Learning Experience in Designing a Dome Test Setup for Sheet Metal Formability Characterization

Monica Dore

Monica Doré currently holds an engineering position at the National Aeronautics and Space Administration (NASA) at Wallops Flight Facility as an Aerospace engineer. Ms. Dore has received an Undergraduate Degree with Honors in Engineering with a Mechanical Specialization from the University of Maryland Eastern Shore (UMES) in December 2014. Prior to receiving her undergraduate degree she worked as an intern with NASA from 2012-2014. She works supporting Airborne Science missions aboard flight platforms such as the P-3 and C-130. She develops mechanical structures for integrating Earth science instruments into NASA aircrafts. The work involves developing, fabricating, and testing flight vehicle structures or their components and recommending optimum configurations, structural design, materials, and techniques. Her work is multi-faceted as she is responsible for completing a design from the concept stages and gathering requirements, to fabrication and assembly. This often entails being the design engineer, the structural analyst, and the fabrication drafter.

Mr. Rodrigo Arturo Ramos, University of Maryland, Eastern Shore

Rodrigo Ramos’ Biography for 2017 ASEE Conference

Rodrigo Ramos, born in Santiago, Chile, is currently the Quality Control Manager at Sonic Tools LP, a business developing and supplying high quality, specialty tooling across a variety of manufacturing industries. He is also in a work committee dedicated to studying the ERP system in place, and correcting processes when necessary. Rodrigo received his Undergraduate Degree in Engineering with a specialization in Mechanical Engineering from the University of Maryland Eastern Shore. Prior to receiving his Undergraduate Degree, Rodrigo programmed and designed the Quality Control database during an internship at Sonic Tools LP. As an intern with MaTech Solutions, Rodrigo built 3D models of shop machines to be used for simulating programs and preventing machine crashes.

Dr. Payam Matin, University of Maryland, Eastern Shore

Dr. Payam Matin is currently an Associate Professor in the Department of Engineering and Aviation Sciences at the University of Maryland Eastern Shore (UMES), Princess Anne, Maryland. Dr. Matin has received his Ph.D. in Mechanical Engineering from Oakland University, Rochester, Michigan in May 2005. He has taught a number of courses in the areas of mechanical engineering and aerospace at UMES. He has served as departmental ABET committee chair through a successful accreditation visit in Fall 2012. Dr. Matin’s research has been mostly in the areas of Computational Mechanics and Experimental Mechanics with applications in Solid Mechanics, Plasticity and Sheet Metal Forming. Dr. Matin has published more than 25 peer-reviewed journal and conference papers. Dr. Matin is the recipient of NSF MRI award as a Co-PI. Dr. Matin worked in Automotive industry for Chrysler Corporation from 2005 to 2007. He joined UMES in August 2007. He is affiliated with ASME and ASEE professional societies

Monai Stinnett, University of Maryland Eastern Shore

Monai Stinnett graduated in December 2014 with a Bachelor of Science Degree in General Engineering Specializing in Mechanical Engineering from University of Maryland Eastern Shore. Monai is currently enrolled at University of Maryland College Park Master Program pursuing in Mechanical Engineering, Energy and Environment. She wants to further her knowledge in Energy Engineering to focus on methods to effectively increase efficiency and to use energy in cleaner ways. For her Senior Design Project, Ms. Stinnett Designed a Dome Test Setup for Sheet Metal Formability Characterization. During the summer of 2014, Ms. Stinnett had a Maryland Space Grant Consortium Summer Exchange Student Internship. She developed lab experiments that aid the learning of multiple manufacturing concepts through hands-on completion of the lab exercise. Integrating learning-based assessment tools into the designs of the experiments.
Learning Experience in Designing a Dome Test Setup for Sheet Metal Formability Characterization

Limiting Dome Height (LDH) test conventionally known as Dome Test is one of the methods that is used in industry for characterization of sheet metal formability. Designing a dome test set up can provide undergraduate students with significant learning experience. The objective of this paper is to discuss the learning experience that has been gained by a team of undergraduate students in designing a dome test setup in a form of a senior design project. A dome test set up is consisted of a punch, die, and blank holder along with instrumentation. Through a contact, the punch forms a flat sheet metal specimen, which is held on its edge by the blank holder, into the cylindrical die cavity in a shape of a dome. The height of the dome is considered as a measure of formability. To achieve a set of pre-defined educational objectives, the design project is defined with specific design requirements and design constraints. Students have worked in a team framework to develop their detailed design approach under supervision of faculty. Students have learned to utilize commercial software ABAQUS for FEM forming simulation of dome test to estimate the maximum punch force needed for successful testing. Students have improved their Computer Aided Design skills in designing the punch, die, and blank holder including the draw beads that can stand the required force estimated by FEM. A force and position sensors are integrated in the design as instrumentation to measure the force and dome height at different stages of deformation. Students have collaborated to build the solid model of the system in SolidWorks. The proper design documentations have been prepared. The learning outcomes and educational benefits of the projects that have been gained throughout the course of two semesters are discussed.

Introduction

Sheet metal forming is generally referred to manufacturing processes in which sheet metal is deformed plastically into a desired geometry of a product. Sheet metal forming has wide applications in today’s industries such automotive, aerospace, defense, and so on. There are several sheet metal forming processes including stamping, hydroforming, deep drawing, roll forming, etc. The mechanics of sheet metal forming is mainly introduced in [1-2]. Formability is defined as ability of sheet metal to be deformed plastically without any failure. In the recent years, several studies have been conducted to characterize sheet metal formability [3-7]. In today’s industries, the demand for stronger material with high ductility and formability is high. The automotive industry is constantly looking for lighter material with improved material properties. Traditionally, the standard tensile test has been used to obtain information on stress-strain graph of materials under deformation. However, this test is limited because it only measures uniaxial stress-strain behavior and is only useful up to the ultimate strength point where “necking” is about to occur. In contrast to the tensile test where the mode of deformation is uniaxial, the Limiting Dome Height (LDH) test is equi-biaxial. Consequently, necking would not occur throughout deformation. Thus, the LDH test allows for a higher magnitude of biaxial state of true stress and strain. In a LDH test, the stress and strain of materials continue to increase all the way until the failure of the material. The dome test setup consists of a thin flat specimen held between a blank holder and a cylindrical die. Force is applied to the specimen by the punch fixture that will be attached to the ram of a press. Figure 1 depicts the setup of the standard Limiting Dome Height test. In this test, rectangular blanks are clamped firmly and stretched over
A hemispherical punch. The height of the dome at a maximum load (near failure) is used as a measure of formability.

![Figure 1. Limiting Dome Height (LDH) Test Setup.](image)

The LDH test can be used to determine the stress-strain graphs for larger strains similar to the Viscous Pressure Bulge test [8]. Although, rather than using a viscous medium, the LDH test uses a solid hemispherical punch to form the sheet. The LDH test is widely used in industry to evaluate lubricants and formability of sheet metal. Friction at the tool interface can affect the formability and thinning distribution. To obtain an accurate stress-strain graph, maximum thinning should occur at or very near the apex of the dome. This can be achieved by using a very good lubricant. In practice, it might be difficult to achieve near-frictionless conditions.

**Educational Goals**

The objective of this project is to engage a team of undergraduate students to utilize the fundamentals of mechanics along with Finite Element Simulation to design a small size low-cost dome test setup for instructional purposes.

While the project is a senior design project, it follows the following main educational goals:

1- The project aims to improve the ability of the students to design a realistic system and its components under realistic design requirements and constraints.
2- The project aims to improve the ability of the students to apply fundamental of solid mechanics (such as Mechanics of Materials, Plasticity and Metal Forming) for design purposes.
3- The project aims to advance students ability in finite element simulation of a complex system.
4- The project intends to improve students’ skills in solid modeling of components and assemblies.
5- The project is to improve the ability of the students to apply modern engineering tools (such as SolidWorks, ABAQUS, Matlab, Excel) to analyze and design a realistic system and its components.
6- The project is to engage students in selecting appropriate instrumentation for force and dome height measurement.
7- The project aims to improve the students’ written and oral communication skills.
The educational goals of the project correlate closely with most of the ABET student outcomes (a-k), which are widely accepted in engineering education community. These outcomes have introduced and mandated by ABET for engineering programs to ensure the quality of engineering graduates. Projects similar to this project would help engineering educators to cover many student outcomes in senior design classes, which improve the quality of engineering education.

Three senior level students worked on this project over the course of two semesters under senior design project I and II classes. One student was mainly in charge of FEM forming simulation. The other student was mainly focused on part and assembly design and stress analysis of the system. The third student worked on the instrumentation and had a supporting rule helping the other two students. The students worked in Spring and Fall semesters on their projects. Unfortunately, no funding was available for the project. It is intended that the project complements the prior works of the other educators in improving senior design classes [9-12].

**Problem Statement**
The project is to research and design the apparatus of the Limiting Dome Height Test. The system will contain instrumentation that will obtain the measurement of the force applied by the punch and displacement of the specimen as a function of time during the experiment. Using the recorded data, the stress-strain graph and subsequently the material properties can be determined.

**Design Requirements**
The Limiting Dome Height Test must be designed to the following design requirements:
- The setup should measure the Applied Force and the Dome Height at Fracture (with instrumentation).
- The setup should be equipped to a Data Acquisition System.
- The setup should feature a die of 51 mm and a Punch of 50.8mm radii to support a standard test.
- The setup should support reasonable sheet metal strength (K-value as high as 1500MPa, and n-value as low as 0.1) and sheet metal thickness in the range of: 0.25 mm < t < 1 mm.

**Design Constraints**
The final design of the Limiting Dome Height Test must be within the following limitations:
- The cost must be within the limits of the department costs budget and less than $1,500.
- The system must be compatible with the Big Red-Torin Press available in the department.
- The system must be removable from the press.
- The system size must fit within the working range of the Big Red Torin Press.
- The weight of the system should not exceed 100 lbs.
Design Approach
To design the Limiting Dome Height Test, the students have developed the following steps:

1. Determine the Design Requirements
2. Determine the Design Constraints
3. Collect background information on The Limiting Dome Height Test
4. Estimate Punch Force based on Finite Element Forming Simulations of a blank specimen with the required strength and thickness with ABAQUS
5. Select the required devices
   a. Verify the applicability of the hydraulic shop press
   b. Choose position and force sensor
   c. Choose the proper material that would be most fitted for the Limiting Dome Height Test Setup
6. Design components using SolidWorks
   (Ram from the press, Punch, Blank Holder, Lower Die Cavity, Specimen, Base, Bolts, Nuts, Position sensor, Force sensor)
7. Check if the weight constraint is within the limit
8. Conduct stress analysis based on Finite Element simulation to verify design
9. Modify and finalize design
10. Purchase materials
11. Fabricate Design
12. Test and validate the performance of the Limiting Dome Height Test setup

Figure 2 shows the design approach describing the process of the design in the simplest form.

![Design Approach Diagram]

Figure 2. Design Approach

Conceptual Design
For the conceptual design, several parts need to be designed. A limiting dome height test consists of a punch, a die, a blank holder, a force application system, a clamping system, instrumentation to measure the force and height of the dome, and a lubrication system. Below are the initial dimensions considered for the different parts:

- Length of specimen = 7 in
- Width of specimen = 4.29 in - 5.63 in (Width will be varied to find Limiting Dome Height)
- Punch radius = 2 in
- Inner diameter of both the Die and Blank Holder is 4.16 in.

The standard setup of the LDH test is shown in figure 3 [13]. As shown, the standard punch in the industry for the limiting dome height test is a hemispherical punch with a 2 inch radius. This radius is used for the conceptual design.

![Figure 3. Limiting Dome Height Test Setup Design Dimensions [13].](image)

For the lower die cavity, enough room should be considered to allow for the instrumentation to measure the dome height. The blank holder, also called the upper die, needs to fit on top of the lower die. Draw beads should be placed in both the blank holder and the lower die to prevent the specimen from slipping into the cavity of the lower die. For the force application system, the students considered a hydraulic press from Torin Manufacturing. This press has a hydraulic ram that can be lowered sufficiently enough, that if the punch is attached to the ram, it will create the necessary deformation. Bolts have been chosen for clamping the lower and upper die together with a sufficient clamping force to prevent slipping of the specimen. Also, a position sensor is considered to measure the height of the dome at failure. Lubrication is necessary to reduce the effects of friction between the punch and the specimen. Teflon based dry coolant has been selected as the lubricant as recommended in [14].

**Finite Element Forming Simulation for Punch Force Approximation**

The purpose of the forming simulation is to obtain the force required to fail the specimen and the height at which the specimen fails. A finite element forming simulation is conducted using ABAQUS to model each part of the system. The ABAQUS simulation is to obtain the contact reaction force on the punch (during a test), and stress and strains in the specimen. After the simulation, the ABAQUS results are compared against the existing experimental data for verification purposes.

**Failure Criterion for Forming Simulation:**

The first step of the design process is to calculate the maximum suitable punch force that needs to be provided to the setup. The suitable punch force is the one that will rupture the strongest materials (at maximum thickness) that are planned to be tested. The industry considers failure to
occur at thickness strain ($\varepsilon_3$) of 20%, known as “thinning.” A negative sign is used to illustrate the decrease in thickness of the sheet metal as it experiences higher magnitude of strain.

$$\varepsilon_3 = -0.2$$  \hspace{1cm} (1)

The theoretical final thickness is calculated from this calculated failure strain based on [2]:

$$\varepsilon_3 = \ln\left(\frac{t}{t_0}\right)$$  \hspace{1cm} (2)

Starting with the thinning criterion as failure, $\varepsilon_1$ and $\varepsilon_2$ can then be calculated using the concept of volume preservation or incompressibility, which mandates [2]:

$$\varepsilon_1 + \varepsilon_2 + \varepsilon_3 = 0$$  \hspace{1cm} (3)

The dome deforms symmetrically, therefore the in-plane strains are equal (biaxial) [1].

$$\varepsilon_1 = \varepsilon_2 = \varepsilon$$  \hspace{1cm} (4)

Substituting (4) into (3) yields:

$$2\varepsilon + \varepsilon_3 = 0$$  \hspace{1cm} (5)

Substituting (1) into (5) yields:

$$\varepsilon = 0.1$$  \hspace{1cm} (6)

The strain values calculated are used in equation (7) to estimate the effective strain (Von-Mises) at the failure. Then, the effective stress is calculated by substituting the specimen material properties and the calculated effective strain into the power law presented equation (8) [2]. For design of the Limiting Dome Height Test, the material model is defined (in ABAQUS) for the plastic deformation zone based on the power law up to the effective failure strain, which is calculated approximately as 0.35.

$$\varepsilon_{\text{eff}} = \frac{1}{3}[(\varepsilon_1 - \varepsilon_2) + (\varepsilon_1 - \varepsilon_3) + (\varepsilon_2 - \varepsilon_3)]^{1/2}$$  \hspace{1cm} (7)

$$\sigma_{\text{eff}} = K\varepsilon_{\text{eff}}^n$$  \hspace{1cm} (8)

where $K$ is the material strength coefficient (or K-value) and $n$ is the strain hardening exponent (or n-value) of the material being tested.

**Forming Simulation Model:**

The finite element forming simulations for the Limiting Dome Height Tests are completed using ABAQUS. All four components are modeled as instances to obtain an accurate force required to fail the specimen. The specimen is modeled as a thin-shelled member with a given thickness of the tested material. The dimension of all the FE model parts are designed to be consistent with the geometries of punch, die and blank holder in the experimental LDH testing. The punch, die, and blank holder are revolved geometries that are considered as rigid bodies. When ABAQUS is used for quasi-static forming analyses, rigid bodies are ideally suited for modeling the tooling (such as punch, die, draw bead, blank holder, roller, etc). Defining the tooling as rigid body models results in lower run times and higher computational efficiency. Shown in figures 4, 5 and 6 are the part instances and their respective mesh of the die, blank holder, punch and the specimen. The die and blank holder have the same geometry, but are mirrors of each other in the assembly.
To verify the results, simulations are run using the parameters specified by the other researchers as shown in figure 7 [14-16]. The thickness, strength coefficient (K-value), and strain hardening exponent (n-value) of the material are defined in ABAQUS based on the values specified in [14].
The exact characterization of the steel used in [14] is unspecified in this reference; however, most galvannealed steel sheets of deep drawing quality have a yield stress in the range of 150 MPa to 225 Mpa. The specimen to be simulated is defined with the properties of steel as shown in Table 1.

<table>
<thead>
<tr>
<th>Density</th>
<th>Poisson’s Ratio</th>
<th>Young’s Modulus</th>
<th>Yield Stress</th>
<th>Thickness</th>
<th>n Value</th>
<th>K Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>7,800 kg/m³</td>
<td>0.3</td>
<td>200 GPa</td>
<td>158 MPa</td>
<td>0.69 mm</td>
<td>0.2</td>
<td>1,000 MPa</td>
</tr>
</tbody>
</table>

Table 1: Material Properties of Steel Defined in ABAQUS.

A stress-strain graph is generated using the power law presented in (8) based on the n and K values of the selected material. As shown in Table 2, the stress-strain graph is defined in the material behaviors section of ABAQUS.

<table>
<thead>
<tr>
<th>Yield Stress</th>
<th>Plastic Strain</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td>2</td>
<td>0.001</td>
</tr>
<tr>
<td>3</td>
<td>0.01</td>
</tr>
<tr>
<td>4</td>
<td>0.02</td>
</tr>
<tr>
<td>5</td>
<td>0.03</td>
</tr>
<tr>
<td>6</td>
<td>0.04</td>
</tr>
<tr>
<td>7</td>
<td>0.05</td>
</tr>
<tr>
<td>8</td>
<td>0.1</td>
</tr>
<tr>
<td>9</td>
<td>0.15</td>
</tr>
<tr>
<td>10</td>
<td>0.2</td>
</tr>
<tr>
<td>11</td>
<td>0.25</td>
</tr>
<tr>
<td>12</td>
<td>0.3</td>
</tr>
<tr>
<td>13</td>
<td>0.35</td>
</tr>
</tbody>
</table>

Table 2: Material behavior in Plastic Deformation zone Defined in ABAQUS.
As depicted in figure 8, the parts are assembled in ABAQUS. A friction contact is defined between the die and the specimen and the blank holder and the specimen to prevent the specimen from slipping during the test. For ideal situations, the contact between the punch and specimen is considered frictionless so that the specimen will fail at the apex of the dome.

**Forming Simulation Results:**
Several simulations are conducted to evaluate the effect of the number of elements and the yield stress on the simulation. The model for specimens are meshed with elements that are refined enough in the area where a high deformation is expected. Figure represents the Von-Mises stress contours at the onset of failure of the specimen. Failure of the specimen is observed as the Punch Force becomes maximum during the test. Observed in the figure, it is seen that after failure, the force decreases rapidly. As expected in the figure, the contour shows that the stress is highest at the center of the sheet, which corresponds to the apex of the dome, and lowest closer to the edge of the sheet where it is only being affected by the contact of the die and blank holder.

The values obtained from the simulations are compared to the values obtained from the Ohio State University’s Limiting Dome Height Test [14]. Figure depicts the punch force versus punch stroke for 3 runs of simulation against simulation and experimental results conducted by Ohio State [14]. The 3 runs are associated with 3 effects of the plastic deformation zone of the material. Run 1, defined as the “basic” simulation, has all the material properties as defined in Table 1; however the Yield Stress is defined as 350 MPa. Run 2, defined as the “Higher Strain”
simulation, has all the material properties as defined in Table 1; however the Yield Stress is defined as 375 MPa and the plastic deformation zone is extended to a plastic strain of 0.5. Run 3, defined as the “Lower Yield Point” simulation, and is defined exactly as defined in Table 1. As expected, the punch force increases as the punch stroke increases. Failure is defined at the maximum punch force as appears in the simulation. After failure, the punch force will decrease. The simulation results are relatively closed to the simulation results of Ohio State.

![Graph showing punch force vs. punch stroke](image)

**Figure 10. ABAQUS Limiting Dome Height Result Comparison.**

Table 3 summarizes the maximum punch force and strokes for each run based on the graphs in Figure. Table 3 also presents the percent errors of the maximum punch force and strokes obtained from the simulations relative to the experimental results conducted by Ohio State. It is shown that the simulation run with an appropriate mesh and lower yield strength had the lowest percent error for the force required to fail the specimen at the apex of the dome. Thus, the FEM model developed is considered as a valid model for further design and analysis purposes.

<table>
<thead>
<tr>
<th>Force (Newtons)</th>
<th>Displacement (mm)</th>
<th>% Error Force</th>
<th>% Error Displacement</th>
</tr>
</thead>
<tbody>
<tr>
<td>Run 1 (Basic)</td>
<td>Run 2 (Higher Strain)</td>
<td>Run 3 (Lower Yield Point)</td>
<td>OHIO FEM Simulation</td>
</tr>
<tr>
<td>73,153.00</td>
<td>78,353.00</td>
<td>72,673.80</td>
<td>80,000.00</td>
</tr>
<tr>
<td>34.49</td>
<td>35.88</td>
<td>34.51</td>
<td>41.19</td>
</tr>
<tr>
<td>16.6%</td>
<td>22.1%</td>
<td>16.1%</td>
<td>23.8%</td>
</tr>
<tr>
<td>19.4%</td>
<td>14.8%</td>
<td>19.3%</td>
<td>0.0%</td>
</tr>
</tbody>
</table>

**Table 3: ABAQUS Limiting Dome Height Result Comparison**

Effects of Material Properties on Punch Force
The maximum force that the press can theoretically deliver is 295 kN. With a safety factor of 1.6, the system will be designed for a force of 180 kN. To design for this level of punch force, the effect of the thickness, the strength coefficient (K), and the strain hardening exponent (n) on the Punch Force must be evaluated. It is expected, that as the strength coefficient and the thickness
increases, the maximum Punch Force required to fail the material increases. In contrast, as the strain hardening exponent increases, the maximum Punch Force required to fail the material decreases. Figure illustrates that as the strength coefficient increases, the maximum Punch Force increases and as the strain hardening exponent increases, the maximum Punch Force decreases. Figure illustrates that as the thickness increases, the maximum Punch Force increases.

![Effect of Material Properties on Punch Force](image1)

**Figure 11. Effects of Material Properties on Punch Force; thickness = 0.69 mm.**

![Effect of Thickness on Punch Force](image2)

**Figure 12. Effects of Thickness on Punch Force with n= 0.2, K=1000 MPa**

The value of the strain hardening exponent varies between 0 and 1. However, n-values typically range from 0.1 to 0.5. In practice, an n-value of 0.1 is the lowest value that will lead to the largest punch force. Simulations are conducted to study the punch force for the maximum thickness and minimum n-value for varying K. Figure 1 shows how with the maximum thickness and minimum n-value, the design force of 180 kN will limit the design to a K-value of 1700
MPa. Figure 1 also shows how the thickness affects the slope of the punch force versus K-value graph.

![Comparision of Thickness on Maximum Punch Force](image)

Figure 13. K-value (strength) limit for targeted punch force for minimum n-value and maximum thickness

**Design of a Solid Model for Dome Test Setup**

A hemispherical punch is designed with a 2 inch radius as required for the standard test. The solid model of the punch is shown in Figure 14. The punch was designed to lock into the press ram using a lock screw. The punch is inserted into the ram with the lock screw being used to hold the punch in place.

![Solid Model of Punch](image)

Figure 14. Solid Model of Punch.

As shown in figure 15, a cylindrical die is designed as the lower die. This design is relatively standard in the industry. The ring outside of the die cavity is the female lock bead of the lower die. The blank holder contains a male lock bead that fits into the female lock bead to hold the specimen in place and prevent it from slipping into the die cavity. The four holes around the
outside of the lower die are the bolt holes. These holes are where the bolts clamp the blank holder and lower die together. Two small rectangular male features are designed on the lower die. These features are locators and the specimen is placed in between them. To allow room for the position sensor, which is used to measure the dome height, the lower die is elevated so that the sensor can be placed underneath it. To accommodate this, circular columns are designed under the lower die, raising it to the desired level. Figure 15 also depicts the bottom of the lower die, which encompasses four larger holes that allow for the lower die to be elevated by the circular columns of the base.

As shown in Figure 16, the design of the blank holder (the upper die) is basically a mirror image of the lower die design. The draw beads on the bottom surface of the blank holder are designed as a male feature to fit into the female draw beads on the top surface of the lower die, locking the specimen in place and preventing it from slipping into the lower die cavity. The upper die is also not as thick as the lower die is. Such design is intended to incorporate the bolts as well as lower the cost since less material is used. The specimen is placed on top of the draw beads on the lower die and is locked into place as the blank holder is placed on the lower die.
Figure 17 shows an exploded view of the blank holders, specimen, and lower die assembly. The male and female draw beads are seen in the blank holder and the lower die, respectively. Additionally, the male and female specimen locators are seen in the lower die and blank holder, respectively.

![Figure 17. Exploded View of Blank Holder, Specimen, and Lower Die Assembly](image)

The base and the circular columns that hold the lower die in the desired level are seen in figure 18. The base consists of four cylindrical columns. Only partial of the length of each column is inserted into the bottom holes of the lower die. That provides room to place the position sensor and the associated instrumentation needed to collect data. The position sensor is placed in the central hole of the base. The output terminal of the position sensor is designed to fit into the central hole of the base. To ensure contact around each column, tolerances are assigned to both the lower die holes and the cylindrical columns of the base.

![Figure 18. Base](image)

Figure 19 depicts the entire Dome Test setup assembly including the specimen as a solid model. The assembly features the die assembly on the bottom, which is shown in a clamped position and placed on the base assembly, and the punch assembly on the top. The position sensor is placed underneath the die assembly. Figure 19 also shows the labeled exploded view of the assembly with all components. The press ram and the force washer as well as position sensor are also included in the exploded view.
SolidWorks is used to calculate the weight of the system to compare with the predefined design constraints. According to the constraints, the Limiting Dome Height Test setup has to weigh less than one 100 lbs or 45 Kg. The weight of each part is shown in Table 4. The total weight of the Limiting Dome Height Test setup is 8437 grams; therefore the weight constraint of the Test setup has been met.

<table>
<thead>
<tr>
<th>Parts</th>
<th>Mass (Kg)</th>
<th>Weight (lbs.)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Punch</td>
<td>.971</td>
<td>2.141</td>
</tr>
<tr>
<td>Die</td>
<td>2.095</td>
<td>4.618</td>
</tr>
<tr>
<td>Blank Holder</td>
<td>3.981</td>
<td>8.777</td>
</tr>
<tr>
<td>Base</td>
<td>2.359</td>
<td>5.200</td>
</tr>
<tr>
<td><strong>Total</strong></td>
<td><strong>8.437</strong></td>
<td><strong>18.600</strong></td>
</tr>
</tbody>
</table>

Table 4: Weight of Limiting Dome Height Test setup bases on Solid Model

**Stress Analysis of Limiting Dome Height Test Setup**

The purpose of the stress analysis of the setup using SolidWorks Finite Element Simulation is to ensure that the individual components (punch, die, blank holder, and base) will not fail under stresses due to press force. The components must not deform permanently under the stress. In the simulations, 6061 Aluminum Alloy is used as the material for the fixtures. This material is chosen because of its low cost while featuring yield strength comparable to that of steel.

As shown in Figure , the fixed boundary conditions are defined on the top of the punch where the surface of the punch is in contact with the load cell. The punch is designed to allow for a washer load cell to be placed between the punch and the ram in such a way that the load cell takes the entire punch force. The punch is in constant contact with the sheet metal specimen. Thus, the
load on the punch will be in a form of a pressure being applied from the bottom to the punch by the specimen. The maximum pressure is calculated by dividing the targeted punch force by the normal area of the hemispherical section of the punch.

\[
P = \frac{F_v}{A_n} = \frac{F_v}{\pi(r^2)} = \frac{180 \times 10^3 N}{\pi(50.8 \times 10^{-3} m)^2} = 22.2 MPa
\]  

(9)

The calculated pressure of 22.2 MPa is used as an external load to simulate for stress analysis of the model. Figure 20 depicts the maximum stresses that the punch is expected to endure. The maximum stress on the punch does not surpass the yield strength of the material. This means that the punch will not permanently deform. This maximum stress is occurred on the fillet of the cylindrical section of the punch. The results of the simulation feature a factor of safety of 1.5 under the given conditions.

![Figure 20. Von Mises Stress Contour of Punch](image)

The load on the base is simulated by distributing the calculated punch force of 180 kN across the columns. The force is divided by four and distributed over each column. The fixed boundary conditions are defined for the base at the bottom of the plate in all directions. Clamps ensure such boundary conditions during the test. From Figure , it can be seen that the maximum stress that the base endures is less than the material yield strength. The maximum stress occurs at the base of the columns. The results lead to a factor of safety of 1.3 for the base.
As shown in figure 22, the next component that is simulated is the lower die. The fixed boundary conditions are defined for the lower die on the four holes where the columns from the base are inserted. The load is defined by the punch force of 180 kN being distributed over the entire location of the specimen. The clamping force of 13.87 kN is considered by partitioning the area where the nuts are in contact with the die, then imposing the force to each section. From the simulation results, a stress can be seen throughout the height of the dome, with the maximum stress occurring on the top face of the lower die. On the bottom of the die, stress can be seen strongest around the holes where the columns are inserted. In overall, the results show the maximum stress is lower than the yield strength of the material. This leads to a factor of safety of 1.75 on the lower die.

In Figure , the results of the FEM simulation on the blank holder can be seen. The clamping conditions provided by the bolts defines the boundary conditions. The load required to simulate forces acting on the blank holder is the same distributed 180kN force from the lower die. This load is used to simulate the reaction forces from the deforming specimen. The maximum stress on the die is occurred on the bottom face of the blank holder, close to the edge of the die cavity. The results of the simulation show the maximum stress that is less than the yield strength featured in the material. This provides a factor of safety of 1.375 to the blank holder.
Limiting Dome Height Test Sensory Devices

Data Acquisition System
The data acquisition system shown in Figure is a system that processes the signals that are measured and collected by the instrumentation from the physical conditions and converts them into digital numeric values that can be viewed by a computer. The purpose of the data acquisition system is to generate the punch force versus stoke as explained in the “Test Procedure” section. The data acquisition systems include a position sensor and a force sensor shown in Figure and Figure, respectively.

Position Sensor
A position sensor is a device that permits position measurements. The position sensor used for the dome test is an ultrasonic sensor which utilizes sound waves for measurements. The position sensor is intended to measure the dome height. The ultrasonic sensor has a sensing range of 30mm-300mm; the range of the dome test is about 40mm. The dimensions of the position sensor selected are shown in figure 25. The position sensor is placed on the base underneath of the lower die.
Force Washer load Cell

Force Washer Transducers are miniature load cells designed for measuring fastener forces. The Force washer load cell shown in figure 26 is selected for the dome test setup.

Table shows the dimensions of the force washer load cell selected [17]. For the final design, the load cell is placed between the ram of the press and the hemispherical punch. The force washer load cell has a measuring range of 220kN [17]. The targeted punch force used for the design is 180kN. The students have decided to select the 220kN force washer load cell to meet the amount of the force needed.

Cost Analysis
Table 6 lists the raw materials and the items needed for the Limiting Dome Height test setup and the cost associated with them. The total cost is more than what was originally expected. The noticeable prices of position and force sensor have significantly contributed into the bottom line. If these items are designed in house, the design will be more cost effective.
<table>
<thead>
<tr>
<th>Name</th>
<th>Quantity</th>
<th>Price</th>
<th>Model #</th>
</tr>
</thead>
<tbody>
<tr>
<td>Position Sensor</td>
<td>1</td>
<td>$225</td>
<td>UB300-18GM40-U-V1</td>
</tr>
<tr>
<td>Force Sensor</td>
<td>1</td>
<td>$749</td>
<td>054216-01144</td>
</tr>
<tr>
<td>Bolts/nuts</td>
<td>1(10pk)</td>
<td>$57.15</td>
<td>1RU96</td>
</tr>
<tr>
<td>Coolant</td>
<td>3</td>
<td>$21.69</td>
<td>31736929</td>
</tr>
<tr>
<td>C Clamps</td>
<td>4</td>
<td>$12.42</td>
<td>03134277</td>
</tr>
<tr>
<td>Raw Materials</td>
<td>2</td>
<td>$35.84</td>
<td>7392T49</td>
</tr>
<tr>
<td></td>
<td>1</td>
<td>$47.49</td>
<td>9050K21</td>
</tr>
<tr>
<td></td>
<td>1</td>
<td>$175.19</td>
<td>9143K75</td>
</tr>
<tr>
<td></td>
<td>1</td>
<td>$22.66</td>
<td>8924K6</td>
</tr>
<tr>
<td>Manufacturing Cost</td>
<td></td>
<td>$700</td>
<td></td>
</tr>
<tr>
<td>Total</td>
<td></td>
<td>$2,018.44</td>
<td></td>
</tr>
</tbody>
</table>

Table 6: Cost Analysis

Assessment of Student Outcomes

For the purpose of assessing this project, a few specific learning outcomes have been selected. These learning outcomes include 1) Design, 2) Modern tools, 3) Oral Communication, 4) Written Communication and 5) Teamwork. For each selected learning outcome, a set of performance indicators have been defined as metrics for assessment purposes. Table 7 summarizes the performance indicators that have been developed to assess each of the learning outcomes selected. For a given learning outcome, the performance of the participating students have been assessed on each of the performance indicators (associated with the outcome) using a four scale scoring system. The scoring system considers scores of 1-4 for student performance of unsatisfactory/need improvement/satisfactory or competent. The student performance is estimated in percentage for each performance indicator for any selected learning outcome.

Figure 27 shows the assessment results for Design learning outcome where student performance is depicted for each of the performance indicators associated with Design outcome. As seen, the students performed above 80% for all the performance indicators. Within such range, which is encouraging, it appears that student performance is the lowest in setting up design constraints, developing design strategy/planning/timeline, conducting design analysis using governing equations along with design modification and optimization. These performance indicators can be even further improved with more attentions for future projects.
<table>
<thead>
<tr>
<th>Performance Indicators</th>
<th>1) Design Outcome</th>
<th>2) Modern Tools Outcome</th>
<th>3) Oral Communication Outcome</th>
<th>4) Written Communication Outcome</th>
<th>5) Teamwork Outcome</th>
</tr>
</thead>
<tbody>
<tr>
<td>Design requirements</td>
<td>Basic computer-based software such as Word, PowerPoint, etc</td>
<td>Organization</td>
<td>Organization</td>
<td>Attendance in team meetings</td>
<td></td>
</tr>
<tr>
<td>Design constraints</td>
<td>Spread Sheet software</td>
<td>Clarity</td>
<td>Clarity</td>
<td>Preparation for team meetings</td>
<td></td>
</tr>
<tr>
<td>Design conceptualization</td>
<td>Programming languages such as C, Java, etc</td>
<td>Sufficiency</td>
<td>Sufficiency</td>
<td>Contribution to team</td>
<td></td>
</tr>
<tr>
<td>Design strategy/planning/timeline</td>
<td>Math/Computational tools such as Matlab, Mathematica, MathCAD, etc</td>
<td>Flow/Sequence</td>
<td>Flow/Sequence</td>
<td>Interaction/ cooperation with others</td>
<td></td>
</tr>
<tr>
<td>Design analysis: (using governing equations)</td>
<td>Computer-Aided Design (CAD) software such as SolidWorks, etc</td>
<td>Use of Charts/Graphs/ Tables/…</td>
<td>Use of Charts/Graphs/ Tables/…</td>
<td>Sharing information with others</td>
<td></td>
</tr>
<tr>
<td>Design analysis: (using modern tools)</td>
<td>Computer-Aided Engineering (CAE) software such as Abaqus, FlowSimulation, PSpice, MultiSim, etc</td>
<td>Time</td>
<td>Proper English: (Grammar, spelling, writing Style, etc)</td>
<td>Listening to others</td>
<td></td>
</tr>
<tr>
<td>Design verification</td>
<td>Presentation Soft Skills: (Eye contact, being heard, non-monotonic, body language, nervousness, not-blocking screen, etc)</td>
<td>Results</td>
<td></td>
<td>Respectfulness to others</td>
<td></td>
</tr>
<tr>
<td>Design modification/optimization</td>
<td>Professional appearance</td>
<td>Discussion of results</td>
<td></td>
<td>Encouraging others for participation</td>
<td></td>
</tr>
<tr>
<td>Cost analysis</td>
<td>Proper English</td>
<td>Citation/References</td>
<td></td>
<td>Sharing credit with others</td>
<td></td>
</tr>
<tr>
<td>Design documentation</td>
<td>Handling questions</td>
<td>Formatting</td>
<td></td>
<td>Accepting designated team roles</td>
<td></td>
</tr>
<tr>
<td>Physical/virtual prototype</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table 7 Summary of Performance Indicators used for Assessment for each Learning Outcomes
Figure 27 Assessment of Design Outcome; Student Performance (%) for different Performance indicators

Figure 28 Assessment of Modern Tools Outcome; Student Performance (%) for different Performance indicators
Figure 29 Assessment of Oral Communication Outcome; Student Performance (%) for different Performance indicators

Figure 30 Assessment of Written Communication Outcome; Student Performance (%) for different Performance indicators
Figure 31 Assessment of Teamwork Outcome; Student Performance (%) for different Performance indicators

Figure 32 Assessment of Learning Outcomes; Student Performance (%) for Learning Outcomes
Figure 28 shows the assessment results for Modern tools outcome where student performance is depicted for each of the performance indicators associated with Modern Tool outcome. As shown, the students performed above 80% for all the performance indicators. Figure 29 depicts the assessment results for Oral Communication outcome. Student performance for all the performance indicators is above 75%. Within such range, it appears that sufficiency of the materials presented by the students is the lowest performance, which can be improved for similar future projects. Figure 30 shows the assessment results on Written Communication outcome. Although all the performance indicators have been assessed above 75%, sufficiency of the materials included in the written reports and writing style (in terms of proper English) can be further improved. Figure 31 depicts the assessment results for Teamwork learning outcome. It is observed that student performance is above 75% for all the performance indicators associated with Teamwork outcome. The contribution of the teammates to the team and information sharing among team members are among the performance indicators that have been assessed the lowest. These performance indicators need further attention for future similar projects. Figure 32 summarizes the assessment results for all the learning outcomes. As shown, all the outcomes have been assessed above 80%, which is very encouraging.

**Further Observations on Student Learning**

1. The project exposed a team of students to the design process of a real world Limiting Dome Height test setup with realistic design requirements and design constraints.
2. The students developed a logical design approach to design the main components of the dome test setup.
3. The students learn how to conduct an FEM forming simulation to validate the performance of the setup designed.
4. The students learned how to apply the fundamentals of mechanics to design for the main components such as lower die, blank holder, punch, and bolted joints.
5. The students learned how to build a solid model of the system, and how to run a finite element simulation to verify the design.
6. The students gained hands-on experience working with different modern math and engineering software such as MATLAB, SolidWorks, ABAQUS, and etc.
7. The students gained valuable experience on how to select instrumentation for the apparatus.
8. The students improved their oral communication skills by making weekly presentation to the audience of the senior design class and a faculty advisor.
9. The students improved their written communication skills by documenting the design, design verification and instrumentation selection.
10. The students had a chance to improve their project management skills by setting up project plan, time line, budge and cost table and etc.

**Student Feedback on Learning Experience**

Faculty involved in this project received very positive feedback from the students who conducted the project. At the beginning of the project, the students thought that the topic was uncommon and unconventional. However, they became very interested in the topic as they read and learned more about the project. They were very convinced that they had the opportunity to work on a design project, which involved them in applying math and engineering fundamentals toward the design of a practical system. The students were very satisfied that they gained practical experience with modern engineering tools and software. The students noticed that their oral and
written communication skills have improved remarkably as a result of this project. The students viewed the project as a challenge since many tasks needed to be completed in a short period of time. The students realized that while the course materials are very helpful, they are not enough to conduct real world projects. To this end, they learned a great deal of extracurricular materials to successfully complete the project.

**Pedagogy**

The teaching style adopted for this project is project-based learning and learning-by-doing. The faculty member briefly introduces the design concepts and general terms for design requirements and constraints. The students gather information on their own, and define the project in a form of a design problem with very specific design requirements and constraints. On weekly basis, the students present (in a form of PowerPoint) their progress and future plans to the faculty member and an audience of students. The faculty member reviews the materials and provides detailed feedback interactively during the presentation. If further discussion is needed, the students meet with the faculty member during office hours. The faculty member acts as both customer and advisor and keeps track of the students’ progress and future plans in a log sheet so he can follow up for the upcoming weeks. As the students make progress on their project, they are asked to submit a written report at three different milestones during the semester. Each revision of the report is graded thoroughly by the faculty member. The reports are improved gradually after multiple rigorous grading. The final report should be well-written, formatted, and free of technical errors. A final presentation is presented by the students and evaluated by the faculty member. Assessment data are collected throughout the semester by the faculty member.

**Uniqueness of Project**

Although the project is a senior design project that provides learning experience similar to those of other senior design projects offered by any mechanical engineering program, it provides unique learning experience outlined below:

1. The students are highly exposed to mechanics of plastic deformation, which is nearly overlooked in most of undergraduate mechanical engineering curriculums.
2. As a result of exposure to plastic deformation, the students learn how to work with the concepts such as true strain, true stress, principle of volume preservation, thinning as failure criterion, effective strain, effective stress, and power law. These concepts are not emphasized in most of ME undergraduate programs.
3. The students learn about sheet metal forming (as an important manufacturing process widely used in different industries such as automobile, defense and appliance industries) and the relevant concepts that sheet metal forming can be analyzed and understood.
4. The students learn about forming die setup, and the testing methods commonly used in industry.
5. The students learn how to conduct an FEM-based forming simulation, which is highly complex due to material nonlinearity and contact conditions between multiple components. Such complex simulations would be difficult to be covered in undergraduate level FEM courses.
6. While the project provides the advanced learning topics mentioned above, it is defined in a form of a design project with specific design requirements and constraints so it can be conducted by undergraduate students in a team framework.
Exposure to the advanced topics discussed not only prepare the students for career opportunities in relevant industries, it highly motivates them to pursue graduate study in the same research area due to their prior experience.

Thus, if any of these learning experience is attractive for other mechanical engineering programs, they may be benefitted from this paper to define similar projects to provide similar opportunities for their students.

**Conclusion**

- A Limiting Dome Test setup is designed. The system designed is capable of evaluating formability and delivering the stress-strain graph of the sheet metals.
- The system designed is capable of taking measurement of the force applied by the punch and displacement of the specimen as a function of time during the experiment.
- FEM simulations are implemented in order to understand the forming process in the Limiting Height Dome Test and find the height at which the specimen fails and the force required to fail the specimen.
- Solid model of the Limiting Dome Height Test Setup has been built using SolidWorks. The solid model includes the force washer load cell, position sensor and the Ram of the Hydraulic Shop Press as well.
- SolidWorks has been used to calculate the weight of the Limiting Dome Height Test solid model.
- The data acquisition system is also selected compatible with the position sensor and washer load cell when designing the Limiting Dome Height Test setup.
- The system designed is able to attach and detach from the Hydraulic Shop Press.
- Based on the design of the Limiting Dome Height Test, the testing materials are supported between the range of thickness: $0.25 \text{ mm} < t < 1 \text{ mm}$ featuring a die radius of 51 mm and a punch radius of 50.8 mm.

**References**


14. Sever, Nimet, Kardes, Demiralp, Yurdaer and Dr. Taylan Altan Determination of Flow Stress from Dome Test by Finite Element Based Inverse Analysis Center for Precision Forming, November 2, 2011.

