CHECKSHEET FOR INITIALS ASSIGNMENT

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ME 301-02

Part A. Pre-CAD Plan

✓ Identification/description of basic shapes
✓ Consideration of dimensions (attempting to minimize these)
✓ Consideration of supporting relations
✓ Selection/positioning of origin
✓ Initial thoughts about reference geometry
✓ Itemization/explanation of assumptions

Part B. Process Documentation

✓ Rationale for usage of sketch tools
✓ Clear visualization of relations
✓ Thoughtful use of reference geometry
✓ Details on implementation of SW features
✓ Annotated design tree
✓ Compelling lessons learned

Part C. Products (based on finished model and drawing)

✓ Fully-defined sketches
✓ Creativity/complexity of solid model
✓ Enhancements to appearance of solid model
✓ Use of ME drawing template
✓ Multiple, non-redundant views in 3rd Angle orientation
✓ Thoughtful dimensioning scheme
- Machined initials into a shelled casting with a flange including fastener holes
- JFK initials will be machined into the top surface of the casting
- I will design the part in the following order:
  - On plane 1, sketch rectangular profile, only dimensioning one side and using a ratio of global variables to define the ratio (1 = 2.66)
  - Extend previous sketch up 1 draft angle of 3°
  - On plane 3, I will sketch the base of the extruded boss proportionally to dimensions it similar to the first centerpoint feature.
  - I will extend this up 1 draft angle of 3°
- On plane 2, I will sketch my initials & extend out from into the face.
  - all sketch features will be proportioned to 1 dimension so they will update by editing a single dimension (using global variables)
  - fillets will be included to make it machineable
- the part will be shelled from the backside
- materials will be selected to reflect cast/machined edges
Because most of the lines are straight & orthogonal to one another, mostly horizontal, vertical, perpendicular, parallel, constraints will be used. I will use a lot of dimensions, however, they will be global variables. I will only have to update the other lengths (L) and the rest of the geometry will adjust proportionally. I will use fixed angle dimensions for the centerlines of the legs of the X, with edges offset (if needed), a diagonal construction line (not shown) will be used at a midpoint constraint to place the second on the center of the base set & it is built in on.
Initials Assignment
Parts B (Process Documentation) and C (Products)
I began this part by sketching, on the front origin plane, the rectangular profile of the base flange on my part. The origin is on the top left (as viewed from the first picture on this page) of the rectangle. The shape is constrained using the auto-generated horizontal/vertical constraints that are created with rectangles. It is defined with one user-defined dimension for the height of the rectangle, the horizontal dimension is a global variable [=2.6*(height dimension)] so it remains proportional to the text and can be modified easily. After creating this sketch, I extruded it in the positive z direction and changed my material to cast carbon steel and darkened the color slightly.

I then added a draft to my extruded base flange, I gave it a draft angle of 3 degrees (the same with the rest of the part). I set the back of the part as the Neutral plane and the four sides as the draft faces.

My next step was the sketch and extrude the boss in the center of the part. I located this sketch by projecting the original sketch for the base flange onto the very front plane of the base flange and draw a construction line from the top left corner to the bottom right, with coincident constraints on each end. I then started a center rectangle coincident to the center point of the diagonal construction line I previously sketched, added a user-defined height sketch and a global variable for the width dimension (=2.9*(height dimension)) and created a drafted extrusion of that sketch, building in my 3 degrees of draft.

Next, I created a shell, picking the back as my shell face and using Multi-thickness to leave extra material behind the front face to extrude cut my initials into.
When sketching my initials, my goal was to create a closed loop sketch that could be resized by changing a single dimension without having to reposition. To accomplish this, I began by creating a center point rectangle positioned on the midpoint of a diagonal line stretching across the face I was sketching on (the top of the extruded boss), both of these were set as construction lines, the rectangle was given a user-defined height dimension with a global variable parametric width dimension (=2.6*(height dimension)). This construction geometry rectangle would be the basis of my initials, everything changes proportionally by just updating the height of the rectangle. This view shows my overall dimensions and that it is fully defined.
This view (above) of the sketch shows all of my constraints, and some of my global variable dimensions. Each global variable is a fraction of the height dimension, and every dimension that is applied multiple times (i.e.: 0.45" for the thickness of the font) is set as a fraction of the height in the first instance, and then every other one is set equal to that first instance, so if I wanted to change my font thickness, I could do so by updating the 0.45" dimension farthest to the left and down (see the blow picture of my sketch) and the rest would update automatically. All of the 2D fillets were applied within the same operation and can be updated at once with the R0.10 dimension between the F and the K. The legs of the K are defined by drawing construction lines from the edges of the rectangle to two intersection points as shown, offsetting a line on either side, constraining it with parallel constraints and global variables, and then trimming as necessary and reapplied constraints to the separated line segments. Below is my sketch with constraints set to invisible for clarity. On a side note, the interior radius is set to .100", so that they may be machined with a 3/16" endmill, a 3/8" would be used for the rest of the text.
After creating my fully-defined initials sketch, I used an extrude cut to cut it into the surface of the boss. After doing so, I applied the machined steel appearance to all of the faces that would be milled into the part. I added 3D fillets (below picture) to all of the exterior the edges of the part except for the back of the base flange, as it would be cast.

Next, I created a sketch on the front of the base flange to position the center points of the four holes that would be put around the flange (see below). On each side, I used a construction line across the center of the surface of the flange and a mid-point construction line for the point spacing, I defined one with a user-defined dimension and set the other equal to that with global variables (I should have used a horizontal constraint, although I did not know they worked for points at that time). With my point's positions fully defined, I entered the hole wizard to create a hole at each location, I used a simple countersunk thru hole with a 9/16” diameter (meant to be used with a ½” bolt with .0625” of slop for alignment — it is decorative after all). I also added a machined steel finish to these surfaces, as they would be drilled out and countersunk.
To finish the design of the part, I added a .100” x 45 degree chamfer to the top of the text and applied a machined steel surface finish for aesthetics.

Shown above are screen capture of the modeling environment with my completed model and my design tree.
Lessons Learned:

1. One valuable thing I learned throughout his assignment was that solidworks has a built in draft extrusion feature, the user does not have to take the time to add a draft to faces that have already been created, but can specify and positive or negative draft angle and do it all in one shot.

2. I also learned a bit more about how global variables work within solidworks, and how they differ from autodesk products. They are very useful for creating powerful sketches that can be quickly updated based upon changes to other geometry.

3. Finally, though perhaps more importantly, I became familiar with the drawing enviroment of solidworks. I found it to be quite similar to that of Autodesk, though less intuitive in some ways. The smart dimension feature is very powerful, and the ability to create a set of baseline dimensions from a single datum at once is good.
ALL DRAFT ANGLES 3'
ALL RADIUS .500" UNLESS OTHERWISE NOTED
NOT BASELINE DIMENSIONED - MANUFACTURING PROCESS (CASTING)

EXTENSION OF DRAFT SURFACES

4X Ø .5625 THRU ALL
Ø .61 X 90°, NEAR SIDE

SECTION C-C
1.00 THK
.50 THK TYP

SECTION B-B
.50

4.00 OC
2.50
13.285 OC

DETAILED TEXT DIMS ON PG. 2