ENR212 Lecture 7 Slides and Notes

Slide 1

Hello, everyone and welcome to Lecture Summary 7. (This lecture works through material covered in Chapter 20 of the textbook.)

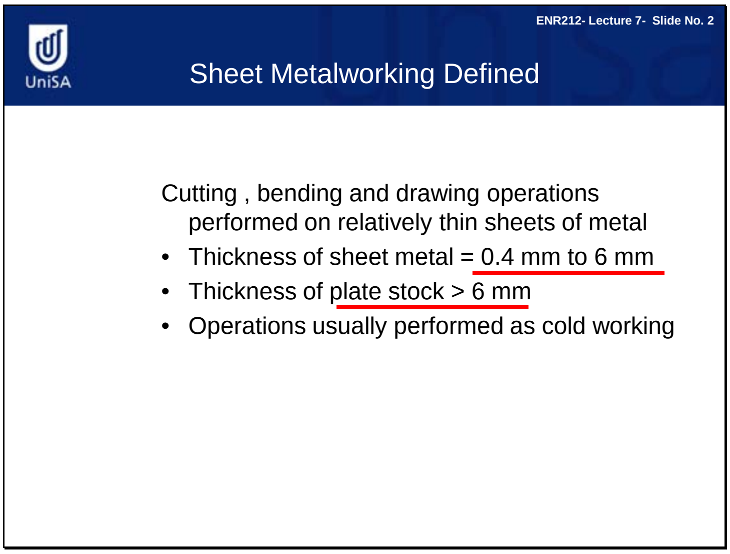
In this lecture, we are going to introduce sheet metalworking. But first we'll review the classification of manufacturing processes. Manufacturing processes include processing operations and assembling operations. Processing operations consist of shaping processes, property enhancing processes, and surface treatment processes. There are four types of shaping processes: solidification processes (such as metal casting), particulate processes, deformation processes (such as bulk deformation and sheet metalworking), and material removal processes.

Today we will look at sheet metal working. There are two basic types of working conditions: hot or warm conditions and cold working conditions. Cold working increases the mechanical properties such as strength. Another major advantage is that, because of strain hardening, it gives a very good surface finish, and accurate dimensions.

There are three types of sheet metal working: cutting, bending and drawing. Cutting is a shearing operation which cuts a metal sheet by using a punch and die. Bending involves straining a metal sheet to take a permanent angle along a straight axis. Deep or cup drawing refers to the forming of a flat metal sheet into a hollow or concave shape, such as a cup, by stretching the metal.

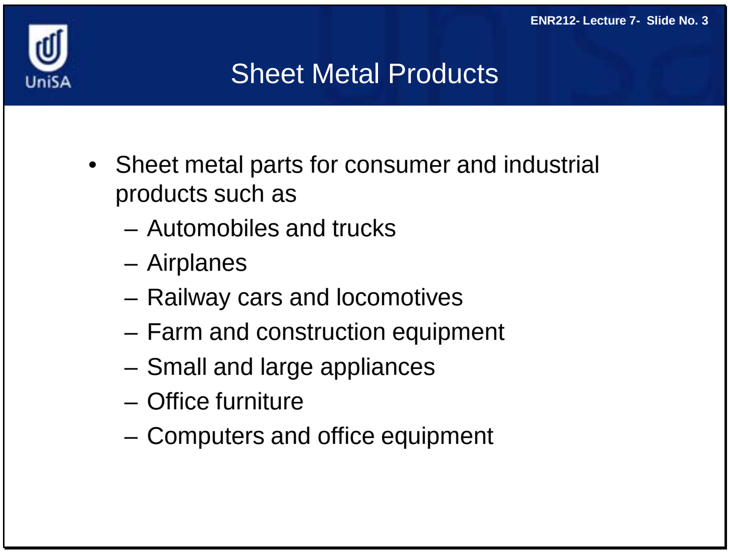
ENR212 Lecture 7 Slides and Notes

Slide 2



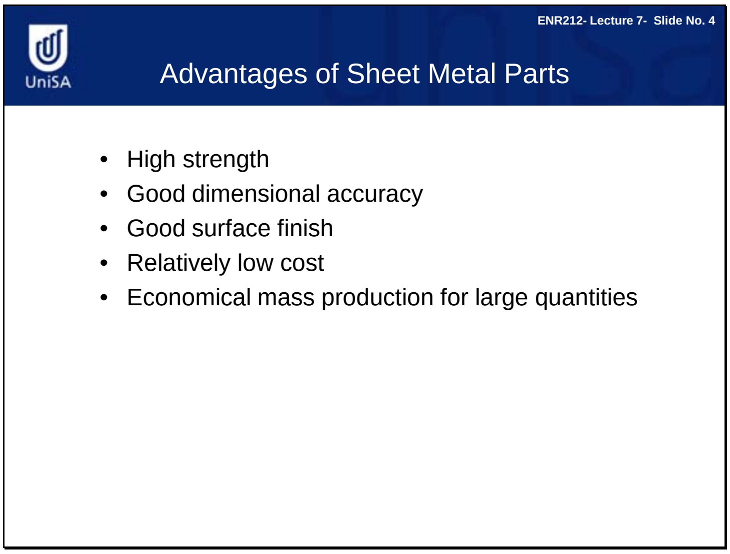
We have already looked at rolling. If you remember, rolling is a metal bulk deformation process. In a typical rolling, the thickness of a metal plate is reduced by two rotating mills. The starting work material in rolling is a metal plate. On the other hand, sheet metalworking operations are performed on relatively thin sheets of metal. Typically, the sheets are between 0.4 mm and 6 mm thick, while plate stock is usually thicker than 6mm.

Slide 3



Sheet metalworking is commercially significant, as you can see from the list. There are plenty of sheet metal products, ranging from transport through farming to office work.

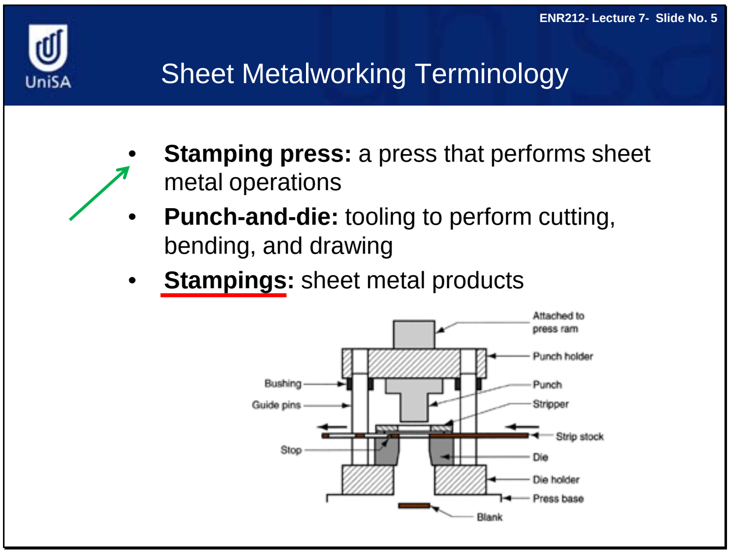
Slide 4



Sheet-metal parts are generally characterized by high strength, good dimensional accuracy, good surface finish, and relatively low cost. If you think back to earlier lectures, you will realize that these features are actually the advantages of cold working, because sheet metal working is often conducted at room temperature (which is cold working).

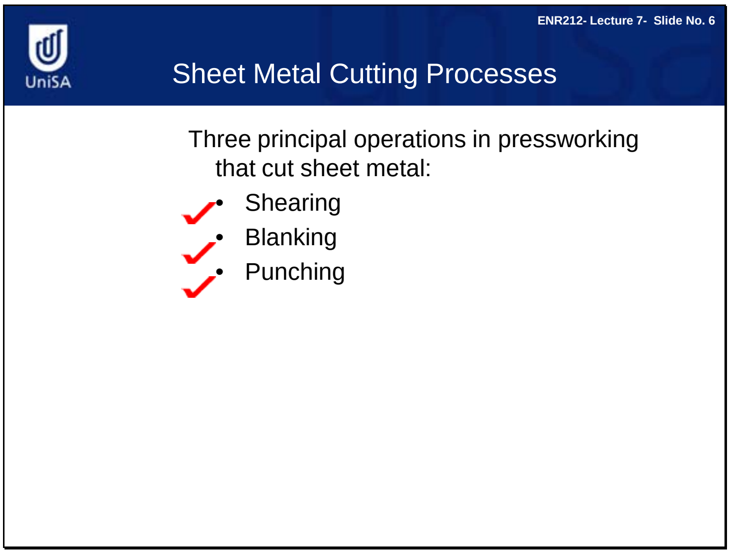
ENR212 Lecture 7 Slides and Notes

Slide 5



A stamping press is a press that performs sheet metal operations. In sheet metal working, the punch-and-die tool is often used to perform cutting, bending, and drawing operations. The products are called stampings.

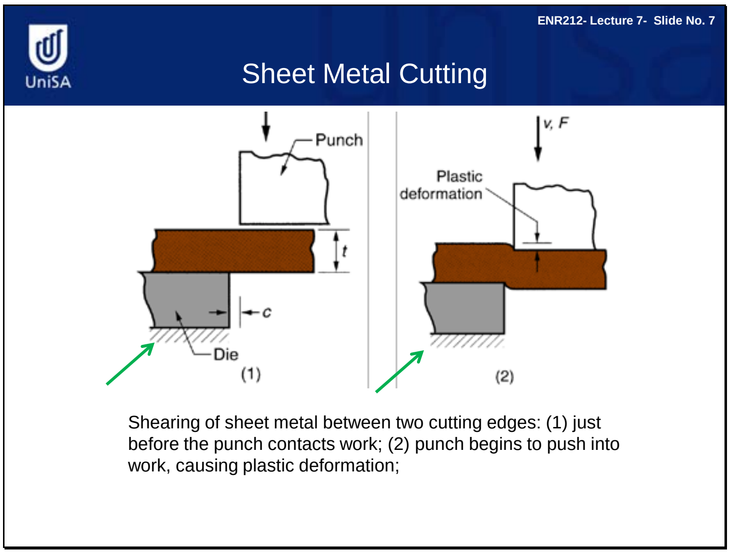
Slide 6



There are three principal operations used to cut metal in pressworking: shearing, blanking, and punching. Shearing involves using two cutting edges along a straight line to separate a metal sheet. Blanking involves cutting sheet metal along a closed outline in a single step to separate a piece from the surrounding stock. Punching is similar to blanking, except the cut piece is scrap.

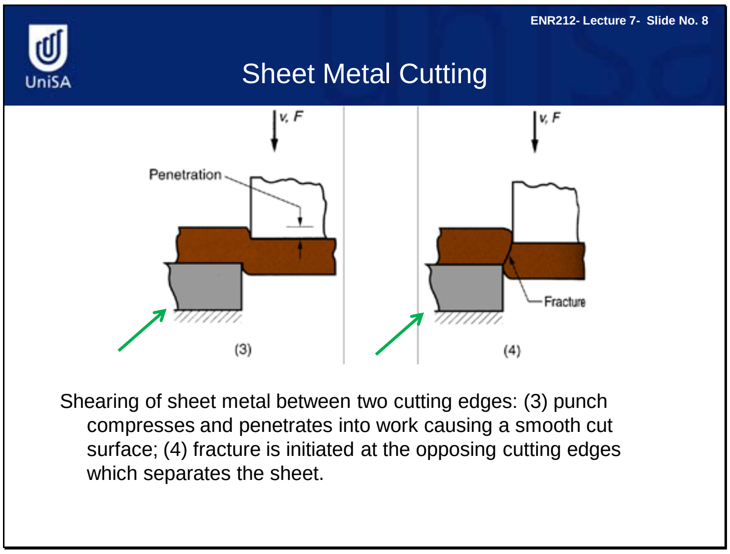
ENR212 Lecture 7 Slides and Notes

Slide 7



Cutting of sheet metal is accomplished using two sharp cutting edges. The shearing action is shown in these four figures. The upper cutting edge is called the punch and the lower is called the die. In Figure 1, the punch sweeps down past the die while the die is stationary. In Figure 2, as the punch begins to push into the metal sheet, plastic deformation occurs in the surfaces of the sheet.

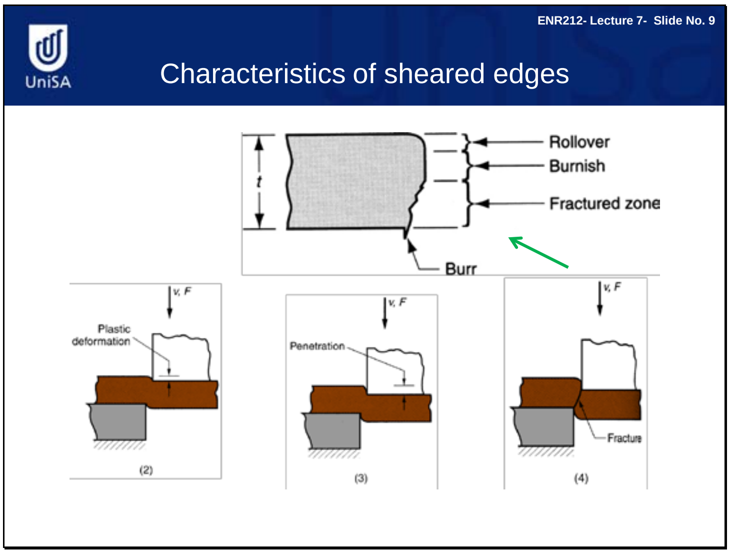
Slide 8



In Figure 3, as the punch continues to move downward, penetration occurs, in which the punch compresses the sheet and cuts into the metal. This penetration zone is generally about one-third the thickness of the sheet. In Figure 4, as the punch continues to travel into the work, fractures initiate and grow in the workpiece at the two cutting edges. If the clearance between the punch and die is correct, the two fracture planes meet, resulting in a clean separation of the work into two pieces.

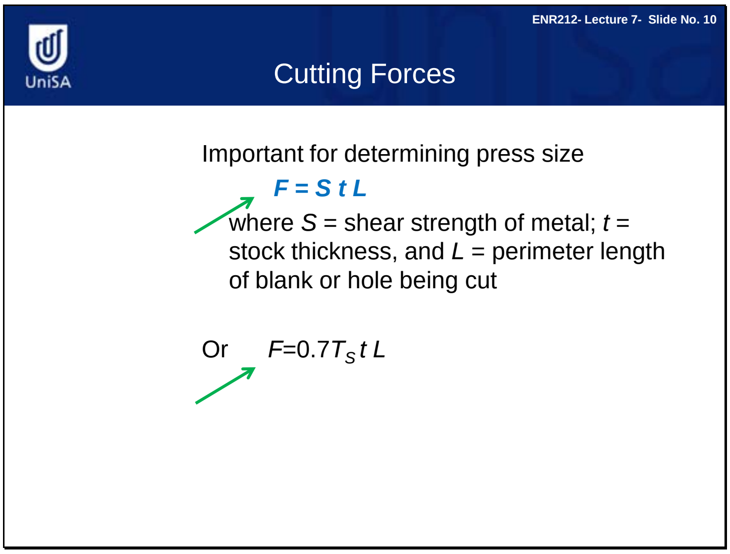
ENR212 Lecture 7 Slides and Notes

Slide 9



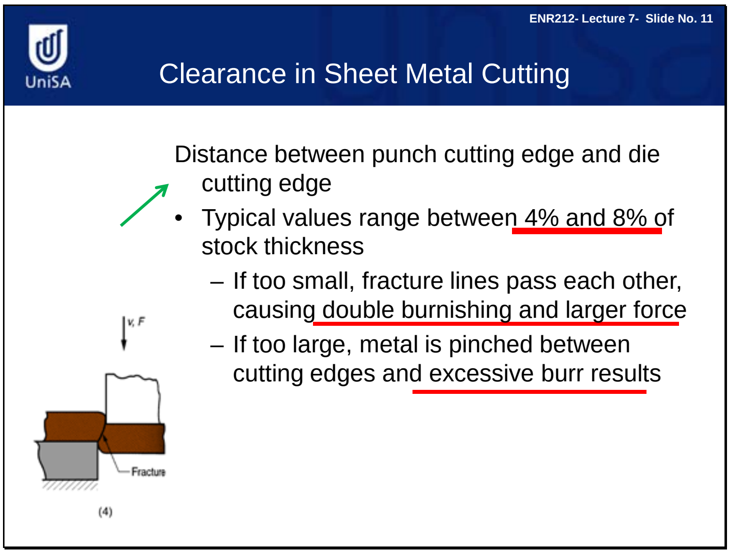
A cut piece of metal consists of four zones: rollover, burnish, fractured zone and burr. The size of the burr can be minimized by adjusting the gap between the punch and the die.

Slide 10



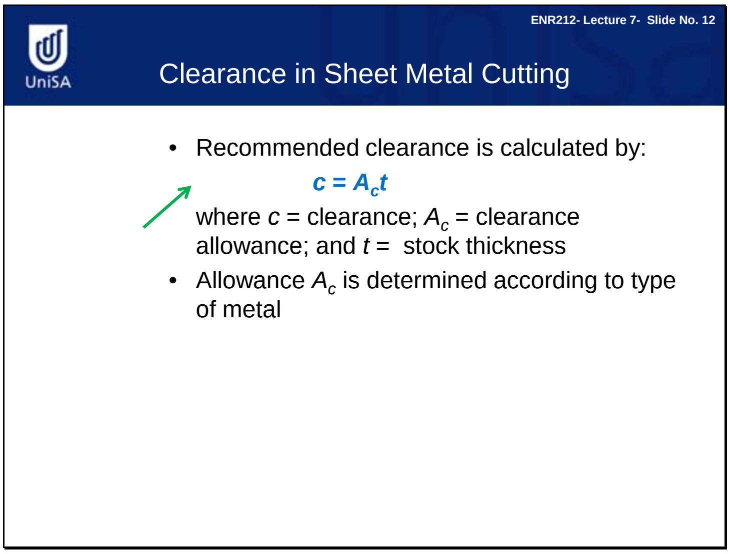
Accurate estimation of the cutting force is important, because it is this force which determines the size of the press needed. The cutting force 'F' in sheet metalworking can be determined by this equation, where S = shear strength of metal; t = stock thickness, and L = perimeter length of blank or hole being cut. Since the testing of shear strength is much more difficult than the tensile strength testing, we need the second equation to estimate a cutting force.

Slide 11



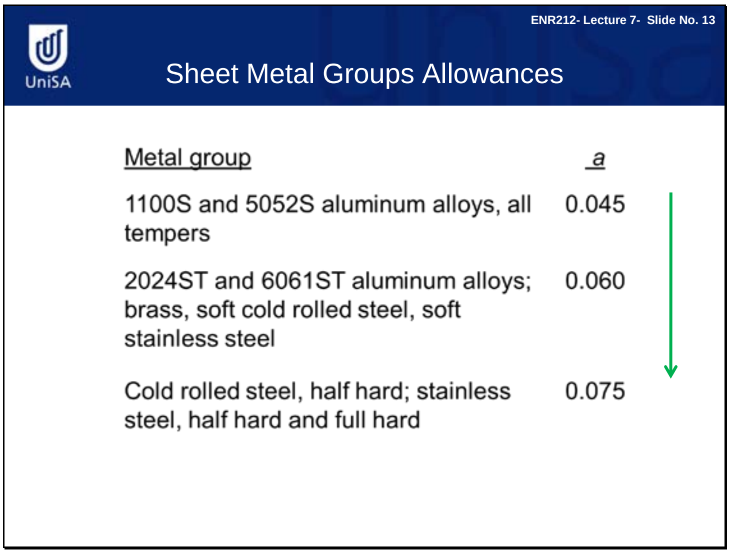
The clearance in sheet metal cutting refers to the distance between the punch cutting edge and die cutting edge. Typical clearance values range between 4% and 8% of stock thickness. If the clearance is too small, the fracture lines pass each other, causing double burnishing and larger cutting forces. If they are too large, the metal is pinched between cutting edges and you will get excessive burr results.

Slide 12



You use the formula above to calculate the recommended clearance, where c equals the clearance, A C equals the clearance allowance, and t equals the stock thickness. Clearance allowance is determined according to the type of metal.

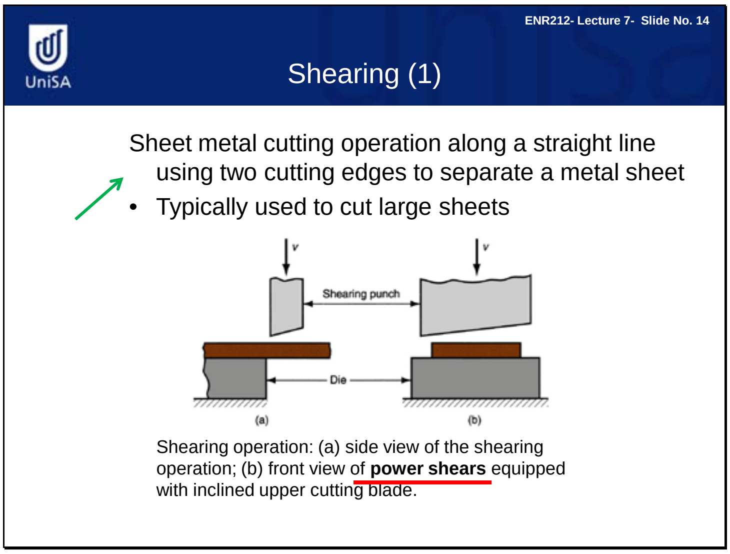
Slide 13



Different metals have different cutting allowances. In general, the more brittle a metal, the more clearance it needs in shearing.

ENR212 Lecture 7 Slides and Notes

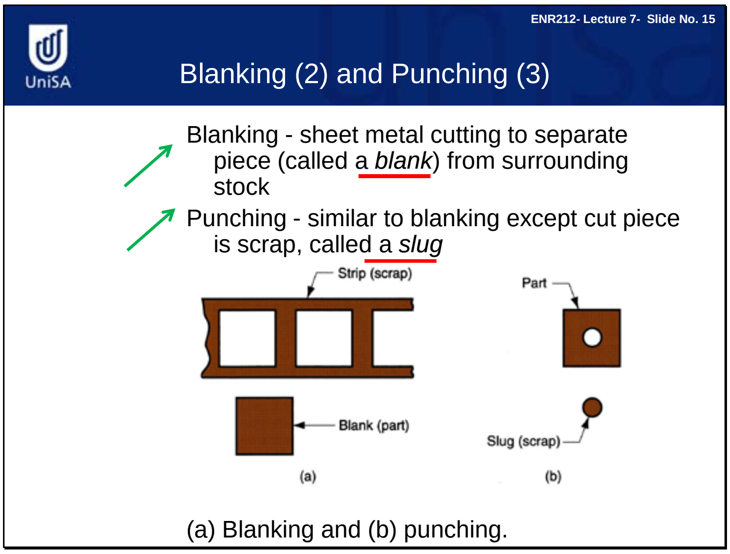
Slide 14



In the first slide of this lecture, we looked at the various classifications of sheet metal working. There are three types of sheet metal working operations: cutting, bending and drawing. Cutting includes shearing, blending and punching. Shearing refers to a sheet metal cutting operation which uses two cutting edges to separate a metal sheet. Shearing is typically used to cut large sheets into smaller sections for subsequent pressworking operations. Shearing is performed on a machine called a power shears. This is the side view and front view. The upper blade of the power shears is often inclined, as shown in the figure, to reduce the required cutting force.

ENR212 Lecture 7 Slides and Notes

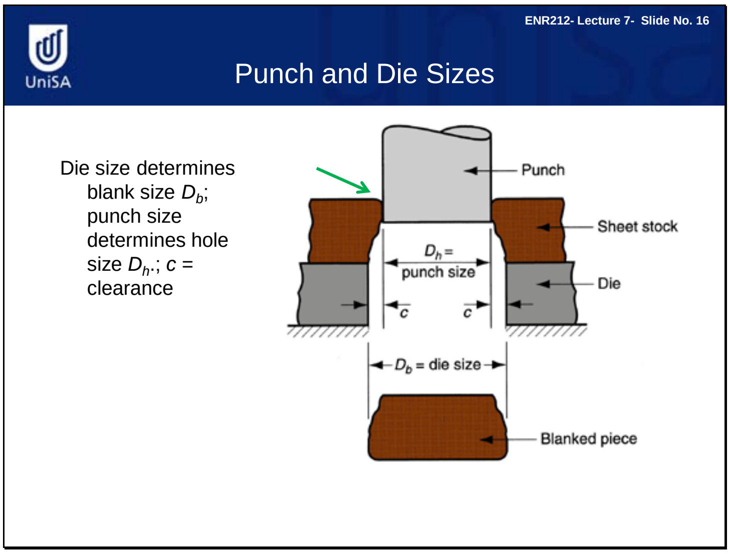
Slide 15



Blanking involves cutting sheet metal along a straight line in a single step, to separate a piece from the surrounding stock. The part that is cut out is the desired product, and is called a blank. Punching is similar to blanking, except that the cut piece is scrap, and is called a slug. The remaining stock is the desired product.

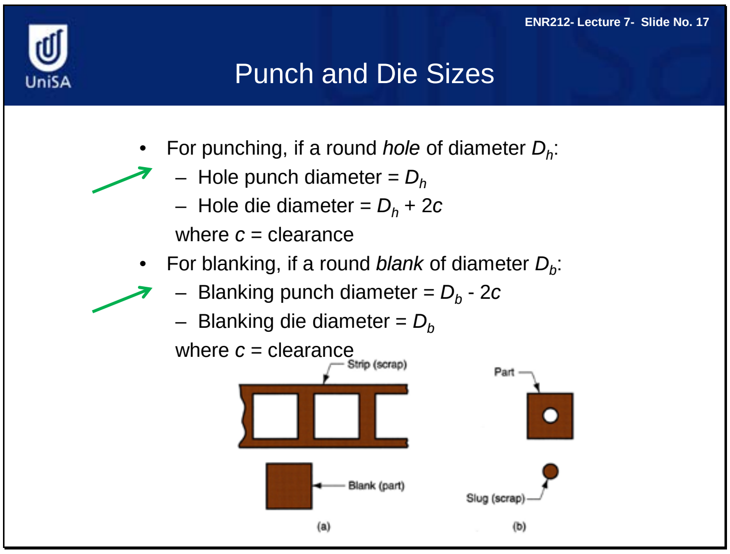
ENR212 Lecture 7 Slides and Notes

Slide 16



This figure shows you a hole produced by a metal cutting process. Note that the hole has a non-uniform diameter. Therefore, blanking and punching processes have different ways to determine the diameter of the punch and die.

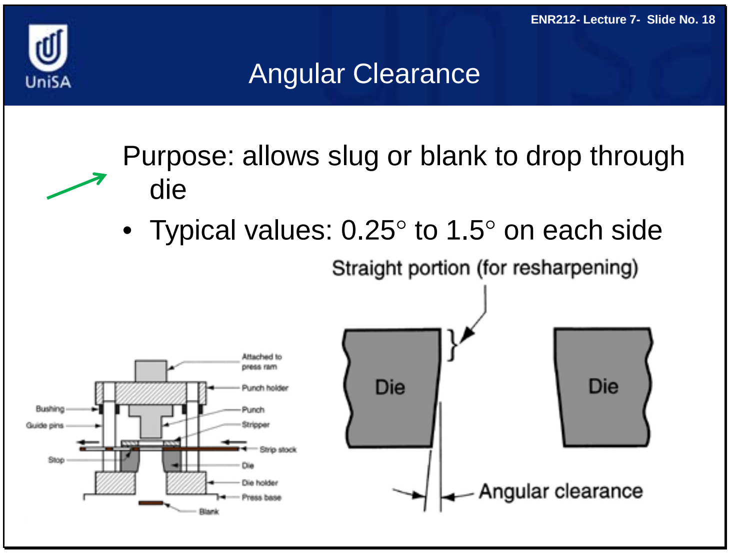
Slide 17



The determination of punch and die sizes in punching and blanking is tricky. (You can assume that this will be one of the tasks which are part of quizzes and the exam.)
For punching, if a round hole of diameter D h is requested, then the hole die diameter is equal to D h plus 2c, where C represents clearance. For blanking, if a round blank of diameter D b is requested, then the blanking punch diameter equals D b minus 2c .

ENR212 Lecture 7 Slides and Notes

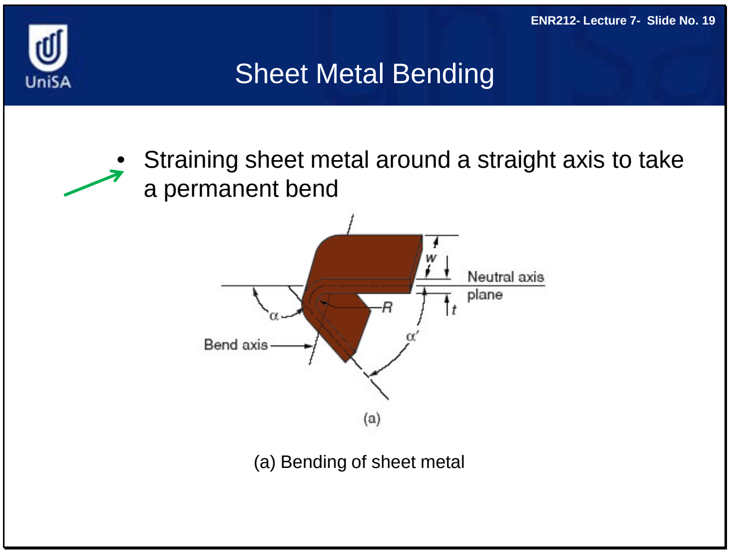
Slide 18



In order for the slug or blank to drop through the die, the die opening must have an angular clearance of 0.25□ to 1.5□ on each side.

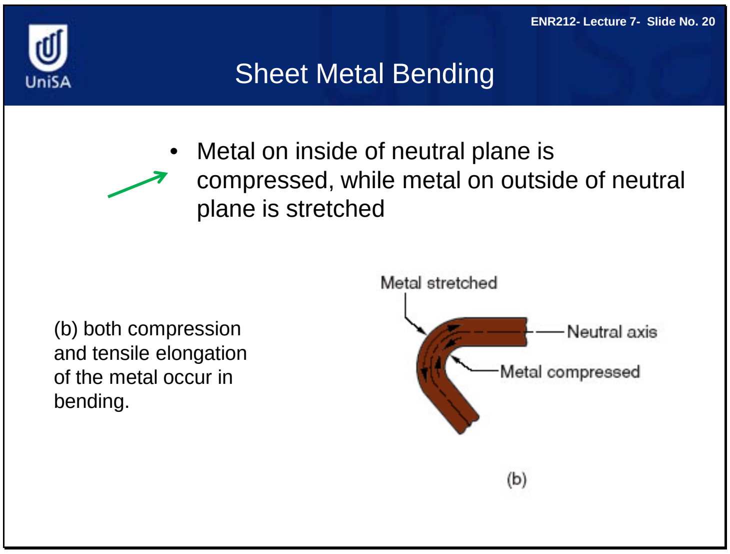
ENR212 Lecture 7 Slides and Notes

Slide 19



Bending in sheet metalwork is defined as straining of a metal sheet to take a permanent angle along a straight axis.

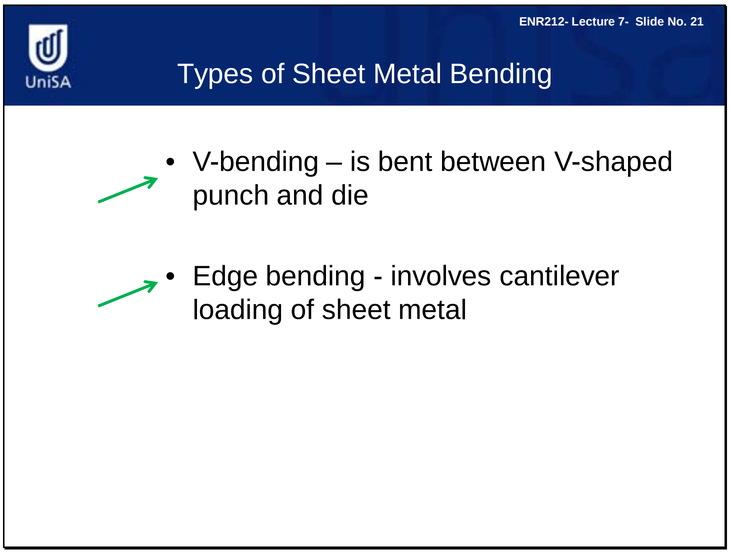
Slide 20



During the bending operation, the metal on the inside of the neutral plane is compressed, while the metal on the outside of the neutral plane is stretched.

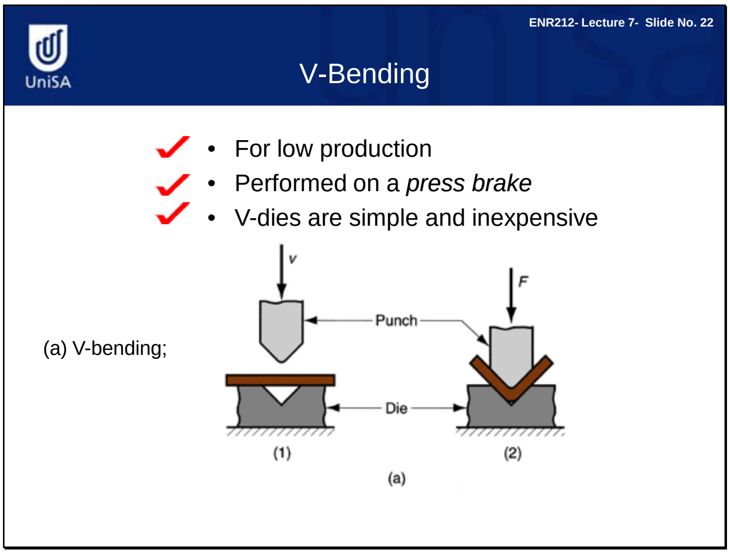
ENR212 Lecture 7 Slides and Notes

Slide 21



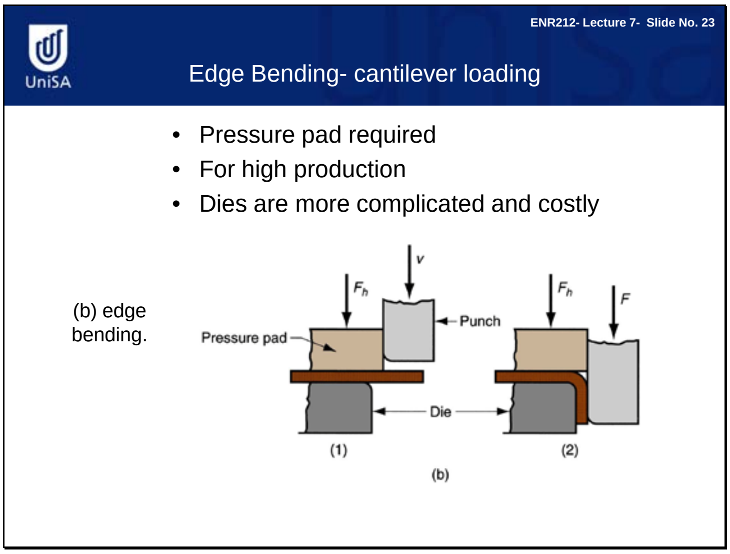
In V-bending, a sheet metal is bent between a V-shaped punch and die. Edge bending involves cantilever loading of the sheet metal.

Slide 22



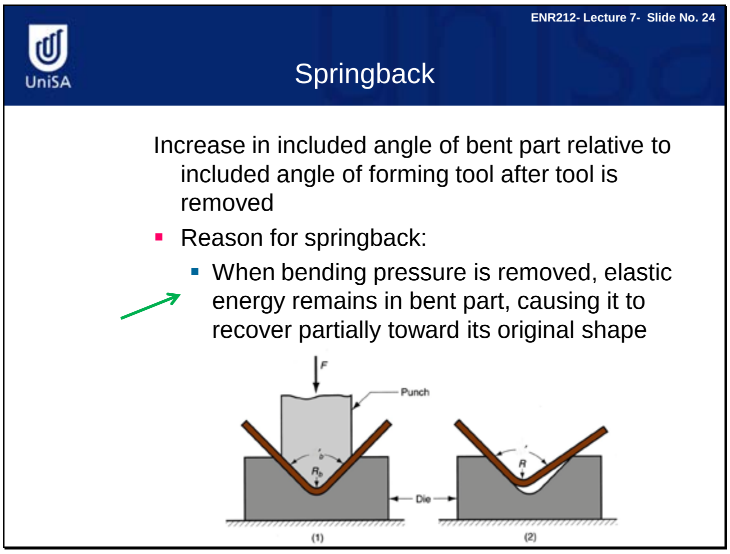
V-bending is generally used for low-production operations. It is often performed on a press brake, and the associated V-dies are relatively simple and inexpensive.

Slide 23



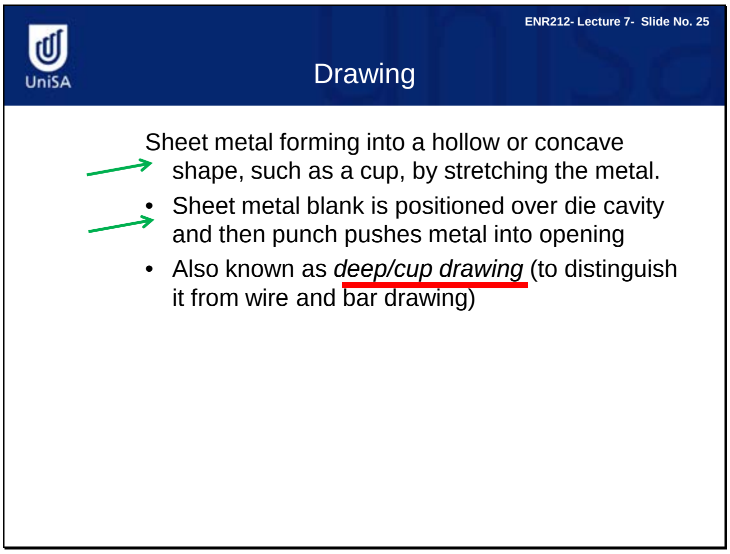
As we have stated, edge bending involves the cantilever loading of a sheet metal. So in edge bending, a pressure pad is used to apply a force to hold the work part against the die, while the punch forces the part to yield and bend over the edge of the die. Because of the pressure pad, the dies are more complicated and costly than V-dies, and are generally used for high-production work.

Slide 24



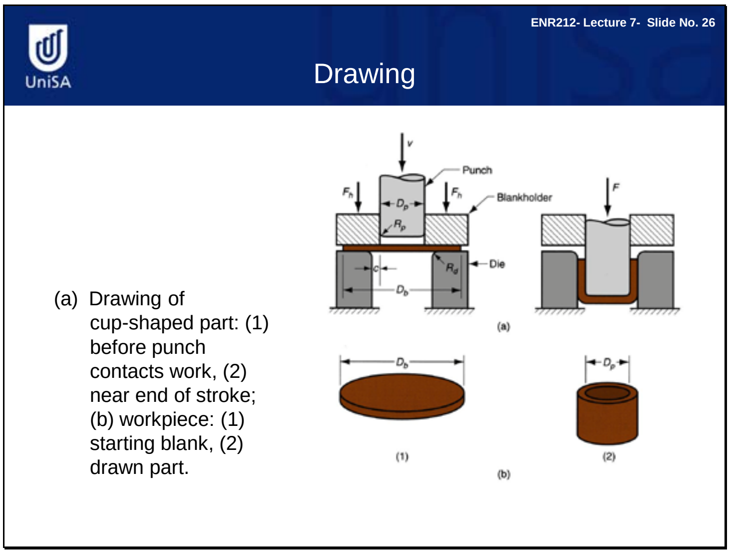
After the punch is removed, the angle increases due to elastic deformation. The increase in the angle is called springback. Springback is illustrated in this figure.

Slide 25



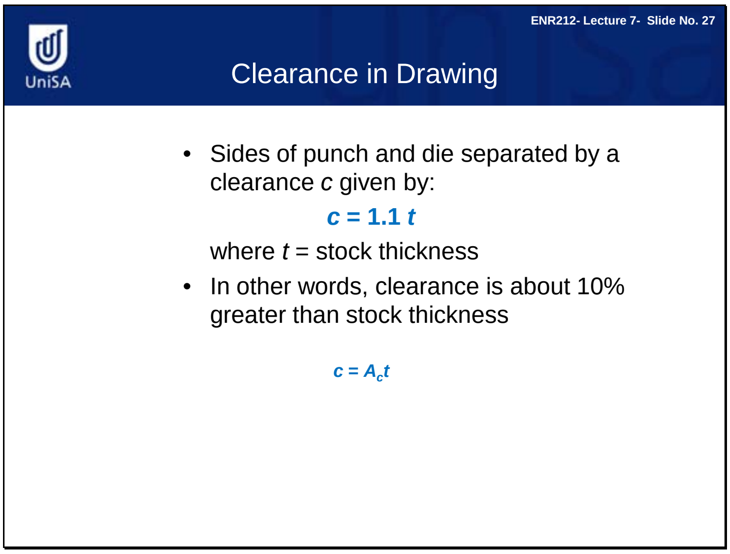
Drawing refers to the forming of a flat metal sheet into a hollow or concave shape, such as a cup, by stretching the metal. The sheet metal blank is positioned by the blankholder over the die cavity and then the punch pushes metal into the opening.

Slide 26



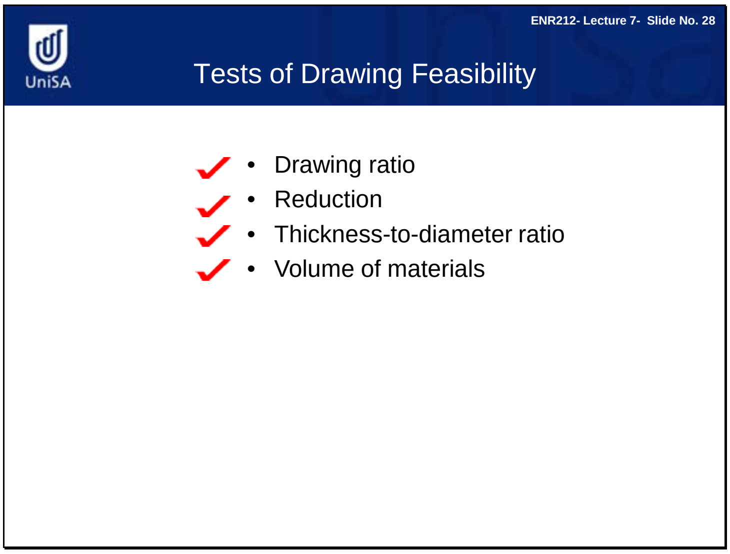
This slide shows you a drawing process. You can see the metal flow is a characteristic of drawing.

Slide 27



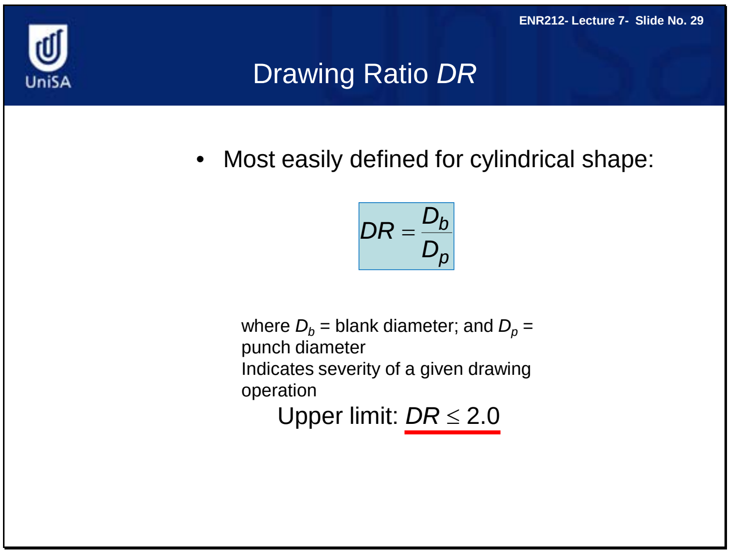
The clearance in drawing is about 10% greater than the stock thickness.

Slide 28



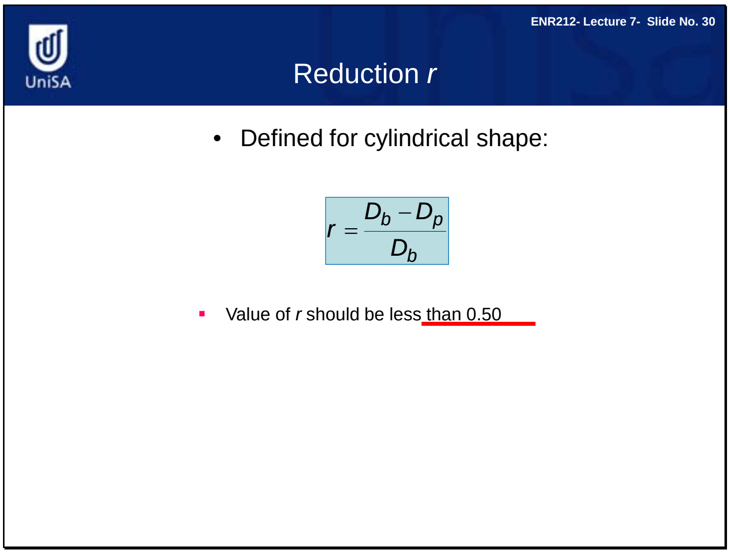
The drawing feasibility of a metal piece is determined by these four conditions: the drawing ratio, reduction, thickness-to-diameter ratio, and the volume of materials.

Slide 29



For a cylindrical shape, the drawing ratio is defined as the ratio of blank diameter to punch diameter. The drawing ratio should be larger than 2.0 for the operation to be feasible. The equation for calculating the drawing ratio is DR = blank diameter divided by punch diameter.

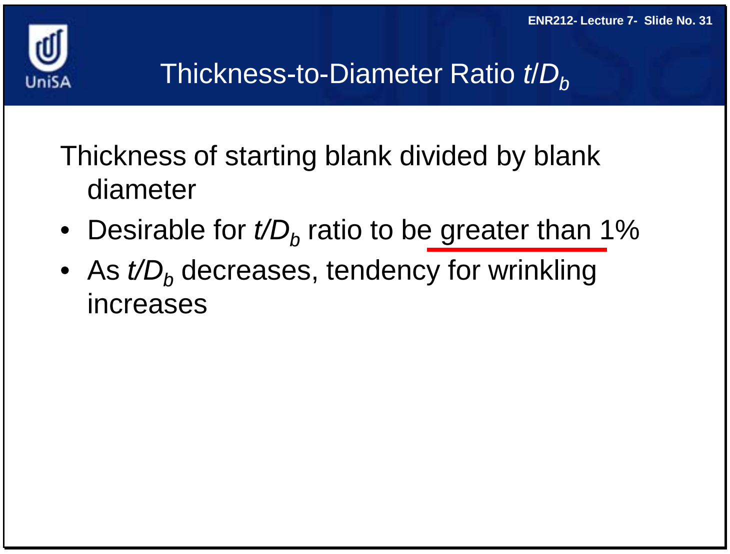
Slide 30



The reduction for a drawing operation can be obtained from this equation. The reduction is equal to the blank diameter minus the punch diameter, all divided by the blank diameter. The value of the reduction should be less than 0.5 for the operation to be feasible.

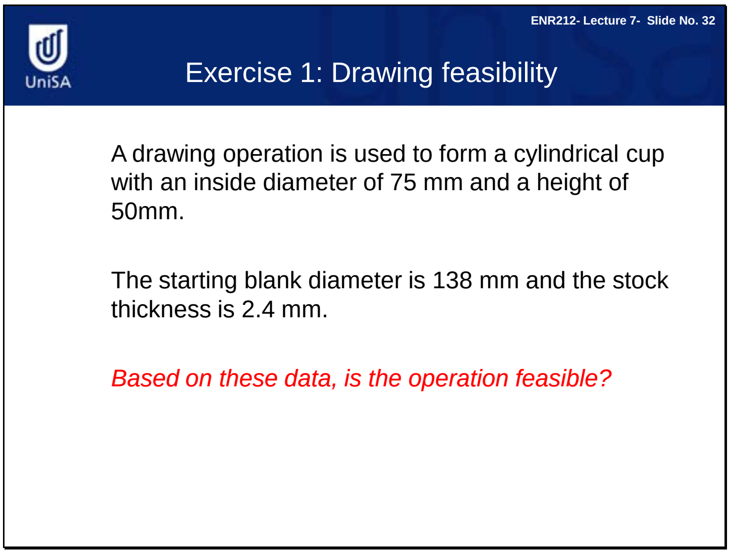
ENR212 Lecture 7 Slides and Notes

Slide 31



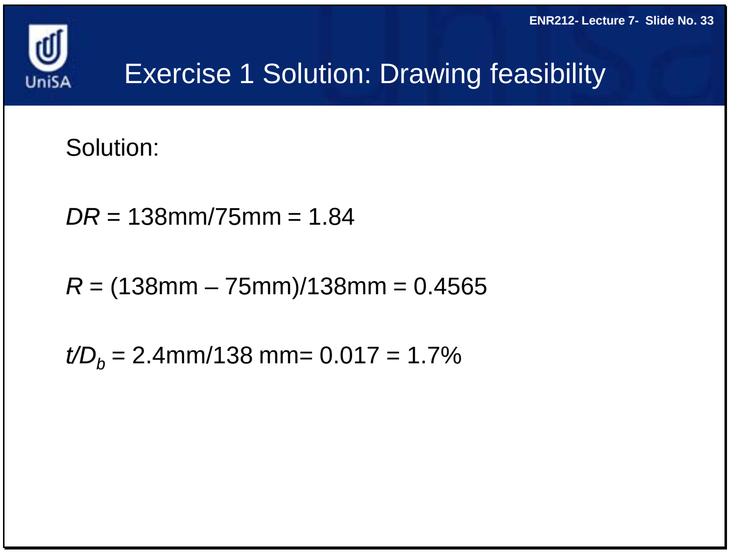
In deep drawing, the thickness-to-diameter ratio of the starting piece must be greater than 1.0 for the operation to be feasible. The thickness to diameter ratio is calculated by dividing the thickness of the starting blank by the blank diameter.

Slide 32



Now it’s time to attempt a drawing operation calculation. If you feel confident, you can try to work this out yourself (you may need to look back over the last few slides in this presentation). Just pause the presentation, and work out the answer, and then check your answer against the solution on the next slide.

Slide 33



The problem asks you to consider a drawing operation which is used to form a cylindrical cup with inside diameter = 75 mm and height = 50 mm. The starting blank diameter equals 138 mm and the stock thickness equals 2.4 mm. For the operation to be feasible, there are three parameters to consider: the drawing ratio, the reduction, and the thickness-to-diameter ratio. Let's look at each of these in turn. Compare your answers to ours:

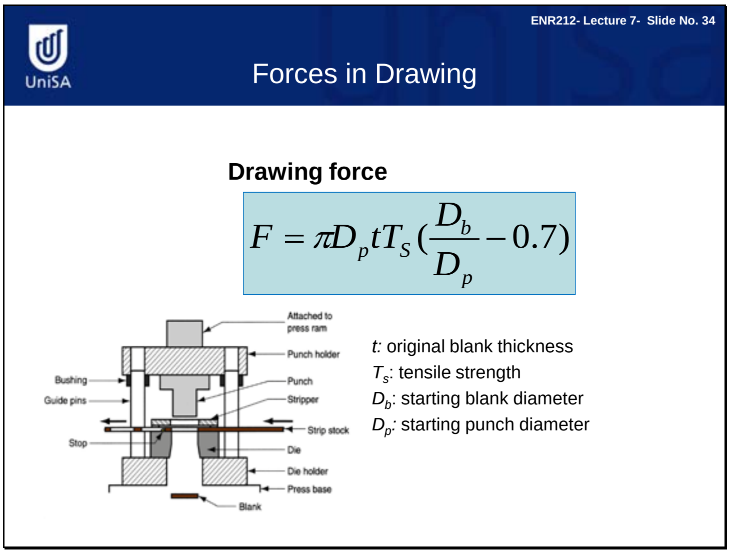
*The drawing ratio (DR) is the ratio of blank diameter to punch diameter, and needs to be under or equal to 2.0. By using the Drawing Ratio equation, which states that the DR equals the blank diameter divided by punch diameter, we get a DR of 1.84. This is under 2.0, so the drawing ratio for this scenario is feasible.

*The reduction is calculated by subtracting the punch diameter from the blank diameter, and then dividing the result by the blank diameter. The reduction must be less than 0.5 for the operation to be feasible. Using the reduction equation, we get a reduction here of .4565, which is less than .5, so the reduction for this scenario is feasible.

*Finally, the thickness to diameter ratio needs to be greater than 1% for the operation to be feasible. We calculate the thickness to diameter ratio by dividing the thickness of the starting blank by the blank diameter. In this case, that equals 1.7%, which is greater than 1%, so the thickness to diameter ratio for this scenario is feasible.

Because all three conditions are met, the operation is feasible.

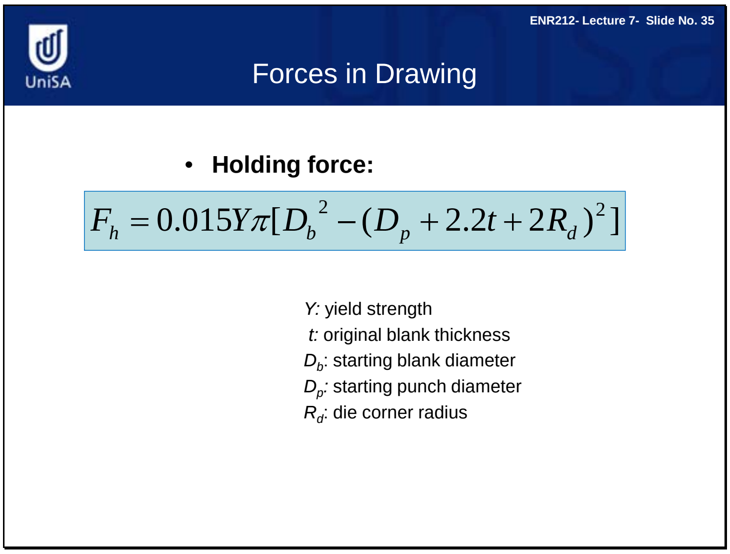
Slide 34



You can roughly estimate the drawing force required to perform a given operation by the formula given on this slide.

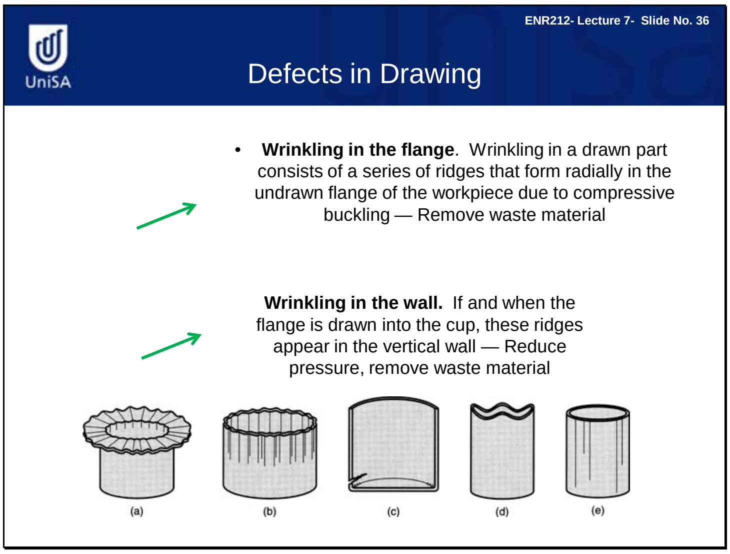
ENR212 Lecture 7 Slides and Notes

Slide 35



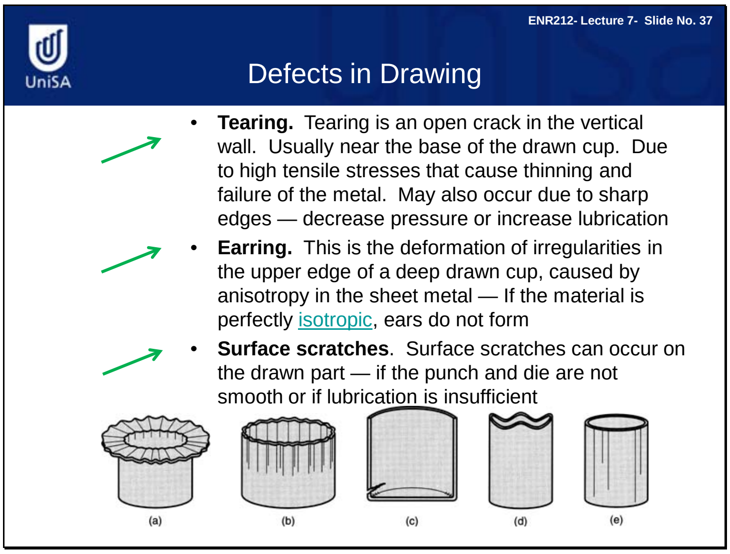
The holding force in drawing is given by this equation. In some of the assessment activities for this unit, you will be asked to calculate the holding force and compare your calculation with the measured value.

Slide 36



There are five major defects in drawing. The first is wrinkling in the flange. When we talk of wrinkling in a drawn part, we mean a series of ridges that form radially in the undrawn flange of the workpiece, due to compressive buckling. This defect can be fixed up by removing waste material. The second defect is wrinkling in the wall. This can be cured by reducing pressure and removing waste material.

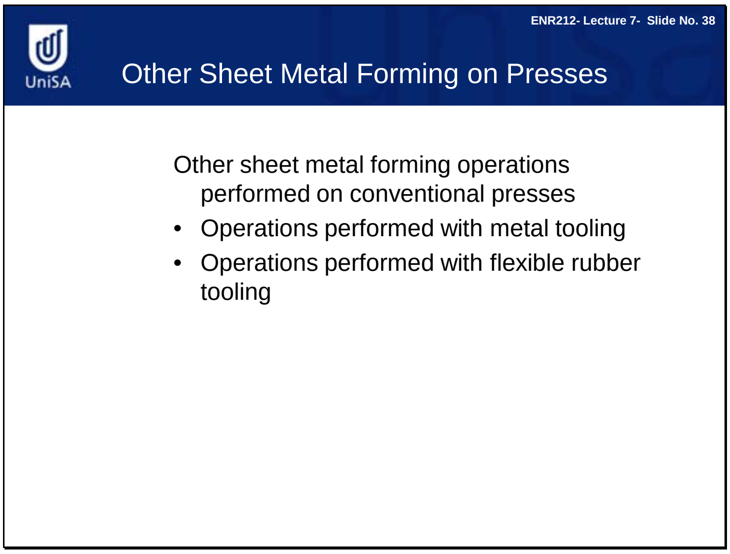
Slide 37



The third defect is tearing. Tearing produces an open crack in the vertical wall, usually near the base of the drawn cup. It is produced by high tensile stresses that cause thinning and failure of the metal. It can also happen because of sharp edges. The answer is to decrease pressure or increase lubrication. The fourth defect is earring. This is a deformation which produces irregularities in the upper edge of a deep drawn cup. It is caused by anisotropy in the sheet metal. If the material is perfectly isotropic, ears do not form.

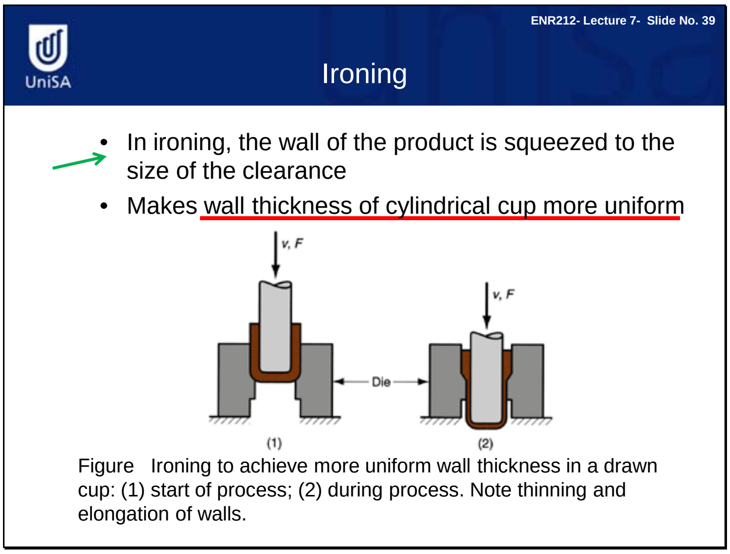
The fifth and final defect is surface scratches. Surface scratches can occur on the drawn part if the punch and die are not smooth, or if there is insufficient lubrication.

Slide 38



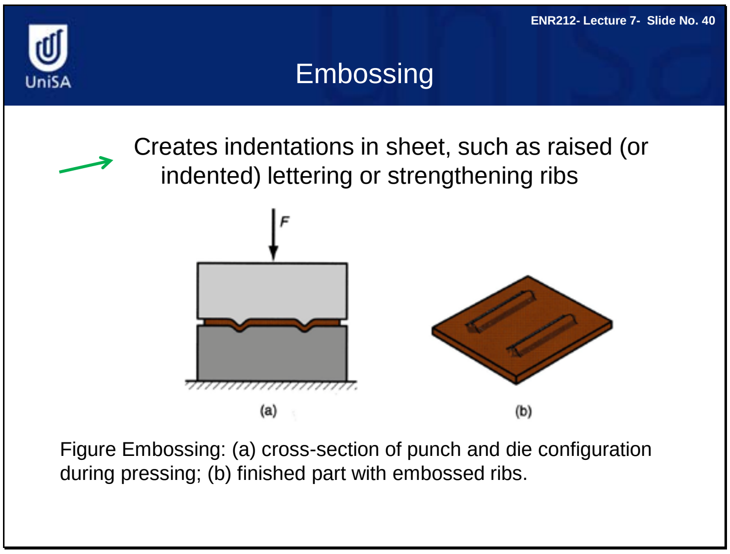
We have learnt about the three major types of sheet metalworking, which are cutting, bending and drawing. However, there are other sheet metal processes and some of these operations are performed with metal tooling, in the same way as the three major types. There are also operations performed with flexible rubber tooling. (Rubber is a type of polymer.)

Slide 39



There is sometimes a problem with the deep drawing process. Because the flange is compressed by the squeezing action between the punch and the die, its perimeter becomes smaller as it is drawn towards the die opening, and the wall thickness obtained is often not uniform. Ironing is a process which doesn't have this problem, because the wall of the product is squeezed to the size of the clearance, so the wall thickness is more uniform than in deep drawing.

Slide 40



Embossing is a forming operation used to create indentations in the sheet. It involves some stretching and thinning of the metal.

ENR212 Lecture 7 Slides and Notes

Slide 41

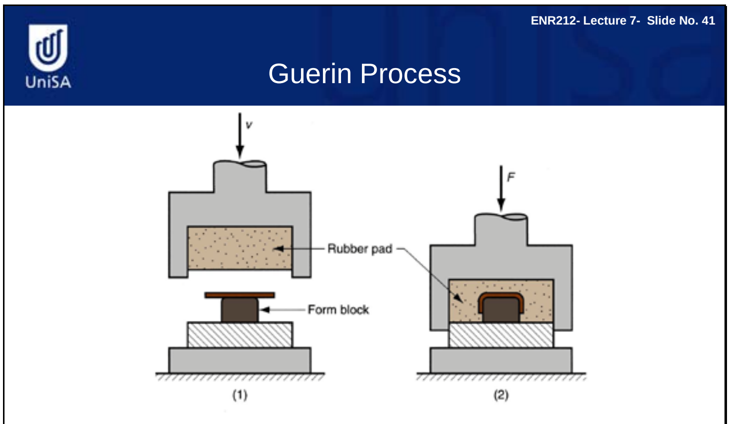
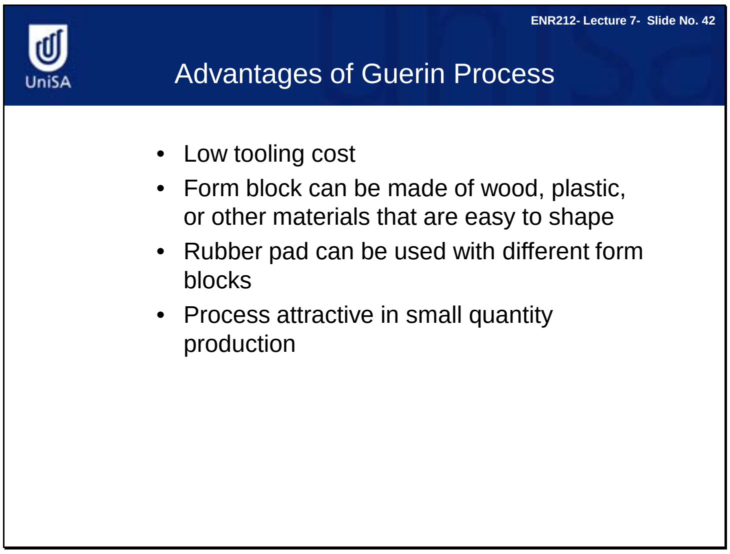


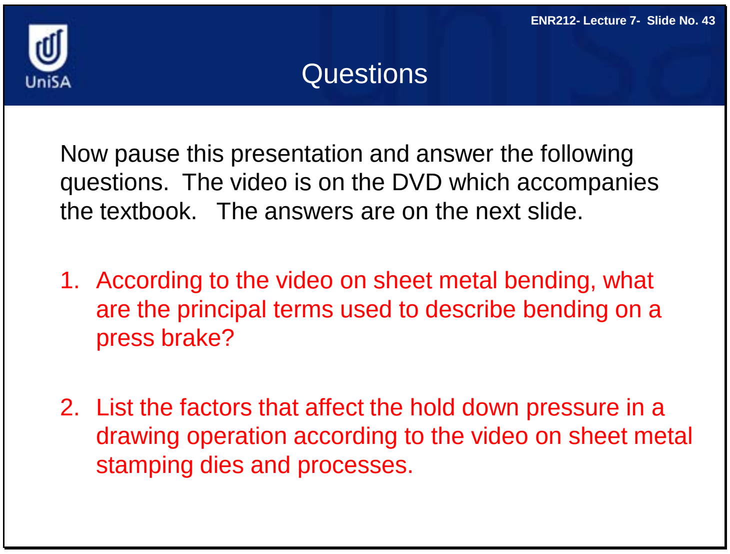
Figure Guerin process: (1) before and (2) after. Symbols *v* and *F* indicate motion and applied force respectively.

For all of the processes we have discussed on the last few slides, the tooling is made of metal. Now we will look at a tool which is not made of metal. The tooling in the Guerin Process is made of a thick rubber pad, which is confined in a steel container. As the ram descends, the rubber gradually surrounds the sheet, applying pressure to deform it to the shape of the form block. The form block can be made of wood, plastics, or other materials that are easy to shape. This type of tooling is relatively low cost.

Slide 42

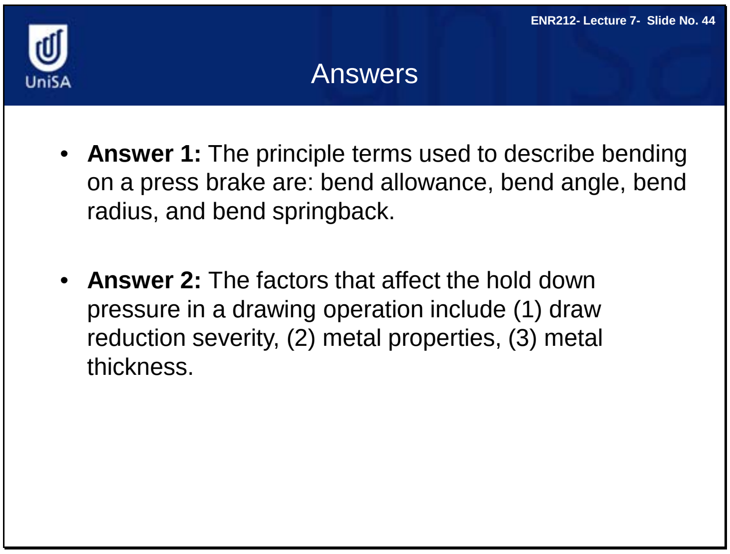

ENR212- Lecture 7- Slide No. 42
UniSA
Advantages of Guerin Process
• Low tooling cost
• Form block can be made of wood, plastic, or other materials that are easy to shape
• Rubber pad can be used with different form blocks
• Process attractive in small quantity production

Slide 43

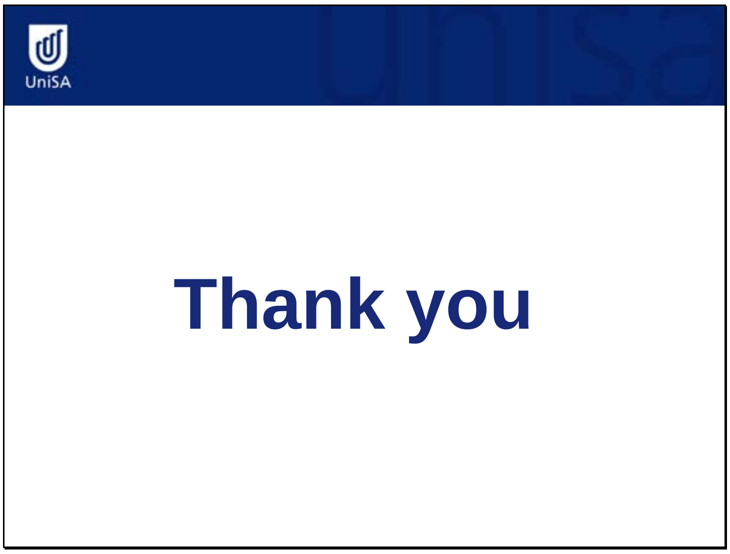


Now pause this presentation, and watch the DVD which came with the textbook (the section on Sheet Metal). When you have watched the DVD, answer these questions. The answers are on the next slide.

Slide 44


ENR212- Lecture 7- Slide No. 44
UniSA
Answers
Answer 1: The principle terms used to describe bending on a press brake are: bend allowance, bend angle, bend radius, and bend springback.
Answer 2: The factors that affect the hold down pressure in a drawing operation include (1) draw reduction severity, (2) metal properties, (3) metal thickness.

Slide 45



Thanks for your attention