retain the drawing file as we will need it for a future assignment!**

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