Slide 1

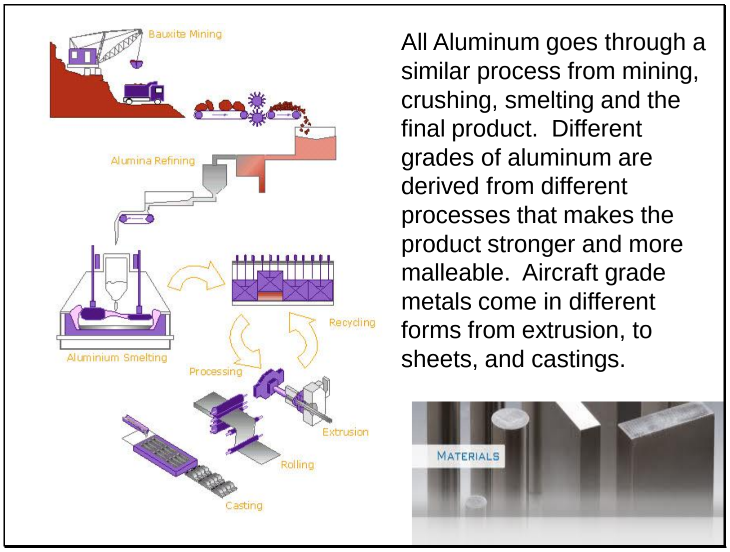
Blue Tuna

Slide 2

BASIC AIRCRAFT SHEET METAL INTRODUCTION

Welcome to Part One of Blue Tuna's Basic Aircraft Sheet Metal Introduction. This course will familiarize the technician with metal types and will assist in correct cutting and layout techniques consistently to specific standards used in aircraft.

Slide 3



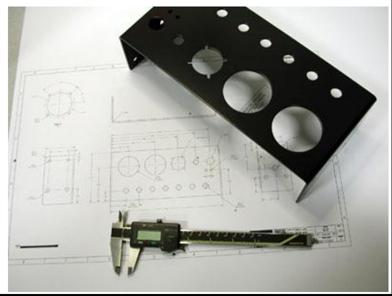
All Aluminum goes through a similar process from mining, crushing, smelting and the final product. Different grades of aluminum are derived from different processes that make the product stronger and more malleable. Aircraft grade metals come in different forms from extrusion, to sheets, and castings.

Slide 4

Receiving Products

All material used in the building of an aircraft must have a material inspection program to receive in parts..

Your Quality Assurance program is built by your company and approved by the FAA or your local authorities.



All material used in the building of an aircraft must have a material inspection program to receive in parts..

Your Quality Assurance program is built by your company and approved by the FAA or your local authorities. Your program includes manuals, processes, and procedures to receive sheet metal parts.

Receiving Inspectors check the Material name and number stamped on the material to the Purchase order (P/O)
The materials Type, Lot Number, and thickness is then recorded on a acceptance type tag (Green Tag) and put into stock

Slide 5

BASIC SHEETMETAL

Aluminum is one of the most widely used metals in modern aircraft construction. It is vital to the aviation industry because of its high strength-to-weight ratio and its comparative ease of fabrication. The outstanding characteristic of aluminum is its lightweight.
Aluminum melts at the comparatively low temperature of 1,250 degrees Fahrenheit.
It is nonmagnetic and is an excellent conductor.
Aluminum alloys, although strong, are easily worked because they are malleable. Most aluminum alloy sheet stock used in aircraft construction range from .016 to .096 inches in thickness; however, some of the larger aircraft use sheet stock, which may be as thick as .356 inches.
The various types of aluminum may be divided into two general calluses: (First) The casting alloys (those suitable for casting in sand, permanent mold, die-castings), and (Second) the wrought alloys (those that may be shaped by rolling, drawing, or forging). Of these two, the wrought alloys are the most widely used in aircraft construction, being used for stringers, bulkheads, skin, rivets, and extruded sections

Popular Types of designations:
- **0** Wrought products only applies to softest temper

•6061 -0
•5052-0
•2024-0

-T3 Solution heat-treated and cold-worked **by flattening**
6061 –T3
•2024-T3

-**T6** Solution heat-treated and annealed **to artificially age**
-6061-T6

The terms “Alclad” and “Pureclad” (which are tradenames) are used to designate sheets that consist of an aluminum alloy core coated with a layer of pure aluminum to a depth of approximately 5 ½ percent on each side.
The pure aluminum coating affords a dual protection for the core, preventing contact with any corrosive agents, and protecting the core electrolytically by preventing any attack caused by scratching or from other abrasions

Slide 6

DEBURRING

Deburring is an important first step Prior to Forming
Deburring makes the edges of metal extremely smooth The higher the temper of the metal the smoother the edge must be to stop stress cracking.

The following are some rules to follow during this step

- Jagged, burred, cut, or cracked edges of metal parts shall be removed by edge smoothing to remove any conditions that might act as stress risers.

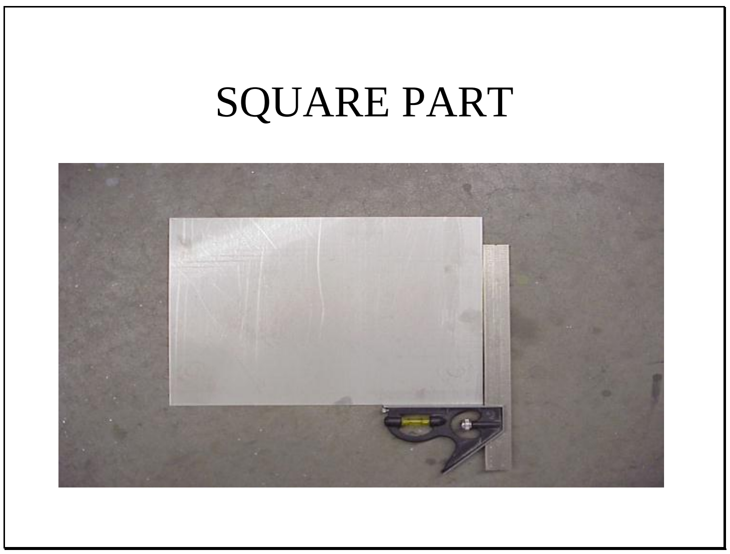
Machining, filing, and wire brushing, sanding or tumbling to limits as specified may perform edge smoothing.

Another rule to follow is - Any edge smoothing or deburring process that results in embedding of steel abrasives into the metal surface shall not be used

.

Blanking, shearing, notching, or nibbling to final or net size is permitted on annealed or normalized parts provided all edges are smoothed to remove all burrs, jagged edges, cracks, and other conditions that might act as stress risers after trimming.

Slide 7



Now we have a part deburred, we must square the metal to build a template (Shop Aid) for other parts to be built from. It’s will become the setup tool for blanking metal and setting the sheet metal break up to build the part.
It’s the Shop Aid tool P/N marked with 9 X’s(XXXXXXXXX)

As a Note: You never scribe on a part that is used on the aircraft

Due to the fact that Scratches and scribe marks create the area that will result in stress risers. (Cracks)

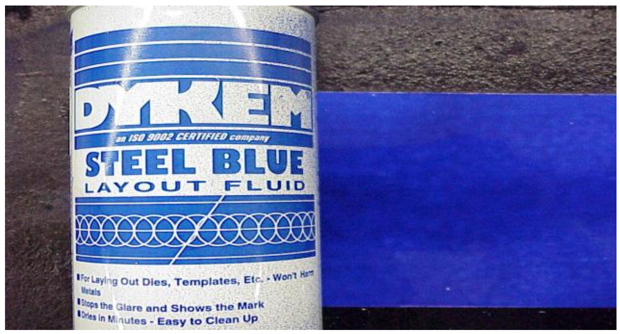
Slide 8

DYKEM

Dykem’s Steel Blue Layout Fluid is a blue spray that is used to see scribe lines on the Master Tool.

Slide 9

DYKEM



The following are the steps to use Dykem,

- Check material to ensure two square sides
- Scotch brite or sand lightly and wipe clean. (This makes, dykem adhere better)
- Using dykem, spray a light coat. (This makes the scribe lines visible)

SCALES Are available in different lengths from 6 inch to 24 inches these are the most common they read in tenths and hundredths.

6-INCH Calipers are used to make precision measurements**.**

Scribes Used to make marks on material for trim lines or bend lines on templates and reference marks

Protractor Used to check degree of angle or layout degree of angle

Beam square. Precision square for checking 90-degree angles**.**

Automatic center punch Makes small indentations on metal for drill starts, center of holes and using a compass to swing an arc

Combination Square Used to make precision 90 degree or 45 degree lines or check material for square.

Compass dividers Used to make circles, equal spaces and parallel lines

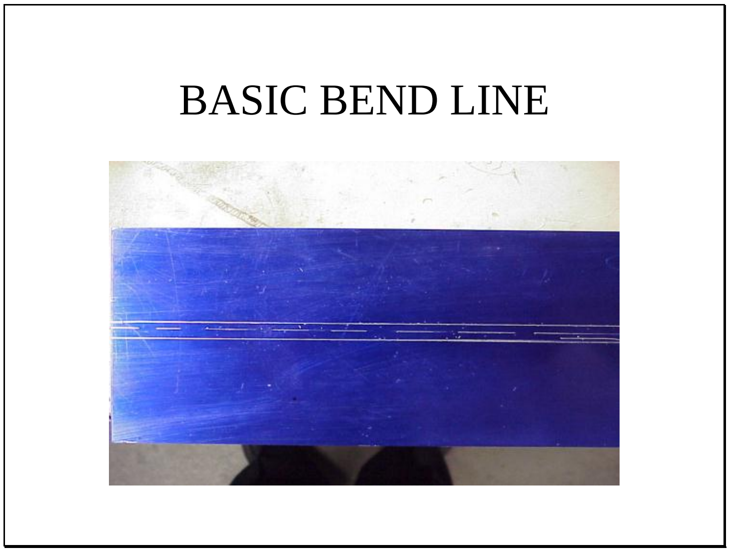
Radius Gages Used to make inside and outside radiuses.

RADIUS GAUGES

Usually the radius of a sharp internal corner shall not be less than .06 inches or greater than .16 inches
The outside corners are radiuses for safety_ to Stop from cutting your arm or hand when reaching into tight places after the part is installed.
Inside and outside Radius's are Specifically stated and otherwise are on the Engineering Drawings .

The minimum corner relief radius shall not be less than the bend radius specified on the drawing for the part, except that the bend radius on each side of the flattened area may be 1/32 inches less than the flange bend radius.

Slide 12

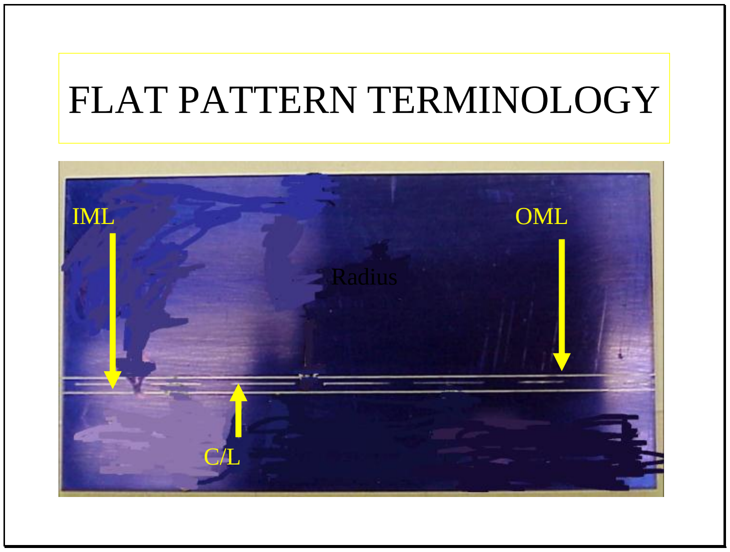


The basic layout is as follows. What you need to know includes, material thickness, degree of bend and bend radius

You will need a Sheet Metal Table of Bend Allowances and setback such as the one by Carl Jants 1951 or several calculators on the market

There are a few free Excel Sheet Metal Table of Bend Allowances on the Web

Slide 13

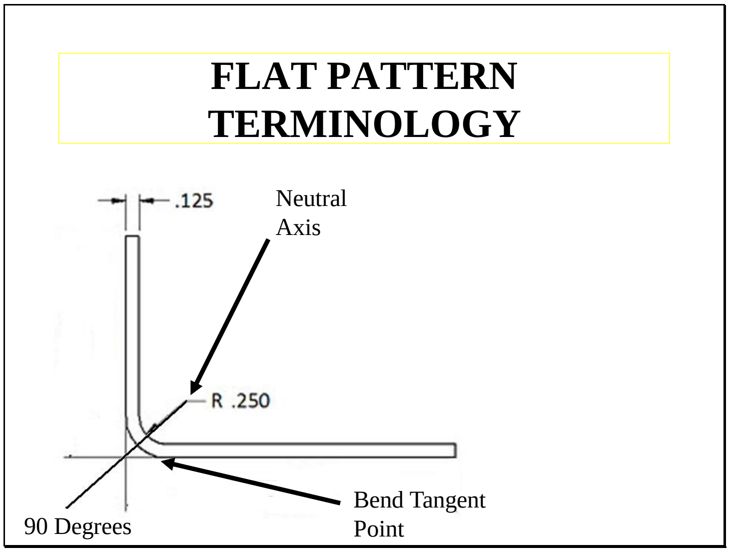


FLAT PATTERN - The outline definition of the part in it's flat state before forming. This can be a physical drawing or template or a computer model. Lets go ahead and include all the fastener locations.

OML = **OUTSIDE MOLD LINE**- The intersection of two outside surfaces of a part extended past the bend radius. This intersection is not actually something that can be measured on a part directly unless the bend angle is 90 degrees. Most drawing dimensions show the formed sheet parts are to the **outside surfaces**

IML = INSIDE MOLD LINE - This is the same as the OML except using the **inside surfaces.**
C/L = Center Line of Bend Radius= BR Radius = R
The center line is used to set the Back stop on mechanical breaks so the break tool center line and the part center line match

Slide 14



BEND TANGENT LINE - The line created where the radius of the bend meets (is tangent to) either flange. There are bend tangent lines for each bend. It is the very top of the radius

NEUTRAL AXIS - The arc through the bend between the bend tangent lines where the metal neither stretches nor compresses.

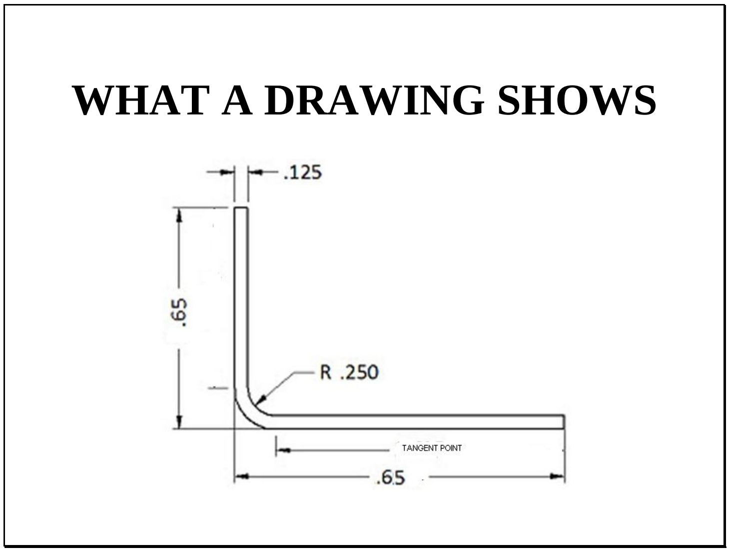
T is The sheet metal thickness.

R is The bend radius. This is always the inside radius, not the outside.

A is The angle of the bend in degrees.

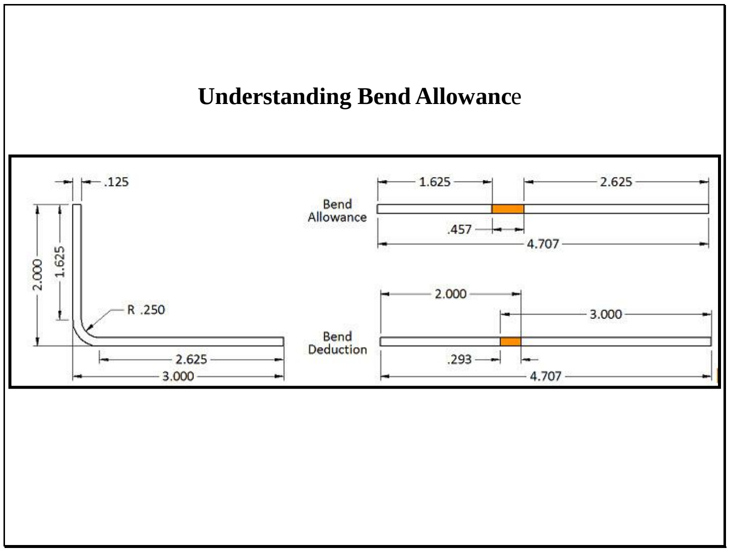
Not too difficult is it? It's just a little tedious. There are sheet metal books available that have charts made up with the numbers calculated for each of the common material thickness and bend radius combinations. The charts are calculated for roughly every 2 degrees of bend.

Slide 15



FLANGE LENGTH - The distance measured from the edge of the flange to the **outside mold line.**

Slide 16

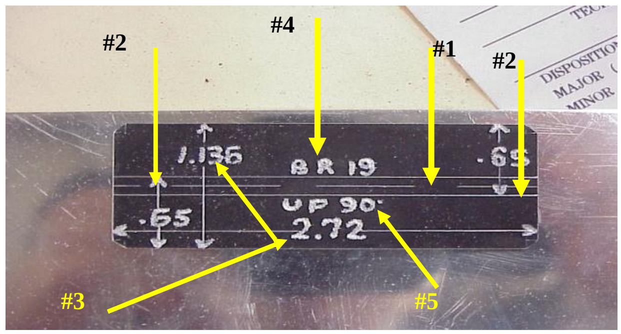


Bend Allowance
Understanding the Bend Allowance and consequently the Bend Deduction of a part is a crucial first step to understanding how sheet metal parts are fabricated. Example _ **.125 R - .032 material is .219 setback**

When the sheet metal is put through the process of bending the metal around the bend is deformed and stretched.
Outside Mold line = the Part surface actually stretches and the Inside Mold Line - Inside Part actually compresses
As this happens you gain a small amount of total length in your part.
Likewise when you are trying to develop a flat pattern you will have to make a deduction from your desired part size to get the correct flat size.
The Bend Allowance is defined as the material you will add to the actual leg lengths of the part in order to develop a flat pattern. The leg lengths are the part of the flange which is outside of the bend radius.
In our example above the part with flange lengths of 2” and 3” with an inside radius of .250” at 90° will have leg lengths of 1.625” and 2.625” respectively. When we calculate the Bend Allowance we find that it equals **.457”.** In order to develop the flat pattern we add .457” to 1.625” and 2.625” to arrive at 4.707”.

Slide 17

SPT



Shown is a finished Sample Part Shop Aid with the Dimensions shown

#1. Center of bend

#2 .65 =the leg length of part after its bent

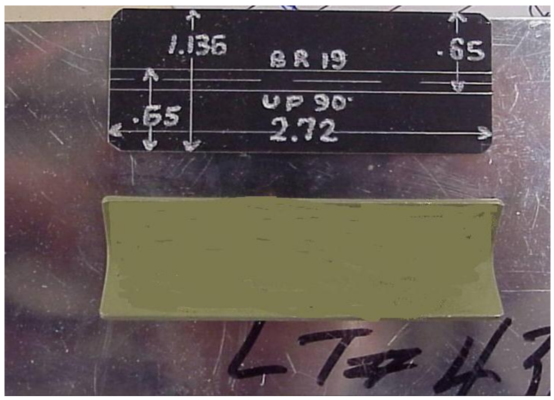
#3 1.136 x 2.72 are the blanking dimensions

#4 BR 19 is the inside Bend Radius after forming on a .190 forming die used on the break

#5 90˚ = what degree the part will be bent to

Slide 18

SPT AND PART



Shop Aid tool with the finished Part