

PLTW
Introduction to Engineering Design
Notes

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Project Lead the Way
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(Uses some power points)

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1. Intro To Design

Brainstorming

Generate Ideas

No Bad Ideas

Set up questions and parameters before you start brainstorming

Have a monitor to keep the group on task

Can do it individually on paper

Brainstorm

1. Focus on problem
2. Moderation
3. Communicate
 - Written
 - Pictures
 - Spoken Word

Design Process- 5 Simple Steps

Design is solving of a specific problem

- Not a random process
- Incorporates a logical sequence of steps

1. Identify The Problem- Who does it affect

- Self
- Others

2. Plan Project- Determine the

- Layout
- Resources
- Schedule
- Direction

3. Problem Specifications

- Verify the exact problem
- Don't add unneeded items
- Is money something that is of importance (Does the design require extra cost due to complexity?)
- DO NOT PREDETERMINE THE SOLUTION

4. Conceptual Design

- Concept Generation- Brainstorm and research possible solutions
- Concept Evaluation- Look at all the possibilities and take components of those possible
 - ✓ Use design selection matrix to compare 2-4 designs
 - ✓ Use one as a datum and compare the rest to it
 - ✓ Give + or – for each area then take the best and use as your datum and repeat again and again until only one is left

5. Final Design

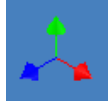
- Engineering Analysis- stress analysis /analytical software (Use analytical tools before you build)
- Prototype Testing- need to build to test functionality of design in an Engineering Analysis
- Design Refinement
- Final Testing
- Documentation- What gets developed for the consumer
 - ✓ -Drawings Presentation that includes data, testing results, and maintenance manuals
 - ✓ -Justify that your solution is the best

AutoDesk 3D Inventor Notes

Cartesian Coordinates

X Y Z

Orthogonal- Perpendicular to each other



Polar Coordinates

$$r^2 = x^2 + y^2$$

$$\text{tangent } \theta = y/x$$

Cylindrical Coordinates

Polar coordinates with the distance of the z axis (depth).

Absolute Coordinates

Coordinates that start from an established origin (0,0).

Relative Coordinates

Coordinates that start from a designated point

Spherical Coordinates

Coordinates that contain two angles and one distant

Sketching

Orthographic

Orthogonal Projections

Orthographic Principal Views

Oblique

Isometric

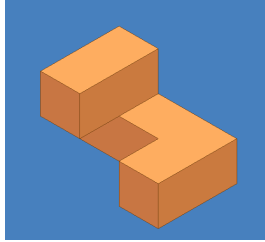
Perspectives

Puzzle

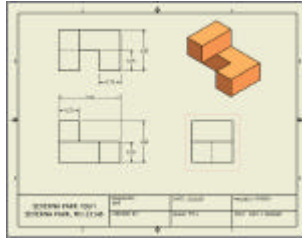
6 Puzzle Pieces

2 Solutions

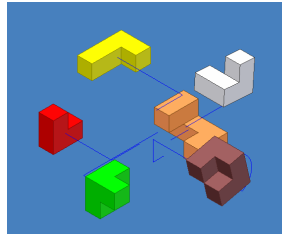
Iso's and CAD Both



Ortho's and CAD Both

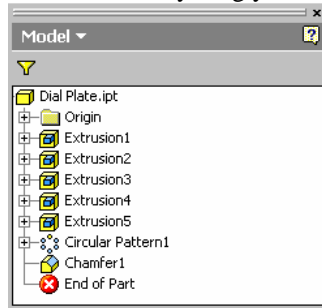


Exploded ISO Assembly 1 Puzzle
Dimension Ortho's 1 Puzzle
Assembled ISO 1 Puzzle

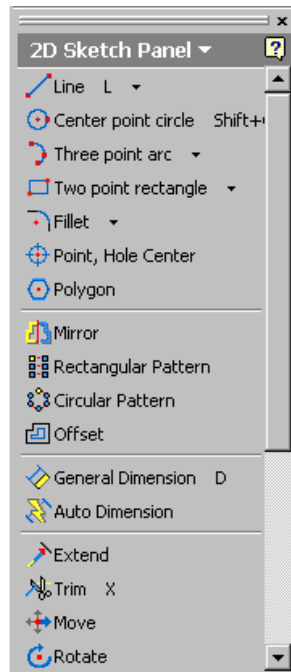


Inventor

Browse Bar- Tracks everything you've done



Panel Bar-



Most Parts standard

.ipt (Inventor Part File) – For Parts
 .iam (Assembly)
 .ipm (Presentation) – For Animation
 .idw (Drawing Orthographic)



New Project with multiple parts- Start a “new project”
 Every new project needs a folder!!!!!!!!!!!!!!
 Open



Click New Puzzle Project

Name Puzzle Cube

Designate File Location- Add name at the end of the file "Puzzle Cube"

Finish Naming and click Done

Hold Wheel and pull (0,0) to bottom left on pan tool

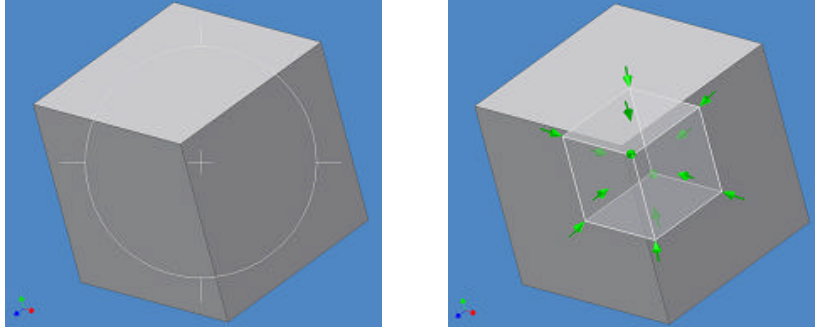
Minor Grid- 1/8"

Major Grid- 1"

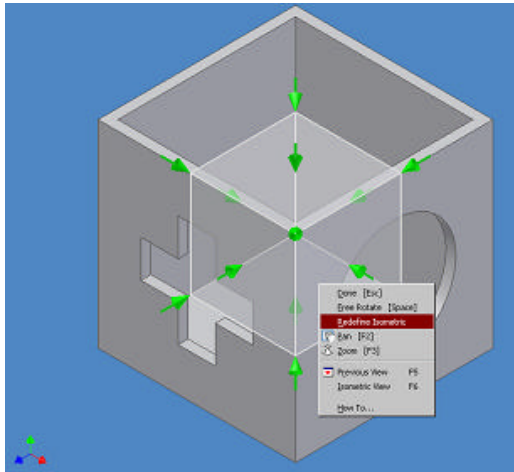
Right Click Isometric (Do NOT Click 3D Editor)

Look At Button (Opening Book)

Rotate Tool "Space Bar" allows you to toggle to common view or Right Click



To reset the isometric view, right click while in the common view rotate tool, and click "redefine isometric."



Unconsumed Sketches

Any sketch that is in purple is not a consumed sketch, meaning it has not been used up by an extrusion, hole, etc. These sketches must be deleted. To delete, go into the browser bar, select the sketch and delete it.

Assembly – Standard iam

1. Place Component
2. Click and use
3. Use rotate to get orientation close then space for common view to click “nose on” arrow to provide the isometric view
4. Place the component and all other pieces
5. Rotate component close to its desired position

Constraints

- Contain starting points
- 1st part surface
- 2nd part surface
- Use mate and finish to assemble parts

*You can edit parts using the “browser bar” (Model)

Custom Templates

Standard idw

To add automatic name- Left Box- Type Properties

Browser

Sheet 1 – Right Click

Editor Sheet- Select ANSI A- then ok

Delete ANSI Large and delete default boarder

To get to default border- right click and insert new border.

Right click title block and define as new

Draw title block- then right click and save as

Browser- select new title block

File save C- Program files

Autodesk Inventor 10 Completers- Give it a name and save

To change open browser and select edit definition

After the SPHS template is opened- bring in all of your views.

When prompted for text- make a text box
Type- PROMPT ENTRY

To Edit Title Block

- Browser
- Sheet 1 or Appropriate Sheet
- Expand Title Block
- Field Text
- Edit Field Text

Bring in projected view and arrange right top isometric- right click create

Panel Bar change drawing view panel to drawing animation

Part 2

“NEW” SPHS Title Block
Enter Info

Preview and select direction

Projected View

- Top
- Right
- Isometric
- Right Click Create

Drawing View Panel To Drawing Animation

Select Isometric- Right click
Then edit view – Style- Shade

Parts Sketch

Tools- Document Settings

“Sketch” -change grid minor line 8 for $\frac{1}{8}$
4 for $\frac{1}{4}$

Tools

Application Options

Colors recommended depending on presentation or presenter

Display- take off edge display to eliminate lines on round objects

Then swap to Grid

- Right click done
- Right click snap to grid

Assembly Animation

Standard .ipn- Create View (from Assembly)

Open and tweak components

- Move components to finish and hit close to move to next piece

Animation

- Intervals of 15 are good
- Play to see if it is good
- Edit if needed
- Record- save- ok- ok- record button- play

When finished take off record and import into power point

Plane of Origin

Default is XY

Strategize How You Are Going To Draw

You are going to be adding and subtracting material

Think about how to orient your drawing

Lofting

Sketch 1st face to be lofted; Finish Sketch

Add a new work plane and move it back to sketch 2nd face; Finish Sketch

(Make sure that each face is closed, otherwise the loft will not work properly)

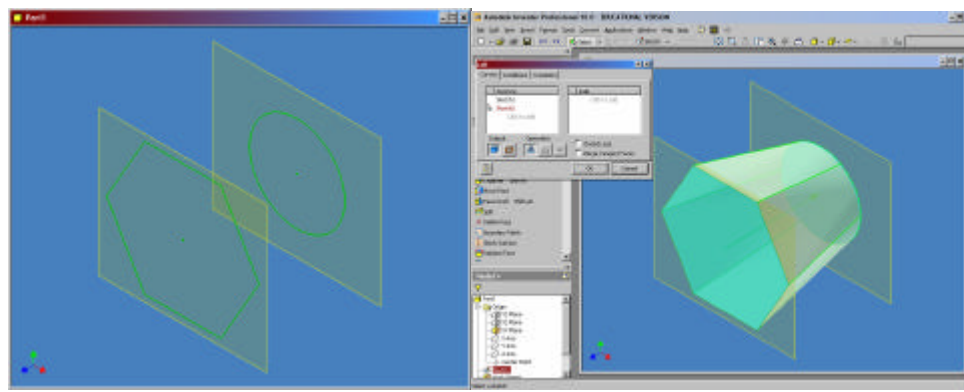
Select Loft from part features

Click Sketch 1 from “model”

Click Sketch 2 and click “Ok”

Should pick up intro

Select remove then ok



Orthographic

Center lines- Drawing animation

Center Mark

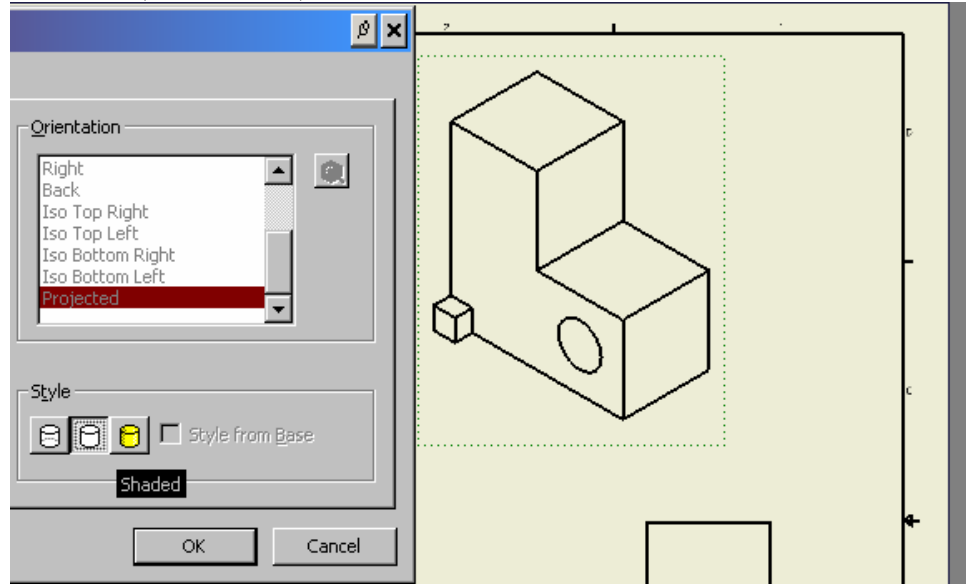
Center Bisector

Center Platter

Shade isometric in orthographic by selecting view (red marching ants)

Right Click “Edit View”

Shade Icon (Yellow Barrel)



Work Axis

Part Feature

Work Axis

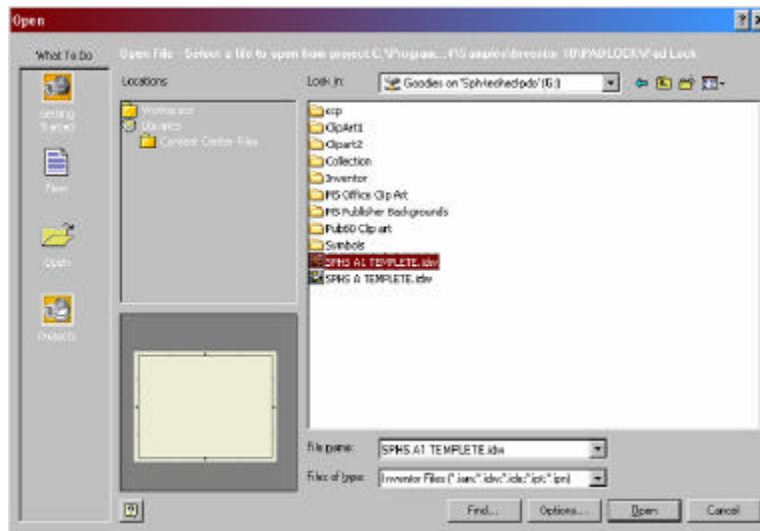
Select Sketch- Model Expand Origin and Select YZ

Right Click Slice Graphic

Sketch Panel and Project Geometry to show edges

Balloons (Puzzle Project)

New SPHS .IDW



Fill in information- Base View

Open Puzzle

Open IPN

Select View- Red Marching Ants

-Right Click

-Edit View

If to big select view again

Edit View and Scale To Size

Then Drawing Animation

Select Balloon

-Select part and right click on an edge (Don't hide arrow on surface)

Dialogue Box is OK- Left Click- Right Click- Continue

Parts List

-Drawing Annotation

-Parts list- OK

Double Click on List to EDIT

Move color to description

Drilled Holes

1. Make a center point in sketch mode
2. Select hole in Part Feature

Keyways

1. Complete Hole
2. Draw Rectangle Any Size
3. Dim To Correct Size
4. Right Click- Done
5. Right Click- Constraints
6. Select Coincident
7. Click Circle- Click Corner Of Rectangle
8. Click Circle- Click Other Corner

Share Sketch- Allows you to do more than 1 extrusion

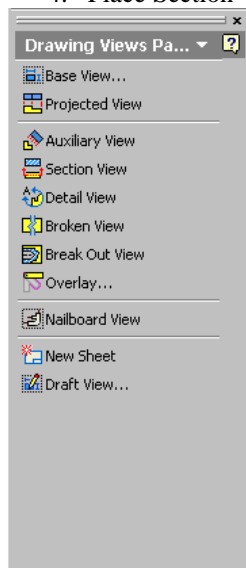
1. To eliminate Dimension in isometric right click on sketch in model and click off visibility
2. In model select the extrude and expand
3. Select sketch- Right Click and shape

Sections

Drawing View Panel

Section View

1. Click on view and get solid red box
2. Click to section points at solid red line
3. Right click continue
4. Place Section View



Note: You can select centers and midpoints by moving the cursor in the general area

Editing Text

Symbols Such As Depth

Select the text, dimension estimate to edit

Right click symbols are right side

Assembly iam

Open- Part

- Orient the isometric how you want to make true isometric in common view.
- Right click redefine isometric
- Rotate component is needed
- To do centers click an edge of one hole and move other part to use its hole with vertical line.
- You can also move components without changing constraints by selecting “move component”
- Inserting pins in holes- Select constraints and insert icon
 - ✓ Inserting Pins
 1. Applying insert constraint you need to have face of head pin highlighted as wide as the face
 2. Flip icon in constraint allows you to reverse direction of component
 3. May constrain work axis and work planes

To Animate an Object - Drive Constraints

Complete ipt's for Air Compressor

Complete iam assembly

- Select Flywheel
 - Place Work Plane
 - XZ or YZ
 - Bring it close to center
- Select constraints- Angles
- Select work plane- Select base and apply
 - Right Click in model (browser)
 - Find the work plane and right click and turn off visibility
- Model Browser- Select angle and right click
- Drive constraint and fill in end degrees (change increments)

Applying Drive Constraints

Used for such things such as gears

Activate the assembly panel

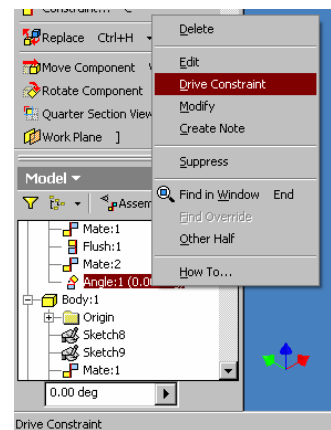
Go to the angle in the Model

-Right Click "drive constraint"

To see if tools activate contact solver

-Hover over gear (make real)

-Right Click and make sure the contrast is set



Curved Lines In A Sketch

Spline- In line fly out

-Right click done- click on mode and you can move

Polygon Tool- Just like in turbo CAD but you can pick the corner or the middle of the line.

-You can also constrain with constrain fly out just below locate.

Shell- Tool hollows out extruded object

Work Plane- Under Parts feature

-You can select an existing plane

Ribs- Inside hollow object

-Draw a line at the angle designated for a rib

Sweep- Draw something circular and will finish sketch

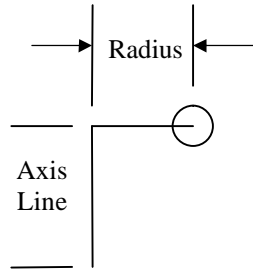
-Draw a path (starting from the center point of circle) perpendicular to circle and trim the excess

-Under parts feature- Sweep

Coil

Select new IPT

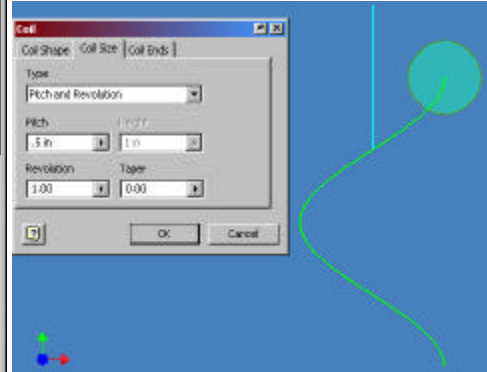
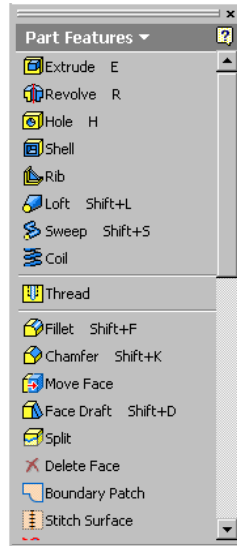
-Vertical line is



-Finish sketch

Coil

- Select axis
- Select direction
- Coil eye tab
- Pitch
- Revolutions
- OK- Finish

**Changing Hidden Lines And Center Lines**

- Can stretch by click and pull
- Can change scale by selecting and right clicking

Rectangular Pattern

Select geometry (hole, cube)

Direction- Click on edge of object and turn geometry on.

Hole Threads

- On drawing annotation panel

Scale

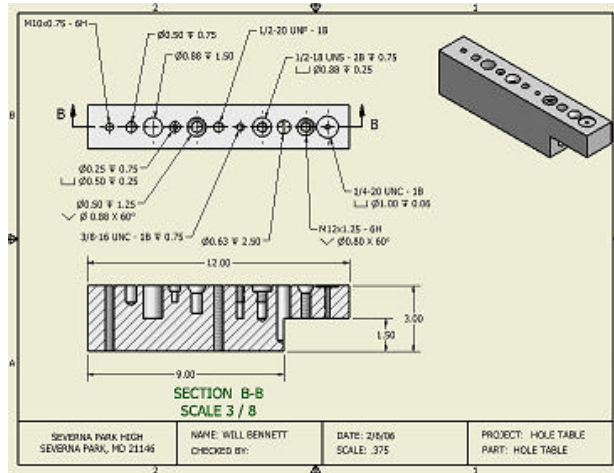
- Right Click on drawing surface
- Edit drawing and change scale

Precision Inputs

- Utilizes the X, Y, and Z-axes.
- View- Toolbar- Precise Input

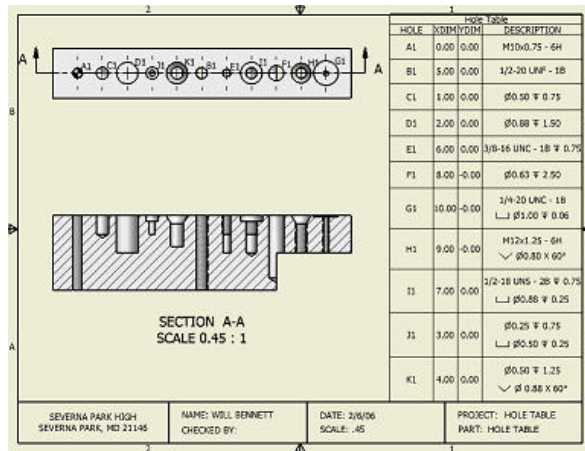
Hole Callout Assignment

1. Draw
2. Holes on separate sketch
3. Section



4. Hole Table

- a. Drawing Annotation
- b. Down to hole table- hole table view
- c. Click at the corner of the first hole
- d. Place Table



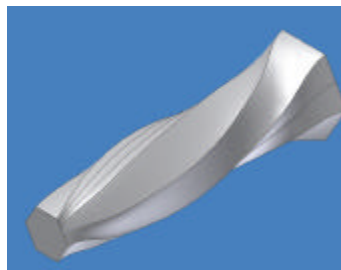
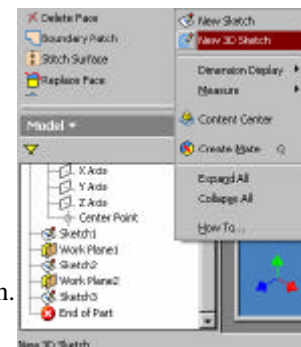
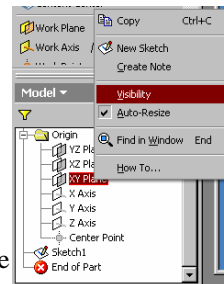
5. Make Threads in "HOLE TOOL" NOT USING THE THREAD TOOL!!!

Blending 2 Shapes

- Draw Shape and Extrude
- Insert Work Plane
- Offset Work Plane
- Draw Another Shape And Extrude
- Select Loft
- Select At Face- Done

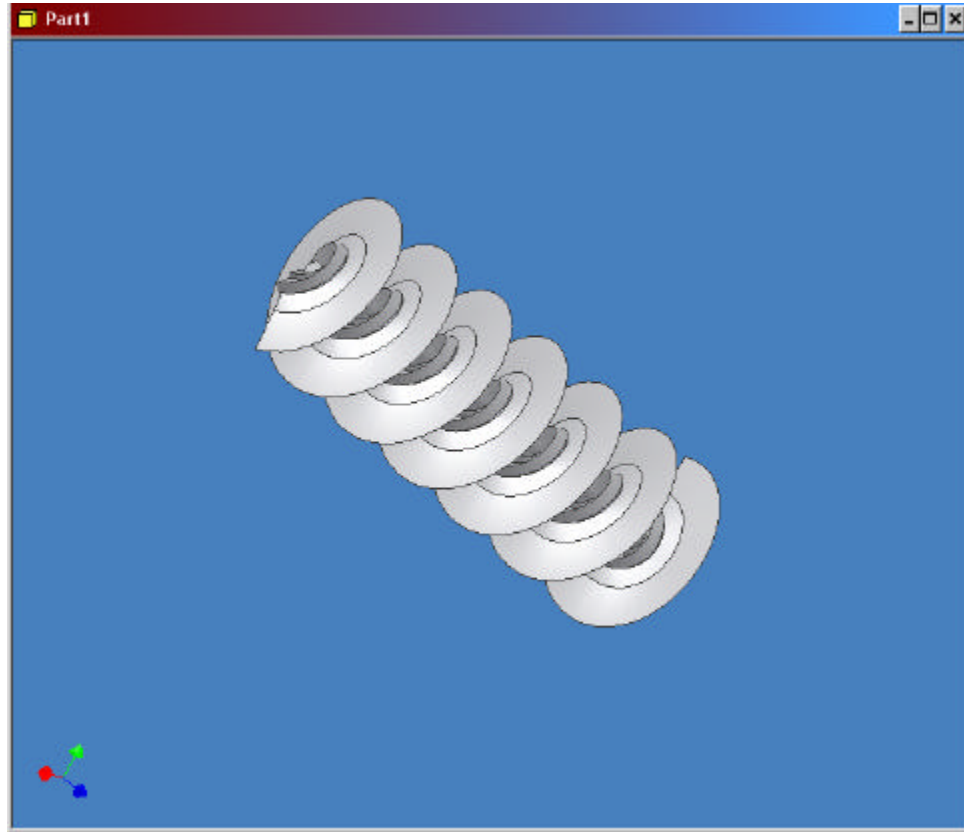
Twists

1. Open .ipt
2. Draw a shape
3. Finish Sketch
4. Go into Origin, apply visibility to XY Plane
5. Go to Isometric view
6. Use New Work Plane feature to offset the XY plane (makes Work Plane 1)
7. On new work plane, draw same shape, only angled 45°
8. Finish Sketch
9. Offset Work Plane 1 to make Work Plane 2 (continue in same direction as before)
10. Draw same shape, again angled 45° from the previous sketch
11. Finish Sketch
12. Go to Isometric View
13. Begin new 3D Sketch
14. Use the Spline tool (flyout of Line) to connect corresponding points starting from the farthest sketch (Origin Work Plane) to the middle (Work Plane 1) and finally Work Plane 3.
15. Use Loft feature to select first the sketch of Origin Work Plane, then Work Plane 1, finally Work Plane 2 (MUST be done in order). Then select the “rails” that were created in the 3D Sketch.
16. Finish Loft.

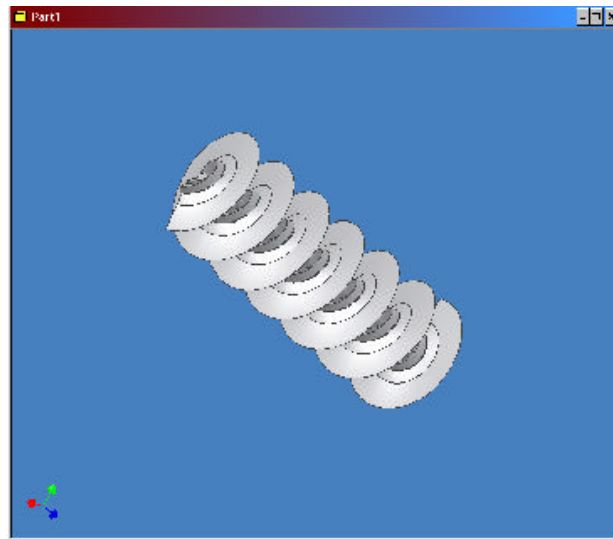


How to Use the Coil Feature

By: Ryan Janes & Shawn Laser



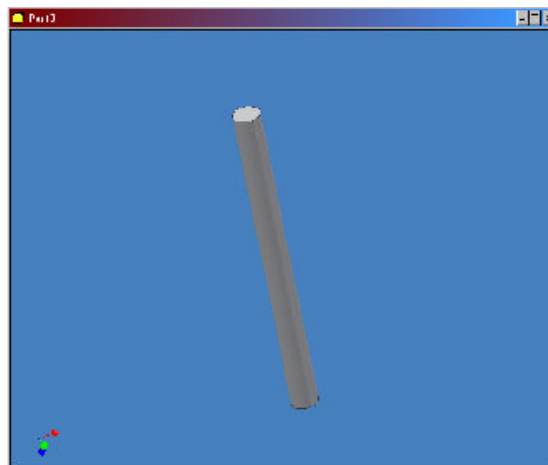
How to Use the Coil Feature



The coil feature is a very easy feature to use once you understand the process but it can also create some of the most impressive designs. For example, this piece only took about 2 minutes.

How to Make This...

1. First begin by making a simple cylinder with the extrusion or revolution feature. The size does not matter as long as you begin with a cylinder.



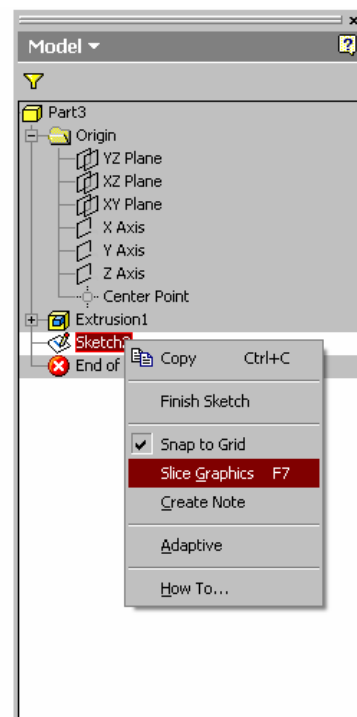
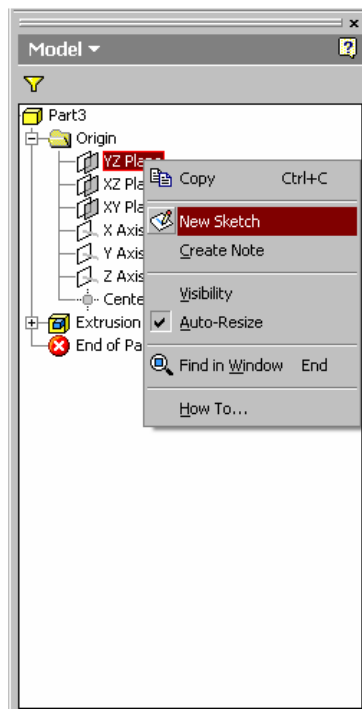
2. In the Model Tool Bar, select the pending axis. The correct axis will demonstrate a vertical slice your cylinder.

-Revolution: XY Axis

-Extrusion: YZ Axis

Right click and select *New Sketch* and you will see your plane cutting the cylinder.

3. Now that you plane has divided your cylinder, the *Sketch (#)* is highlighted. Right click the Sketch (#) and select *Slice Graphics (F7)*.



After selecting the Slice Graphic Feature, Your sketch plane will be working on the divided section of your cylinder drawing.

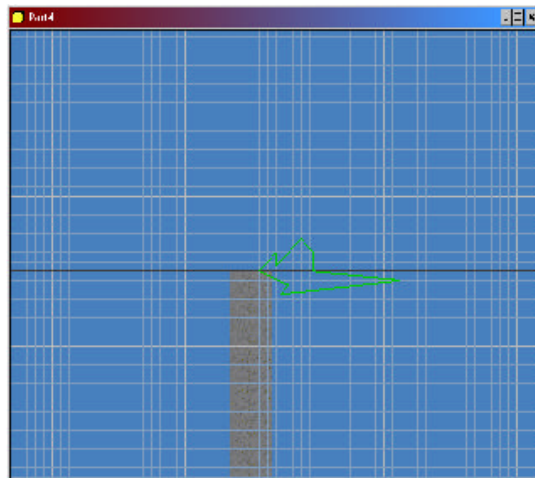


4. Next create a square shape in the upper left corner as shown here. This is the only part that involves creativity as it will generate the shape of the thread.

5. For now, we're using this shape because it is a simple. Later when you better understand the process, you may wish to complicate the thread. I used a different sketch to imitate an auger bit.

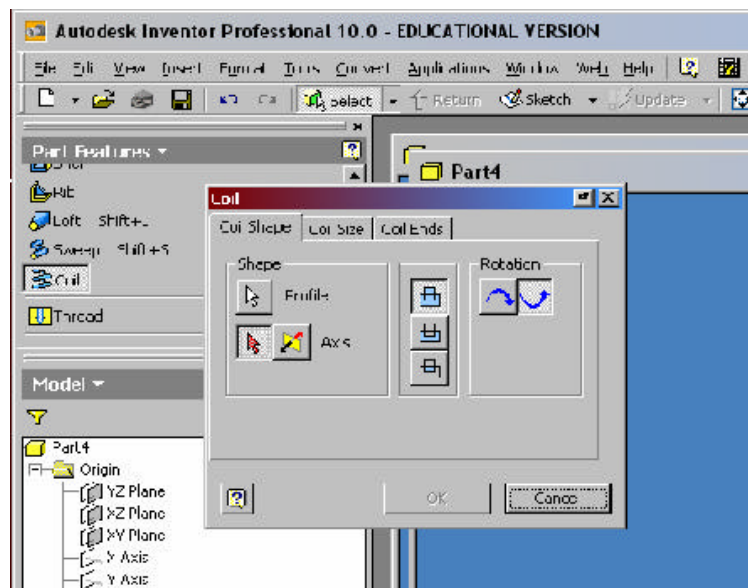


This is my sketch.



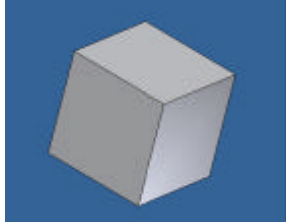
6. Exit sketch mode and select the Coil feature. Next select the axis that generates the centerline of your cylinder.

7. Using the various attributes available to you, you can create different pitches and sizes that can make your coil more appealing.

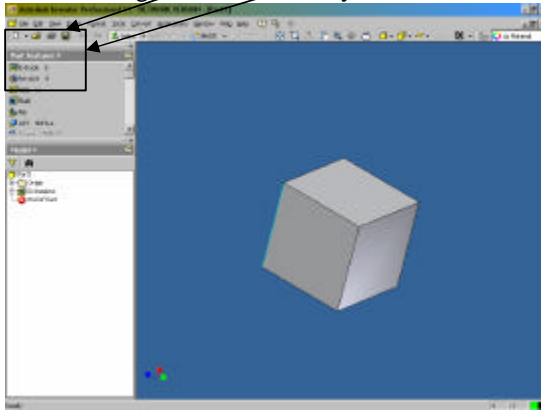


Stress Analysis on Autodesk Inventor 11 By Andrew Bork

1. To start, make a simple 1" square cube

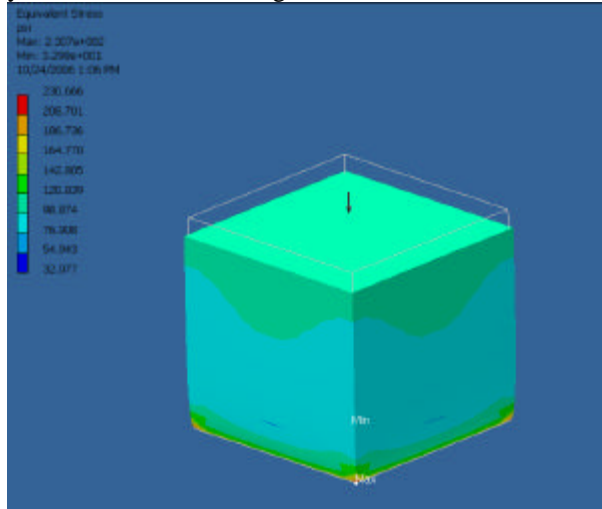


2. Then Click on the Part Features toolbar and click on the dropdown menu to change it to Stress Analysis



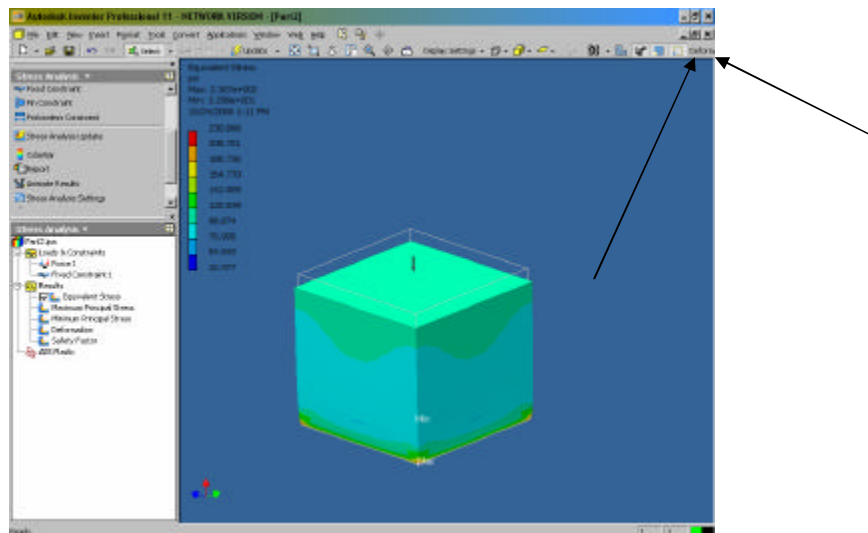
3. A popup menu will appear. Chose your specified material. The rest of this tutorial will use ABS Plastic.
4. Click on the force button in the new panel.
5. Now click on the cube where you would like the force applied.
6. The amount of force applied is represented by the number in the white box. You can change it to however much you want.
7. Without constraints all the force would do is push the block in the specified direction. Click on the fixed constraint button and click on any face or edge except the one you applied the force to. You can constraint multiple faces or edges to see different results.
8. now click on the "stress analysis update" button

9. you should see something like this



10. The gray outline represents the former position and the colored section represents the deformation based on the force applied and the constraints. I applied a 100lb force to the top and constrained the opposite face so that the block was compressed.

11. If you change the amount of force and it looks like the same picture, click on the “Deformation Style” button on the toolbar and make sure the scale is set to “Actual”



Physical Properties Of a Part

Use for Sports Bottle - volume

File- Properties- Physical Properties

Cones/ Bottles Etc.

- -Make a bottle using a shell
- -Draw half of the cone and add center line
- -Revolve the cone

Screws

Content Counter- Categories