EML2322L – MAE Design and Manufacturing Laboratory

Design for Manufacturability (DFM) Examples

Simply stated, DFM is the ability to design components and products that are easier and thus more affordable to manufacture. There is an old adage that *a good engineer can do for a dollar what anyone else can do for two.* In addition to designing parts that don't fail under the normal operating conditions, this is a second defining characteristic of a great engineer.

This document illustrates design for manufacturability concepts by contrasting poor and improved designs for three components: a milling machine part, a lathe part, and a sheetmetal part. For each component, a poor design drawing is presented, followed by an improved design drawing, followed by a detailed explanation of why the changes facilitate DFM.

A more complete list of tips for reducing part cost by designing for manufacturability using traditional processes can be found in the comprehensive Design and Manufacturing Tips document on the course website; *that handout is without question the most important document offered in the entire course.*
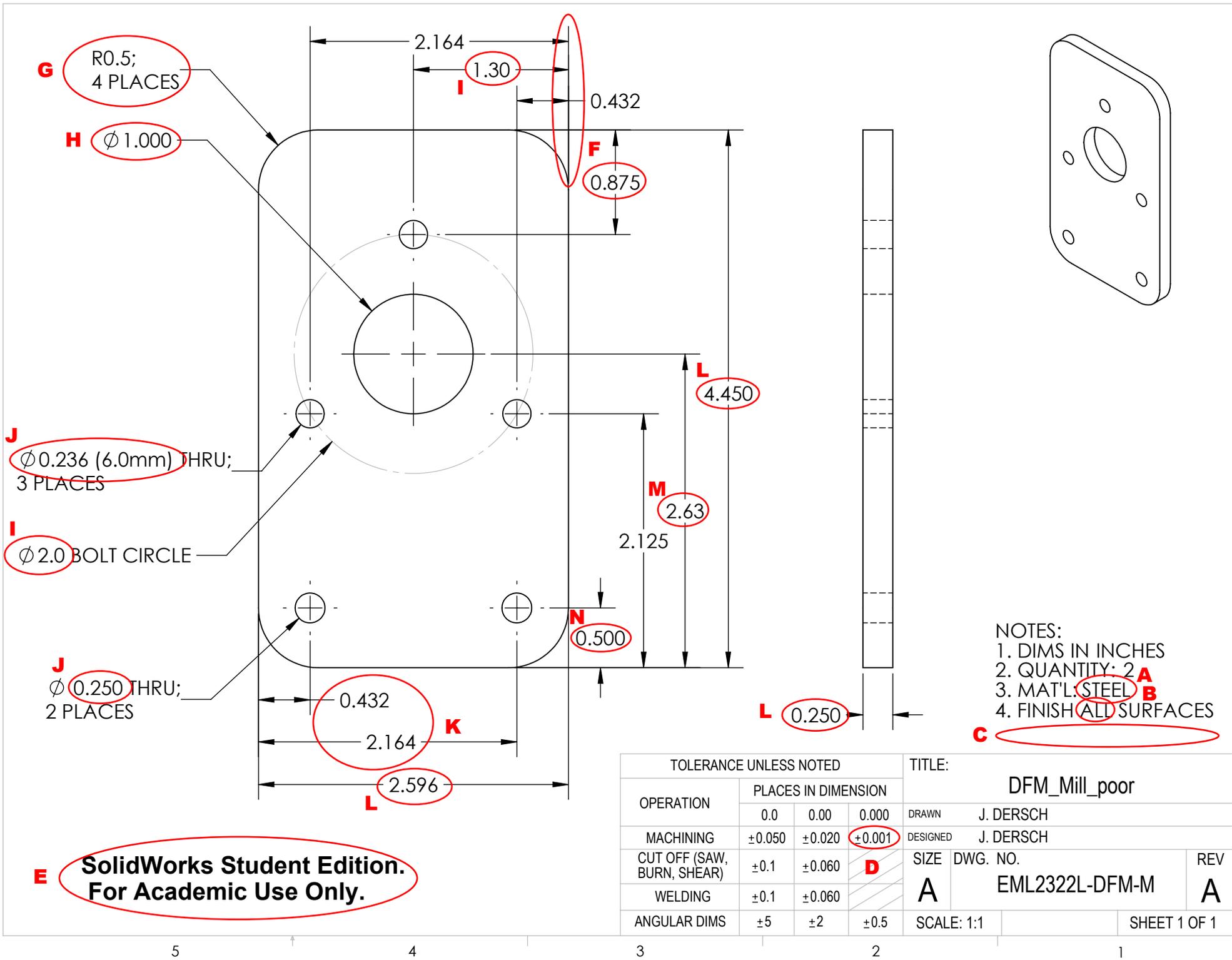
Table of Contents

Milling Machine Part

Lathe Part

Sheetmetal Part

G R0.5;
4 PLACES

H ⌀1.000

2.164

1.30

I 0.432

F 0.875

J ⌀0.236 (6.0mm) THRU;
3 PLACES

I ⌀2.0 BOLT CIRCLE

L 4.450

M 2.63

2.125

N 0.500

J ⌀0.250 THRU;
2 PLACES

0.432

K 2.164

L 2.596

L 0.250

NOTES:
1. DIMS IN INCHES
2. QUANTITY: 2 **A**
3. MAT'L: STEEL **B**
4. FINISH ALL SURFACES

C

TOLERANCE UNLESS NOTED				TITLE:		
OPERATION	PLACES IN DIMENSION			DFM_Mill_poor		
	0.0	0.00	0.000	DRAWN	J. DERSCH	
MACHINING	±0.050	±0.020	**D** +0.001	DESIGNED	J. DERSCH	
CUT OFF (SAW, BURN, SHEAR)	±0.1	±0.060		SIZE A	DWG. NO. EML2322L-DFM-M	REV A
WELDING	±0.1	±0.060				
ANGULAR DIMS	±5	±2	±0.5	SCALE: 1:1		SHEET 1 OF 1

5   4   3   2   1

2.116

1.250

0.384

⌀0.5 THRU

⌀0.248 (6.3mm) THRU;
3 PLACES

⌀2.000 BOLT CIRCLE

⌀ 0.266 THRU;
2 PLACES

4.0

3.625

2.6

2.125

0.5

0.5

1.25

2.0

2.5

0.19

NOTES:
1. DIMS IN INCHES
2. QUANTITY: 2
3. MAT'L: ALUMINUM
4. FINISH NOTED SURFACES
5. BREAK ALL EDGES

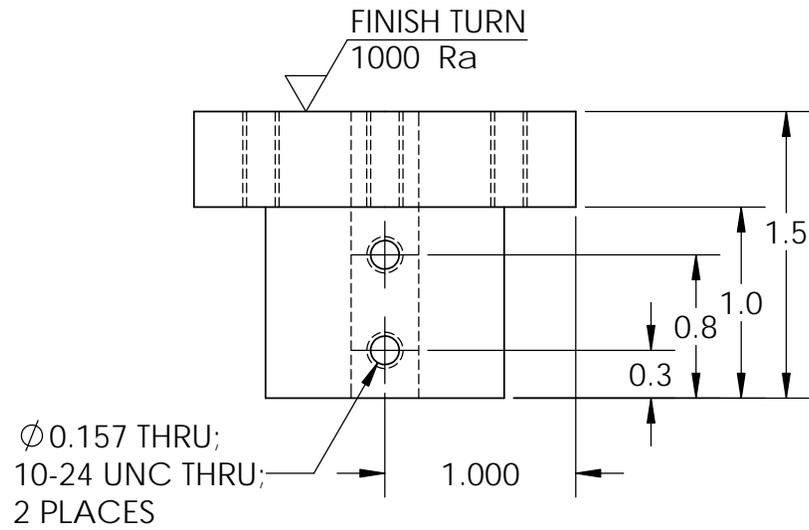| TOLERANCE UNLESS NOTED				TITLE:		
OPERATION	PLACES IN DIMENSION			DFM_Mill_better		
	0.0	0.00	0.000	DRAWN	J. DERSCH	
MACHINING	±0.050	±0.020	±0.005	DESIGNED	J. DERSCH	
CUT OFF (SAW, BURN, SHEAR)	±0.1	±0.060		SIZE	DWG. NO.	REV
WELDING	±0.1	±0.060		A	EML2322L-DFM-M	B
ANGULAR DIMS	±5	±2	±0.5	SCALE: 1:1		SHEET 1 OF 1

4

3

2

1

Explanation of DFM Improvements for Milling Machine Part

A. Steel is more difficult to machine than aluminum. Whenever possible, use the weakest material that satisfies the strength requirement (or other material constraints) which in this case, is aluminum.

B. Finishing surfaces takes time. Unless a finished surface is required for function or appearance, clearly indicate no surface finish is required.

C. For safety during assembly and part handling, always require that sharp edges are broken (removed).

D. A tolerance of $\pm$ 0.001" is difficult to achieve and increases manufacturing time. Always design parts with the loosest tolerances that allow the part (and future assemblies) to function correctly. Consider ± 0.005" to be the tightest reasonable tolerance for lab parts (with the exception of reamer applications).

E. Watermarks covering the drawing are unprofessional and may obscure important information. Always print SolidWorks drawings using a PDF printer (rather than performing a save-as PDF command) to move the watermark to the page border (or remove it).

F. Each dimension datum requires an additional "zeroing" and thus increases manufacturing time and locational error. Generally, only use one datum per axis unless an overriding reason exists.

G. Fillets increase manufacturing time and are generally unneeded, so remove unless absolutely necessary for part function.

H. Tight tolerances and larger hole sizes increase manufacturing time. Always design parts with the loosest tolerances that allow the part (and future assemblies) to function correctly. In this case, a clearance hole is needed through which the motor shaft can pass. If the motor shaft measures Ø 0.354" (9mm), choosing 0.5" guarantees a clearance fit with the loosest tolerances on both the hole diameter (± 0.050") and hole locating dimensions (± 0.020").

I. When matching a hole pattern on another part, tight tolerances are required to ensure proper fitment between parts. Ensure the largest tolerances (i.e. "worst-case dimensions") of both parts will still allow part function and assembly. In this case, a tolerance of ± 0.005" is appropriate.

J. Line fits (i.e. zero clearance) increase assembly time for fasteners. Design using standard clearance hole sizes from the [tap and drill chart](#) for the fastener(s) used to assemble parts.

K. Non-nominal dimensions make reading a drawing more difficult which may result in manufacturing mistakes and/or increased manufacturing time. Design with nominal dimensions when possible. Always design with the loosest tolerances that allow the part (and future assemblies) to function correctly.

L. Use nominal stock sizes and loose (greater than ± 0.010") tolerances when possible. Rev. A requires a 3" x 4.5" x 0.25^{+}" piece of material to manufacture while Rev. B only requires a 2.5" x 4.0" x 3/16" piece of material. Note that, due to the tight tolerances on the thickness of the part, only stock material with a thickness greater than 0.250" may be used. In lab, stock material is usually within ± 0.010" of the nominal size. Additionally, Rev. B reduces part cost by eliminating excess material above the top hole. Finally, always leave at least 1 diameter of material between the edge of a hole and the edge of the part.

M. Loosening this tolerance will result in a part that is easier to manufacture. See point H.

N. Tight tolerances increase manufacturing time. Always design parts with the loosest tolerances that allow the part (and future assemblies) to function correctly. Since this motor mount bracket attaches to a mobile platform which will operate on a floor that is far from flat, ± 0.020" is an appropriate tolerance.

I,J 0.161 THRU;
10-32 UNF;
2 PLACES

L

K 0.750
K 0.250
M 0.03 TYP.
0.03 TYP.

N

O,P 0.44

1.500

K

H ⌀1.5 BOLT CIRCLE

E,F,G ⌀0.15 ▽ 0.250;
10-32 UNF ▽ 0.250;
3 PLACES <120° APART>

Q ⌀2.10
R ⌀0.64
S,T ⌀0.4

NOTES:
1. DIMS IN INCHES
2. QUANTITY: 1
3. MAT'L: STEEL A
4. BREAK ALL EDGES
5. FINISH ALL SURFACES
   B

TOLERANCE UNLESS NOTED				TITLE:	
OPERATION	PLACES IN DIMENSION			DFM_Lathe_poor	
	0.0	0.00	0.000	DRAWN	J. DERSCH
MACHINING	±0.050	±0.020	±0.005	DESIGNED	J. DERSCH
CUT OFF (SAW, BURN, SHEAR)	±0.1	±0.060		SIZE A	DWG. NO. C EML2322L-XXX
WELDING	±0.1	±0.060			
ANGULAR DIMS	±5	±2	±0.5	SCALE: 1:1	SHEET 1 OF 1

REV
A

5   4   3   2   1

FINISH TURN
1000 Ra

1.5
1.0
0.8
0.3

1.000

⌀0.157 THRU;
10-24 UNC THRU;
2 PLACES

⌀1.3
⌀2.0
⌀0.354 +0.001 -0.000 (9mm) THRU;

0.750
0.000
0.375

0.650
0.000

⌀0.157 THRU;
10-24 UNC THRU;
3 PLACES <120° APART>

⌀1.500 BOLT CIRCLE

NOTES:
1. DIMS IN INCHES
2. QUANTITY: 1
3. MAT'L: ALUMINUM
4. BREAK ALL EDGES
5. FINISH NOTED SURFACES

TOLERANCE UNLESS NOTED				TITLE: DFM_Lathe_better		
OPERATION	PLACES IN DIMENSION					
	0.0	0.00	0.000	DRAWN J. DERSCH		
MACHINING	±0.050	±0.020	±0.005	DESIGNED J. DERSCH		
CUT OFF (SAW, BURN, SHEAR)	±0.1	±0.060		SIZE A	DWG. NO. EML2322L-DFM-L	REV B
WELDING	±0.1	±0.060				
ANGULAR DIMS	±5	±2	±0.5	SCALE: 1:1		SHEET 1 OF 1

4   3   2   1

Explanation of DFM Improvements for Lathe Part

A. Steel is more difficult to machine than aluminum. Whenever possible, use the weakest material that satisfies the strength requirement (or other material constraints) which in this case, is aluminum.

B. Finishing surfaces takes time. Unless a finished surface is required for function or appearance, clearly indicate no surface finish is required.

C. Provide a unique drawing name and number for organizational purposes.

D. Watermarks covering the drawing are unprofessional and may obscure important information. Always print SolidWorks drawings using a PDF printer (rather than performing a save-as PDF command) to move the watermark to the page border (or remove it).

E. Blind holes often increase manufacturing time. Design with thru holes when possible.

F. Specifying tolerances that are too loose can result in broken taps or weak threads. Specify tap drill sizes to $\pm0.005"$ (or tighter) tolerances. Note the specified tap drill dimension does not match the size listed for aluminum in the tap and drill chart. This will again result in broken taps or weak threads.

G. When the material is changed from steel to aluminum, a coarse thread becomes the correct choice, since it will be stronger in this weaker material; therefore, specify a 10-24 UNC instead of 10-32 UNF.

H. When matching a hole pattern on another part, tight tolerances are required to ensure proper fitment between parts. Ensure the largest tolerances (i.e. "worst-case dimensions") of both features will still allow part function and assembly. In this case, a tolerance of $\pm0.005"$ is appropriate.

I. When the material is changed from steel to aluminum, a coarse thread becomes the correct choice, since it will be stronger in this weaker material; therefore, specify a 10-24 UNC instead of 10-32 UNF.

J. The specified tap drill dimension does not match the size listed in the tap and drill chart. This is will result in broken taps or weak threads.

K. Tighter tolerances increase manufacturing time, so always design parts with the loosest tolerances that allow the part (and future assemblies) to function correctly. The separation distance between the setscrews is not important for part function, so increase the allowable tolerance range.

L. Each dimension datum requires an additional "zeroing" and thus increases manufacturing time and locational error. Generally, only use one datum per axis unless an overriding reason exists.

M. Chamfers increase manufacturing time and are generally unneeded, so remove unless absolutely necessary for part function.

N. Poor drawing format is unprofessional, increases the likelihood of manufacturing errors, and increases the cost of outsourced parts. Only dimension holes (size and location) in a view in which they are circular.

O. Non-nominal dimensions make reading a drawing more difficult which may result in manufacturing mistakes and/or increased manufacturing time. Design with nominal dimensions if possible. Always design parts with the loosest tolerances that allow the part (and future assemblies) to function correctly.

P. Increasing this dimension reduces the amount of material that must be removed and thus decreases the manufacturing time.

Explanation of DFM Improvements for Lathe Part (cont)

Q. Use nominal stock sizes and looser (larger) tolerances when possible. Rev. A requires turning down a 2.5" diameter piece of material while Rev. B only requires a 2" piece of material. In lab, stock material is usually within ±0.010" of the nominal size. Finally, always leave at least 1 diameter of material between the edge of a hole and the edge of the part.

R. Removing material increases manufacturing time, so remove the smallest amount of material possible for part function (unless weight is a functional goal that justifies the associated increase in part cost).

S. Specifying tolerances that are too large can result in unintended consequences. In this case, the motor shaft on which this wheel is to be attached measures Ø 0.354" (9mm). Consequently, the loose tolerance (±0.050") could result in an interference fit between the hub and motor shaft, which would damage the precision motor shaft and gearing. Alternatively, the loose tolerance could result in a large (0.050") clearance fit between hub and motor shaft, causing unacceptable runout between the axis of rotation of the wheel hub and motor shaft.

T. For properly function, the wheel hub should **very precisely** mate with the motor shaft to which it is attached so the two components rotate concentrically. If the motor shaft measures Ø 0.354" in diameter, the hole size through the center of the wheel hub should be between Ø 0.354" (a line fit) and Ø 0.355" (one thousandth of an inch larger), as denoted by the tight tolerances in Rev. B.

General Notes:

- Provide rectangular coordinates for holes in a bolt pattern to facilitate manufacturing.

- Once the holes in the bolt pattern become thru holes (for easier manufacturability), the part is clearly (and entirely) dimensioned with two orthographic views, so the third view can be removed to unclutter the drawing.
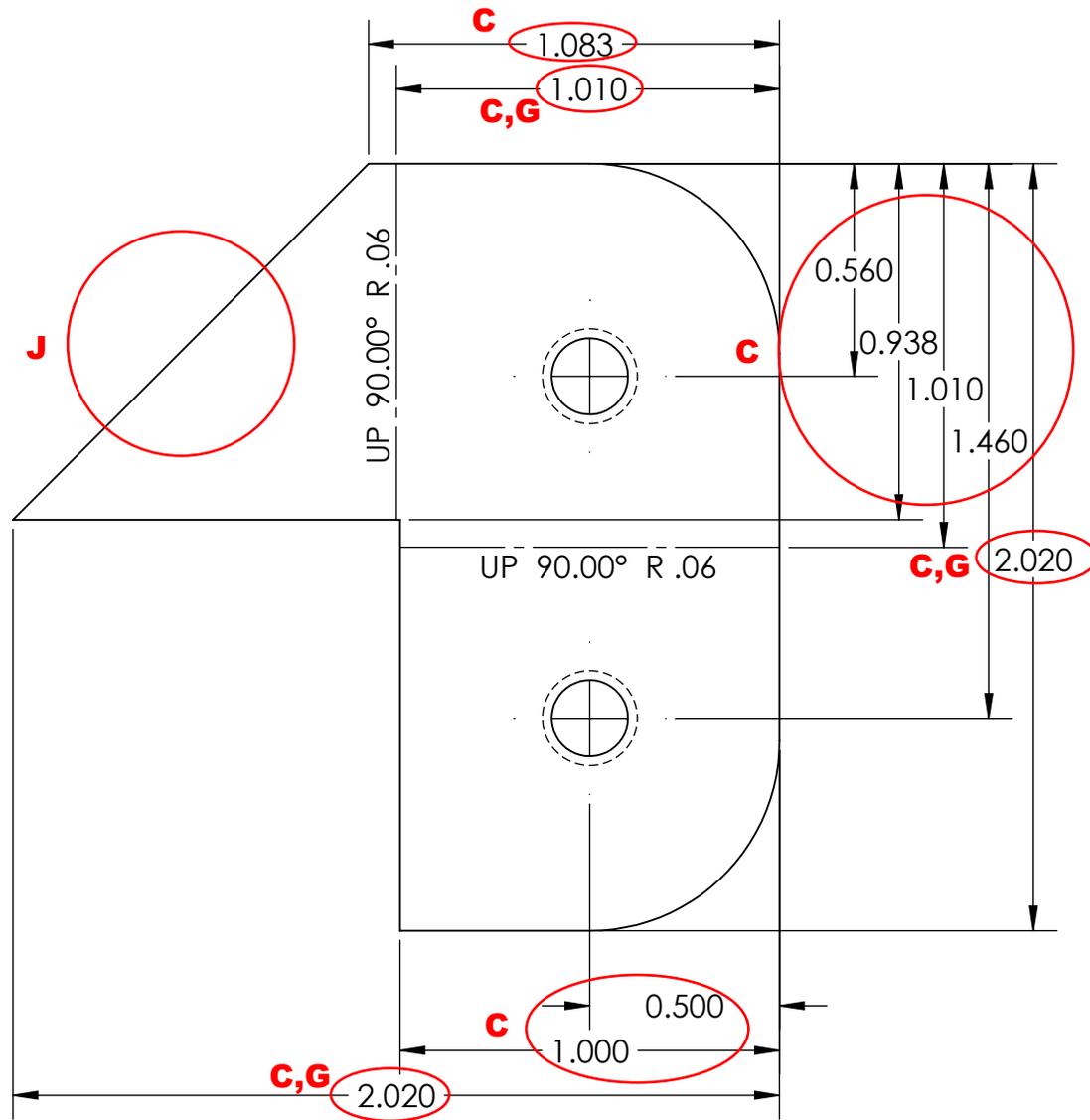
C R0.500

I ⌀0.201 THRU;
1/4-20 UNC THRU;

F

C 0.500

1.060 **C,G**

H
0.56

C 0.500

1.060 **C,G**

WELD

1.060 **C,G**

0.123 **C**

J

C,K
0.938

1.060 **C,G**

C R0.500

I ⌀0.201 THRU;
1/4-20 UNC THRU;

F

C 0.500

C 0.560

NOTES:
1. DIMS IN INCHES
2. QUANTITY: 5
3. MAT'L: 18 GA ALUMINUM **A**
4. FINISH ALL SURFACES
B

TOLERANCE UNLESS NOTED				TITLE:		
OPERATION	PLACES IN DIMENSION			DFM_Sheetmetal_poor		
	0.0	0.00	0.000	DRAWN	J. DERSCH	
MACHINING	**C**			DESIGNED	J. DERSCH	
SHEETMETAL (SHEAR, BEND)	±0.1	±0.060	±0.005	SIZE	DWG. NO.	REV
WELDING	±0.1	±0.060		A	EML2322L-DFM-SM	A
ANGULAR DIMS	±5	±2		SCALE: 1:1		SHEET 1 OF 2

C
1.083
C,G
1.010

J

UP 90.00° R .06

0.560
0.938
1.010
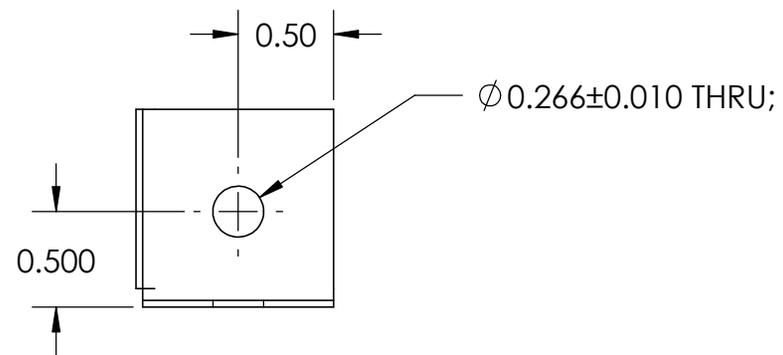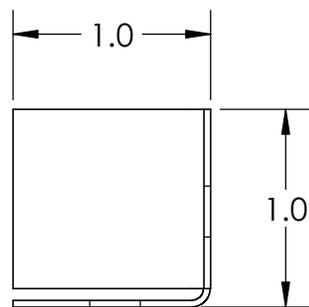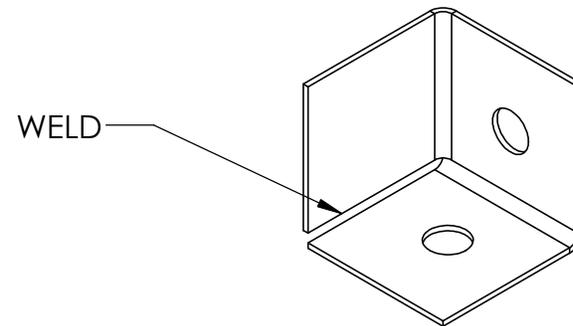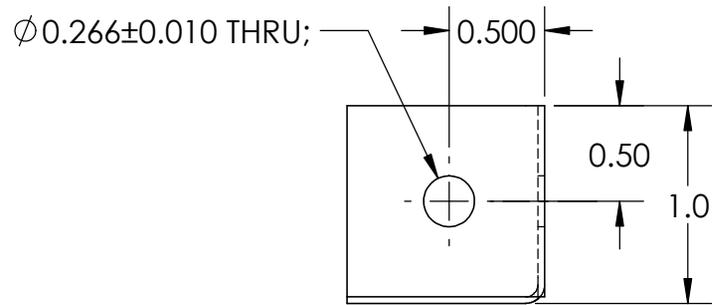1.460

C

UP 90.00° R .06

C,G 2.020

C
0.500
1.000

C,G 2.020

NOTES:
1. DIMS IN INCHES
2. QUANTITY: 5
3. MAT'L: 18 GA ALUMINUM
4. FINISH ALL SURFACES

A

B

D,E

UNFOLDED VIEW
(IF APPLICABLE)

TOLERANCE UNLESS NOTED			TITLE:		
OPERATION	PLACES IN DIMENSION		DFM_Sheetmetal_poor		
	0.0	0.00	0.000	DRAWN	J. DERSCH
MACHINING	C			DESIGNED	J. DERSCH
SHEETMETAL (SHEAR, BEND)	±0.1	±0.060	±0.005	SIZE	DWG. NO.	REV
WELDING	±0.1	±0.060		A	EML2322L-DFM-SM	A
ANGULAR DIMS	±5	±2		SCALE: 2:1		SHEET 2 OF 2

5   4   3   2   1

⌀0.266±0.010 THRU;

0.500
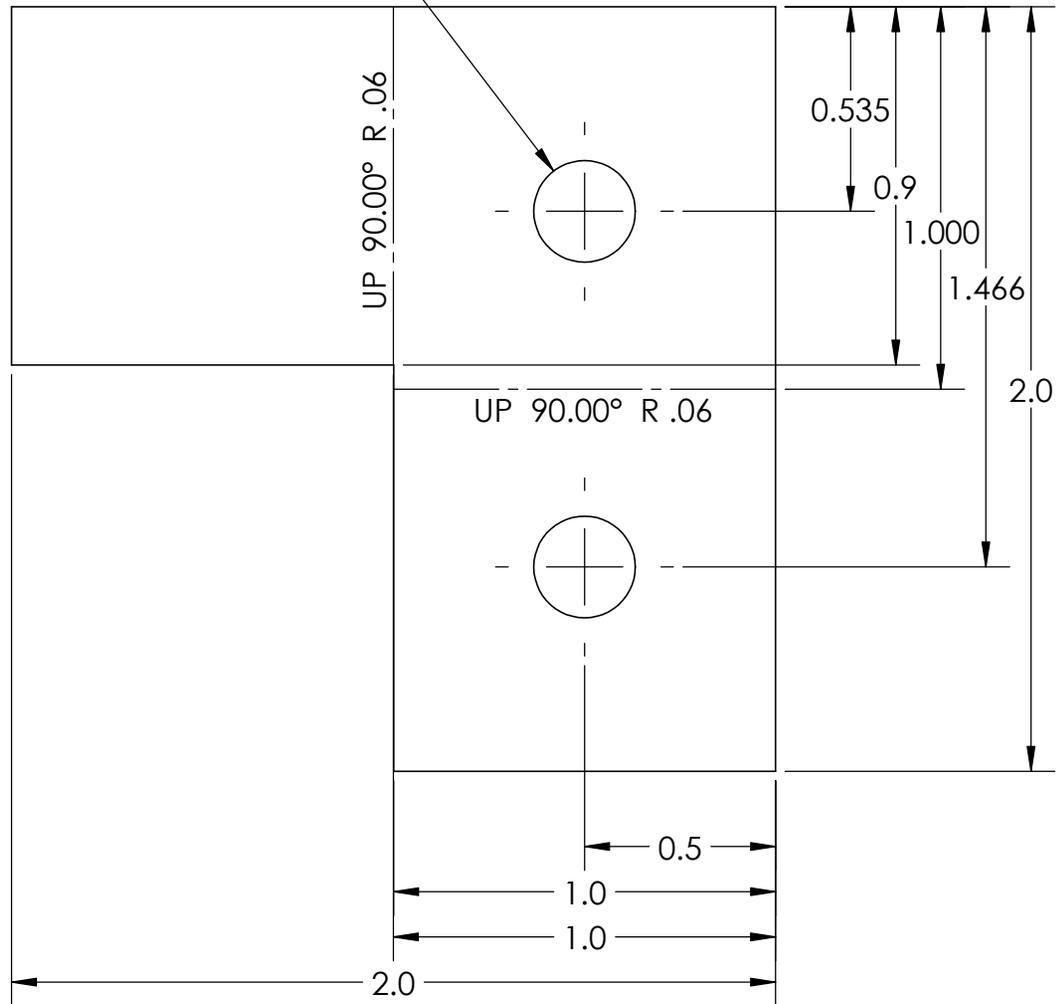
0.50

1.0

1.0

1.0

0.50

WELD

⌀0.266±0.010 THRU;

0.500

NOTES:
1. DIMS IN INCHES
2. QUANTITY: 5
3. MAT'L: 18 GA STEEL
4. BREAK ALL EDGES
5. FINISH NO SURFACES

FOLDED VIEW

TOLERANCE UNLESS NOTED				TITLE: DFM_Sheetmetal_better			
OPERATION	PLACES IN DIMENSION						
	0.0	0.00	0.000	DRAWN	J. DERSCH		
MACHINING				DESIGNED	J. DERSCH		
SHEETMETAL (SHEAR, BEND)	±0.2	±0.100	±0.060	SIZE A	DWG. NO. EML2322L-DFM-SM		REV B
WELDING	±0.1	±0.060					
ANGULAR DIMS	±8	±5		SCALE: 1:1			SHEET 1 OF 2

4   3   2   1

⌀0.266±0.010 ;
2 PLACES

UP 90.00° R .06

UP 90.00° R .06

0.535

0.9

1.000

1.466

2.0

0.5

1.0

1.0

2.0

NOTES:
1. DIMS IN INCHES
2. QUANTITY: 5
3. MAT'L: 18 GA STEEL
4. BREAK ALL EDGES
5. FINISH NO SURFACES

UNFOLDED VIEW

TOLERANCE UNLESS NOTED				TITLE:			
OPERATION	PLACES IN DIMENSION			DFM_Sheetmetal_better			
	0.0	0.00	0.000	DRAWN	J. DERSCH		
MACHINING				DESIGNED	J. DERSCH		
SHEETMETAL (SHEAR, BEND)	±0.2	±0.100	±0.060	SIZE	DWG. NO.		REV
WELDING	±0.1	±0.060		A	EML2322L-DFM-SM		B
ANGULAR DIMS	±8	±5		SCALE: 2:1		SHEET 2 OF 2	

4   3   2   1

Explanation of DFM Improvements for Sheetmetal Part

A. Thin aluminum is more difficult to weld than thin steel, so use steel for sheetmetal parts that require welding. Alternatively, you can use rivets or adhesives to attach sheetmetal pieces.

B. Finishing surfaces takes time. Unless a finished surface is required for function or appearance, clearly indicate no surface finish is required.

C. Tighter tolerances increase manufacturing time, so always design parts with the loosest tolerances that allow the part (and future assemblies) to function correctly.

D. Watermarks covering the drawing are unprofessional and may obscure important information. Always print SolidWorks drawings using a PDF printer (rather than performing a save-as PDF command) to move the watermark to the page border (or remove it).

E. Delete ambiguities to avoid confusion. Remove "(IF APPLICABLE)" from drawings. If you are providing a drawing, it is clearly applicable.

F. Fillets increase manufacturing time and are generally unneeded, so remove unless absolutely necessary for part function.

G. Use nominal sizes to reduce manufacturing time and errors due to transposing unusual dimensions.

H. Each dimension datum requires an additional "zeroing" and thus increases manufacturing time and locational error. Generally, only use one datum per axis unless an overriding reason exists.

I. Five full threads are generally required for fastener interfaces to achieve full strength. Consequently, never design threads in sheetmetal parts; rather, use thru holes or rivet nuts.

J. Unnecessary features increase manufacturing time. Design parts with the simplest and minimal number of features possible.

K. Do not provide redundant / unnecessary dimensions.