Bending Sheet Metal

RAINBOW Aviation Services™

We are often surprised by the number of aircraft builders who seem to be intimidated by the process of calculating bend allowance, setback, and simply creating a flat layout for bending a simple part. In this article we are going to back up a bit, and provide some of the theory necessary to understanding how we go about the process of converting a flat piece of sheet metal into a complex sheet metal component. Learning to accurately layout and bend sheet-metal is a very useful exercise. Once you have mastered the process, you will find that it not only saves a great deal of time, but also can save you a great deal of wasted material.

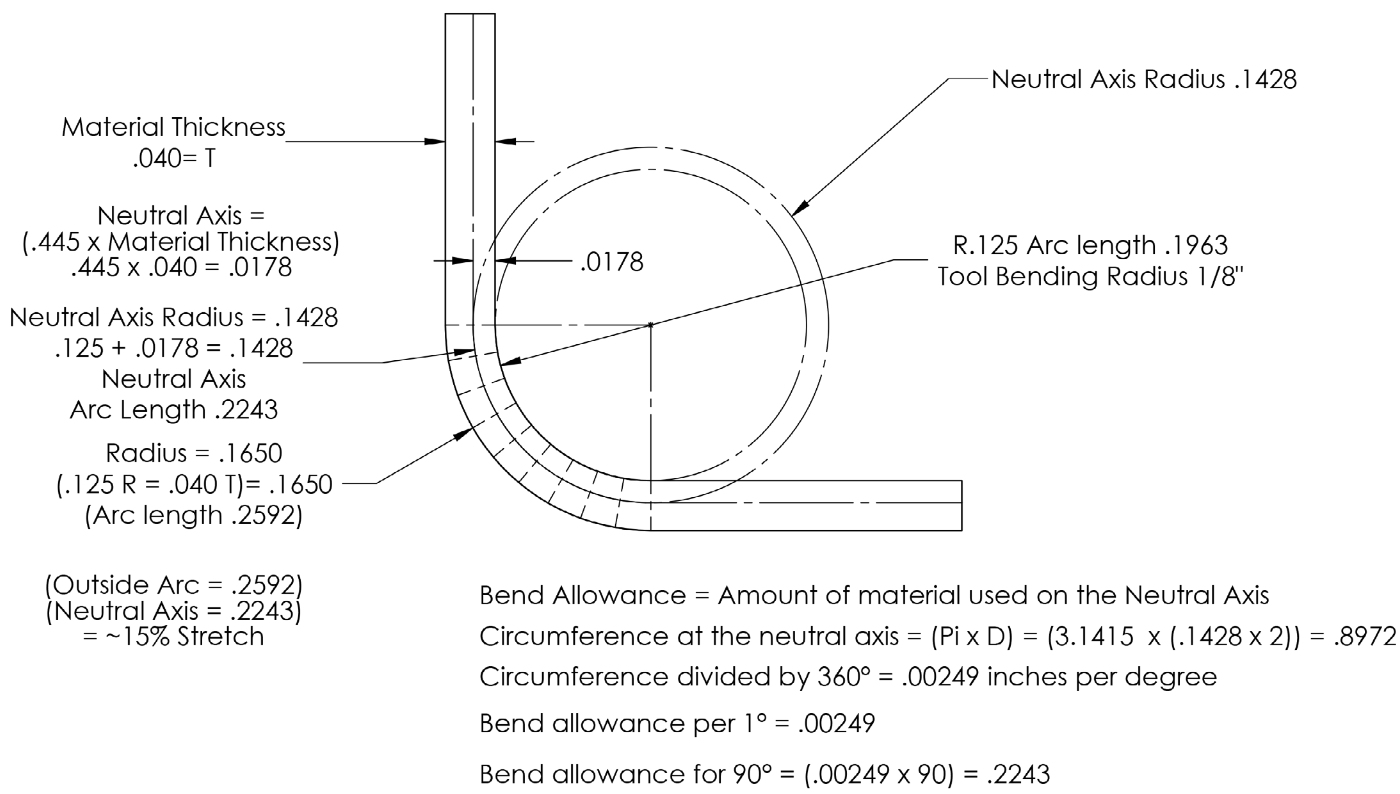

Figure: 1 Bending Aluminum

To start with, let's examine some of the properties of aluminum sheet metal used in aircraft. The two most common alloys of aluminum used in the experimental aircraft world are, 6061 T6, and 2024 T-3. 6061 is one of the least expensive and most versatile of the heat-treatable aluminum alloys. 6061 T6 has a tensile strength of approximately 40,000 psi. It has good corrosion resistance in comparison to 2024 T-3. And current cost per square foot of .040" is $2.53. On the other hand, 2024 is one of the best known of the high strength aluminum alloys. With its high strength, about 50,000 psi tensile strength in the T-3 condition, it is used on structures and parts where good strength-to-weight ratio is desired. Since corrosion resistance is relatively low, 2024 is commonly used in clad form ("Alclad") with a thin surface layer of high purity aluminum. The cost of 2024 T-3 is about 50% more than 6061 T6 aluminum, costing about $3.94 per square foot for .040" sheet.

Minimum Bend Radius		
Material Thickness	6061-T6	2024-T3
0.020	1/16	1/16
0.025	1/16	1/16
0.032	1/16	3/32
0.040	3/32	3/32
0.050	3/32	1/8
0.063	1/8	5/32
0.080	3/16	1/4

Figure: 2 Sample Bend Radius Chart

Understanding the properties of each of the aluminum alloys becomes very important during the sheet-metal layout and bending process. In particular, the malleability and ductility. By definition, ductility is a solid material's ability to deform under tensile stress. And malleability is a material's ability to deform under compressive stress. The ductility of 2024 T-3 is about 18%. When bending aluminum around a radius, we can see that we are both stretching one side of the aluminum (ductility), and compressing the other side of the aluminum, (malleability). (Figure: 1). Extensive testing has shown that the "neutral axis" (neither, under compression or tension) during the bending process is about .445 times the thickness of the material. The smaller the radius that the metal is bent around, the greater the differential between the neutral axis and the outside arc of the skin. Additionally, the greater the thickness of material, the greater the differential between the neutral axis and the outside arc of the skin. Stretching the outer skin beyond its limits will normally result in cracking. Of course, there isn't a necessity for calculating the minimum bend radius because most sheet-metal manuals, including AC 43.13-1B, have a minimum bend radius chart available for quick reference. (Figure: 2)

The tool, which we use for bending sheet-metal, is called a "brake". A sheet-metal

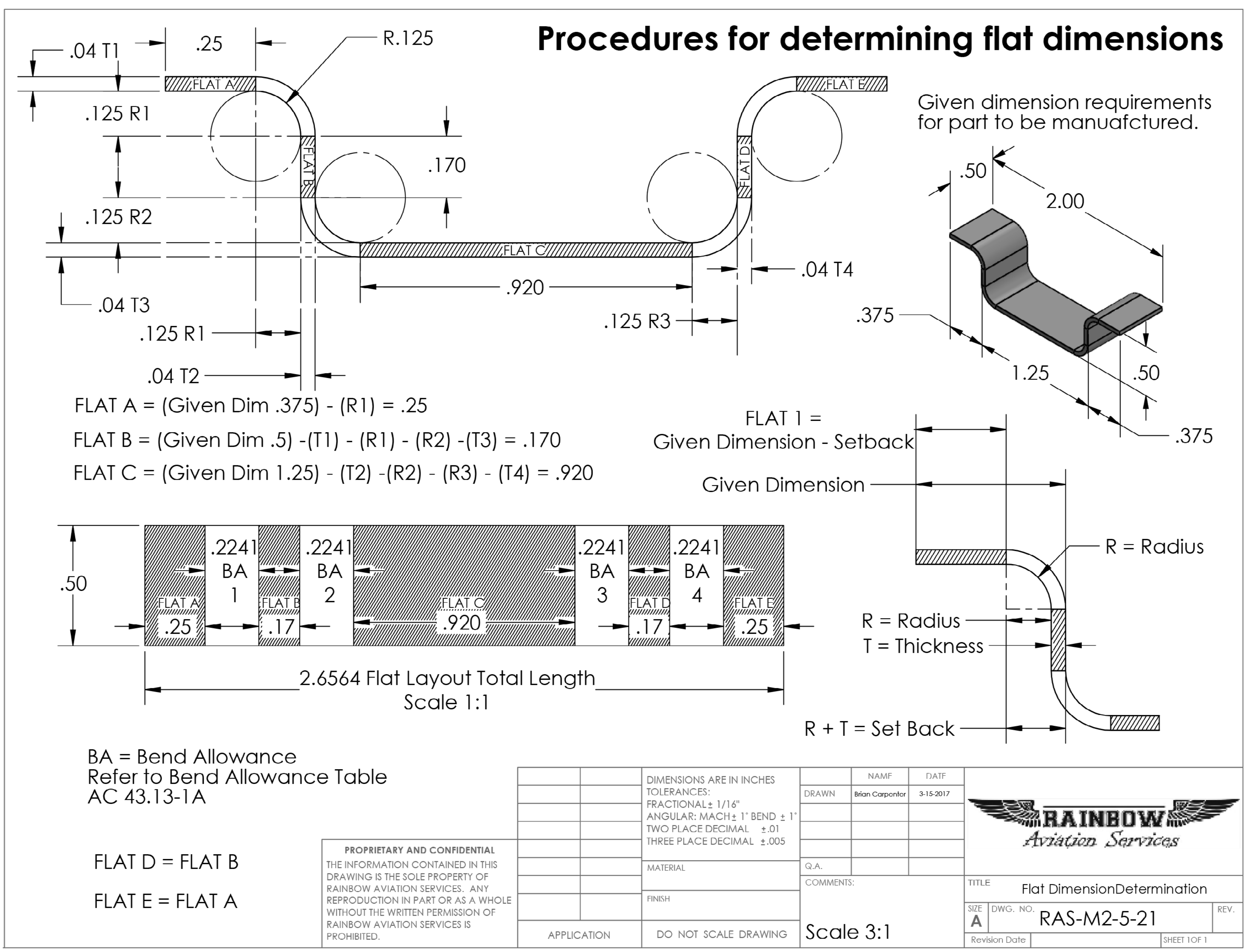

Figure: 3 Procedures For Determining Flat Layout brake used for aircraft aluminum has either fixed or interchangeable jaws with a very specific radius built into the jaws. In our shop, we use a 1/8 inch radius which allows us to bend up to .063" 6061 T6 aluminum. This allows us the ability to bend the majority of sheet metal sizes used in small experimental aircraft. Understanding the necessity for utilizing a radius during the bending process will, now, help us to understand how to calculate bend allowance. Bend allowance is nothing more than the amount of material that is used for the bent portion of the sheet-metal.

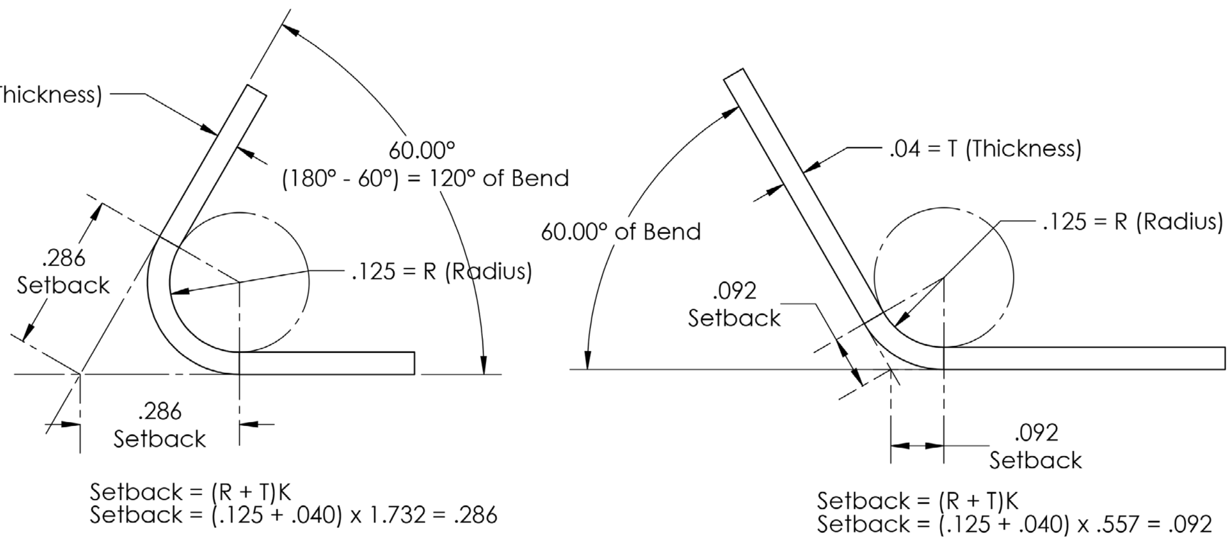

Figure: 4 *Calculating Setback for Acute and Obtuse Angles*

The radius of the bend at the neutral axis is the tooling radius + (.445 times the thickness of the sheet metal). Multiplying the radius times 2 will give us the diameter, and multiply that times pi (3.1415) will give us the circumference. Taking the circumference and dividing by 360° will leave us with a dimension per 1° degree of bend. Multiplying that times 90° will give us the bend allowance for a 90° bend. (Figure: 1) Although the process of calculating bend allowance is relatively simple, it's made even easier by the use of a bend allowance table. A bend allowance table has a matrix of the most common sheet-metal sizes and the standard bending radii already calculated for both a 1° bend as well as the most common, 90° bend. A bend allowance table can be found in Advisory Circular 43.13-1B. When we prepare a piece of metal for bending, we are doing what we call a flat layout. All sheet metal components are simply a series of flat sections and bends. Prior to bending up a sheet-metal part we get out a piece of scratch paper and simply formulate a layout similar to what we see in (Figure: 3). We will lay out each flat section with the bend allowance required for each of the bends. In this case, because the bends are all 90°, the material thickness is the same, and the radius for each of the bends are also the same, we only need to calculate or look up bend allowance once. The amount of material (bend allowance) used for each of the bends is identical. Next we simply

Deg.	K	Deg.	K	Deg.	K	Deg.	K	Deg:	K
1	0.0087	37	0.3346	73	0.7399	109	1.401	145	3.171
2	0.0174	38	0.3443	74	0.7535	110	1,428	146	3.270
3	0.0261	39	0.3541	75	0.7673	111	1.455	147	3.375
4	0.0349	40	0.3639	76	0.7812	112	1.482	148	3.487
5	0.0436	41	0.3738	77	0.7954	113	1.510	149	3.605
6	0.0524	42	0.3838	78	0.8097	114	1.539	150	3.732
7	0.0611	43	0.3939	79	0.8243	115	1.569	151	3.866
8	0.0699	44	0.4040	80	0.8391	116	1.600	152	4.010
9	0.0787	45	0.4142	81	0.8540	117	1.631	153	4.165
10	0.0874	46	0.4244	82	0.8692	118	1.664	154	4.331
11	0.0963	47	0.4348	83	0.8847	119	1.697	155	4.510
12	0.1051	48	0.4452	84	0.9004	120	1.732	156	4.704
13	0.1139	49	0.4557	85	0.9163	121	1.767	157	4.915
14	0.1228	50	0.4663	86	0.9324	122	1.804	158	5.144
15	0.1316	51	0.4769	87	0.9489	123	1.841	159	5.399
16	0.1405	52	0.4877	88	0.9656	124	1.880	160	5.671
17	0.1494	53	0.4985	89	0.9827	125	1.921	161	5.975
18	0.1583	54	0.5095	90	1.000	126	1.962	162	6.313
19	0.1673	55	0.5205	91	1.017	127	2.005	163	6.691
20	0.1763	56	0.5317	92	1.035	128	2.050	164	7.115
21	0.1853	57	0.5429	93	1.053	129	2.096	165	7.595
22	0.1943	58	0.5543	94	1.072	130	2.144	166	8.144
23	0.2034	59	0.5657	95	1.091	131	2.194	167	8.776
24	0.2125	60	0.5773	96	1.110	132	2.246	168	9.514
25	0.2216	61	0.5890	97	1-130	133	2.299	169	10.38
26	0.2308	62	0.6008	98	1.150	134	2.355	170	11.43
27	0.2400	63	0.6128	99	1.170	135	2.414	171	12.70
28	0.2493	64	0.6248	100	1.191	136	2.475	172	14.30
29	0.2586	65	0.6370	101	1.213	137	2.538	173	16.35
30	0.2679	66	0.6494	102	1.234	138	2.605	174	19.08
31	0.2773	67	0.6618	103	1.257	139	2.674	175	22.90
32	0.2867	68	0.6745	104	1.279	140	2.747	176	26.63
33	0.2962	69	0.6872	105	1.303	141	2.823	177	38.18
34	0.3057	70	0.7002	106	1.327	142	2.904	178	57.29
35	0.3153	71	0.7132	107	1.351	143	2.988	179	114.59

Figure: 5 *K Factor Chart for Determining Setback* need to calculate the length of each one of the flat sections. The normal formula for calculating the flat section is given dimension, minus setback. “Setback” by definition is the radius plus the thickness used during the bend. If all given dimensions were given from the outside of the material to the end of the flat section this formula would work great. However there are many cases where you’re going to have to extrapolate on this formula in order to calculate the flat section length. For example, in order to calculate the length of FLAT A, the given dimension is from the inside of the bend. In this case FLAT A = (given dimension .375) - (bend radius .125) = .25. In order to keep comprehension to a higher level, we normally start by teaching bend allowance as we have shown here, with all bends being conducted at 90°. Once we have mastered the process calculating for 90° bends, we can now venture into the calculations necessary for bends that are more acute or obtuse. (Figure: 4) We still use setback which is radius + thickness however this time we multiply a K factor. A K factor chart is available in Advisory Circular 43.13-1B (Figure: 5). This is simply another complex mathematical calculation distilled into a matrix which correlates to the correction factor to the angle of the bend. To calculate the length of each flat, use the same procedures as we used in calculating for a 90° bend. Simply take the given dimension and subtract the setback. When calculating the bend allowance for the bends that are other than 90° simply multiply the bend allowance for 1° times the number of degrees that the metal is bent. This is the same number of degrees used when calculating setback utilizing K factor.

You may have become very proficient at bending aluminum using the old standby method where you start with an extra-large sheet, bend it to the appropriate angle, then cut off the excess material to come up with your final dimension. There’s nothing particularly wrong with utilizing this method, however if you have more than one bend you’re going to be in big trouble. This is where I see individuals getting fairly creative-by guessing at the dimensions, bending the metal, and re-measuring to see how far off they are. Then changing their original dimensions by the amount of error in the original part and re-bending a new piece. After about three or four tries, they can typically get pretty close. But as you might imagine, this can be quite time-consuming, expensive, and frustrating. If you find yourself working on aluminum aircraft on a regular basis, the amount of effort required to learn to do sheet-metal layout is really quite minimal. Once you’ve practiced a bit, you can develop confidence and accuracy worthy of a professional. It’s very rewarding to go through the process of laying out a fairly complex part with multiple bends and have it fit into the aircraft on the first shot. In part 2 of this article, we will address some of the more practical aspects of bending aluminum such as how to place the metal into and set up the sheet-metal “brake”. Establishing a site line. And some other tips and tricks that will get you on your way to becoming a sheet-metal whiz.

Carol and Brian Carpenter are owners of Rainbow Aviation Services in Corning, California. For more Information visit www.rainbowaviation.com