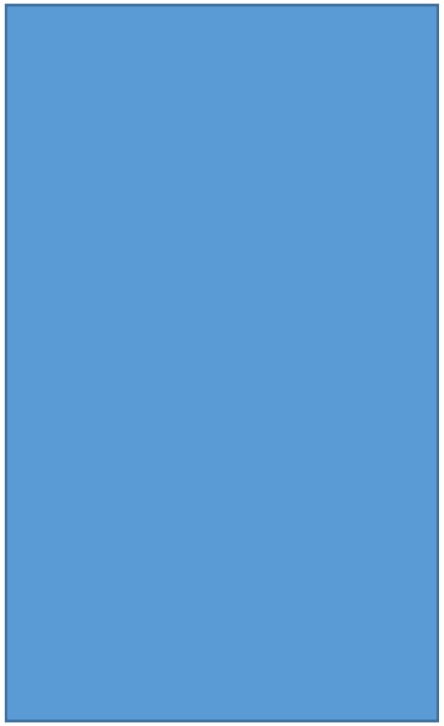
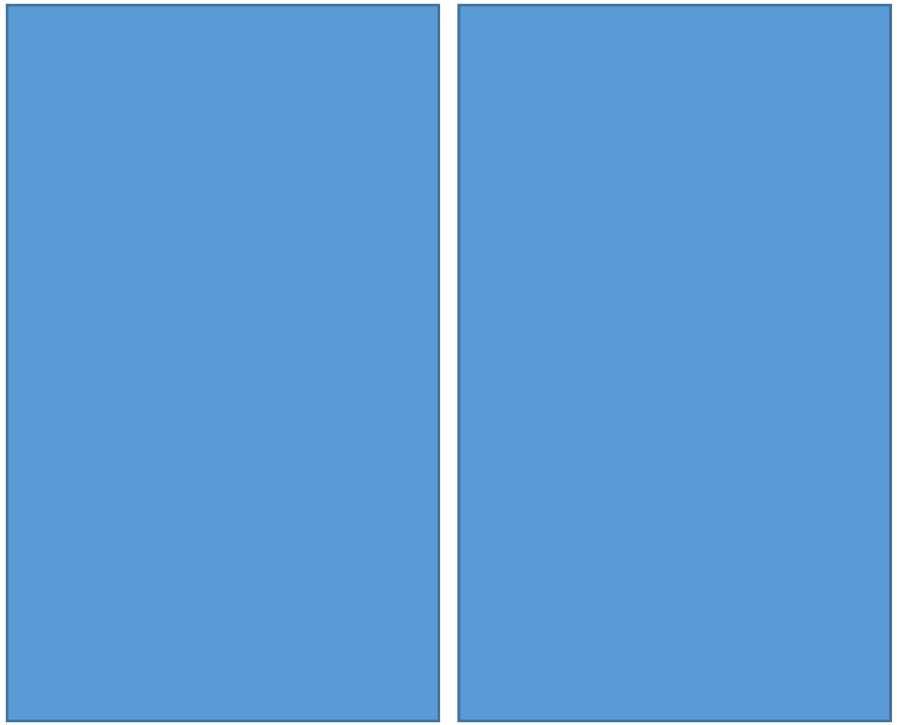
DRAFTING MADE EASY
9/29/2015

Let’s start simple. The standard sizes for drafting paper are defined by letters, ie. A, B, C, or D. They are based on the standard 8.5X11 sheet of letter paper.



8.5X11 = “A” size

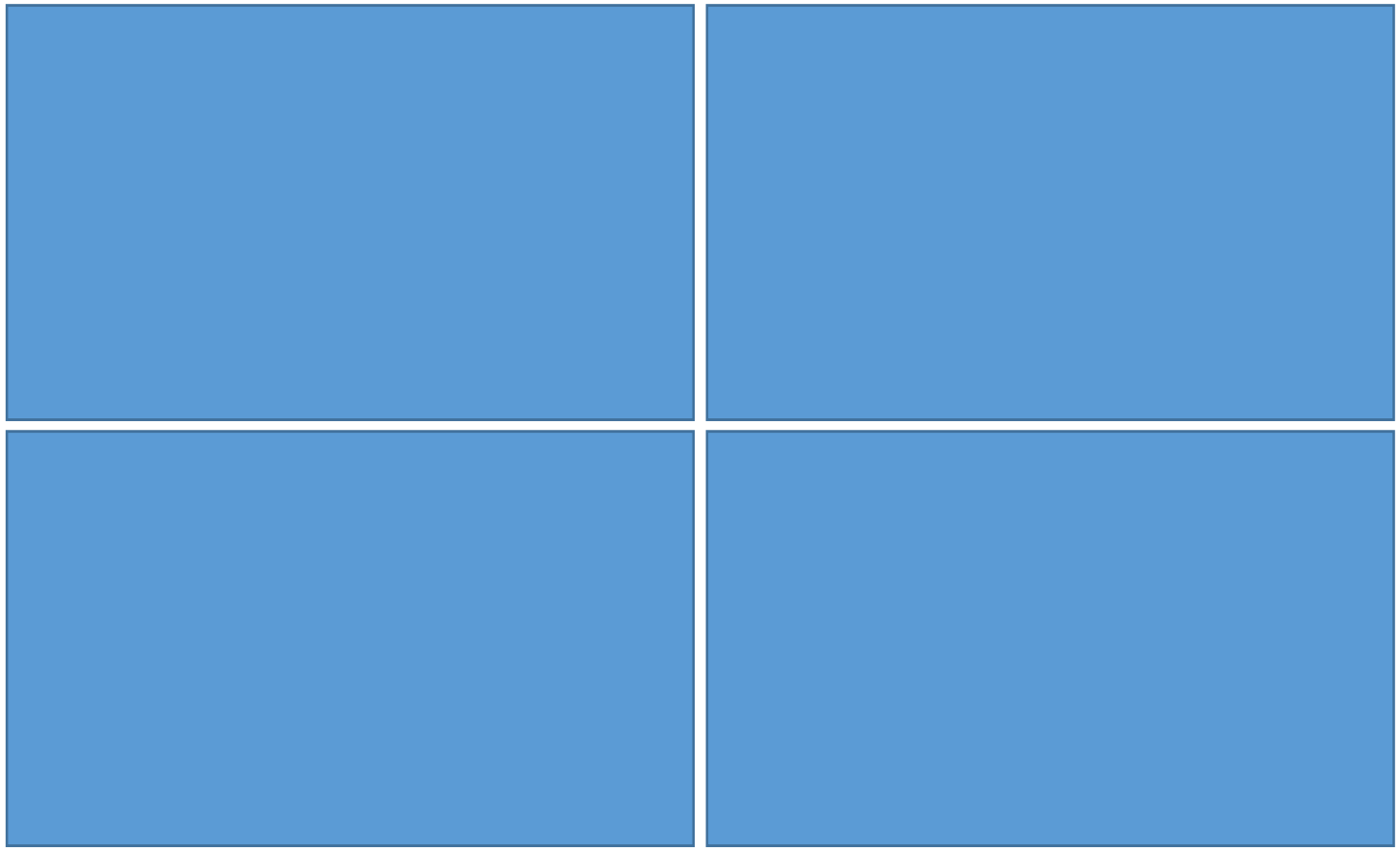
Now double the shortest dimension and we get the next size up….



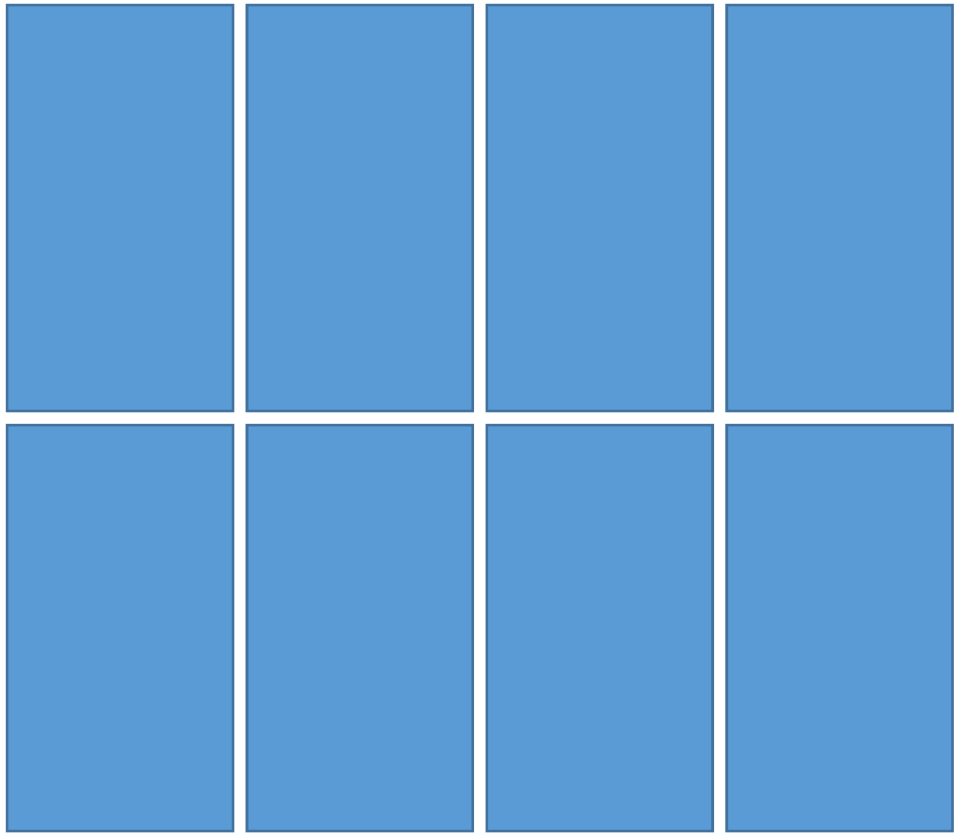
11X17 = “B” size

“B” size is by far the most popular size because it can be printed on many standard office laser printers.

And so on…..



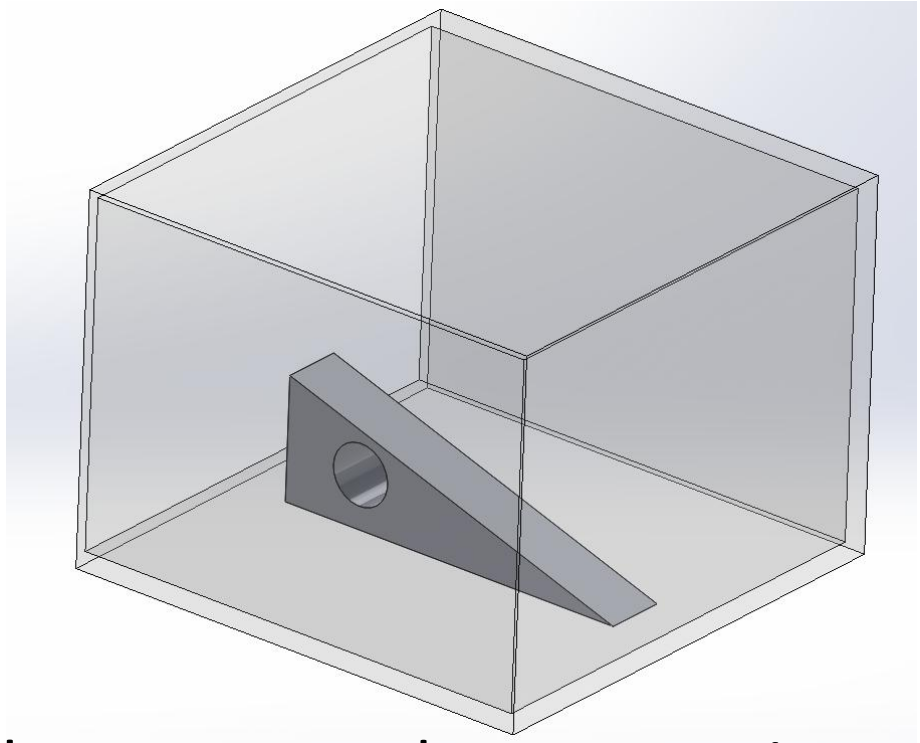
22X17 = “C” size



22X34 = “D” size

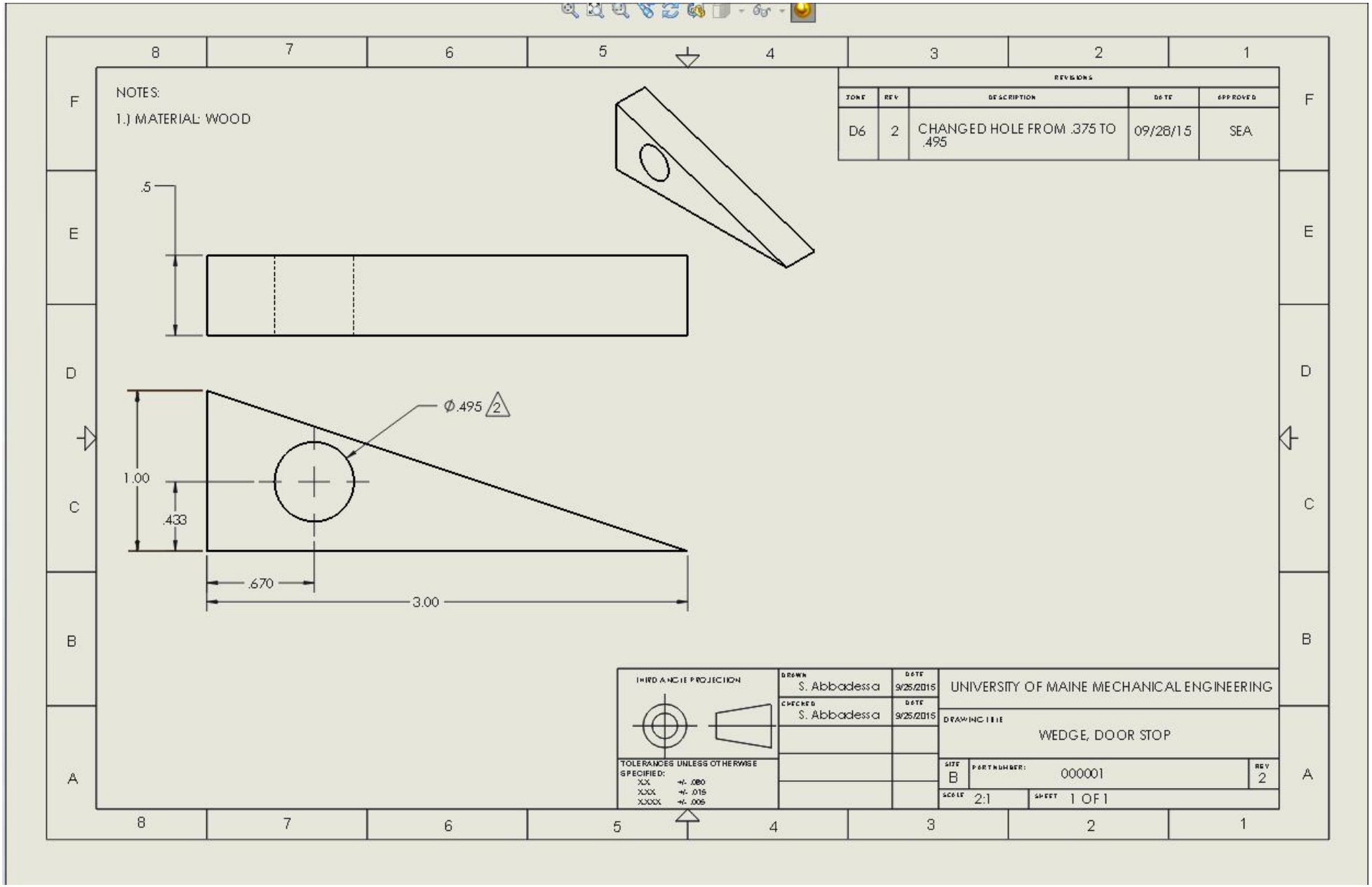
Creating Views:

The most fundamental concept in drafting is the idea of the “glass box”. Imagine your part is in a glass box. If we project the edges of the object onto the surfaces of the box and then “unfold” each side, each glass face will have one of our drafting views on it.

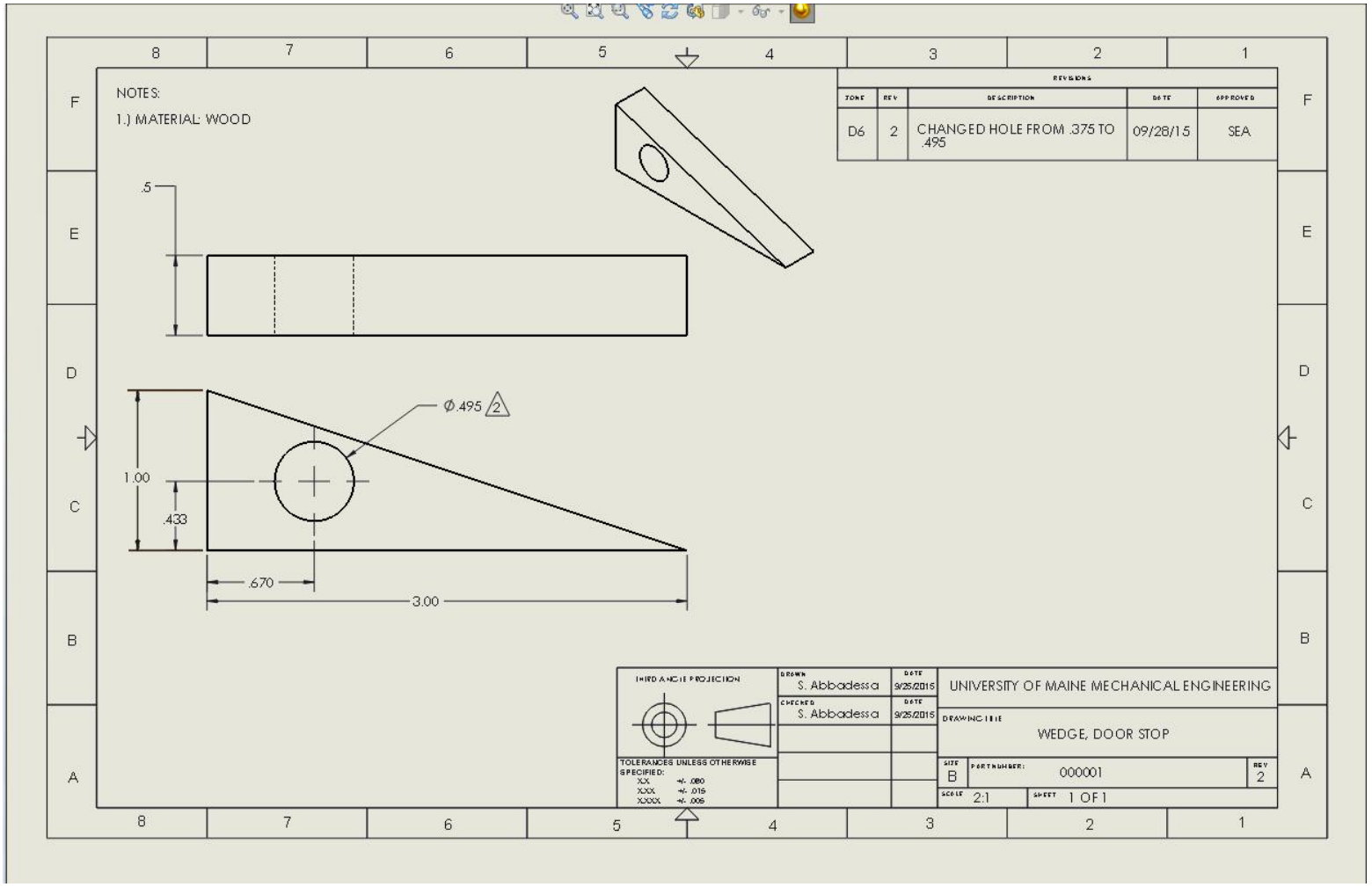


Internal or unseen edges get projected as dotted lines.

The “fold lines” of the glass box are not shown on a blueprint and are simply assumed to be there.



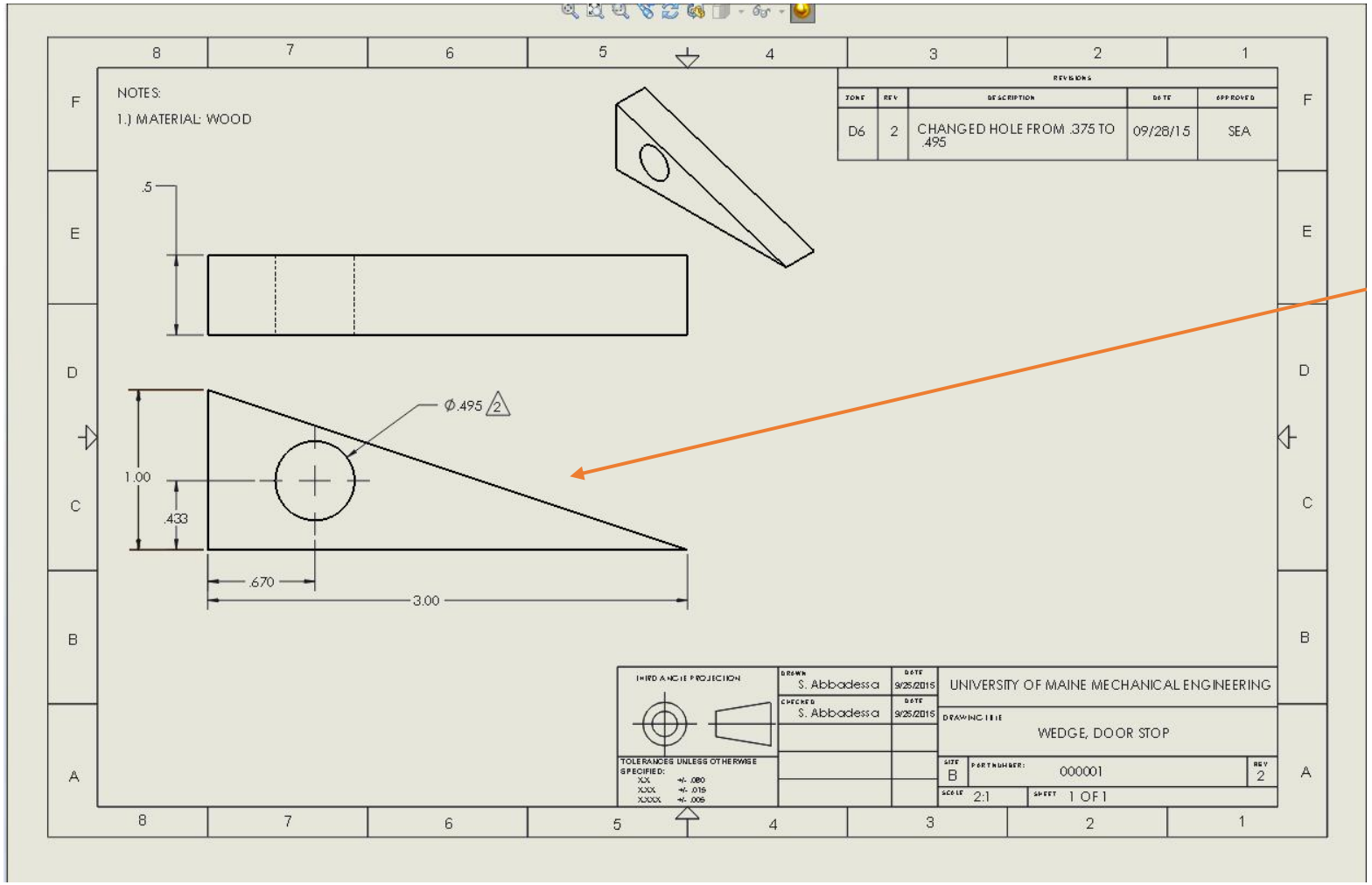
Not all of the six sides of the box need to be shown if the geometry can be fully described with fewer views.



The isometric is generally added as just a visualization aid.

Choosing the front view:

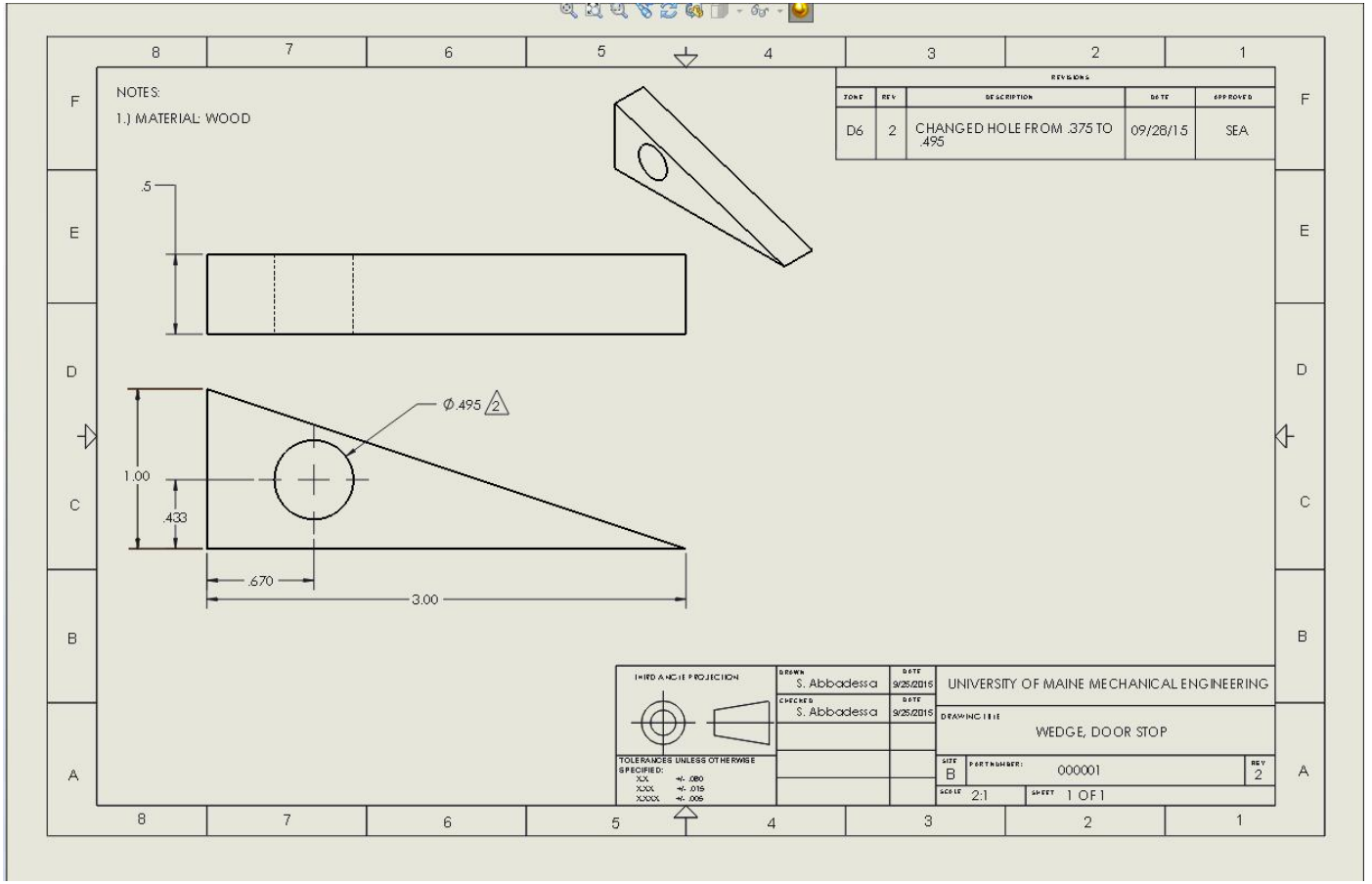
Since today we work in 3D CAD, any of the glass box faces could be turned to be considered the front view. You choose the front view by selecting the view with the fewest hidden lines.



No hidden lines means this is the best choice for the front view.

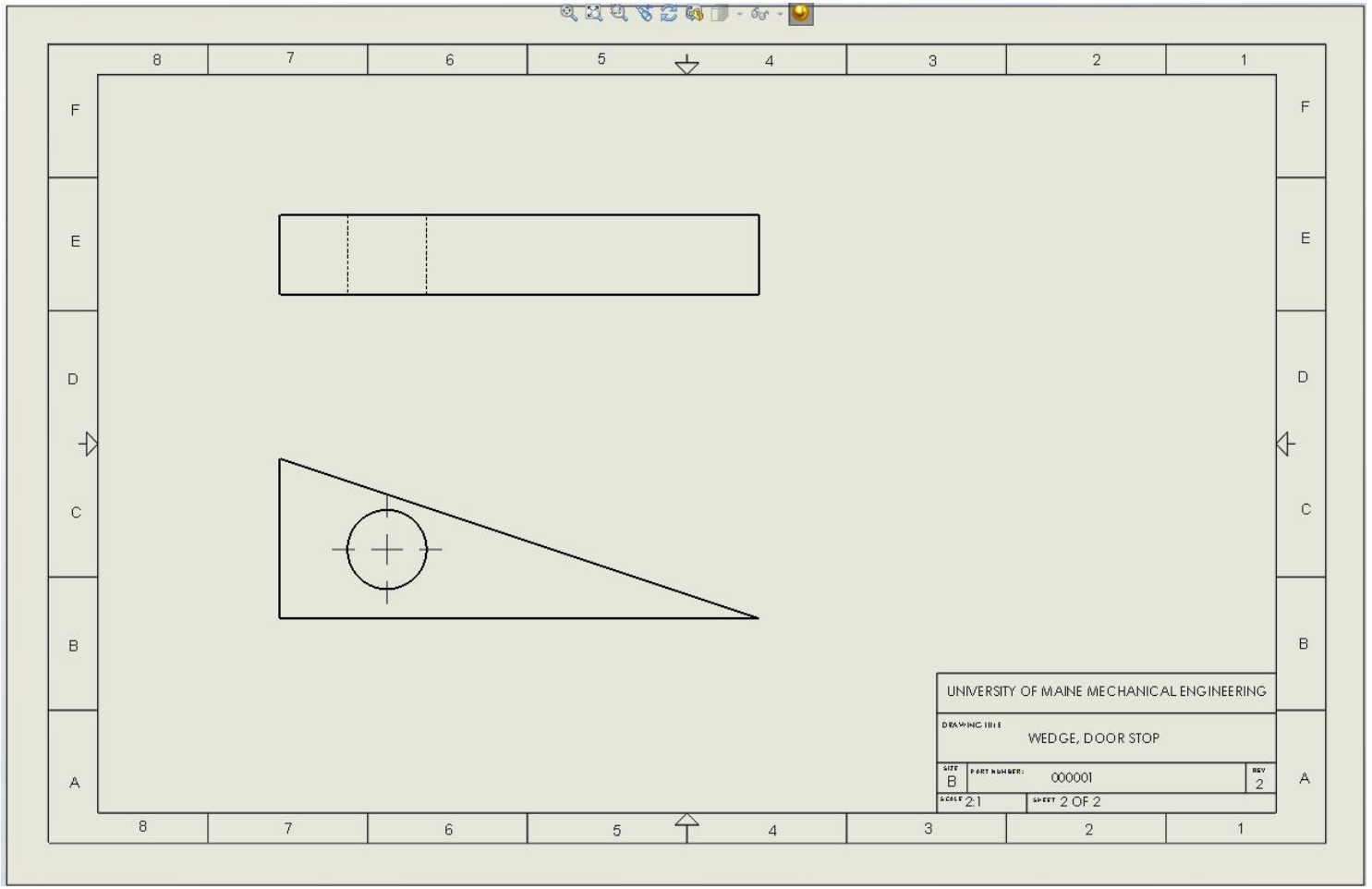
The Format:

The format is the border of the drawing. It contains the information needed interpret the drawing such as the projection angle (1st for Europe and Asia, 3rd for the US mainly), tolerancing information, approval status, revision history….. Every company makes up their own based on the needs of their product. Below is a modified version of the one I use.



The Format:

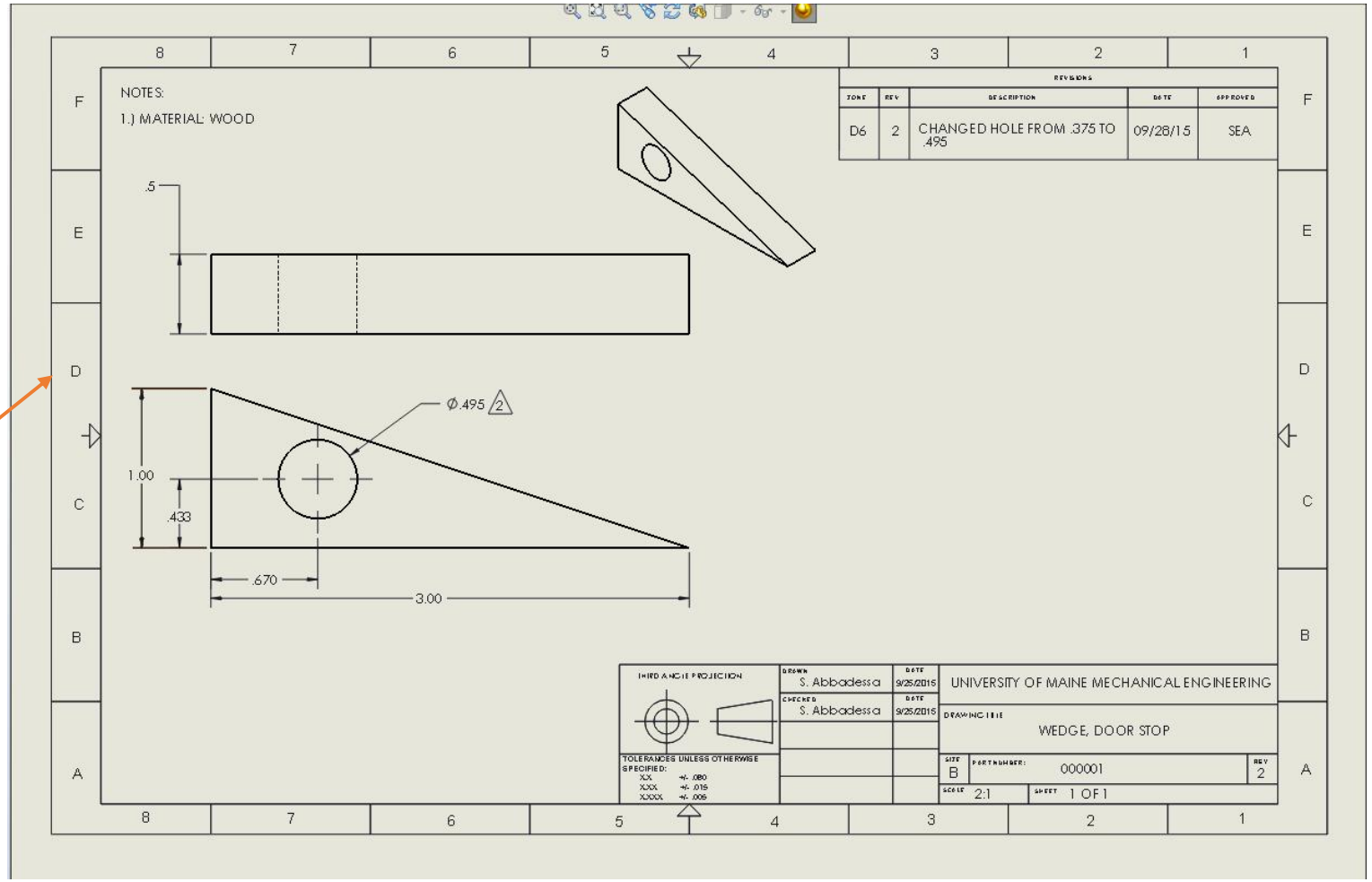
Often the second page will have a less information in the format because page one's information is expected to be in effect throughout.



The Perimeter:

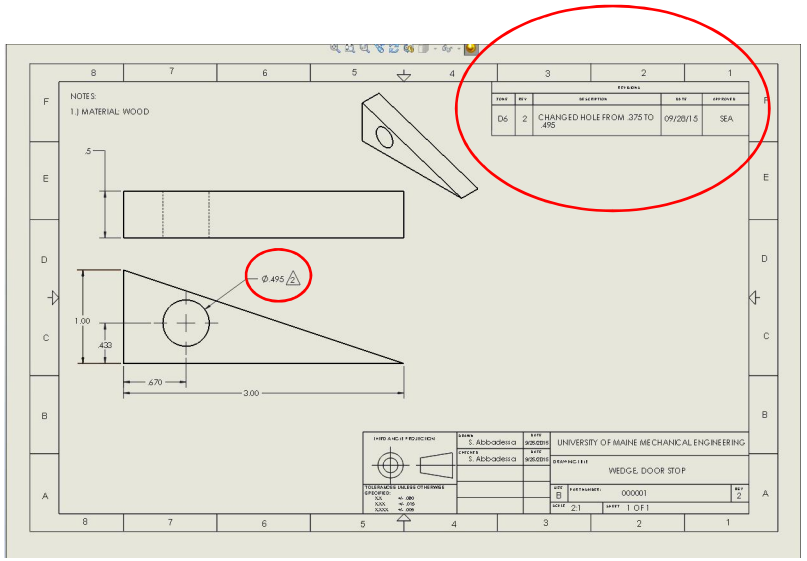
These coordinates, such as E8, allow you to locate changes called out in the revision block. It is also nice to use when discussing the part dimensions with another person over the phone.

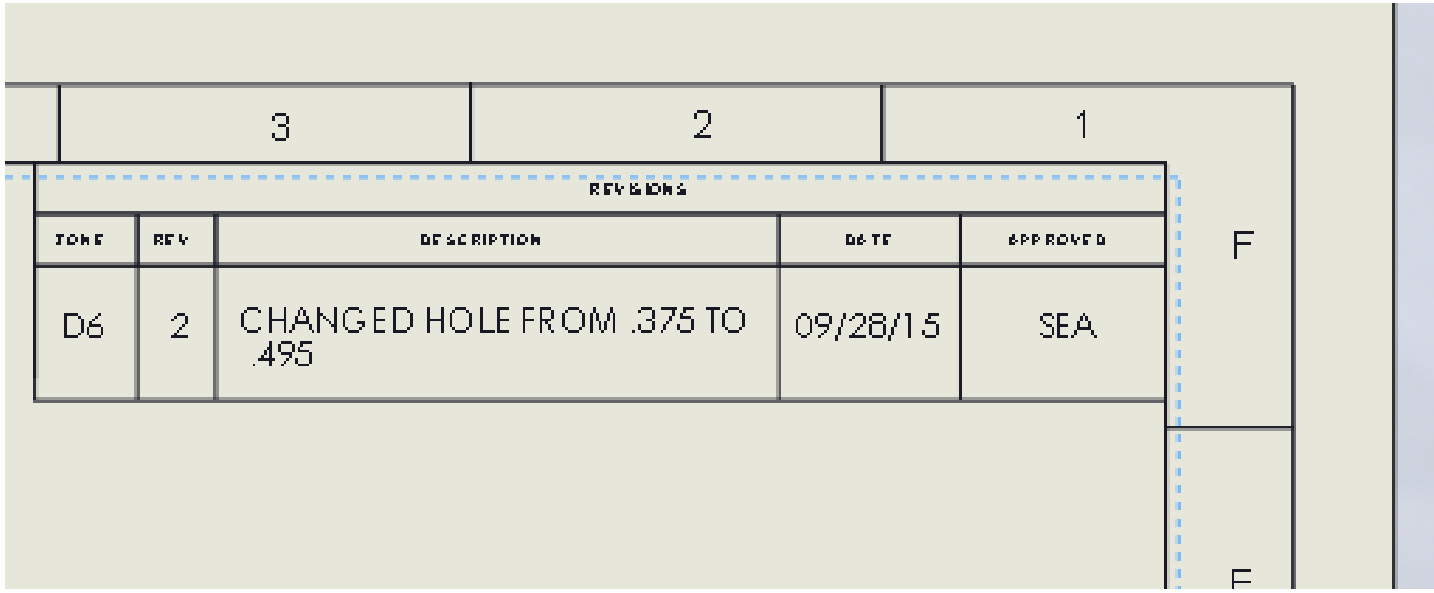
Perimeter



The Revision History Block:

This is where you record all the changes to the part. This includes changes to the bill of material if it is an assembly, any dimensional changes, paint or coating changes, material changes, new features, or controlled process changes. There should be enough information so that each revision can be undone to the previous revision level.

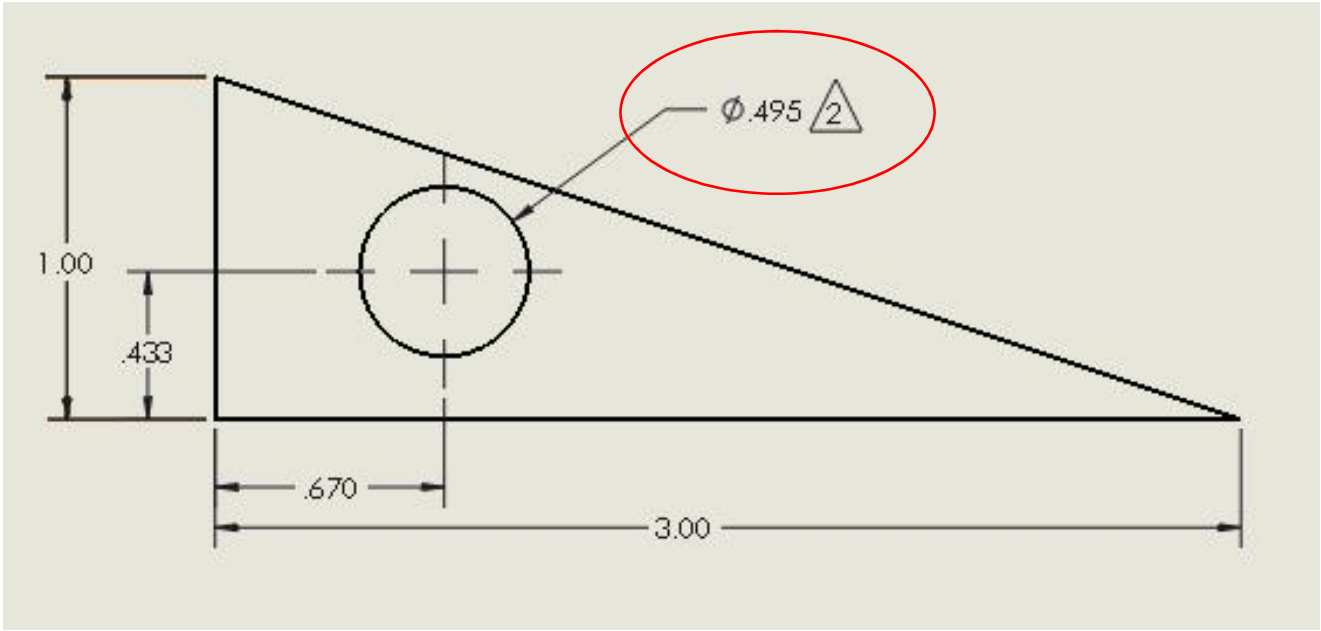




If the change to the part is severe enough that you could not install it on all assemblies back to the original usage, then change the part number, begin at the initial revision, and revise the assembly(s) it is used in. We often refer to this as violating “form, fit, or function”.

The Revision History Block:

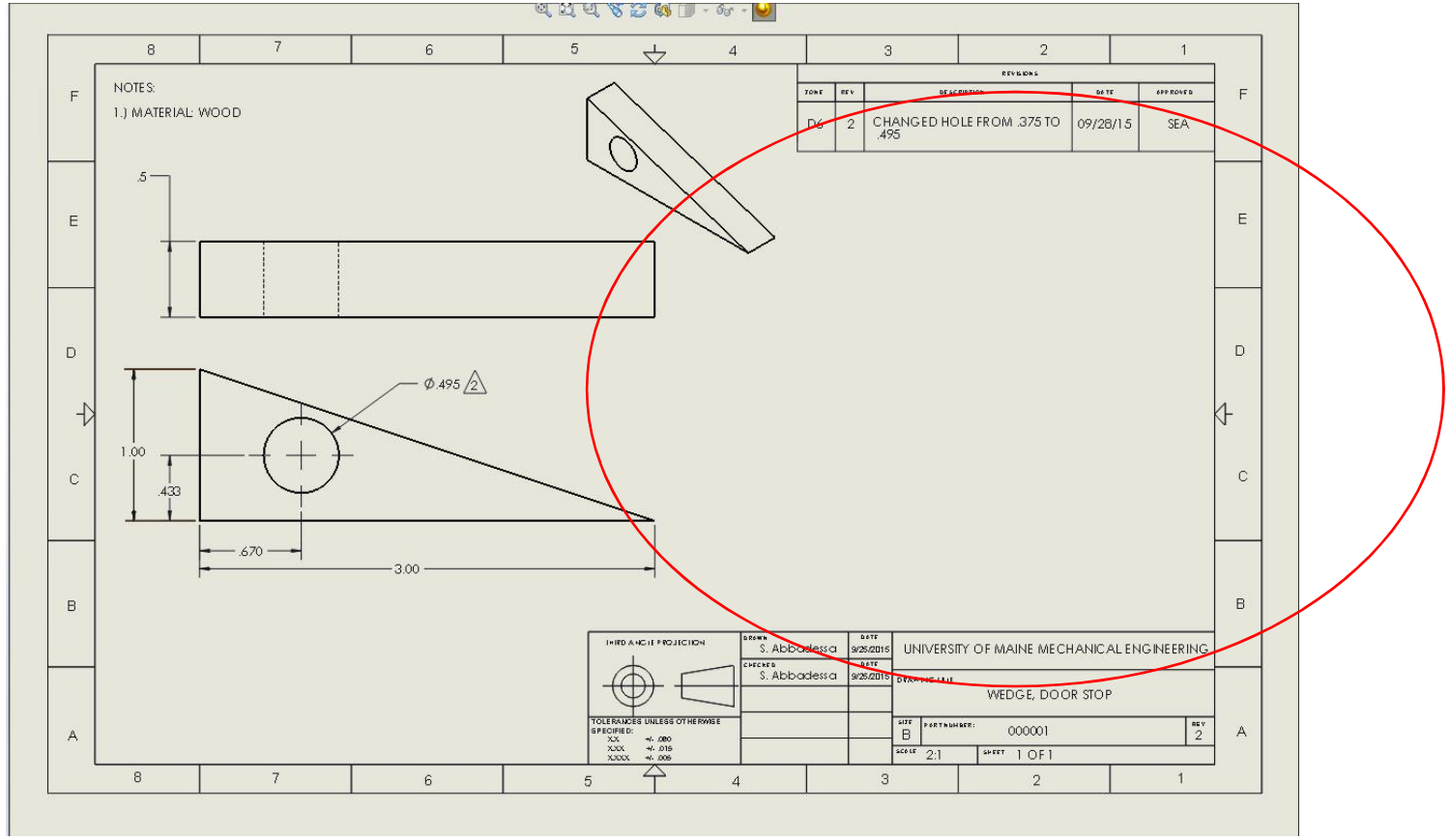
One way to mark what was changed in a revision is to use “delta” symbols.



This tells the reader that the .495 dimension was changed in revision 2. The details are placed in the revision history block. Every company does this differently.

The Revision History Block:

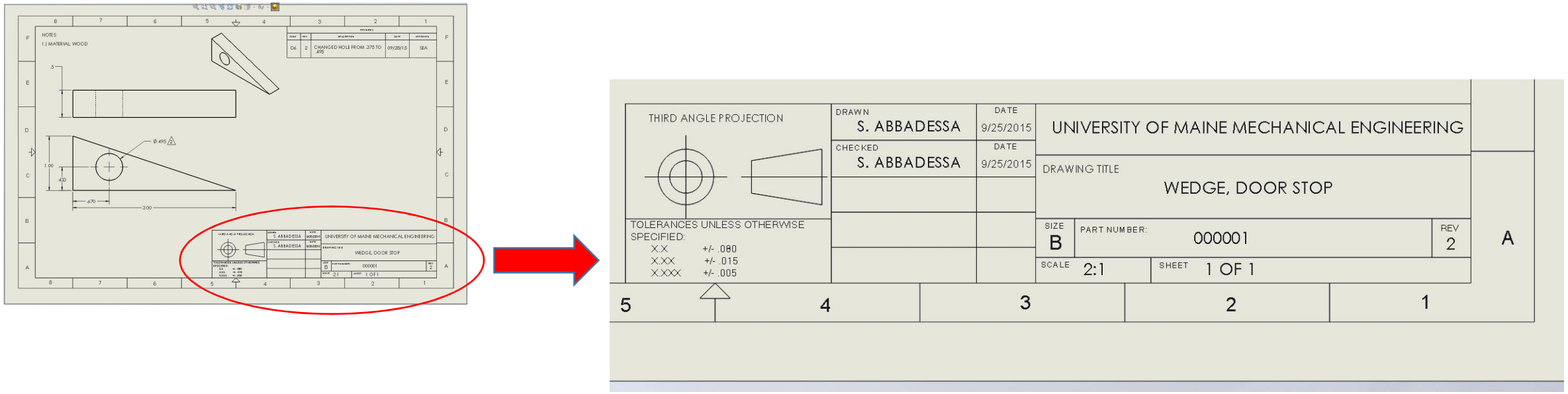
Do not place information below the revision history list area!!! Views and dimensions should not be moved after the initial revision gets approved.



Remember…the print is not for you!!! It is a vehicle to get your ideas to purchasing and manufacturing. Don't make their lives difficult.

The Title Block

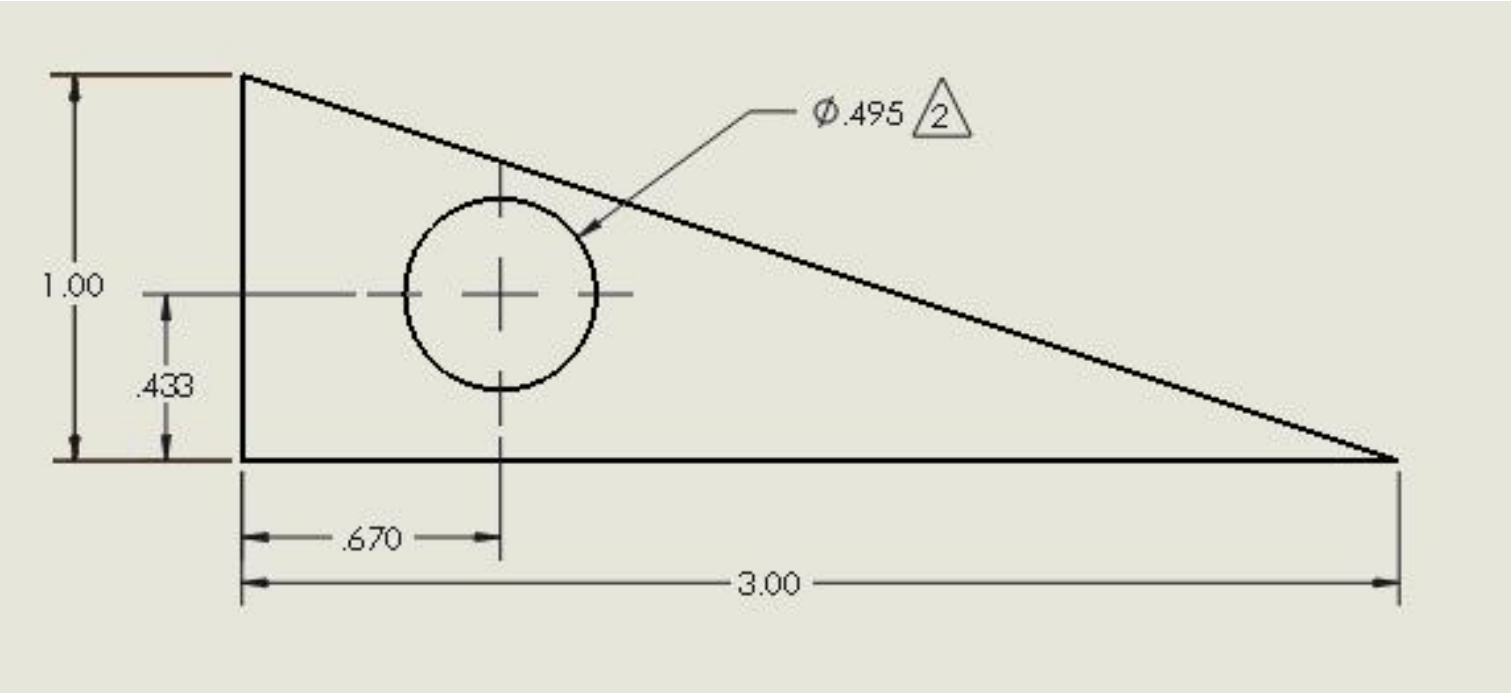
The title block generally contains the projection angle, standard tolerancing, sign offs, the company name, the drawing title, part number, standard view scale, and document revision.



ALL TEXT ON A BLUEPRINT SHOULD ALWAYS BE IN ALL CAPITALS!!!!!

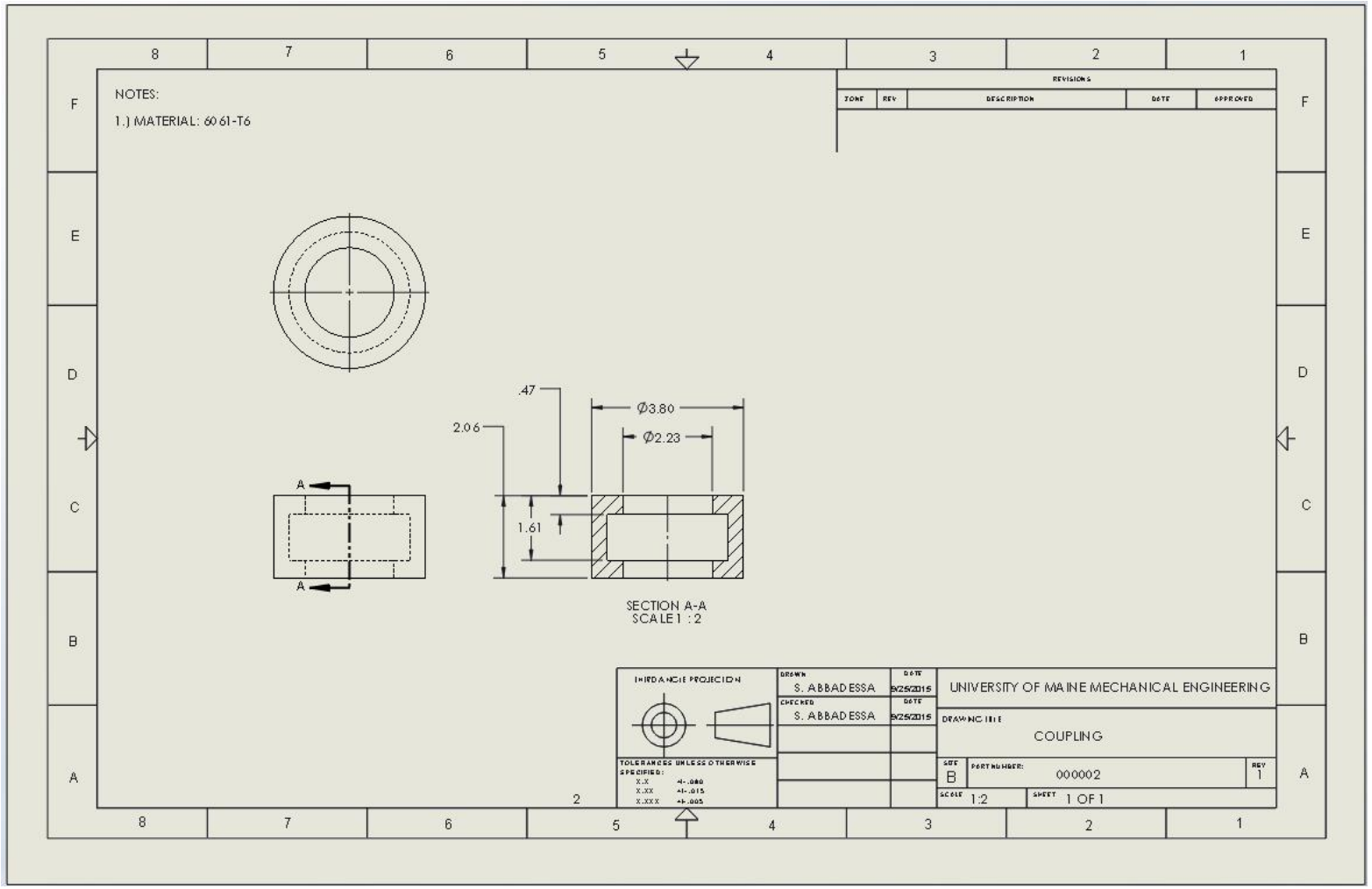
Dimensioning

The meat and potatoes of drafting….and generally the cause of the most disputes. If you end up in a small company with a model shop or a company that makes its own parts, ask how the machinists set up their machines and dimension accordingly. Remember most metal forming machines are XY devices with encoders for distance tracking. Dimension accordingly!! Calling out an angle and distance is not nearly as useful as XY coordinates.



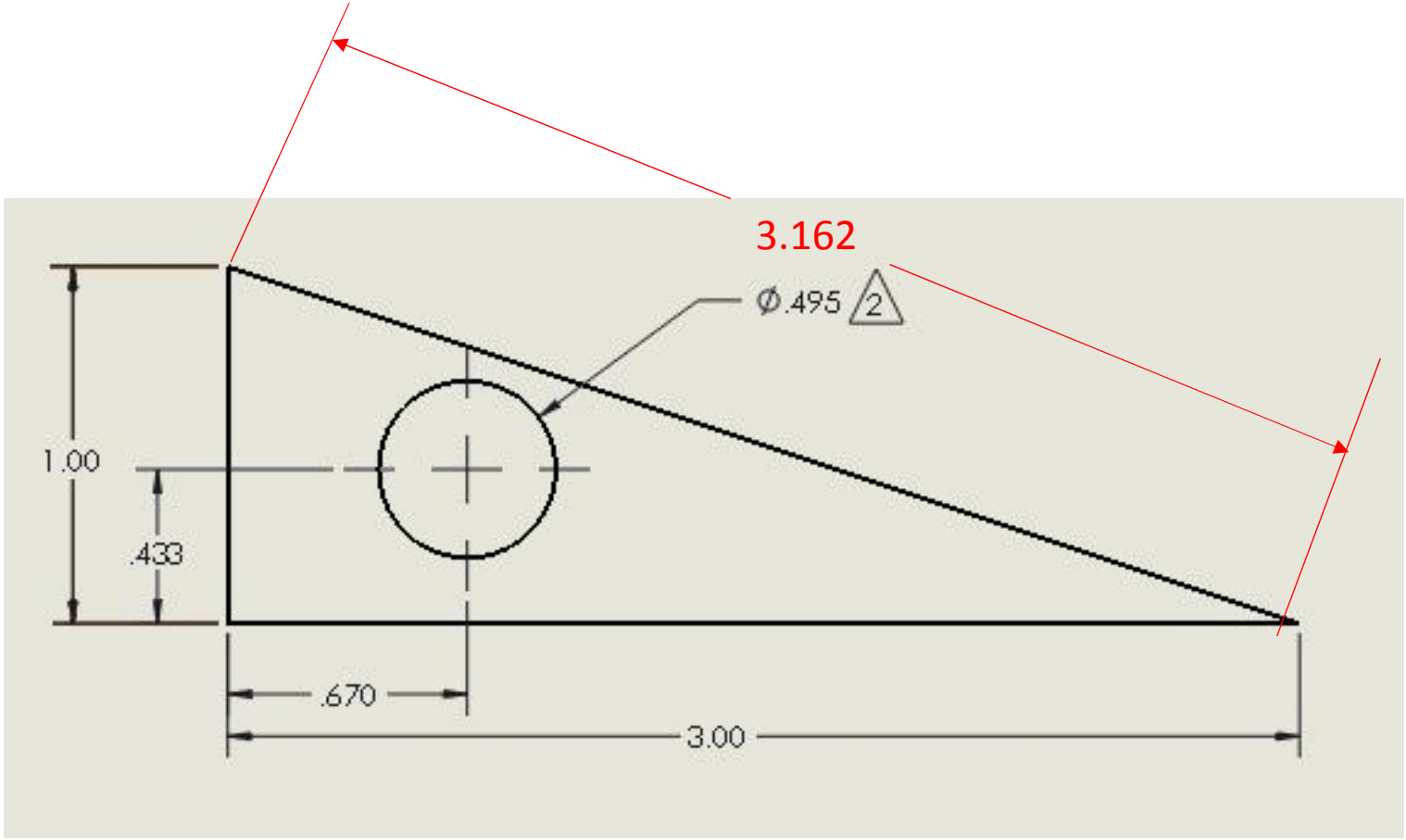
Dimensioning

There are a few general rules to dimensioning. The first important one is never dimension to a hidden(dotted line). Draw additional views if you need them like this section view.



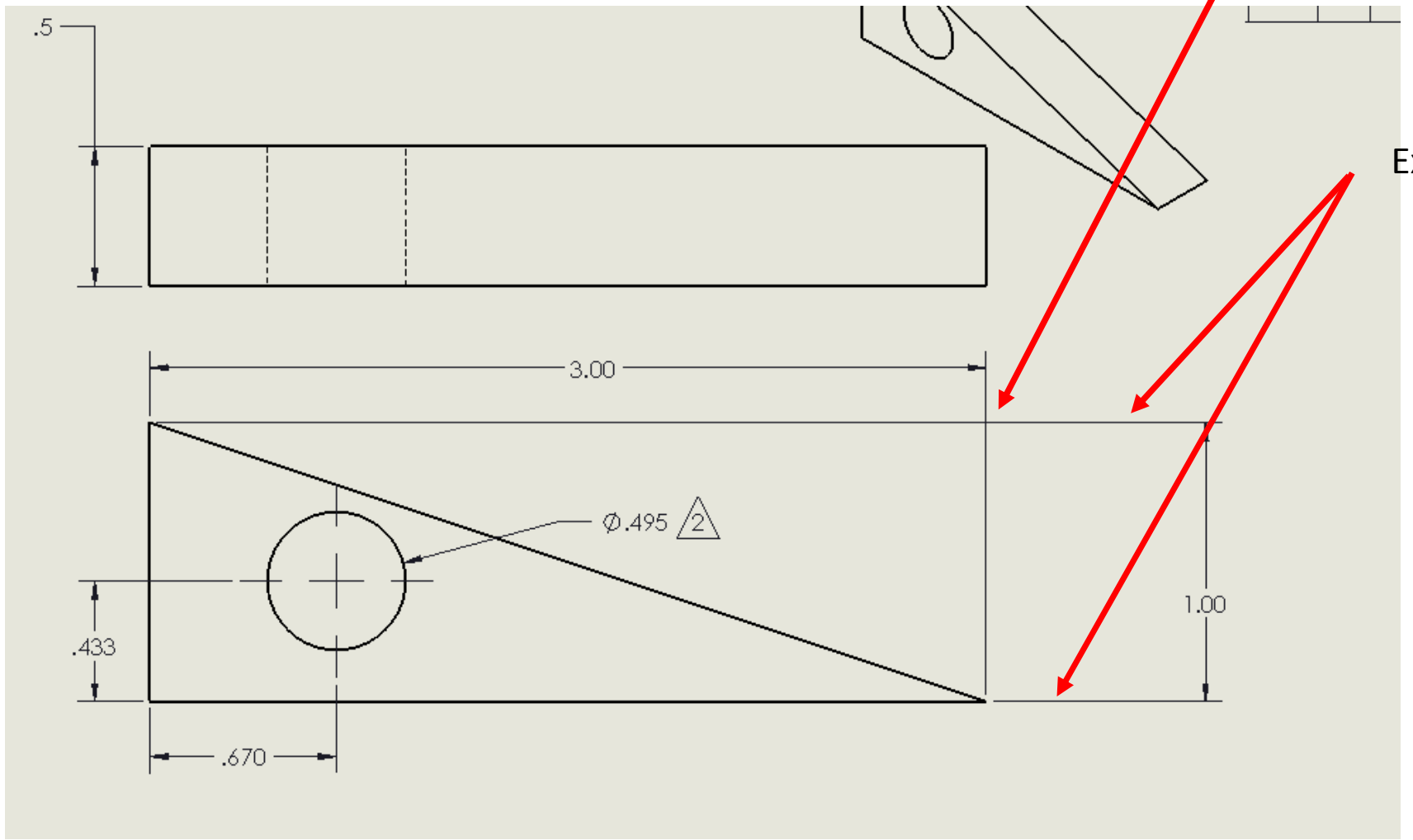
Dimensioning

There should only be one callout for a distance. There should be just enough information to define the geometry and no more.



Dimensioning

Do not cross your extension lines.



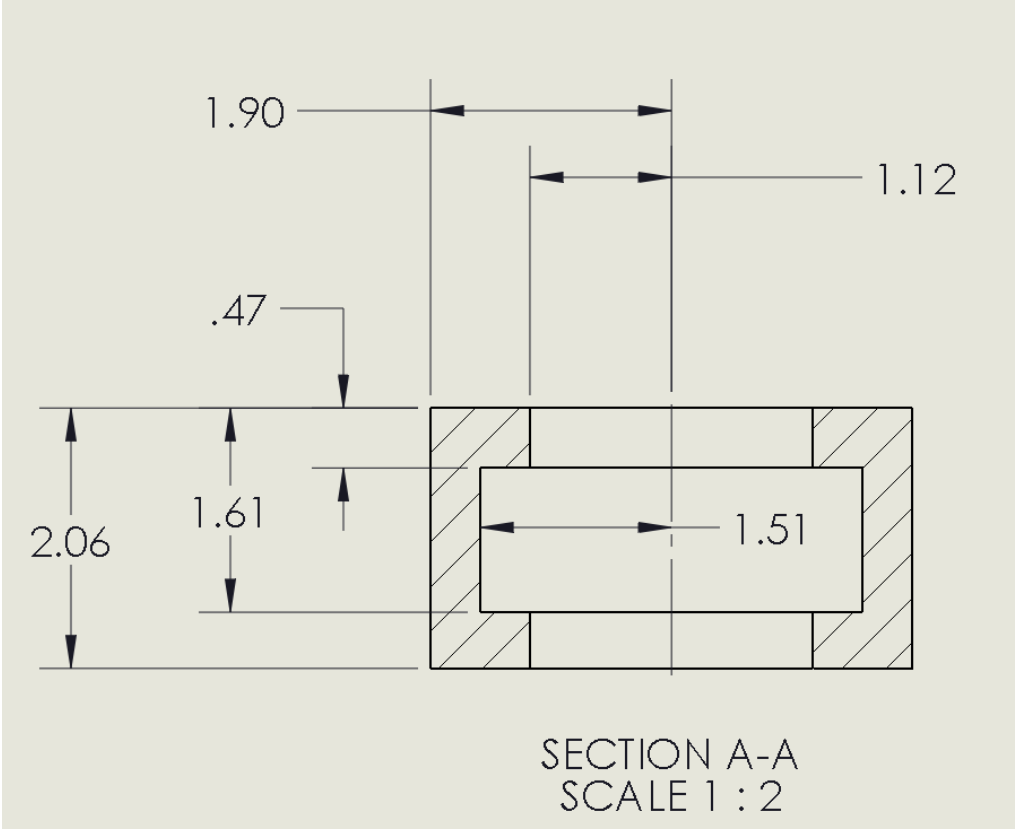
Extension line crossing is bad form

Extension lines

Dimensioning

Whenever possible dimension your part so it can be checked with calipers.

BAD!!



BAD!!

Don’t do this.

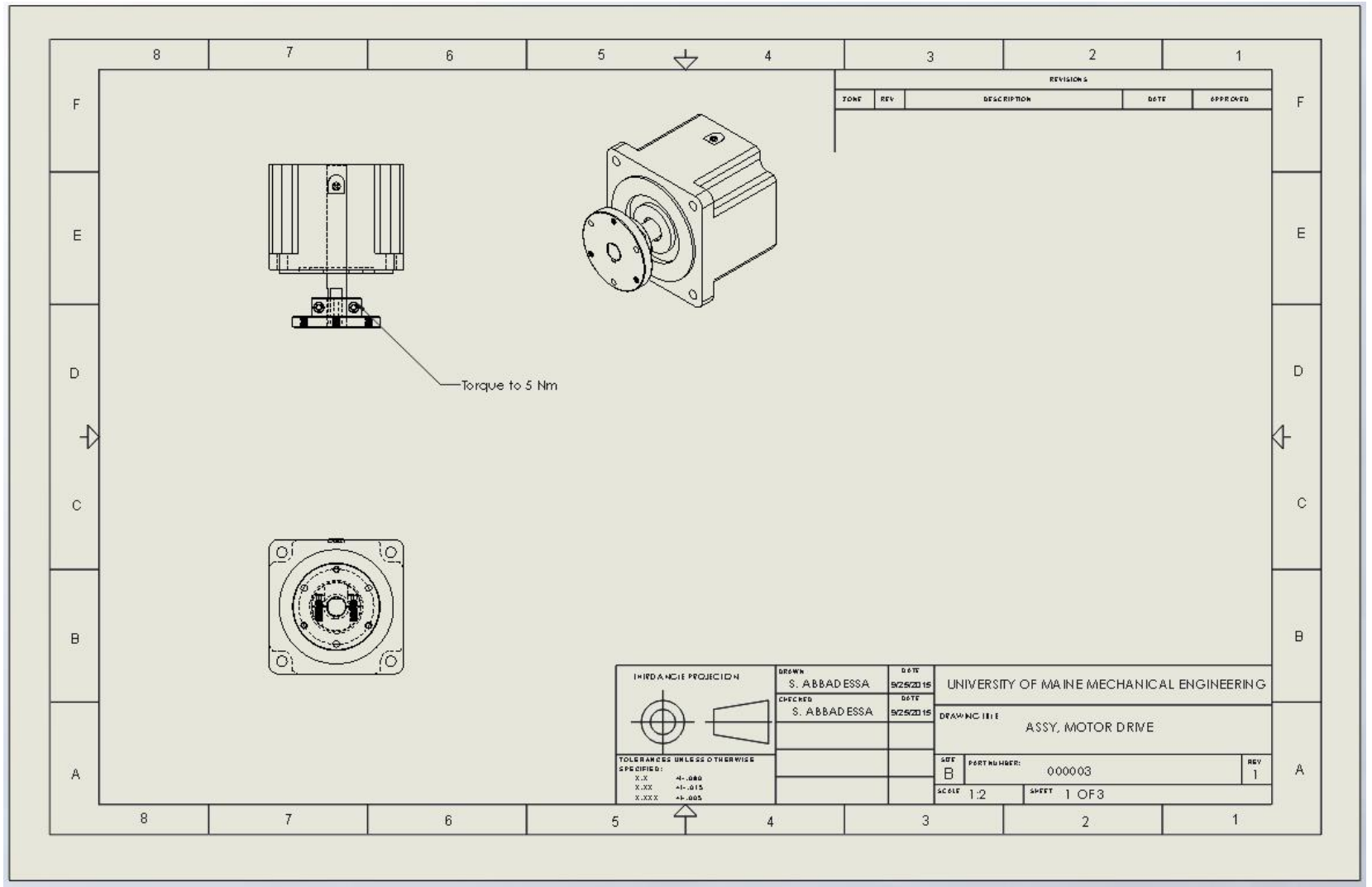
Assembly Drawings:

Assembly drawings show you how to put the pieces together. They include any welding, bonding, and post assembly machining.

Here is what I like to see:

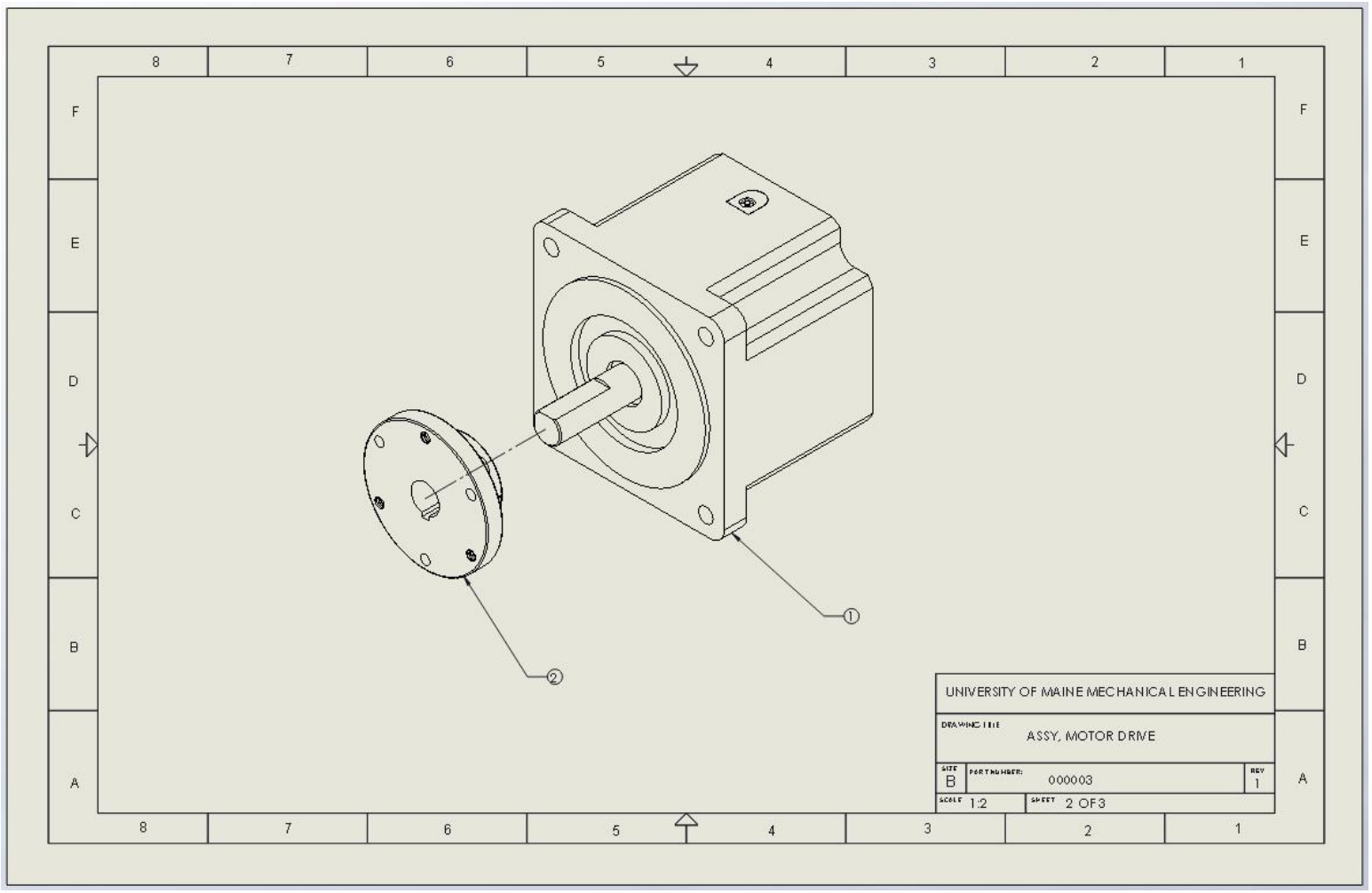
Assembly Drawings:

Page 1: Just some overalls to show the manufacturing folks and purchasing people what they are making.



Assembly Drawings:

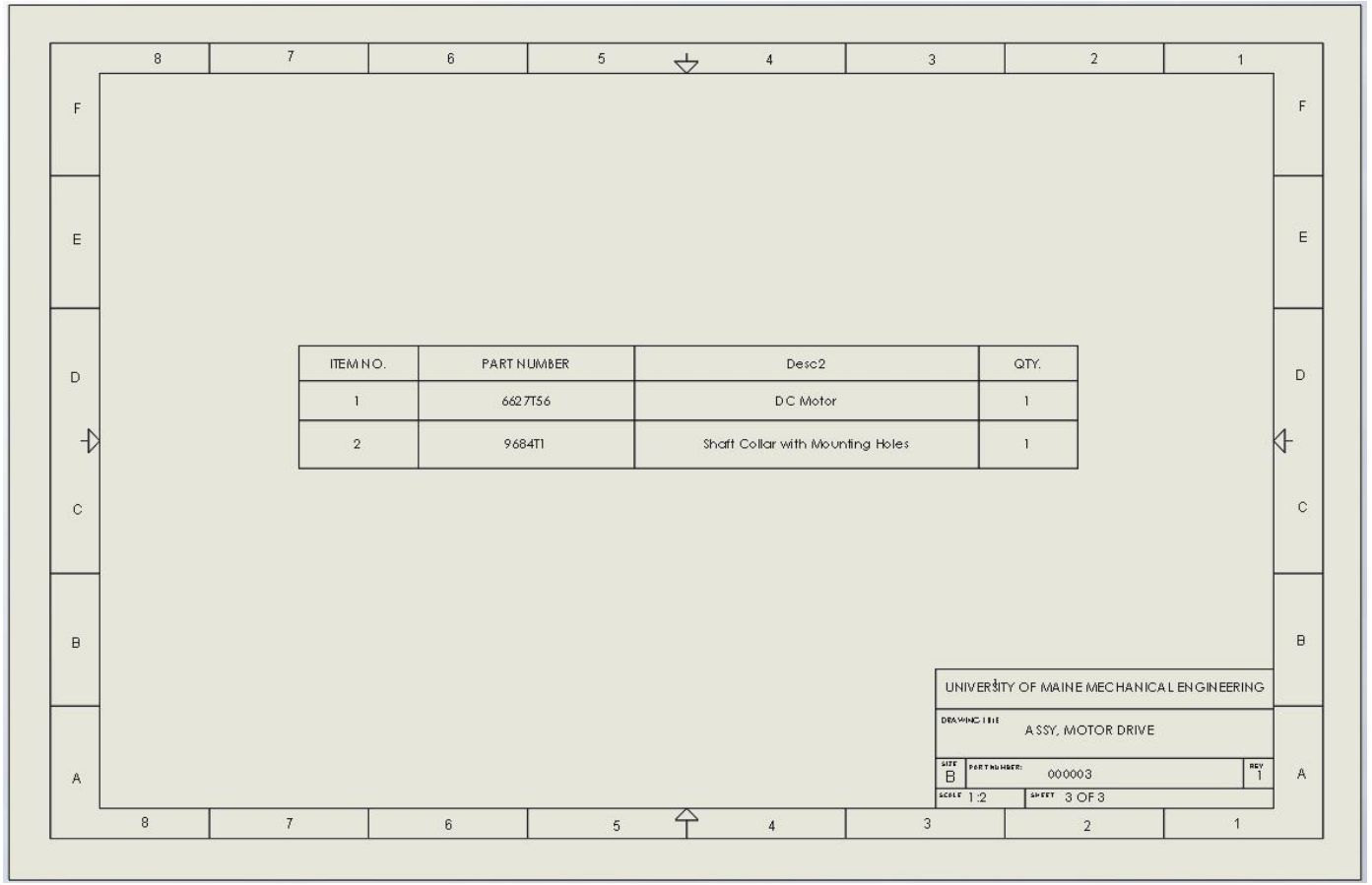
Page 2: An exploded view showing where everything goes



The numbered balloons show the line item on the bill of materials on the next page.

Assembly Drawings:

Last Page : The bill of materials. I put it on the print so it stays associated to the solid model.



ITEM NO.	PART NUMBER	Desc2	QTY.
1	6627T56	DC Motor	1
2	9684T1	Shaft Collar with Mounting Holes	1

UNIVERSITY OF MAINE MECHANICAL ENGINEERING

DRAWING TITLE: ASSY, MOTOR DRIVE

SIZE B | PART NUMBER: 000003 | REV 1

SCALE 1:2 | SHEET 3 OF 3

I like to include my company's part number, the vendor, and the vendor part number in the BOM so purchasing can tell where I found an item.

Tolerancing

This is probably the biggest problem I have had with new engineers. On a typical print we dimension to the ideal size we would like to see. That is NEVER what we get so we need to set some allowable variation….a.k.a. tolerancing.

People always seem to think that putting narrow limits on their parts will make it always fit together……WRONG!!!! Narrower limits WILL invariably drive the cost of the parts up due to an increased scrap rate and inspection requirements. Let's consider some basic sizes:

A sheet of paper is typically .003 inches thick.
Fits on a 1" shaft are typically in the .0002 range
The CNC mill in the Crosby machine shop was only rated to hold +/-.0016" when it was brand new.

Tolerancing

The rules of thumb that I use for typical parts are:

CNC milled or lathe parts

+/- .010 can be done blindfolded by a good machinist
+/- .005 reasonably easy to hit
+/- .003 not so easy but possible
+/- .001 forget it. The thermal expansion and flexing of the machine tool is more than this.

Laser cut sheet metal:

+/- .010 Best possible in thicknesses under roughly 3/8”, except for bend locations.

Tolerancing

CNC plasma cut sheet metal

+/- .030 Best possible, but highly dependent on the material thickness and alloy

Drilled holes

+/-.003 Best possible. Never even think about making a pressed fit from a drilled hole.

Drilled and reamed holes

+/-.001 Best possible

Tolerancing

Lost wax casting

+/-.045 Best possible

Anything that needs to be held closer that +/-.001 needs to be ground or lapped to fit. $$$$

Any surface which does not mate with another part can generally have a large allowable tolerance. TAKE ADVANTAGE OF THIS TO REDUCE THE COST OF YOUR PARTS!!!!