CHAPTER 1

INTRODUCTION

1.1 Project Background

The Limiting Dome High Test is used to evaluate press formability of sheet metal. The Limiting Dome Height (LDH) test is a modified hemispherical dome test. Instead of a fully clamped blank, strips of varying widths are clamped on end by a lock bead and then deformed with hemispherical punch. In order to understand the origin of the widely varied results, a sensitivity study was performed using finite element modeling (FEM). In a structural simulation, finite element modeling (FEM) helps tremendously in producing stiffness and strength visualizations and also in minimizing weight, materials, and costs.

1.2 Problem Statement

There is a lot of manufacturing industrial using sheet metal. There are large variety of shapes and sizes, ranging from simple bends to double curvatures with shallow or deep recesses products made by sheet-forming processes. The main problems with sheet metal forming is complex process because it involves many stress state and plastic deformation, so analysis in Limiting dome high test are perform to analyzed the criteria of sheet metal.

1.3 Objective

To perfom simulation of Limiting Dome High Test using Finite Element Analysis. Using Finite Element to access characteristic and formability of differnt sheet metal material using Limiting Dome High test and to investigate the effect of punch dome high and draw beads at edge of sheet metal in Limiting Dome High test.

1.4 Scope of work

There are three scope of work that involve in this project. this is explain the work involves in this project.

1.4.1 Simulation of the LDH test sheet metal forming using Finite Element Package ALGOR

1.4.2 The material that is used Aluminums Alloy 5052-T4, Brass Red, Titanium, AISI 1010 steel fully annealed.

1.4.3 Validation simulation between different material of sheet metal properties of Limiting dome High.

CHAPTER 2

LITERATURE REVIEW

2.1 Finite Element Analysis

The finite element method (FEM) sometimes referred to as finite element analysis (FEA) is a computational technique used to obtain approximate solution of boundary value problems in engineering (David V. Hutton, 2004).

The solution approach in finite element is based either on eliminating the differential equation completely (steady state problems), or rendering the PDE into an approximating system of ordinary differential equations, which are then numerically integrated using standard techniques such as Euler's method, Runge-Kutta, etc. In solving partial differential equations, the primary challenge is to create an equation that approximates the equation to be studied, but is numerically stable, meaning that errors in the input data and intermediate calculations do not accumulate and cause the resulting output to be meaningless. There are many ways of doing this, all with advantages and disadvantages. The Finite Element Method is a good choice for solving partial differential equations over complex domains (like cars and oil pipelines), when the domain changes (as during a solid state reaction with a moving boundary), when the desired precision varies over the entire domain, or when the solution lacks smoothness. For instance, in a frontal crash simulation it is possible to increase prediction accuracy in "important" areas like the front of the car and reduce it in its rear (thus reducing cost of the simulation); Another example would be the simulation of the weather pattern on Earth, where it is more important to have accurate predictions over land than over the wide-open sea (Fix and George, 1973).