1.0 MACHINING - GENERAL

1.1 The type of machining equipment currently in use at de Havilland is considered to be **"Conventional CNC"** machining equipment. That is, the machining spindle operates at up to 7000 rpm. A new generation of high speed machining equipment is becoming available to the industry which utilizes higher machining rates. This **"High Speed"** equipment operates at 16000 to 48000 rpm. This high speed equipment has several advantages over the conventional CNC equipment:

Thinner machined sections (See Figure 6)
Closer Tolerances on machined sections
Better Surface Finish (Ra 63 or better)
Less Inspection
Less Deburring
Less Distortion

1.2 **When to Design for High Speed Machining**
A component should be designed for high speed machining whenever the weight advantage due to the capability to machine thinner sections and due to the closer tolerances is 10% or more over conventional CNC machining.

(B) Where parts have been specifically designed for high speed machining, the drawing must include a general note to define the part as follows:

"PART TO BE FABRICATED USING HIGH SPEED MACHINING"

1.3 **Machine Capacity**
The majority of high speed machines currently in operation have a part size limitation of approximately 40" by 30". There are however, machines coming into service with larger capacities.

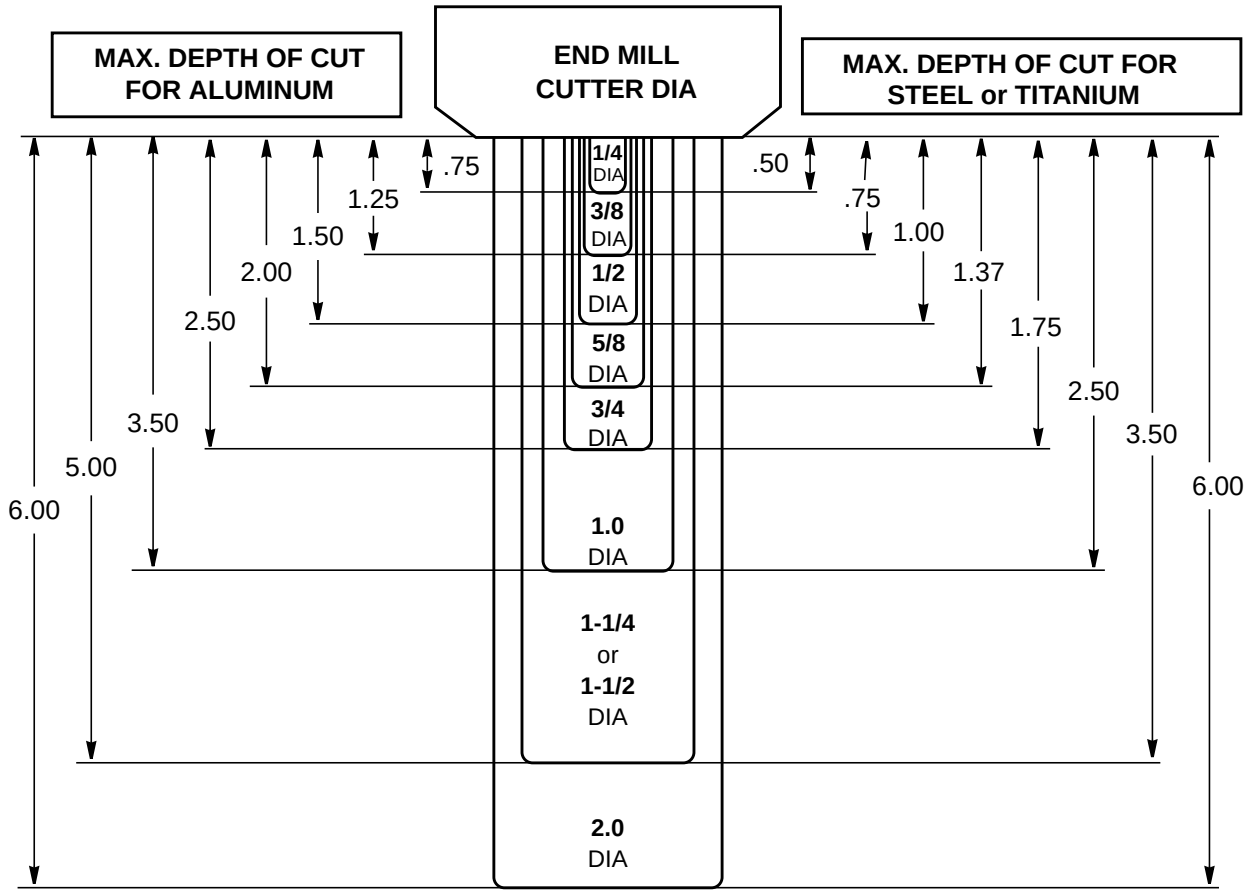
1.4 **Special Inspection Procedures**
All CNC machined 7000 series aluminum alloy parts require inspection for soft spots (to detect local overheating) in accordance with PPS 32.03 as follows:
- For a first-off part tape proving and whenever a change is made to the N/C tape, anodize the parts at 40 volts as per PPS 32.03 and inspect for soft spots as per PPS 32.03 and QAP 4.11/09. Re-anodize at 22±2 volts.
- For all production parts, anodize as per PPS 32.03 and inspect for soft spots as per PPS 32.03 and QAP 4.11/09.

(B) PARA 1.2 - REF HIGH SPEED MACHIINING DRAWING NOTE ADDED

REV: | REV: | REV: | REV: | REV: B - 12 MAR 99 | REV: A - 27 DEC 95

2.0 Depth of Cut

2.1 The maximum cutter depth vs cutter diameter for aluminum, steel and titanium are shown in Figure 1 for conventional CNC machining and Figure 2 for high speed machining.



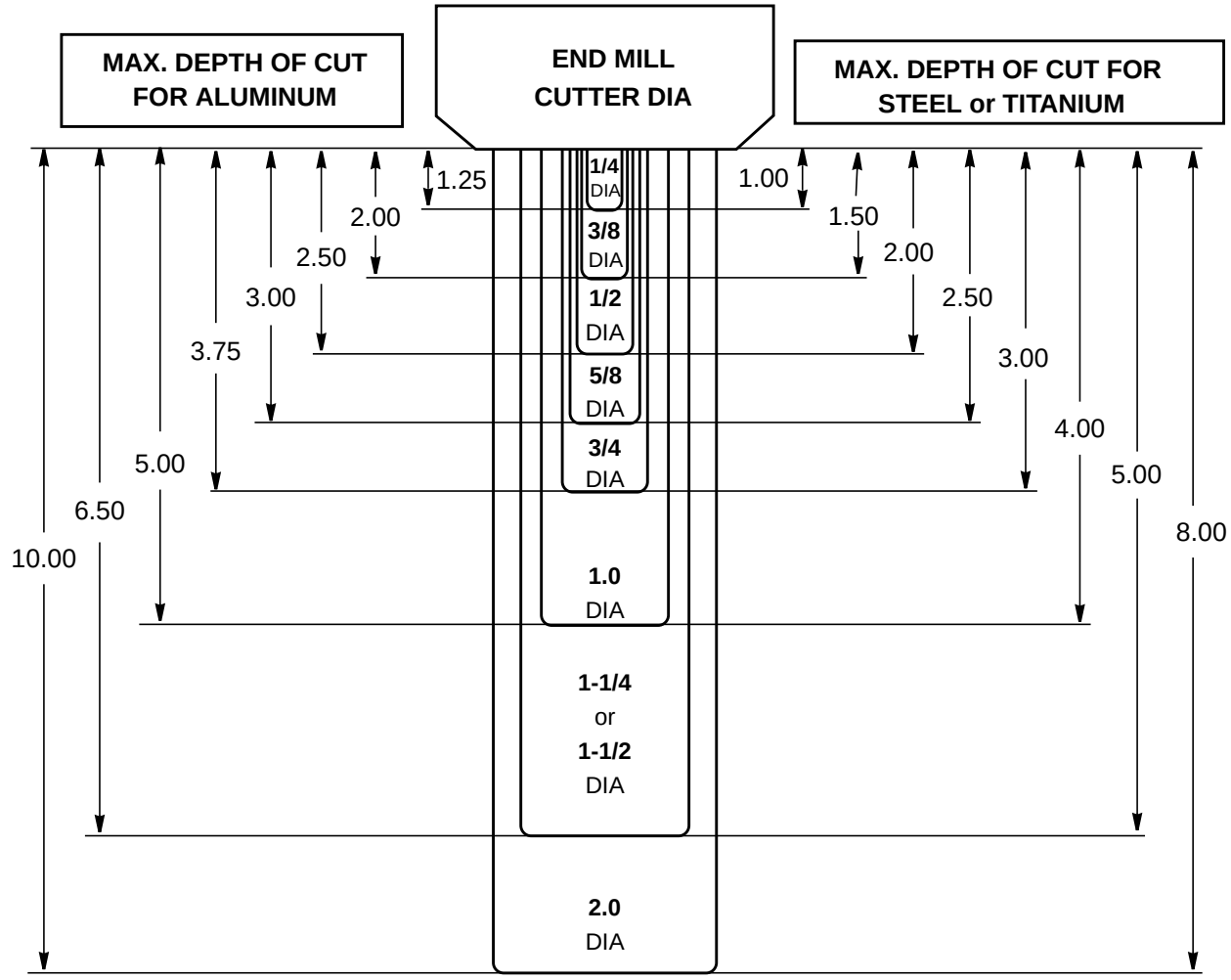
Ⓑ **FIGURE 1: MAXIMUM CUTTER DEPTH/CUTTER DIAMETER - CONVENTIONAL CNC MACHINING**

Ⓑ FIGURE 1 WAS TABULAR DATA

SEE ENGINEERING STANDARDS APPROVAL RECORD FOR ORIGINAL SIGNATURES AND CHANGE SUMMARY

DRAWN	J. ROTSCH	MACHINING	DS 121
CHECKED	S. SCHRATTNER		
STRESSED	E. CROMIE		SHEET: 2
APPROVED	S. SCHRATTNER		

REV: A - 27 DEC 95 | REV: B - 12 MAR 99 | REV: | REV: | REV: | REV:



Ⓑ **FIGURE 2: MAXIMUM CUTTER DEPTH/CUTTER DIAMETER - HIGH SPEED MACHINING**

3.0 CORNER AND FILLET RADII

3.1 Inside Corner Radii - Conventional CNC Machining

The inside corner radius is created by the side of the cutter. For conventional CNC machining specify an inside corner radius 0.02 - 0.03” greater than the cutter radius to ensure continuous loads in order to avoid whiplash, under cuts, and rope twist finishes. See Table 1 for the range of radii that may be specified on the drawing. See Figure 3.

Ⓑ FIGURE 2 WAS TABULAR DATA

3.2 **Inside Corner Radii - High Speed Machining**

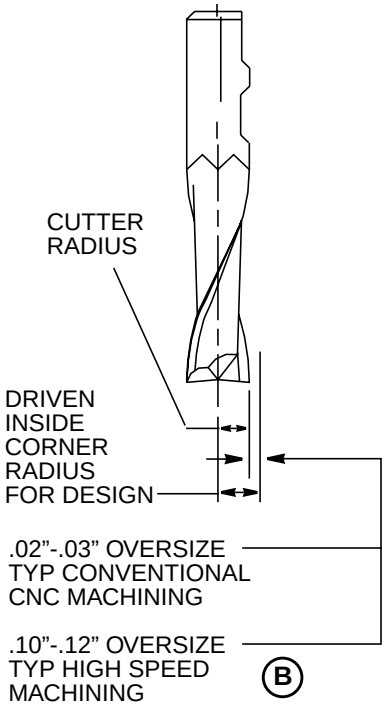
For high speed machining, greater allowance is made but this is achieved by simply using a smaller diameter cutter. See Table 1 for the range of radii that may be specified on the drawing.

3.3 To avoid excessive tool changes use the same radius for all inside corners, one radius for all corner rounding and one radius for all fillet radii on a component. Avoid using more than two (2) decimal places for inside corner, corner chamfering or corner rounding and fillet radii on engineering drawings unless the tighter tolerance is necessary.

TABLE 1

STANDARD CUTTER RADIUS (inches)	MINIMUM DRIVEN INSIDE CORNER RADIUS (to be specified on drawing) (inches)	
	CONVENTIONAL CNC MACHINING	(B) HIGH SPEED MACHINING
.125	.15	.22
.187	.22	.28
.250	.28	.34
.312	.34	.40
.375	.40	.53
.500	.53	.65
.625	.65	.78
.750	.78	1.03
1.000	1.03	1.12

FIGURE 3



(B) HIGH SPEED MACHINING DATA ADDED TO TABLE 1 AND FIGURE 3

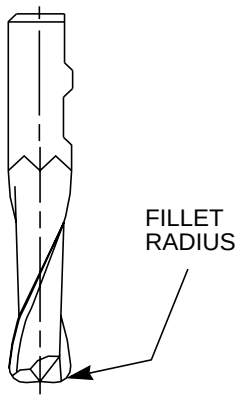
3.4 **Fillet Radii**

The fillet radius is created by the end of the cutter. See Table 2 and Figure 4. Specify a fillet radius on the drawing selected from Table 3. This applies equally to conventional CNC and high speed machining

TABLE 2

FILLET RADIUS (to be specified on drawing) (inches)
.03
.06
.09
.12
.19
.25
.38
.50

FIGURE 4



3.5 **Corner Chamfering**

Corner chamfering is preferred over corner rounding (see 3.6). The standard angle is 45°. Chamfer dimensions shall be 2 decimal places only.

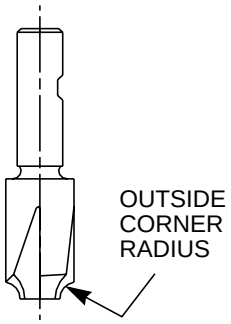
3.6 **Standard Corner Rounding Radii**

Avoid outside corner radii, instead use corner chamfer (see 3.5) or specify "BREAK SHARP EDGES", especially for radii less than .13". Where corner rounding is necessary, specify a radius selected from Table 3. See also Figure 5. The radii apply equally to conventional CNC and high speed machining.

Ⓑ **TABLE 3**

OUTSIDE CORNER RADIUS (to be specified on drawing) (inches)	
.13	.31
.16	.38
.19	.44
.25	

FIGURE 5



Ⓑ .50 CORNER RAD DELETED FROM TAB;E 3

SEE ENGINEERING STANDARDS APPROVAL RECORD FOR ORIGINAL SIGNATURES AND CHANGE SUMMARY

DRAWN	J. ROTSCH	MACHINING	DS 121
CHECKED	S. SCHRATTNER		
STRESSED	E. CROMIE		SHEET: 5
APPROVED	S. SCHRATTNER		

REV: A - 27 DEC 95 | REV: B - 12 MAR 99 | REV: | REV: | REV: | REV:

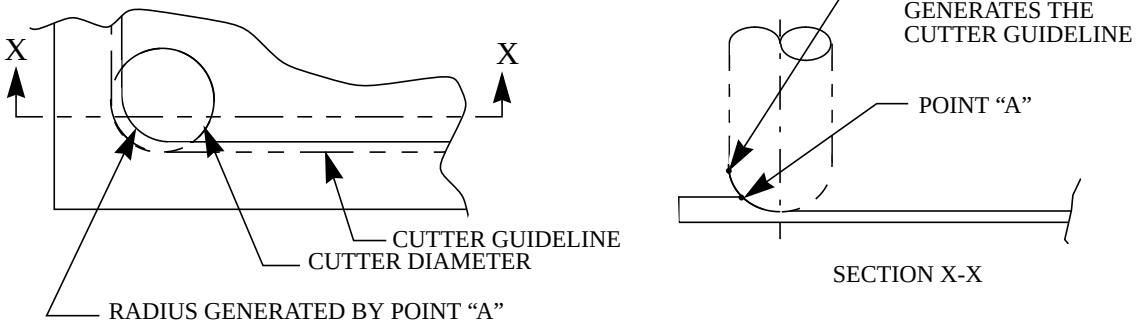
3.7 **Partial Fillet Radii**

When designing a stepped part with a partial fillet radius, the designer should also provide the “cutter guideline” generated by the outside of the cutter. See Figure 6.
NOTE - CATIA Ver. 4.17 will automatically generate the “cutter guideline”.

(B)

FIGURE 6

EXAMPLE OF GENERATED RADIUS:



4.0 FLOORS AND WEBS

4.1 Floors and Webs - Conventional CNC Machining

The minimum floor and web thicknesses and tolerances are shown below.

Minimum thickness: .045” (up to 20 sq. in.)
.060” (over 20 sq. in.)

Minimum tolerance: ± .010” (preferred)
± .005” (allowed for parts up to 48” in length)

4.2 Floors and Webs - High Speed Machining

The minimum floor and web thicknesses and tolerances are shown below.

Minimum thickness: .020” (may be restricted by area)

Minimum tolerance: ± .005”

(B) PARA 3.7 - NOTE RE CATIA VER 4.17 ADDED

REV: A - 27 DEC 95 | REV: B - 12 MAR 99 | REV: | REV: | REV: | REV:

5.0 WALLS AND FLANGES

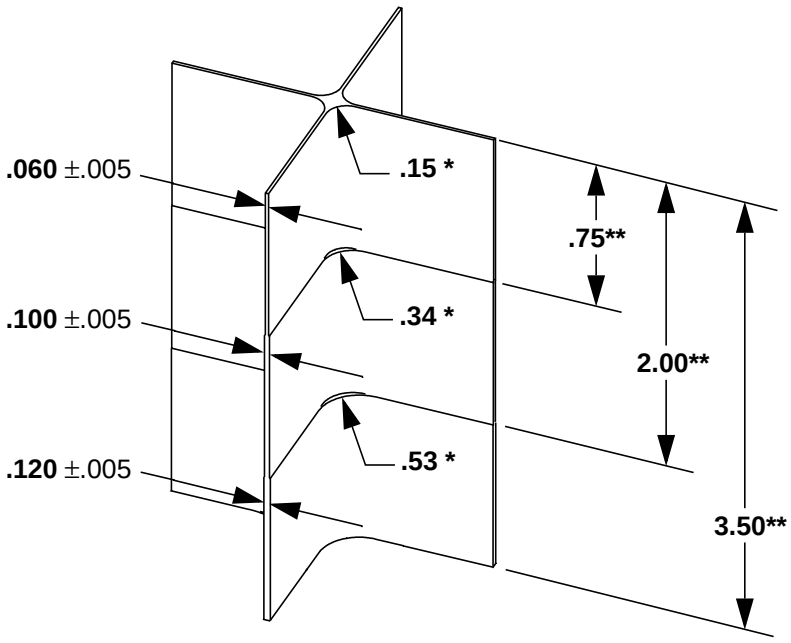
5.1 Walls and Flanges - Conventional CNC Machining

The minimum wall and flange thicknesses and tolerances for various heights are shown in Figure 7. All dimensions are in inches.

(B) Angular tolerance: ± 0° 30' Preferred'
± 0° 15' Minimum

Positional accuracy: ± .010" Preferred
± .005" Minimum

(B) **FIGURE 7 - Conventional CNC Machining**



*** Driven Inside Corner Radius per Table 1**
**** Based on Maximum Cutter Depth for Aluminum per Figure 1**

(B) PARA 5.1 - PREFERRED ANGULAR TOL ADDED
FIGURE 7 REVISED AS PER FIGURE 1AND TABLE 1 DATA

REV: A - 27 DEC 95 | REV: B - 12 MAR 99 | REV: | REV: | REV: | REV:

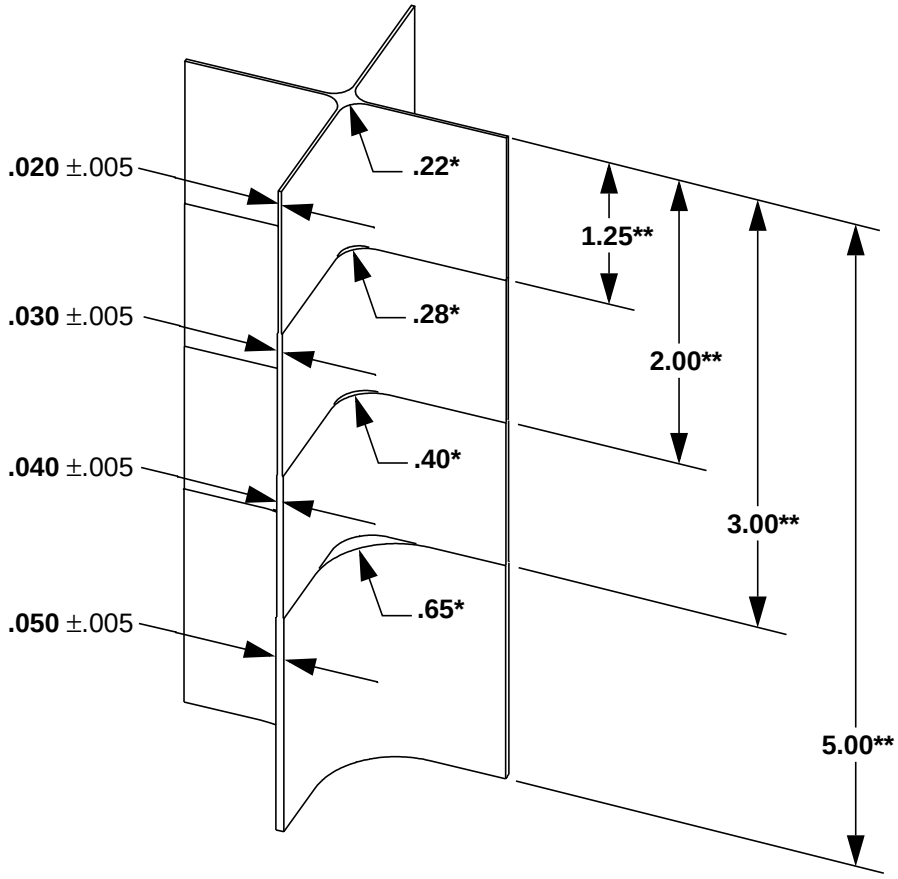
5.2 **Walls and Flanges - High Speed Machining**

The minimum wall and flange thicknesses and tolerances for various heights are shown in Figure 8. All dimensions are in inches.

Minimum angular tolerance: ± 0° 15'

Minimum positional accuracy: ± .005"

Ⓑ **FIGURE 8 - High Speed Machining**



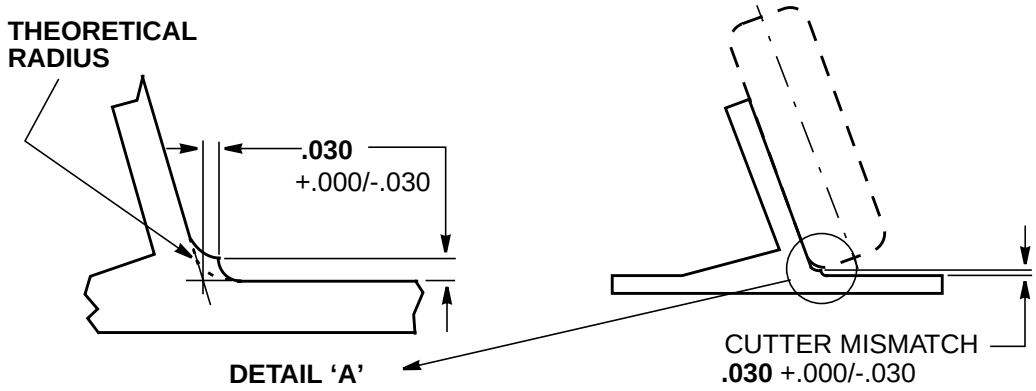
*** Driven Inside Corner Radius per Table 1**
**** Based on Maximum Cutter Depth for Aluminum per Figure 2**

Ⓑ FIGURE 8 REVISED AS PER FIGURE 2 AND TABLE 1 DATA

(B) 6.0 Open M/C Angles

6.1 When designing OPEN M/C angles, the designer should be aware that a cutter mismatch of 0.030" +.000/-.030 must be allowed at the heel of the angle to accomodate the intersection of the machine cutter paths, to avoid undercutting of the web during profiling of the standing flange. See Figure 9.

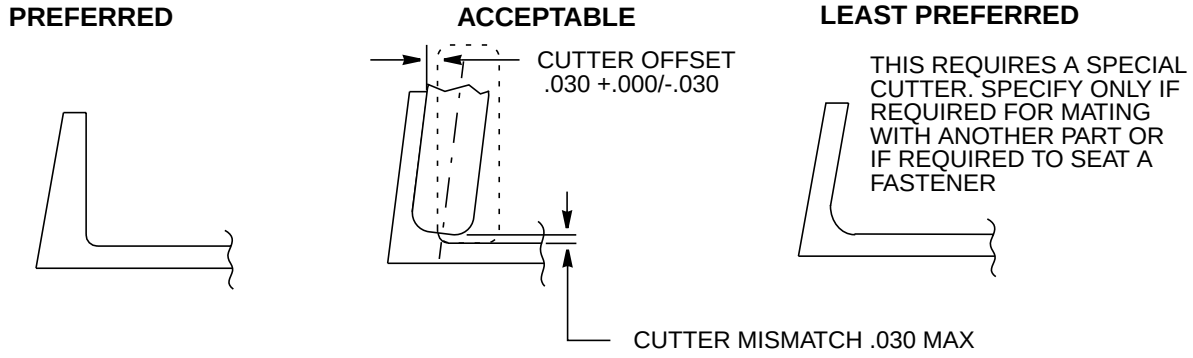
FIGURE 9 - Open Machined Angle



7.0 Closed M/C Angles

7.1 When machining closed angles, it is **preferred** to specify a vertical inner wall. Specifying a sloped inner wall is **acceptable**, but will result in a ridge as shown in Figure 10. Manufacturing will not blend ridges less than .030" (ref. PPS 27.03). If a true radius is required in order to seat a mating part or to seat a fastener, Engineering must specify this in a note on the drawing. This is the **least preferred** option since it requires an additional pass with a ball end mill or bell cutter. Consult NC Methods for specific information regarding special form cutters.

(B) FIGURE 10 - Closed Machined Angles



(B) SECT 6.0 - OPEN M/C ANGLES ADDED
FIGURE 10 - CLOSED M/C ANGLES CLARIFIED

SEE ENGINEERING STANDARDS APPROVAL RECORD FOR ORIGINAL SIGNATURES AND CHANGE SUMMARY

DRAWN	J. ROTSCH	MACHINING	DS 121
CHECKED	S. SCHRATTNER		
STRESSED	E. CROMIE		SHEET: 9
APPROVED	S. SCHRATTNER		

REV: A - 27 DEC 95 | REV: B - 12 MAR 99 | REV: | REV: | REV: | REV:

8.0 Miscellaneous

8.1 Design or Stress Engineering must specify the maximum thickness of material (on the drawing or nomenclature) which Manufacturing is permitted to use if the physical properties are reduced with increases in plate or bar thickness and the integrity of the design would be affected by using thicker material.

8.2 For long thin components it may be advantageous to use bar stock in lieu of plate (cut into strips) where bar stock of the appropriate size is available. On the other hand, Methods may prefer plate material where it is advantageous to make multiple parts per billet. Consult NC Methods for guidance.

8.3 Hole diameters shall be shown on the Engineering Drawing as follows:

Bolts	Specify hole diameter (incl. tolerance) as per DS 126
Solid Rivets	Specify fastener diameter dash number in rivet symbol only, do not specify hole diameter (Production uses hole diameter specified in PPS)
Blind Rivets	Specify fastener diameter dash number, do not specify hole diameter (Production uses hole diameter specified in PPS)
Two Piece Permanent Fasteners	Specify fastener diameter dash number, do not specify hole diameter (Production uses hole diameter specified in PPS)
Tooling Holes	Specify tooling, index and part location holes as specified in the Drafting Room Manual (DRM)

8.4 The drawing must include one of the following notes regarding the grain direction:

Grain Direction <-------> Tolerance $\pm 5^{\circ}$

- or -

Grain Direction Optional

8.5 Consult NC Methods for additional information on unique or special conditions, or when tighter tolerances are required.