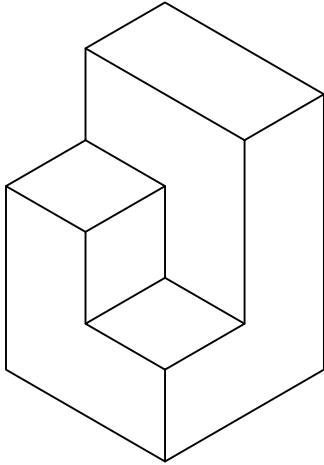
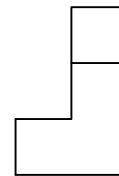
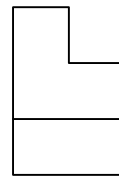
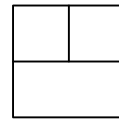


UNIVERSITY OF NEW HAVEN
Tagliatela College of Engineering
EASC 1107: Introduction to Engineering
Fall 2018

Orthographic Drawing and Inventor
Handout-7

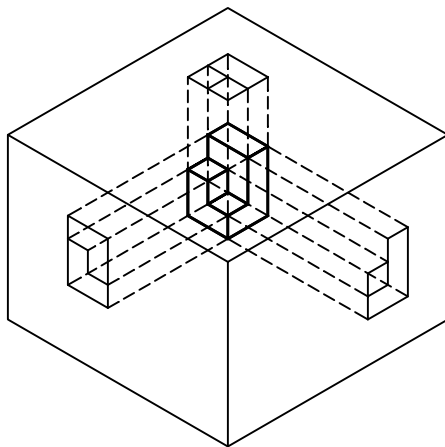


Isometric Drawing

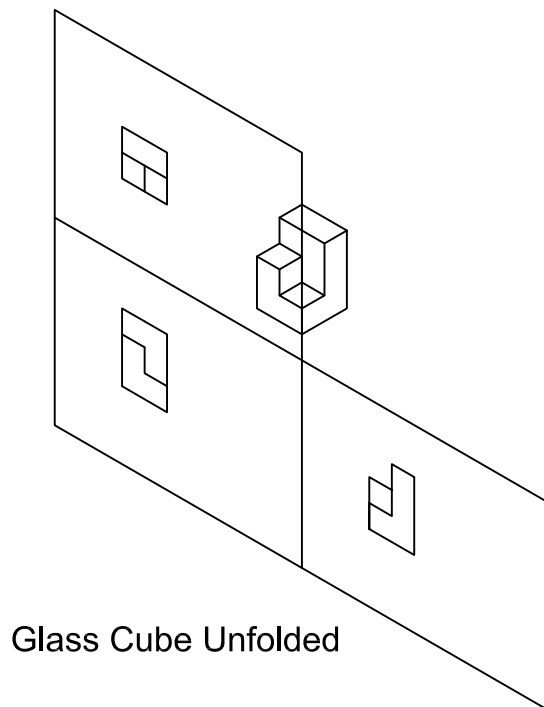


Orthographic Drawing

In creating orthographic views, you imagine that the object is surrounded by a transparent glass cube. The edges and surfaces of the object are projected onto the panes of the glass cube and the cube is unfolded so the panes of glass lie in one plane.

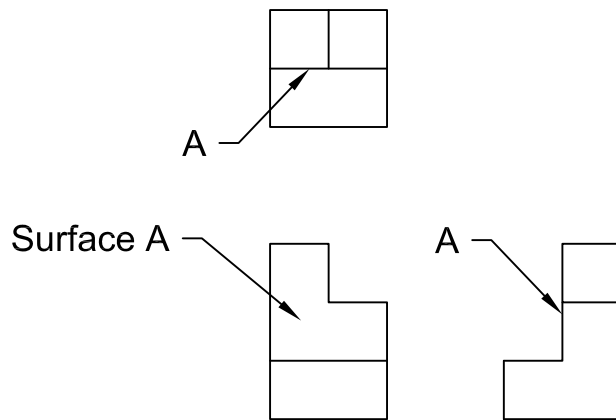


Object Surrounded by
Glass Cube

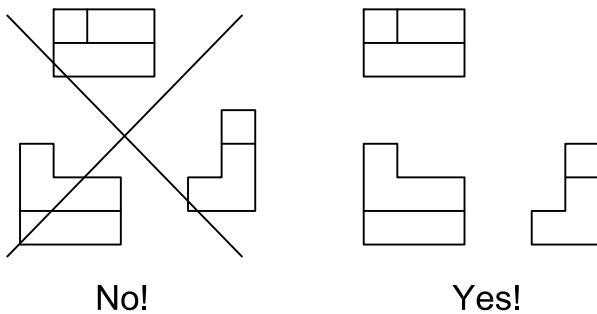


Glass Cube Unfolded

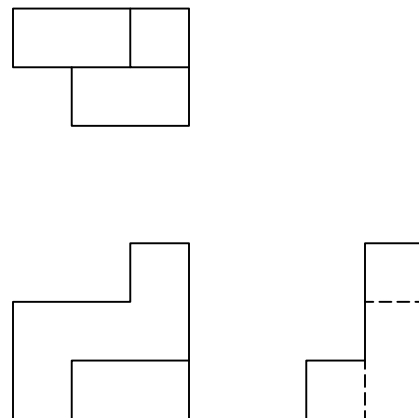
Normal surfaces are defined as being parallel to either the top, front, or side views. A normal surface is seen as a surface in the view to which it is parallel and is seen as an edge in the other views.



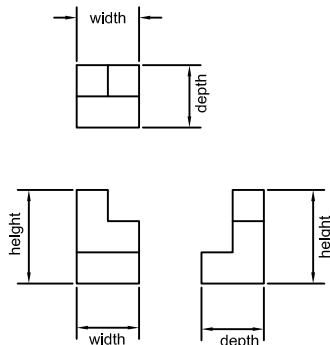
Rules in orthographic projection:



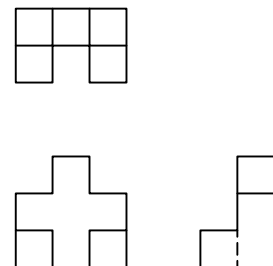
The orthographic views of an object should be aligned with one another.



Edges of an object that are hidden from a given viewpoint are shown as dashed lines in that view.

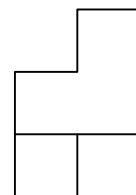
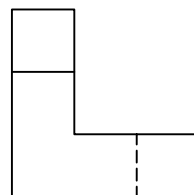
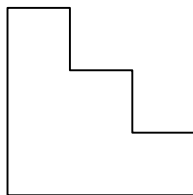
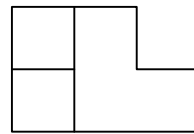
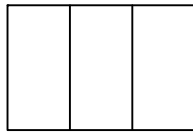


The top view shows the width and depth of an object, the front view shows the height and width, and the side view shows the height and depth.



When a hidden line coincides with a solid line on a view, show only the solid line.

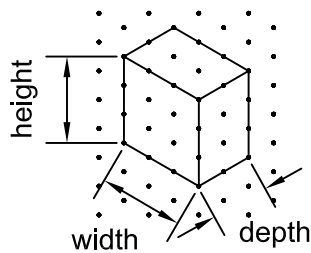
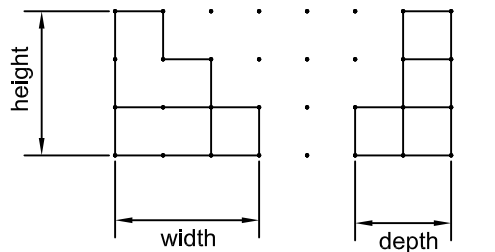
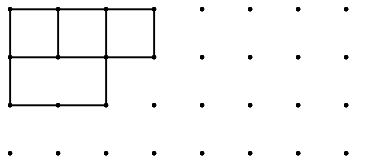
Sometimes only two views are required to completely describe an object. However, most times, three views are shown (top, front, and right side).



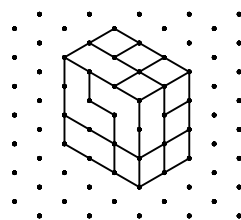
Only 2 Views Required

3 Views Required

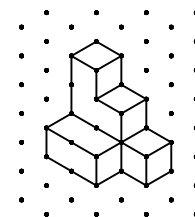
The box method is sometimes used to create an isometric drawing from three orthographic views. Step1: Construct an isometric drawing of a box that is the same overall size as the object seen in the orthographic views. Step2: Draw the top, front, and side views on this box. Step3: Add and erase lines from the isometric drawing to complete it.



Step 1



Step 2



Step 3

An isometric view of an object is shown below along with its top and front views. Circle the letter corresponding to the correct side view from the choices given

1.

A B C

2.

A B C

3.

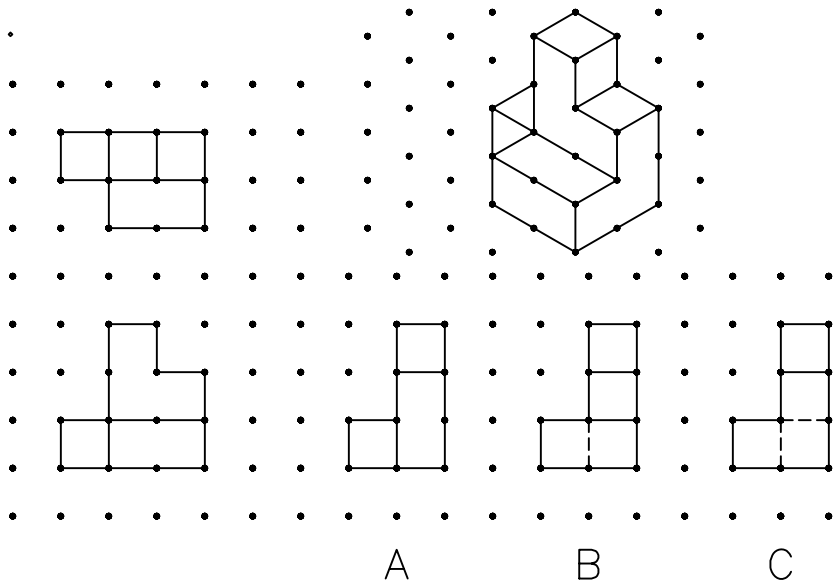
A B C

4.

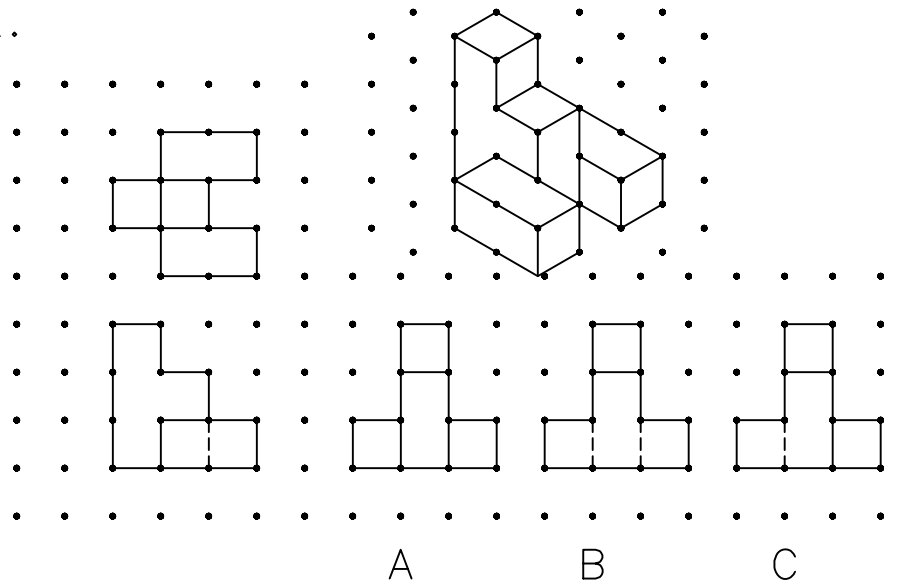
A B C

An isometric view of an object is shown below along with its top and front views. Circle the letter corresponding to the correct side view from the choices given

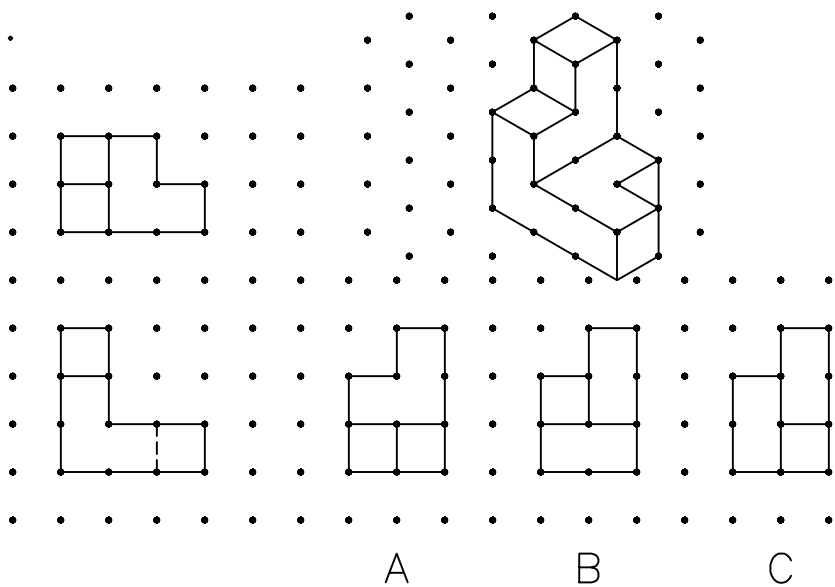
1.



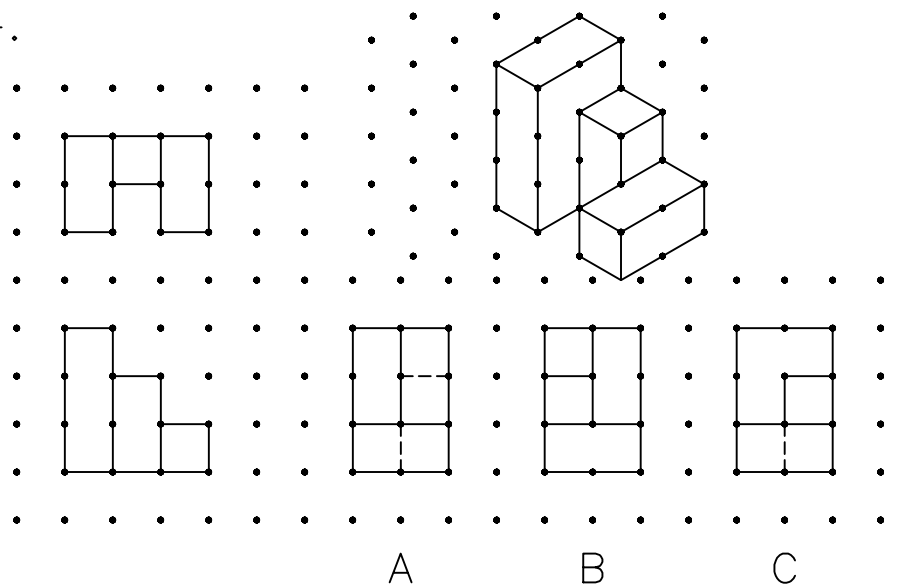
2.



3.



4.



An isometric view of an object is shown below along with its top and front views. Circle the letter corresponding to the correct side view from the choices given

1.

Isometric view of a 3D object. The object consists of a base rectangular prism with a smaller rectangular prism on top of its left side. The top view shows a 3x2 grid. Three side view options are provided: A shows a 2x2 grid with a dashed line; B shows a 2x2 grid with a dashed line; C shows a 2x2 grid with a dashed line.

A B C

2.

Isometric view of a 3D object. The object consists of a base rectangular prism with a smaller rectangular prism on top of its left side, and another smaller rectangular prism on top of its right side. The top view shows a 3x3 grid. Three side view options are provided: A shows a 2x2 grid with a dashed line; B shows a 2x2 grid with a dashed line; C shows a 2x2 grid with a dashed line.

A B C

3.

Isometric view of a 3D object. The object consists of a base rectangular prism with a smaller rectangular prism on top of its left side, and another smaller rectangular prism on top of its right side. The top view shows a 3x3 grid. Three side view options are provided: A shows a 2x2 grid with a dashed line; B shows a 2x2 grid with a dashed line; C shows a 2x2 grid with a dashed line.

A B C

4.

Isometric view of a 3D object. The object consists of a base rectangular prism with a smaller rectangular prism on top of its left side, and another smaller rectangular prism on top of its right side. The top view shows a 3x3 grid. Three side view options are provided: A shows a 2x2 grid with a dashed line; B shows a 2x2 grid with a dashed line; C shows a 2x2 grid with a dashed line.

A B C

An isometric view of an object is shown below along with its top and front views. Circle the letter corresponding to the correct side view from the choices given

1.

A

B

C

2.

A

B

C

3.

A

B

C

4.

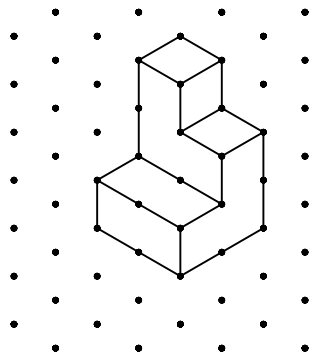
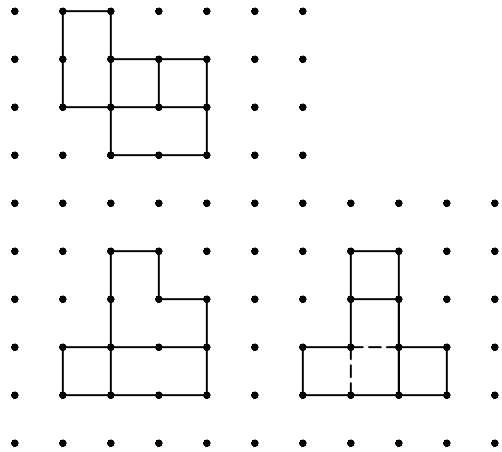
A

B

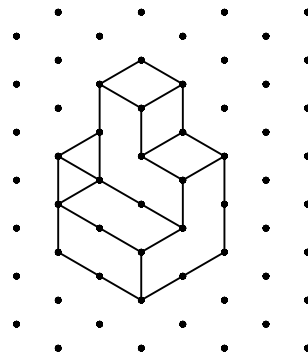
C

For the object shown in orthographic projection on the left, circle the letter of the correct corresponding isometric view.

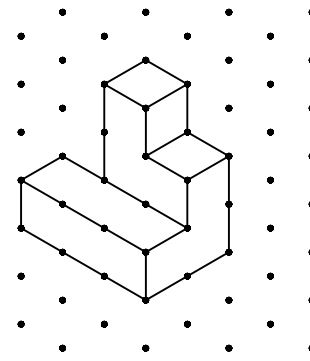
1.



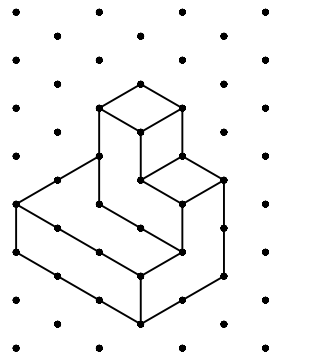
A



B

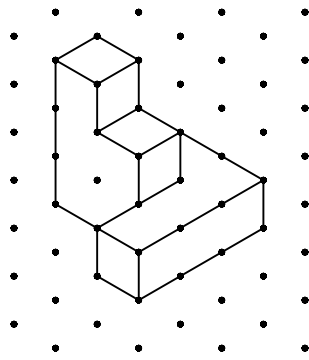
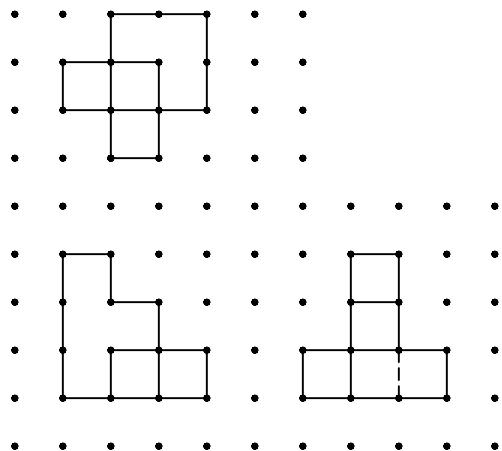


C

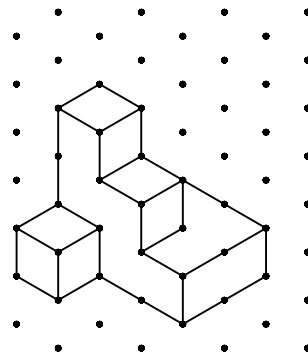


D

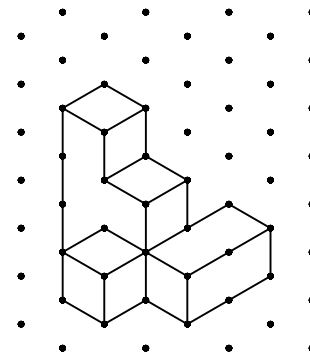
2.



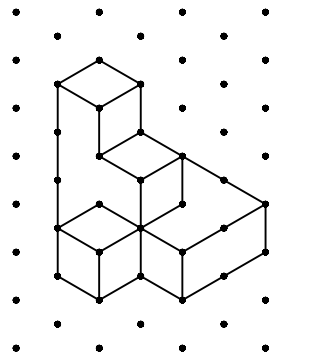
A



B



C



D

Name:

Class:

Section:

Date:

Developing Spatial Thinking

Grade:

Page

ortho-09

Circle the letter corresponding to the correctly aligned set of orthographic views.

1.

A B C D

2.

A B C D

Circle the letter corresponding to the correctly aligned set of orthographic views.

1.

A B C D

2.

A B C D

Circle the letter corresponding to the correctly aligned set of orthographic views.

1.

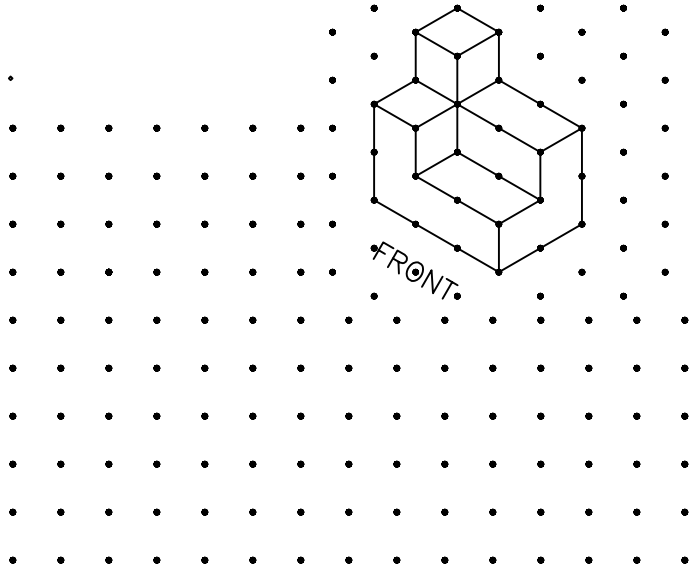
A 4x4 grid of orthographic views. The first row shows four different sets of three views (top, front, and side) of a 3D object. The second row shows four sets of three views labeled A, B, C, and D. Each set consists of a top view, a front view, and a side view. The goal is to identify which set is correctly aligned.

2.

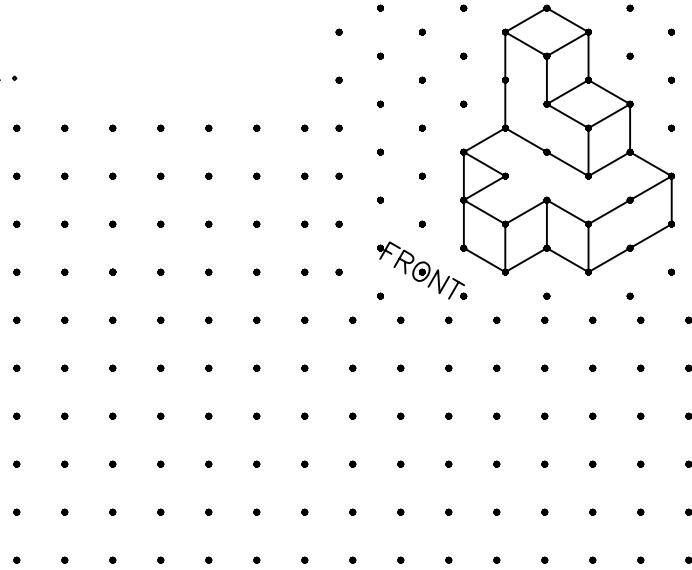
A 4x4 grid of orthographic views. The first row shows four different sets of three views (top, front, and side) of a 3D object. The second row shows four sets of three views labeled A, B, C, and D. Each set consists of a top view, a front view, and a side view. The goal is to identify which set is correctly aligned.

For the objects shown in isometric below, sketch the top, front, and right side views in the space provided. Make sure that your views are properly aligned.

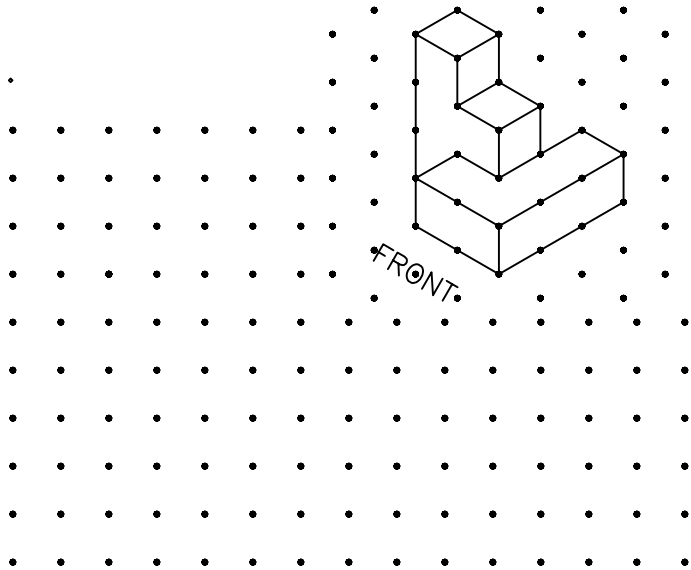
1.



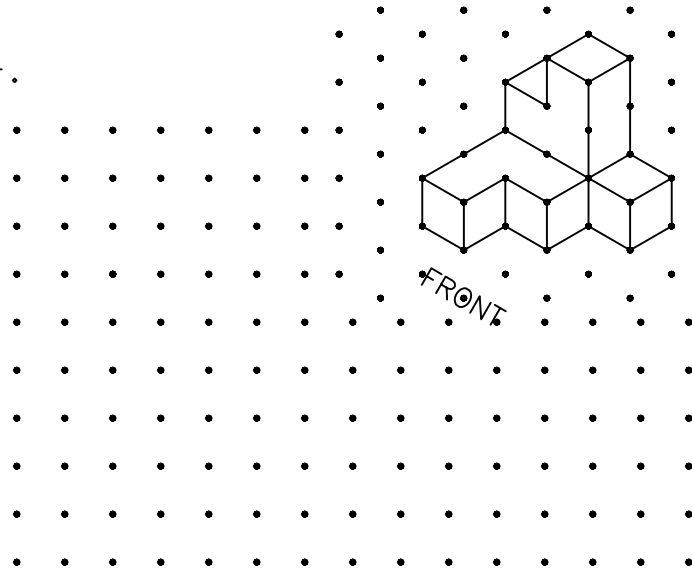
2.



3.

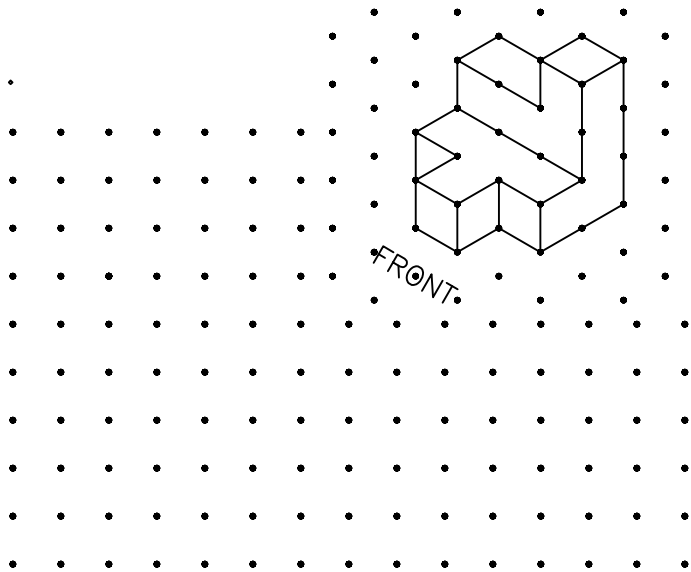
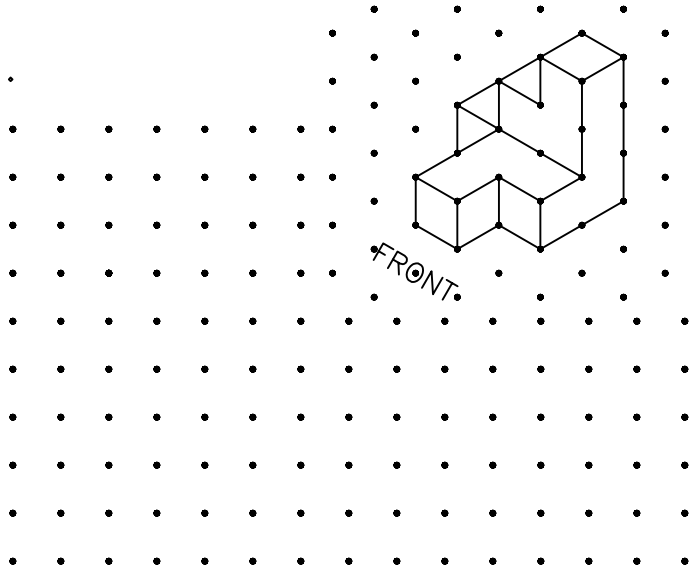


4.

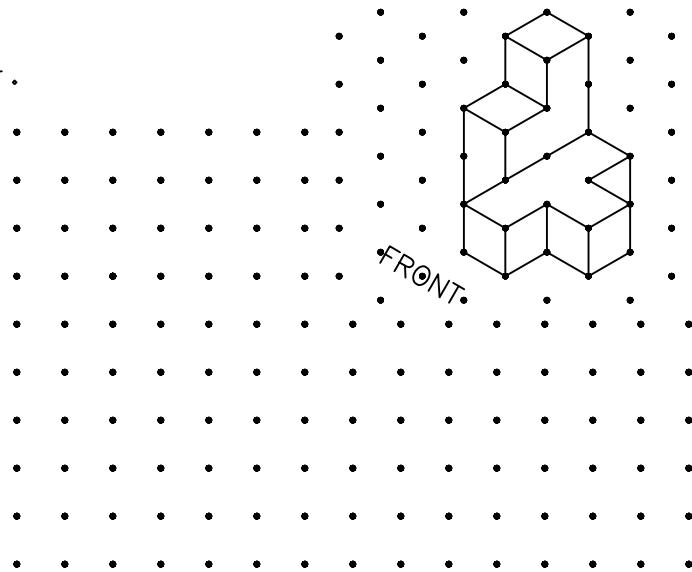
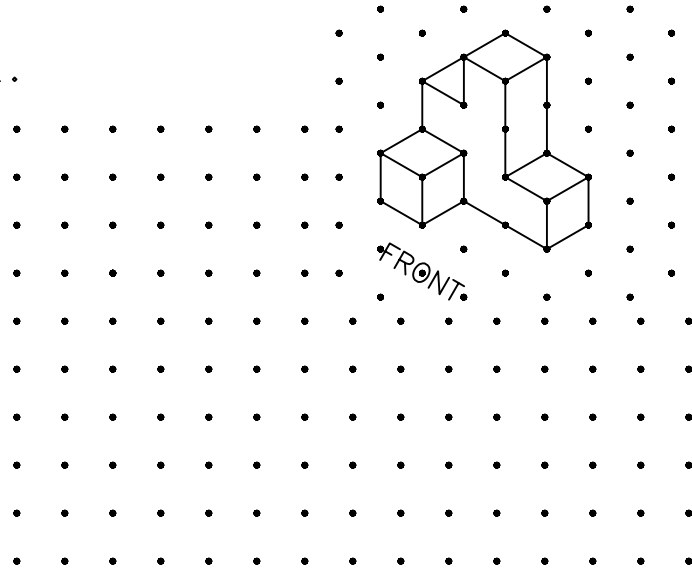


For the objects shown in isometric below, sketch the top, front, and right side views in the space provided. Make sure that your views are properly aligned.

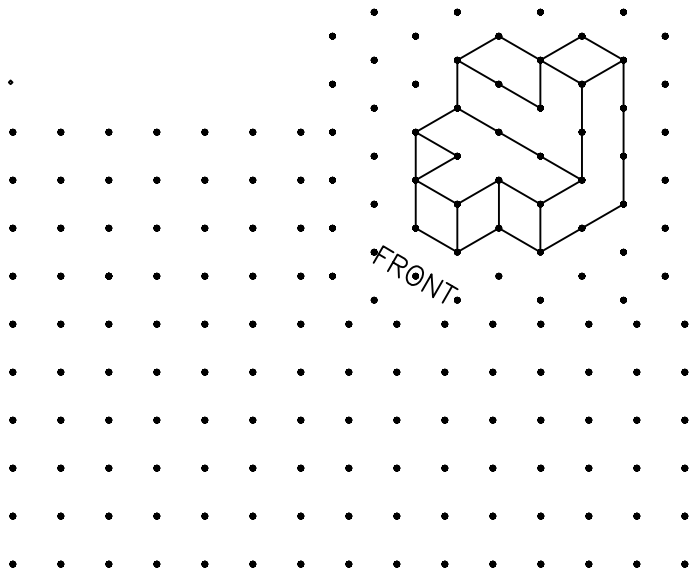
1.



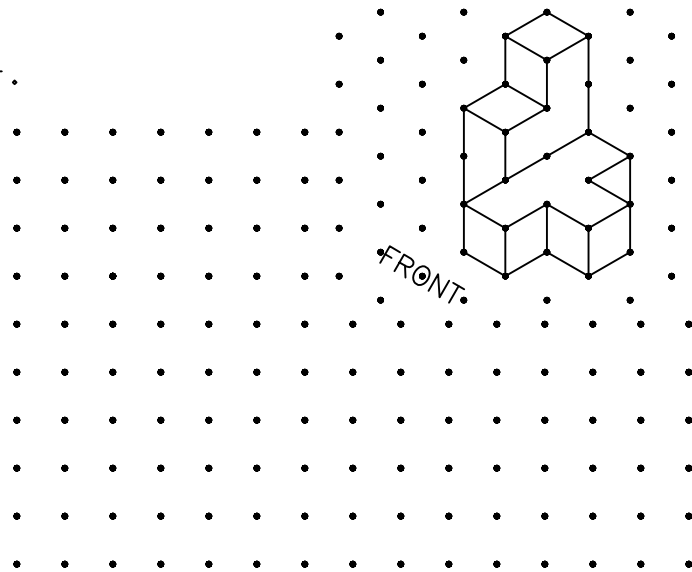
2.



3.

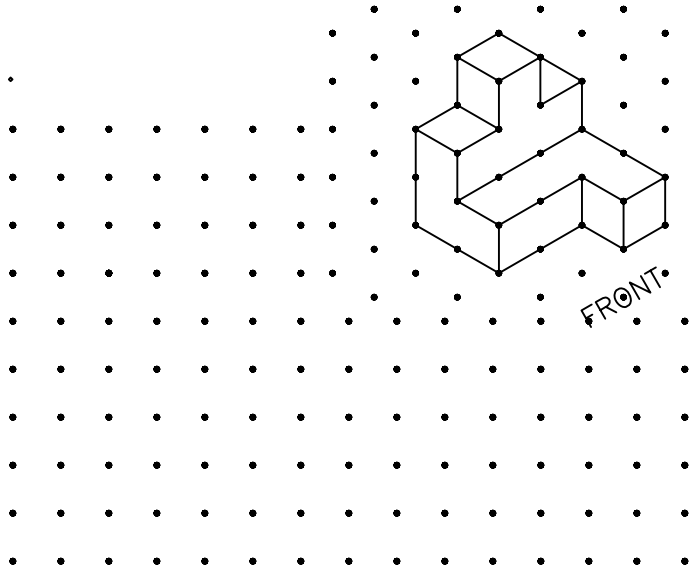


4.

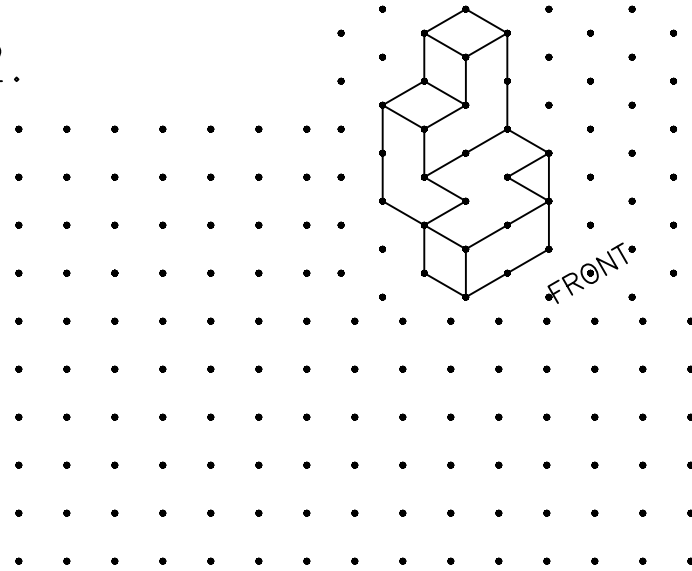


For the objects shown in isometric below, sketch the top, front, and right side views in the space provided. Make sure that your views are properly aligned.

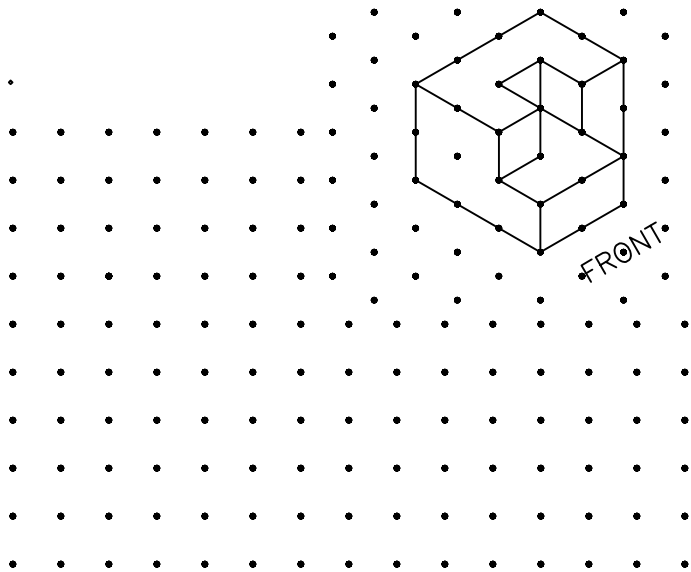
1.



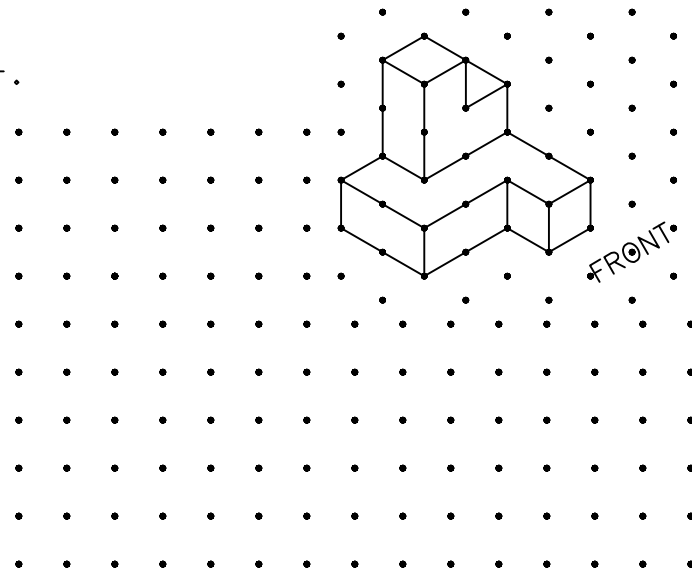
2.



3.

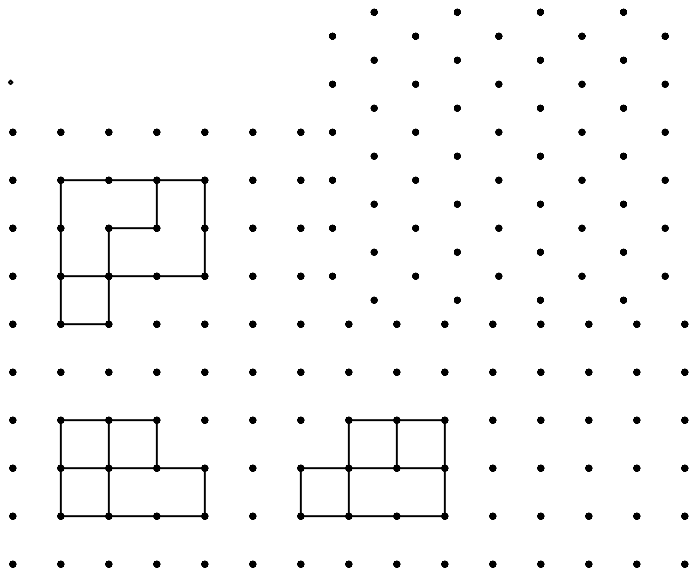


4.

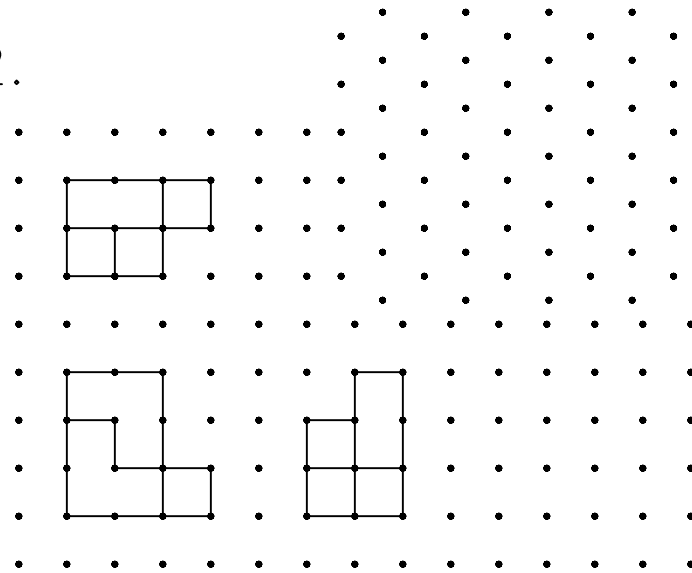


For the objects shown in orthographic projection below, construct an isometric view in the space provided. Use the box method to assist you if necessary.

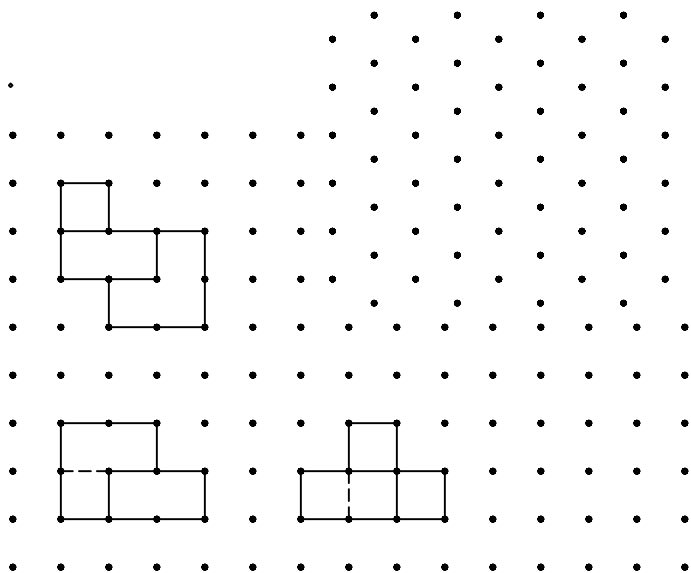
1.



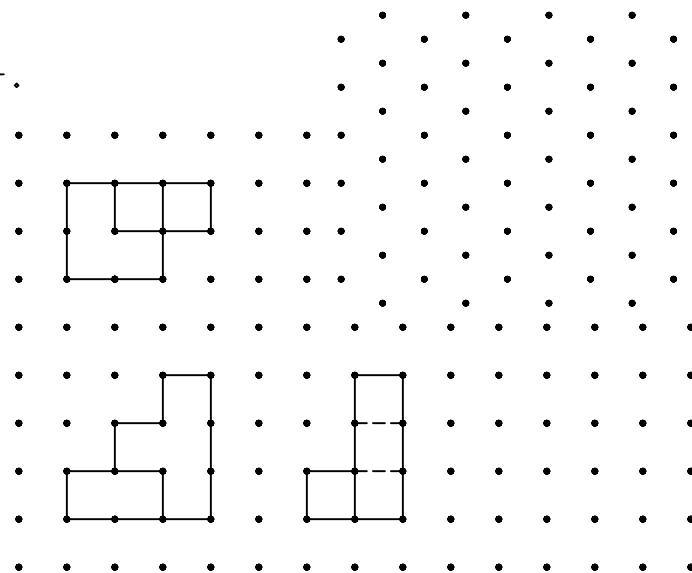
2.



3.

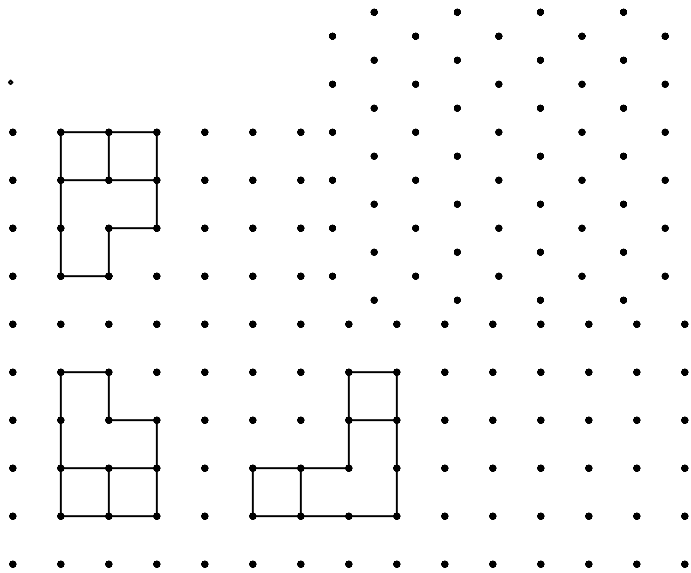


4.

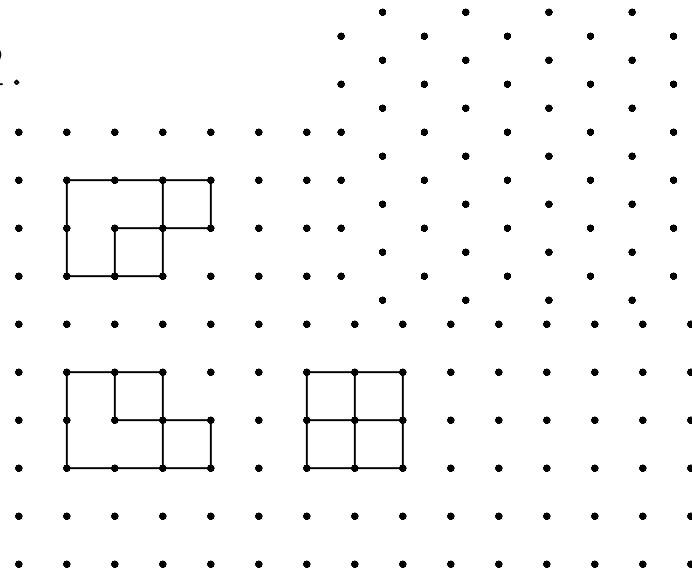


For the objects shown in orthographic projection below, construct an isometric view in the space provided. Use the box method to assist you if necessary.

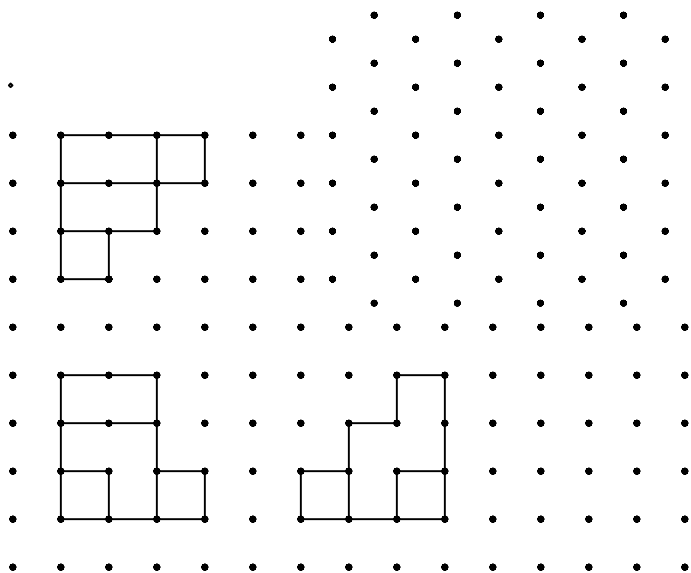
1.



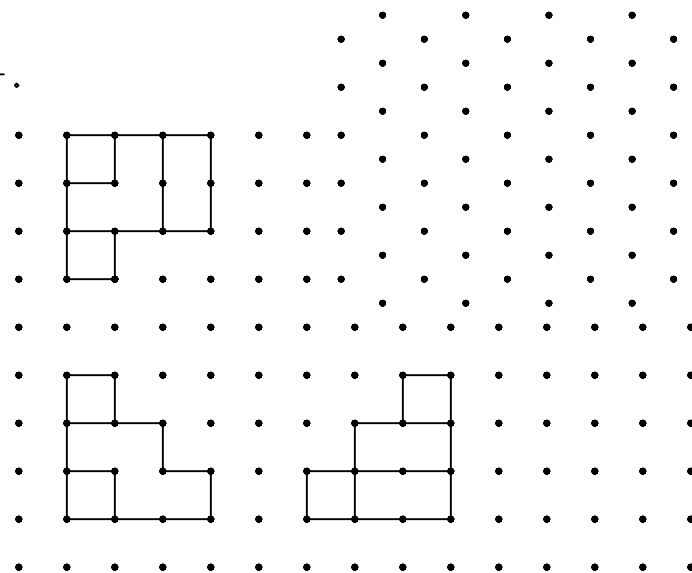
2.



3.

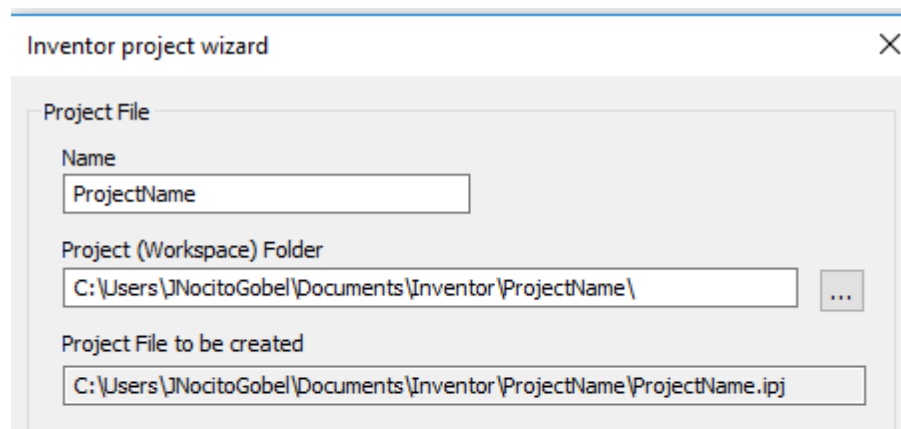


4.



UNIVERSITY OF NEW HAVEN
Tagliatela College of Engineering
EASC 1107: Introduction to Engineering
Fall 2018
Introduction to Solid Modeling with Inventor

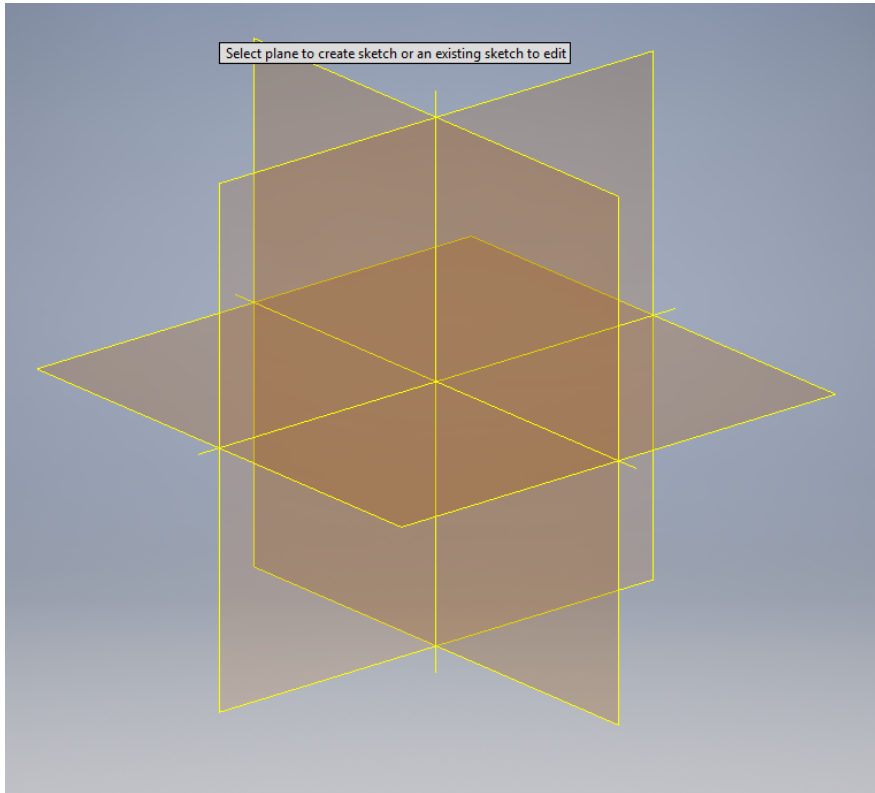
1. Open Inventor. Work off your flash drive. Do not assume that you can move or copy your files. Problems can potentially occur when you do that. If you have to copy or move files, do so within the program only – save as, etc, from within the program.
2. Create Project – Everything you create will be associated with a project. The project is used by Autodesk Inventor as a container to keep all files associated with your project in one place.
 - Click Project on menu bar– On screen click New, select single user and Next. Following screen will appear.



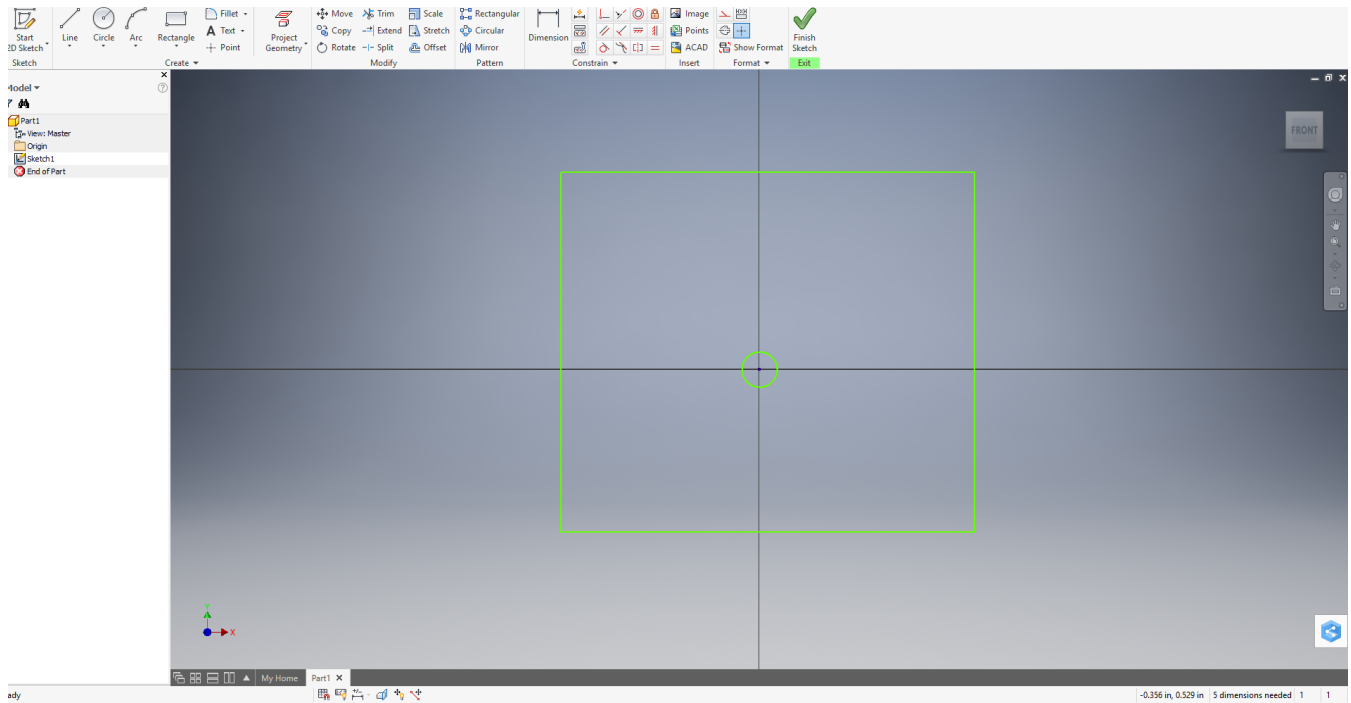
- On this screen, type in a name.
 - Under Project (Workspace) Folder, click on the icon to the right and locate your flash drive. Set your project workspace folder to a location on your flash drive.
 - Select Finish.
3. Create Part File for each unique part.
 - Click New
 - Select Standard.ipt for Inventor part
 - Be sure the project file you just created is displayed.
 - Click Create.

Let's create a 3in x 3 in x 0.5 in block with a 0.15 in pin at the center.

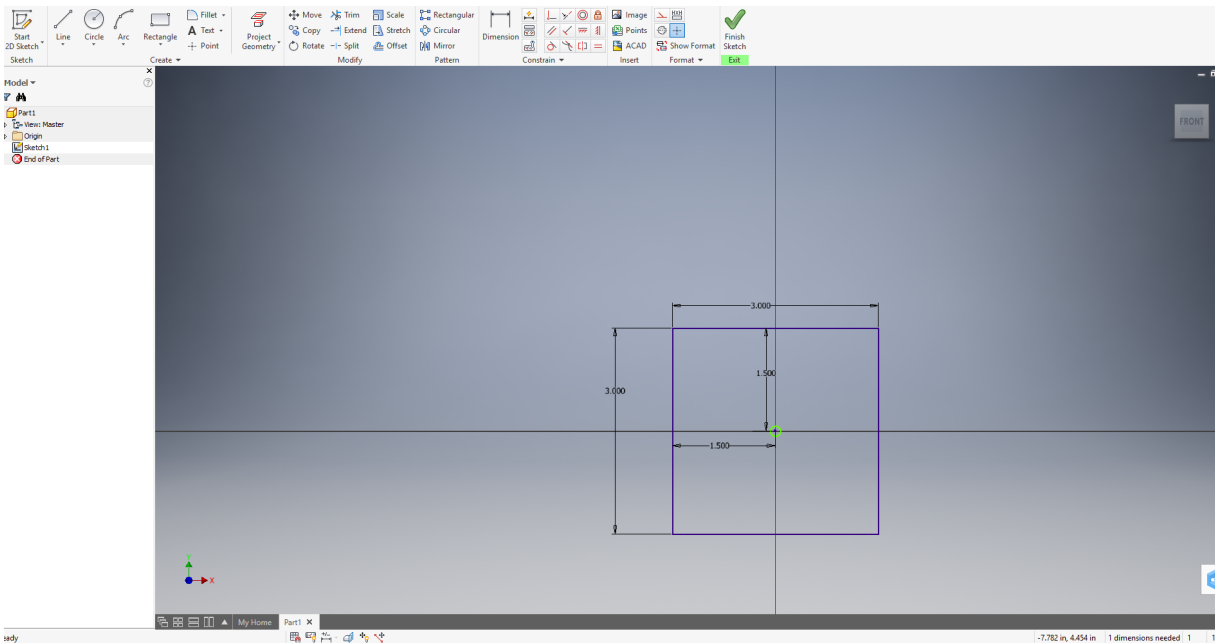
- First, create a 2-D sketch. Click on Create 2-D Sketch on top menu.
- Select the X-Y plane on screen below. The sketch pad will then appear.



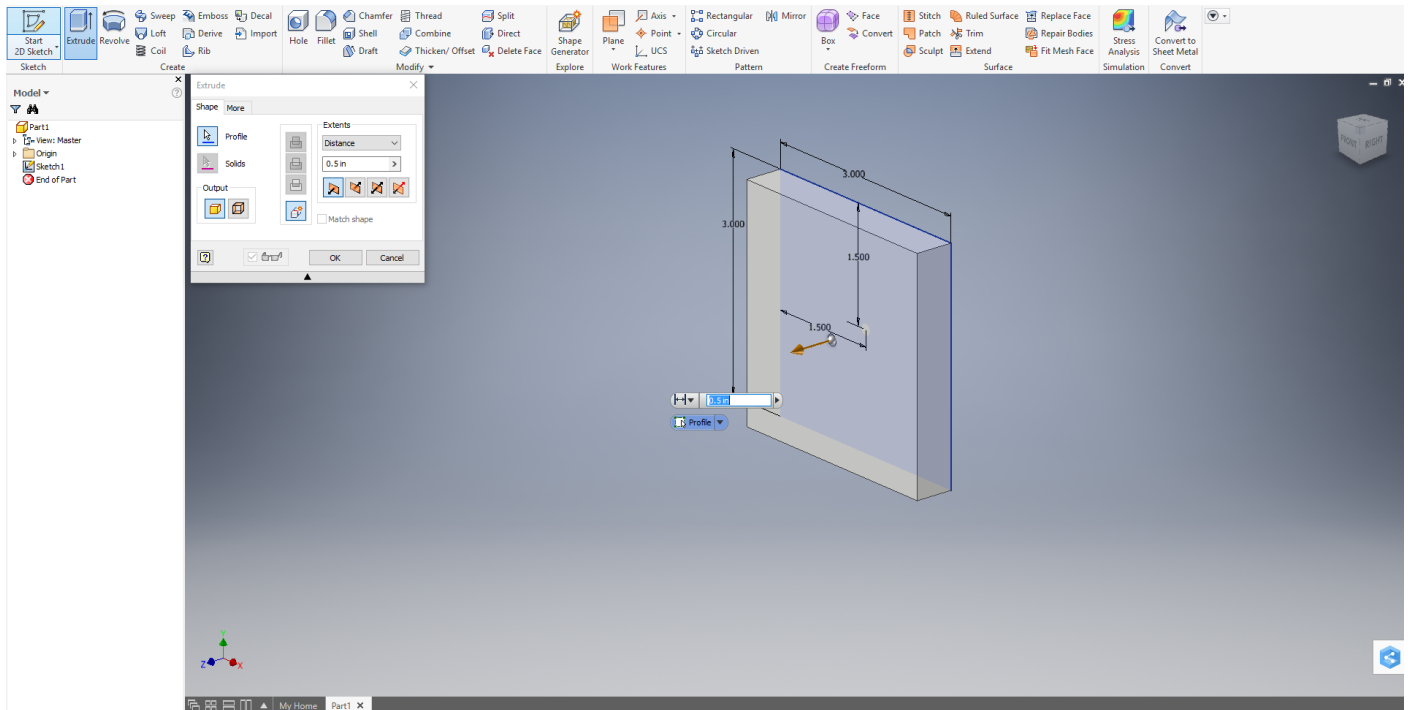
- Sketch a 2-D version of your object. Use the Line, Rectangle, circle or arc tools in the menu to sketch your object. To create a 3 in x 3 in x 0.5 in block with a 0.15 in hole at the center (location of the pin), first use rectangle tool to create the block in 2-D. Start anywhere except at the origin, left click and then move outward to create the rectangle and left click again. At this point do not worry about the dimensions of the rectangle. Right click and hit OK to create the block (see below). Hit esc to exit the tool you are using.
- Select circle, move cursor to the origin (0.0, 0.0), left click and drag out, type in 0.15 inch in the box, left click, right click and select OK.



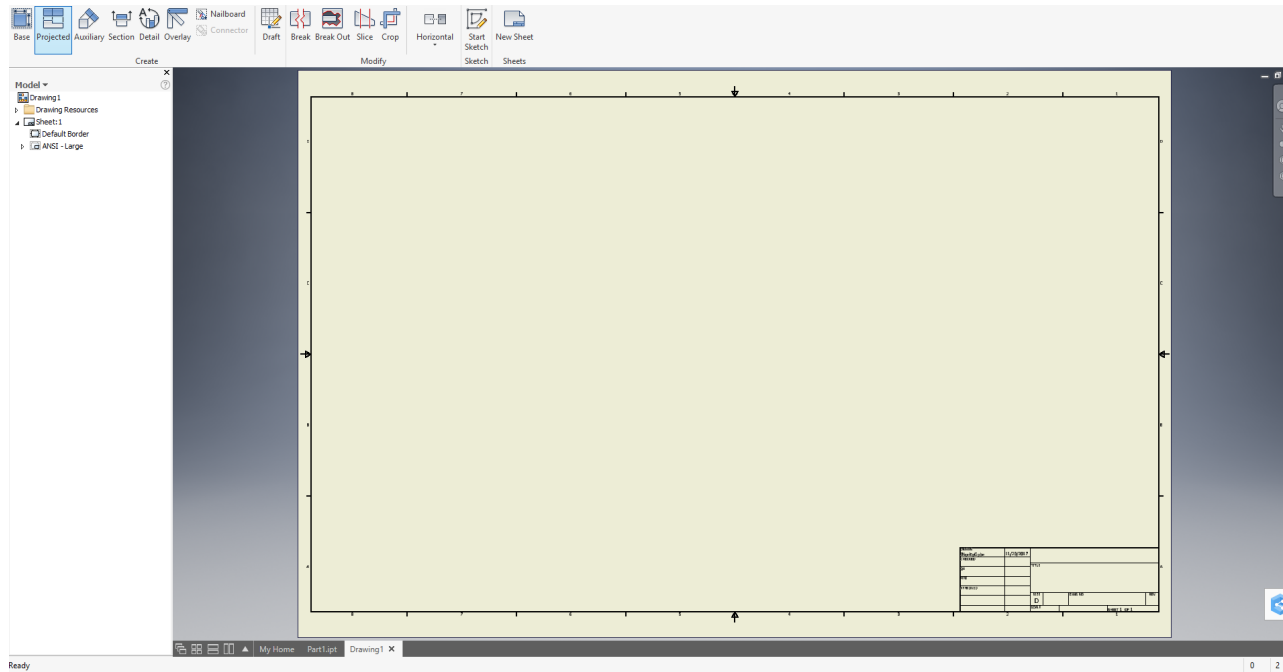
- After drawing the sketch, click on Dimension in the menu to dimension your object. Sketch must be fully constrained (all dimensions assigned – all lines are blue) before you can move to 3-D. For this 2-D block, select Dimension in the menu bar, move the cursor to a line you want to dimension, click and drag outward from the line, double click, enter the dimension of 3.0 in and click the green check mark. Do this for each dimension of the object until it is fully constrained. To get the rectangle centered about the origin (0.0, 0.0), click on the origin and then a side of the block, move the cursor until you see the dimension, enter 1.5 in, and hit OK.



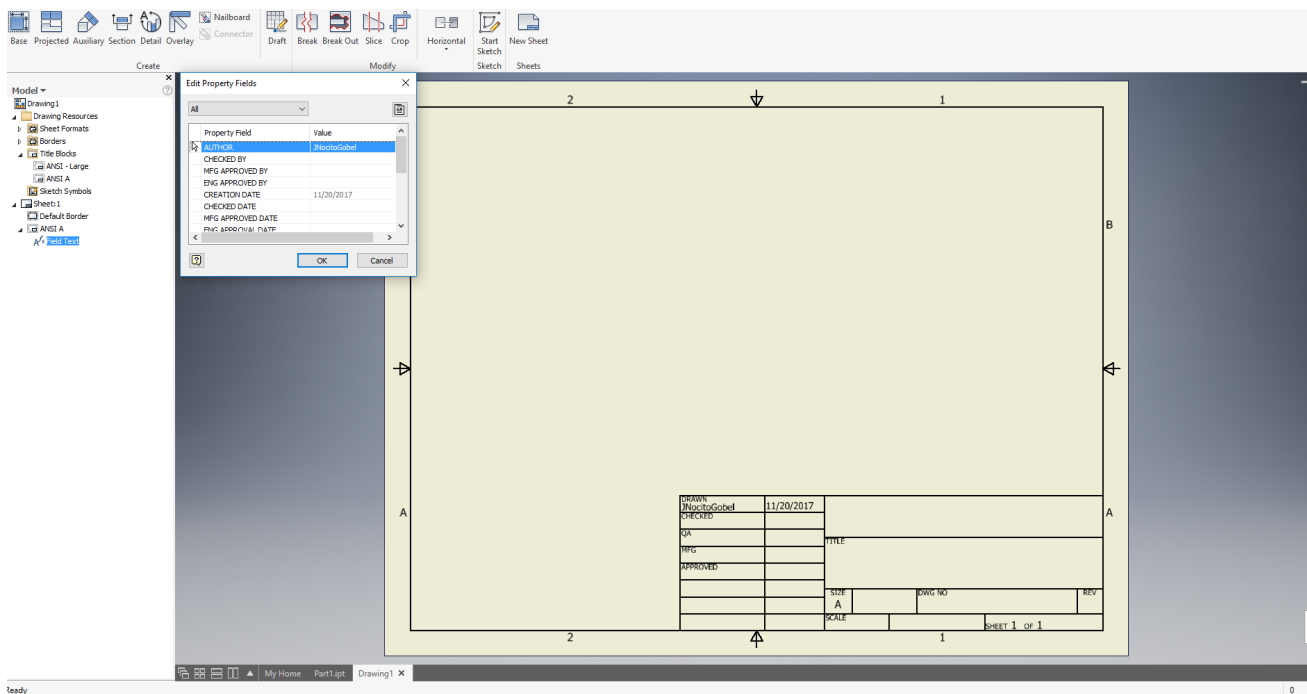
- Click on Finish Sketch when the drawing is complete. When you select Finish Sketch, your object/part will be in isometric view.
- Extrude object into 3-D: To do this, select 3-D Model on the top menu bar. Select Extrude, click on the face you want to extrude. A small box will appear. Enter how far to extrude the object and direction, and click OK. For instance, we will extrude the block 0.5 in (see below. Save the file.



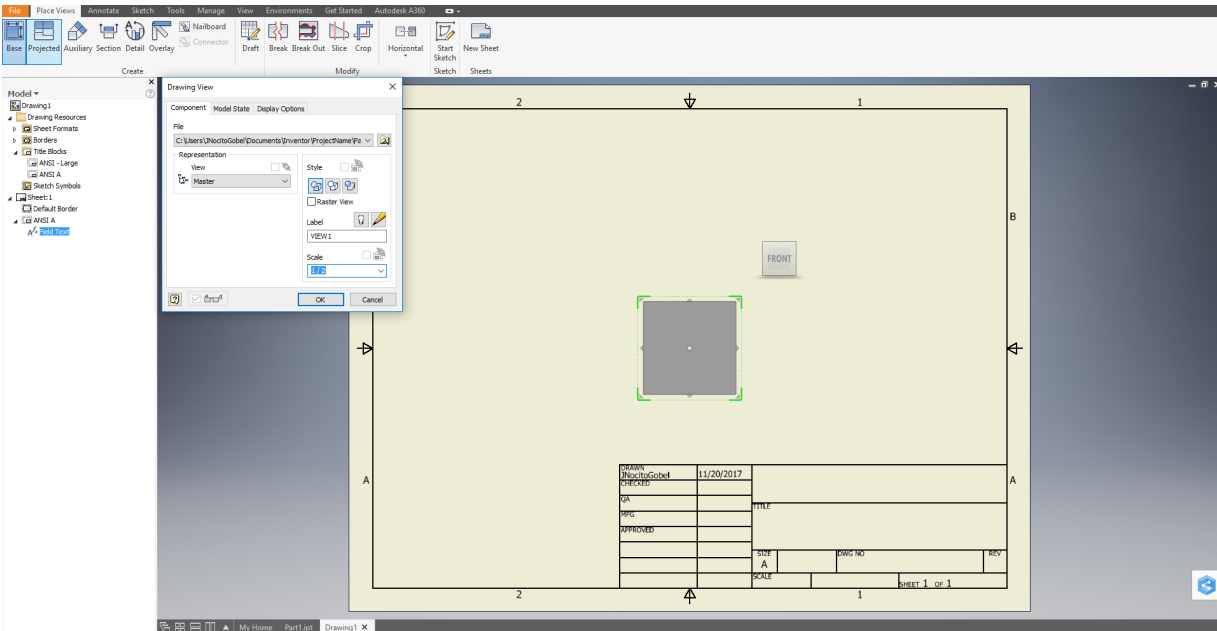
- Repeat all the above steps to create the pin (cylinder with 0.15in diameter and 1 in length)
4. Create stl file for Makerbot for each unique part.
 - Open a Part file created – choose Print – 3-D Printer Preview. A 3-D file will be created. At top of the screen, click on the 2nd icon for Save As. File extension should be stl file. Click on Save.
 5. Create Drawing file to create an orthographic view of each part.
 - Click New
 - Select Standard.idw for Inventor part.
 - Click Create. The following screen will appear.



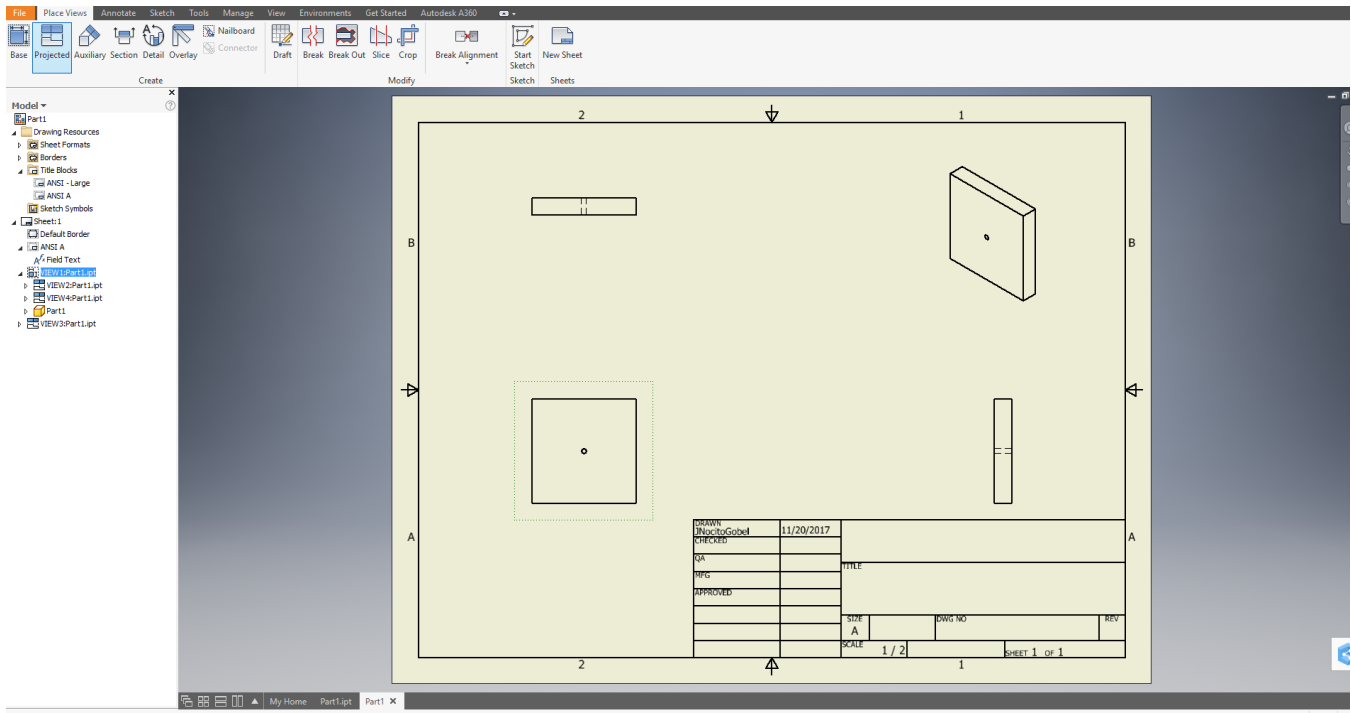
- First, change sheet size from D to A. Go to Sheet:1 and right click the mouse. On the menu, click on Edit Sheet. When the Sheet menu appears, click on A and hit OK.
- Next, change the Title block size. Go to ANSI-Large and right click the mouse. On the menu, click delete.
- Expand the Drawing Resources options (click on arrowhead). Go down to Title Block, expand options (click on arrowhead). Right click on ANSI A and Insert.
- You can edit the Title Block by right clicking on Field Text.



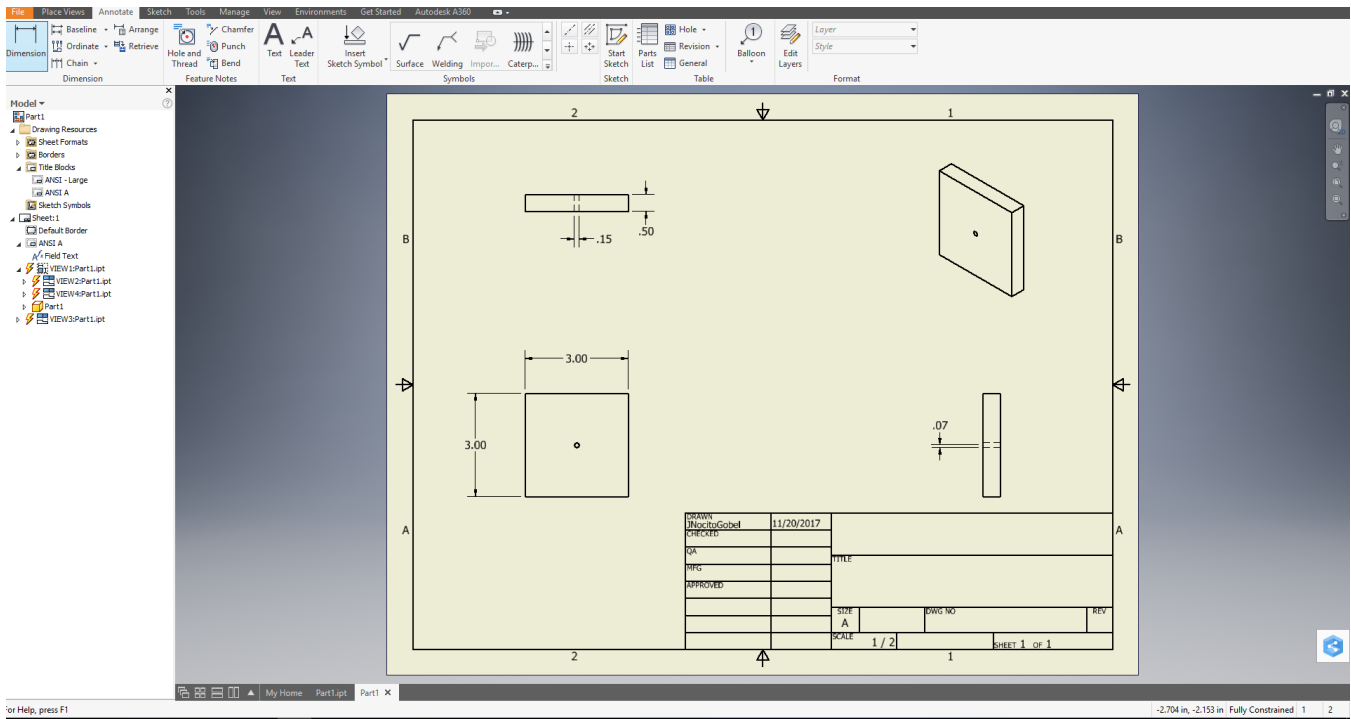
- On the menu that appears, click on Iproperties icon on the right side of the menu. Click on the respective choices to change information in the Title Block.
- Next insert front view of your part/object using Place Views – Base. Click on Base, and select the part for the orthographic drawing you want to create. If needed you can change the scale of the part; e.g. 1:2, then place the front view in the drawing before clicking OK.



- Bring in other views using Projections. Click on Projected, then click on your base view, drag it upward for top view, double click, move to the right for isometric view, double click, and then down for the side view, double click, right click and select create.
- To move any of the views, move your mouse and click on object. A dashed square will surround the object. Place your cursor on this dashed border, and you will be able to move the part/object.

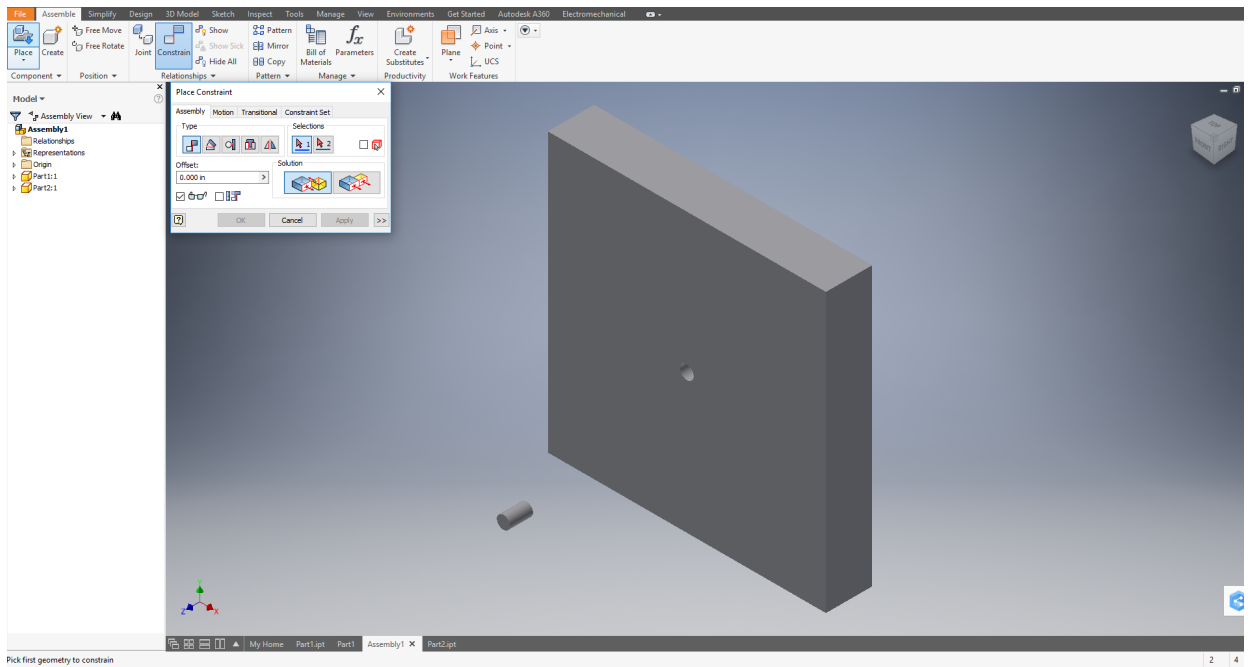


- To complete your orthographic drawing, Click on Annotate. Properly dimension your views using Dimension tool. Click on a side to dimension, drag the cursor outward, double click and select OK. Do this for each dimension needed on your base and projected views. For holes, click on each side of the hole and then drag outward to create the dimension. **Do not dimension isometric view.** Save file.

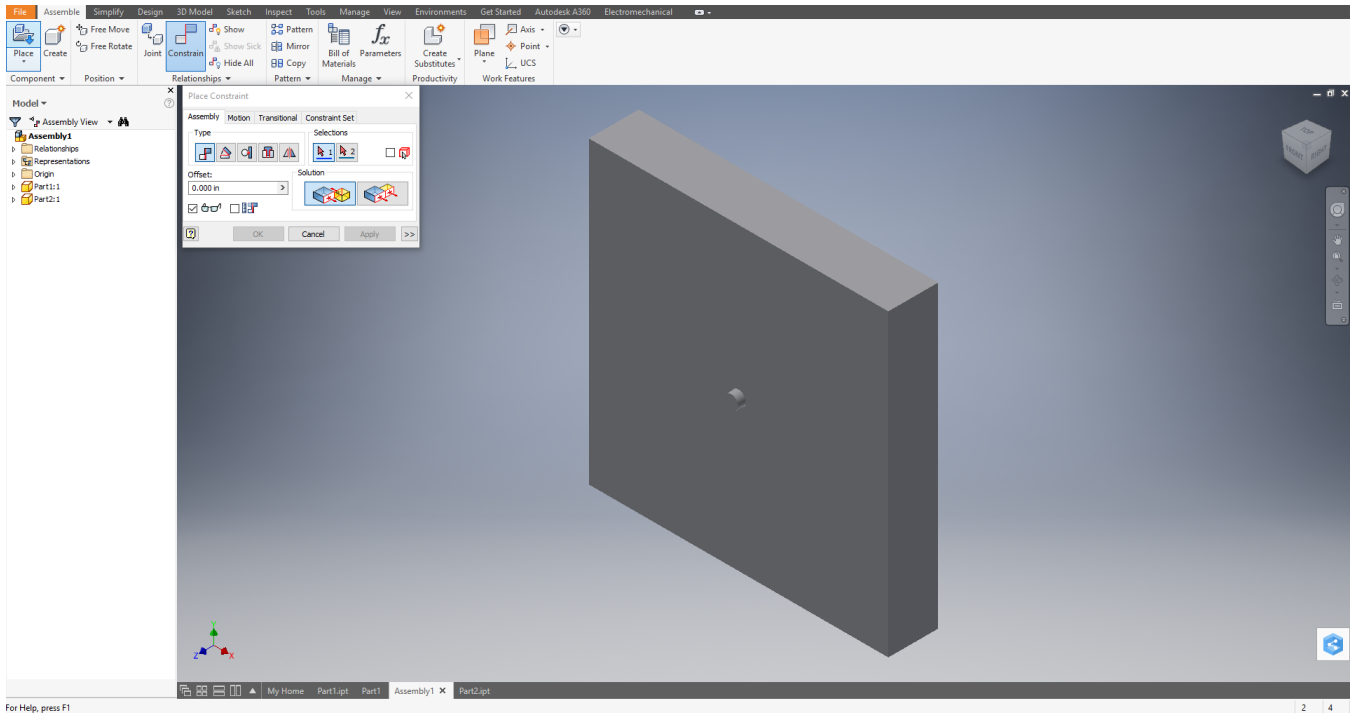


- Repeat the above steps for the pin part.

6. Create Assembly Drawing: After drawing each of your unique parts, create an Assembly drawing of your object.
- Click New
 - Select Standard. iam file and click create.
 - Next, bring in your parts using Place tool. When you click on Place, a menu will appear. Click on the file you want to bring in first, click on Open (at bottom of menu), and place the part in the drawing space. Right click and enter OK.
 - After bringing in all your parts, you want to assembly the parts. To do this, use the constrain tool. There are two commands, either mate or flush parts (see below).



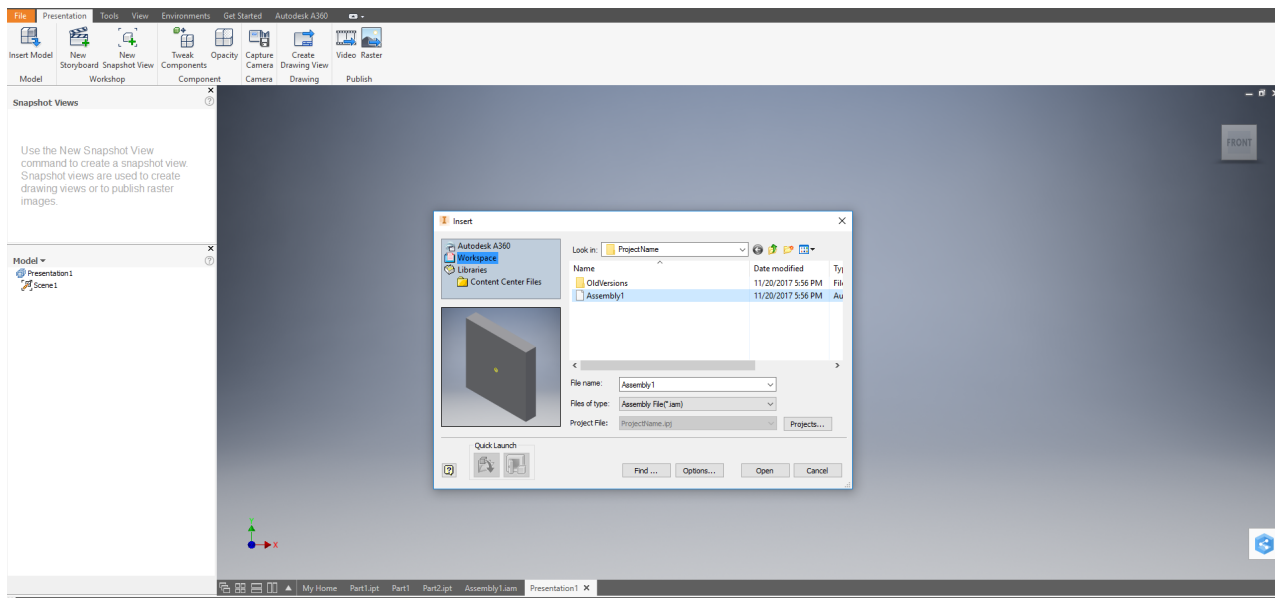
- Mating is used when you want to align parts along the same axis (see picture). Flush is used to constrain surfaces side-be-side. You must constrain your completely assembled object in at least 3 ways, using these two commands. This means that the parts will not move when the object is fully assembled.
- For instance, the peg (cylindrical object) will be inserted into the hole in the block by first mating these two parts. To align the axis of these two parts, click on mate, then click on the cylinder (should see axis) and click on hole in block (see axis), and then apply. See picture on next page.



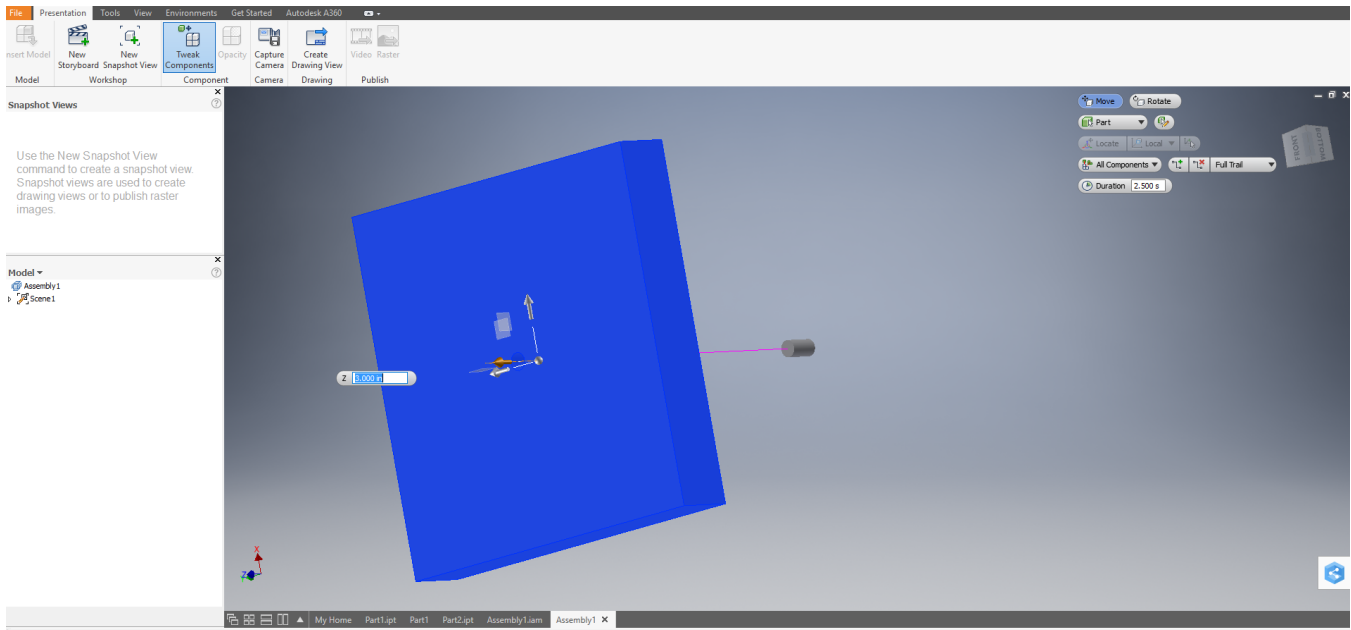
- This constrains these two parts in only one direction. Need to do two more times to fully constrain these parts. If you have more than two parts, then the total number of ways you constrain all of your parts is still at least 3 ways.
- Save the file, once your object is completely assembled.

7. Create Presentation Drawing: Presentation file is an exploded view of the Assembly file.

- Click New
- Select Standard. ipn file and click create. A window will open where you select your Assembly file and click Open. Your Assembled object will be brought into the space.



- To create an exploded view, use the Tweak Components tool. Select Tweak Components, click on one of the parts. A 3-D axis appears. Click on the arrowhead in the direction you want to pull the object. Repeat this step for each part. Save the file once your object is completely pulled apart (exploded view).



- Create a drawing file that contains both the Assembly and Presentation (Exploded View) files. To do this, open a New drawing file (Standard.idw). Follow step 5 above to change the sheet size from D to A and bring in proper title block (ANSI A). Then using Base tool, first bring in the exploded view (.ipt file) in isometric view. Then bring in Assembly file using Base tool again, in isometric view. To complete this drawing, select Annotate. Click on Parts List, select your Assembly file, and place the Parts List at the top of the drawing. Use the Balloon tool to identify each part by number. Double click to move from one part to another using this tool.
- Complete the Excel worksheet where the Parts files are listed, along with the quantity of each unique part, and the volume for each part. To find the volume for an individual part:
 - Open a Part file.
 - Right click on the name of the part file (left hand side of screen).
 - In the menu that appears, click on iProperties.
 - Along the top of this screen, click on Physical.
 - Change the Material to PVC, unplastericized.
 - Record the volume of the part.

10. Create the following drawing.

- All dimensions are in millimeters.

