FINITE ELEMENT MODELING SWIFT SHEET METAL FORMABILITY TEST

MOHAMAD FAIZUL BIN KHORUDDIN

Thesis submitted in partial fulfillment of the requirements
for the award of the degree of
Bachelor of mechanical engineering

Faculty of Mechanical Engineering
UNIVERSITI MALAYSIA PAHANG

NOVEMBER 2009

CHAPTER 1

PROJECT FRAMEWORK

1.1 SHEET METAL FORMING

Sheet metal is one of the most important semi finished products used in the steel industry, and sheet metal forming technology is therefore an important engineering discipline within the area of mechanical engineering. Sheet metals are characterized by a high ratio of surface area to thickness. Sheet metal forming is basically conversion of a flat sheet metal into a product of desired shape without defect like fracture or excessive localized thinning [1].

In automobiles the sheet metal is deformed into the desired and brought into the required form to get auto body pressings like bonnet, bumpers, doors, etc. In aircraft's sheet metal is used for making the entire fuselage wings and (body). In domestic applications sheet metal is used for making many parts like washing machine body and covers, iron tops, timepiece cases, fan blades and casing, cooking utensils etc [1].

The products made by sheet-forming processes include a large variety of shapes and sizes, ranging from simple bends to double curvatures with shallow or deep recesses. Typical examples are metal desks, appliance bodies, aircraft panels, beverage cans, auto bodies, and kitchen utensils. In many cases while deforming the sheet metal, the component fractures at certain point. The causes of failure are parameters related to forming process [1].

Traditional evaluation of formability is based on both intrinsic tests and simulative tests. The intrinsic tests measure the basic characteristic properties of materials that can be related to their formability. These tests provide comprehensive information that is insensitive to the thickness and surface condition of the material. Examples of intrinsic tests are Uniaxial tensile test, Plane strain tensile test, Marciniak Biaxial Stretching test, Hydraulic Bulge test, Marciniak In-Plane Sheet torsion test, Miyauchi shear test, Hardness test. The simulative tests subject the material to deformation that closely resembles the deformation that occurs in a particular forming operation. Examples of these tests include Ericksen , Olsen, Fukui, Swift tests[1].

1.2 PROJECT BACKGROUND

Finite element modeling (FEM) of Swift sheet metal formability test is important in sheet metal forming process. In this research, FEM is used to monitor the Swift cupping test.

The sheet metal is a very important thing nowadays, because of many applications use this material. Even though, there are limitation of sheet metal formability and many cases while deforming the sheet metal, the component fractures at certain point. The causes of failure are parameters related to forming process. The project is expected to find the sheet formability by using finite element method. The project is using finite element modeling of Swift Sheet Metal Formability Test to investigate the ability of sheet metal to be shaped.

This research will overcome this problem with finite element analysis with find the elastic and plastic deformation of sheet metal with four different materials with same thickness. The aluminum, brass, steel and iron are the variables of this simulation with the constant thickness.

.

1.3 PROBLEM STATEMENT

In this project, this question will be answered:

i. Sheet metal forming is complex process because it involves many stress and plastic deformation.
ii. Swift formability test can be used to investigate the formability state of sheet metal.
iii. In addition finite element can used as investigate tool.

1.4 PROJECT OBJECTIVE

The objectives of this thesis are as follow:

i. To developed working model finite element model of swift formability test.
ii. To investigate elastic and plastic material model in the finite element.
iii. To investigate different element type of plane stress and axisymmetric.

1.5 OUTCOME PROJECT

The outcomes of this project are:

i. Access edge movement in swift forming using plasticity and elastic module.
ii. Simulation of finite element method with different element type and material method

1.6 PROJECT SCOPES

This project will be focus on: Swift Formability die design

i. Swift formability test
ii. The diameter 50mm of punch
iii. Diameter of 127mm of sheet metal
iv. Finite element modeling with using ALGOR
v. Element type: Plane stress and Axisymmetric
vi. Element type: 2-D
vii. Material model: Elasticity elastic (Isometric) and Plasticity plastic (von Mises with Isotropic Hardening)
viii. Prescribed movement: -50mm
ix. Used common 1 mm thickness of sheet metal
    - Steel AISI 1010 annealed
    - Brass,red
    - Aluminum 6063 –T6

CHAPTER 2

LITERATURE REVIEW

2.1 FINITE ELEMENT ANALYSIS

FEA consists of a computer model of a material or design that is stressed and analyzed for specific results. It is used in new product design, and existing product refinement. A company is able to verify a proposed design will be able to perform to the client's specifications prior to manufacturing or construction. Modifying an existing product or structure is utilized to qualify the product or structure for a new service condition. In case of structural failure, FEA may be used to help determine the design modifications to meet the new condition [2].

There are generally two types of analysis that are used in industry: 2-D modeling, and 3-D modeling. While 2-D modeling conserves simplicity and allows the analysis to be run on a relatively normal computer, it tends to yield less accurate results. 3-D modeling, however, produces more accurate results while sacrificing the ability to run on all but the fastest computers effectively. Within each of these modeling schemes, the programmer can insert numerous algorithms (functions) which may make the system behave linearly or non-linearly. Linear systems are far less complex and generally do not take into account plastic deformation. Non-linear systems do account for plastic deformation, and many also are capable of testing a material all the way to fracture [2].

2.1.1 Material Properties

This section refers to material types used to model; there are two types of isotropic material involved in this model: linear elastic-plastic (for sheet metal) and linear elastic (for the punch and die). The basic material properties required for this model are given in Table 1

Table 2.1: Material Properties of the Model [3]

Sheet Metal (Aluminum) Linear Elastic - Plastic		
Material Property	Symbol	Property Value
Young Modulus	E	78 GPa
Uniaxial yield strength	G_y	550MPa
Poison's Ratio	N	0.3
Material density	P	2700 kg/m^3
Punch and Die (Steel) Linear Elastic		
Young Modulus	E	78 Gpa
Uniaxial yield strength	G_y	4000 Mpa
Poison Ratio	v	0.3
Material density	p	7800 kg/m^3

2.1.2 Nonlinear Analysis

Linear finite element analysis assumes that all materials are linear elastic in behavior and that deformations are small enough to not significantly affect the overall behavior of the structure. While nonlinear analysis is much more complicated than simple linear analysis because it is required many variables such as changes in geometry, permanent deformations, structural cracks and buckling. A fundamental difference between geometrically linear and geometrically nonlinear analysis is that in linear analysis equilibrium is satisfied on the initial un-deformed configuration, whereas in nonlinear analysis equilibrium must be satisfied in the deformed configuration. Nonlinear geometric and material effects may be incorporated in this analysis. To achieve final equilibrium in a nonlinear analysis, we solve the model many times, constantly adjusting the applied forces based on the current state of equilibrium, and modifying the geometry based on the current displacements. Convergence nonlinear analyses are generally solved by an iterative procedure and the level of convergence reached tells us the likely error in the solution. Not all nonlinear analysis was guaranteed to converge but often convergence can be helped along by modifying some of the parameters. For nonlinear analysis, since it is no longer possible to directly obtain results according to external loads, a solution procedure is usually adopted in which the total required load is applied in a number of increments. The analysis allows you to set any desired load level for each increment or time step of a nonlinear solution. For the analysis of nonlinear problems, the solution procedure adopted may be of significance to the results obtained. In order to reduce this dependence, wherever possible, nonlinear control properties incorporate a series of generally applicable default settings, and automatically activated facilities [3].

2.2 DRAWING

Formability of sheet metal generally understood to mean capability of being extensively deformed into desired shape without any fracture or defects in the finished part. This process is called deep drawing or press forming. The deep drawing process is the most common sheet metal forming method. Generally, this process is involved with a flat blank is formed into a finished shape between a pair of matched dies. A blank of sheet metal is restrained at the edges, and the middle section is forced by a punch into a die to stretch the metal into a cup shaped drawn part. This drawn part can be circular, rectangular or just about any cross-section. Drawing can be either shallow or deep depending on the amount of deformation. Shallow drawing is used to describe the process where the depth of draw is less than the smallest dimension of the opening; otherwise, it is considered deep drawing.

Other forming methods exist, but in all of them two principal kinds of deformation, drawing and stretching are involved [4]. This drawing process, the clamping force of the hold down dies is just sufficient to permit the material to flow radically into the die cavity without wrinkling. Drawing leads to wrinkling and puckering at the edge where the sheet metal is clamped. This is usually removed by a separate trimming operation.

2.3 STRETCHING

A wavy piece of sheet metal can be straightened by gripping it at two opposite sides and stretching it a bit beyond its elastic limit. It will come out straight, flat and undistorted. Sheet metal so treated by a mill is termed stretcher leveled, and is supplied to meet flatness requirements [5].

Parts blanked from aluminum and other soft metals can also be stretched if product designers and tooling designers agree that it is needed. In the operation sequence, one or more stations before the blanking station, a station is provided for impressing a grid of closely-spaced pinpoint indentations on both sides of the material. These indentations, although tiny, shallow and barely visible, have the same effect as stretching. They flatten and stiffen the metal in the area where the blanking will occur, so that the finished part will have the required flatness, yet with the indentations hardly noticeable [5].

The periphery of the tray has a short vertical rim and a narrow horizontal flange. When this part is draw-formed, an unequal compressive stress builds up along the rim and flange, and continues down into the bottom panel. This stress sometimes warps a part like this and introduces the objectionable "snap" action. To avoid this defect, the bottom of the tray is stretched a little in a way that cancels the compression stress there. This gives the metal a new granular orientation, eliminates the objectionable twist and allows the tray to lie steady on the table [5].

The stretching that is introduced could be designed into the drawing die, or it could be done as a second operation after the part is completed. In the first method, the blank is drawn and the bottom stretched in one operation. In the second, the drawing only is performed in the first of two dies. The stretching is done later in another die. Both methods produce the same results. Depending upon the ingenuity and discretion of the die designer, the stretching could comprise a pattern of shallow rectangular depressions pressed into the bottom panel to stretch the excess metal, remove stress, and prevent it from twisting. Unfortunately, it is the nature of these procedures to render unpredictable results, so they must be worked out experimentally in each case [5].

2.4 SWIFT CUP

This test simulates the drawing operation and involves drawing of a flat or hemispherical bottom parallel side cup. The sheet is held under a blank holder, but is well lubricated with polyethylene and oil and ensures that the blank can be drawn in under the blank holder. Typical Swift cup test forming tolls are available in 19, 30 and 50 mm diameters for use with specimens ranging in thickness form 0.3 to 1.24, 0.32 to 1.30, 0.45 to 1.86 mm, respectively. For drawing 40 mm square cups from 80 mm diameter round specimens from 0.2 to 2 mm thick, a 40 mm square forming tools is recommended [6].

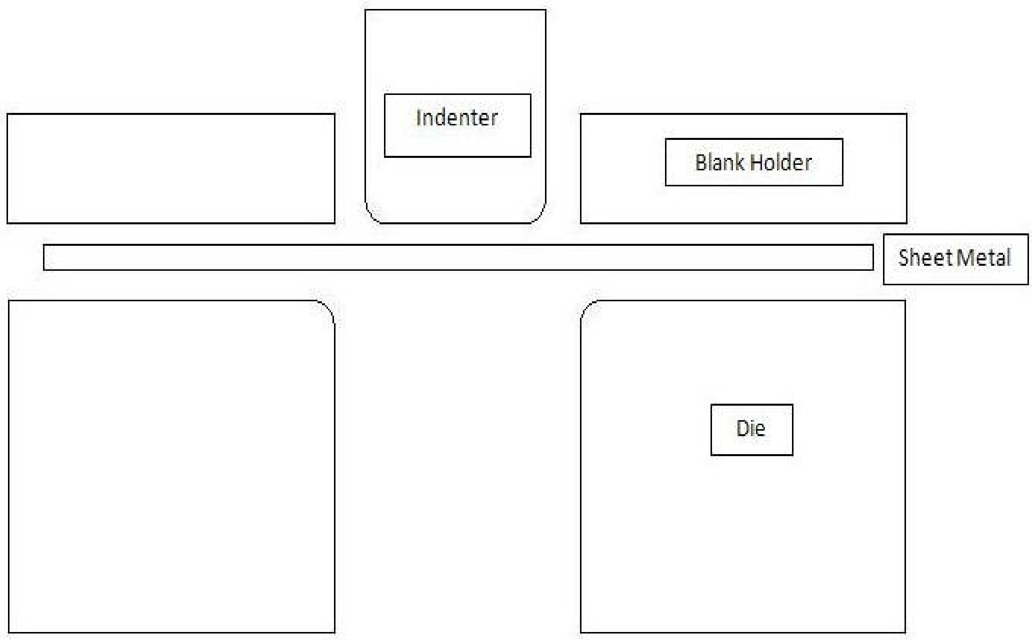

Figure 2.1 Swift Forming [6]

Figure 2.2: Swift Cup [6]

The draw ability of the metal is estimated by drawing a series of blanks of increasing diameter. The maximum blank size that can be drawn without fracture occurring over the punch nose is used to calculate the limiting draw ratio. For example, forming a 66 mm diameter disk using a 33 mm forming tool provides an LDR of 2.0. Because the condition of the edge of each blank can have an important effect on the test result, the blank edges usually are turned in a lathe to ensure strain-free, hurt-free edge [6].

The result of this test correlates well with the performance of sheet metal in deep drawn components, but because of shape and alignment, reproductility between laboratories is not good. The main problem with this test, however is that it is time consuming, and a large number of blanks of different sizes must be tested to obtain a reliable result[6].

Apart from measuring draw ability, this test also can be used as a quality control check to measure the tendency toward earring of the sheet metal. In case, a blank of fixed diameter is drawn, and the height between the peaks and through in the cup wall are measured [6].

The Englehardt or draw fracture test is variation of the Swift cup test for measuring draw ability to overcome the problems of complexity and time involved in the test. The draw fracture test involve drawing of a cup to the points of maximum drawing load, then clamping of flange and continuing the punch travel to fracture [6].

This result depends on strips thickness and usually is corrected, using a empirical relationship, to a nominal thickness. Because of its simplicity of operation and reproducibility, the draw fracture test is the most suitable for testing of draw ability on a routine basis [6].

2.5 ALGOR SIMULATION

2.5.1 Plane Stress

Plane stress is defined to be states of stress in which the normal stress, σ_z and shear stress τ_{xz} and τ_{yz} directed perpendicular to the x-y plane are assume to be zero [7].

The geometry of the body is essentially that of a plate one dimension much smaller than the others. The loads are applied uniformly over the thickness of the plate and act in the plane of the plate as shown. The plane stress condition is the simplest form of behaviour for continuum structures and represents situations frequently encountered in practice.

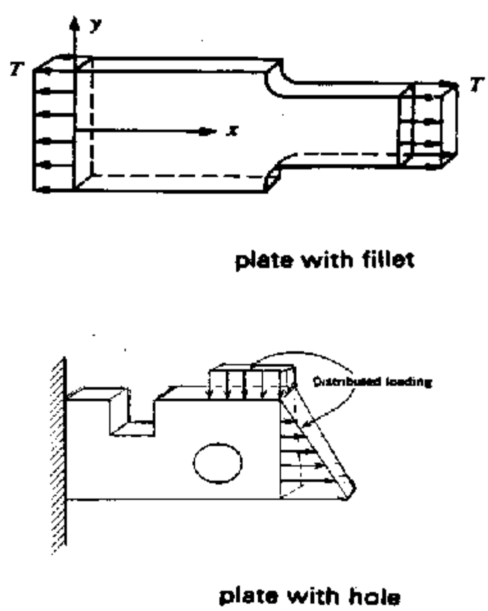

Figure 2.3: Plane Stress [7]

2.5.2 2D Element Type

2-D elements are 3- or 4-node isoparametric triangles or quadrilaterals which must be input in the global Y-Z plane. Figure 1 shows some typical 2-D elements. The element can represent either planar or axisymmetric solids, as illustrated in Figure 2. In both cases, each element node has two translational degrees of freedom [8].

When the element is used to represent an axisymmetric solid or shell, the global Z-axis is the axis of revolution. All elements must be located in the +Y half-plane where Y is the radius axis. Figure below illustrates these conventions [8].

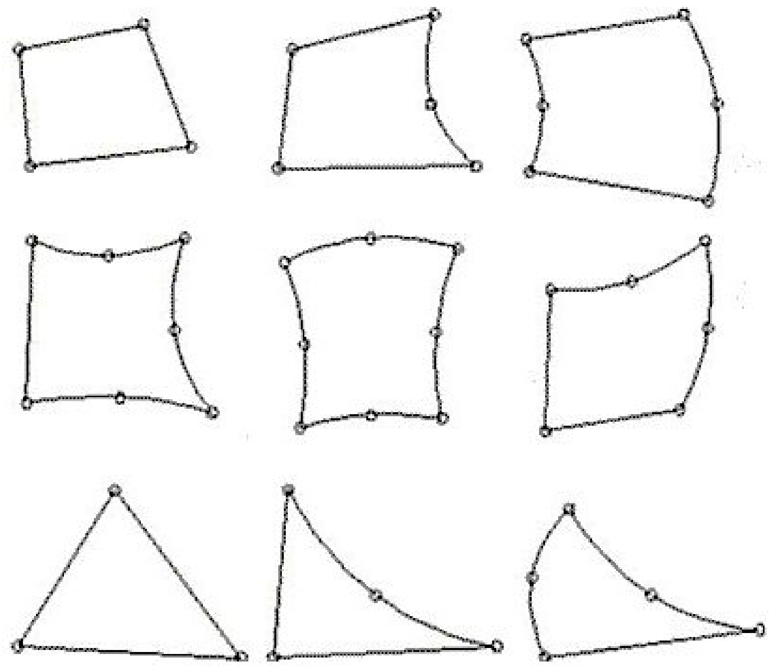

Figure 2.4: Node Configuration for 2-D Elements [8]

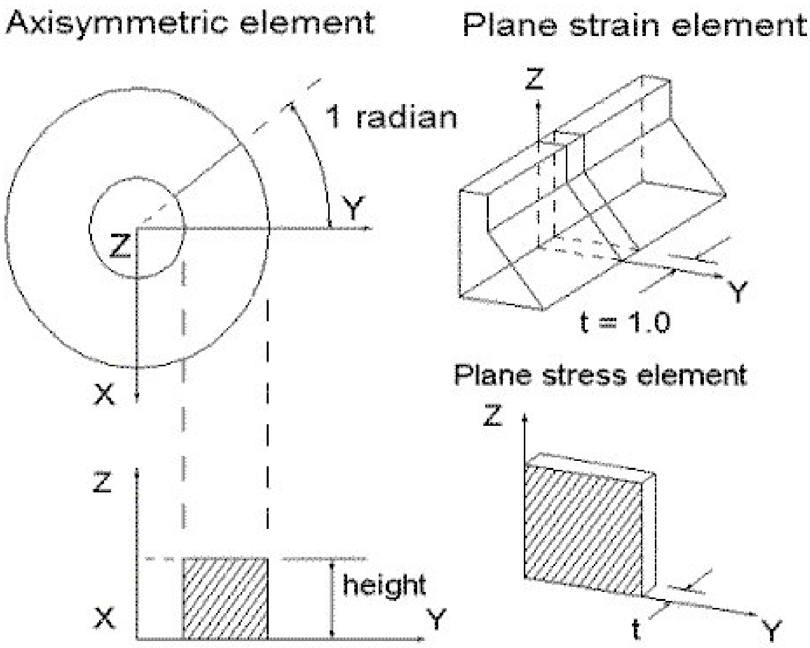

Figure 2.5: Applications of 2-D Elements [8]

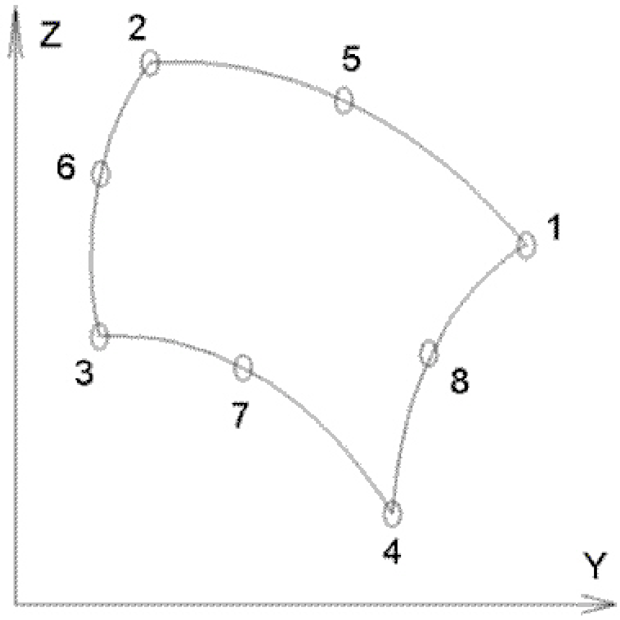

Figure 2.6: Sample 2-D Element [8]

2.5.3 Surface to Surface Contact

In order for loads to be transferred between elements, the nodes must be connected together. For example, if two bodies begin an analysis separated, no interaction will occur during the analysis. The bodies will pass through each other [8].

Surface-to-surface contact in Mechanical Event Simulation and nonlinear stress analysis (but not natural frequency analysis) allows you to create pairs of surfaces that may come into contact with each other during the analysis thereby connecting the nodes on the surfaces together. The processor will determine the distance between the nodes on this surface at each time step of the analysis. When the nodes are sufficiently close to each other, a force will be applied to prevent penetration [8].

Before the user starts a contact analysis, he or she must clearly identify where the contact interaction might occur during the analysis. (Although possible, it would be very inefficient to define every part, every surface as contacting every other part, every surface.) Not only could multiple target surfaces interact with one master surface, but self contact is also possible in a large deformation problem such as a rubber elasticity analysis. In such cases, the user must define multiple contact pairs that cover all potential contact interaction [8].

The contact pairs can consist of any two arbitrary surfaces. Surface-to-surface contact will connect the nodes on one surface to the faces of the other surface (and optionally vice versa). However, in order to speed up the contact search, the user should only specify the contact pairs that will definitely interact within the given event duration. Especially in problems that involve small amounts of sliding contact, the analysis will converge more rapidly if the number of contact elements is minimized. This can be achieved by specifying a contact radius, which will ensure that all generated contact elements have a length that is initially shorter than the contact radius. Provided the contact radius is larger than the distance the parts move relative to each other, contact will be maintained over the entire range of motion [8].

To define where contact can occur, put the surfaces to come into contact on a unique surface number, namely, the highest surface number of any of the lines making the elements that come into contact. To determine which of the six possible sides of a brick element are in contact, the solver checks the surface number of each line making an element. Each face that has a majority of these lines (3 of 4 sides, or 2 of 3 sides) on the highest surface number can participate in contact. Faces whose lines are not on the highest surface number on the element cannot participate in contact [8].

For cases where it is difficult for the user to predict the relative motion of contact pairs, the processor provides an automatic updating scheme to help the user set up the contact pairs efficiently with only a few contact surfaces covering the entire contact area. However, this may require a large amount of memory in a 3-D analysis. If the necessary memory is not available, the user must split the large contact surface into several smaller contact surfaces [8].

2.5.4 Node

A node is a coordinate location in space where the degrees of freedom (DOFs) are defined. The DOFs for this point represent the possible movement of this point due to the loading of the structure. The DOFs also represent which forces and moments are transferred from one element to the next. The results of a finite element analysis, (deflections and stresses), are usually given at the nodes [8].

In the real world, a point can move in 6 different directions, translation in X, Y, and Z, and rotation about X, Y, and Z. In FEA, a node may be limited in the calculated motions for a variety of reasons. For example, there is no need to calculate the out of plane translation on a 2-D element; it would not be a 2-D element if its nodes were allowed to move out of the plane. The DOF for the generic element types are given in table below [8].

Table 2.2: Degree Of Freedom for Element Type [8]

Element type	Translation			Rotation		
	X	Y	Z	**X**	**Y**	**Z**
Truss, spring, gap	Yes	Yes	Yes			
Beam	Yes	Yes	Yes	Yes	Yes	Yes
2-D		Yes	Yes			
Membrane		Yes	Yes			
Plate, shell	Yes	Yes	Yes	Yes*	Yes*	*
Brick, tetrahedral	Yes	Yes	Yes			

Yes indicates the deflections are calculated. Transmission of forces or moments is supported. * Rotational DOF for plate elements are based on local direction 1, 2, 3 instead of global direction X, Y, Z. Rotations about the local axes 1 and 2 (axes in the plane of the element) are calculated. Rotation about the local axis 3 (axis perpendicular to the element) is not calculated [8].

The DOF of a node (which is based on the element type) also relates what types of forces and restraints are transmitted through the node to the element. A force (axial or shear) is equivalent to a translation DOF. A moment is equivalent to a rotational DOF. Thus, to transfer a moment about a certain axis, the node must have a rotational DOF about the axis. If a node does not have that rotational DOF, then applying a moment to the node will have no effect on the analysis. This fact may also place requirements on how two parts are connected together. Additional modelling may be required to insure that the connection between the parts does not produce a hinge. See the page "Setting Up and Performing the Analysis: Linear: Element Types and Parameters. Combining Element Types" for examples [8].

2.5.5 Element

An element is the basic building block of finite element analysis. There are several basic types of elements. Which type of element for finite elements analysis that is used depends on the type of object that is to be modeled for finite element analysis and the type of analysis that is going to be performed [8].

An element is a mathematical relation that defines how the degrees of freedom of a node relate to the next. These elements can be lines (trusses or beams), areas (2-D or 3-D plates and membranes) or solids (bricks or tetrahedral). It also relates how the deflections create stresses [8].

2.5.6 Prescribed Displacement

A prescribed displacement will cause a node to translate or rotate through a certain distance specified in the magnitude field. This translation or rotation can occur along any vector specified in the direction section. The speed at which this translation or rotation occurs will be controlled by the load curve to which it is applied. If you want to simulate a part accelerating during an analysis, you can create a load curve defined by a second order equation that would represent the displacement of the node under the acceleration. (To have smooth acceleration, the points defined on the load curve must be at a smaller time interval than the smallest time step that occurs during the analysis.)The displacement of the node at any time in the analysis will be related to the magnitude assigned to the prescribed displacement and the multiplier at that time given in the load curve. The prescribed displacement can be removed and re-inserted using the active range. This feature is handy for a variety of situations where the model needs to be "released", such as time dependent boundary conditions. The force required to move the node at the prescribed displacement can be calculated and output [8].

2.5.7 Contact Element

Contact elements allow the user to easily model impact problems. Contact is modeled using contact stiffness. This stiffness is calculated using the modulus of elasticity, a contact area and the length of the element. It should be noted that when the element is longer than this contact length, it has no stiffness. The modulus of elasticity should be representative of the materials making contact [8].

2.5.8 Material Model – Isotropic Hardening

Isotropic hardening describes the increase in material strength due to the plastic strain. The stress – strain response for isotropic hardening is shown in figure 2.7. This figure assume a von Mises yield surface, which define the combination of shear and tensile stress that cause the imitation of plastic deformation

$$\delta = \sigma_y = \sqrt{\sigma^2 + \tau^2}$$

The onset of plastic flow begins at point A. Plastic deformation will cause work hardening in the material as dislocation interacts with each other. If the material is unloaded from point B to zero stress then reloaded the material will yield at new stress $\overline{\sigma_B}$ and plastic deformation will continue along its original stress – strain path. This is termed "material memory "because upon reaching point B during reloading the material 'remembers" it's prior loading. As loading continuous to point C, isotropic hardening considers $\overline{\sigma_c}$ as the new yield strength of the material. If the material is loaded in comparison, yielding will not occur until point D at stress of -δ_c. The yield surface has expanded evenly in all direction during plastic deformation with no charge in shape and no translation of yield surface canter [9].

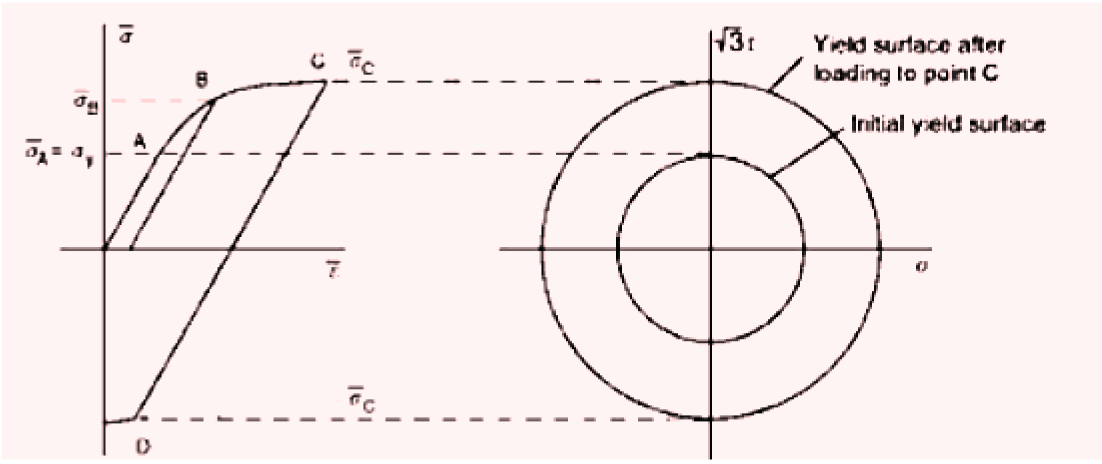

Figure 2.7: Schematic Illustration of Isotropic Hardening [9]

2.6 SHEET METAL DUCTILITY

Conversion of sheet metal into useful shape is a common goal for producers and fabrication. The premise behind all such operation is that the metal be sufficiently ductile to undergo the required that the given degree of residual ductility remain after forming to meet service application [10].

Ductility is a mechanical property that reflects the ability of a material to undergo plastic deformation. Unfortunately, there is no unique testing procedure with which to evaluate ductility. Maximum reduction of area is the most common measure. However, that measurement is difficult, if not impossible, to obtain for sheet metal. Total percentages elongation at fracture is therefore used as an alternative. Here variations in specimen shape, gage, length, and machining technique have a bearing on the result. The value of ductility therefore is influence almost as much by test procedure as by metal quality [10].

Measuring ductility by the amount of plastic deformation before fracture overlooks a practical limitation on the amount of ductility available for many forming operation. Fabrication of some special shape is limited by the onset if instability, or necking, which may lead to severe localization of deformation. This event terminates useful deformation and could be considered, in a broad sense, as a measure of ductility.

In other application, the transition temperature separating ductile and brittle behavior can serve as an index of ductility [10].

Ductility is considered to be the ability of a sheet metal to undergo the plastic deformation require of the forming operation. Measurement of this ductility may include the largest diameter blank that can be successfully drawn into cup of fixed size, the height of the hydraulically bulged dome, or the ratio of tensile stress to yield stress. The practical measure ductility, and often the only measurement that is meaningful, is the performance of sheet metal in the actual forming [10].

CHAPTER 3

METHODOLOGY

3.1 METHODOLOGY

The methodology of the project involving several steps is illustrated in Figure 3.1. The methodology describes the steps in conducting the project from start until its finish. A good methodology can be the guideline in managing the project. In developing a project, methodology is most important element to be considered to make sure that the development of research is smooth in order to get expected result.

In this project, analysis by using ALGOR software V22 is the main step in getting result. Through the analysis, comparison of the result will get. It is important that the analysis that have going through follow the objective and also the project scope. The results also have to achieve the project objective. Some analysis sequence has been to run through the ALGOR software.

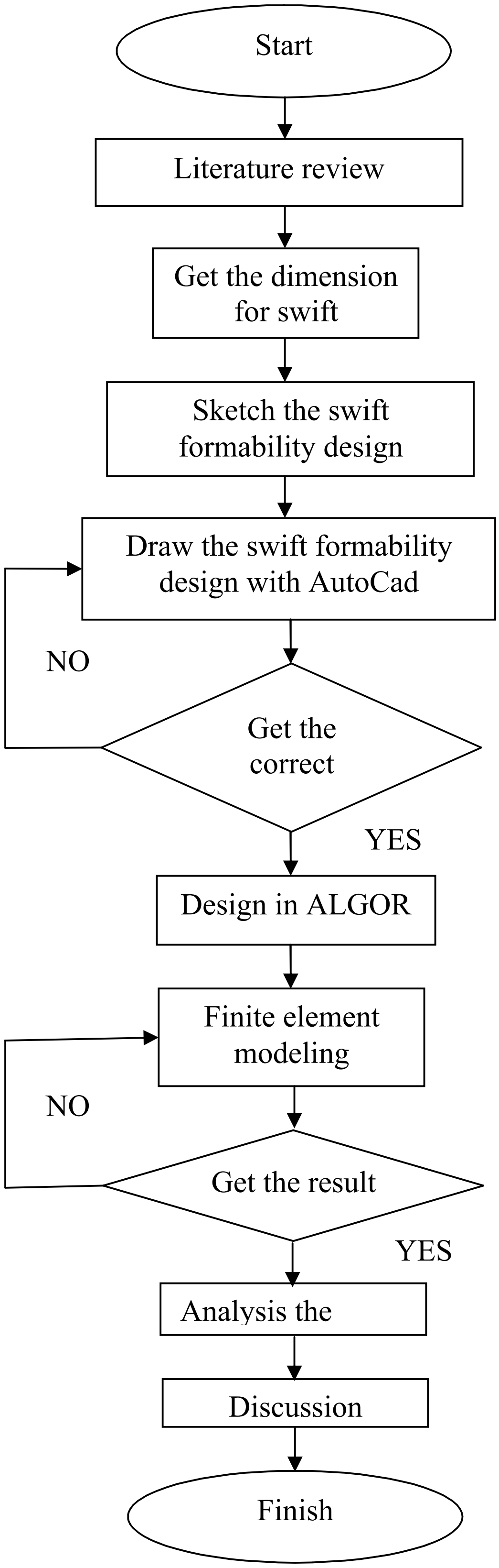

Figure 3.1: Flow Chart

3.2 PROJECT TOOLS

There is the method we use for do this project. This method is very important to make this project run with successfully.

3.2.1 Die Design

The sketching of the die design is further processed when it fulfills the stated criteria and approved by the supervisor. For the first step is the die needs to design using the AutoCAD. The specimens that use for this project are aluminum, brass and steel. The all raw materials came out with same dimension 127 mm of diameter and 1mm thickness. The dies design by using AutoCAD 2008 with designing part by part to making die tools. The die design is constructed with fine line. The sketching also drawn out into 3D solid model by using SolidWork. The drawing is using mm units. The design in making die tools is starting with the punch. The punch designs with 50 mm of diameter. Then follow by the bottom die. The width of the die is 54mm and 50 mm of depth. The die design with use AutoCAD 2008 is shown in figure 3.2 below. The 3-D view of Swift forming is shown in figure 3.3.