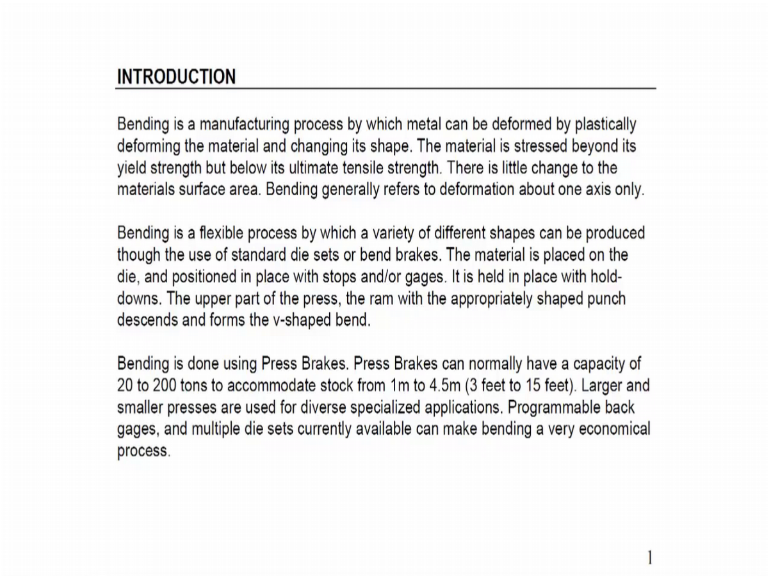
Enclosure Design of Electronics Equipment
Prof. N V Chalapathi Rao
Department of Electronic Systems Engineering
Indian Institute of Science, Bangalore

Lecture - 24
Making a quick model

Hello. So, I am back like that say I hope both of us are back, yesterday I covered a part of that bending allowance from one of the posters on the net with a permission saying yes you can use all these information. Now I will continue with this from another place which also in the same this thing, I suggest has we go along you look at the original.

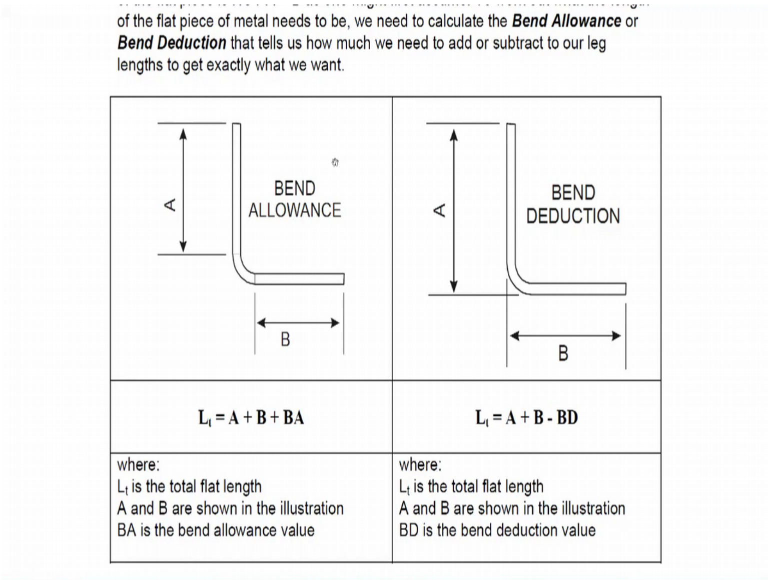
(Refer Slide Time: 00:46)



In case I have skipped something. So, you see here it is a manufacturing process by which metal can be deformed by plastically deformed the material and changing it is shape. Material is stressed beyond it is yield strength, all these I spoke to you yesterday saying we have a yield stress and then afterwards there will be a permanent set that can occur.

So, if you take a normal stress strain diagram, you see there is a small kink in it and then after the plastic deformation is done, eventually before it tastefully below it is ultimate tensile strength, you can continue to make operations. And you can improve life or improve the process by intermediate stages of annealing and let the grain grow back. Well, that is the speciality by itself as part of the material thing it has a bearing on this of course, I cannot say you cannot you do not need it here, but we can concentrate on what we need to do here.

(Refer Slide Time: 01:59)



So, so far the bending allowances which we have talked about are about taking the internal dimensions and adding something to it as a bend allowance. This is what we have bend doing all along. Another way of doing it is saying in some cases we probably have only the external dimensions which are measurable; alternatively only the external dimensions are of interest to us.

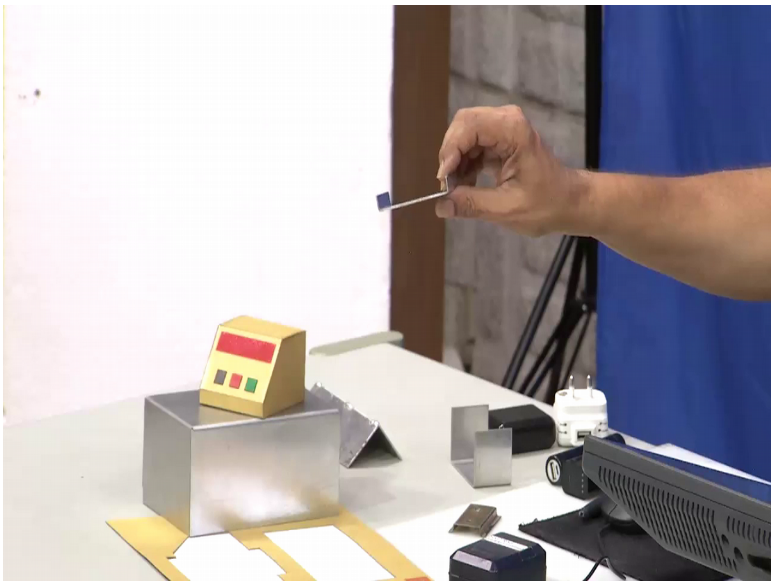
So, please have a look at the sample which I would like to show you here.

(Refer Slide Time: 02:31)

This is what I was talking to about this being a hat section. This particular thing actually is from a box which holds a switch gear switch gear meaning all those various types of things, which use that in seek lamp mounting. So, if you see here you are not that much really interested in the internal dimensions as we are interested in the total height, and then we are interested in the various features, what is the depth here and so on.

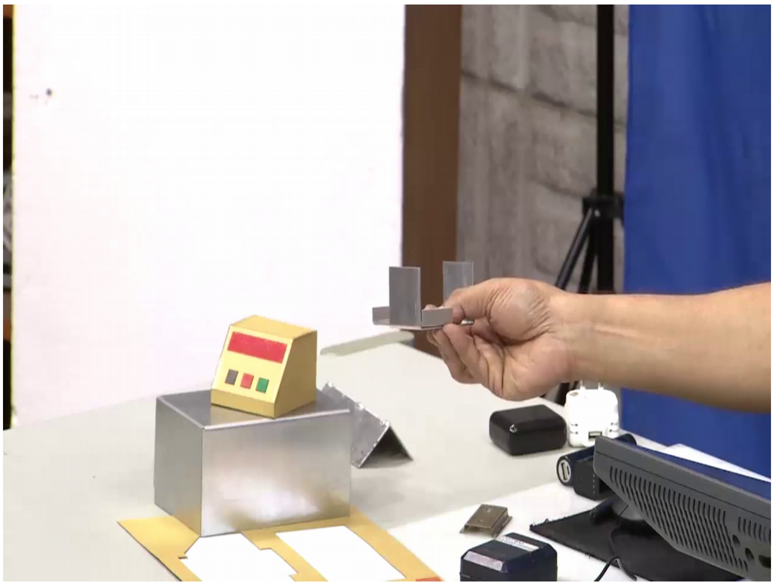
So, in such cases we never ably end up with having to find out the total dimensions outside, and eventually subtract the extra material may come about.

(Refer Slide Time: 03:26)



I will get back again I will give you a special lecture on this, in this as you can see is a small aluminium sample in which I have used earlier to make sure that what I am talking as a practical and same thing.

(Refer Slide Time: 03:42)



And now this lecture today is going to talk to you about imagine there are two sections, in which part of it is like this and then we have a corner.

How does one cover the corners? So, as I go along I will also show you have a small method.

(Refer Slide Time: 04:05)

Actually if you remember I started this in the first lecture saying you learn how to make a card board simulation. This is not a actually a real protect, but then I just wanted to show you that in a very short time is very much possible for us to simulate items like this with too much of bother of having to fabricate a piece and then, what you call end up with not being able to modified.

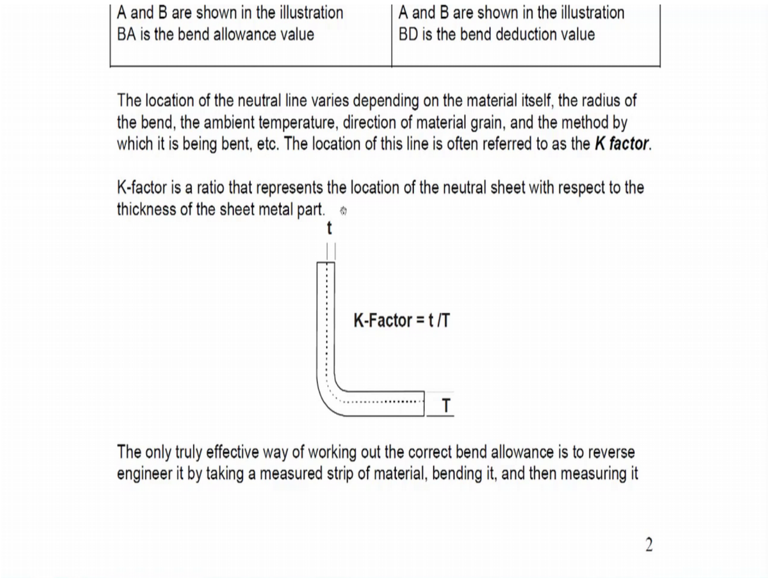
(Refer Slide Time: 04:43)

It is not as serious as this, this one is my hearing aid box in the case this hearing aid box tremendous amount of detail has been worked inside, such that it can be opened well, but even then if you start with these things, things like where do you split the box, (Refer Time: 05:07) know; where you split the box and then how do the top and bottom joint together and then if we have class fear, how does it work and so on.

As I have pointed to earlier a lot of it depends on how will what is the amount of money, we are ready to spend in making the detailing perfect. But it is a simple throw a item; obviously, you do not add too much to it, but if it is a cosmetic item or in my case it is a very very highly functional item, which I need to carry all these things. The tremendous amount of interest and detailing time is spent so that things look. So, if you have to come to cosmetics including powder compacts, ravish and then your various types of sprays and what you will find in a shower lot of the shower gels and all that, a lot of effort goes into making the detailing on the top.

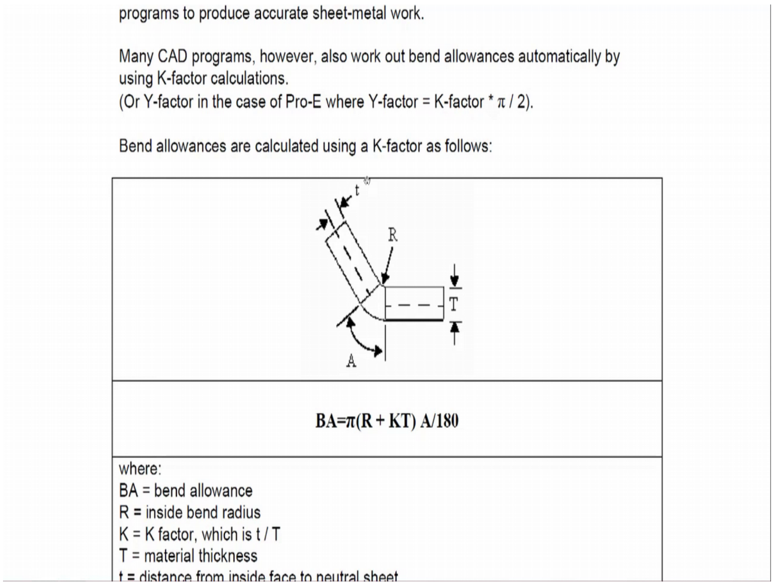
Now, getting back to my exercise

(Refer Slide Time: 06:22)



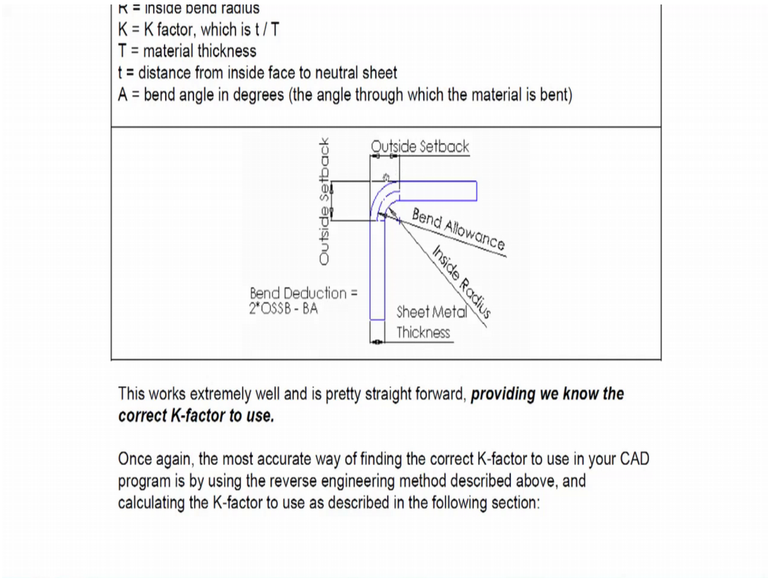
I mean the actual lecture here you will notice that same thing the K factor seems to be uniform is a ratio that represents location of the neutral sheet with respect to the thickness of sheet metal part.

(Refer Slide Time: 06:39)



So using the radian calculations and so on they have come out with lot of these things we had tried to tell you about yesterday.

(Refer Slide Time: 06:51)


R = inside bend radius
K = K factor, which is t / T
T = material thickness
t = distance from inside face to neutral sheet
A = bend angle in degrees (the angle through which the material is bent)

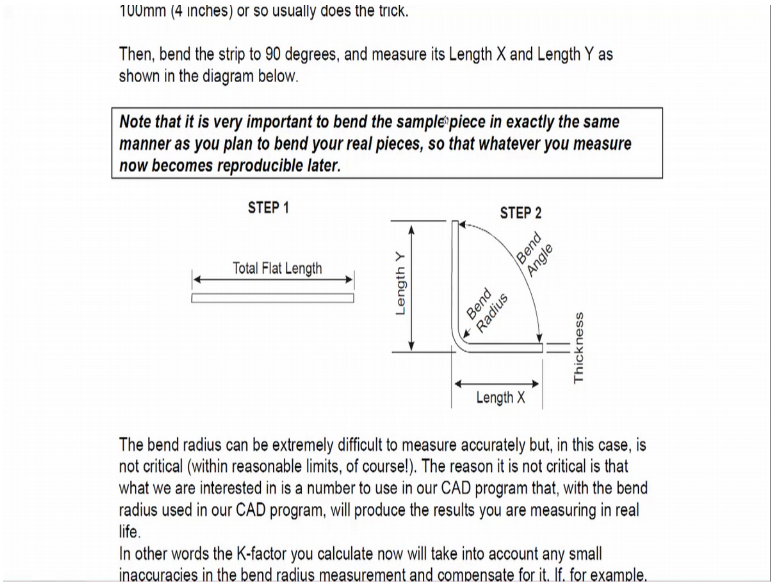
This works extremely well and is pretty straight forward, ***providing we know the correct K-factor to use.***

Once again, the most accurate way of finding the correct K-factor to use in your CAD program is by using the reverse engineering method described above, and calculating the K-factor to use as described in the following section:

When there is this a issue of a setback and bending allowance and so on, extremely well and straight forward providing provided we know the correct K factor to use.

Once again most accurate way of finding the K factor, is by using reverse engineering method described and calculating the K factor use as described later. They talk about reverse engineering it means make a sample bend it and get back. Because it is a physical phenomena the equation does not follow the I mean the equation does not cost phenomena it only describes the phenomena and then how we can use it as we go down.

(Refer Slide Time: 07:34)


100mm (4 inches) or so usually does the trick.

Then, bend the strip to 90 degrees, and measure its Length X and Length Y as shown in the diagram below.

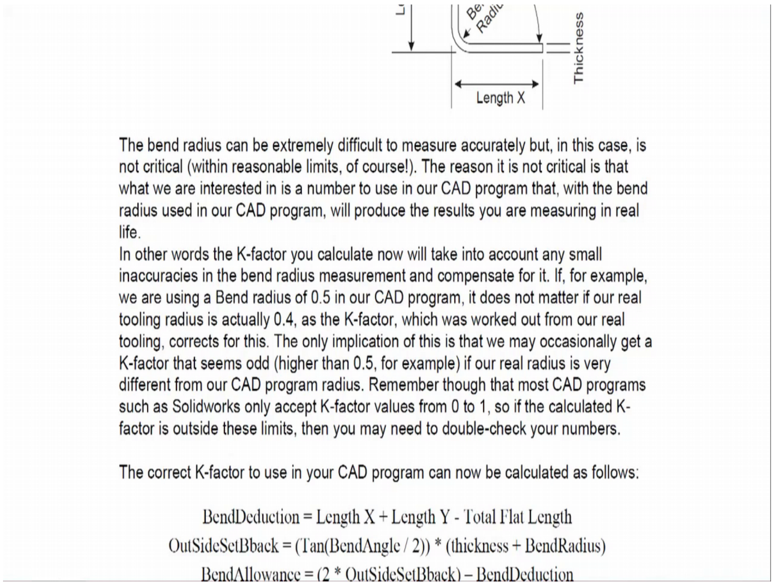
Note that it is very important to bend the sample piece in exactly the same manner as you plan to bend your real pieces, so that whatever you measure now becomes reproducible later.

The bend radius can be extremely difficult to measure accurately but, in this case, is not critical (within reasonable limits, of course!). The reason it is not critical is that what we are interested in is a number to use in our CAD program that, with the bend radius used in our CAD program, will produce the results you are measuring in real life.
In other words the K-factor you calculate now will take into account any small inaccuracies in the bend radius measurement and compensate for it. If, for example,

You see here one more I may have repeated even here know.

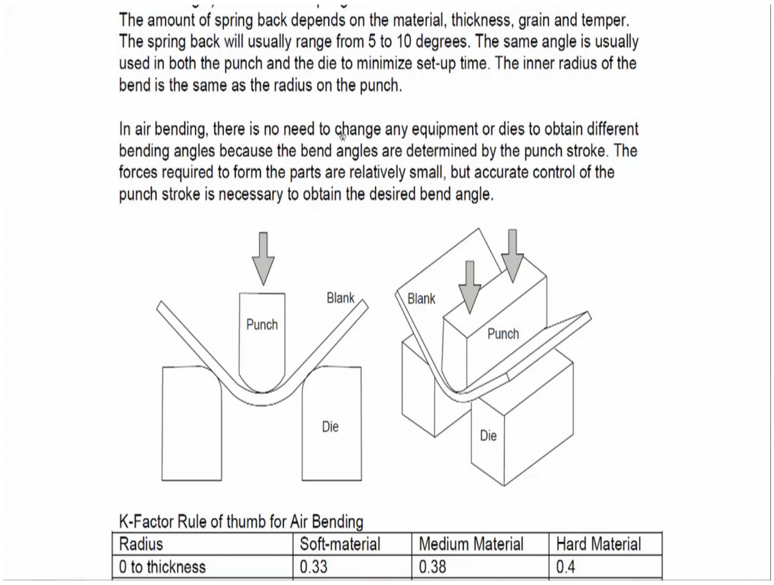
It comes from a probably either it is called t thing bend works, but do not know which program is it, but you see the caution everywhere. It is important to bend the ample piece in the same manner as planned to bend your real pieces so that whenever you measure now becomes reproducible later seen that no total flat length versus what you get here.

(Refer Slide Time: 08:10)



The bend radius can be extremely difficult to measure accurately, in the case of a not critical application we are interested in a number to use in our cad program that the band radius used in our program will produce results you are measuring in your real life. So, if you go down further a lot these things are explained below.

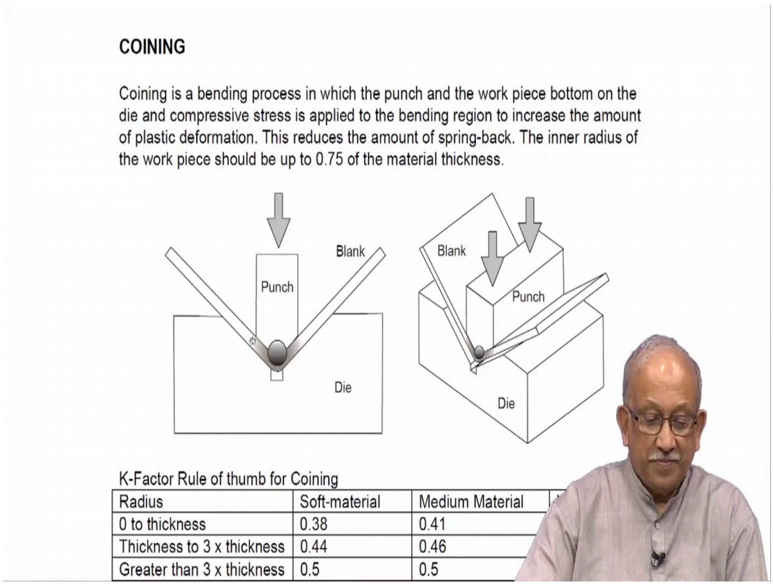
(Refer Slide Time: 08:34)



So, is have told you about it know saying what exactly is a air bending. Air bending there is no need to change any equipment or dies to obtain different bending angles.

What is the done is see this a top and bottom, you have two dies and then you see the radius here and then as it moves inside it comes back.

(Refer Slide Time: 08:57)



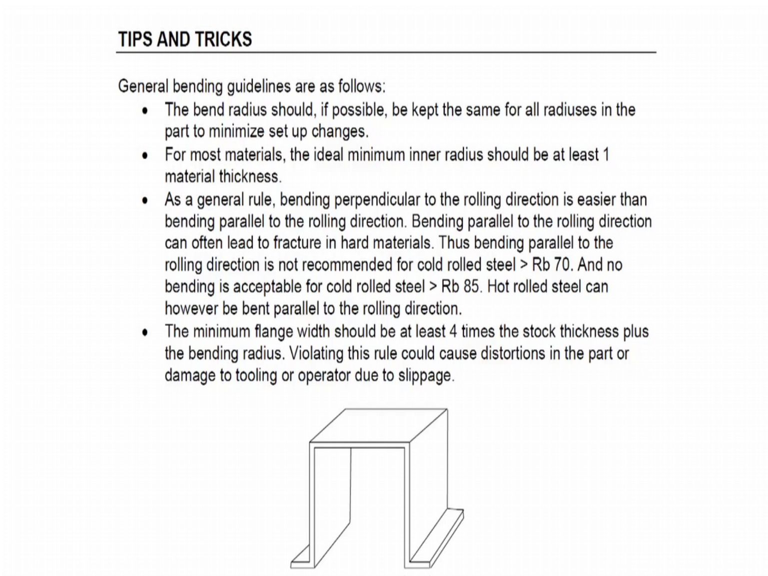
You end up with soft material medium and hard material, what is the K factor. So, as you make the punch move down a little you can get more or less. So, included angle and then something else also is because of these air bending natural flow of the material and especially we use an hydraulic then slowly, it seems to give you consistent better results.

The other option is called bottoming. Bottoming is a bending process where the punch and work piece bottom on the die, this makes for a controlled angle with very little spring back. The tonnage area of this type of press is more than air bending the inner radius of the work piece should be minimum of one material thickness earlier I have told you that can be 0.1, 0.2 and then 3 times and so on.

In this case you know they have given. So, you see here if you carefully see here this punch is you know holding it hard and making it bend, seeing this know you have a die and a punch and then it is pushing all the materials inside. You see here there are still small bits of a gap here. Now going down you really have a bending process in which the punch in the wok piece bottom on the die and compressive stress is applied to binding region, to increase the amount of plastic deformation.

So, you see here a small detail, this detail actually we need not to have in a typical vertical equipment constructor, need not break is a too much about it except to know that a fabricator goes through all these things and then when you make a design put in to consult with them first. So, you see here if you see the except corner here you see here there is a small relief also given here, then there is another some device here it pushes the thing here .

(Refer Slide Time: 11:14)

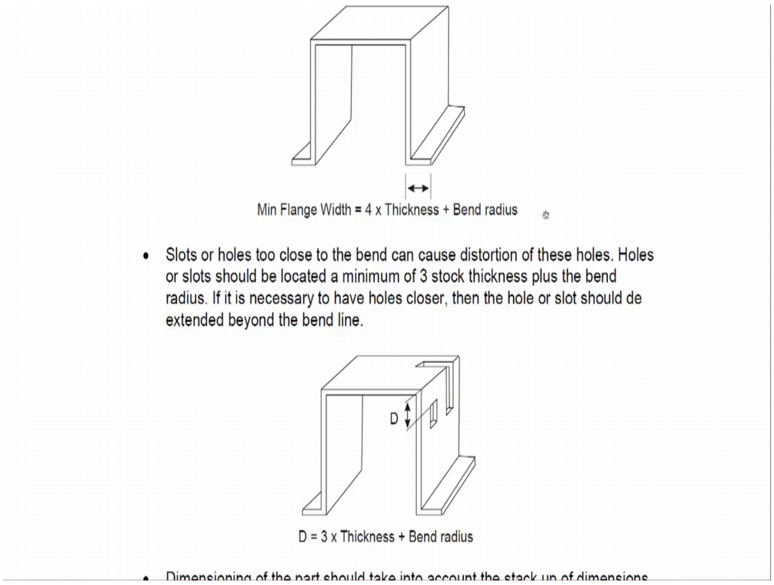


So, I see that various types of these operations are even here bend radius should if possible to be kept at the same for all radiuses is in the part to minimise set up, ideal minimum radius should be at least one, bending perpendicular to the rolling directions easier than bending parallel to the rolling direction. In rolling you mean on the material which comes out of what you call rolling presses, because of a rolling press you have a continuous strip. So, if the strip is available in this direction, it means you can bend it here like this. It comes continuously and then bending it here is like that if you try to bend it in the other direction chances are it get crack.

Now, you see here hot rolled steel can be parallel to the rolling, but in for cold rolled steel we have a little problem about this thing. Minimum flange width should be at least 4 time the stock thickness, plus the bedding violating the rule should cause distortions in the part a tooling or operator due to slippage once again relatively important point.

So, we have here you have seen this, if you try to hold this and use chances are it will pull inside you understood know. If you take the corner and trying to do something chances are this edge will get pulled inside, especially the object is wrong. So, to overcome that I have means generally what they do is make something longer, and then share of the unwanted portion even have it one thickness. So, they will put another one thickness sheet here remaining the share of. So, you have it the original operation is now repeated by three more operations. After the original operation it would take it out measure probably or check with the gauge and then pass it through another notching stage, after notching again you know maybe you have to do dressing and all that you end up with multiple handling of the equipment if you violate this here.

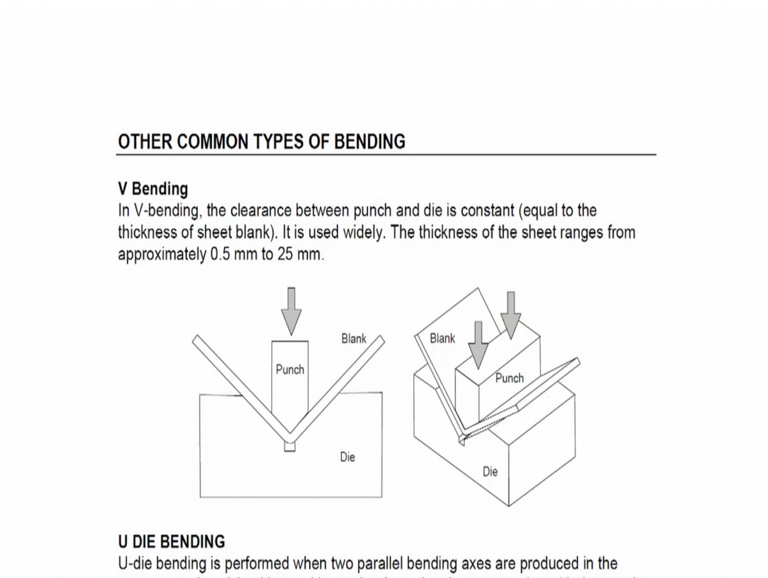
(Refer Slide Time: 13:29)



So, flange width it just a matter of convenience they have given here saying thickens and bend radius and so on.

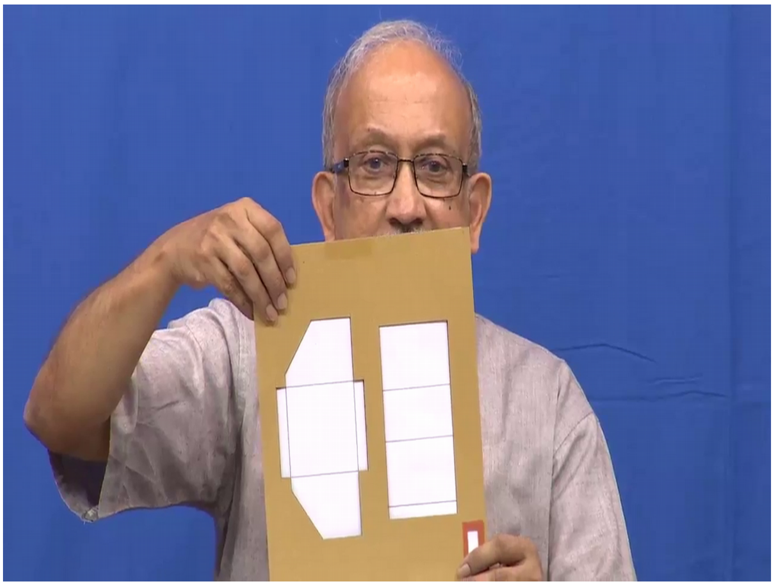
Slots are holes too close to the bend can cause distortion, holes should be located minimum. These are all matters of simple design guide rules, dimensioning the part should take into account the shakeup of dimensions that can happen and mounting holes and we made oblong should be made, and then parts should be inspected and you know in the constraint portion and so on.

(Refer Slide Time: 14:00)



The issue being I think while he was marking it there, you are while the marking was going on there, you were told that cumulative dimensions are used like this you seen this know.

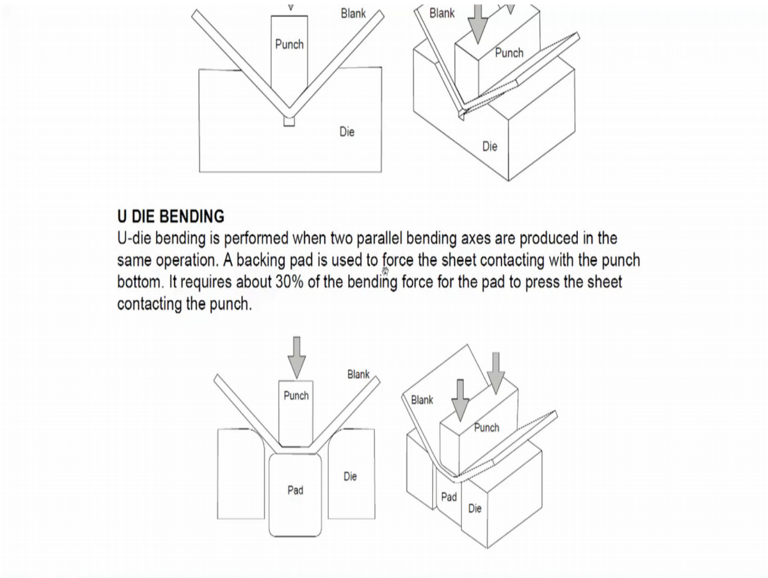
(Refer Slide Time: 14:17)



So, something has been made and then after that starting from a datum line things are added, and then wherever we have this bending a bending allowance has been added, and all marking has been done with respect to a datum line that is such an advantages to this except when it is symmetrical and depending on accumulation of errors. If you see my colleague there, he did it in a slightly convenient way he had marks something from this side turned it over and again marked something from that side.

So, that the symmetry is maintained in our case you know is something becomes little more or less does not really makes such a lot of a difference. And then you can see a small bit of detail here in the corner which I will come back to later, you see here there is a small step and all that I will come back to you later about it. Now coming back to my this notes in V bending, the clearance between punch and die is constant equal to thickness of the sheet blank. It is used widely sheet ranges from 0.5 mm to 25 mm tough really really tough and then you see here.

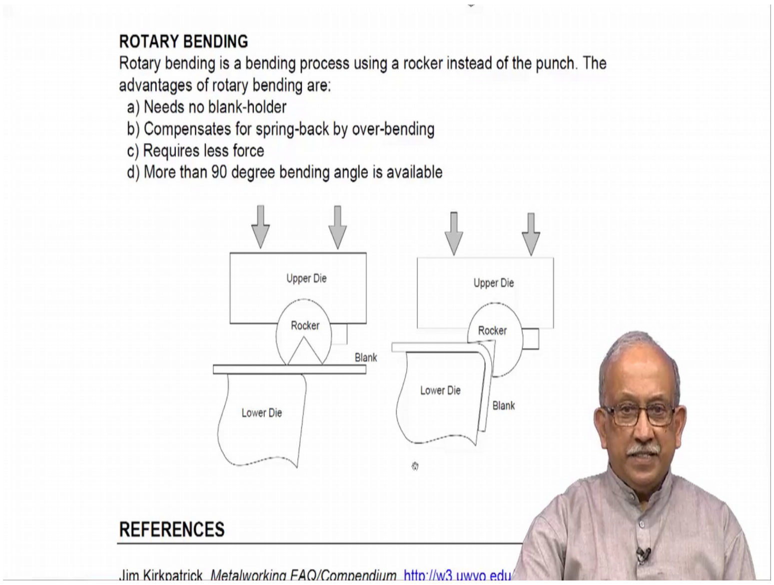
(Refer Slide Time: 15:39)



Things like that hat section which I showed you, U die bending is performed when two parallel bending axes are produced the same operation, a backing pad is used to force the sheet containing with the punch, it requires 30 percent of bending force for the pad to press and so on and so on and so on. Why this is done is if you to have to a cumulate bending and then there may not be may not be possible for us to. Put something else here and then clamp it here, once the u is formed clamping the u again it is a little problem there is no way of continuing the other binding.

So, binds are close together typically this operation is done and sometimes even in the case of the hat section even the other operation can be done here if you put the other material easily will be able to make such things here.

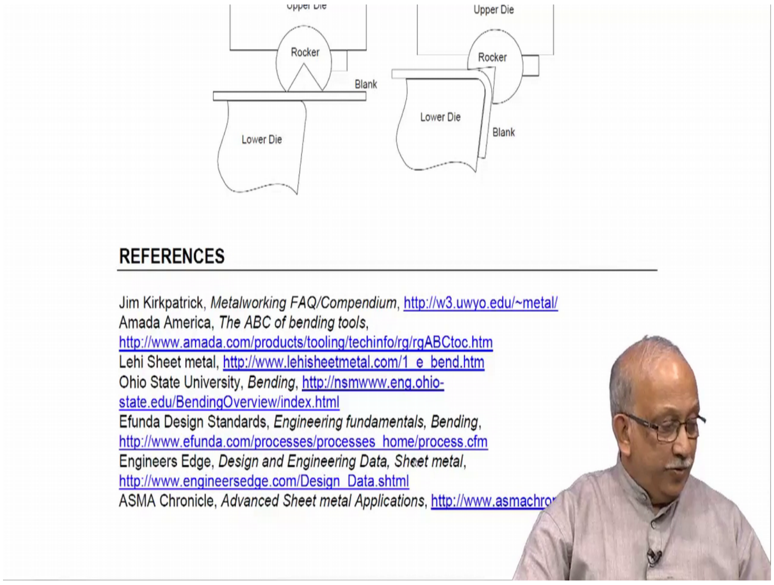
(Refer Slide Time: 16:59)



Then you here slowly were getting into if you have specially type of bedding. Saying why do we need to just follow those things. The blank holder and pad are there flange length can be changed and the bend angle can be controlled by the strop position of the punch, that is the little specialty items, but then maybe it is just worth knowing if you go down, we have stuff here you have seen this know various types of other operations. Double die bending can be seen as two wiping operations acting the work piece one after another.

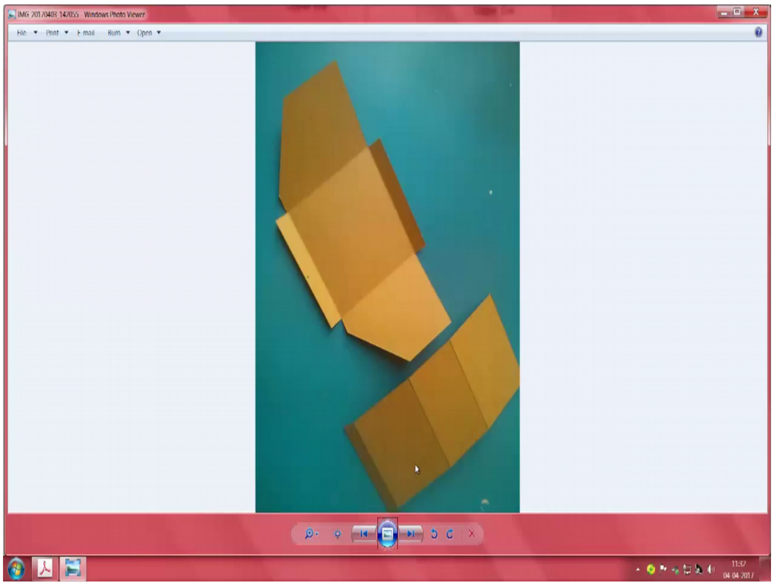
So, he has something here which first makes a contact in those then afterwards more and he does something. Then we have all these beautiful rotary bendings looks good and then a big list of references here.

(Refer Slide Time: 17:30)



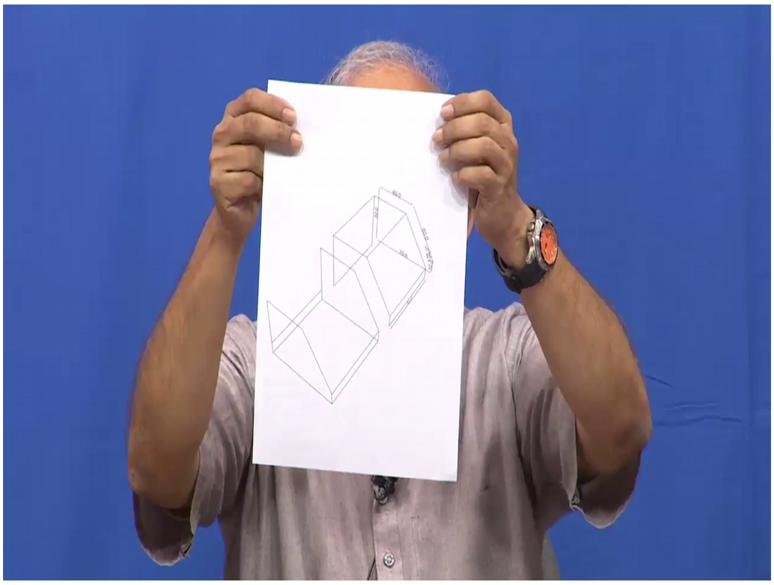
I suggest you to look up these things, now will show you few other things from my nice collection of samples here.

(Refer Slide Time: 17:38)



Yesterday, I wanted to show a sample for you of this item.

(Refer Slide Time: 17:49)



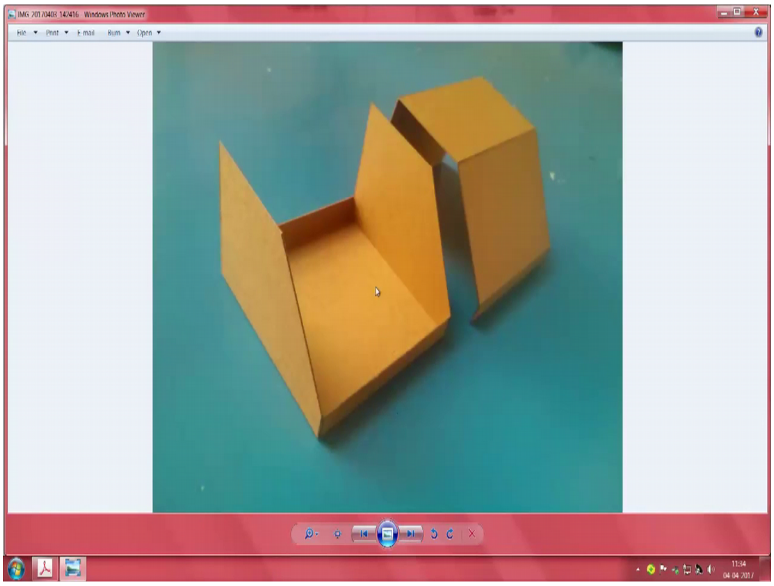
See here if you remember I was holding that small box.

So, try to make a what you call a design like this and in this design oh which will I am bending oh I think I should be this side. We have several nominal dimensions which covers the top here then they become here and so on here, and then somehow I need to make sure that they top and bottom fits here, from the cad convenience we will ignore the material thickness, but we cannot get away with it in reality.

But when we make a cardboard box I will just show you the thing this was given to our same professional who does all these things you see here. After developing the sheet this is the development drawing, is the development drawing as he has done he has happily removed all the materials we want very very carefully. On two counts why it is being shown is that you will understand what it is. Secondly, it is a little like the magic sculptor, somebody asked as sculptor it seem saying how you make all these beautiful art pieces (Refer Time: 19:15) oh nothing I look at the rock and remove all the unwanted pieces, and then remains; what remains is the wanted piece it tough for has to do.

But then if you look at my what you call monitor you will be able to see think that have been done. So, after the sheet has been cut out from there, very carefully a lot of interest you know has been taken to ensure that cuts are taken perfectly, and then this one is the bottom portion this is the top portion, no allowances of any sort has been given just make sure that things fit. Seen that what the picture earlier shows you given directions lightly it better here and now you see

(Refer Slide Time: 20:14)



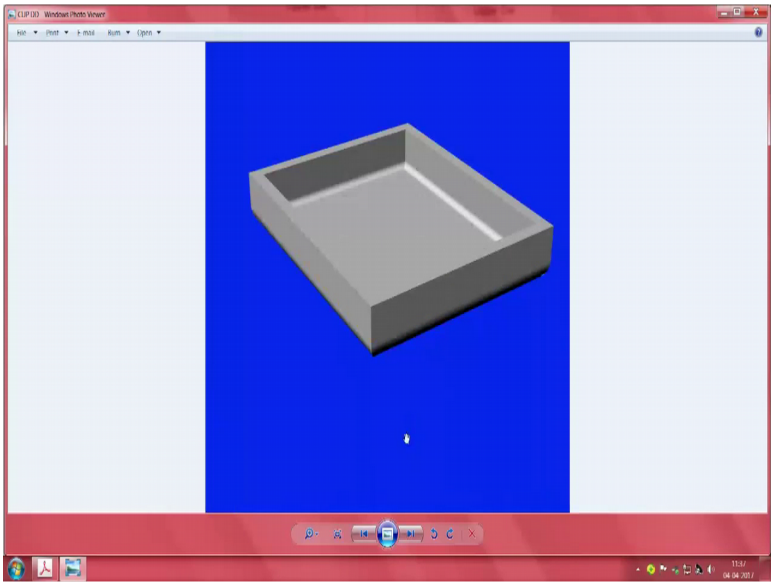
Miraculously these things seem to join with each other without any problem as seen here I will try to focus in the corner you have noticed actually there is a little pile up, though we started with no thickness we seem to end up with a small you know something which is trying to push into the other corner which happens here also.

Though the thickness is very small in this case, that sheet thickness is probably around 0.6 or 0.7 mm cardboard, still we still see the problem. Little closer view it is actually a small object and then to make it hold shape another of my colleagues is trying to hold them together and then you as applied this fevicol. Fevicol is nothing it is a trade name of these modern cardboard glue which would I used in school.

And then after that see the top and bottom have been attached together. See here so far bottom is made, and then the top top was only a a pre form seen that know that is just pre form. After sticking all these parts together at the bottom is just attached to the top and you see we have a beautiful nice compact box this is what I was talking to in my first lecture remember my first lecture, I told you should learn to make all these small items two things are here in this one of them is the earlier extension we just you know took a standard existing some unit we had and then I said multimeter is probably some which is good, and then this one is imaginary.

But more for the purposes of showing you how sheet metal is affected by these things. So, you can see I have a as now I say go a long I wanted to show you a very very interesting thing.

(Refer Slide Time: 22:36)

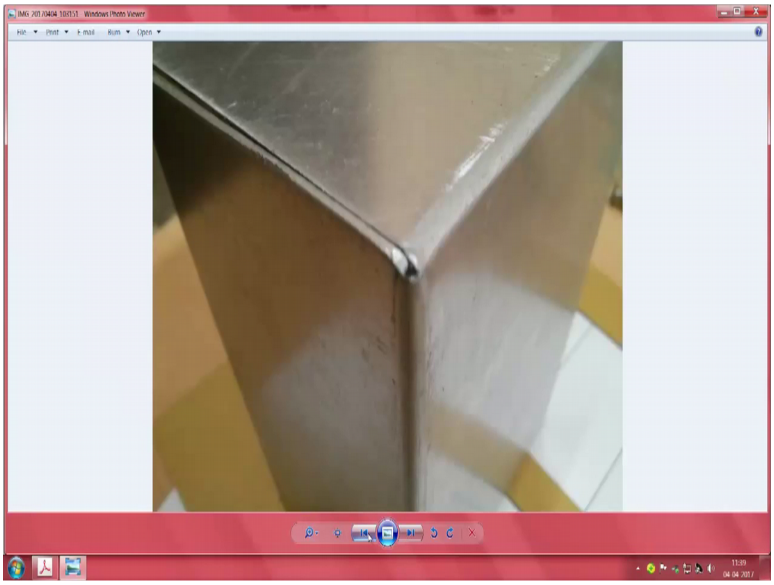


See if I have to make a sheet metal cup, it is possible for me you understood the internal reduces have been taken care of external radius also has been taken care of, but the only thing point in reality it is unlikely there you will have shop corner shape.

If you want to make a cup this, which is quite a lot like a cup I have here. Seen here another of my boxes it has radii all the directions that one is a little similar to this cup even internal radius and then thickness has been added. To the internal radius thickness has been added and the corners also externally they have these corners. Looks nice and then one small bit of detail which is not visible is that small thickness you know half the thickness has been removed here, and in the stop there is a small what you call catch here and the whole thing snaps and I have a smooth enclosure which goes neatly into my point I carry my hearing it is.

So, coming back my monitor what you shown here the sort thing can be done only we using a technique called deep drawing. The case of deep drawing we have a cavity outside cavity which forms this and then we have this inside cavity. That inside cavity and outside cavity form an object like this, but in real life we will notice that this cannot be done by a simple sheet metal operation.

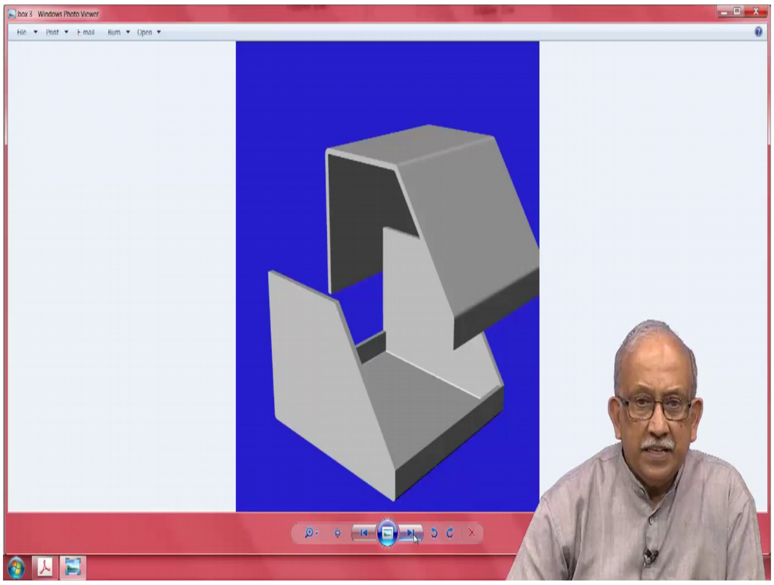
(Refer Slide Time: 24:23)



When we do sheet metal you end up with this small issue about how join the corners. You see here we have a thickness here, we have a thickness there and something happens on the whole thing because two thicknesses are involved. Something overlaps there you not as easy as we think it is corner is still tough. You want to completely avoid the corner we end up with giving radius fully here, I have seen that there is a radius radius and complete the radius has been relieved, but these are not suitable for external operation.

So, how we overcome this small problem this corners and this you know we need to work a lot more about it to see where we come with these things, I do not know if you remember.

(Refer Slide Time: 25:22)



This quite a little like the boxes trying to show you yesterday. Very conveniently I glassed over this corner just glassed over; because to early if we if we confuse our self with all these details now you will suddenly notice, well in a cad drawing. It is easy to make these a 3 D render on checks in realty what you will end up is most likely this corner is most likely to you look like this, seen that know it is not a matter of just ugly is not fully functional, not at all fictional.

So, you need to spend a lot of time worrying about this detail. So, right now I will go back to a few of my samples, if are people can shown these things. Well this enclosure all has rounded things, this one know seems to be a little more flat flatter things. This can with a little bit of what say interest is very much possible to a for us to fabricate a small unit like this.

(Refer Slide Time: 26:46)

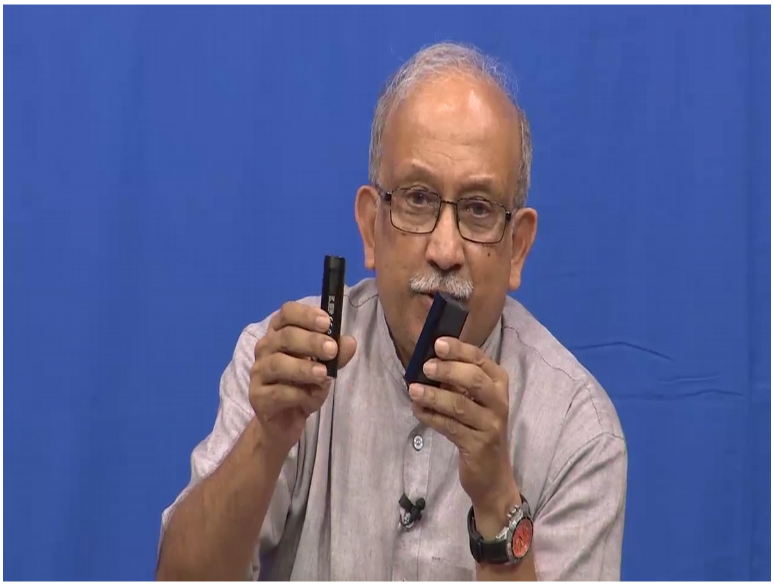
This is the unit which is why it has been shown. So, while things look when we try to show them in workshop; in reality the corners and all do not match well and there is a gap here actually. It is not escalating. In fact, it is not even touching it properly; 2 of the limbs have been they have been able to manage it, but in this limp because of the closure in the end and. So, on that is not possible to close these things.

(Refer Slide Time: 27:26)

So if you really want a perfect square in the corner, if you want to make a perfect edge our choices is probably to continue with welding. This is one of those which I have shown you earlier a shield a dark welding in which you know either r corner some other gas has been used and then fill it, it have been put inside and upside it looks flush similarly on under thing if you were to you know weld it on the outside, inside will probably be flush.

So, in play around a little with these things it looks a little like that gasket with I have shown you the other based lecture, and it is not as if rounded edges only the answer to all our problems.

(Refer Slide Time: 28:21)



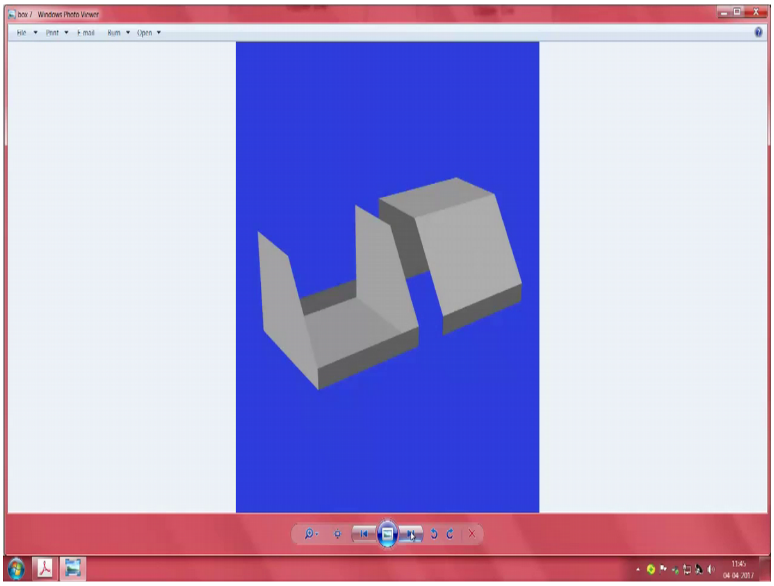
So, in real life see here I showed you 2 things, both of them use a that same 18650 cell inside round some benefits, it makes you know it may looks little like some other object in this case see that extra volume actually volume appears to be about the same cell is in this.

And then we have something which gives us slightly more work. This can be also done by sheet metal this can probably done by taking a stub like the mu highway h pvc or mu pvc tubes, and then building up all our things inside. Same things can be done probably if you have an aluminium extrusion you can make it this case it is made out of some thermoplastic, so, that injection moulding can be done, but if you have if there were to be heat generating components.

We will probably be using an aluminium extrusion or you can even assemble is using is ordinary whatever work shop items you have. Just like I have shown you the section where we have two things being close together, at a you will see this in most of these things you see here we have a top and we have a bottom and then.

If you remember I was talking to something about if you cannot hide something highlights it. Next time look around all the injection moulded parts you have they do not really close because the human eye can easily locate relative differences. So, intentionally they leave a gap then make a step and all that overall that point looks quites precise absolutely no problem with it, having shown a few of these samples now permit me to go back this whole thing.

(Refer Slide Time: 30:49)



So, in reality this is not easy to produce you have seen this know, well it looks nice it is not that easy to produce or anything. Same box we I showed you here this did have a thickness, this is what has been done in the cardboard thing. Then when as you come here slowly we have added the thicknesses here, the movement we add the thickness we end up with two issues which I thought know I will highlight one more time here.

One is you have seen here there is a bending radius inside it is real. Similarly there is a bending radius here real and then the movement you have an internal bending radius, chances are they will be an outside radius also which will be part of ;you have seen this here if I go a little closer while the inside bending radiuses clear here outside also sheet thickness internal plus sheet thickness will come. And you need the internal bending radius yes otherwise the sheet is likely to clear apart.

This is the reality of sheet metal; well again very conveniently and to avoid confusion I have ignored several things, one of the features which will notice is well there is an internal bending radius this bending radius is supposed to sit here in this corner, seen here. So, there will be a clash this will not fit here unless this is smooth and out on the outside. So, in the corner here a little bit of smoothing is required, and same thing happens here also if you are to come with the bottom here it is not honkey dowry is (Refer Time: 32:41) a shop corner shop corner here and then conveniently it has been stopped here.

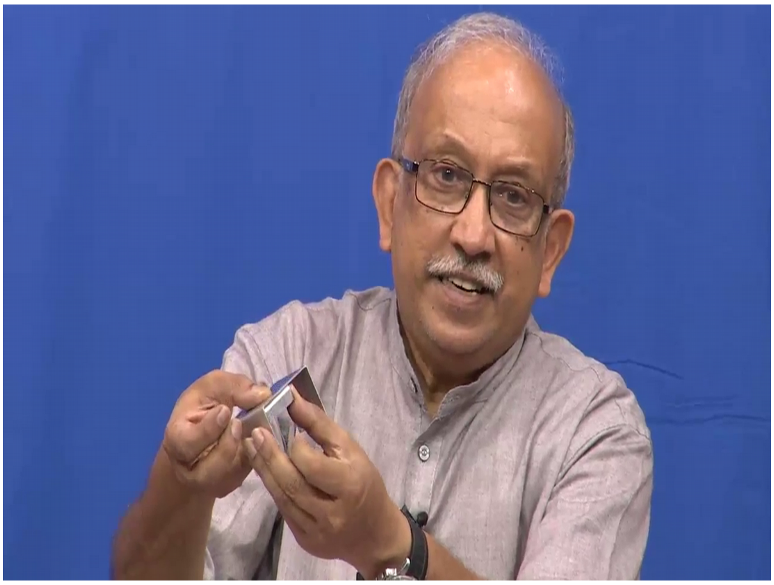
That is not likely to easily fit into this corner you need a little rounding off here or you need some other operations there. So, what looks otherwise as a simple; these things know it really does not close so well. It will be a lot like the other sample which I have shown; yes seen this this is more like what is likely to happen. So, you need to take decision about which part overlaps what in this case there is a bend here, internal plus external radius is taken there is another bend here internal plus sectional has been taken care here. If you want to avoid you can you need to probably leave both the thickness is visible here and with an another internal radius also which is the simple which I have shown, but in reality what they are do is they make this overlap a little, and make some special what you call care is taken to make this follow in to the other thing when it bend it is normally it becomes a square.

It will just be a square after that after the bending is over they have whether jointly you know append it or they have a filed it to fit the other operation. This is sheet metal reality, but then why do you need to learn? You see here it started here you see here it is sharp and, but in the corner we have a radius here we have a radius here and then there is a unspecified thing. If you are to make it flow here you need to do the other adjustments which have been done.

So, probably this lecture has been hopefully you know enough informative you have seen here it is not we do not want any clash to end up with a completely open thing. So, this is go a this is one of the what you call I will see whether I can continue or I will stop here. Now going back here you see here everywhere whenever there is a corner and whenever this type of operations are done, sometimes these are all done to accommodate other limbs also image any one of these ends were to be closed.

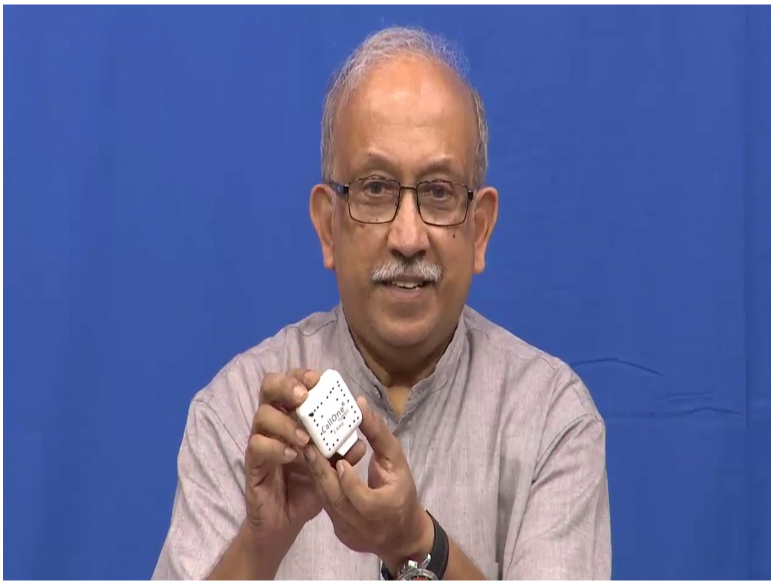
So, in blank there would have been a limb, which is a little longer and then just as you have a punch and to sider operation is there, you can have it on all the four sides. So, you have a cup like thing the punch forces inside, and then when it comes out you have a beautifully formed part. So, we have two angles.

(Refer Slide Time: 35:35)



Flat imagine the whole thing is flat. So, you have all the four type of these operations are there you make a punch push it inside, and all four of them close very well. Only in nature and an organic they grow into that, but in this case we still end up with probably small corners like this or you have to do something to take care of all these edges which are there. I suggest you go over the whole this thing one more time.

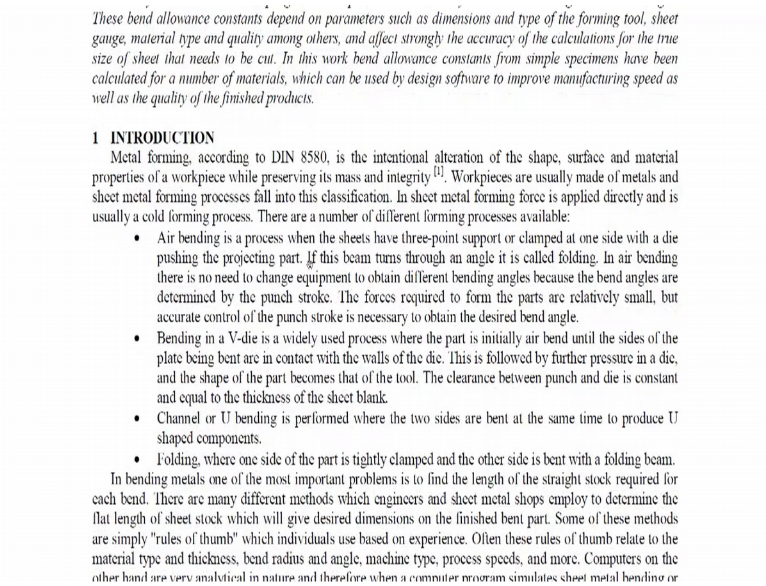
(Refer Slide Time: 36:13)



And then it is not as if plastic things are easy, if you make something out of injection moulding this one is taken from a some mobile charger which we have probably put to a bad use and then it burnt out.

But you see here even here know they have all these nice corners everywhere and in this corner we need to give slots. So, the air passes it did not help just hilling holes all over the place, did not help just have a look and see and then after that I need not bring you back.

(Refer Slide Time: 36:53)



These bend allowance constants depend on parameters such as dimensions and type of the forming tool, sheet gauge, material type and quality among others, and affect strongly the accuracy of the calculations for the true size of sheet that needs to be cut. In this work bend allowance constants from simple specimens have been calculated for a number of materials, which can be used by design software to improve manufacturing speed as well as the quality of the finished products.

1 INTRODUCTION

Metal forming, according to DIN 8580, is the intentional alteration of the shape, surface and material properties of a workpiece while preserving its mass and integrity [1]. Workpieces are usually made of metals and sheet metal forming processes fall into this classification. In sheet metal forming force is applied directly and is usually a cold forming process. There are a number of different forming processes available:

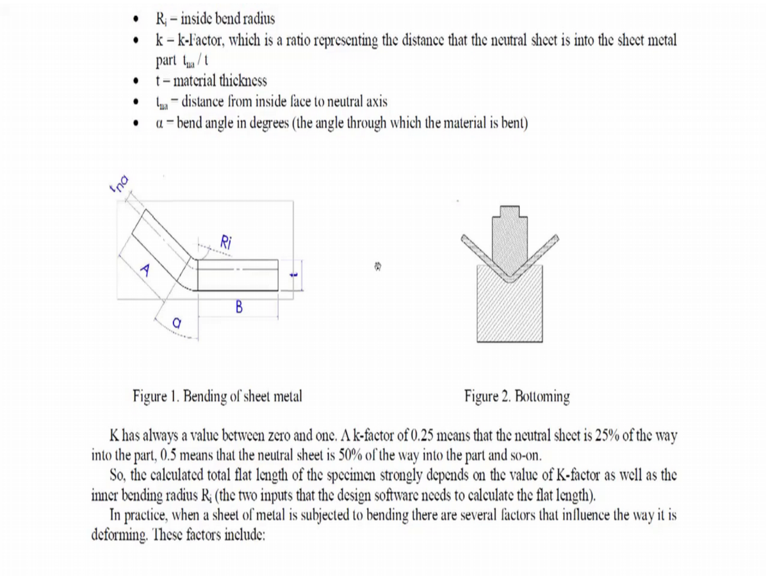
- Air bending is a process when the sheets have three-point support or clamped at one side with a die pushing the projecting part. If this beam turns through an angle it is called folding. In air bending there is no need to change equipment to obtain different bending angles because the bend angles are determined by the punch stroke. The forces required to form the parts are relatively small, but accurate control of the punch stroke is necessary to obtain the desired bend angle.
- Bending in a V-die is a widely used process where the part is initially air bend until the sides of the plate being bent are in contact with the walls of the die. This is followed by further pressure in a die, and the shape of the part becomes that of the tool. The clearance between punch and die is constant and equal to the thickness of the sheet blank.
- Channel or U bending is performed where the two sides are bent at the same time to produce U shaped components.
- Folding, where one side of the part is tightly clamped and the other side is bent with a folding beam.

In bending metals one of the most important problems is to find the length of the straight stock required for each bend. There are many different methods which engineers and sheet metal shops employ to determine the flat length of sheet stock which will give desired dimensions on the finished bent part. Some of these methods are simply "rules of thumb" which individuals use based on experience. Often these rules of thumb relate to the material type and thickness, bend radius and angle, machine type, process speeds, and more. Computers on the other hand are very analytical in nature and therefore when a computer program simulates sheet metal bending or

To this for sometime it is not as if last two thing about bending allowances has been done.

Just as it is treated as a empirical operation without understanding what is going on inside, here there are people who keep working on these what you call this is a sheet metal. Few operations especially you have something called metal forming definition and then later on some people have been working on it to find out what is the bending allowance to be given and so on well. So, far what I have shown you has been what you call more about a industrial practice, the thing like K factor and all have already people have been working on it and released notes are available in the public.

(Refer Slide Time: 38:02)



- R_i – inside bend radius
- k – k-Factor, which is a ratio representing the distance that the neutral sheet is into the sheet metal part t_{na} / t
- t – material thickness
- t_{na} – distance from inside face to neutral axis
- α – bend angle in degrees (the angle through which the material is bent)

Figure 1. Bending of sheet metal

Figure 2. Bottoming

K has always a value between zero and one. A k-factor of 0.25 means that the neutral sheet is 25% of the way into the part, 0.5 means that the neutral sheet is 50% of the way into the part and so-on.

So, the calculated total flat length of the specimen strongly depends on the value of K-factor as well as the inner bending radius R_i (the two inputs that the design software needs to calculate the flat length).

In practice, when a sheet of metal is subjected to bending there are several factors that influence the way it is deforming. These factors include:

So, if you see here this is not very different from what you have shown earlier well that is you know they claim a intellectual property, and this is in keeping with the tradition of our you know open access to various documents and all that. A lot of this stuff is also available of the academic what you call unit so.

(Refer Slide Time: 38:32)

Evangelos Karachalios and Achilles Vairis

combination of the above parameters is described below:

1. A piece of metal (rectangular) is cut and measured so as to know its original dimensions = original flat sheet dimensions (length and thickness are the important values – width is not taken into consideration so it is assumed constant)
2. This sheet of metal is then subjected to two consecutive bends of 90° and forms a piece with a 3D shape as shown in figure 2
3. The specimen is then measured so as to obtain its external formed dimensions a, b, c (figure 2)
4. A specimen with the above measured dimensions (a, b, c & t) is being drawn in the CAD software (Solidworks 2003 in this case)[2]
5. The tool tip radius R_i is assumed constant and equal to 1 as all the tools used in the current investigation have a punch radius equal to 1mm
6. The value of k (k-factor) is altered in the software until the calculated flat length of the specimen matches exactly the initial cut length of the specimen in step 1

This value of k-factor can now be used to calculate:

1. The flat length of a specimen with known external dimensions (a,b,c) and thickness (t)
2. The exact positions where bending should occur during production

So, for a specimen of any size, with the same thickness, of the same material which is produced with the same combination of die & punch a single k-factor value can be used.

In the experimental work [5, 6], this procedure was repeated for all the different combinations of parameters (material type, sheet thickness, dies & punches dimensions) for a set of three specimens for each setting and average values were used in the calculations.

The upper and lower tools used are listed in tables 2 and 3. The CAD/CAM programme can estimate the dimensions of the sheet metal piece required to bent to produce the 3D object that the engineer is drawing. The dimensions of this initial sheet metal piece are different from the final bend product and the size of this difference is determined by the K-factor that the programme uses. The K-factor depends on the material selected, the type of bending, the thickness of the specimen and the radius of the tool. All experiments were performed

Experience have been carried out and then gauges you know a millimetre gauge and then type material, and then the things have been reported which are verifiable, whole thing I have reported and then there are several publications other than trade journals where it is possible for us to take care of all these things .

(Refer Slide Time: 39:02)

mm.

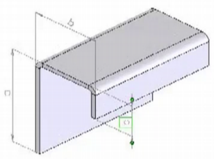


Figure 3. Air bend specimen

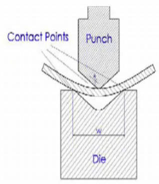


Figure 4. Air bending

Type (Trumpf)[7]	D20	D2	D23	D24
Die width W (mm)	6	8	12	16
Opening angle	86°	30°	86°	86°

Table 2. Single-V dies

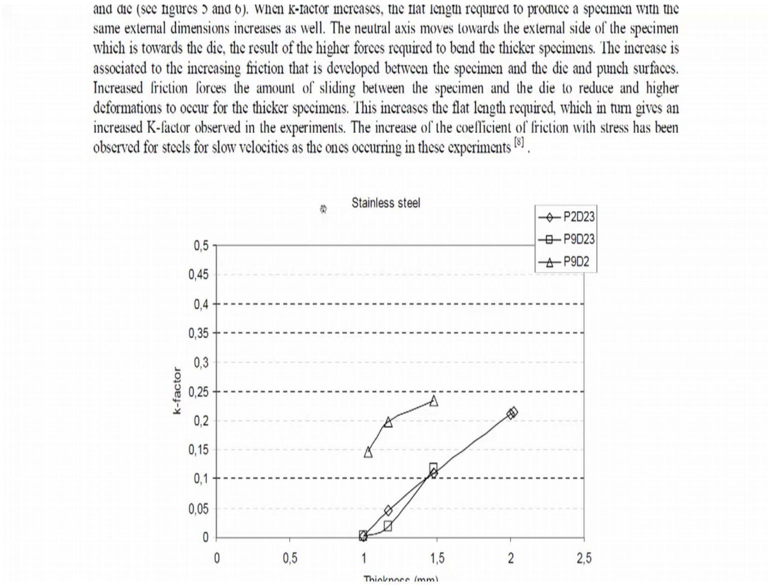
Type (Trumpf)[7]	P9	P2
Opening angle	28°	86°
Tool tip radius (mm)	1	1

Table 3. Punches

Now, the issue being you see here, cad cam programs can estimate the dimension the sheet metal piece required to bend produce 3 d object when the engineer is drawing. Life has been made simpler with analytics already having gone into various types of operations.

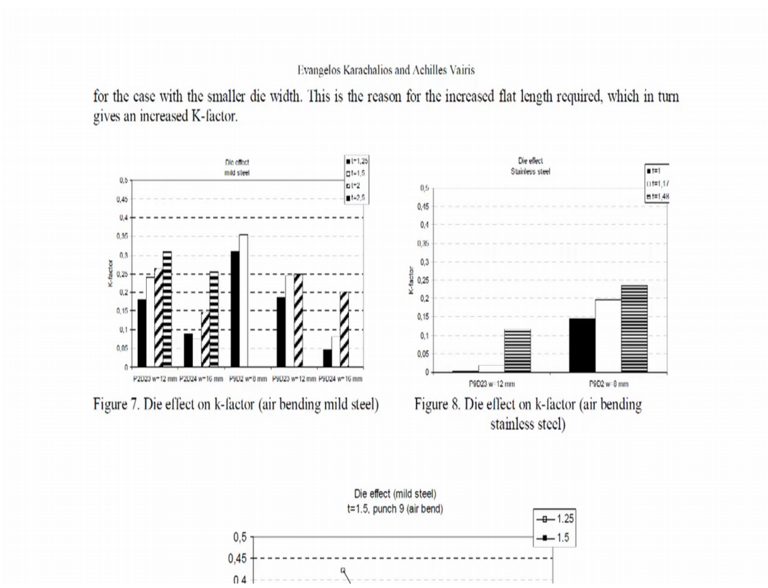
So, we have here how the K factor gets you know accommodated and so on.

(Refer Slide Time: 39:35)


and die (see figures 5 and 6). When K-factor increases, the flat length required to produce a specimen with the same external dimensions increases as well. The neutral axis moves towards the external side of the specimen which is towards the die, the result of the higher forces required to bend the thicker specimens. The increase is associated to the increasing friction that is developed between the specimen and the die and punch surfaces. Increased friction forces the amount of sliding between the specimen and the die to reduce and higher deformations to occur for the thicker specimens. This increases the flat length required, which in turn gives an increased K-factor observed in the experiments. The increase of the coefficient of friction with stress has been observed for steels for slow velocities as the ones occurring in these experiments [8] .

So, and then most important have you seen here saying coefficient of friction with stress has been observed for steels for slow velocities as the ones occurring and so on. So, the amount of different types of forces and then a reproducibility all of it is available in books.

(Refer Slide Time: 40:06)


Evangelos Karachalios and Achilles Vairis

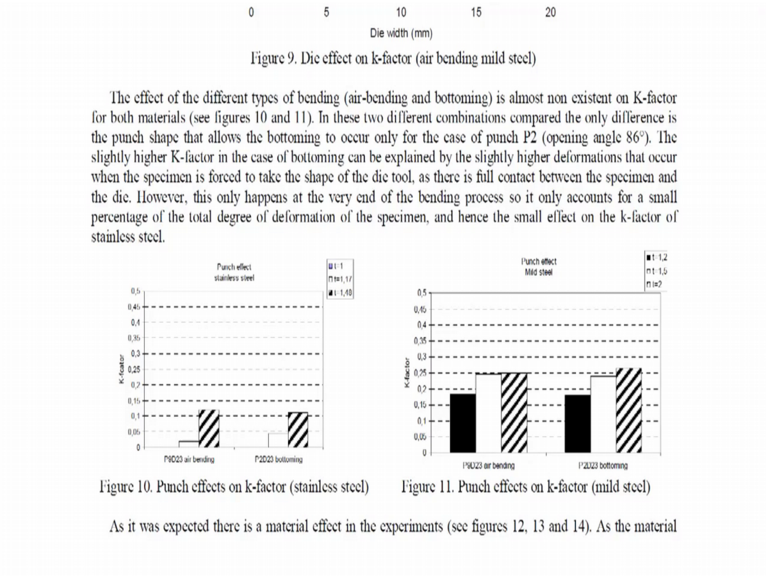
for the case with the smaller die width. This is the reason for the increased flat length required, which in turn gives an increased K-factor.

Figure 7. Die effect on k-factor (air bending mild steel)

Figure 8. Die effect on k-factor (air bending stainless steel)

Still small compensation know saying for a smaller die width what happens for a longer die width what happens, a lot of this information is available since some of you are the electrical engineers they are familiar with.

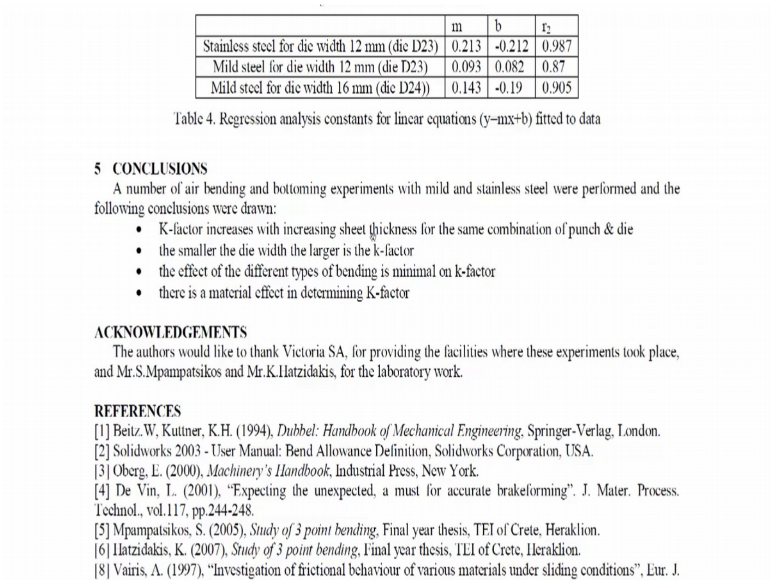
(Refer Slide Time: 40:24)



You look at a anything a small tharmistor or if you look at an (Refer Time :40:24) or anything there is usually the title sheet contains a little bit of the extracts bottom sheet gives you the full actual heavier and what is it that is committed.

So, you see here lot of stuff about punch shape and all this stuff know is available of the publications. So, you can always go about and then try to do understand these things.

(Refer Slide Time: 41:01)



	m	b	r_2
Stainless steel for die width 12 mm (die D23)	0.213	-0.212	0.987
Mild steel for die width 12 mm (die D23)	0.093	0.082	0.87
Mild steel for die width 16 mm (die D24))	0.143	-0.19	0.905

Table 4. Regression analysis constants for linear equations (y=mx+b) fitted to data

5 CONCLUSIONS

A number of air bending and bottoming experiments with mild and stainless steel were performed and the following conclusions were drawn:

- K-factor increases with increasing sheet thickness for the same combination of punch & die
- the smaller the die width the larger is the k-factor
- the effect of the different types of bending is minimal on k-factor
- there is a material effect in determining K-factor

ACKNOWLEDGEMENTS

The authors would like to thank Victoria SA, for providing the facilities where these experiments took place, and Mr.S.Mpampatsikos and Mr.K.Hatzidakis, for the laboratory work.

REFERENCES

[1] Beitz,W, Kuttner, K.H. (1994), *Dubbel: Handbook of Mechanical Engineering*, Springer-Verlag, London.
[2] Solidworks 2003 - User Manual: Bend Allowance Definition, Solidworks Corporation, USA.
[3] Oberg, E. (2000), *Machinery's Handbook*, Industrial Press, New York.
[4] De Vin, L. (2001), "Expecting the unexpected, a must for accurate brakeforming". J. Mater. Process. Technol., vol.117, pp.244-248.
[5] Mpampatsikos, S. (2005), *Study of 3 point bending*, Final year thesis, TEI of Crete, Heraklion.
[6] Hatzidakis, K. (2007), *Study of 3 point bending*, Final year thesis, TEI of Crete, Heraklion.
[8] Vairis, A. (1997), "Investigation of frictional behaviour of various materials under sliding conditions", Eur. J.

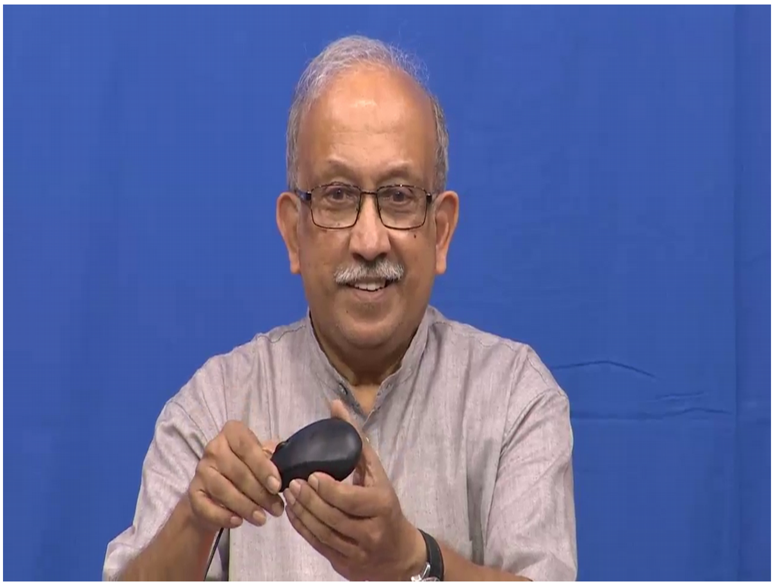
So, that you see in case there is any mistake is that mistake of so far we assuming that all the machines are maintained well, there is no gap or shake or anything of those things I think split in the punch and die alignment, aware the forces are centralized are not.

If you are sure about these conditions you can now work on all these things and then end up with a very convenient reproducible width of blanks. Once your blanks reproducible width things will work very very well, and then two things are there one is the continuously you need to operate all the tooling. Secondly, the material condition should be checked.

So, thank you for today and then we will meet again with techniques level of things. So, far I have talked you about the highest abstraction level saying, how do you try to you know concept I think of products and so on. And second level I have try to take you to the our vacation facility here, saying using sheet metal we can fabricate such items and then for purposes of sheet metal I told you it is very easy for us to make all these nice models. It is card board it is not necessary child's play, and then if you want accommodate thickness you can always make thing sort of a thicker board.

And then finally, if the same thing can be reproduced in plastic materials typically (Refer Time: 42:56) the if you are only concentrating on the shape commercial polished iron sheets are available, and then you also have form boards, then you have polycarbonate, then you have a acrylic, then you have I do not know what are the new types of materials in there are say various types of forms that are available.

(Refer Slide Time: 43:20)



If you here to make a mouse you use an acrylic form to model this, you take it and then you try to script the form and the magic is now with the advent of rapid prototyping you can scan the object and reproduce it directly in a 3 d printer. So, in 3 d printer also next lectures I will see will I can arrange for a demonstration, the easiest I suggest for you is that learn how to make things out of sheet metal and then also plastics.

So, most of this thing the advantage main advantage of plastic is the insulator, and you need not worry too much about what you call how to make sure there are no short circuit something in the same way the disadvantage is that not the mimdnc proof. So, you need to worry about it. Sheet metal has the positive advantage of it is being strong and then if you make it in magnetic materials the shielding is better, and if you make a nonmagnetic materials critical conductivity is still ensured and normal conduct to EMI and those things know behave much much better.

And then ultimately strong beyond belief as compared to plastic and then sheet metal if you make it if you make the drawings and use all the various techniques, there are enough people out there will convert it to a cad drawings. So, if I can take a through a little bit of cad exercise similarly if I can take you through a little bit of a plastic exercise saying how do I join plastics and so on and then show you one or two materials or I am sorry projects which completely are fabricated for each type of thing.

Thank you for today we will probably meet again.

Thank you.