CAD/FEA TOOLS AND THE ANALYSIS OF DESIGN FOR OPTIMIZATION

A Thesis

Presented to

The Faculty of the College of Science and Technology

Morehead State University

In Partial Fulfillment

of the Requirements for the Degree

Master of Science

By

Patil Yogesh

October 21, 2006

Accepted by the faculty of the College of Science and Technology, Morehead State University, in partial fulfillment of the requirements for the Master of Science degree.

Director of Thesis

Master's Committee: ______________________, Chair

____

____

1/08/2007
Date

CAD/FEA TOOLS AND THE ANALYSIS OF DESIGN FOR OPTIMIZATION

Yogesh Patil, M.S.
Morehead State University, 2006

Director of thesis: ____________________

This thesis is an experimental study of predicting an optimized design of a product using a simulation software. The main objective of this thesis is to emphasize and explore the use of integrated computer aided design and finite element analysis (CAD/FEA) software in optimizing the design and to reduce the physical testing steps. To support this objective, a work-holding clamping plate (master design) is designed on the Autodesk Inventor accompanied with the FEA package; the design is then analyzed, using the stress analysis package (ANSYS) integrated with Inventor version 10. The design and the analysis results are then compared with the series of modified designs of the master work-holding plate results. In the CAD/FEA application tool, the model is constrained, loaded, and then the design is simulated for stress analysis visualizing the magnitude of the stresses that occur throughout the part and the deformation of the part, the stress factor of safety.

Accepted by: ____________________, Chair

____

____

CONTENTS

CHAPTER – I

INTRODUCTION

General Area of Concern

The trend of globalization has forced the manufacturing facilities to redesign the manufacturing system to follow the steps of lean manufacturing and Just-In-Time (JIT) production. The companies are trying to design the process technology and the system to be flexible, controllable, and efficient. To be compatible with this technological progress, it is relatively essential to emphasize the design and durability of the product prior to sending the product to final production.

Basically, manufacturing is a value adding activity and needs to be done in the most efficient way, considering the least amount of time possible, optimum material selection, and manpower. At the same time, it is necessary to have the right attributes such as durability, light-weight, easy to maintain and cheaper product to remain competitive. Design optimization is the only way to achieve this goal. Design optimization is possible using design analysis tools. There are several finite element analysis tools, independent or collaborative, can be used for predictable analysis of the design for maximum allowable stress, maximum applied loads, breaking point of the product, etc (Moaveni, S).

Objectives

The main objectives of this study are (a) to emphasize and explore the use of integrated Computer Aided Design and Finite Element Analysis (CAD/FEA) software in optimizing the design, design time, and shorten the time-to-market as well as reduce the physical testing steps by predicting the structure of design using CAD/FEA simulation; and (b) to perform linear static analysis of the product to configure the stresses and deformation on the design, reducing the physical testing procedure. Four steps must be taken to achieve these goals:

1. Design a master model of an object (main base plate/support plate of vise) to be optimized on Autodesk Inventor professional.
2. Simulate the design to check structural analysis using an integrated CAD/FEA application package in Autodesk Inventor professional.
3. Modify the design into different geometries, taking into consideration the structural analysis parameters, such as stresses, loads, and displacements (design-analysis-test-built) for each design.
4. Compare the results of the master model with the results of the modified designs using the data obtained by simulation tools.

Significance of the Study

For almost 100 years, it was the traditional role of physical tests, with a special focus placed on quality and testing, which was mostly used to consider a concept of "build and ship". However the rapid proliferation of the technological development in the last few years has changed the process of traditional physical testing, adding into it the CAE tool and the finite element analysis tool. Design analysis software helps to improve the product design, reduce material waste, save time and get the product on the market faster than the competitors.

Inspired from the manufacturing revolution, industries are finding solutions to minimize the overall costs of manufacturing and optimize the designs, in which the design itself consumes 60% of the total cost. Design also includes physical testing and analysis of the product.

The object of this study is to examine the feasibility of the integrated design analysis CAD/FEA software application tool in optimizing the design of a support plate for a vise taking into consideration the structural analysis parameters, such as stress, displacements, and load. This simulation technique of design is used to predict the real-world performance and the effect of design changes.

The study does not seek to develop a new and sophisticated analytical or numerical model of a base plate, but to demonstrate the potentiality and the effectiveness of the integrated CAD/FEA design analysis application tool to design an optimized model of plate.

Problem Statement

The support plate is subjected to a distributed load horizontally on the clamping face. The bottom of the plate is constrained to imitate the real-life situation where the vise is tight fitted on a table with the help of nuts and bolts. The plate design is determined using a designed load. The part geometry is modified in six different designs, each design subjected to variation in load (10lbf, 20lbf, 30lbf, 40lbf, 50lbf, and 60lbf) to determine the geometric condition of the product. Plot the deformed shape, and Von-Mises stress distribution from the results obtained, which means, (a) Simulate the effect of material and geometric variations on the model due to stresses; (b) Determine the displacement of the structure when the support plate is subjected to known loads; and (c) Check the designed geometry for varying loads, and redesign the model to ensure optimum results.

Layout of the Thesis

Chapter 2 "Review of Literature," presents the historical overview of the integrated use of CAD/FEA applications. The chapter involves some of the basic terminology and concepts involved with CAD/CAM/FEA analysis. The chapter gives a brief overview of manufacturing system, mastercam, CNC machine, ANSYS, CAD/FEA software (AutoDesk Inventor 10). The chapter then goes into more detailed information regarding the effective use of integrated CAD/FEA analysis.

Chapter 3 of this Thesis is entitled "Methodology." This chapter explains the methodology used to obtain the goal. Several methods of lab work (such as designing several clamping plate models, analyzing the design models and monitoring the results) are discussed.

Chapter 4 of this Thesis is entitled "Findings." Which includes the comparisons of the data collected using graphical representations to obtain the proposed conclusion. The chapter also presents the future improvements to be made in the usage of CAD/FEA.

Definition of Terms

1. CAE (Computer Aided Engineering)

CAE is a computer graphic simulation technology used to evaluate and analyze the part geometry produced in CAD. CAE analysis includes two categories, the finite element modeling and structural analysis, and mass property analysis.

2. CAD (Computer Aided Design)

CAD is a mechanical designing tool which uses a computer system to assist the designer in creation, modification, analysis, or optimization of a design.

3. CAM (Computer Aided Manufacturing)

CAM is a form of automation where computers communicate work instructions directly to the manufacturing machinery. CAM is a manufacturing tool which is used to plan, manage, and control the operations of a manufacturing plant by producing a tool-path of the CAD geometry and creating CNC (computer numerical control) codes, either through direct or indirect computer interfacing.

4. Finite Element Analysis

The finite element analysis is a numeric modeling technique in which a continuum is discretized into simple geometric shapes called elements. The elements are joined together by shared nodes, and the collection of nodes and elements are called mesh (Rao, S).

6. ANSYS

ANSYS is a comprehensive general-purpose finite element computer program that is capable of performing all kinds of analysis such as static, dynamic, heat transfer, fluid flow, and electromagnetism analyses.

7. Stress

Stress is defined as the ratio of applied load to the cross-sectional area of an element in tension. It is expressed in pounds per square inch or kg/mm2.

8. Strain

Strain is the ratio of change in length to the original length. It can also be defined as a measure of the deformation of material that is dimensionless.

9. Ductile Material

Ductile material demonstrate large amount of plastic deformation before the onset of fracture. Also, the properties of ductile material can be enhanced through the use of one of the strengthening mechanisms.

10. Von Mises Stress

Von Mises stress is used to estimate the yield criteria for ductile material. It is calculated by combining stresses in two or three dimensions, with the result compared to the tensile strength of the material loaded in one direction.

11. Young's Modulus

The proportionality constant, or ratio of stress to strain, is known as young's modulus or modulus of elasticity.

12. Poisson's Ratio

Poisson's ratio is the ratio of transverse contraction strain to longitudinal extension strain in the direction of stretching forces.

13. CNC

The abbreviation CNC stands for computer numerical control, and refers to a computer "controller" that reads G-code instructions and drives the machine tools, a powered mechanical device typically used to fabricate metal components by the selective removal of metal.

CHAPTER – II

REVIEW OF LITERATURE

Historical Overview

In the first quarter of the 20^{th} century, a rigid mode of mass production replaced batch-order production and job-order fabrication of products. There was a massive increase in the household incomes in North America and Europe, followed by the large-scale production of household appliances and motor vehicles. However, product complexity and manufacturing inadaptability led to long product life cycles, resulting in slow innovation and modifications. Then came the boom in international manufacturing industries in post World War II in Western Europe, U.S.A., and Japan. These global industries competed for their respective market shares with the domestic companies within these countries. Due to globalization, the industries reconstructed their prior-to-manufacturing techniques, emphasizing the design to obtain a quality product that would be competitive, aesthetically and ergonomically. The engineering design starts with the customer's viewpoint or feedback or in response to an idea developed by research team, initiated on the CAD (Benhabib, B).

Before creating a solid geometric model, there are several factors that need to be considered:

1. Create optimum or optimize product architecture and conceptual product design.
2. Pursue off-the-shelf and modularity opportunities.

3. Allocate an appropriate material selection.
4. Understand the potential processes that will manufacture the parts.
5. Design for efficient manufacturing, considering the total cost.
6. Correct dimensioning and tolerance consideration (Anderson, D).

Software Review

CAD (Computer Aided Design)

Introduced in the 1960s, CAD tools are the most common geometric modeling tools with advanced improvement of designing 3D modeling. Geometric modeling is the first step in CAE (Computer-Aided Engineering) analysis of a design. The objective is to encapsulate all geometric data pertaining to the part in a single model and specify all necessary material properties as additional information. Solid modeling, as a branch of geometric modeling, refers to the geometric description of solid objects in their entirety (Groover, P., Zimmers, W).

Autodesk Inventor

Autodesk Inventor, in this thesis, is a CAD software used to design a solid model (support plate for a vise). Autodesk Inventor also includes an advanced feature of third-party analysis tool ANSYS, a finite element analysis tool, to analyze the design for stresses and deformations. Autodesk Inventor a is mechanical design software which includes assembly-centric solid modeling and drawing production systems, tools, and commands to complete fully parametric three-dimensional parts, assemblies, and presentations, and two-dimensional drawings. Autodesk Inventor provides a combination of industry-specific tools that extend the capabilities of solid modeling techniques for completing complex machinery and other product designs. Figure 1 shows the Autodesk Inventor interface (Madsen, P).

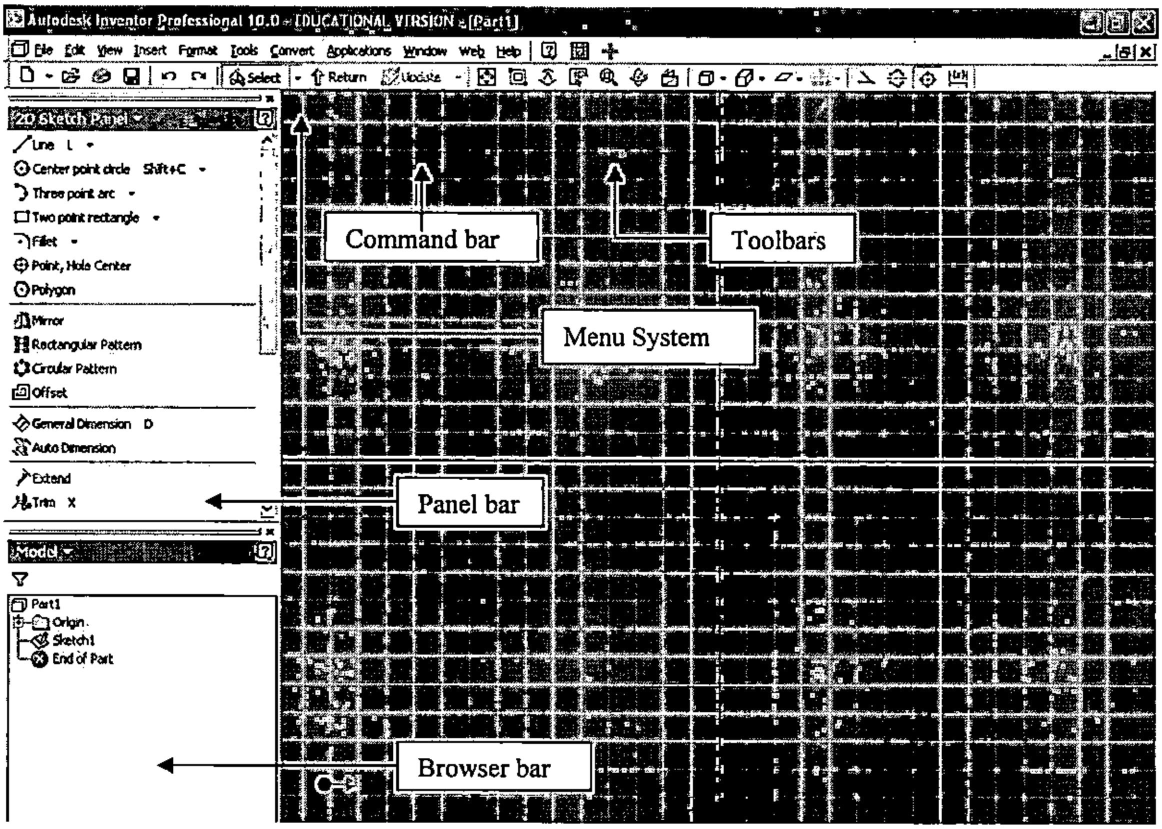

Figure 1. Autodesk Inventor Professional Environment.

The term "interface" used here is to describe the tools and techniques used to provide and receive information to and from a computer program. The interface window includes the following:

Menu system bar

The menu system interface has several groups of pull-down menu features which provide the text-based, menu configuration. A designer can pick any of these pull-down menus to reveal the options for that menu, and move the cursor down and up to highlight any of the options.

Panel bar

The panel bar provides the list of several design tools available for creating models in a specific work environment where we are currently working. For example, when working on 2D drawing or sketch mode, only sketching tools are available; when moved to finish sketch, then the features menu becomes available.

Browser bar

The browser stores and displays all the objects in the model. The design can be modified using the right-click on the processes done on the model, using the edit mode.

Toolbars

Toolbars and toolbar icons are used to perform specific tasks which help in modifying the model. Some of the features of tool bars are rotate, zoom to fit, pan, zoom area, etc.

Command bar

Command bar is one of the important tools used in Autodesk Inventor, which contains select, sketch, update, and color or style drop down lists.

Finite Element Analysis Review

The finite element analysis tool in this thesis is a tool to understand better how a design will perform under certain conditions. Here the stress analysis tool is a third-party application tool supported by ANSYS. Autodesk Inventor Professional stress analysis provides tools for determining structural design performance directly on the Autodesk Inventor model. The stress analysis includes tools to place loads and constraints on a part and calculate the resulting stress, deformation, safety factor, and resonant frequency modes. A completely virtual product development system saves money and moves products to market faster (Madsen, P). Figure 2 shows the AIP stress analysis environment.

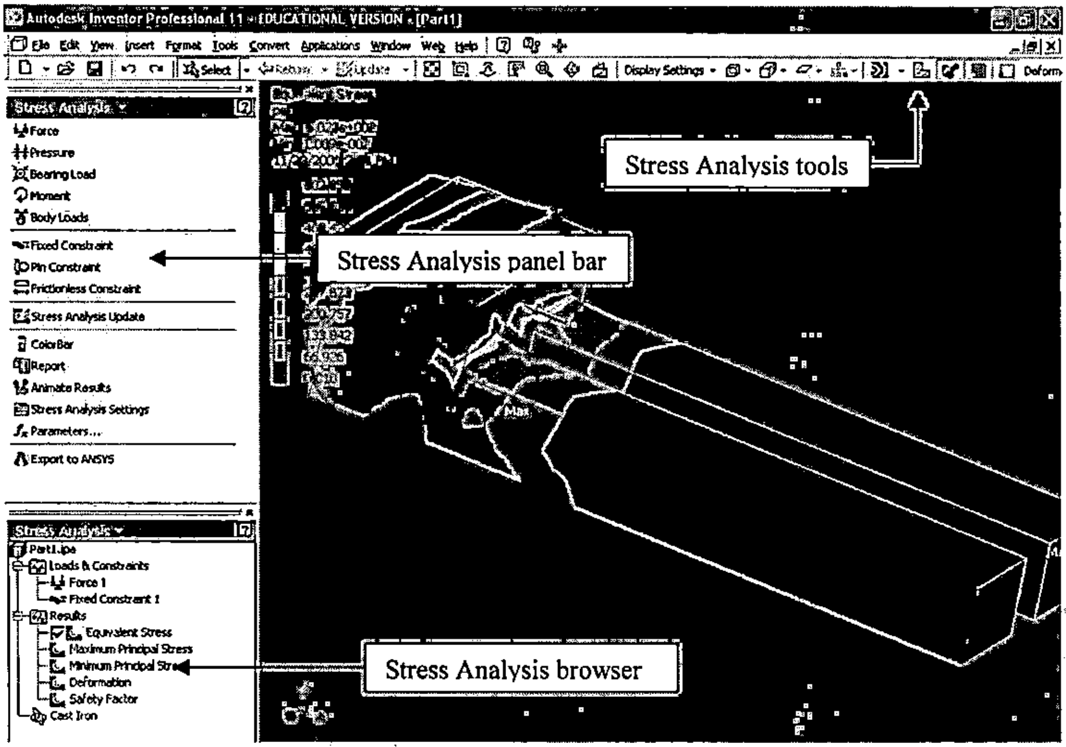

Figure 2. Stress Analysis Environment.

The design phase is the key phase of a manufacturing system, so it is of prime significance to perform an analysis of a mechanical part designed in the design phase to bring a better product to market in less time. The analysis of design helps determine if the part is strong enough to withstand expected loads or vibrations without breaking or deforming inappropriately (Hryniewieki, J). AIP stress analysis also helps in gaining valuable insight at an early stage when the cost of redesign is small and helps in predicting whether the part can be redesigned in a more cost-effective manner and still performs satisfactorily under expected use. Autodesk Inventor professional stress analysis environment consists of the following menus:

Stress analysis environment

The stress analysis environment enables the designer to simulate the part's behavior under externally imposed loads and frequencies. The Stress Analysis Update tool then performs an analysis to evaluate the performance of the part in response to the applied force and constraints.

The stress analysis panel bar

The stress analysis panel bar includes stress analysis tools briefly described below:

Force

The force tool is used to apply forces of the selected magnitude on the selected faces, edges, or vertices, to analyze the behavior of the design under the applied force.

Pressure

The pressure tool is used to apply forces of the selected magnitude on the selected faces to analyze the behavior of the design under the applied pressure.

Bearing load

The bearing load tool is used to apply bearing load of the selected magnitude on the selected cylindrical faces to analyze the behavior of the design under the applied bearing load.

Moment

The moment tool is used to apply the moment of selected magnitude on the selected faces to analyze the behavior of the design under the applied moment of inertia.

Body loads

The body load tool is used to apply the moment of selected magnitude on the part to analyze the behavior of the design under the applied body load.

Fixed constraint

The fixed constraint tool is used to place a fixed constraint on any of the faces, edges, or vertices defined in the part. To apply a fixed constraint you must first select a valid material and enter the stress analysis environment.

Pin constraints

Pin constraint tool is used to apply pin constraint of the selected combination on cylindrical and curved faces.

Frictionless constraint

The frictionless constraint tool is used to apply frictionless constraint on faces. Frictionless constraint prevents the surface from moving or deforming in the normal direction relative to the surface. The surface is free to rotate, move, or deform in tangential direction to the applied frictionless constraint.

Stress analysis update

Stress analysis update is used to make change to the part, or stress analysis settings after the part is analyzed. Use the Stress Analysis Update tool to run a new analysis on the part. The tool analyzes the part considering the changes made and generates a valid solution.

Report

The Stress Analysis Report documents the design and analysis information created. It is divided into sections that correspond to objects in the browser. Each scenario in the report represents one complete engineering simulation. The definition of a simulation includes known factors about a design such as material properties of the part, and types and magnitudes of loads and constraints.

Animate results

Using the Animate Results tool, you can visualize the part through various stages of deformation. The designer can also animate stress, safety factor, and deformation under frequencies.

Stress analysis setting

Using Stress Analysis Settings on the stress analysis panel bar, you can modify the mesh fineness on the scale of -100 to 100. The element size is determined based on the size of the body box, the proximity of other topologies, body curvature, and the complexity of the feature.

Export to ANSYS

The Export to ANSYS tool is used to create a copy of the analysis file which is compatible with ANSYS Workbench. The file is stored with a .dsdb extension.

Stress analysis browser

The stress analysis browser displays all the defined loads and constraints, input conditions, and results of the part. Loads and constraints are placed at the top of the hierarchy after the origin folder, and before all other folders. You can also view the type of material selected for the analysis.

CHAPTER – III

METHODOLOGY

General Description

This chapter discusses the design and analysis of a vise base plate which undergoes deformation due to the forces applied when clamping the parts.

The vise was assumed to be modeled with the material properties of cast iron. It was also assumed that the cast iron used in this product has isotropic properties. An *Isotropic* material implies that the material properties do not vary with direction. The elastic properties of an *Isotropic* material are fully defined by a single value of *Young's Modulus* and a single value of *Poisson's Ratio* (Black, J., DeGarmo, Kohser, E., Ronald A). The controlled model shown in Figure 4 (page 20) was a variation of a model (a replica of the support plate as shown in Figure 3 on page 19) with the statistics as shown in Table 1 (page 18). The test models were modifications and improvements from the control model. The models were built and analyzed using an integrated CAD/FEA Software from Autodesk inventor 10. These models have the same length but different geometries; the change in geometries is the result of the improvement analysis.

Basic Assumptions

For the purpose of this thesis, several assumptions were made to simplify the analysis of the model. These basic assumptions are:

- For solid FEA support plate models, the material selected is assumed to be isotropic, i.e. the material properties do not vary with direction.
- Force applied on the face of the support plate model is uniformly distributed on the face.
- The temperature effects are neglected, i.e. the model is assumed to be analyzed at normal conditions.

Material Properties

- Linear - stress is directly proportional to strain.
- Constant - all properties are temperature-independent.
- Homogeneous - properties do not change throughout the volume of the part.

Table 1. Material property Assumptions for Cast Iron

Young's Modulus	1.748e+007 psi
Poisson's Ratio	0.3
Mass Density	0.2621 lbm/in³
Tensile Yield Strength	2.901e+004 psi
Tensile Ultimate Strength	4.003e+004 psi

Modeling of the SP1

The control model was designated as SP1. It was 15.06 inches long, 6 inches wide, and 2.555 inches in height as shown in Figure 4 (page 20). This model design is a modification to the clamping vise manufactured by the Kurt Work holding Manufacturing Inc. http://www.kurtworkholding.com/workholding/versatile_lock.php) as shown in Figure 3 (page 19). The modification was achieved by eliminating the support and clamping jaws. The model was design and analyzed on Autodesk Inventor Version 10. Figure 4 (page 20) shows the geometry of the model SP1.

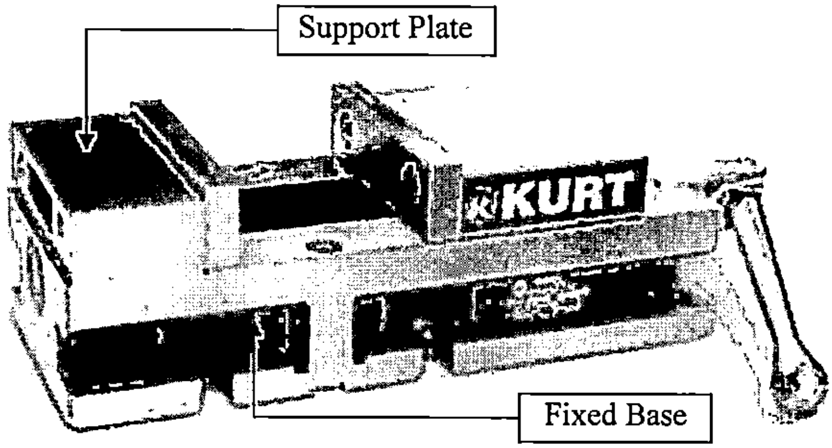

Figure 3. Work-holding Vise (Courtesy: Kurt Manufacturing Inc.)

Table 2. Statistics of SP1

Bounding Box Dimensions	Length, L: 15.06 inches Width, W: 6 inches Height, H: 2.555 inches				
Part Mass	22.42 lbm				
Part Volume	85.54 in^2				
Mesh Relevance Setting	-100	-40	0	40	100
Nodes	4145	4282	5815	10066	28539
Elements	2526	2589	3497	5715	18451

The statistics shown in Table 2 were obtained from the CAD/FEA software, which can be calculated theoretically using the formula as below:

Part Volume = L x W x H

Where L = Length, W = Width, and H = Height

Part Mass can be determined using the formula:

Density = Mass / Volume

Model Formation of SP1

The model formation starts with the actual measurement of the vise using a standard inches scale. The profile of the Kurt Workholding Manufacturing Inc. vise was carefully measured by using vernier calipers and rulers. Using the measurements, a profile was drawn on the standard inches drawing window of Autodesk Inventor Professional as shown in Figure 4.

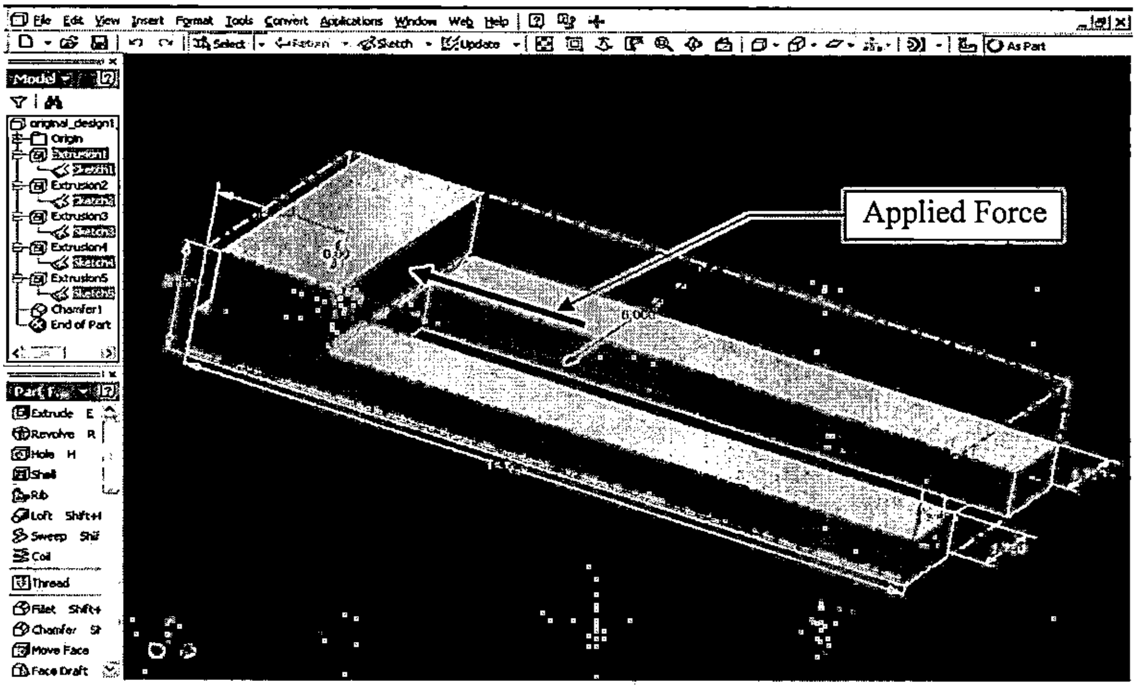

Figure 4. Designed Model of SP1

Model Analysis of SP1

The designed model SP1 was then taken to the stress analysis environment using an integrated and powerful tool of finite element software ANSYS. The stress analysis environment was made active using just one click of the mouse. The panel bar changes from part feature tools to stress analysis tools.

At first, material was selected; the product was considered to be built of cast iron, so cast iron is selected from the drop-down menu Format, which gives a document setting window as shown in Figure 5.

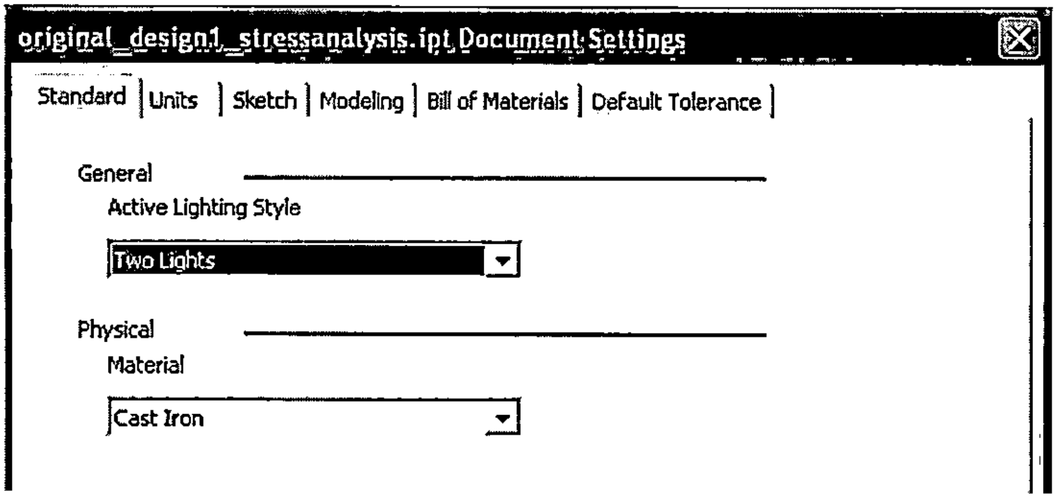

Figure 5. Document Setting Window

Constraint was then applied to the clamping plate which helps to imitate the real-life situation, where the vise is tight fitted on a table (can be located on CNC table, shop floor table, etc). Figure 6 (page 22) shows the window for applying fixed constraints.

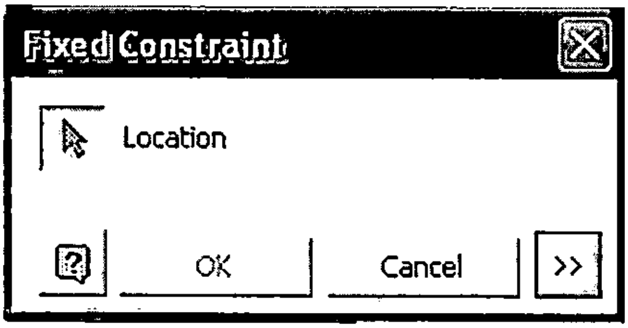

Figure 6. Window Enabling Fixed Constraint

Once the plate was constraint, force 10lbf was applied to SP1 on the face as shown in Figure 4 (page 20), setting the mesh relevance setting to -100. Figure 7 shows the window for selecting forces and its magnitude.

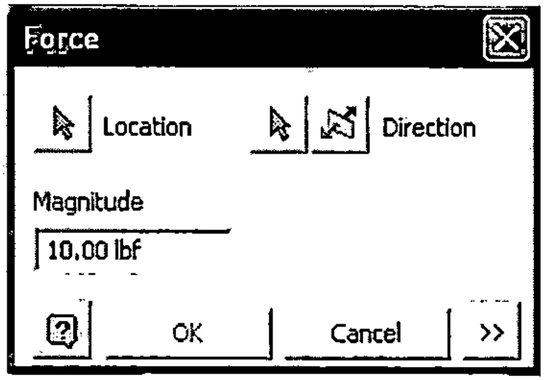

Figure 7. Window for Applying Forces

All the parameters were first defined and then analysis was performed using the stress analysis update tool located on the panel bar; the stress analysis update tool allows making changes to the part, or changing the analysis setting after the part is analyzed.

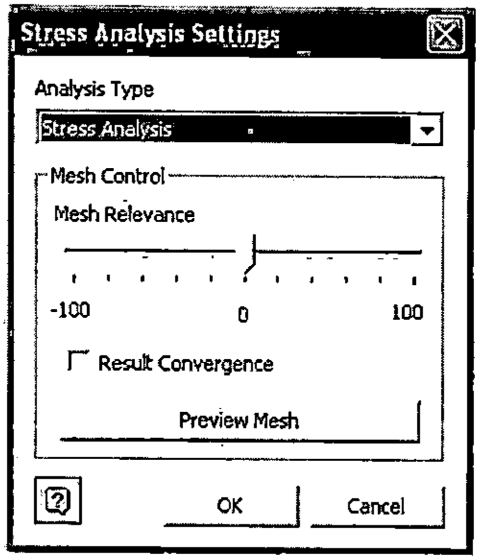

Figure 8. Mesh Relevance Setting Diagram

The Integrated ANSYS selects the number of nodes and elements automatically with the set mesh relevance setting of Zero as shown in Table 2 (page 19). The mesh relevance setting controls the fineness and coarseness of the mesh used in the analysis. As shown in Figure 8, a setting of -100 produces a coarse mesh, fast solutions, and results that may include significant uncertainty; a setting of +100 generates a fine mesh, longer solution times, and the least uncertainty in results. The change in mesh relevance setting has significant effect on the results, showing relative change in the values. The model SP1 is simulated for six different forces with five different mesh settings for each force, using the same geometry. The results recorded are in the form of deformation and equivalent stresses.

The stress analysis tool has the ability to document the design and analysis information the designer creates, and the Report tool on the panel bar creates the document in HTML format. The report also shows the diagram of deformation caused, due to the load on the SP1.

Modeling of the SP2

Model Formation of SP2

The second design to be modeled was designated SP2. It was 15.06 inches long, 6 inches wide and 2.555 inches high. This model was a basic model using the same bounding box as used for SP1. Figure 9 (page 25) shows the model of the modified vise with no fillets and cuts. This model was the basic model with simple design and time consuming machining process. The browser panel bar as shown in Figure 10 (page 25) shows the step-by-step process to design the part. Table 3 shows the statistics of SP2, showing the five different meshes with automatic generated nodes and elements for each mesh. Due to the change in geometry, there is an increment in the part mass and part volume compared to model SP1.

Table 3. Statistics of SP2

Bounding Box Dimensions	Length, L: 15.06 inches Width, W: 6 inches Height, H: 2.555 inches				
Part Mass	23.06 lbm				
Part Volume	87.98 in^2				
Mesh Relevance Setting	-100	-40	0	40	100
Nodes	620	1217	1726	3300	5963
Elements	280	594	833	1743	3352

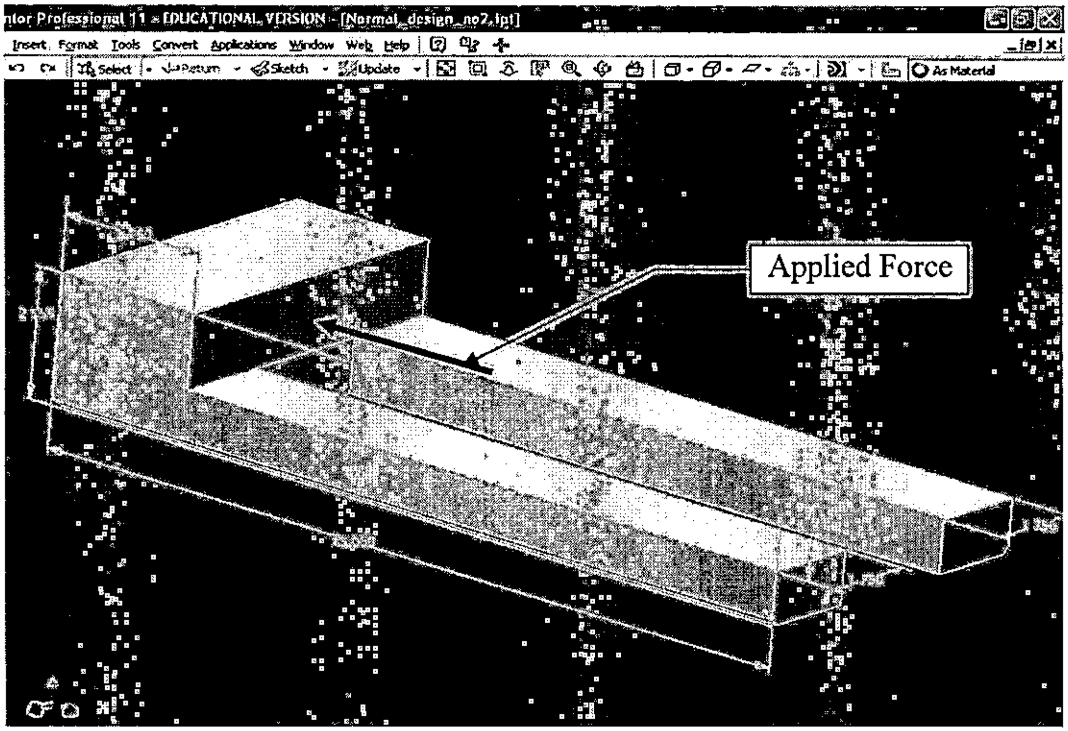

Figure 9. Designed Model of SP2

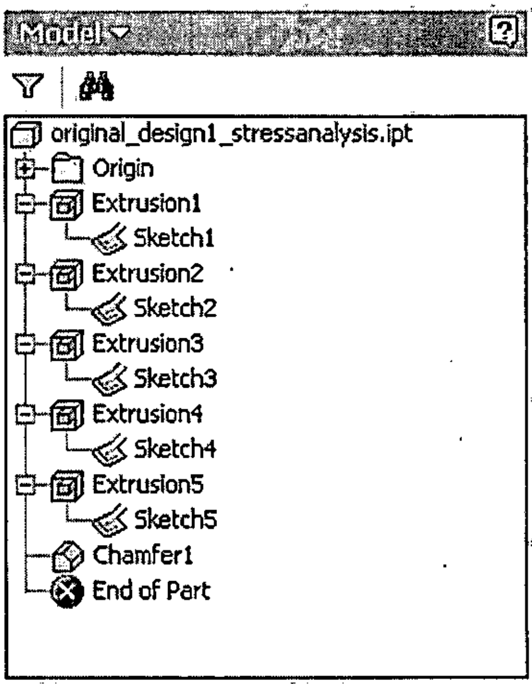

Figure 10. Solid Modeling Steps

Model Analysis of SP2

The FEA mesh for SP2 was regenerated from the method described under the model analysis of SP1. Material Selection is the first step in the model analysis to assign the part with the material properties as shown in Table 1 (page 18). The model SP2 was considered to be built with cast iron. Then the stress analysis setting tool shown in Figure 8 (page 23) is made active to select the required mesh from coarser to finer. The methodology uses five different meshes of -100, -40, 0, 40, and 100. Then constraint was applied at the bottom of the SP2 with the default zero load setting holding the vise to a fixed position. Load 10lbf is applied on the face of the clamp as shown in Figure 9 (page 25). Once the parameters are set the Stress Analysis Update tool was selected to start the analysis. After obtaining the result for 10lbf force, mesh -100, the model was analyzed for mesh -40, similarly for mesh 0, mesh 40 and mesh 100. Then force 20lbf, 30lbf, 40lbf, 50lbf, and 60lbf are applied to SP2 for five different meshes. The results indicating deformations and equivalent stresses are gathered and recorded in ascending order as shown in Table 9 (page 45).

Modeling of SP3

Model Formation of SP3

The design model 3 was designated as SP3. SP3 was modified using the same bounding box, but with a slight change in the geometry, increasing the part mass. The modification of SP3 was random modification with the result unknown. Figure 11 (page 27) shows'the solid model of SP3. The model formation steps are as shown in Figure 12 (page 28) which includes the time consuming model formation features.

Table 4 lists the statistics of SP3 showing the five different meshes with automatic generated nodes and elements for each mesh. Due to the change in geometry, there is a small change in the part mass and part volume.

Table 4. Statistics of SP3

Bounding Box Dimensions	Length, L: 15.06 inches Width, W: 6 inches Height, H: 2.555 inches				
Part Mass	22.71 lbm				
Part Volume	86.63 in^2				
Mesh Relevance Setting	-100	-40	0	40	100
Nodes	687	1376	1971	3842	6532
Elements	315	686	989	2127	3738

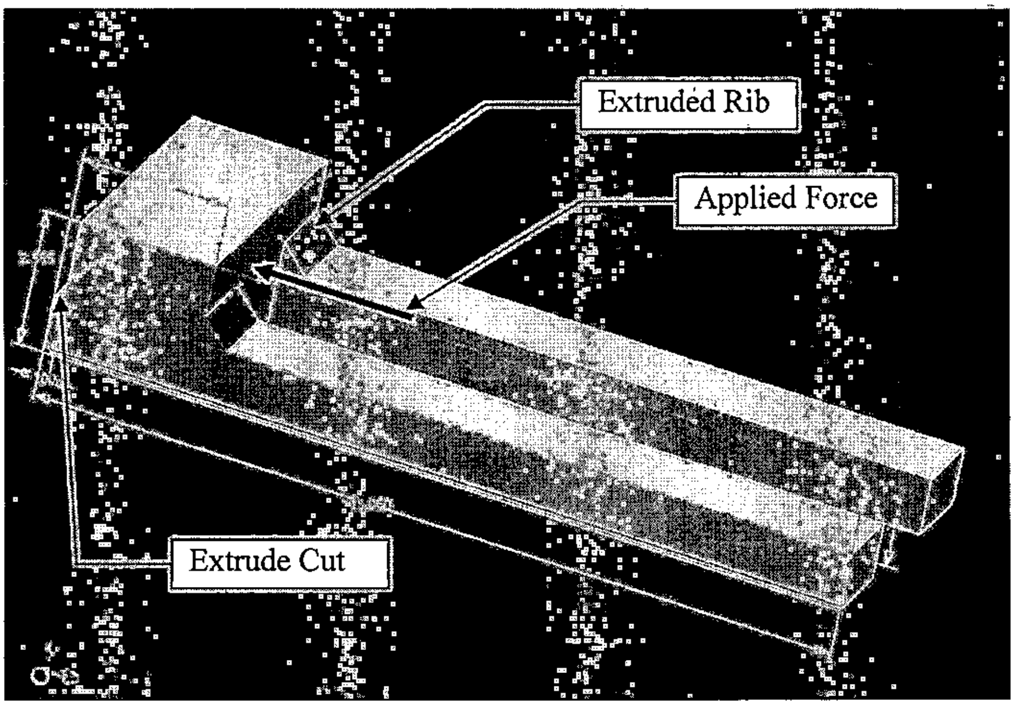

Figure 11. Designed Model of SP3

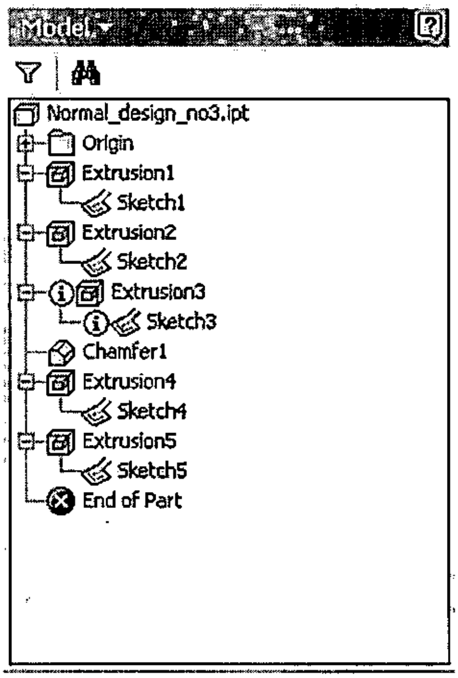

Figure 12. Solid Model Steps

Model Analysis of SP3

The FEA mesh for SP3 was regenerated from the method described under the model analysis of SP1. The first step to be considered in the model analysis is to select material and assign the part with the material properties as shown in Table 1 (page 18). The model SP3 was considered to be built with cast iron. The stress analysis setting tool shown in Figure 14 (page 29) is made active to select the required mesh from coarser to finer. The methodology uses five different meshes of -100, -40, 0, 40, and 100. The constraint is applied at the bottom of the model SP3 with the default zero load setting, holding the vise to a fixed position. Figure 13 (page 29) shows the fixed constraints tool activated, enabling the designer to apply constraints on the highlighted face of SP3.

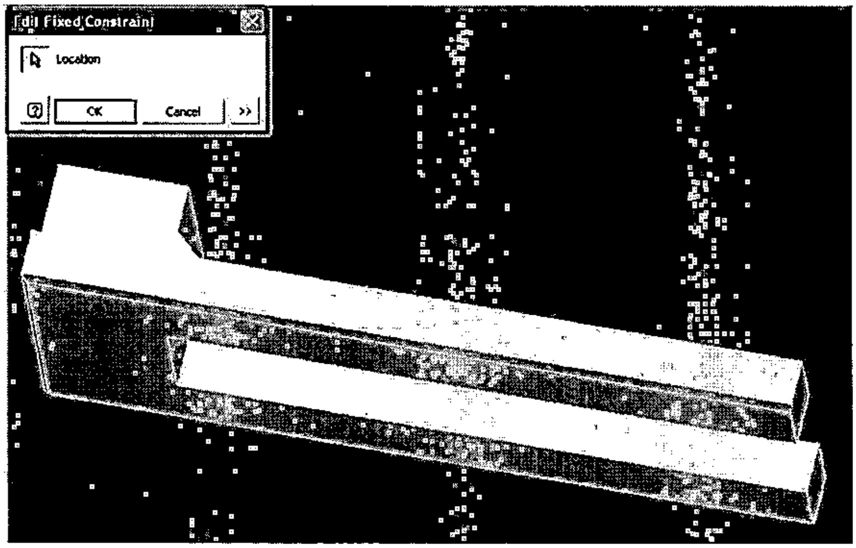

Figure 13. Shows Fixed Constraints Application.

Load 10lbf was applied on the face of the clamp as shown in Figure 11 (page 27). Then the mesh relevance setting was adjusted on -100 as shown in Figure 14, which makes the mesh coarser. Once the parameters are set the Stress Analysis Update tool was selected to start the analysis.

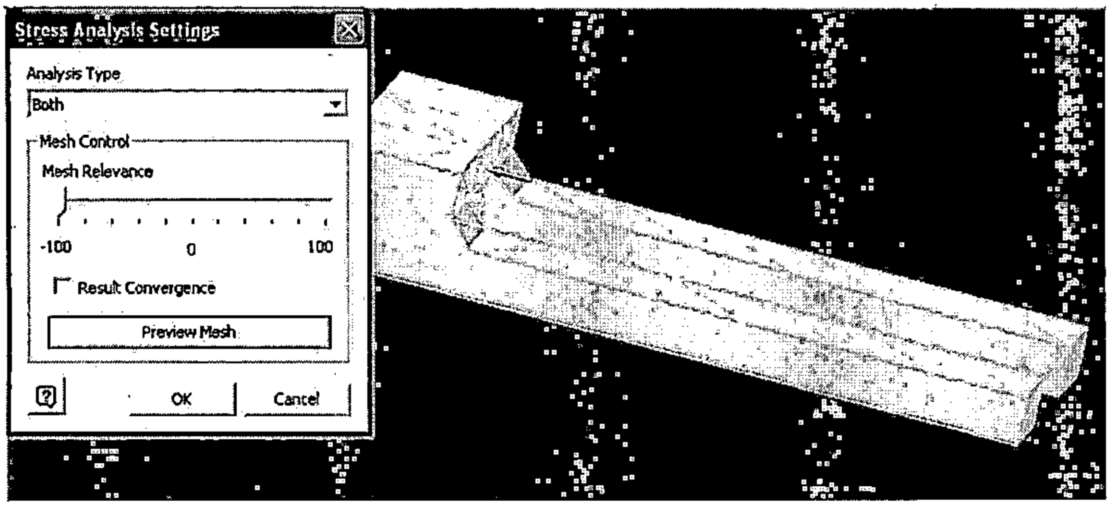

Figure 14. Stress Analysis Setting

After obtaining the result for 10lbf force, mesh -100 the model was analyzed for mesh -40, similarly for mesh 0, mesh 40 and mesh 100. Then force 20lbf, 30lbf, 40lbf, 50lbf, and 60lbf are applied to SP3 for five different meshes. The results in the form of deformations and equivalent stresses are gathered and recorded in ascending order as shown in Table 10 (page 49).

Modeling of SP4

Model Formation of SP4

The design model 4 was designated by SP4. SP4 was designed using the same bounding box as used for SP1 with length 15.06 inches, width 6 inches, and height 2.555. Figure 15 (page 31) shows the solid model of SP4. The model formation steps are as shown in Figure 16 (page 31) which includes the time consuming model formation features. Table 5 lists the statistics of SP4, showing the five different meshes with automatic generated nodes and elements for each mesh. Due to the change in geometry, there is a small change in the part mass and part volume, compared to SP3.

Table 5. Statistics of SP4

Bounding Box Dimensions	Length, L: 15.06 inches Width, W: 6 inches Height, H: 2.555 inches				
Part Mass	22.37 lbm				
Part Volume	85.33 in^2				
Mesh Relevance Setting	-100	-40	0	40	100
Nodes	597	1194	1747	3449	6297
Elements	261	574	837	1845	3579

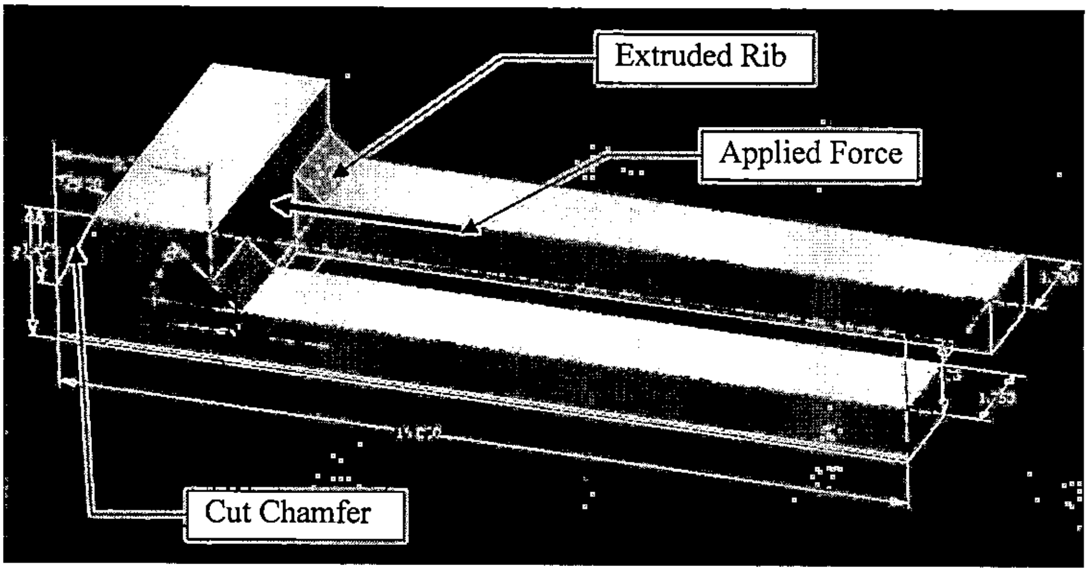

Figure 15. Design Model of SP4

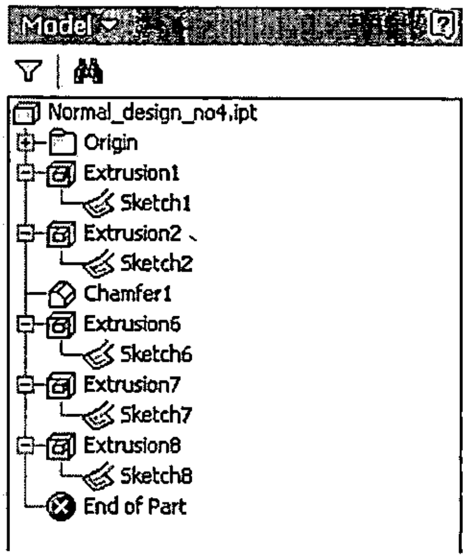

Figure 16. Solid Modeling Steps

Model Analysis of SP4

The FEA mesh for SP4 was regenerated from the method described under model analysis of SP1. Material Selection is the first step in the model analysis to assign the part with the material properties as shown in Table 1 (page 18). The model

SP4 was considered to be built with cast iron. Then the stress analysis setting tool shown in Figure 18 (page 33) was made active to select the required mesh from coarser to finer. The methodology uses five different meshes of -100, -40, 0, 40, and 100. Then constraint was applied at the bottom of the SP4, with the default zero load setting holding the vise to fixed position. Figure 17 shows the fixed constraints tool activated, enabling the designer to apply constraints on the highlighted face of SP4.

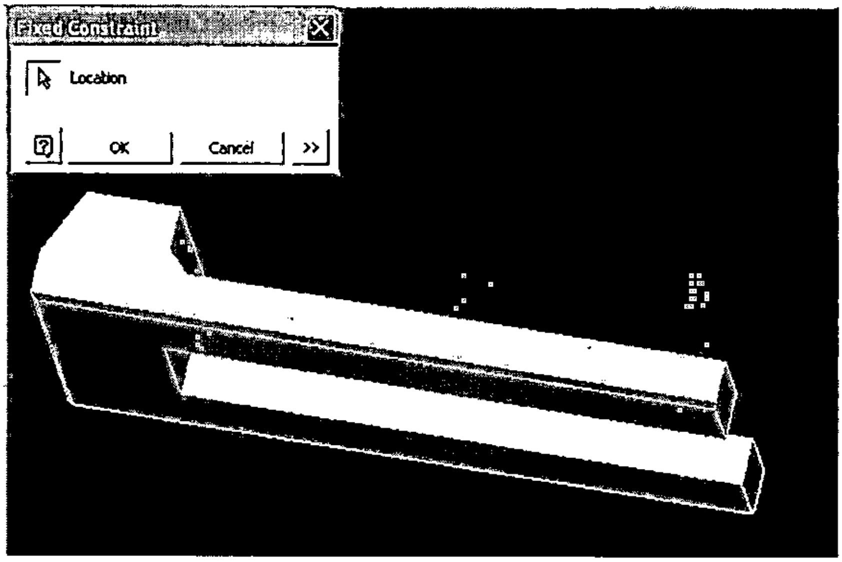

Figure 17. Fixed Constraint Application

Load 10lbf is applied on the face of the clamp as shown in Figure 15. Then the mesh relevance setting was adjusted on -100 as shown in Figure 18 (page 33), which makes the mesh coarser. Once the parameters are set, the Stress Analysis Update tool is selected to start the analysis.

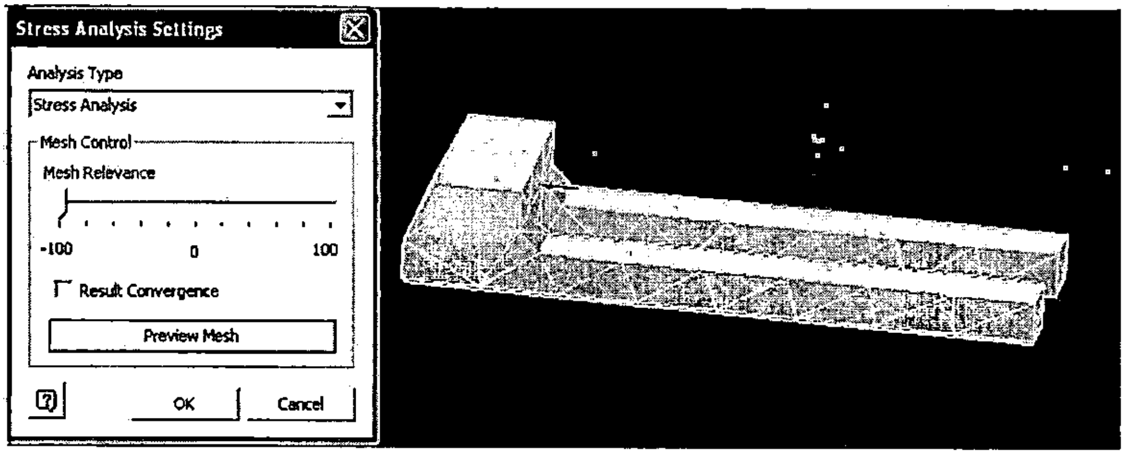

Figure 18. Stress Analysis Settings

After obtaining the result for 10lbf force, mesh -100 the model was analyzed for mesh -40, similarly for mesh 0, mesh 40 and mesh 100. Then force 20lbf, 30lbf, 40lbf, 50lbf, and 60lbf are applied to SP4 for five different meshes. The results in the form of deformations and equivalent stresses are gathered and recorded in ascending order as shown in Table 11 (page 53).

Modeling of SP5

Model Formation of SP5

The design model 5 was designated by SP5. SP5 was designed using the same bounding box as used for SP1 with length 15.06 inches, width 6 inches, and height 2.555. Figure 19 (page 34) shows the solid model of SP5. The model formation steps are as shown in Figure 20 (page 35) which includes the time consuming model formation features. Table 6 (page 34) lists the statistics of SP5, showing the five different meshes with automatic generated nodes and elements for each mesh. Due to the change in geometry, there is an increase in the part mass and part volume.

Table 6. Statistics of SP5

Bounding Box Dimensions	Length, L: 15.06 inches Width, W: 6 inches Height, H: 2.555 inches				
Part Mass	22.44 lbm				
Part Volume	85.60 in^2				
Mesh Relevance Setting	-100	-40	0	40	-100
Nodes	1175	1751	3185	5231	7648
Elements	625	915	1780	3021	4408

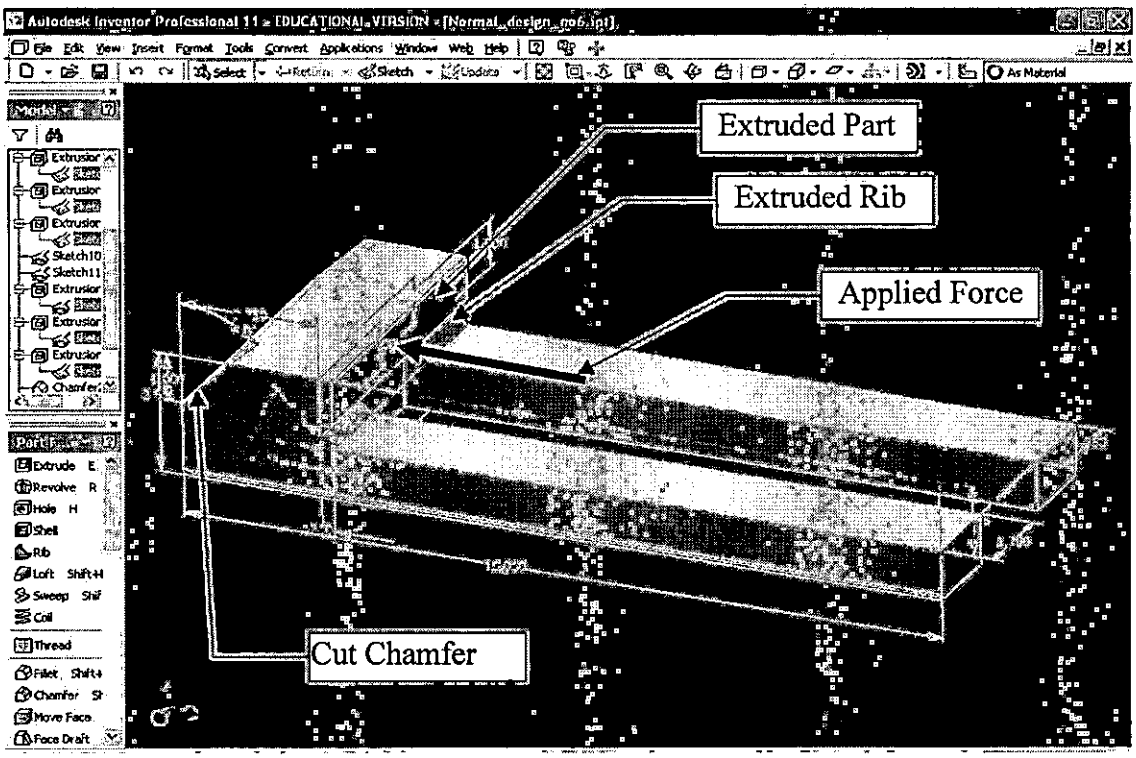

Figure 19. Design Model of SP5

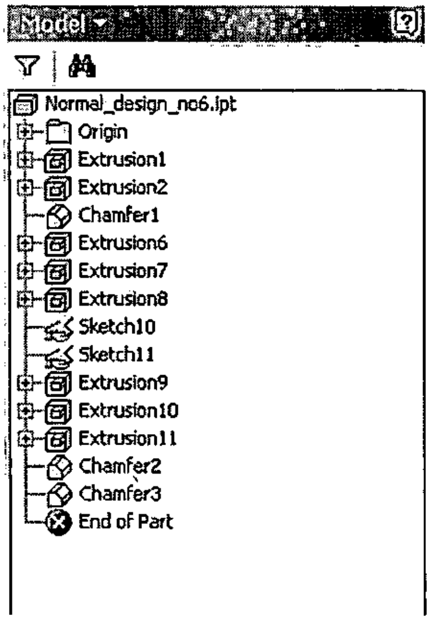

Figure 20. Solid Modeling Steps

Model Analysis of SP5

The FEA mesh for SP5 was regenerated from the method described under the title, model analysis of SP1. As discussed in the model analysis of SP4, the first step in performing model analysis is to select an appropriate material to assign the part with the material properties as shown in Table 1 (page 18). The model SP5 was considered to be built with cast iron. Then the stress analysis setting tool shown in Figure 22 (page 36) was made active to select the required mesh from coarser to finer. The methodology uses five different meshes of -100, -40, 0, 40, and 100. Then constraint is applied at the bottom of the SP5 with the default zero load setting holding the vise to a fixed position. Figure 21 (page 36) shows the fixed constraints tool activated enabling the designer to apply constraints on the highlighted face of SP5.

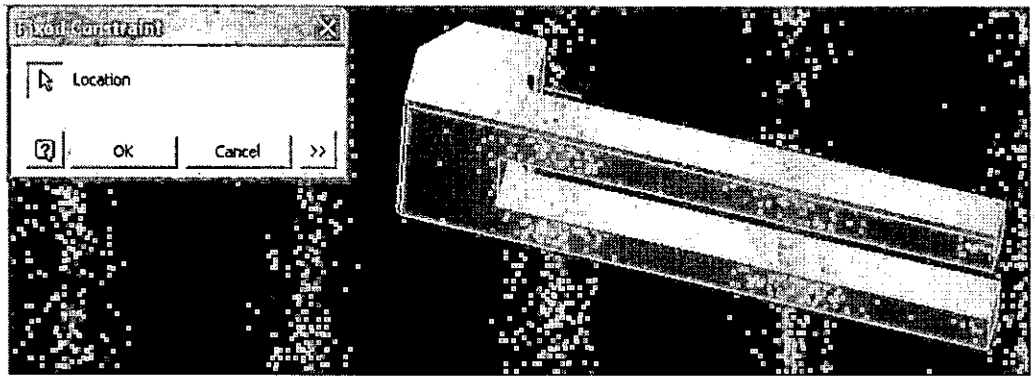

Figure 21. Fixed Constraint Application

Load 10lbf was applied on the face of the clamp as shown in Figure 19 (page 34). Then the mesh relevance setting was adjusted on -100 as shown in Figure 22, which makes the mesh coarser.

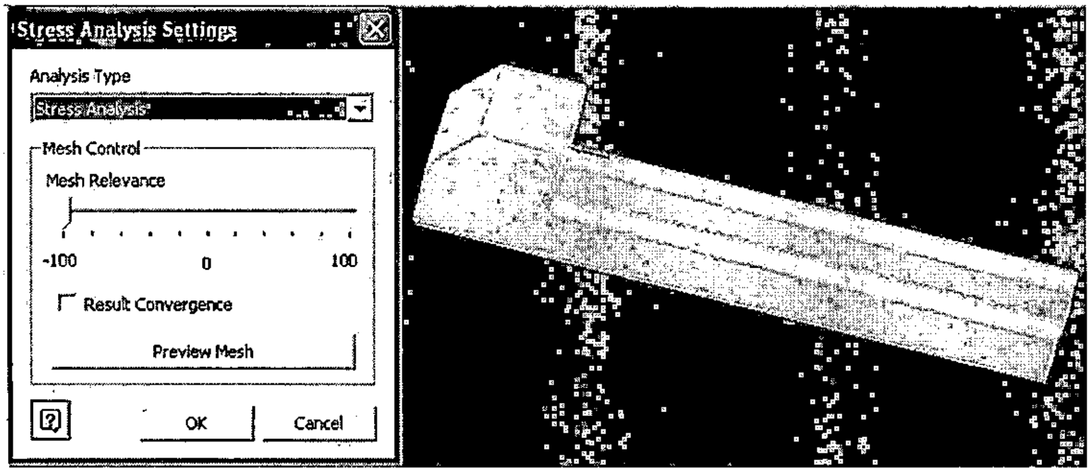

Figure 22. Stress Analysis Settings

Once the parameters are set the Stress Analysis Update tool is selected to start the analysis. After obtaining the result for 10lbf force, mesh -100 the model is analyzed for mesh -40, similarly for mesh 0, mesh 40 and mesh 100. Then force 20lbf, 30lbf, 40lbf, 50lbf, and 60lbf are applied to SP5 for five different meshes. The results in the form of deformations and equivalent stresses are gathered and recorded in ascending order as shown in Table 12 (page 57).

Modeling of SP6

Model Formation of SP6

The design model 6 was designated by SP6. SP6 was designed using the same bounding box as used for SP1 with length 15.06 inches, width 6 inches, and height 2.555. Figure 23 (page 38) shows the solid model of SP6. The model formation steps are as shown in Figure 24 (page 38) which includes the time consuming model formation features. Table 7 lists the statistics of SP6, showing the five different meshes with automatic generated nodes and elements for each mesh. Due to the change in geometry there is decrement in the part mass and part volume, compared to SP5.

Table 7. Statistics of SP6

Bounding Box Dimensions	Length, L: 15.06 inches Width, W: 6 inches Height, H: 2.555 inches				
Part Mass	21.1 lbm				
Part Volume	80.47 in^2				
Mesh Relevance Setting	-100	-40	0	40	100
Nodes	2160	2675	4186	6434	8765
Elements	1147	1369	2315	3654	4965

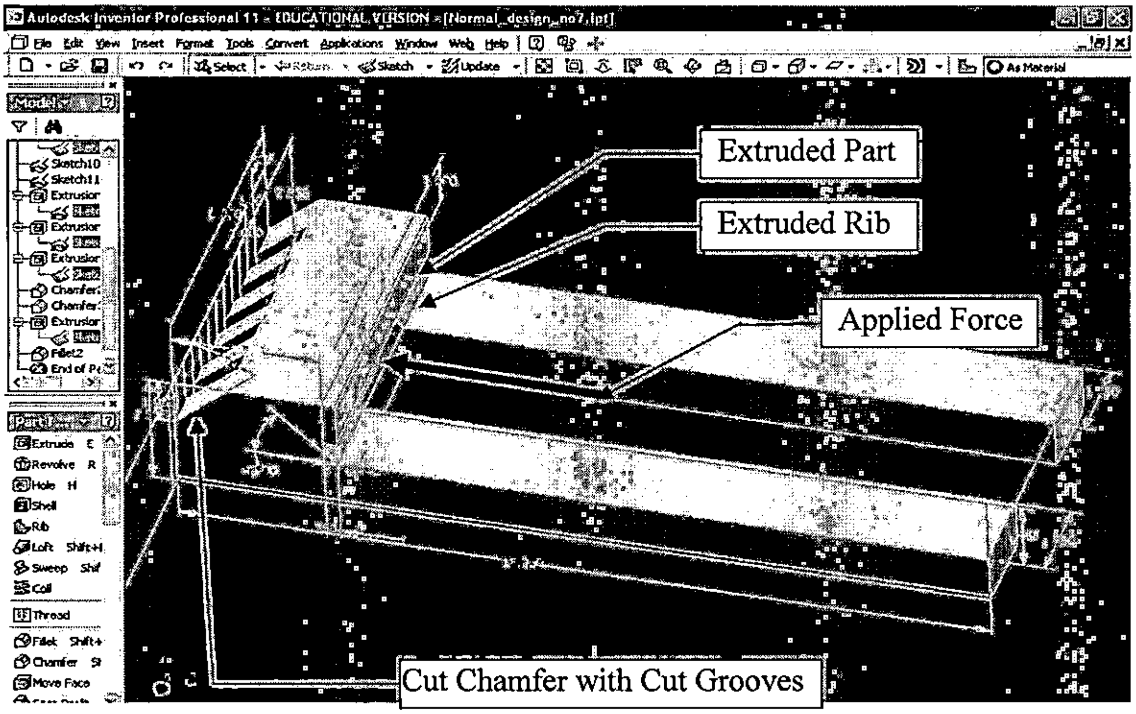

Figure 23. Designed Model of SP6

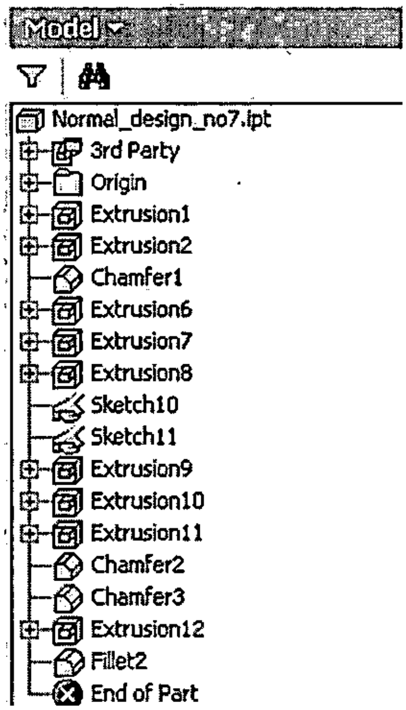

Figure 24. Solid Modeling Steps

Model Analysis of SP6

The FEA mesh for SP6 was regenerated from the method described under title model analysis of SP1. Material Selection is the first step in the model analysis to assign the part with the material properties as shown in Table 1 (page 18). The model SP6 was considered to be built with cast iron. Then the stress analysis setting tool shown in Figure 26 (page 40) was made active to select the required mesh from coarser to finer. The methodology uses five different meshes of -100, -40, 0, 40, and 100. Then constraint was applied at the bottom of the SP6 with the default zero load setting holding the vise to a fixed position. Figure 25 shows the fixed constraints tool activated, enabling the designer to apply constraints on the highlighted face of SP6.

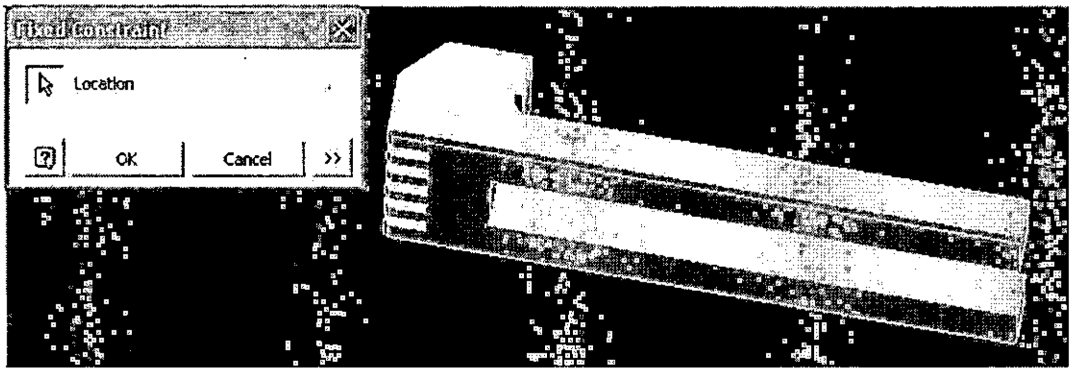

Figure 25. Fixed Constraint Application

Load 10lbf was applied on the face of the clamp as shown in Figure 23 (page 38). Then the mesh relevance setting was adjusted on -100 as shown in Figure 26 (page 40), which makes the mesh coarser.

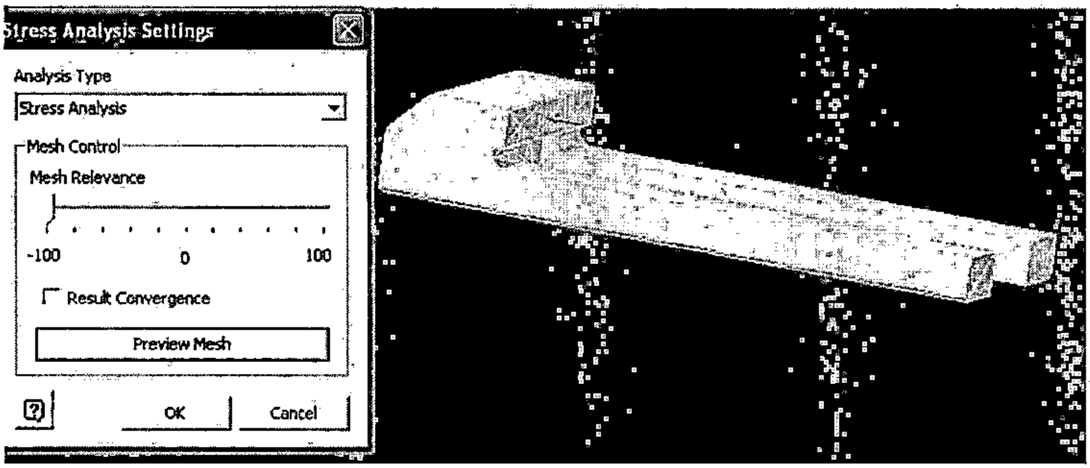

Figure 26. Stress Analysis Settings

Once the parameters are set the Stress Analysis Update tool was selected to start the analysis. After obtaining the result for 10lbf force, mesh -100, the model was analyzed for mesh -40, similarly for mesh 0, mesh 40 and mesh 100. Then force 20lbf, 30lbf, 40lbf, 50lbf, and 60lbf are applied to SP6 on the face as shown in Figure 23 (page 38) for five different meshes. The results in the form of deformations and equivalent stresses are gathered and recorded in ascending order as shown in Table 13 (page 60).

CHAPTER IV

FINDINGS AND RESULT ANALYSIS

General Remarks

This chapter describes the results obtained from the integrated CAD/FEA analysis tool of Autodesk Inventor Professional required to optimize the design SP1. The results obtained are in the form of deformations and equivalent stresses. The results are analyzed for significant differences among the six designs, using Minitab version 14.

Analysis Results of Solid Model SP1

The results for SP1 are based on the simulation of the model. The results are in the form of deformation (nm) and equivalent stress (psi) which were theoretically unknown when designing the model. SP1 was an identical design of one of the parts of a work-holding vise of Kurt Manufacturing Inc. (http://www.kurtworkholding.com/ workholding/versatile_lock.php) which was experimented upon for design optimization using CAD/FEA. Table 8 shows the collection of results recorded from separate individual simulation of six different forces on SP1. Each force was simulated for five different meshes, giving the results in the form of deformations in inches and equivalent stress in psi. The deformation was converted from inches to nanometer using the conversion of 1 inches = 25400000 nm. The conversion is a good practice, because the deformation values in terms of inches are very small to plot the results. In order to magnify the graphical representation of the statistical data, inches to nanometer is a practiced method (Burkhardt, M., DeRusseau, S., Werner, S.).

Table 8. Structural Results Table for SP1

Forces (lbf)	Mesh	Deformation (nm)	Equivalent Stress (psi)
10	-100	10.9474	7.379
10	-40	10.9474	5.761
10	0	11.176	7.297
10	40	11.1252	9.962
10	100	11.2776	9.879
20	-100	21.8694	14.76
20	-40	21.8948	11.52
20	0	22.352	14.59
20	40	22.2758	19.92
20	100	22.5806	19.76
30	-100	32.766	22.14
30	-40	32.766	17.28
30	0	33.528	21.89
30	40	33.528	29.89
30	100	33.782	29.64
40	-100	43.688	29.52
40	-40	43.942	23.04
40	0	44.704	29.19
40	40	44.45	39.85
40	100	45.212	39.52
50	-100	54.61	36.9
50	-40	54.864	28.8
50	0	55.88	36.48
50	40	55.626	49.81
50	100	56.388	49.4
60	-100	65.532	44.28
60	-40	65.786	34.57
60	0	67.056	43.78
60	40	66.802	59.77
60	100	67.818	59.28

Table 8 shows the data for 6 different forces with combination of 5 relevance mesh settings giving an output of 30 readings for equivalent stress and deformation.

The results are represented in graphical form as shown in Figure 27 and Figure 28 (page 44).

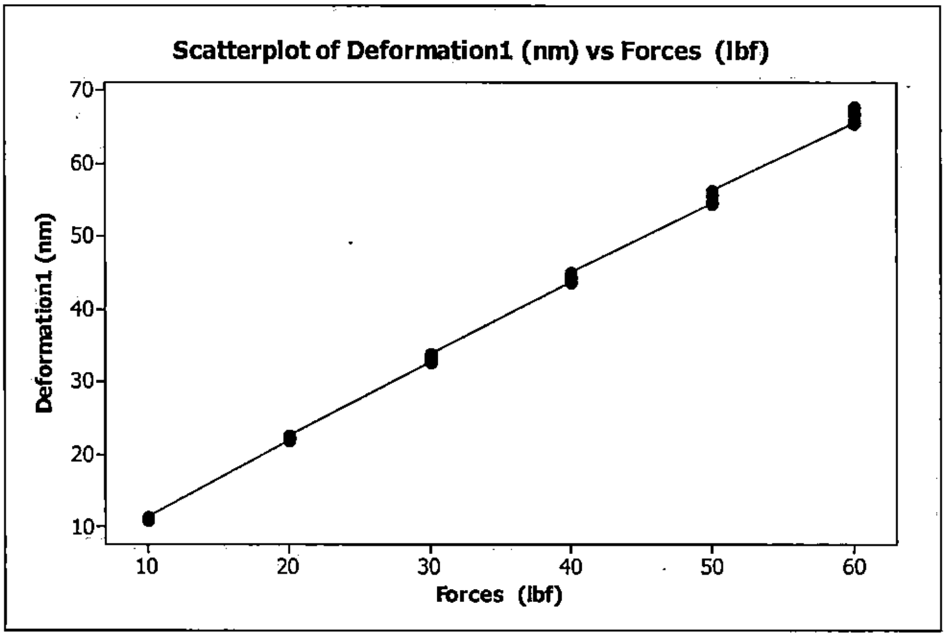

Figure 27. Deformations Vs Force Graph for SP1

The scatter plot as shown in Figure 27 reveals an association between the forces and deformation. The deformation on the vertical axis is represented as the response variable. The graph shows a strong linear (positive correlation) increment, interpreting that an increment in force increases the deformation on SP1. The graph also reveals that, with linear progression of the data between the forces and deformation, there is also a linear increment of data among the force, which interprets the variation in the deformation due to the mesh relevance setting.

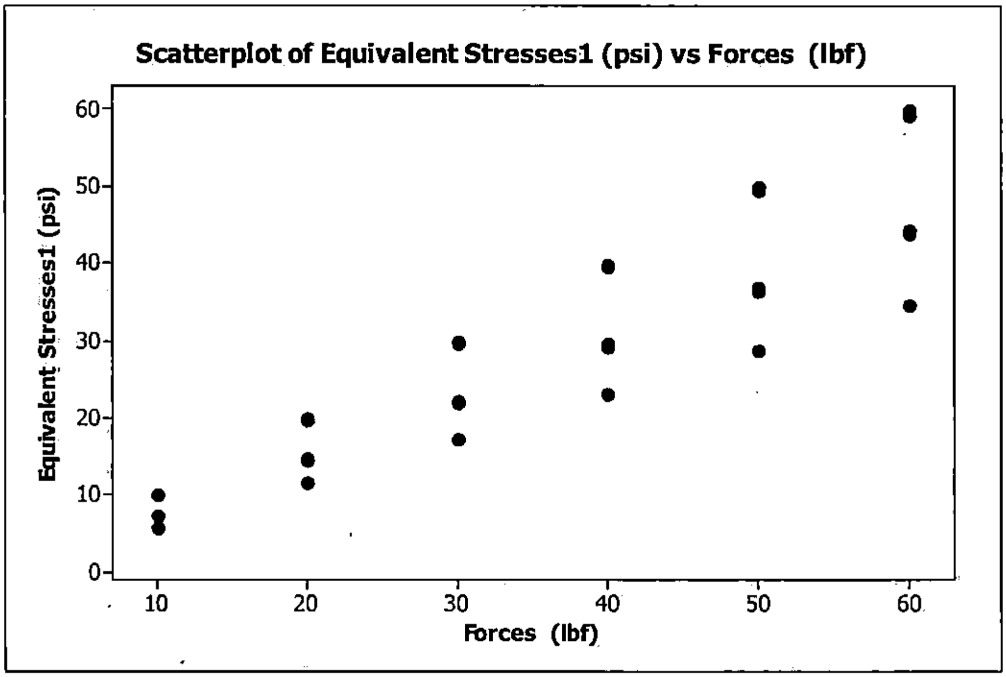

Figure 28. Equivalent Stresses Vs Forces Graph for SP1

The scatter diagram shown in Figure 28 represents that, as the force increases, the equivalent stress on the model also increases. The increment is heteroscedastic, which means there is a non-constant variation in equivalent stress over the values of forces. The graph reveals that, at force 10lbf the equivalent stress remains low and even the values of stresses for force 10lbf are close; however as the force increases (say 60lbf), the equivalent stress also increases and the values of stresses for 60 lbf are farther than those for 10lbf.

Analysis Results of Solid Model SP2

SP2 was design as the first base model randomly created using the same bounding box as used for SP1, to compare the results and reduce the built-up time using the simulation CAD/FEA software. Table 9 (page 45) shows the collection of results recorded from separate individual simulation of six different forces on SP1. Each force was simulated for five different meshes giving the results in the form of deformations in inches and equivalent stress in psi. The deformation was converted to nanometer using the conversion of 1 inches = 25400000 nm.

Table 9. Structural Results Table for SP2

Forces (lbf)	Mesh	Deformation (nm)	Equivalent Stress (psi)
10	-100	7.9248	1.811
10	-40	8.0772	2.227
10	0	8.255	2.584
10	40	8.3058	3.142
10	100	8.4074	3.325
20	-100	15.8496	3.621
20	-40	16.1544	4.453
20	0	16.51	5.168
20	40	16.637	6.823
20	100	16.7894	6.65
30	-100	23.7998	5.432
30	-40	24.2316	6.68
30	0	24.765	7.753
30	40	24.9428	10.24
30	100	24.9428	10.24
40	-100	31.75	7.242
40	-40	32.258	8.906
40	0	33.02	10.34
40	40	33.274	13.65
40	100	33.528	13.3
50	-100	39.624	9.053
50	-40	40.386	11.13
50	0	41.402	12.92
50	40	41.656	17.06
50	100	41.91	16.62
60	-100	47.498	10.86
60	-40	48.514	13.36
60	0	49.53	15.51
60	40	49.784	20.47
60	100	50.292	19.95

Table 9 (page 45) shows the analysis results for SP2 in the form of maximum equivalent stresses and deformations with respect to forces applied. The results are shown in the form of graphical representation, using Minitab. Figure 29 represents a linear progression graph of deformation versus forces showing linear increment in deformation with the increase in force. Figure 30 (page 47) represents the graphical representation of equivalent stresses versus forces with the linear increment in stresses with forces.

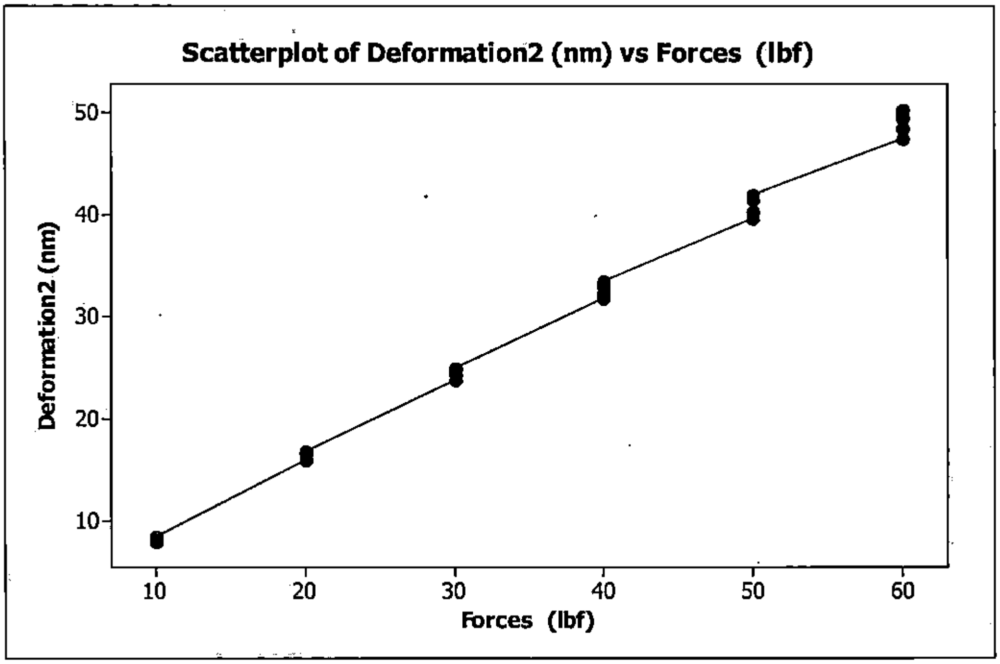

Figure 29. Deformations Vs Forces Graph for SP2

The scatter plot shown in Figure 29 shows the linear progression of data, which reveals that, as force increases, there is an increment in deformation, showing a non-constant variation in deformation over the values of forces.

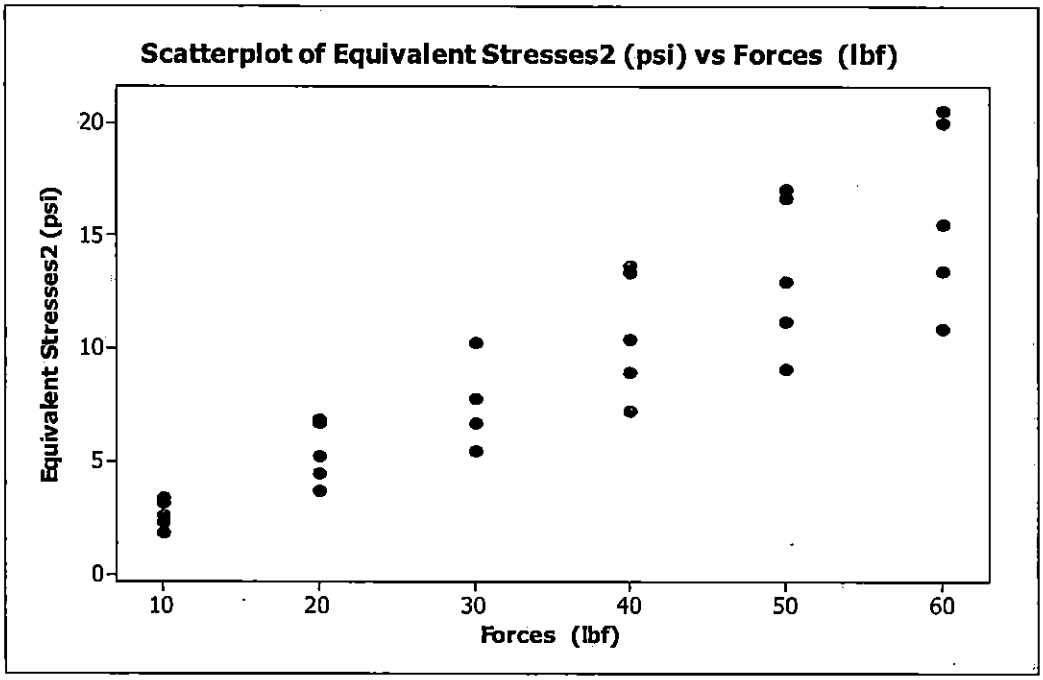

Figure 30. Equivalent Stresses Vs Forces Graph for SP2

The scatter diagram shown in Figure 30 represents that, as the force increases, the equivalent stress on the model also increases. The increment is heteroscedastic, which means there is a non-constant variation in equivalent stress over the values of forces. The graph reveals that, at force 10lbf, the equivalent stress remains low and even the values of stresses for force 10lbf are close; however as the force increases (say 60lbf), the equivalent stress also increases and the values of stresses for 60 lbf are farther than those for 10lbf.

Comparing the results of SP1 and SP2

There is a small difference between the results of SP1 and SP2; referencing from Table 8 (page 42) and Table 9 (page 45) of model SP1 and SP2, the deformation value in Table 10 (page 49) for force 10lbf and mesh -100 is 10.947 nm, whereas for the same force of 10lbf and mesh of -100, the deformation value of SP2 is 7.925 nm. The difference is 3.022 nm, which is due to an increase in mass at the applied force region of SP2. The design with less deformation will be able to hold repeated force cycles. SP2 is also capable of holding 60lbf with the deformation of 50.292 nm, whereas SP1 shows almost 68 nm. The model SP2 also shows lesser stress than SP1. Consequently, the model SP2 shows higher stiffness as compared to SP1, but SP2 uses slightly more material than SP1, increasing the volume of the model.

Analysis Results of Solid Model SP3

The structural results in Table 10 (page 49) show the data for 6 different forces with the combination of 5 relevance mesh settings giving output of 30 readings for equivalent stress and deformation. The results are represented in graphical form as shown in Figure 31 (page 50) and Figure 32 (page 51).

Table 10. Structural Results Table for SP3

Forces (lbf)	Mesh	Deformation (nm)	Equivalent Stress (psi)
10	-100	8.382	1.866
10	-40	8.5344	2.469
10	0	8.7122	2.34
10	40	8.7884	2.701
10	100	8.8392	3.022
20	-100	16.7386	3.732
20	-40	17.0688	4.937
20	0	17.4244	4.68
20	40	17.5514	5.403
20	100	17.6784	6.044
30	-100	25.0952	5.599
30	-40	25.654	7.406
30	0	26.162	7.02
30	40	26.416	8.104
30	100	26.416	9.066
40	-100	33.528	7.465
40	-40	34.036	9.875
40	0	34.798	9.359
40	40	35.052	10.81
40	100	35.306	12.09
50	-100	41.91	9.331
50	-40	42.672	12.34
50	0	43.688	11.7
50	40	43.942	13.51
50	100	44.196	15.11
60	-100	50.292	11.2
60	-40	51.308	14.81
60	0	52.324	14.04
60	40	52.578	16.21
60	100	53.086	18.13

Figure 31 (page 50) shows the scatterplot of deformation versus forces, interpreting that, as force increases on the SP3, there is a significant increase in deformation. Figure 32 (page 51) shows the scatterplot of equivalent stresses versus forces on SP3.

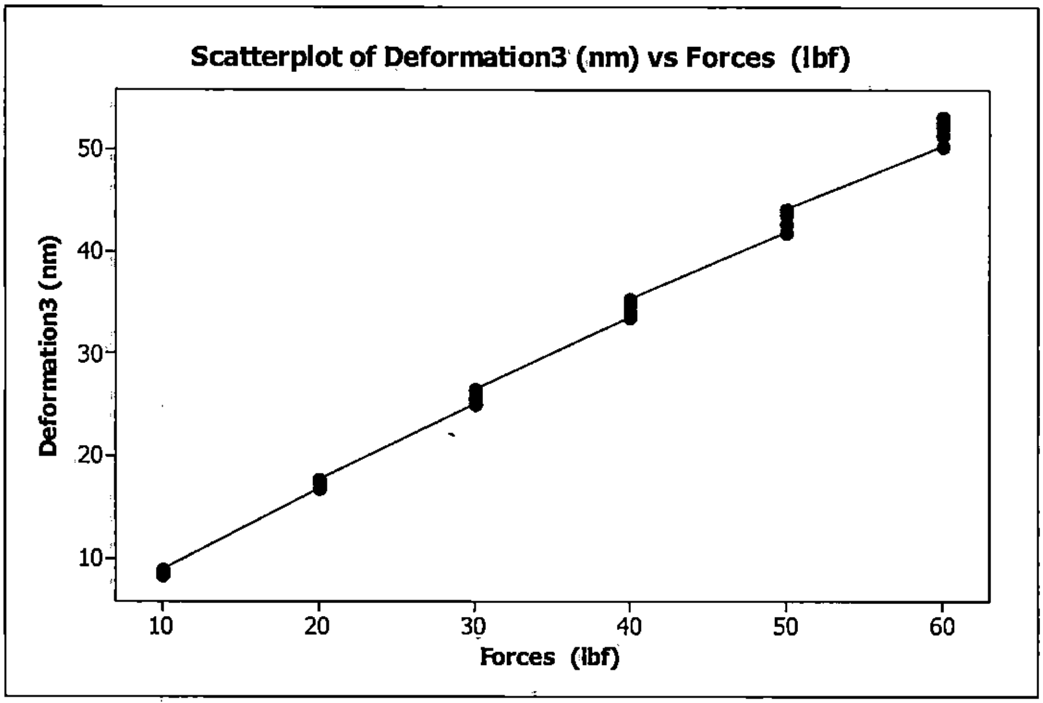

Figure 31. Deformations Vs Forces Graph for SP3

The scatter plot shown in Figure 31 shows the linear progression of data, which reveals that, as force increases, there is an increment in deformation, showing a non-constant variation in deformation over the values of forces.

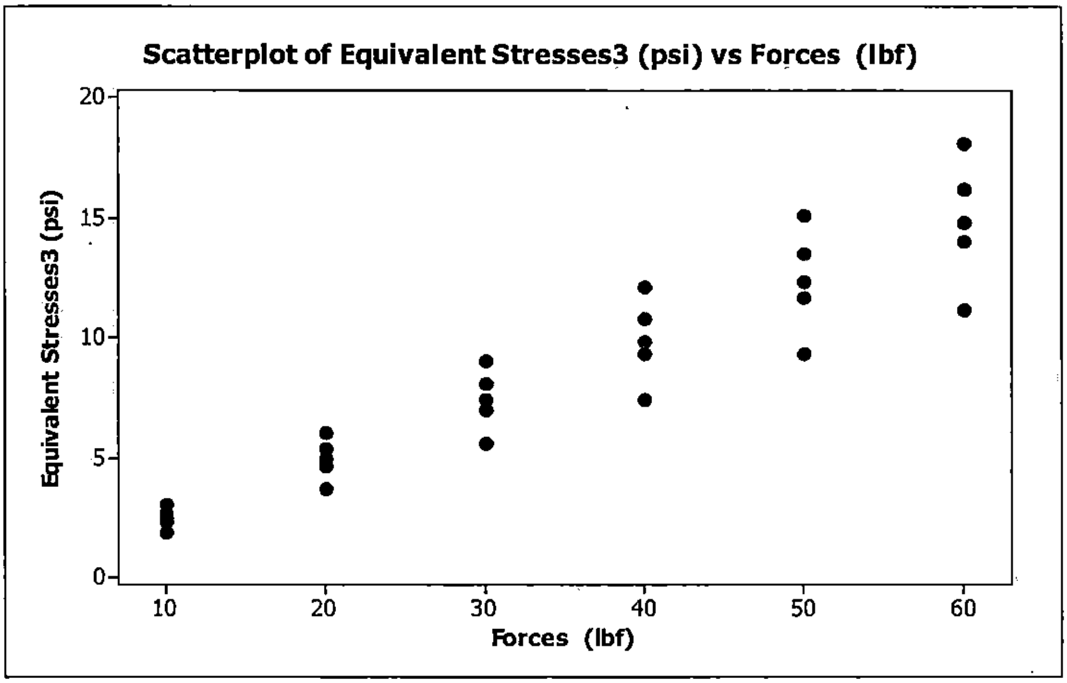

Figure 32. Equivalent stresses Vs Forces Graph for SP3

The scatter diagram shown in Figure 32 represents that, as the force increases, the equivalent stress on the model also increases. The increment is heteroscedastic, which means there is a non-constant variation in equivalent stress over the values of forces. The graph reveals that, at force 10lbf, the equivalent stress remains low and even the values of stresses for force 10lbf are close, but as the force increases (say 60lbf), the equivalent stress also increases and the values of stresses for 60 lbf are farther than those for 10lbf.

Comparing the results of SP1 and SP3

The graphical representation of SP3 for deformation listed in Figure 31 (page 50) shows similar pattern of linear progression as shown by Figure 28 (page 44), but the data representation in Figure 31 (page 50) shows more variation in the results due to mesh, as compared to Figure 28 (page 44). Figure 19 (page 34) shows the same consistent non-constant variation, whereas Figure 29 (page 46) shows more variation in the values of mesh 0 and 40. The deformation value in Table 7 for force 10lbf and mesh -100 is 10.947 nm, whereas for the same force of 10lbf and mesh of -100, the deformation value of SP3 is 8.382 nm. The equivalent stress values of SP3 are less than that of the equivalent stress of SP1. Hence, considering the result parameters, model SP1 has a greater (or a lesser) maximum percent rate of failure due to repeated force than model SP3.

Analysis Results of Solid Model SP4

Table 11 (page 53) shows the data for 6 different forces with a combination of 5 relevance mesh settings, giving an output of 30 readings for equivalent stress and deformation.

Table 11. Structural Results Table for SP4

Forces (lbf)	Mesh	Deformation (nm)	Equivalent Stress (psi)
10	-100	8.4328	1.948
10	-40	8.6106	2.492
10	0	8.6868	2.936
10	40	8.8138	3.089
10	100	8.8646	3.135
20	-100	16.8656	3.895
20	-40	17.2212	4.985
20	0	17.3736	5.873
20	40	17.6022	6.178
20	100	17.7292	6.271
30	-100	25.2984	5.843
30	-40	25.908	7.477
30	0	26.162	8.809
30	40	35.56	9.267
30	100	26.67	9.406
40	-100	33.782	7.791
40	-40	34.544	9.969
40	0	34.798	11.75
40	40	35.306	12.36
40	100	35.56	12.54
50	-100	42.164	9.738
50	-40	42.926	12.46
50	0	43.434	14.68
50	40	43.942	15.45
50	100	44.196	15.68
60	-100	50.546	11.69
60	-40	51.562	14.95
60	0	52.07	17.62
60	40	52.832	18.53
60	100	53.086	18.81

The results are represented in graphical form as shown in Figure 33 (page 54) and Figure 34 (page 55).

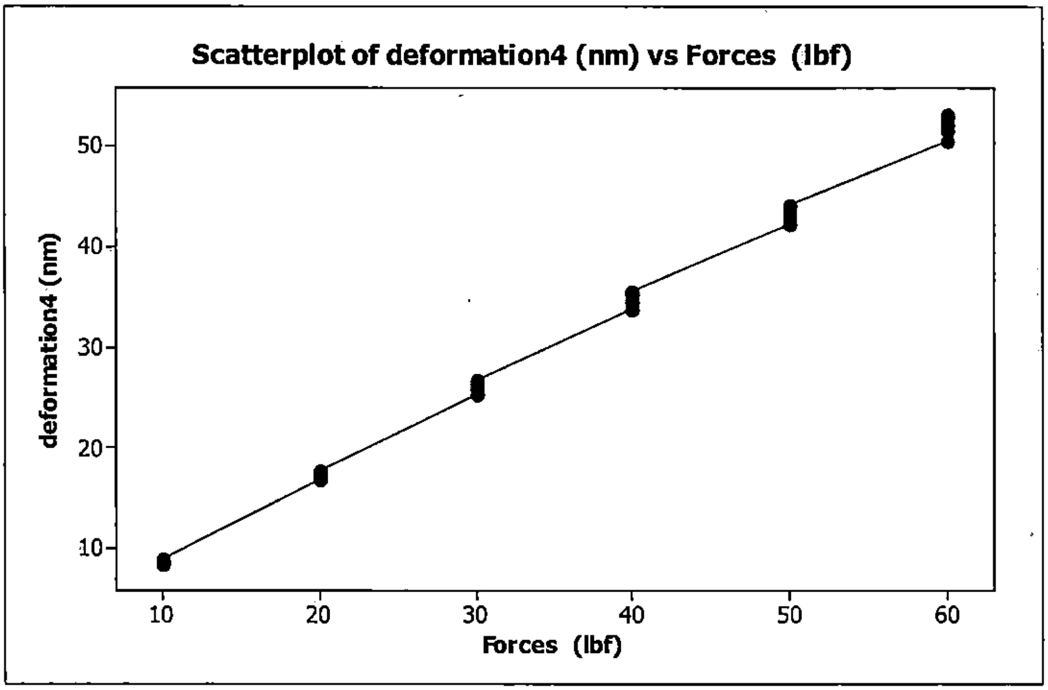

Figure 33. Deformations Vs Forces Graph for SP4

The scatter plot as shown in Figure 33 reveals an association between the forces and deformation. The deformation on the vertical axis is represented as the response variable. The graph shows a strong linear (positive correlation) increment, interpreting that an increment in force increases the deformation on SP1. The graph also reveals that, with linear progression of the data between the forces and deformation, there is also a linear increment of data among the force, which interprets the variation in the deformation due to the mesh relevance setting.

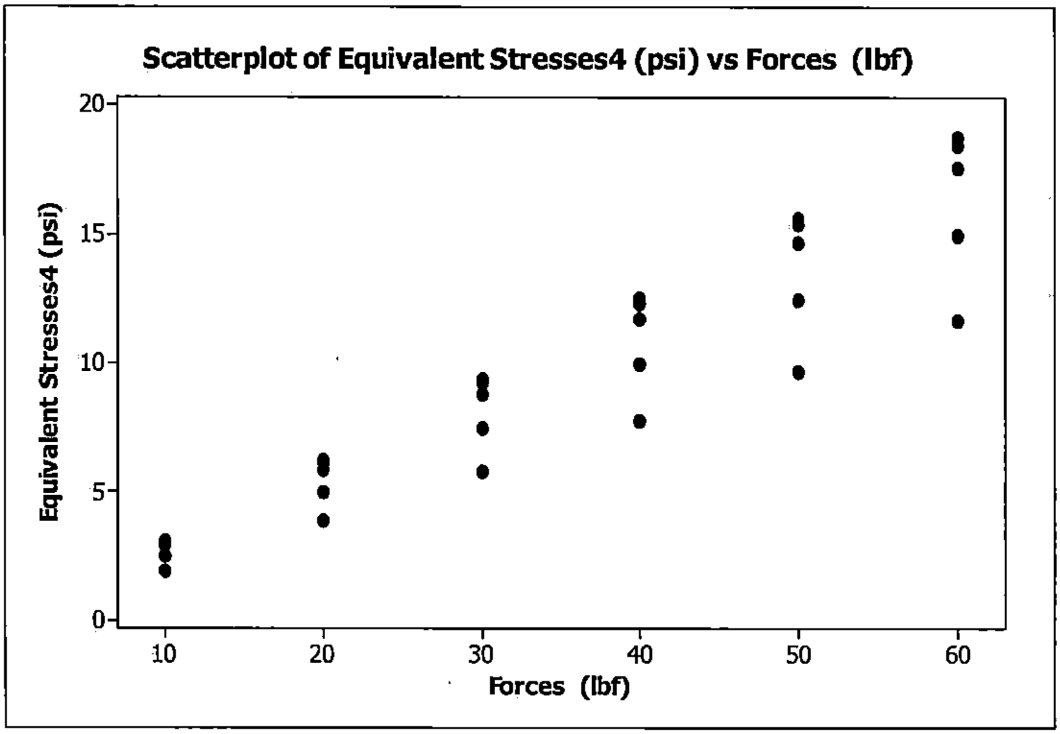

Figure 34. Equivalent Stresses Vs Forces Graph for SP4

The scatter diagram shown in Figure 34 represents that, as the force increases, the equivalent stress on the model also increases. The increment is heteroscedastic, which means there is a non-constant variation in equivalent stress over the values of forces. The graph reveals that, at force 10lbf, the equivalent stress remains low and even the values of stresses for force 10lbf are close, but as the force increases (say 60lbf), the equivalent stress also increases and the values of stresses for 60 lbf are farther than those for 10lbf.

Comparing the results of SP1 and SP4

According to the results, model SP4 shows maximum stiffness as compared to model SP1, as the deformation of SP4 at force 10lbf and mesh -100 is 8.433 nm, compare to the deformation of SP1, which is 10.947 nm. Also model SP1 shows maximum equivalent stress as compared to the equivalent stress of SP4. Hence, from the results, SP4 has maximum resistance against the repeated force as compared to SP1.

Analysis Results of Solid Model SP5

Table 12 (page 57) shows the data for 6 different forces with a combination of 5 relevance mesh settings giving output of 30 readings for equivalent stress and deformation.

Table 12. Structural Results Table for SP5

Forces (lbf)	Mesh	Deformation (nm)	Equivalent Stress (psi)
10	-100	14.3002	3.676
10	-40	14.5288	4.141
10	0	14.5796	4.548
10	40	14.6558	4.955
10	100	14.7574	5.218
20	-100	28.702	7.352
0	-40	28.956	8.282
20	0	29.21	9.096
20	40	29.21	9.909
20	100	29.464	10.44
30	-100	42.926	11.03
30	-40	43.688	12.42
30	0	43.688	13.64
30	40	43.942	14.86
30	100	44.196	15.65
40	-100	57.15	14.7
40	-40	58.166	16.56
40	0	58.42	18.19
40	40	58.674	19.82
40	100	58.928	20.87
50	-100	71.628	18.38
50	-40	72.644	20.7
50	0	72.898	22.74
50	40	73.152	24.77
50	100	73.66	26.09
60	-100	85.852	22.06
60	-40	87.122	24.85
60	0	87.376	27.29
60	40	87.884	29.73
60	100	88.646	31.31

The results are represented in graphical form as shown in Figure 35 (page 58) and Figure 36 (page 59).

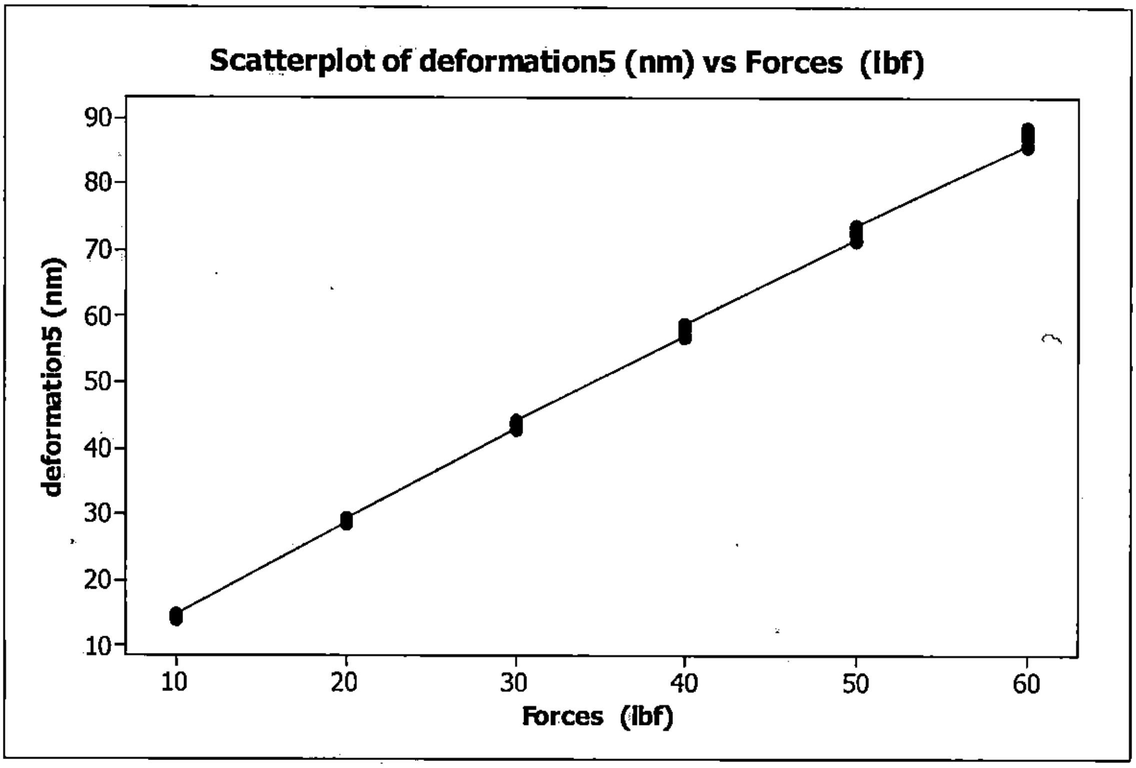

Figure 35. Deformations Vs Forces Graph for SP5

The scatter plot shown in Figure 35 shows the linear progression of data, which reveals that, as force increases, there is an increment in deformation, showing a non-constant variation in deformation over the values of forces.

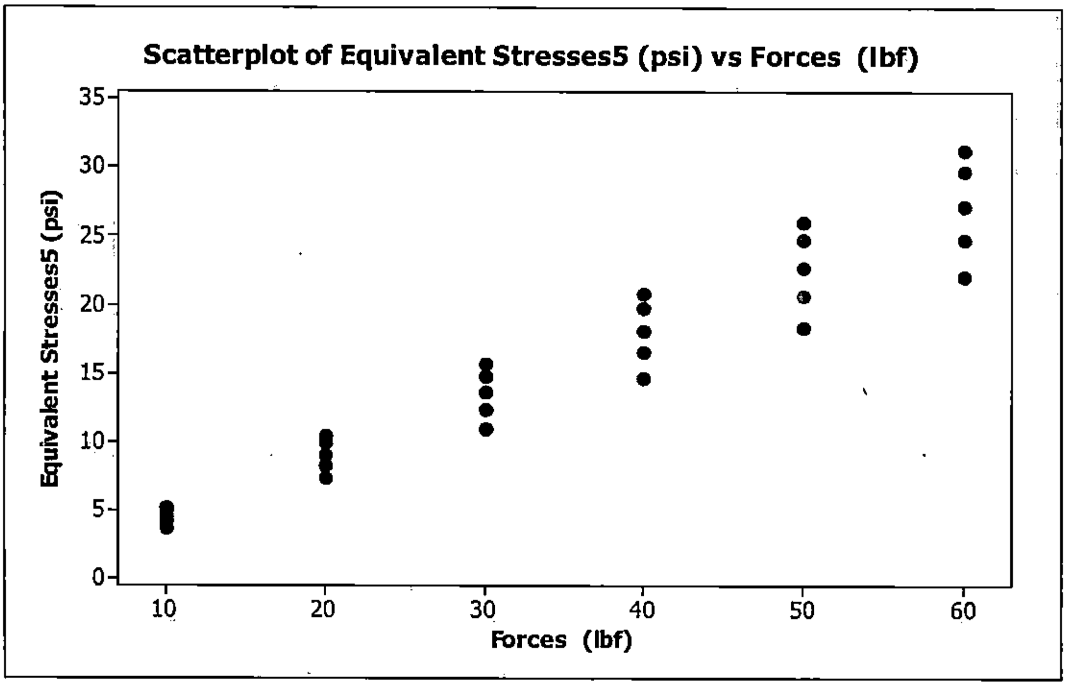

Figure 36. Equivalent Stresses Vs Forces Graph for SP5

The scatter diagram shown in Figure 36 represents that, as the force increases, the equivalent stress on the model also increases. The increment is heteroscedastic, which means there is a non-constant variation in equivalent stress over the values of forces. The graph reveals that, at force 10lbf, the equivalent stress remains low and even the values of stresses for force 10lbf are close, but as the force increases (say 60lbf), the equivalent stress also increases and the values of stresses for 60 lbf are farther than those for 10lbf.

Analysis Results of Solid Model SP6

Table 13 shows the data for 6 different forces with a combination of 5 relevance mesh settings giving an output of 30 readings for equivalent stress and deformation.

Table 13. Structural Result Table for SP6

Forces (lbf)	Mesh	Deformation (nm)	Equivalent Stress (psi)
10	-100	17.196	4.169
10	-40	17.323	4.439
10	0	17.424	7.281
10	40	17.501	7.297
10	100	17.628	6.064
20	-100	34.29	8.338
20	-40	34.544	8.878
20	0	34.798	14.56
20	40	35.052	14.59
20	100	35.306	12.13
30	-100	51.562	12.51
30	-40	51.816	13.32
30	0	52.324	21.84
30	40	52.578	21.89
30	100	52.832	18.19
40	-100	68.834	16.68
40	-40	69.342	17.76
40	0	69.596	29.12
40	40	70.104	29.19
40	100	70.612	24.25
50	-100	85.852	20.85
50	-40	86.614	22.2
50	0	87.122	36.41
50	40	87.63	36.48
50	100	88.138	30.32
60	-100	103.124	25.02
60	-40	103.886	26.63
60	0	104.394	43.69
60	40	105.156	43.78
60	100	105.918	36.38

The results are represented in graphical form as shown in Figure 37 and Figure 38 (page 62).

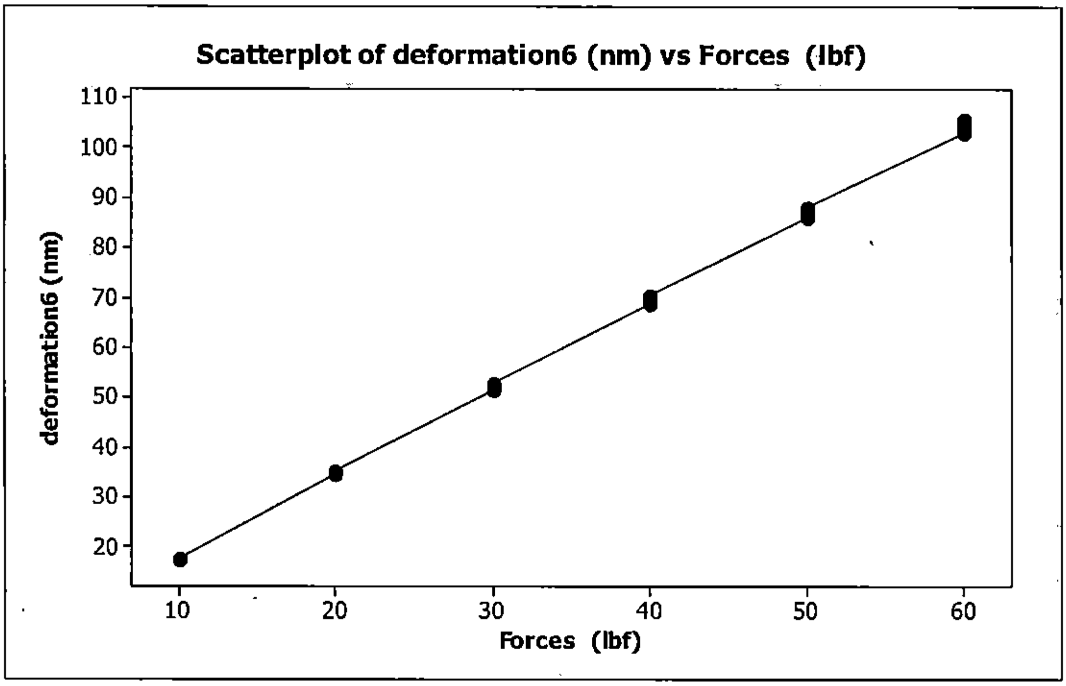

Figure 37. Deformations Vs Forces Graph for SP6

The scatter plot as shown in Figure 37 reveals an association between the forces and deformation. The deformation on the vertical axis is represented as the response variable. The graph shows a strong linear (positive correlation) increment, interpreting that an increment in force increases the deformation on SP1. The graph also reveals that, with linear progression of the data between the forces and deformation, there is also a linear increment of data among the forces, which interprets the variation in the deformation due to the mesh relevance setting.

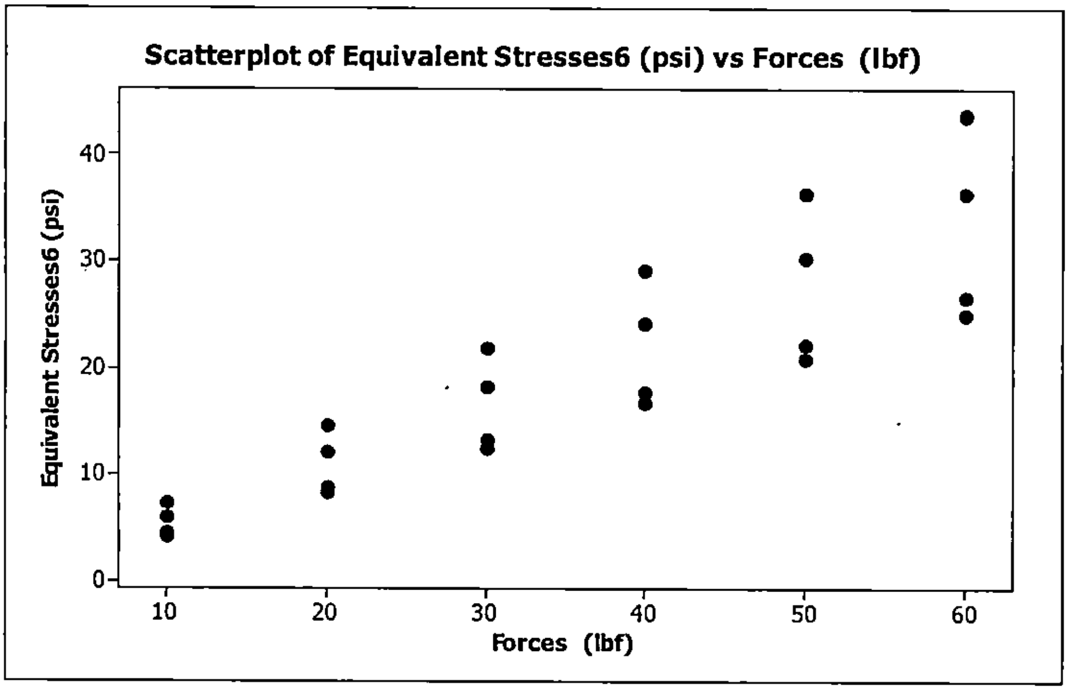

Figure 38. Equivalent Stresses Vs Forces Graph for SP6

The scatter diagram shown in Figure 38 represents that, as the force increases, the equivalent stress on the model also increases. The increment is heteroscedastic, which means there is a non-constant variation in equivalent stress over the values of forces. The graph reveals that, at force 10lbf, the equivalent stress remains low and even the values of stresses for force 10lbf are close, but as the force increases (say 60lbf) the equivalent stress also increases and the values of stresses for 60 lbf are farther than those for 10lbf.

Comparing the results of SP1 and SP5, and SP6

From the result Table 8 (page 42) and Table 13 (page 60) and graphical representation the deformations for model SP5 and SP6 are more than the deformation for model SP1. The equivalent stresses for SP5 and SP6 are less as compared to the equivalent stress of SP1. The comparison of the design model is based on the results obtained from processing of each individual design. These data are collected and organized in tabular form to understand the relationship between the results and identify the optimized or optimal design based on the simulation technique, considering deformation and equivalent stresses.

Mathematical Design:

The design used was a 3 factor factorial model for deformation and equivalent stresses. The following model was used to relate the response variable with the other factors.

$$\text{Response} = \mu + D + F + M + D * F + D * M + F * M + E$$

Where,

D represents Design effects.

F represents Force effects.

M represents Mesh effects.

E represents Error.

μ represents an overall average response deformation (equivalent stresses)

D*F represents an interaction effect between design and force.

D*M represents an interaction effect between design and mesh.

F*M represents an interaction effect between force and mesh.

Note: The D*F*M term was omitted from the model. This was added to the error term.

Analysis of Variance Output for the Model When Deformation is the Response Variable.

Factor	Type	Levels	Values
Design	fixed	6	D1, D2, D3, D4, D5, D6
Forces (lbf)	fixed	6	10, 20, 30, 40, 50, 60
Mesh	fixed	5	-100, -40, 0, 40, 100

Analysis of Variance for Deformations (nm), using Adjusted SS for Tests

Source	DF	Seq SS	Adj SS	Adj MS	F	P
Design	5	26283.3	26283.3	5256.7	669702.97	0.000
Forces (lbf)	5	68617.0	68617.0	13723.4	1748375.93	0.000
Mesh	4	50.7	50.7	12.7	1615.19	0.000
Design*Forces (lbf)	25	6268.4	6268.4	250.7	31943.93	0.000
Design*Mesh	20	2.0	2.0	0.1	12.77	0.000
Forces (lbf)*Mesh	20	12.4	12.4	0.6	78.76	0.000
Error	100	0.8	0.8	0.0		
Total	179	101234.6				

S = 0.0885959 R-Sq = 100.00% R-Sq(adj) = 100.00%

Unusual Observations for Deformations (nm)

Obs	Deformations (nm)	Fit	SE Fit	Residual	St Resid	
1	10.947	11.092	0.059	-0.145	-2.19	R
4	11.125	10.968	0.059	0.157	2.38	R
14	33.528	33.393	0.059	0.135	2.05	R
26	65.532	65.385	0.059	0.147	2.23	R
29	66.802	66.958	0.059	-0.156	-2.36	R
119	52.832	52.688	0.059	0.144	2.18	R
122	14.529	14.684	0.059	-0.155	-2.35	R
127	28.956	29.108	0.059	-0.152	-2.30	R
178	104.394	104.548	0.059	-0.154	-2.33	R

R denotes an observation with a large standardized residual.

Figure 39. Analysis of Variance Output for the Model When Deformation is the Response Variable.

Figure 39 first displays factors (design, forces, and mesh), their types (all fixed), number of levels, and the level values. The second table gives an analysis of variance table. This is followed by a table of coefficients, and then a table of unusual observations.

```
S = 20.75    R-Sq = 25.96%   R-Sq(adj) = 23.84%
                                 Individual 95% CIs For Mean Based on
                                 Pooled StDev
Level   N   Mean  StDev  --+---------+---------+---------+------
D1     30  38.84  19.29          (-----*------)
D2     30  28.67  14.24  (-----*-----)
D3     30  30.27  15.05   (-----*-----)
D4     30  30.38  15.07   (-----*------)
D5     30  50.97  25.30                     (-----*------)
D6     30  60.95  30.27                                (-----*-----)
                         --+---------+---------+---------+------
                          24        36        48        60

Pooled StDev = 20.75


Tukey 95% Simultaneous Confidence Intervals
All Pairwise Comparisons among Levels of Design

Individual confidence level = 99.56%


Design = D1 subtracted from:

Design   Lower  Center  Upper  ---------+---------+---------+---------+
D2      -25.63  -10.17   5.29           (-----*-----)
D3      -24.03   -8.57   6.89           (------*-----)
D4      -23.92   -8.46   7.00           (------*-----)
D5       -3.33   12.13  27.59                    (-----*-----)
D6        6.65   22.11  37.57                        (-----*-----)
                               ---------+---------+---------+---------+
                                      -25         0        25        50


Design = D2 subtracted from:

Design   Lower  Center  Upper  ---------+---------+---------+---------+
D3      -13.85    1.61  17.07               (------*-----)
D4      -13.75    1.71  17.17                (-----*-----)
D5        6.84   22.30  37.76                        (-----*-----)
D6       16.82   32.28  47.74                            (-----*-----)
                               ---------+---------+---------+---------+
                                      -25         0        25        50


Design = D3 subtracted from:

Design   Lower  Center  Upper  ---------+---------+---------+---------+
D4      -15.35    0.11  15.57               (-----*-----)
D5        5.23   20.69  36.15                       (-----*-----)
D6       15.22   30.68  46.14                           (-----*-----)
                               ---------+---------+---------+---------+
                                      -25         0        25        50
```

```
Design = D4 subtracted from:

Design  Lower   Center  Upper   ---------+---------+---------+---------+
D5       5.13    20.59  36.05                     (-----*-----)
D6      15.11    30.57  46.03                          (-----*-----)
                                ---------+---------+---------+---------+
                                       -25         0        25        50

Design = D5 subtracted from:

Design  Lower   Center  Upper   ---------+---------+---------+---------+
D6      -5.48     9.98  25.44                 (-----*-----)
                                ---------+---------+---------+---------+
                                       -25         0        25        50
```

Figure 40. Tukey's Multiple Comparison for Average Deformation for the Six Different Models

Figure 40 shows the P-value is 0.000 for design, indicating that there is sufficient evidence of equality between the means when the level of significance is set at 0.05, which confirms to reject the hypothesis of no difference. The difference between the means is shown in Tukey 95% simultaneous confidence intervals for all Pairwise Comparisons among the six designs.

Tukey's Comparisons

Tukey's test provides 5 sets of multiple comparison confidence intervals.

- Design 1 mean substracted from design 2, 3, 4, 5, and 6 means: The first interval in the first set of the Tukey's output (–25.63, 5.29) gives the confidence interval for the average of design1 subtracted from the average of design 2. For this set of comparisons, none of the means is statistically different because all of the confidence intervals include 0. The means for design 1 and 6 are statistically different because the confidence interval for this combination of means (6.65, 37.57) excludes zero.

- Design 2 mean substracted from design 3, 4, 5, and 6: The means for design 2, 5, and 6 are statistically different because the confidence interval for this combination of means excludes zero.
- Design 3 mean subtracted from design 4, 5, and 6: The mean for design 3, 5, and 6 are statistically different from one another, because the confidence interval for this combination of means excludes zero.
- Design 4 mean subtracted from design 5 and 6: The mean for design 4, 5, and 6 are statistically different from one another, because the confidence interval for this combination of means excludes zero.
- Design 5 mean subtracted from design 6: The mean for design 5 and 6 are not statistically different from each other, because the confidence interval for this combination of means includes zero.

General Conclusion

Figure 40 (page 67) shows that D2 is a better design as compared to other designs. D5 and D6 are the worst designs as compared to D2. Even D1 shows a worse design than D2. Designs D3 and D4 are close to D2, indicating better design.

Analysis of Variance Output for the Model When Equivalent Stress is the Response Variable.

```
Factor          Type   Levels  Values
Design          fixed       6  D1, D2, D3, D4, D5, D6
Forces   (lbf)  fixed       6  10, 20, 30, 40, 50, 60
```

```
Mesh            fixed       5  -100, -40, 0, 40, 100
```

```
Analysis of Variance for Equivalent Stresses (psi), using Adjusted SS for
Tests
```

Source	DF	Seq SS	Adj SS	Adj MS	F	P
Design	5	9177.70	9177.70	1835.54	942.80	0.000
Forces (lbf)	5	10080.72	10080.72	2016.14	1035.57	0.000
Mesh	4	1200.06	1200.06	300.02	154.10	0.000
Design*Forces (lbf)	25	2183.61	2183.61	87.34	44.86	0.000
Design*Mesh	20	818.00	818.00	40.90	21.01	0.000
Forces (lbf)*Mesh	20	286.73	286.73	14.34	7.36	0.000
Error	100	194.69	194.69	1.95		
Total	179	23941.50				

```
S = 1.39531    R-Sq = 99.19%    R-Sq(adj) = 98.54%
```

```
Unusual Observations for Equivalent Stresses (psi)
```

Obs	Equivalent Stresses (psi)	Fit	SE Fit	Residual	St Resid
2	5.7610	2.0137	0.9302	3.7473	3.60 R
3	7.2970	5.1148	0.9302	2.1822	2.10 R
4	9.9620	12.5607	0.9302	-2.5987	-2.50 R
5	9.8790	12.6256	0.9302	-2.7466	-2.64 R
7	11.5200	9.2662	0.9302	2.2538	2.17 R
22	28.8000	31.0543	0.9302	-2.2543	-2.17 R
27	34.5700	38.3207	0.9302	-3.7507	-3.61 R
28	43.7800	45.9723	0.9302	-2.1923	-2.11 R
29	59.7700	57.1516	0.9302	2.6184	2.52 R
30	59.2800	56.5286	0.9302	2.7514	2.65 R
153	7.2810	10.5749	0.9302	-3.2939	-3.17 R
178	43.6900	40.4020	0.9302	3.2880	3.16 R

```
R denotes an observation with a large standardized residual.
```

Figure 41. Analysis of Variance Output for the Model When Equivalent Stress is the Response Variable.

In Figure 41, all p-values are printed as 0.000, meaning that they are less than 0.0005. This indicates significant evidence of effects if the level of significance, a, is greater than 0.0005. The significant interaction effects of deformation with design, forces, and mesh terms imply that the coefficients of second order regression models of the effect of forces and mesh upon deformation depend upon the design. The R-square value shows that the model explains 99.19% of the variance in equivalent stresses, indicating that the model fits the data extremely well for predicting deformation from design, forces, and mesh factors.

Tukey's Multiple Comparison for Average Equivalent Stress for Six Different Models.

```
Source   DF       SS      MS      F      P
Design    5   9177.7  1835.5  21.63  0.000
Error   174  14763.8    84.8
Total   179  23941.5

S = 9.211   R-Sq = 38.33%   R-Sq(adj) = 36.56%


   Individual 95% CIs For Mean Based on
                                 Pooled StDev
Level    N    Mean   StDev  --+---------+---------+---------+-------
D1      30  28.195  15.409                               (---*----)
D2      30   9.351   5.262   (---*----)
D3      30   8.679   4.569  (---*----)
D4      30   9.521   5.047   (----*---)
D5      30  15.776   8.131           (----*---)
D6      30  20.475  11.468                  (---*----)
                            --+---------+---------+---------+-------
                            7.0      14.0      21.0      28.0

Pooled StDev = 9.211

Tukey 95% Simultaneous Confidence Intervals
All Pairwise Comparisons among Levels of Design

Individual confidence level = 99.56%

Design = D1 subtracted from:

Design     Lower   Center    Upper  --+---------+---------+---------+-------
D2       -25.706  -18.845  -11.983   (----*-----)
D3       -26.378  -19.516  -12.655  (-----*----)
D4       -25.536  -18.675  -11.813   (----*-----)
D5       -19.281  -12.419   -5.558        (-----*----)
D6       -14.582   -7.720   -0.858            (-----*----)
                                    --+---------+---------+---------+-------
                                    -24       -12         0        12

Design = D2 subtracted from:

Design    Lower  Center   Upper  --+---------+---------+---------+-------
D3       -7.533  -0.672   6.190                (----*-----)
D4       -6.692   0.170   7.032                (-----*-----)
D5       -0.436   6.425  13.287                      (----*-----)
```

```
D6         4.263  11.125  17.986                                        (----*-----)
                                     --+---------+---------+---------+-------
                                     -24       -12         0        12
Design = D3 subtracted from:

Design    Lower  Center   Upper     --+---------+---------+---------+-------
D4       -6.020   0.842   7.703                      (-----*----)
D5        0.235   7.097  13.959                           (-----*-----)
D6        4.935  11.796  18.658                               (-----*-----)
                                     --+---------+---------+---------+-------
                                     -24       -12         0        12

Design = D4 subtracted from:

Design    Lower  Center   Upper     --+---------+---------+---------+-------
D5       -0.606   6.255  13.117                          (-----*-----)
D6        4.093  10.954  17.816                              (-----*-----)
                                     --+---------+---------+---------+-------
                                     -24       -12         0        12

Design = D5 subtracted from:

Design    Lower  Center   Upper     --+---------+---------+---------+-------
D6       -2.162   4.699  11.561                        (-----*-----)
                                     --+---------+---------+---------+-------
                                     -24       -12         0        12
```

Figure 42. Tukey's Multiple Comparison for Average Equivalent Stress for Six Different Models

Result Interpretation

Figure 42 shows the P-value is 0.000 for design and indicates that there is sufficient evidence of no equality between the means when alpha is set at 0.05, which confirms to reject the hypothesis of no difference. The difference between the means is shown in Tukey 95% simultaneous confidence intervals all Pairwise Comparisons among Levels of Design.

Tukey's Comparisons

Tukey's test provides 5 sets of multiple comparison confidence intervals.

- Design 1 mean substracted from design 2, 3, 4, 5, and 6 means: For this set of comparisons, none of the means is statistically different because all of the confidence intervals include 0.
- Design 2 mean substracted from design 3, 4, 5, and 6: The means for design 2 and 6 are statistically different because the confidence interval for this combination of means excludes zero.
- Design 3 mean substracted from design 4, 5, and 6: The mean for design 3, 5, and 6 are statistically different from one another, because the confidence interval for this combination of means excludes zero.
- Design 4 mean substracted from design 5 and 6: The mean for design 4 and 6 are statistically different from one another, because the confidence interval for this combination of means excludes zero.
- Design 5 mean substracted from design 6: The mean for design 5 and 6 are not statistically different from each other, because the confidence interval for this combination of means includes zero (Joiner, R).

General Conclusion

Figure 42 (page 71) shows that D3 is the most efficient design as compared to other designs. Figure 42 (page 71) also shows that D1 is the worst design, when compared with D3. D5 and D6 also show worse design.

Conclusion

It has been shown by the simulation using CAD modeling and finite element analysis integration and confirmed by the results, a time consuming design formation and optimization of the designed model were possible. The prediction of deformation and stress was the major factor in designing an optimized model. Without using CAD/FEA for design modification and one-click transformation to stress analysis environment to predict the deformation and stress, the designers will have to sacrifice their time and money in testing physical models. To be able to obtain the final design and meet the strength requirements, 6 design models were created and analyzed for 6 different forces and 5 different mesh settings obtaining 30 results for deformation and equivalent stresses each. Without CAD/FEA it would have taken months to design these models, build each for the physical model, and test for the analysis.

The analysis results shown by the result table and the general conclusion from the analysis of variance of deformation and equivalent stresses entails that model SP3 shows relatively optimized design as compared to the results of other design models. The results also show that the model does not necessarily have to be heavy to sustain force, but design plays an important role in sustaining force to reduce deformation. Therefore, an optimal design should incorporate a combination of good control over the geometry.

Suggestions and Recommendations

There are several other methods to optimize the design.

1. The next step for the further research is considering the material selection factor. Material selection also affects the cost and appearance of the product. Best material selection could result in reduced deformation and stresses for the same forces.
2. Vibration effect consideration is another factor to be researched to manufacture a product.
3. Temperature affects when the vise is holding a part under the CNC machine for longer machining operations.

Limitations

The prediction in this kind of CAD/FEA integration is limited to uniformly distributed load only. Also the nodes and element settings are selected automatically for the set relevance mesh setting, which makes it difficult to obtain precise readings from the congested features.

Future Development on CAD/FEA Integration

1. The designer should be able to customize the elements and nodes selection to configure the precise stress and deformation in the part.
2. The CAD/FEA integration should have the ability to choose the nodes and configure the results on each node selected.

Reference

Anderson, D. M. (2004). Design for Manufacturability and Concurrent Engineering. California: CIM Press.

Bahr, D. F., Moody N. R. ((2005). Development of Experimental Verification Techniques for Non-linear Deformation and Fracture on the Nanometer Scale. California: Sandia National Laboratories.

Benhabib, Beno (2003). Manufacturing: Design, Production, Automation, and Integration. Marcel Dekker Inc.

Black, J. T., DeGarmo, Kohser, E. P., Ronald A. (1997). Materials and Processes in Manufacturing. New Jersey: Prentice Hall.

Burkhardt, M. R., DeRusseau, S. N., Werner, S. L. U.S. Geological Survey. Open-File Report 96-216. (http://ca.water.usgs.gov/pnsp/rep/ofr96216.pdf).

Groover, P. M., Zimmers, W. E., (1989). Computer Aided Design and Computer Aided Manufacturing. New Delhi: Prentice Hall of India.

Hughes, J. R. (2000). The Finite Element Method *Linear Static and Dynamic Finite Element Analysis*. New York: Dover Publications, Inc.

Hryniewiecki, J. (2005). Stress Analysis in Automotive Brake Vacuum Booster. Indiana: University of Notre Dame.

Joiner, R. (1994). *Minitab Handbook*, Third Edition. California: International Thomson Publishing.

Lin, F., Sun, W. (2001). Computer Modeling and FEA Simulation for Composite Single Fiber Pull-out. Journal of Thermoplastic Composite Materials, Vol. 14 (4), pp.327-343.

Lund, E. (1994). Finite Element Based Design Sensitivity Analysis and Optimization, Report no. 23. Denmark; Aalborg University.

Madsen, P. D. (2003). Autodesk Inventor 5/5.3 *Basics Through Advanced.* New Jersey: Prentice Hall.

Moaveni, S. (1999). Finite Element Analysis *Theory and Application with ANSYS.* New Jersey: Prentice Hall.

Rao, S. S. (1999). The Finite Element Method in Engineering. Boston: Butterworth Heinemann.

Rutherford, A. (2001). Introducing ANOVA and ANCOVA a GLM Approach. Great Britain: Sage Publications.