2017

# GroovePin performance of threaded inserts

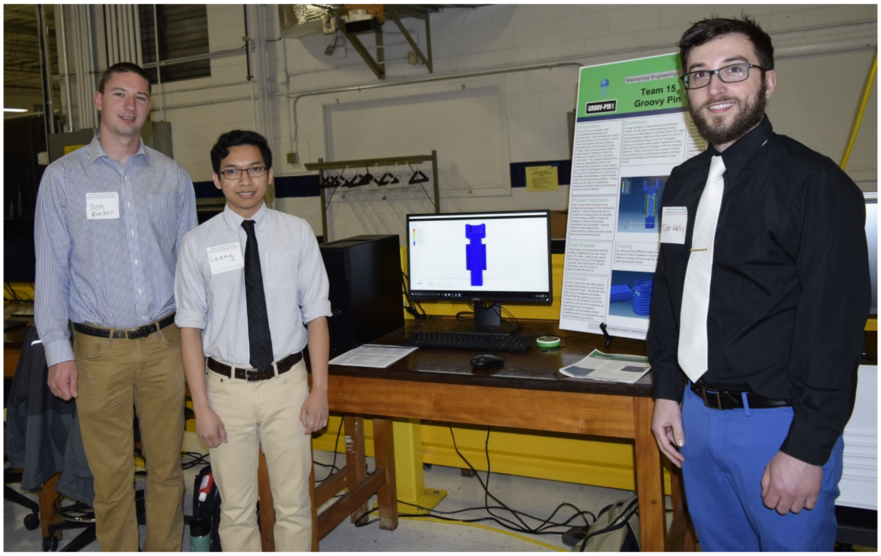
Laeng Khoun
*University of Rhode Island*

Josh Blacker
*University of Rhode Island*

Tyler Kelly
*University of Rhode Island*

## Recommended Citation

Khoun, Laeng; Blacker, Josh; and Kelly, Tyler, "GroovePin performance of threaded inserts" (2017). *Mechanical Engineering Capstone Design Projects.* Paper 16.
https://digitalcommons.uri.edu/mechanical-engineering-capstones/16



# Final Design Report
# Team 15 : Groovy-Pin

Team Members:
Laeng Khoun - Project Manager
Josh Blacker - Research Engineer
Tyler Kelly - CAD/FEA Engineer

May 8, 2017

Sponsor: Groov-Pin Corporation

Faculty Advisor: David Taggart

Professor: Bahram Nassersharif

Department of Mechanical, Industrial and Systems Engineering
University of Rhode Island

# Abstract

Groov-Pin corporation presented team Groovy-Pin with the capstone design project at the beginning of the fall semester. The capstone design project with Groov-Pin was primarily focused on the performance of the threaded inserts that the company manufactures. The company presented a plan of action to analyze the forces, torque and deformation caused through thread forming and thread failure using finite element analysis in Abaqus. The project goal was to create a model in Abaqus that would test the specific insert that Groov-Pin would like to analyze by creating a force or a torque on the fastener.

Over the course of this year team Groovy-Pin has performed successful tests in Abaqus that replicate the physical tests that Groov-Pin performs on their inserts. Last semester, team Groovy-Pin was successfully able to come up with a proof of concept and run a pull-out test in Abaqus. With Groov-Pin and other resources helping along the way, the team was able to get a better understanding of the Abaqus software and examine the results of the pull-out test. The pull-out test showed that the threads of the base material sheared before the bolt did. It also showed the threaded area took on the highest amounts of stress.

This semester Team Groovy-Pin was able to create a more successful pull-out test and a deformation model. The pull out test was refined so that the base material did not shear fully and instead the bolt was pulled out by only one thread pitch. In the deformation simulation, Groovy-Pin also was able to dent an insert inward successfully and show the deformation that takes place. The dented insert is used by Groov-Pin to increase prevailing torque by increasing contact between the threads of the bolt and internal threads of the insert. Groovy-Pin attempted to create a prevailing torque simulation by screwing in a bolt into the dented model but was not able to achieve acceptable results due to difficulty with modeling the complex interactions.

# Contents

# Nomenclature

| | |
|---|---|
| $l$ | length (in) |
| $P$ | Pitch |
| $D_{maj}$ | Major diameter (in) |
| $D_{min}$ | Minor diameter (in) |

# Acronyms

| | |
|---|---|
| $TPI$ | Threads Per Inch |
| $CAD$ | Computer Aided Design |
| $FEA$ | Finite Element Analysis |
| $MIL$ | Military Standard |
| $ID$ | Inner Diameter |
| $OD$ | Outer Diameter |
| $SS$ | Stainless Steel |
| $QFD$ | Quality Function Deployment |

# List of Tables

# List of Figures

# 1 Introduction

Groov-Pin is a leading company that is involved with the manufacture and design of threaded inserts. Founded in 1926, Groov-Pin set out to develop the very first press-fit fastener of its kind: a Grooved Pin. After many years of exploration in the grooved pin industry, Groov-Pin began development of the Tap-Lok threaded insert. Designed to be user friendly, these threaded inserts are an invaluable tool in many mechanical engineering applications. All of the threaded inserts developed by Groov-Pin are made from metal bars on screw machines in their Georgia facility. They provide an immense amount of strength and permanent threads in softer metals such as aluminum and magnesium. They allow the design engineers to utilize the maximum amount of strength available which is pivotal to military applications.

Threaded inserts are fasteners that permit a much stronger bond in softer material. They are helical in nature with a thread pattern on the exterior and interior. Some threaded inserts are self-tapping while others fit into a pre-tapped hole. Both of these types of threaded inserts will be examined in our digital experiments.

Groov-Pin presented Team Groovy-Pin with an engineering problem involving their entire line of threaded inserts. With the popularity of threaded inserts rapidly rising, Groov-Pin was interested in developing modular FEA simulations for all of their threaded inserts to primarily solve the following three scenarios: pull-out resistance, prevailing torque, and deformation of the threaded insert. The purpose of these FEA experiments are to enable engineers from various industries across the globe to utilize an FEA program such as Abaqus to determine what role the aforementioned scenarios play in any given application. This will allow the design engineer to better predict how all of these variables may ultimately lead to failure; which is something to be avoided whenever possible. The following brief descriptions of the three scenarios Team Groovy-Pin have been and will continue to investigate will shed some light on the significance of this project.

Pull-out resistance is where the threads between the threaded insert and the bolt interact with each other via a pull-out force. These thread patterns are to resist these pull-out forces in order to create a stronger mate in the assembly. It is always ideal that the bolt breaks before the threaded insert and this test was explored for our proof of concept. After some alterations to the original design, primarily the threaded inserts boundary conditions, the team was able to identify this prediction and see a fracture of the bolt. The pull-out test was also concerned with the external threads of the insert and the base material. The base material that the insert sits in is a softer material then the insert and will also fail before the insert does. Groov-Pin was interested in examining the stress areas and deformations on the threads of the base material as the insert is being pulled out.

Prevailing torque is a concept that involves the torquing of the screw into the threaded insert. In essence, the prevailing torque is the amount of torque necessary to fasten the screw into the threaded insert without any forces that will loosen them such as vibrations. This concept seems trivial but is actually quite complex. Deformation of the threaded insert is where the threaded insert is squeezed out of round prior to installation and post manufacture. Generally, the threaded insert is placed around a mandrel and the threaded insert is then squeezed out of round with two punching tools from either side until the internal threads come in contact with the mandrill or pin on the interior of the threaded insert. This is done to provide more resistance and friction once the bolt is fastened into the threaded insert. However, there is currently no standard for how much the threaded insert should be deformed which is why further investigation is required.

Now that these concepts have been explained, the remainder of the report will explore the work that Team Groovy-Pin has completed and their findings from the results of the Abaqus models.

# 2 Project Planning

The planning for this project was divided into two levels: a minor level and a major level. At the minor level, daily tasks were scheduled and completed which kept the team organized on day to day operations and ensured deadlines were met. The major level encompassed the larger overall scope of the project which allowed the team to manage the project at large starting with defining the problem, progressing through development stages, and finally ending with the goal of publishing the team's findings. In the major level, planning for this project was divided even further and generally consisted of following design process flow charts similar to those presented in class lectures.

## Minor Level Planning

## 2.1 Microsoft Project and Gantt Chart

The focus of minor level planning was to keep track of day to day operations and and meeting deadlines. The team used Microsoft Project in order to keep track of all work contributed to the project as well as the hours associated with the work. Within the Microsoft Project document, weekly events are scheduled and kept track of which includes weekly conference calls, Abaqus exercises, and meetings. The document is used to record other non-weekly meetings such as consultation with advisors and sponsor representatives. The document allows team members to keep track of their personal contributions to the project. A copy of the Microsoft Project document can be seen represented by a Gantt Chart shown in Figures 107 and 108 shown in the Appendix. The Gantt Chart shows the timeline, tasks, milestones, and resources which has been an invaluable component of managing the project. In addition to the full Gantt Charts in the appendix, a short preview of the Gantt chart can be seen in figure 1 below.

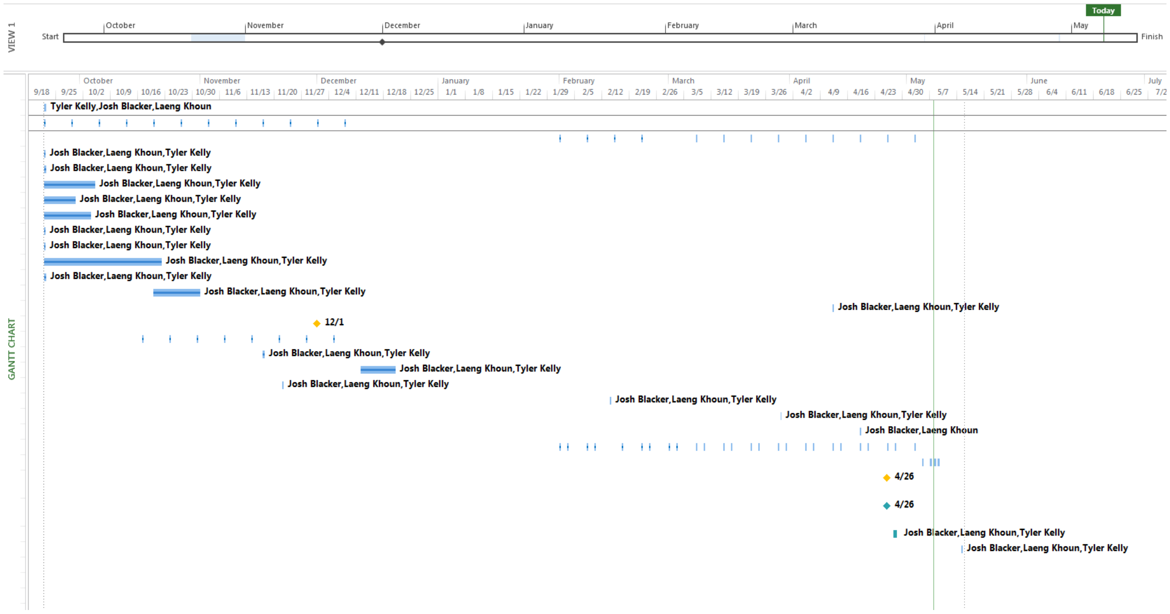


Figure 1: A preview of the Gantt Chart. Full versions can be seen in the appendix.

## 2.2 Team Member Roles and Titles

Each member of Team Groovy-Pin was given a role and job title which exemplifies their greatest strengths and major contributions to the team. The given roles and titles can be seen in Table 1 below. Although titles were given, each member was expected to contribute equally to the project. Team members also did work outside of their titles when necessary. Most responsibilities were shared in order to ensure quick turnaround of results and progress. Sharing responsibilities beyond each member's roles and titles proved to be effective in that daily operations and goals/deadlines were met. The team however, also found it was appropriate to assign titles in order to give recognition to each member's major contributions and strengths.

Table 1: Team Member Job Titles

| Team Member | Job Title |
|---|---|
| Laeng Khoun | Project Manager |
| Josh Blacker | Research Engineer |
| Tyler Kelly | CAD/FEA Engineer |

## Major Level Planning

## 2.3 Define

In the "Define" stage of the project, the team's goal was to understand and define the problem presented by Groov-Pin. The team reviewed the presentation by delivered by the sponsor. After reviewing the presentation, the team scheduled a meeting with the sponsor at Groov-Pin's facility in Smithfield, Rhode Island. At the meeting, the team members met with Groov-Pin representatives Scot A. Jones, Chief Executive Officer, and Jonathan Dupre, Junior Engineer. At the meeting, Scot and Jonathan defined the problem in further detail and described their intentions for the project and their overall goals. After meeting the meeting at the Groov-Pin facility, the team members better understood the problem and entered the research stage of the project.

## 2.4 Research

In the "Research" stage of the project, the team reviewed the given problem definitions and conducted research relevant to the project goals. This included researching patents relevant to the project. Each team member conducted a patent search exercise using the United States Patent and Trademark Office's online database. A number of patents were identified that were relevant to the problem and were evaluated for their usefulness to the project.

The team was also granted access to a Dropbox folder with a collection of resources including journal articles and documents that proved to be helpful when conducting research. Although the team has mainly moved past the research phase of the project, the team has often returned doing research in order to investigate finite element analysis. The primary software used for the project, Abaqus, is a very complex and powerful software which warrants constant research and self-teaching in order to take advantage of it's many advanced and intricate features.

## 2.5 Brainstorm

The "Brainstorm" stage of the project consisted of each team member generating 30 different design concepts for a total of 90 design concepts. Although splitting up different aspects of the problem to avoid concept overlap was considered, the team ultimately decided against setting constraints on ideas to allow for total creative freedom in generating designs. After generating concepts, a conference call with Groov-Pin helped the team narrow the number of concepts down to three unique designs: a pullout resistance simulation, a deformation of insert simulation, and a prevailing torque simulation.

## 2.6 Create

The Create stage of the project consisted of the process of creating the simulations. The group chose to start with the the pullout resistance simulation as it was believed to be the simplest to model and a good starting point for learning Abaqus. Several iterations of the pullout resistance simulation were produced and the team expects there will be more in the future. The Create stage will be revisited throughout the project as the team revises designs and starts creating the deformation of insert simulation and prevailing torque simulation.

## 2.7 Collaborate

The Collaborate stage of the project includes sharing current progress and results with our faculty advisor, Dr. David Taggart and our sponsor, Groov-Pin. The team has weekly conference calls with Groov-Pin in order discuss our progress and to get their guidance and feedback. Using their feedback, we make revisions to our models and revisit the Create stage in order to implement changes. In addition to consultation with Dr. Taggart and the Groov-Pin representatives, the team also receives feedback from Dr. Bahram Nassersharif. The Collaboration stage will be revisited throughout the project as the team makes further progress.

## 2.8 Experiment

The Experiment stage of the project will consist of refining models to get more accurate results as well as verifying the results against physical tests. The team will take the completed simulations and refine meshes in order to get more accurate results. The team will also compare those results to relevant research and known theory. Towards the end of the project the team may find it necessary to perform physical tests of pullout resistance and prevailing torque to compare to the digital experiments. The physical tests may also be out of range of the current scope of the project as creating digital experiments is the main goal.

## 2.9 Evaluate

The Evaluate stage of the project will take place towards the end of the project and will incorporate reviewing the finished simulations and findings with Groov-Pin. The simulations and results will be analyzed for accuracy and completeness. By this stage of the project, the simulations for pullout resistance, deformation of insert, and prevailing torque will all have gone through the cycle between the Create stage through to the Experiment stage so that multiple iterations of the simulations have been made to refine and perfect the simulations. The team will work with Groov-Pin to ensure the results of the project are acceptable and worthy of progressing with to the Publish stage of the project.

## 2.10 Publish

The Publish stage of the project will incorporate using the simulations and results found in order to publish an article in a journal within the engineered fasteners industry. Simulations of the same scale and scope of this project are unprecedented and both Groov-Pin and Team Groovy-Pin expect the results of the project to be worthy of publishing.

# 3 Financial Analysis

## 3.1 Sources of Funding

This project is sponsored and funded by Groov-Pin Corporation. Due to the nature of the project, costs associated with the project are expected to be little to none. No specific budget was given but any costs associated with the project are expected to be kept at a minimum. At this stage of the project, expected costs are likely to be limited to small expenses such as journal article access costs and purchases of bolts and inserts as test specimen.

## 3.2 Man Hour Analysis

Due to the nature of the project and the low costs involved, a hypothetical man hour analysis was done in order to determine the cost of labor for the project. The main component of costs for this project would be the labor component. In a non-academic scenario in which a company employs entry level engineers, it is necessary to account for costs based on hours spent contributing to the project. The team agreed that it was appropriate to assign pay rates based on the United States national average of approximately \$20/hr for entry-level engineers. Using hours recorded on the Microsoft Project software, as of May 8th, 2017 a total of 238 hours were recorded among all three team members. These hours include all time spent meeting, doing research, working on the model, debugging, and other miscellaneous time contributing to the project. Total costs for the project were calculated to be \$4,760 as shown in Table 2.

Table 2: Total Cost of Labor (As of 05/08/2017)

| | |
|---|---|
| Pay Rate | \$20/hr |
| Total Hours | 238 hr |
| Total Cost of Labor | \$4,760 |

## 3.3 Cost of Software

Creating simulations for this project mainly involved the use of 3D CAD design software Solidworks and Finite Element Analysis software Abaqus. Both software packages are licensed products offered by the European multinational software company Dassault Systmes. Through the University of Rhode Island, Solidworks and Abaqus are provided to students for free at point of use so Team Groovy-Pin did not have to pay for licenses to use the software.

Although Solidworks was provided to the team through the university, the team researched the cost of Solidworks to see what additional cost would be added if it was not free. A standard license to use Solidworks has a list price of $3,995 according to TeDesco [1]. A subscription service that covers upgrades and technical support is also offered at a list price of $1,295 for a year of service. For this cost analysis, an assumption will be made that the project will run for a single full year.

# 4 Patent Searches

**Patent Number:** 8,037,772

**Patent Name:** Thread Forming Fasteners for Ultrasonic Load Measurement and Control

**Patent Date:** October 18, 2011

**Company Owning Patent:** Innovation Plus, LLC

**Patent Description:** This patent demonstrates a method of determining load between critical joints in applications such as automotive engines and aerospace. It uses an ultrasonic load measurement transducer with the thread forming fastener to determine the load. Steps can then be taken to accurately measure and control the load in the thread-forming fastener during tightening, and to inspect the load in the thread-forming fastener after assembly. This provides insight into how it may be measured in use.

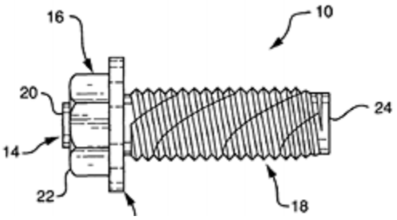


Figure 2: A picture of patent No. 8,037,772 owned by Innovation Plus, LLC.

**Patent Number:** 8,449,235

**Patent Name:** Method for producing a threaded insert with an internal and external thread, and threaded insert

**Patent Date** May 28, 2013

**Company Owning Patent:** Ludwig Hettich Co.

**Patent Description:** This patent owned by Ludwig Hettich Co. shows a method used to produce a threaded insert with an internal and external thread that have differing or same thread pitches where the threaded insert is created from a profile strip. One side of the profile strip is pre-shaped with the profile of the external thread and the other side is pre-shaped with the profile of the internal thread. This patent identifies thread pitch and spacing as crucial factors to performance. This will help in creating models and testing different cases of threads to see which perform the best in simulations. An image associated with the patent can be seen in Figure 3.

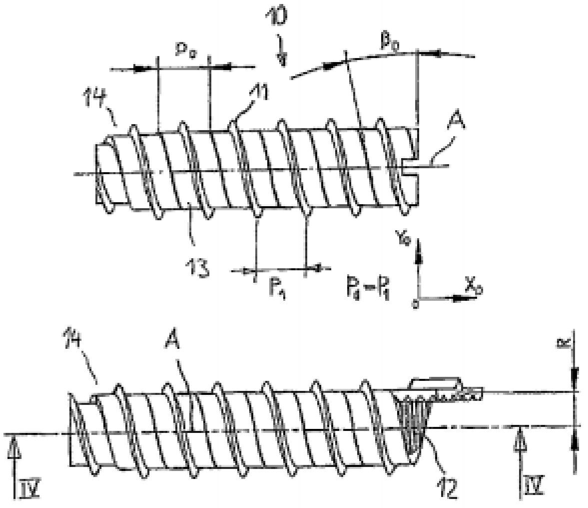


Figure 3: A picture of patent No. 8,449,235 owned by Ludwig Hettich Co.

**Patent Number:** 4,106,540

**Patent Name:** Anti-vibration thread insert

**Patent Date:** August 15, 1978

**Company Owning Patent:** Microdot, Inc.

**Patent Description:** This patent shows a locking thread insert that is designed to resist vibrations. It is wound from wire to create a diamond-shaped cross-section. The wound wire is coiled to a predetermined diameter preferably slightly larger than the diameter of a threaded hole in a female member in which the insert is to be disposed. The threaded screw is free-running in the insert until a predetermined magnitude of loading is encountered at which time a locking and anti-vibration interaction occurs between the crest of the male thread on the screw and the insert. The crest of the male thread is radially compressed when moved axially so as to positively resist loosening under vibration. The interaction between the insert and screw also positively seats and locks the insert in the female member.

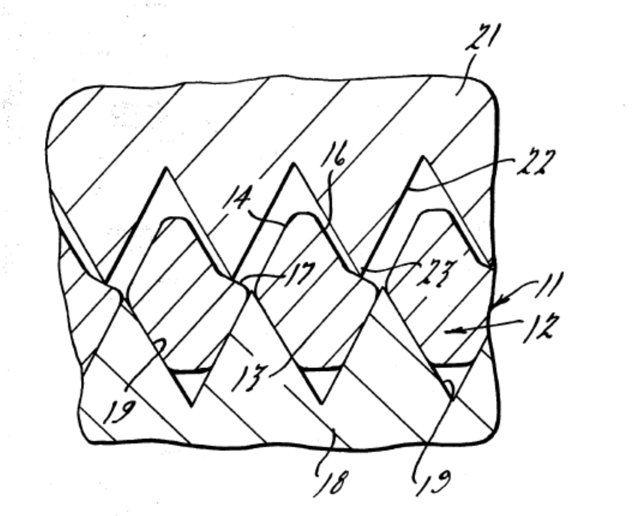


Figure 4: A picture of patent No. 4,106,540 owned by Microdot, Inc.

# 5 Evaluation of Competition

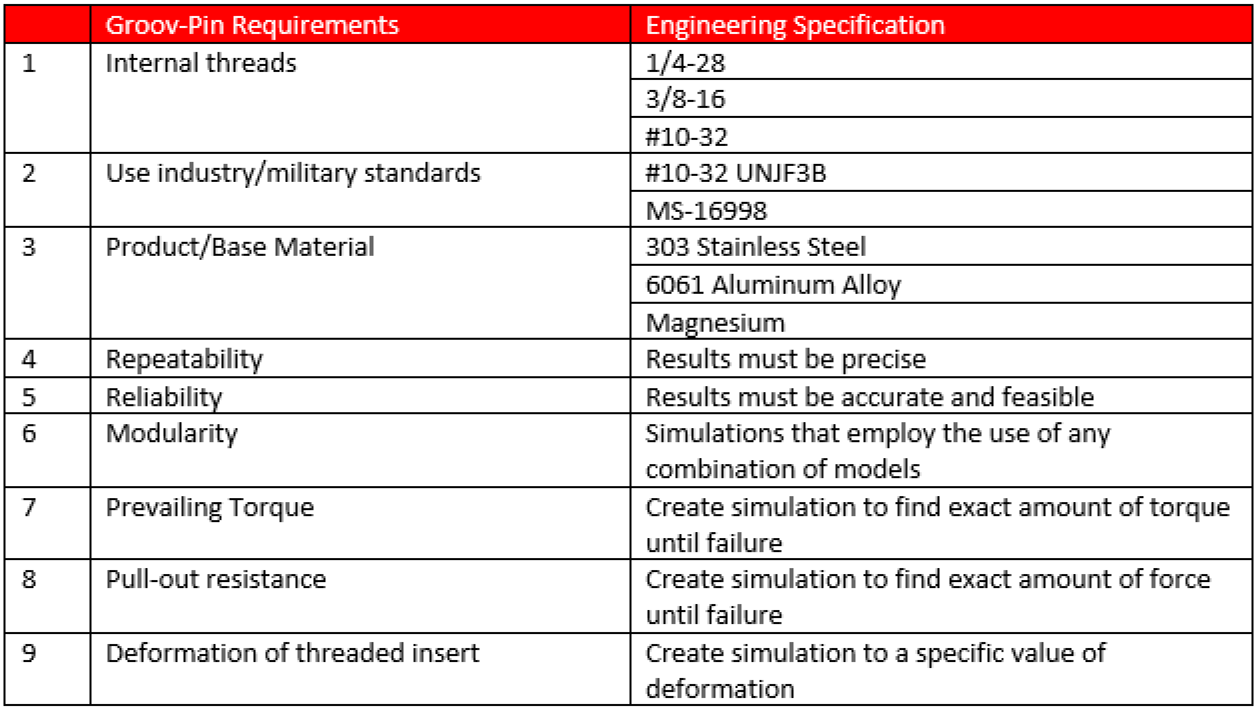
Designing digital experiments for engineered fasteners is unprecedented and pushes boundaries within the engineered fastener industry. Team-Groovy Pin was not able to identify competitors that aim to produce models and simulations within the same scope and scale as planned for this project. The results of this project will likely provide useful information on aspects of threaded insert performance that have not been produced before. The team decided that instead of evaluating outside competition, evaluating competition between different iterations of the team's simulations would be more appropriate.

In the earliest stages of the project, the team planned to start with 2D FEA simulations in order to ease into learning FEA methods and modeling techniques. Despite this reasoning, the team decided that starting with 3D models would be more practical and time efficient. By evaluating the potential benefits from creating a 2D model versus a 3D model, the team concluded that forgoing the 2D model would be the best approach to the project. The team reasoned that the 3D model would more accurately capture the intricacies of interactions between a bolt and threaded insert in a pullout resistance test. The 2D model would not incorporate hoop stresses, torque, or rotation associated with pulling a the threads of the bolt against the internal threads of an insert. Thus, it was decided that spending the team's vital and limited time pursuing a 2D model was not practical. Although the 3D model presented a greater learning curve, progress made in the creation of the 3D model proved to meet the goals of the project faster than if the team started with 2D models.

The team initially started the first 3D model with the design approach of prioritizing accuracy over simulation run times. This proved to be unpractical because the earliest iterations of the 3D model had bugs and issues that caused crashes in running the simulation. This led to situations in which simulations were submitted for processing, 10 to 15 minutes would pass, and the simulation would ultimately crash with an error. Errors could stem from a number of different reasons such as bad interactions, distorted elements, or overconstraining / overloading parts. This also meant that time was being wasted running crashing simulations. The team consulted with Groov-Pin and were advised to simplify the 3D model by removing exterior threads which cut down on number of nodes to run and in turn, time to run the simulation. The mesh sizing was also made more coarse in order to cut down on the number of nodes being run in the simulation. This ultimately allowed the team to run more tests on the simulations which allowed for testing through trial and error. The team was able to produce a working model through this process and the simulation presented working results, proving that the 3D simplified model was the top contender against it's competition.

# 6 Specifications Definition

## 6.1 Groov-Pin Specifications



| | Groov-Pin Requirements | Engineering Specification |
|---|---|---|
| 1 | Internal threads | 1/4-28 |
| | | 3/8-16 |
| | | #10-32 |
| 2 | Use industry/military standards | #10-32 UNJF3B |
| | | MS-16998 |
| 3 | Product/Base Material | 303 Stainless Steel |
| | | 6061 Aluminum Alloy |
| | | Magnesium |
| 4 | Repeatability | Results must be precise |
| 5 | Reliability | Results must be accurate and feasible |
| 6 | Modularity | Simulations that employ the use of any combination of models |
| 7 | Prevailing Torque | Create simulation to find exact amount of torque until failure |
| 8 | Pull-out resistance | Create simulation to find exact amount of force until failure |
| 9 | Deformation of threaded insert | Create simulation to a specific value of deformation |

Table 3: Groov-Pin Engineering Specifications

The following specifications were given to the team by Groov-Pin to use in creating the model.

1. Groov-Pin specified dimensions for internal threads to be used in the model.

2. Groov-Pin specified the specific military standards to be used in the model.

3. Groov-Pin stated the bolt in the model would be 303 stainless steel and the insert would be aluminum or magnesium.

4. The model must be repeatable in order to ensure results aren't simply a one time occurance. Repeatability allows others to verify accuracy of the results.

5. The models must be reliable in that the results are accurate.

6. The models must be modular so that different inserts can be swapped in and out for

testing.

7. A simulation must be created to model prevailing torque.

8. A simulation must be created to model pull-out resistance.

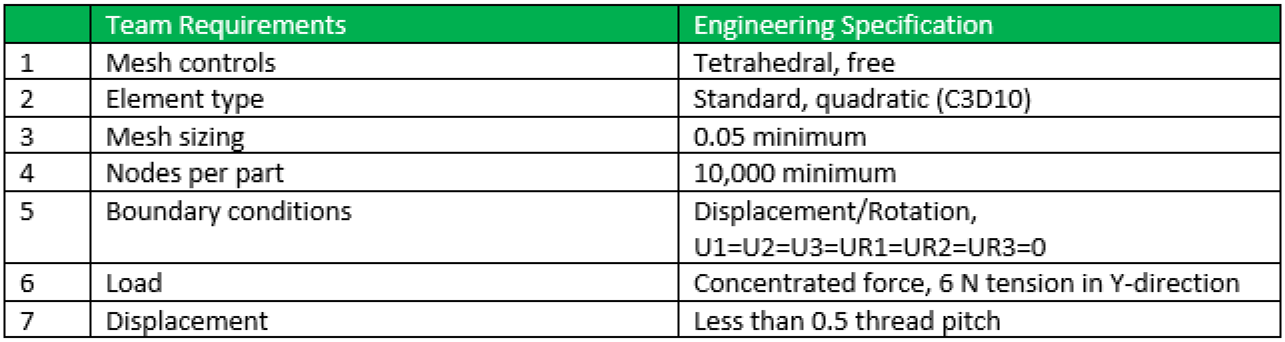
9. A simulation must be created to model the deformation of a threaded insert.

10. Mesh sizing should be 0.05 minimum (mesh size is unitless.)

11. Nodes per part should be 10,000 minimum. This will allow for the most accurate results with minimum processing time.

## 6.2 Team Groovy-Pin Specifications

The team decided that it was also appropriate to define additional design specifications not given by the sponsor in order to set standards for modeling within the group. The team's design specifications can be seen in Table 4 below.



| | Team Requirements | Engineering Specification |
|---|---|---|
| 1 | Mesh controls | Tetrahedral, free |
| 2 | Element type | Standard, quadratic (C3D10) |
| 3 | Mesh sizing | 0.05 minimum |
| 4 | Nodes per part | 10,000 minimum |
| 5 | Boundary conditions | Displacement/Rotation, U1=U2=U3=UR1=UR2=UR3=0 |
| 6 | Load | Concentrated force, 6 N tension in Y-direction |
| 7 | Displacement | Less than 0.5 thread pitch |

Table 4: Team Engineering Specifications

# 7 Conceptual Design

## 7.1 Laeng Khoun's Concepts

### Static 2D Models

These models represent what happens to a threaded insert in different conditions and are fairly easy to model. Although they are a good place to start, they do not show what happens to a 3D insert and are not dynamic. These concepts will most likely be used as the first step in progressing to more advanced models and represent a good starting point for the process of creating an ideal model of an experiment.

**1. Simple Tensile Test with Force Directly Applied to a Threaded Fastener:** This concept involves the creation of a 2D model of a threaded fastener placed inside a material with the counter threads already shaped to match the threads of the fastener. The material is to be anchored along the bottom edge as force is being applied at the top edges of a threaded fastener. In this concept, a force is applied directly to the threaded fastener upwards in order to put it in tension. This model would show the concentrations of strain on the threaded insert and would simulate the performance of the insert when it is tension. Being 2D and static, this model would be easy and feasible while still giving useful data on the performance of a threaded fastener in tension.

**Evaluation:** This concept does not entirely accurately represent how the insert would actually perform in a real use scenario as there is usually a bolt inside the insert. Groov-Pin wants to see interaction between bolt and insert.

**2. Simple Tensile Test with Force Directly Applied to a Bolt:** This concept is similar to Concept 1 however, in this case a bolt is placed inside the threaded fastener and a force is applied on the bolt instead of the fastener directly. This is slightly more realistic and will show strain in both the bolt and threaded fastener.

**Evaluation:** This concept is not entirely accurate as there is no incorporation of a threaded insert. Groov-Pin wants to see interaction between bolt and insert.

**3. Simple Tensile Test with a Force Applied to a Block Assembly with a Bolt and Threaded Fastener:** This concept is built off of Concept 2 in that it has a base material, a threaded fastener, a bolt, and a block of material to be joined by the bolt. This is done to simulate a physical tensile test performed with a testing frame such as an Instron machine. In this experiment, the bolt isnt being pulled directly but instead is joined with a surrounding block and the force is applied to the block. This simulates the performance of the bolt and threaded insert assembly when used in a realistic scenario.

**Evaluation:** The incorporation of a base material and clamping block assembly includes more than is required to model the interaction between bolt and insert. The nodes will be wasted on modeling extraneous parts rather than being dedicated to the interaction surface.

## Static 3D Models

These concept designs represent 3D non-dynamic models. These models blend feasibility with accuracy in that they model experiments in 3 dimensions while still not requiring more advanced techniques such as dynamic modeling. The majority of these experiments will build off of Concept 3 in which they have a full assembly including the top blocks in order to be physically replicable in with an Instron testing machine.

**4. 3D Static Tensile Test of a with Force Applied Directly to the Threaded Insert:** This model shows the performance of a threaded insert in tension where the force is applied onto the insert directly in the upwards direction. This shows in 3D where the concentration of strain is located in a 3D specimen. This method only shows an instance and does not show progression of strains as a dynamic model would. This test also is simplistic and does not show how the insert would perform in a full assembly.

**Evaluation:** This concept does not entirely accurately represent how the insert would actually perform in a real use scenario as there is usually a bolt inside the insert. Groov-Pin wants to see interaction between bolt and insert.

**5. 3D Static Tensile Test with a Force Applied to a Block Assembly with a Bolt and Threaded Fastener:** This model is built off of Concept 3 but is represented in 3D. This concept will show the locations of strain concentration in a 3D specimen and is accurate in that it models a full assembly but does not show the dynamic progression of the strain.

**Evaluation:** The incorporation of a base material and clamping block assembly includes more than is required to model the interaction between bolt and insert. The nodes will be wasted on modeling extraneous parts rather than being dedicated to the interaction surface.

**6. 3D Static Compression Test Followed by Tension Test:** This model is built off of the test performed in Concept 5 but is preceded by a compression test. The assembly model is to be placed in compression first then re-run using the same already compressed specimens in tension. This is done to observe what happens to the specimen in a realistic environment where there is not only a tensile force. This experiment is not dynamic and does not show progression of strain.

**Evaluation:** Putting the assembly into a compression loading scenario does not accurately model pullout resistance. A tensile loading models pullout more accurately than compression.

**7. 3D Static Tensile Test Repeated to 5 Cycles:** This model is built off of Concept 5 but the simulation is repeated with the same specimens over 5 repetitions in order to see the behavior of the specimens in repeated cycles. This is similar to the current standard 15 cycle testing that Groov-Pin uses to test their products but has fewer repetitions. It however does not show any dynamic progression of strain.

**Evaluation:** This concept would be simulation intensive and may not be practical. If accurate results are the goal, time spent processing the simulation would be better spent modeling a single more detailed test.

**8. 3D Static Tensile Test Repeated to 15 Cycles:** This model is built off of Concept 5 but the simulation is repeated with the same specimens over 15 repetitions in order to see the behavior of the specimens in repeated cycles. This is a model of the current standard 15 cycle testing that Groov-Pin uses to test their products. It however does not show any dynamic progression of strain.

**Evaluation:** This concept would be simulation intensive and may not be practical. If accurate results are the goal, time spent processing the simulation would be better spent modeling a single more detailed test.

**9. 3D Static Tensile Test Repeated to Bolt Failure:** This model is built off of Concept 5 but the simulation is repeated with the same specimens until the bolt fails by breaking or fracture. This will reveal the amount of repeated tension the specimen can handle before the bolt fails. Bolt failure is the goal because Groov-Pin expects the bolt to fail before their threaded insert does. It however does not show any dynamic progression of strain.

**Evaluation:** This concept would be simulation intensive and may not be practical. If accurate results are the goal, time spent processing the simulation would be better spent modeling a single more detailed test. Bolt failure can be achieved by increasing load and manipulating material properties.

**10. 3D Static Tensile Test Repeated to Insert Failure:** This model is built off of Concept 5 but the simulation is repeated with the same specimens until the threaded insert fails by breaking, fracture, or removal from base material. This will reveal the amount of repeated tension the specimen can handle before the threaded insert fails. Threaded insert failure is the goal because Groov-Pin expects the bolt to fail before their threaded insert does and it is necessary to find the point at which their product fails. This model must use a high strength bolt so that the bolt does not fail before the insert. It however does not show any dynamic progression of strain.

**Evaluation:** This concept would be simulation intensive and may not be practical. If accurate results are the goal, time spent processing the simulation would be better spent modeling a single more detailed test. Insert failure can be achieved by increasing load and manipulating material properties. Bolt failure is also expected before the insert fails.

**11. 3D Static Tensile Test of Threaded Insert with Gradually Increasing Force:** This model is built off of Concept 5 but the tension is gradually increased over multiple simulation runs. This is a way to try to emulate dynamic modeling but still being a more feasible static mode. It however may not be an entirely accurate substitute for a dynamic model.

**Evaluation:** This concept would be simulation intensive and may not be practical. If accurate results are the goal, time spent processing the simulation would be better spent modeling a single more detailed test.

**12. 3D Static Torque Test of Threaded Insert:** This is a simple torque test of a threaded insert where a rotational force is being applied to the blocks of the assembly to see how a threaded insert and block performs with an applied torque. This model however does not show dynamic progression.

**Evaluation:** This simulation doesn't model pullout resistance and does not take into account prevailing torque in the case of a deformed insert.

**13. 3D Static Torque Test of Threaded Insert Repeated to 5 Cycles:** This model builds off of Concept 12 but is repeated with the same specimens over 5 repetitions in order to see the behavior of the specimens in repeated cycles. This is similar to the current standard 15 cycle testing that Groov-Pin uses to test their products but has fewer repetitions. It however does not show any dynamic progression.

**Evaluation:** This simulation doesn't model pullout resistance and does not take into account prevailing torque in the case of a deformed insert.This concept would be simulation intensive and may not be practical. If accurate results are the goal, time spent processing the simulation would be better spent modeling a single more detailed test.

**14. 3D Static Torque Test of Threaded Insert Repeated to 15 Cycles:** This model builds off of Concept 12 but is repeated with the same specimens over 15 repetitions in order to see the behavior of the specimens in repeated cycles. This is the current standard 15 cycle testing that Groov-Pin uses to test their products. It however does not show any dynamic progression.

**Evaluation:** This simulation doesn't model pullout resistance and does not take into account prevailing torque in the case of a deformed insert.This concept would be simulation intensive and may not be practical. If accurate results are the goal, time spent processing the simulation would be better spent modeling a single more detailed test.

**15. 3D Static Torque Test of Threaded Insert Repeated to Bolt Failure:** This model is built off of Concept 12 but the simulation is repeated with the same specimens until the bolt fails by breaking, fracture, or removal from base material. This will reveal the amount of repeated torque the specimen can handle before the bolt fails. Bolt failure is the goal because Groov-Pin expects the bolt to fail before their threaded insert. It however does not show any dynamic progression of strain.

**Evaluation:** This simulation doesn't model pullout resistance and does not take into account prevailing torque in the case of a deformed insert. Bolt failure can be achieved by increasing load and manipulating material properties.This concept would be simulation intensive and may not be practical. If accurate results are the goal, time spent processing the simulation would be better spent modeling a single more detailed test.

**16. 3D Static Torque Test of Threaded Insert Repeated to Insert Failure:** This model is built off of Concept 12 but the simulation is repeated with the same specimens until the threaded insert fails by breaking, fracture, or removal from base material. This will reveal the amount of repeated torque the specimen can handle before the threaded insert fails. Threaded insert failure is the goal because Groov-Pin expects the bolt to fail before their threaded insert does and it is necessary to find the point at which their product fails. This model must use a high strength bolt so that the bolt does not fail before the insert. This experiment may not be feasible because the point of failure may be at a large number of repetitions and it is not known when the insert will fail. It however does not show any dynamic progression of strain.

**Evaluation:** This simulation doesn't model pullout resistance and does not take into account prevailing torque in the case of a deformed insert. Insert failure can be achieved by increasing load and manipulating material properties. Bolt failure is also expected before the insert fails. This concept would be simulation intensive and may not be practical. If accurate results are the goal, time spent processing the simulation would be better spent modeling a single more detailed test.

**17. 3D Static Torque Test of Threaded Insert with Gradually Increasing Force:** This model is built off of Concept 12 but the torque is gradually increased over multiple simulation runs. This is a way to try to emulate dynamic modeling but still being a more feasible static mode. It however may not be an entirely accurate substitute for a dynamic model.

**Evaluation:** This simulation doesn't model pullout resistance and does not take into account prevailing torque in the case of a deformed insert.This concept would be simulation intensive and may not be practical. If accurate results are the goal, time spent processing the simulation would be better spent modeling a single more detailed test.

## Dynamic 3D Models

These models will be both 3D and dynamic showing the progression of strain in a 3D specimen or assembly as the force is being applied over a certain period of time or number of intervals. These models will most accurately represent the performance of threaded inserts and will meet all the requirements Groov Pin has set for the project.

**18. 3D Dynamic Tensile Test with a Constant Force Over a Period of Time or Standard Intervals:** This model is a dynamic tensile test with a constant force over defined intervals of time. This model aims to show the progression of stress/strain and critical locations on a threaded insert or bolt. It is necessary to have a dynamic model because it shows progression rather than a single instance.

**Evaluation:** Dynamic modeling is advanced and has an associated learning curve thus making it not as feasible as static models. This model would be better saved for later on in the project after a static model has been achieved.

**19. 3D Dynamic Tensile Test with a Constant Force Until Bolt Failure:** This model is a dynamic tensile test with a constant force over defined intervals of time. This model aims to show the progression of stress/strain and critical locations on a threaded insert or bolt until the bolt fails by breaking or fracturing. The goal is bolt failure because

Groov-Pin expects bolts to fail before their threaded inserts do. It is necessary to have a dynamic model because it shows progression rather than a single instance.

**Evaluation:** Dynamic modeling is advanced and has an associated learning curve thus making it not as feasible as static models. This model would be better saved for later on in the project after a static model has been achieved.

**20. 3D Dynamic Tensile Test with a Constant Force Until Insert Failure:** This model is a dynamic tensile test with a constant force over defined intervals of time. This model aims to show the progression of stress/strain and critical locations on a threaded insert or bolt until the threaded insert fails by breaking, fracturing, or removal from base material. The goal is threaded insert failure because Groov-Pin expects bolts to fail before their threaded inserts and it is necessary to find the point at which their product fails. This model must use a high strength bolt so that the bolt does not fail before the insert. This experiment may not be feasible because the point of failure may be at a large number of repetitions and it is not known when the insert will fail. It is necessary to have a dynamic model because it shows progression rather than a single instance.

**Evaluation:** Dynamic modeling is advanced and has an associated learning curve thus making it not as feasible as static models. This model would be better saved for later on in the project after a static model has been achieved. Bolt failure is also expected to occur before the insert fails.

**21. 3D Dynamic Tensile Test with a Constant Force Repeated to 15 Cycles:** This model is a dynamic tensile test with a constant force over defined intervals of time which is then repeated over 15 cycles using the same specimen. This model aims to show the progression of stress/strain and critical locations on a threaded. 15 cycles are used because it is the current standard of repetitions that Groov-Pin uses to test their inserts.

It is necessary to have a dynamic model because it shows progression rather than a single instance. Dynamic modeling however is advanced and has an associated learning curve thus making it not as feasible as static models.

**Evaluation:** Dynamic modeling is advanced and has an associated learning curve thus making it not as feasible as static models. This model would be better saved for later on in the project after a static model has been achieved. This concept would be simulation intensive and may not be practical. If accurate results are the goal, time spent processing the simulation would be better spent modeling a single more detailed test.

**22. 3D Dynamic Torque Test with a Constant Force Over a Period of Time or Standard Intervals:** This model is a dynamic torque test with a constant force over defined intervals of time. This model aims to show the progression of stress/strain and critical locations on a threaded insert or bolt. It is necessary to have a dynamic model because it shows progression rather than a single instance. Dynamic modeling however is advanced and has an associated learning curve thus making it not as feasible as static models.

**Evaluation:** Dynamic modeling is advanced and has an associated learning curve thus making it not as feasible as static models. This model would be better saved for later on in the project after a static model has been achieved. This concept would be simulation intensive and may not be practical.

**23. 3D Dynamic Torque Test with a Constant Force Until Bolt Failure:** This model is a dynamic torque test with a constant force over defined intervals of time. This model aims to show the progression of stress/strain and critical locations on a threaded insert or bolt until the bolt fails by breaking or fracturing. The goal is bolt failure because Groov-Pin expects bolts to fail before their threaded inserts do. It is necessary to have a dynamic model because it shows progression rather than a single instance. Dynamic modeling however is advanced and has an associated learning curve thus making it not as feasible as static models.

**Evaluation:** Dynamic modeling is advanced and has an associated learning curve thus making it not as feasible as static models. This model would be better saved for later on in the project after a static model has been achieved. This concept would be simulation intensive and may not be practical.

**24. 3D Dynamic Torque Test with a Constant Force Until Insert Failure:** This model is a dynamic torque test with a constant force over defined intervals of time. This model aims to show the progression of stress/strain and critical locations on a threaded insert or bolt until the threaded insert fails by breaking, fracturing, or removal from base material. The goal is threaded insert failure because Groov-Pin expects bolts to fail before their threaded inserts and it is necessary to find the point at which their product fails. This model must use a high strength bolt so that the bolt does not fail before the insert. This experiment may not be feasible because the point of failure may be at a large number of repetitions and it is not known when the insert will fail. It is necessary to have a dynamic model because it shows progression rather than a single instance. Dynamic modeling however is advanced and has an associated learning curve thus making it not as feasible as static models.

**Evaluation:** Dynamic modeling is advanced and has an associated learning curve thus making it not as feasible as static models. This model would be better saved for later on in the project after a static model has been achieved. This concept would be simulation intensive and may not be practical. Bolt failure is expected to occur before insert failure occurs.

**25. 3D Dynamic Torque Test with a Constant Force Repeated to 15 Cycles:** This model is a dynamic torque test with a constant force over defined intervals of time which is then repeated over 15 cycles using the same specimen. This model aims to show the progression of stress/strain and critical locations on a threaded. 15 cycles are used because it is the current standard of repetitions that Groov-Pin uses to test their inserts. It is necessary to have a dynamic model because it shows progression rather than a single instance. Dynamic modeling however is advanced and has an associated learning curve thus making it not as feasible as static models.

**Evaluation:** Dynamic modeling is advanced and has an associated learning curve thus making it not as feasible as static models. This model would be better saved for later on in the project after a static model has been achieved. This concept would be simulation intensive and may not be practical. If accurate results are the goal, time spent processing the simulation would be better spent modeling a single more detailed test.

## Dynamic 3D Models with Shearing and Other Advanced Features

These models are 3D and dynamic while including cutting edge modeling features such as shearing of material as a threaded insert is tapped into the material. These models involve a high level of research and represent the greatest learning curve. These experimental models may also stretch the limits and functions of existing software but may still be possible to perform physically.

**26. 3D Dynamic Model of a Self Tapping Threaded Insert as it Taps Into a Base Material:** This model is 3D dynamic model of a self tapping threaded insert tapping into a base material and cutting away at the material as it is installed. This model will show how the insert performs as it is being installed as well as showing stress/strain concentrations on the material as it is being sheared away. This model will provide a lot of insight into the process of installing a threaded insert and will produce a model and associated data that has never been produced or pursued before.

**Evaluation:** This design concept involves cutting edge techniques that although possible in the Abaqus finite element analysis software, are very difficult and are not as feasible. Pursuing a simpler 3D dynamic model without cutting/shearing may be more beneficial as those concepts are more feasible and realistically achievable.

**27. 3D Dynamic Model of a Threaded Insert Assembly under Vibrating Conditions:** This model is a 3D dynamic model of a threaded insert assembly subjected to vibratory conditions. It is not yet known whether it is possible to create digital experiments with vibratory conditions or whether or not the Abaqus software offers this as a feature. Because of the environments the products are being used in (automotive and aerospace) are associated high levels of vibration, it is important to test for performance under vibration.

**Evaluation:** This concept may not as feasible and thus not as useful as a simpler 3D dynamic model of a tension or torque test. Groov-Pin is also mainly concerned with pull out resistance and prevailing torque rather than vibration resistance despite its importance.

**28. 3D Dynamic Model of a Tensile Test for a Threaded Insert with Lubrication:** Groov-Pin mentioned through meetings with the design team that certain assemblies involve the use of lubrication. Therefore, it may be beneficial that a model of an assembly that incorporates lubrication be created.

**Evaluation:** Despite the importance of lubrication in the assembly, this concept may not be as beneficial as a simpler 3D dynamic model without lubrication due to the difficulties and uncertainties involved in modeling lubrication.

**29. 3D Dynamic Model of a Threaded Insert Tapping into a Material, Then Removed and Repeated:** This is a 3D dynamic model of a threaded insert that builds off of Concept 26. This concept would model the performance of the threaded insert after it has already been used once and then used again. This model would provide a lot of insight into the reuse of threaded inserts but assumes that Concept 25 has already been achieved and can be expanded upon.

**Evaluation:** This concept assumes concept 25 has been created. his concept may not as feasible and thus not as useful as a simpler 3D dynamic model of a tension or torque test. Groov-Pin is also mainly concerned with pull out resistance and prevailing torque rather than and material tapping is a secondary goal best saved for ideal scenarios where the project is ahead of schedule.

**30. 3D Dynamic Model of a Threaded Insert Assembly In Tensile and Subjected to Extreme Temperatures:** This design concept is a 3D dynamic model of a threaded insert assembly in a tensile test but also subjected to extreme temperatures. Temperatures will reflect both very cold temperatures and very high temperatures to account for variance in operating conditions in which the products are often used (automotive or aerospace applications that experience extreme fluctuations in temperature). This concept is advanced in that it requires the tensile testing of an assembly in a cold environment and then repeated in a hot environment.

**Evaluation:** This concept would produce useful data on the performance of threaded inserts but goes beyond what is necessary in evaluating critical factors to pull out resistance and prevailing torque.

## 7.2 Josh Blacker's Concepts

## 2-D Models

**1.Tensile test to test for the failure of the insert:** This test would be a 2-D test with the head of the bolt in the threaded insert undergoing a uniaxial tension force. The bolt would need to be made of a material that is stronger than stainless steel so it does not fail before the threaded insert does. The results of this test will give the maximum stress that the insert can withstand before it undergoes plastic deformation.

**Evaluation** This simulation is for the pull-out force test however it does not satisfy the full requirements that Groov-Pin would like the team to use. This concept would be a subset of the pull-out force test.

**2. Tensile test to show stress distribution on different insert threads at different loads:** This test would be a 2-D test with the head of the bolt in the threaded insert undergoing a uniaxial tension force. The size of the threads will influence the stress distribution across the threads. This test will look at the stress on the outer part of the thread and inner part at different loads.

**Evaluation** This simulation is for the pull-out force test however it does not satisfy the full requirements that Groov-Pin would like the team to use. This concept would be a subset of the pull-out force test.

**3. Tensile test at high temp to test deformation:** This test would be a 2-D test with the head of the bolt in the threaded insert undergoing a uniaxial tension force. Because most threaded inserts can not withstand elevated temperatures that occur above 300 Celsius, this test will show at what temperature the different inserts will fail at.

**Evaluation** This simulation is for the pull-out force test however it does not satisfy the full requirements that Groov-Pin would like the team to use. This concept would be a subset of the pull-out force test.

**4. Tensile test at different heights of inserts, (influence thread distortion lengths):** This test would be a 2-D test with the head of the bolt in the threaded insert undergoing a uniaxial tension force. This test will test different heights of inserts and examine the stress on the threads at each length to find a relationship between the thread distortion length and insert height.

**Evaluation** This simulation is for the pull-out force test however it does not satisfy the full requirements that Groov-Pin would like the team to use. This concept would be a subset of the pull-out force test.

**5. Tensile test using different metals for deformation:** This test would be a 2-D test with the head of the bolt in the threaded insert undergoing a uniaxial tension force. This test will examine how a threaded inserts material influences the deformation of the threads.

**Evaluation** This simulation is for the pull-out force test however it does not satisfy the full requirements that Groov-Pin would like the team to use. This concept would be a subset of the pull-out force test.

**6. Compression test (load on insert is uniform):** This test would be a 2-D test with the head of the bolt in the threaded insert undergoing a uniaxial compression force. It is important for the threaded insert to have a uniform load so that it minimizes the stress on the inserts when it undergoes tension or compression. This test will test each insert for the load distribution.

**Evaluation** This simulation is related to the pull-out force test however it does not satisfy the full requirements that Groov-Pin would like the team to use. Groov-Pin is not interested in the bolt undergoing a compressive force.

**7. Tensile test to measure stress in each individual thread:** This test would be a 2-D test with the head of the bolt in the threaded insert undergoing a uniaxial tensile force. This test considers the fact that the first two threads in a bolt are known to take on 80 percent of the load. This test will examine the percentage of load each thread in the insert is taking on as it is in tension.

**Evaluation** This simulation is for the pull-out force test. This concept would be a subset of the pull-out force test. Groovy-Pin is currently looking for ways to test the interaction of a few threads rather than the full interaction of the insert and bolt to allow more nodes to be in a smaller area for more accurate results.

## 3-D models

**8. Torque test to test for friction between threaded insert and bolt:** This will be a 3-D test with the bolt inside the threaded insert undergoing a torque in the counterclockwise direction. Friction between the insert and the bolt influences prevailing torque. The more friction between the bolt and insert, the better the performance of the insert.

**Evaluation** This simulation is for the prevailing torque test. This concept would be a subset of the prevailing torque test. Groovy-Pin is currently looking for ways to incorporate friction between the threads in the Abaqus model.

**9. Torque test to test for friction on outer edge of insert and base material:** This will be a 3-D test with the bolt inside the threaded insert undergoing a torque in the counterclockwise direction. Friction between the insert and the base material influences prevailing torque. The more friction between the base material and insert, the better the performance of the insert.

**Evaluation** This simulation is for the prevailing torque test. This concept would be a subset of the prevailing torque test, however Groov-Pin is not interested in the interaction between the base material and the insert outer threads.

**10. Torque removal test with plating:** This will be a 3-D test with the bolt inside the threaded insert undergoing a torque in the counterclockwise direction. Plating is used in threaded inserts to resist prevailing torque because it adds an extra layer on the threads that creates more friction between the base material and insert. This model will show the interaction between the plating and base material, and measure the torque it takes to rotate 360 degrees.

**Evaluation** This simulation is for the prevailing torque test. This concept would be a subset of the prevailing torque test, however Groov-Pin is not interested in the interaction between the base material and the insert outer threads.

**11. Test Lubricants with different viscosities with torque test:** This will be a 3-D test with the bolt inside the threaded insert undergoing a torque in the counterclockwise direction. Some bolts use a lubricant to ease installation and removal of inserts. This will test different lubricants and examine the torque required to rotate the insert 360 degrees.

**Evaluation** This simulation is for the prevailing torque test. This concept would be a subset of the prevailing torque test.

**12. Test Lubricants with different coefficients of friction with torque test:** This will be a 3-D test with the bolt inside the threaded insert undergoing a torque in the counterclockwise direction. Some bolts use a lubricant to ease installation and removal of inserts.

This will test different lubricants and examine the torque required to rotate the insert 360 degrees.

**Evaluation** This simulation is for the prevailing torque test. This concept would be a subset of the prevailing torque test. This will be of interest to find what method creates more prevailing torque.

**13. Torque test for insert having coating material at high viscosity of coating chemical:** This will be a 3-D test with the bolt inside the threaded insert undergoing a torque in the counterclockwise direction. The coating material is used to control prevailing torque by creating more friction in the threads and the base material. This test will examine the amount of torque required to loosen the insert 360 degrees.

**Evaluation** This simulation is for the prevailing torque test. This concept would be a subset of the prevailing torque test. This will be of interest to find if coating method creates more prevailing torque.

**14. Installation test for insert having coating material at high viscosity of coating chemical:** This will be a 3-D Dynamic test that will examine the torque required for the installation process of an insert containing a coating into a base material.

**Evaluation** This would be a subset of the model that will show the insert cutting into the base material. Groovy-Pin will need to look into how to add a coating material to the insert in Abaqus.

**15. Create test that vibrates the base material at a high frequency in the x,y,z-direction:** This test is a 3-D Dynamic with the bolt inserted in the insert. The test that will show the deformation of the threaded insert and the base material as it undergoes a high frequency of vibration.

**Evaluation** Groov-Pin is only interested in testing the pull-out force with a tensile test and a prevailing torque with a torque applied on the bolt.

**16. Create test that vibrates the base material at a high frequency in the x,y,z-direction with plated insert:** This test is a 3-D Dynamic with the bolt inserted in the insert. The test that will show the deformation of the threaded insert and the base material as it undergoes a high frequency of vibration.

**Evaluation** Groov-Pin is only interested in testing the pull-out force with a tensile test and a prevailing torque with a torque applied on the bolt.

**17. Reverse torque test that measures the torque resistance in each individual thread:** This will be a 3-D test with the bolt inside the threaded insert undergoing a torque in the counterclockwise direction. The test will examine the torque required to rotate the insert 360 degrees. In order to test the torque in each individual thread. The test will begin with just one thread in the base material and the next test will have two threads in the base material and so forth. Each test will measure the total torque and show the relationship between number of threads and prevailing torque.

**Evaluation** Groov-Pin is only interested in the interaction between the inside threads of the insert and the bolt during the prevailing torque test.

**18. Test for deformation between insert and base material while inserting threaded insert:** This test will be a 3-D dynamic test that will observe the stress and strain that the base material and the threaded insert undergo during the insertion process.

**Evaluation** Groov-Pin is only interested in the interaction between the inside threads of the insert and the bolt during the prevailing torque test. Groov-Pin is also not interested in the insertion process.

**19. Test for temperature between insert and base material while inserting threaded insert:** Galling can occur during the installation process of the threaded insert due to an increase in temperature.This simulation will be a dynamic 3-D model that shows the threaded insert being inserted into a tapped hole. This will test for the temperature between the base material and insert threads at different rpms.

**Evaluation** Groov-Pin is only interested in the interaction between the inside threads of the insert and the bolt during the prevailing torque test. Groov-Pin is not interested in the insertion process.

**20. Test for friction between insert and base material while inserting threaded insert:** Galling can occur during the installation process of the threaded insert due to an increase in friction.This simulation will be a dynamic 3-D model that shows the threaded insert being inserted into a tapped hole. This will test for the friction between the base material and insert threads at different rpms.

**Evaluation** Groov-Pin is only interested in the interaction between the inside threads of the insert and the bolt during the prevailing torque test. Groov-Pin is not interested in the insertion process.

**21. Test surface contact area between threads and base material in a self-tapped insert:** The surface contact area between the threads and the base material can influence the pull out resistance and the prevailing torque. This will be a 3-D static model that shows the bolt in the insert locked in place. The self-tapped inserts will be examined in this experiment to find the amount of surface area of the threads that is in contact with the base material.

**Evaluation** This would be a subset of the dynamic model that shows the self tapped insert cutting away at material. It will examine the tolerance between the insert and the base material.

**22. Find shear stress on threads during pull out test:** This test is a 3-D Static model that undergoes a uni-axial tensile test. The shear stress on the inserts will be calculated at different loads.

**Evaluation** This is a subset of the pull-out force test where different forces will be applied to see the results of a higher load and where the failure of the insert is reached.

**23. Temperature test that shows corrosion of stainless steel and other metals:** Galling may take place in the threaded insert during the installation process due to the friction and heat that is produced. This model will show the corrosion of stainless steel during the installation process with a 3-D Dynamic model.

**Evaluation** Groov-Pin is not interested in the installation process.

**24.) Test surface contact area between threads and base material in a tapped insert:** The surface contact area between the threads and the base material can influence the pull out resistance and the prevailing torque. This will be a 3-D static model that shows the bolt in the insert locked in place. The tapped inserts will be examined in this experiment to find the amount of surface area of the threads that is in contact with the base material.

**Evaluation** This would be a subset of the pull-out force test and the prevailing torque test to test for the tolerance between the threads of the insert and bolt as well as the base material.

## Repeatability

**25. Create model that can be used for different types of inserts interchangeably:** This model will be useful for the user to easily test different inserts with the same simulation on the Abaqus software.

**Evaluation** This concept is a subset of the dynamic model where an interchangeable model is a goal for Groovy-Pin.

**26. Create model that simulates prevailing torque test with 15 cycles:** This 3-D dynamic model will simulate a physical prevailing torque test that Groov-Pin uses. This model will show the torque required for the threaded insert to rotate 360 degrees after being installed into a base material. This process runs through this test for 15 cycles.

**Evaluation** This is a subset of the prevailing torque test and will be important for Groovy-Pin to run the same test fifteen cycles. However one cycle would be the bolt getting fully screwed out of the insert rather than just 360 degrees.

**27. Create model that simulates pull-out force test:** This 3-D dynamic model would be a simulation of the pull-out force test that Groov-Pin uses today. It would be a faster testing process because it would be automatic rather than manual.

**Evaluation** This concept is the pull-out force test that Groovy-Pin is trying to simulate in Abaqus with a simple tensile force.

**28. Create model that changes tapped hole size:** This model would allow the user to change the base material and the tapped hole size. Some inserts use a tapered hole at different angles of taper. This would be necessary to test all possible threaded insert installations.

**Evaluation** This concept is a subset of the dynamic model where an interchangeable model is a goal for Groovy-Pin.

**29. Create model that shows insert screwed into base with a lubricant:** This model would be a 3-D Dynamic simulation that shows the lubricants influence on the ease of the threaded insert being inserted into the base material.

**Evaluation** This concept is a subset of the dynamic model. Groovy-Pin will be looking into a way to add lubrication to the Abaqus model.

**30. Create model that shows insert self tapped only partially into the base, taken out, then self tapped in the same hole:** This would be a 3-D Dynamic simulation that shows a self tapped insert cutting away at the base material. The insert would not be fully installed into the material before it is taken out and reapplied into the material to lock it into place. It would observe the way the base material is cut and deformed to allow the threaded insert to sit locked in place. This would simulate a failed installation followed by a correct installation.

**Evaluation** This concept is a subset of the dynamic model that will be rather difficult but is of interest to Groov-Pin.

## 7.3 Tyler Kelly's Concepts

## Physical Experiments

**1. Apply an insert into a tapped hole and performing destructive tests to observe behavior in Aluminum and Magnesium.** By utilizing the Instron machine, tension can be measured until insert failure. An assembly would need to be created to perform this experiment. Start with a base plate composed of 6061 Aluminum Alloy or Magnesium Alloy, approximately five inches by five inches in length and twice the thickness of the insert. In the center of this plate would be a tapped hole that will be used to place the threaded insert into, followed by a plate half as thick as the base plate, and then a bolt. To completely understand the behavior of the threaded insert and all mated components, it would be wise to make sure that the bolt is completely tightened. By using a set of jaws attached to the Instron machine which would grip the head of the bolt, the bolt would be placed into tension until failure. Using the computer software from the Instron machine, several variables may be recorded to provide a better understanding of this phenomena.

**Evaluation:** This method, while very important to gaining a physical understanding of what exactly is going on between the bond, is not applicable to our FEA experiments.

**2.Apply an insert into a tapped hole and perform destructive torque test to observe behavior in Aluminum and Magnesium.** By using an assembly identical to that in Concept 1, the amount of torque limited to the insert and/or bolt may be determined with some basic equipment. According to manufacture specifications, all bolts and threaded inserts will have a calculated maximum torque before it fails. This will be tests by the use of a torque wrench and the aforementioned assembly. By using different sized tapped holes, threaded inserts, and bolts as well as different base materials, a basic understanding of the factors involved in insert and bolt failure may be observed.

**Evaluation:** This method, while very important to gaining a physical understanding of what exactly is going on between the bond, is not applicable to our FEA experiments.

**3.Use a self-tapping insert and performing destructive torque test to observe behavior in Aluminum and Magnesium.** Instead of using a tapped hole, this concept involves the use of a self-tapping threaded insert. This provides greater strength along with a tighter fit between the threaded insert and the base material. Prevailing torque is one of the main factors that needs to be evaluated in regards to self-tapping inserts and by performing the same experiment explained in Concept 1, the variables involved with failure can be recorded and then compared to threaded inserts using tapped holes in the base material.

**Evaluation:** This method, while very important to gaining a physical understanding of what exactly is going on between the bond, is not applicable to our FEA experiments.

**4.Use a self-tapping insert and performing destructive test torque to observe behavior in Aluminum and Magnesium.** By using an assembly identical to that in Concept 3, the amount of torque required for failure of the insert and/or bolt may be determined. By following the same procedure from Concept 3, the results from this concept may be compared to the results in the threaded insert used in a tapped hole.

**Evaluation:** This method, while very important to gaining a physical understanding of what exactly is going on between the bond, is not applicable to our FEA experiments.

## 2D Models

**5.Using a modeled threaded insert inside of a plate with only the first 3 threads attached and applying tension and compression.** By creating a profile of the base material (aluminum or magnesium), the threaded insert, a plate half as thick as the base plate, and the bolt, it would be relatively simple to apply tension to the insert and bolt in Abaqus. It would be wise to use a profile drawn from the exact center of both plates, the threaded insert, and the bolt. This will provide the most accurate results. It is pivotal that all 2D models are created with an exact thread profile and it is imperative to make sure that they are joined together properly. After some research, the maximum stress is achieved within the first few threads upon inserting into the base plate. That is why it is imperative to measure applied loads and forces at the first three threads only.

**Evaluation:** This design concept is limited to just the first few threads and does not provide us with the entire interaction. Groov-Pin also does not want 2D models and it would be impossible to perform any tests due to the nature of our design problem.

**6. Using a modeled threaded insert inside of a plate with all threads attached and applying tension and compression.** By applying the same models created from Concept 5, the finite element method may be applied to a fully tightened assembly. Thus, all three thread profiles must be completely attached and perfectly aligned to provide accurate results. As stated previously, these tests should be performed with the base plate consisting of both aluminum and magnesium and the center plate being half as thick as the base plate.

**Evaluation:** Groov-Pin does not want 2D models and it would be impossible to perform any tests due to the nature of our design problem. It is limited to two dimensions.

## 3D Static Models

**7. Model of an insert with exterior coarse threads without bolt and force applied directly to insert until failure.** For this concept, the use of a threaded insert (standard or self-tapping) with a coarse thread profile will be inserted into the base plate composed of aluminum or magnesium. Once the threaded insert is mated concentrically to the base plate, forces will be applied directly to the insert until it fails. While this concept might involve some large forces to induce failure, it will still be very important to understand all variables involved in creating pull-out resistance and prevailing torque.

**Evaluation:** This method is limited to just a coarse thread pattern and our designed experiments should be diverse enough to be applied to any type of threads.

**8. Model of an insert with exterior fine threads without bolt and force applied directly to insert until failure.** The same exact method will be used in Concept 7 with the main different is that the external thread profile of the insert will be fine rather than coarse. This might provide some important insight into whether a coarse or fine thread profile between the base plate and the exterior of the insert has a dramatic difference on variable such as friction, stress, torque, etc.

**Evaluation:** This method is limited to just a fine thread pattern and our designed experiments should be diverse enough to be applied to any type of threads.

**9. Model of an insert with exterior coarse threads with a bolt inside of the insert and force directly applied to bolt to determine which fails first.** This concept will involve a 3D model of the base plate (aluminum or magnesium), threaded insert with a coarse exterior thread profile (standard or self-tapping), a center plate half as thick as the base plate, and a bolt. All of these models will be mated appropriately and various forces will be applied directly to only the bolt. It is ideal in any application that uses a threaded insert and a bolt that the bolt always fails first. If, by using this concept, certain directed forces are determined to make the threaded insert fail first, then that data may be applied to the design of this project.

**Evaluation:** This method is limited to just a coarse thread pattern and our designed experiments should be diverse enough to be applied to any type of threads.

**10. Model of an insert with exterior fine threads with military standard bold inside of insert and force directly applied to the bolt to determine which fails first.** This concept uses an identical method to the one explained in Concept 9, with the major different being that the exterior thread profile of the threaded insert being fine rather than coarse (the thread profile of the base plate with also be fine). This could potentially uncover some very important information regarding the factors involved in bolt or insert failure.

**Evaluation:** This method is limited to just a fine thread pattern and our designed experiments should be diverse enough to be applied to any type of threads.

**11. Use an insert without bolt and force directly applied to insert at elevated temperatures.** Considering that many threaded inserts are used in automotive and aeronautical applications to provide additional strength where it is needed, it is important to assume that these inserts will be applied at extreme temperatures. Due to the fact that the threaded inserts associated with this project are used in softer metals, extreme temperatures might lead to deformation of the base material and subsequently the threaded insert. This could lead to catastrophic failure. This concept will involve the use of finite element analysis to examine stresses on the base material and the threaded insert alone at elevated temperatures. While this is not practical for use in the real world, it will provide a basic understanding that may be compared to an actual assembly with a bolt that has been fastened to the insert.

**Evaluation:** This method is applicable to our designed digital FEA experiments but is limited to just one scenario. It will, however, be a component of our future designed experiments.

**12. Use an insert with bolt and force directly applied to the bolt at elevated temperatures.** This concept will involve a model of a base plate, threaded insert, center plate half as thick as the base plate, and a bolt. The assembly will be examined using the finite element method but at extreme temperatures. The force will be applied to only the bolt so determine its behavior. The results from this concept may then be compared to the results of Concept 11.

**Evaluation:** This method is applicable to our designed digital FEA experiments but is limited to just one scenario. It will, however, be a component of our future designed experiments.

## 3D Dynamic Models

**13. Model of the self-tapping insert and a plate with a hole for the first 3 threads to observe behavior.** By using a model of a base plate with a hole in the center and a model of the self-tapping threaded insert, the behavior observed for the first three threads of the insert can be analyzed. Considering that this is where most of the stress occurs, a dynamic model can provide an active picture of what truly goes on during this process.

**Evaluation:** This method is ideal due to the fact that the models and analysis are in three dimensions. However, it is limited to just the first few threads.

**14. Model of the self-tapping insert and a plate with a hole during the advancement process to observe behavior.** The second process of a thread forming insert is called the advancement process. This takes place after the first few threads have been formed and before the insert is fully tightened. By using the same models from Concept 13, various stresses may be analyzed.

**Evaluation:** This method is ideal due to the fact that the models and analysis are in three dimensions. But it is limited to just one component of what needs to be analyzed.

**15. Model of the self-tapping insert and a plate with a hole during the tightening process to observe behavior.** The last step in the thread forming process between the insert and the base plate is when it is tightened. By using the same models from Concept 13, various stresses may be analyzed as well as the torque.

**Evaluation:** This method is ideal due to the fact that the models and analysis are in three dimensions. But it is limited to just one component of what needs to be analyzed.

**16. Use an insert with exterior coarse threads without bolt to evaluate torque until failure.** The use of an insert with a coarse thread profile should be used for this concept.

**Evaluation:** This method is limited to a coarse thread pattern. The actual design concepts are much more complex.

**17. Use an insert with exterior fine threads without bolt to evaluate torque until failure.** The use of an insert with a fine thread profile should be used for this concept.

**Evaluation:** This method is limited to a fine thread pattern. The actual design concepts are much more complex.

**18. Use an insert with exterior coarse threads with bolt inside of insert to evaluate torque and see which one fails first.** By using a base plate, thread insert (standard or self-tapping), and a center plate half as thick as the base plate mated together, a bolt may be dynamically fastened into the assembly to measure various torques involved in the process. It would be wise to apply torque until either the bolt or the threaded insert fails.

**Evaluation:** This method is limited to a coarse thread pattern. The actual design concepts are much more complex and this will be a single component of our final design.

**19. Use an insert with exterior fine threads with bolts inside of insert to evaluate torque and see which one fails first.** Apply the same method from Concept 18 with the main difference being that the thread profile for the exterior of the threaded insert and the thread profile of the hole in the base plate are fine rather than coarse.

**Evaluation:** This method is limited to a fine thread pattern. The actual design concepts are much more complex and this will be a single component of our final design.

**20. Use an insert with bolt at elevated temperatures with applied torques directly on the bolt until failure of the insert/bolt.** By creating a model of a base plate, threaded insert, and a center plate half as thick as the base plate, they can be mated appropriately and remain fixed. Then by taking a bolt and dynamically inserting it into this assembly, various data and behavior may be observed and applied to the final design.

**Evaluation:** This method is limited to elevated temperatures. The actual design concepts are much more complex and this will be a single component of our final design.

**21. Apply a self-tapping insert into a plate and over tighten until failure and observe behavior.** Use a model of a self-tapping insert into a plate with a hold and continue tightening until the insert fails.

**Evaluation:** This method is applicable to our digital FEA experiments that will be designed but it is a single component of a much larger design.

**22. Apply a self-tapping insert that has been plated and compare results to the same insert that has not been plated.** This concept will show whether or not any plating will create the insert to fail sooner than a non-plated insert. While the amount of material added to the thickness of the insert is very small, it may be a huge factor into determining prevailing torque of the insert.

**Evaluation:** It has yet to be determined whether any type of plating will have an affect on prevailing torque and will be explored further.

**23. Apply an insert assembly with bolt and apply vibrations and observe behavior.** This concept will involve a full assembly of a base plate, threaded insert, center plate, and a bolt. Once these have been properly mated, vibrations should be applied to the assembly and all behavior should be analyzed.

**Evaluation:** Vibrations will likely have a negative effect on prevailing torque and will need to be explored further. This concept is just a single component of the much larger design.

**24. Apply an insert assembly with a bolt at elevated temperatures and apply vibrations.** This concept involves a full assembly that was depicted in Concept 23 but the assembly will be subject to elevated temperatures. Vibrations should then be applied and any deformation from the elevated temperatures will provide different results that can be analyzed.

**Evaluation:** Vibrations and elevated temperatures will likely have a negative effect on prevailing torque and will need to be explored further. This concept is just a single component of the much larger design.

**25. Apply a threaded insert into a threaded hole and deliberately cross the threads between the insert and the tapped hole in the base plate.** This dynamic model will be comprised of a base plate with a tapped hole in the center and a threaded insert. The threaded insert should be inserted into the tapped hole making sure that the threads have been crossed. Doing this may increase torque and stress at earlier stages in the fastening process and could lead to failure much sooner.

**Evaluation:** Proper installation of the threaded insert needs to be performed in order to have a strong bond. While this concept could potentially affect the strength, it is presumed for our experiments that the threaded insert will be installed correctly.

**26. Apply a self-tapping insert into the base plate at an angle so the bolt will not be parallel when tightened.** This concept will involve a base plate with a hole and a self-tapping insert. The insert will be inserted into the hole of the base plate at a slight angle. Then a center plate and a bolt will be mated to this assembly, creating a non-parallel mate between the bolt and the center plate. Different torques should be applied directly to the bolt until failure and then compared to previous models to compare the importance of concentric mates.

**Evaluation:** Proper installation of the threaded insert needs to be performed in order to have a strong bond. While this concept could potentially affect the strength, it is presumed for our experiments that the threaded insert will be installed correctly.

**27. Apply an abundance of lubrication to a threaded insert and a tapped hole in a base plate to determine friction and stresses.** This concept will involve the use of a threaded insert and a base plate with a tapped hole. By applying a large amount of lubricant between the plate and the insert, torque may be increased as well as friction. All behavior should be analyzed on the insert.

**Evaluation:** Lubrication will reduce friction and will certainly affect all aspects of our digital FEA experiments. This is, however, a small component of the larger design.

**28. Induce galling between a stainless steel insert and a base material to determine its effects.** Galling is a significant problem with threaded inserts and fasteners. By inducing such an effect in Abaqus, the phenomena might be better understood and can aid in the prevention in future use.

**Evaluation:** Inducing galling will prove to be too challenging for the scope of this project.

**29. Increase shear area dramatically.** This concept will allow the finite element method to determine how a larger shear area will affect the average shear stress between the threaded insert and the base plate.

**Evaluation:** This is a small component of the larger design.

**30. Reduce shear area dramatically.** This concept is identical to Concept 29 but will show how a smaller shear area will affect the average shear stress.

**Evaluation:** This is a small component of the larger design.

A quality function deployment chart was necessary for the group to organize demanded qualities and characteristics for the model. The QFD as shown in Figure 112 in the Appendix examines the interactions between each quality and characteristic as well as shows how each characteristic interacts with each other.The QFD can also be used to evaluate competition and determine which characteristic is most important to the design.

# 8 Quality Function Deployment

## 8.1 Demanded Quality

### Pull-Out Resistance

The first demanded quality is that of pull-out resistance. The digital experiments that the team will be developing need to be able to determine and evaluate the amount of pull-out resistance between the interior threads of the threaded insert and the exterior threads of the bolt. The interaction between the external threads of the insert and the base material also must be tested until the base material fails or the insert is pulled out one thread pitch. This is one of our major design concepts.

### Prevailing Torque Resistance

The second demanded quality is identifying the amount of prevailing torque involved between the bolt and threaded insert. Prevailing torque is created when a bolt is screwed into a deformed insert. Prevailing torque creates resistance between the mating of a bolt and insert. Groov-Pin wants a simulation that models this type of interaction in order to determine what attributes can help resist the occurrence of prevailing torque. This simulation involved a series of complex functions that should be explored further in the future.

### Dynamic Models

The third demanded quality is dynamic models. The team will need to develop FEA models that are dynamic, that is, models that are in motion. This demanded quality is of particular importance to that of prevailing torque. Another important aspect to this demanded quality is to design them so that material may be removed during the self-tapping process of those particular threaded inserts. This takes the dynamics models one step further.

### Repeatability of Process

Designing digital FEA experiments for this project is vital but it is even more vital that these processes are repeatable. This brings us to our fourth demanded quality. Any experiment, whether it be physical or digital, needs to be repeatable and output precise results. This demanded quality is no different and needs to be applied to each concept that we will design.

### Minimize Insert Failure

The last demanded quality that the team decided to include is minimizing failure of the threaded insert. Per Groov-Pins advisement, the bolt should always break before the threaded insert does under any circumstance. This dramatically reduces the amount of parts that need to be eliminated if the bolt breaks first. A bolt can be easily removed if it breaks, but attempting to remove a threaded insert from its base material will prove to be complex. It is likely that the base material will need to be replaced if the threaded insert breaks before the bolt, which is not ideal.

## 8.2 Quality Characteristics

### Vibrations

The first quality characteristic is reducing the effect that vibrations have on the bond in the assembly. Vibrations may be detrimental to the stability of the bond between the base material, threaded insert, and bolt by potentially loosening them from each other. This is something that needs to be avoided at all cost. Vibrations involves a series of very complex functions that should be explored further in the future.

## Performance at Extreme Temperatures

Due to the fact that some of these threaded inserts are used in the aerospace industry, performance at extreme temperatures is the teams second quality characteristic. Elevated temperatures tend to make materials more ductile, especially in softer metals such as aluminum and magnesium. These experiments should be able to identify these effects on the materials. Evaluating performance at extreme temperatures would need to be determined experimentally by finding different yield stresses and using these values to analyze them in Abaqus. This should be explored further in the future.

## Materials

Our third demanded characteristic are materials and was rated the highest by the team. Materials of the base material, threaded insert, bolt, and nut are all important to a mechanical design. The materials used in the assembly will change based on what is being used and should be examined cautiously.

## Load Distribution

The fourth demanded characteristic is load distribution. Since the team will be investigating the effects between all of the bonds in the assembly, it will be vital to identify how the varying distributions of load will affect the behavior of the threaded insert.

## Lubrication

Lubrication is the fifth demanded characteristic and is used primarily with pre-tapped threaded inserts. As a team, we wanted to explore how lubrication affects the bond, which brings us to the next demanded characteristic. Lubrication involves evaluating friction and the effects that lubrication has on it. However, the academic version of Abaqus does not allow the use of friction so this became outside the scope of our project.

## Friction

Friction is something that can either have a positive or negative effect in mechanical designs, particularly when it comes to fasteners. This characteristic will need to be included in all parts of our digital experiments so we can closely replicate real life situations. However, the academic version of Abaqus does not allow the use of friction so this became outside the scope of our project.

## Torque

The seventh demanded characteristic is torque which is required for any time of fastener with any type of thread pattern. Having a deep understanding of the amount of torque required to fasten a bolt to the threaded insert without premature failure or failure of any type will be important in the scope of the project. However, it was discovered through the completion of the project that measuring and modeling a simulation that involved torque required extensive research and became outside the scope of the project.

## Insert Size

Another demanded characteristic the group chose for our QFD is how the size of the threaded insert affects every aspect of the assembly. It will be important to identify how smaller or larger threaded inserts behave and if failure is reached faster depending on the size of the threaded insert.

### Plating

The last demanded quality the team chose to include on the QFD is plating. Plating is where another material, generally another metal, is adhered to the threaded insert. The team wanted to examine if there is a direct correlation to the small but noticeable increase in thickness associated with plating. Plating involves added friction and the effects that this friction has on the process. However, the academic version of Abaqus does not allow the use of friction so this became outside the scope of our project.

## 8.3 Quality Characteristics Interactions

The interaction of all of the quality characteristics can be seen at the top of the QFD analysis. This portion is considered the roof of the house and each characteristic has its own diagonal column that collides with each other characteristic. The square box that is shared with two characteristics shows the correlation that they share between each other. The symbols that are used represent how strong the correlation is and whether it is a positive or negative one. The symbol to represent a positive correlation is the + sign and a strong correlation is represented with a ++ sign. A negative correlation is represented with a - sign and a strong negative correlation is represented with an upside down triangle. The only characteristics that had a strong positive correlation was vibration and load distribution. These two characteristics have a strong correlation because with a high amount of vibration on the bolt and insert the amount of load distribution across the insert must be high as well. Vibration also had a positive correlation with the performance of the insert at extreme temperatures. Vibration had a negative correlation with the friction and the plating of the insert because these two characteristics allowed for less empty space between the bolt and the insert as well as between the insert and the base material. Another quality characteristic that shared correlations with other characteristics was friction. Friction had a negative correlation with not only vibration but also the lubrication and the torque characteristics. Friction also had a positive correlation with load distribution and the performance of the insert at high temperatures. The size of the insert was another characteristic that had correlations with other characteristics which include a positive correlation with the amount of torque the insert can handle as well as a positive correlation with the load distribution across the insert.

## 8.4 Relationship Matrix

At the center of the house is the relationship matrix that shows how the demanded qualities interact with the quality characteristics. The relationship between the two could be characterized as having a strong relationship, a moderate relationship, a weak relationship, or no relationship if the square is left blank. The two demanded qualities that were found to have the most weight in the matrix was the pull out resistance and the prevailing torque. The only quality characteristic that both of these demanded qualities did not share a relationship with was the size of the insert. It is no surprise that these two demanded qualities are seen as the most important in Groovy Pins Abaqus models because they are the main reason for the insert to fail.

## 8.5 Difficulty and Relative Importance

As seen in the bottom section of Figure 112, difficulties were assigned to each quality characteristic and weight / importance was calculated. Each characteristic was assigned on a scale from 0, easy to accomplish to 10, extremely difficult. Vibration was given a rating of 5 because through research done by the team, it was found that vibration was possible to model in Abaqus but required extensive research. Performance at extreme temperatures was given 7 because it was also found to be possible to be done in Abaqus but required additional research and presented a learning curve to be overcome. Materials was given a 0 because materials can easily be changed in Abaqus by simply editing material properties. Load distribution was also determined to be easy and was given a 2 because Abaqus presents various options for defining load which can be specified in the load module. Lubrication was given a higher rating of 9 because the team does not know whether or not simulating lubrication is possible in Abaqus and documentation regarding lubrication in Abaqus is extremely limited. Friction interactions were given a 5 because although Abaqus allows specifying friction conditions, the means to do so are complicated and will require research and clarification. Torque is a loading condition that can be specified in Abaqus and was thus given a 3 to indicate it was easy and feasible. The insert size parameter was also deemed to be easy to change because it would require just a simple adjustment in the Solidworks file and was assigned a rating of 2. Lastly, adding plating to the models was deemed to be extremely difficult as the plating is very thin and it is not known how plating would be distinguished from the part in Abaqus or if simulating plating interactions would be accurate or feasible. Thus, plating was given a difficulty rating of 9.

By incorporating the interactions between qualities and characteristics, a relative weight or relative importance was calculated for each characteristic. The characteristic that had the most importance was materials, with a relative weight of 21.2. This proved to be an accurate analysis of importance because in actuality when creating the model, it was found that results from running simulations had the most dramatic differences when material properties were modified.

## 8.6 Competitive Analysis

Competitive analysis in the QFD involved comparing different simulations and iterations against each other. As explained in the Evaluation of Competition section, the models that were evaluated against each other included a 2D model, 3D pullout model, a simplified 3D pullout model, a hypothetical 3D prevailing torque model, and ideal models. An ideal models section was created and were given fives in each demanded quality to symbolize a hypothetical scenario in which all the required models were created and addressed each quality perfectly. To contrast with the ideal models, other iterations were compared. The 2D iteration was deemed to be not accurate in modeling pullout resistance and prevailing torque because the physics required to model those aspects aren't present in a 2D scenario. It did however match the other demanded qualities because of the simplicity involved in creating a 2D model. The first 3D pullout model was not dynamic and did not yield results so it did not meet the first three demanded qualities. It was however, modular in that parts could be swapped out easily. The 3D simplified model performed slightly better as it was able to yield results. A hypothetical 3D prevailing torque simulation was also compared and rated to be the best in each demanded quality but did not rank in the pullout resistance category as pullout resistance is not a goal of the simulation.

# 9 Design for X

## 9.1 Repeatability

One design for excellence that Groovy-Pin would like to incorporate in their Abaqus model is repeatability. Groovy-Pin would like to create a simulation that can run more then once with the same output. This means that the model must have reliable results and a precise output that does not change when the simulation is repeated with different mesh sizes. It is important for Groovy-Pin to have accurate results, and a repeatable simulation is a must for this goal to be accomplished. Groovy-Pin was able to create repeatable experiments by running the test multiple times at different mesh sizes in Abaqus to ensure the results were accurate.

## 9.2 Reliability

In order to design for reliability, the team designed the simulations for accuracy in FEA results. This involved ensuring the models for parts and materials were accurate and are in agreement with a real-life scenario in which threaded inserts are subjected to different conditions. At a more detailed level, in Abaqus, it is necessary to ensure the meshing is as fine as reasonably achievable. This involves using appropriate element controls and types as well as making element sizes small to get finer meshes which yield better results. A mesh convergence study will be performed on final iterations of the simulations in order to ensure the results are accurate and agree with each other. The team compared results from simulations to known data for the materials used. For example in the pull-out resistance test, it was found that the force required to deform the steel bolt was comparable to the tensile strength known for the steel.

## 9.3 Modularity

In order to design for excellence for the project, modularity is a key component to the success of the design. Not only does Groov-Pin want to have digital FEA experiments for pull-out resistance, prevailing torque, and deformation of the threaded insert, they want these FEA experiments to be modular. Modularity, in our case, is where any 3D model of a threaded insert and its corresponding bolt, nut, and base materials to be fastened, may be inserted into the designed experiments without much effort. The purpose of this is to have a user friendly environment where any engineer may utilize these experiments not matter their level of expertise in FEA.

With the use of a computer-aided design program, such as Solidworks, we were able to create modular assemblies. The three-dimensional models that we created, particularly the deformation model, can be effortless adjusted to fit Groov-Pin's needs. Manuals were created for the deformation and the pull-out resistance experiments. These manuals include a section on how to easily change the threaded insert to accommodate different sizes and simply import the updated model into Abaqus.

# 10 Project Specific Details & Analysis

## 10.1 Market Analysis:

The method that Groov-Pin and most other companies that manufacture threaded inserts use to test for prevailing torque is a fifteen cycle test using a torque wrench to record the minimum and maximum torque. This fifteen cycle test is currently the only test that Groov-Pin uses to tests the prevailing torque of their inserts and there is currently no other option that is more effective. The test that Groov-Pin uses to find the pull-out resistance of their inserts is also a physical evaluation that involves a tensile test machine to apply a force on the fastener. Groov-Pin takes great pride in the strength of their inserts and their locking capabilities. The torque test is essential in successfully validating the inserts durability. An abaqus model that can simulate this type of test would be an asset to Groov-Pin because it will be digital rather than physical so it will cut down on physical labor. The goal of the Abaqus model of both a prevailing torque test and a pullout force test is also to cut back on testing time and create more accurate results for each test. An Abaqus model would also allow Groov-Pin to analyze the wear, as well as the stresses and strains of the bolt, threaded insert and base material undergoing a given force or torque.

The third concept of Groovy Pins project is to create an Abaqus model that will simulate the cutting away of a base material with a self tapped insert. This model has not been created yet and would allow Groov-Pin to get a good understanding of where the material that is being cut away is going. Another goal of Groovy-Pin is to allow the model to be interchangeable and user friendly so every insert can be tested without creating a new Abaqus simulation and starting form scratch.

## 10.2 Demand Forecasting

The demand for an application like this is high because it allows the company to test the threaded inserts qualities that are most important to any customer that is interested in purchasing a high-grade insert. Groov-Pin has discussed with Groovy-Pin the importance of prevailing torque and pull-out force and how these two factors have been the two major reasons for threaded inserts to fail. The demand to improve these two factors and minimize the failure rate of threaded inserts is high. With this being known, it is assumed that any information from the results of the Abaqus models regarding these two factors will be a great asset to both Groov-Pin and the threaded insert industry.

## 10.3 Cost versus Price Information

Team Groovy-Pin is using the computers at the University of Rhode Island Engineering Computer Center to work on Abaqus. The Abaqus software that is in the computer lab is free for students to work on. Because Team Groovy-Pin is only creating 3-D digital models there are no manufacturing costs.

## 10.4 Surveys of Potential Users

Groov-Pin would like the group Groovy-Pin to publish an article on a threaded insert journal to contribute to the threaded insert industry. Other companies that manufacture threaded inserts will find this useful if there are any new findings on prevailing torque and pull-out force that will create a stronger insert. Potential users for a user friendly Abaqus model include Groov-Pin and customers that are interested in buying the threaded inserts so they can get a better idea of the inserts strength.

# 11 Detailed Product Design

Once the team had developed a total of ninety design concepts and submitted them to Groov-Pin, we decided to pursue three separate concepts for the remainder of our project. These concepts, as previously mentioned, will now be explained in greater detail.

## 11.1 Design Concept 1: Simulation of Pull-Out Resistance

Pull-out resistance is an important factor to consider when measuring the strength of a threaded insert and all inserts are put through a physical pull-out test. Groov-Pin asked the team to simulate the physical test in Abaqus. Pull-out resistance is the amount of force required to pull the insert free of the base material. The physical test is set up with the insert sitting in the base material and the fastener screwed into the insert. Next an upward force is applied to the fastener until the insert gives away from the base material. Te base material must be clamped down and is can range from wood, plastic, or softer metals. Groov-Pin wanted the team to use 60-61 Aluminum and the threaded insert was made of 303 stainless steel. The pull-out test is primary concerned with the outer threads of the insert and the base material.

To simulate this test Groovy-Pin created a simple model of a 303 stainless steel bolt that was sitting in a cylindrical base material. The bolt was acting as the threaded insert in this model and its threads were acting as the outside threads of the threaded insert. The reason Groovy-Pin decided to not include the insert in the model was to try to first get results with a simpler model and show that the base material was deforming. A concentrated force was applied on the top of the bolt to force the bolt upward and the base material was given boundary conditions on the side of the cylinder to keep it in place. The team was successful in showing that the base material deformed and that the stresses were along the threads, however the aluminum was sheared past failure and the bolt had one up three thread pitches. Groovy-Pin attempted to decrease the force on the bolt and decrease the time step but the results were still showing the threads of the base material completely shearing off. The team changed the boundary conditions of the bolt to only move upward in the y-direction one thread pitch. This resulted in an accurate simulation of the pull-out test and showed the base material deforming before its threads failed.

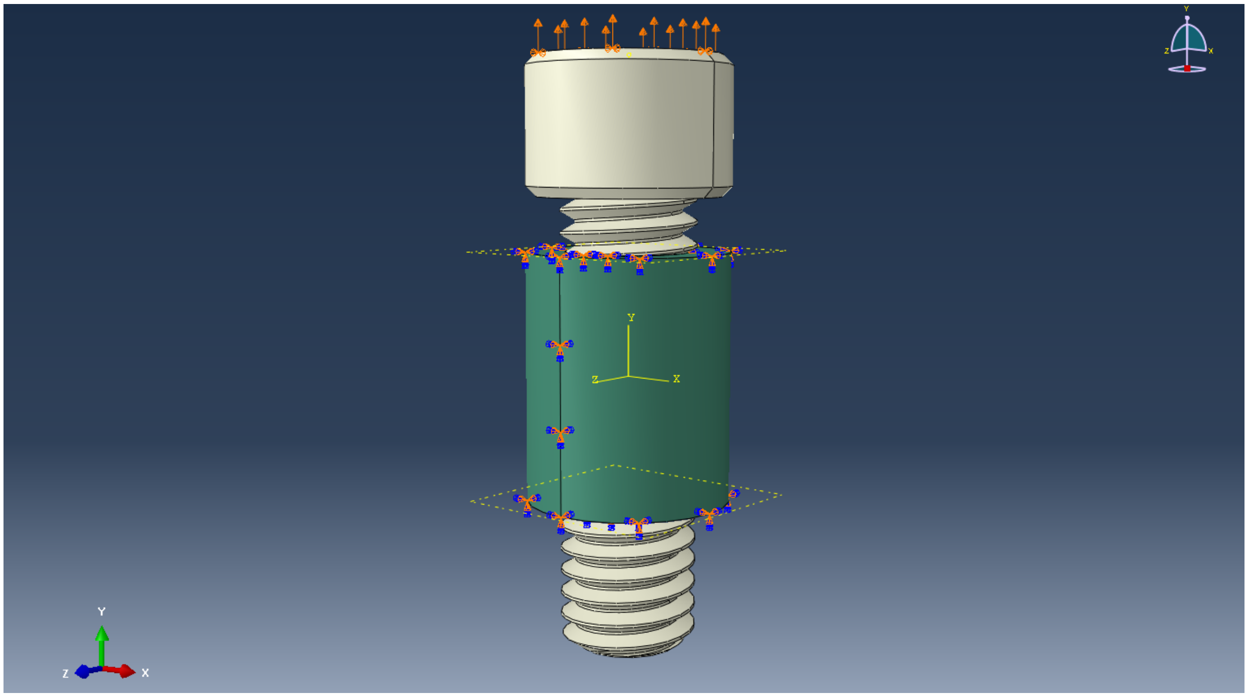


Figure 5: Setup for the pull-out test.

The next pull-out model that Groovy-Pin worked on included a base material in the form of a square plate and a threaded insert siting in the base material. The base material was set to have boundary conditions so it would not move and the insert had an applied pressure force on the top of the insert to pull it out of the base material. The base material was partitioned so that the mesh closer to the insert was finer so that the number of elements in the model was below the maximum number of nodes. This simulation however gave us many errors due to contact surfaces and the mesh was not fine enough. By reducing the area of the square Groovy-Pin was able to run a simulation but the results did not accurately represent a pull-out test because the base material did not deform enough to analyze the pull-out resistance.

## 11.2 Design Concept 2: Deformation of a Threaded Insert

Designing a digital experiment of a threaded insert was pivotal to Groov-Pin as they perform this on a number of different threaded inserts that they offer. For their process, the take a threaded insert, place a mandrel inside of the threaded insert and use two separate punching tools to squeeze the threaded insert inward at the four different points that the punch tools make contact. The mandrel is generally 0.002 inches smaller than the minor diameter of the threaded insert. This process deforms the threaded insert just enough to permit a tighter fit once a screw is fastened through the threaded insert. Several steps were taken to design this process digitally.

Initially, a three-dimensional model of the threaded insert and the punch tools were designed on the Solidworks platform. The threaded insert was subsequently mated at the origin be using the x-, y-, and z-planes. This detail is important so that modular designs may be used by utilizing the replace components function in Solidworks. Once the threaded insert was mated to the origin, two of the punch tool models were mated at their appropriate places, ensuring that both tools were placed at equal distances from the origin. Once this three-dimensional assembly was created in Solidworks, the file was saved as a STEP file and opened in Abaqus for analysis.

Once the model was imported into Abaqus, the desired material properties were defined. It was vital to create two separate materials; one that was linear and one that was non-linear. The enabled the threaded insert to deform during the finite element analysis. For the linear material, three properties were defined: density, Young's Modulus and Poisson's Ratio. For the non-linear material, the three aforementioned properties were defined but in addition to those, the plasticity property was also defined. It required two rows of data. The first row defined the yield stress and the plastic strain at the yield point for steel. The second row defined the yield stress and plastic strain at a single point after the yield. This allows Abaqus to calculate the remaining data. Once these properties were defined, sections were added to each part accordingly.

The three parts were then added to the assembly. The beauty of using a STEP file is that when each part was added, they were arranged exactly like they were in Solidworks. The assembly is shown in figure 6. A static, general step was created ensuring that non-linear geometry was turned on and the automatic stabilization was also activated. All defaults were left for the dissipated energy fraction.

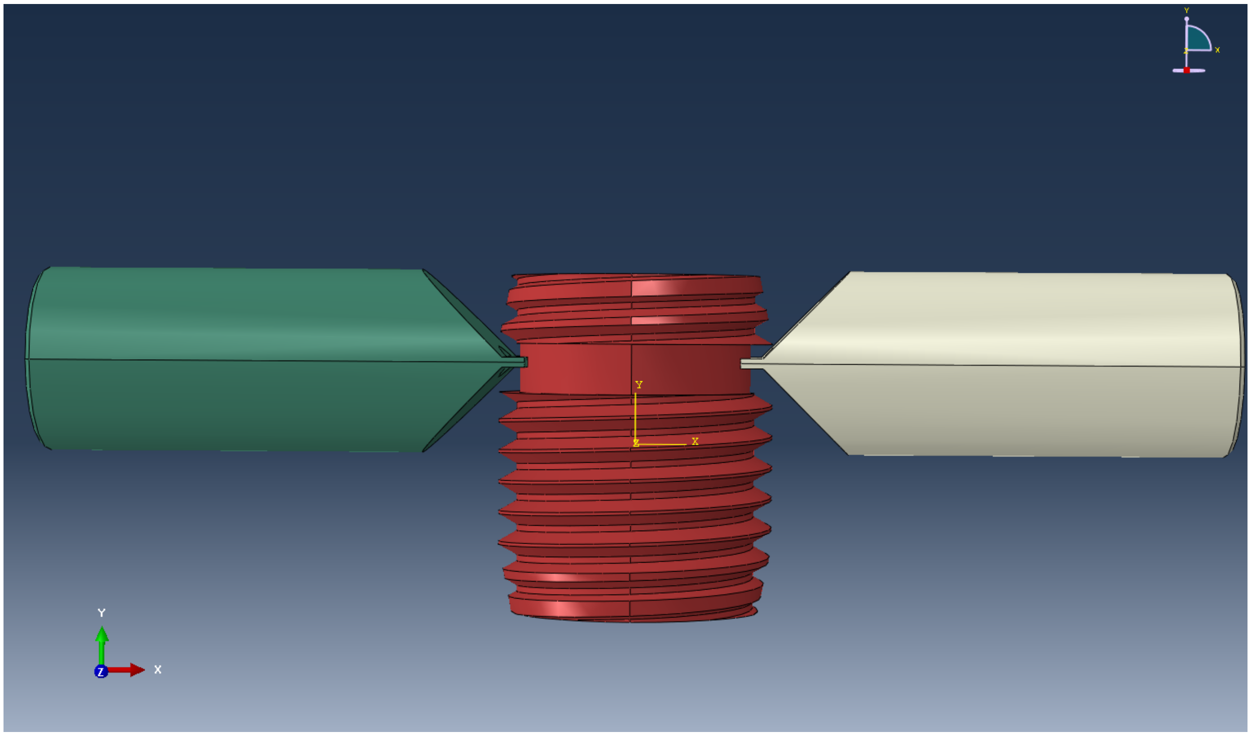


Figure 6: Setup for the deformation of insert simulation.

The contact surfaces were created, making sure that the threaded insert was always the master surface and the four faces at the end of the punch tool acted as the slave surfaces. These contact surfaces were added as surface-to-surface contacts and the interaction property was set to tangential behavior. The boundary conditions were subsequently set for each part. The top and the bottom face of the threaded insert were fixed in both the y- and z-directions. The rear face on each of the lock punches were also fixed in the y- and z-directions. The same face on the lock punch had a value of 0.3 and -0.3 in the x-direction accordingly. This allowed the lock punches to move toward the threaded insert and ultimately deform them.

Once the boundary conditions were set, each part was meshed appropriately. The element shape was set to "Tet." A job was created, leaving everything at default and the analysis was run.

## 11.3 Design Concept 3: Simulation of Prevailing Torque

After speaking with Groov-Pin, developing a digital experiment that can identify the amount of prevailing torque allowed until failure became our second design concept. This concept involves the use of FEA and more specifically, dynamic models to see exactly what is happening between the base material, threaded insert, bolt and nut. Prevailing torque is the amount of torque required to fasten a bolt into the threaded insert, or the threaded insert into the base material while trying to minimize the amount of resisting forces, i.e. vibrations.

After achieving a successful deformation simulation, the next phase of the project was to use the deformed insert for use in a prevailing torque test. This involves setting up an assembly consisting of a steel bolt and the deformed insert model from the previous simulation. This is possible through Abaqus by exporting the deformed model geometry available in the results file as a part. It is then necessary to import the deformed part into the new assembly for use in the simulation. The team discovered that in order to simulate conditions for a screwing motion was far more complex than expected.

The required interaction between a bolt and deformed insert involves a screwing motion which involves the combination of rotation with a specific angular velocity and displacement along an axis established by the insert itself. The team researched tutorials for the best way to achieve such complex interaction between parts. Through our research, it became apparent that the simulation would require a complex process of defining and using new functions that rely on each other consecutively. Defining a screwing motion involves the use of a series of functions such as creating wires, assigning connectors to the wires, creating fasteners, and defining fastener interaction properties. Not only is each step for the process complex in itself, but each step is reliant on the last in order to work. This adds more possible "points of failure" to the simulation which would require a considerable amount of more time for research and troubleshooting issues. It became apparent to the team that pursuing a simulation involving such a complex process would require several more weeks of work beyond the scope of the project.

Alternatively, the team created a prevailing torque simulation that represented the interaction as closely as possible without the use of wire or connector functions. The team expanded on the deformation simulation by adding a bolt which moves down into the insert as the insert is deformed. The model can be seen in figure 7. Although this simulation produced working results, the team decided that the results don't accurately enough represent the given problem because the bolt doesn't screw into the insert. A complete prevailing torque simulation would require several more weeks of research into new Abaqus functions as well as time required to troubleshoot issues with the simulation.

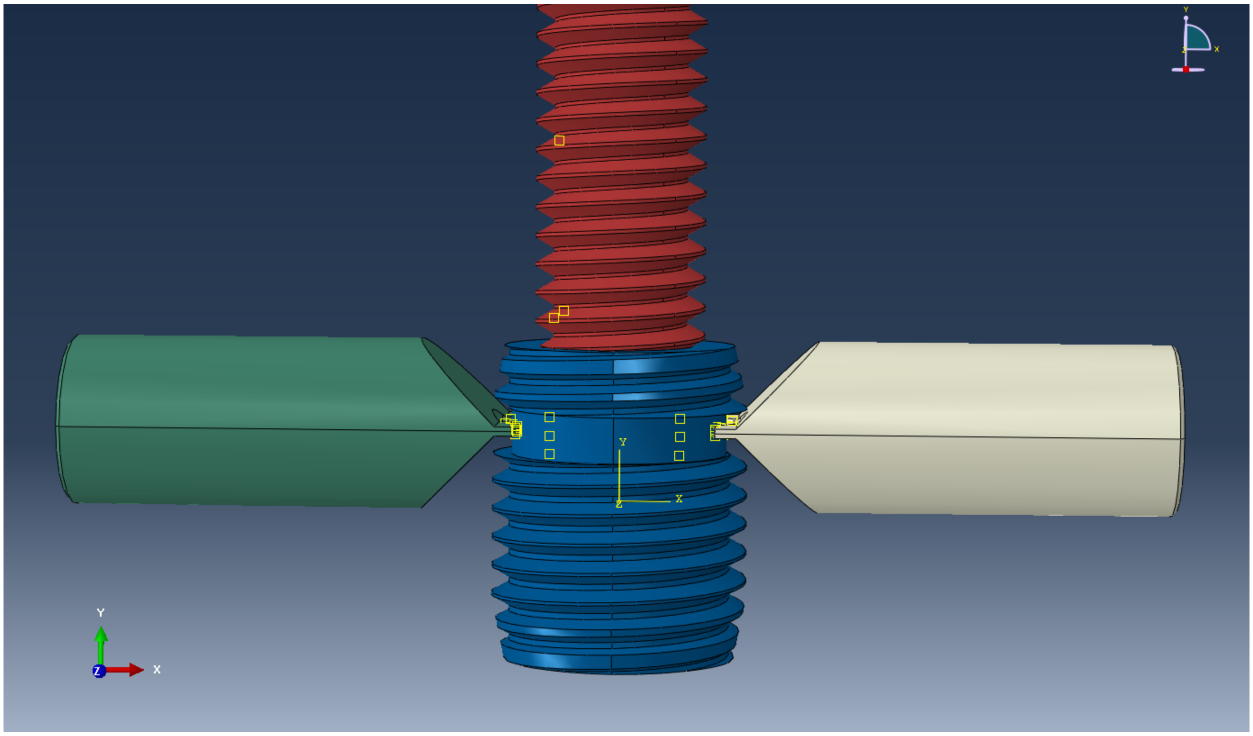


Figure 7: Setup of the alternative prevailing torque simulation.

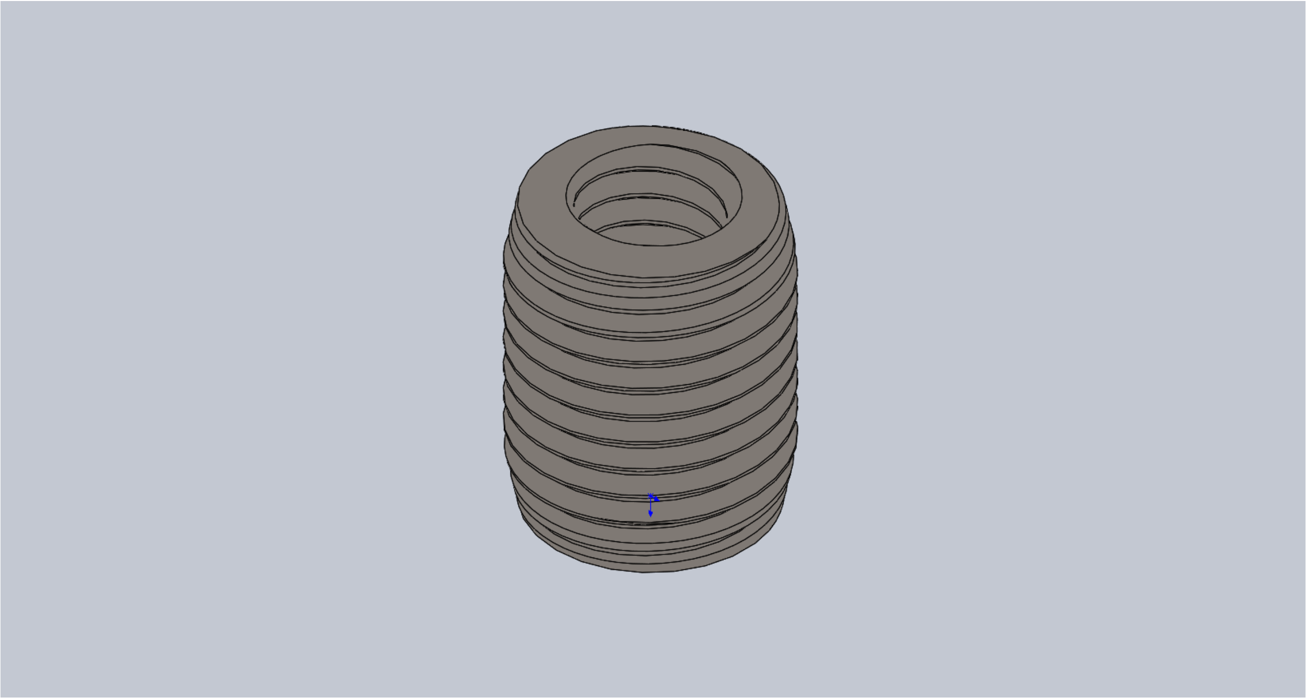


Figure 8: 3D Model of a Threaded Insert

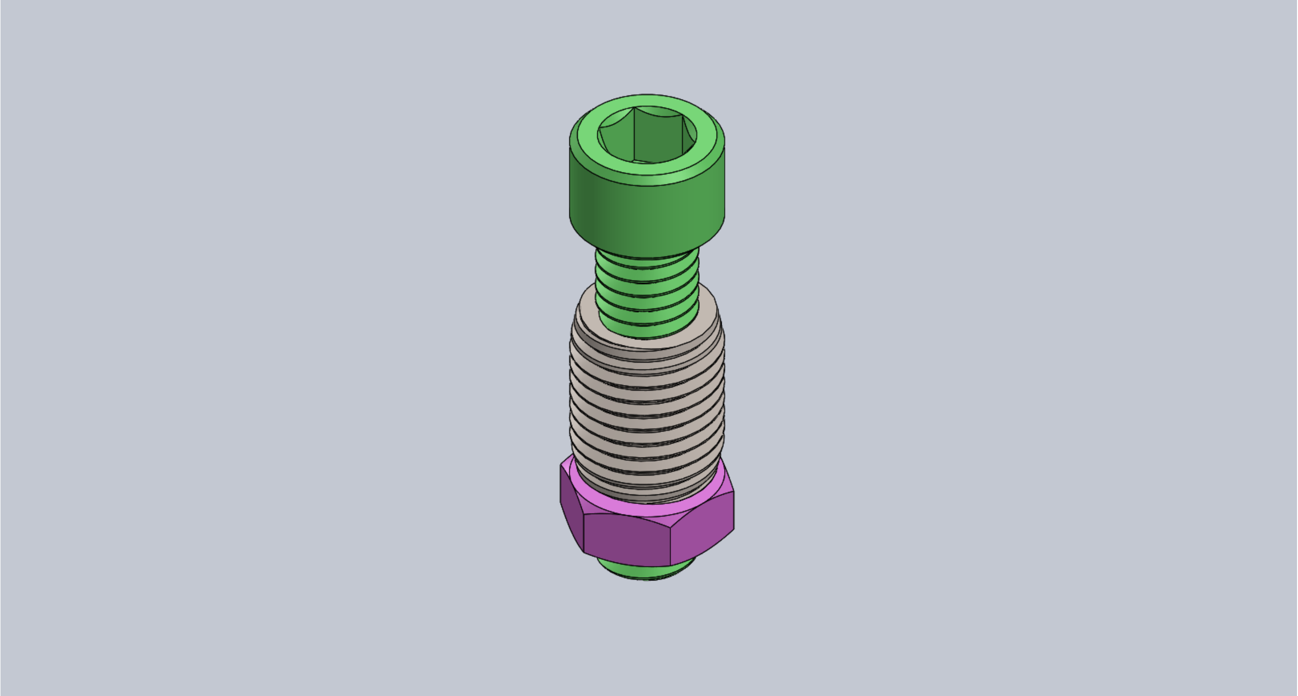


Figure 9: 3D Model of Screw, Threaded Insert, and Nut

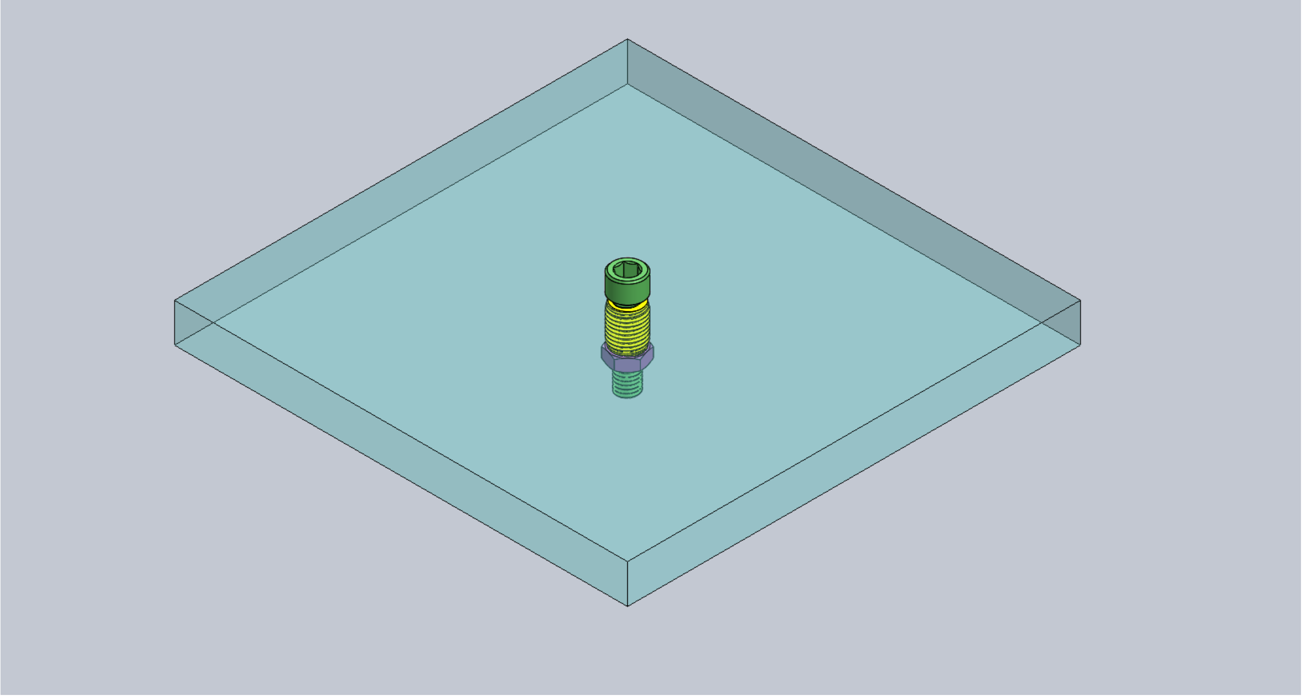


Figure 10: 3D Model of an Ideal Assembly

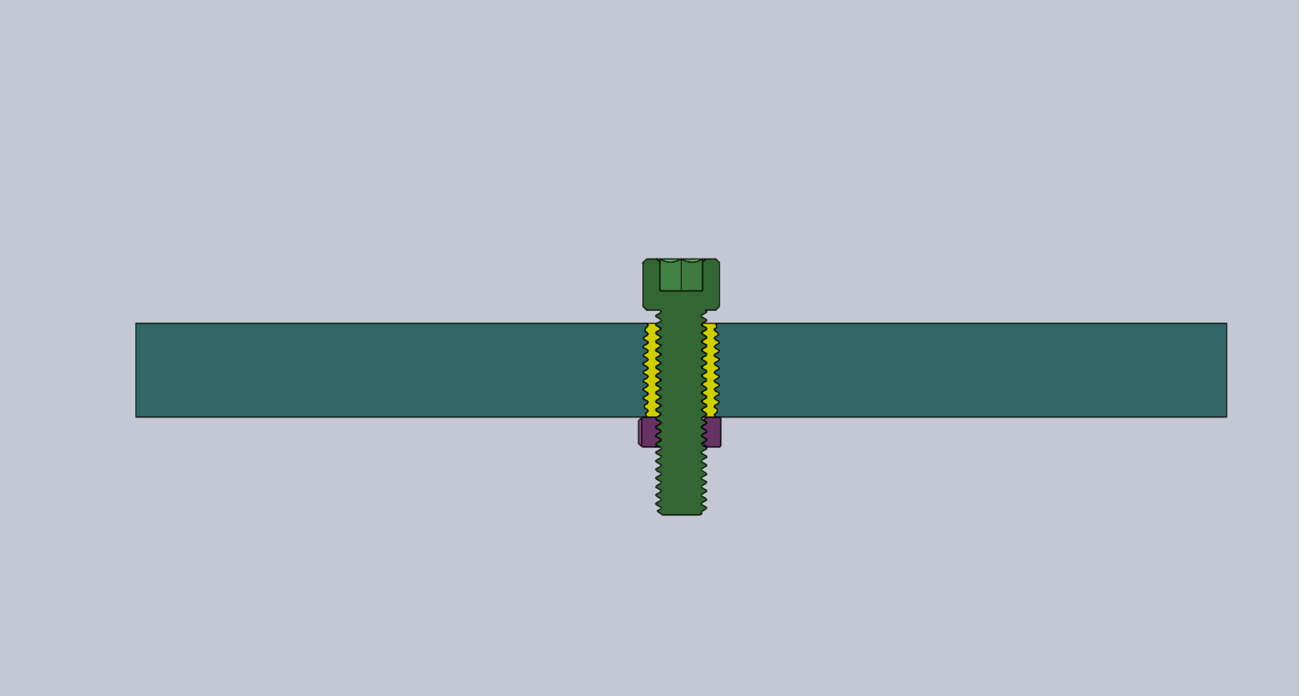


Figure 11: Section View of an Ideal Assembly

As you can see from Figures 8 to 11, these are the Solidworks models that we have been working with and will continue to work with in Abaqus for our digital experiments.

# 12 Engineering Analysis

Engineering analysis for this project includes the analysis and verification of results presented from running different iterations of simulations created for the Pullout Resistance and Deformation of Insert Models. The figures below show results in each stage of the project and will show that results become more understandable and demonstrates that the simulation behaves as expected given defined aspects of the model such as materials, load, and boundary conditions. Each result in the figures below have been the product of mesh convergence studies conducted in order to ensure the results are accurate and do not change even when element sizes are reduced.

## 12.1 Initial Results for Pullout Resistance

As seen in Figure 12, the 3D simplified model yielded the team's first set of results but the results achieved were not as expected. The results shown in the diagram shows the stresses located at seemingly random locations and neither the bolt nor insert deformed in a manner that was expected. The team suspected designed the simplified model for shorter run times by increasing mesh size may have lead to poor results. The team then decided to create another iteration of the model with finer meshes.

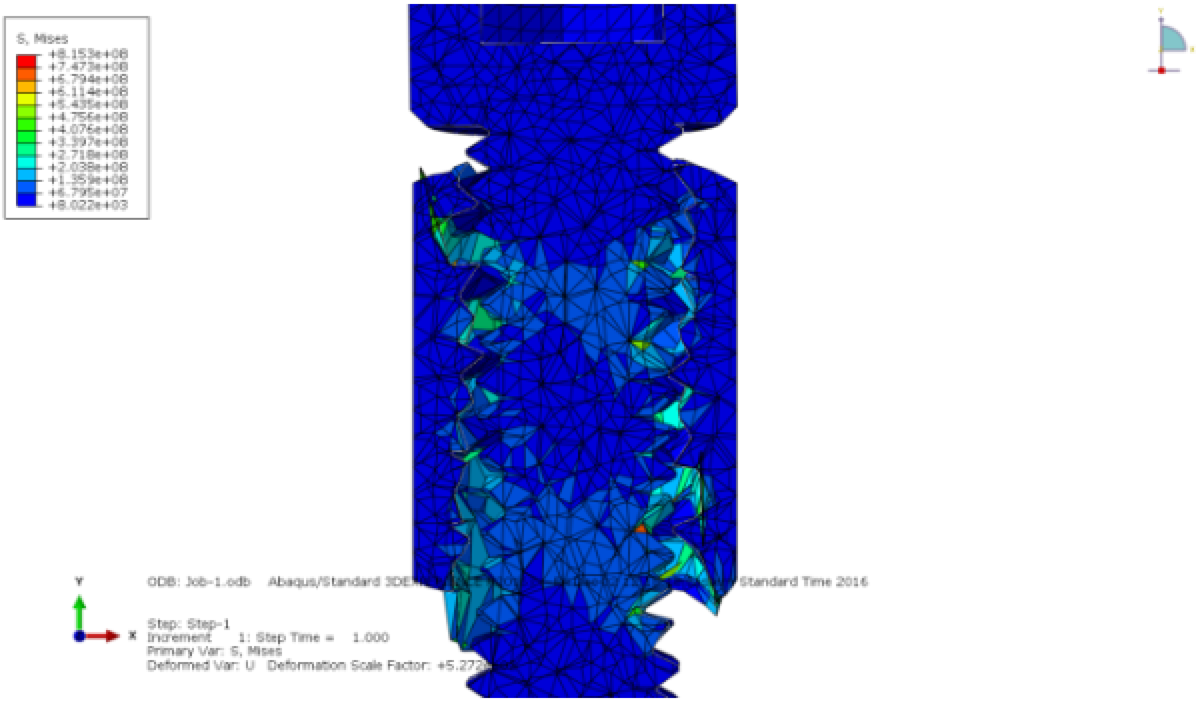


Figure 12: Initial Results

## 12.2 Secondary Results for Pullout Resistance

In the secondary model of the 3D simplified pullout resistance test, the meshes on both the bolt and insert were refined by more than 40% in order to yield more accurate results. The simulation run time was increased significantly for this model but as seen in Figure 13 the resulted achieved were much better. The results from the secondary model shows that stresses are concentrated on the edges of the bolt threads. The threads of the insert have also been completely destroyed which is expected because the aluminum is softer than the steel bolt.

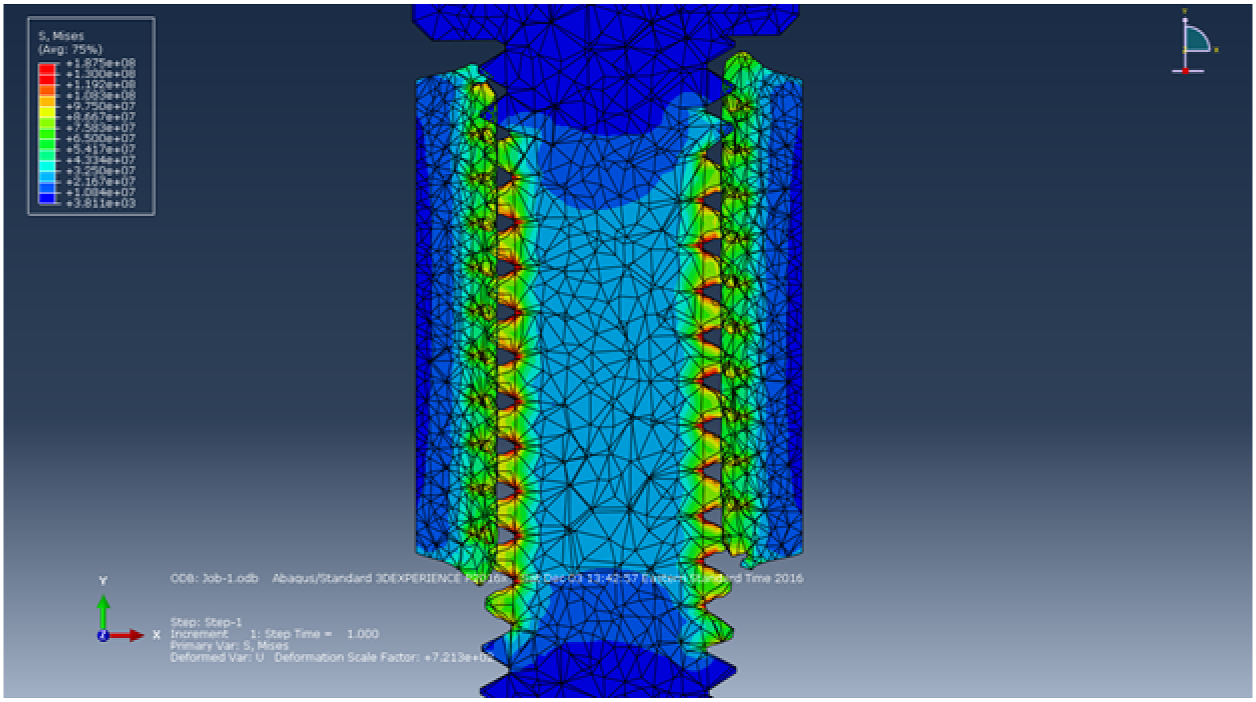


Figure 13: Secondary Results

## 12.3 Tertiary Results for Pullout Resistance

The team decided the next goal in the project was to work on the pullout resistance model further in order to get the bolt to deform or break. As discussed with Groov-Pin the bolt generally fails before the insert does. To achieve this, the boundary conditions were added to the bottom surface of the insert so that the insert could not deform downwards. The results shown in Figure 14 to Figure 16 show that the bolt has deformed and the greatest stresses are focused on the top 3 threads of the bolt which is in agreement with the known pullout resistance theory.

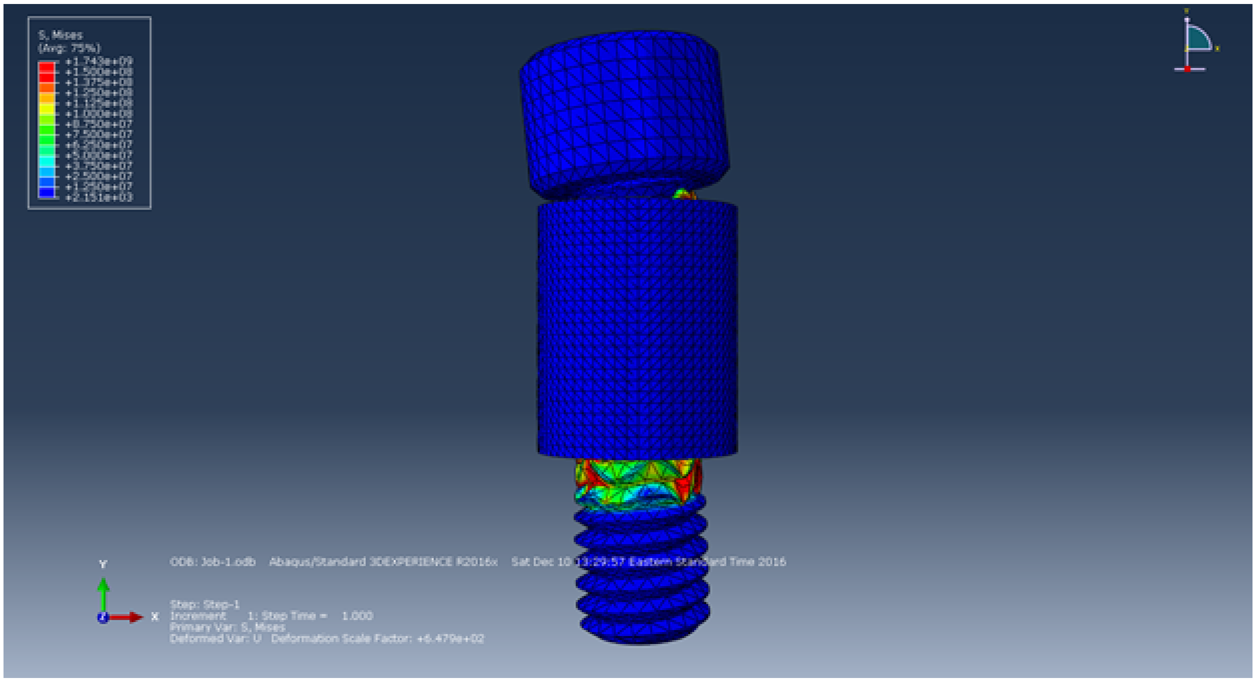


Figure 14: Tertiary Results: Full View

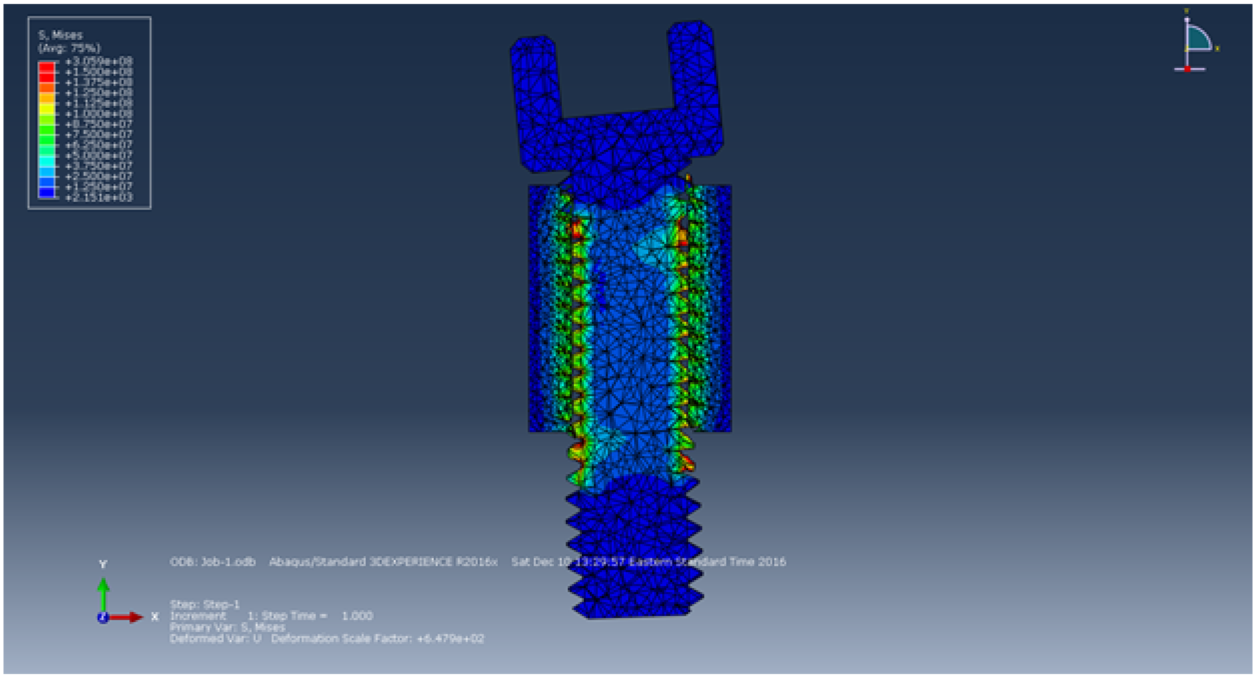


Figure 15: Tertiary Results: Section View

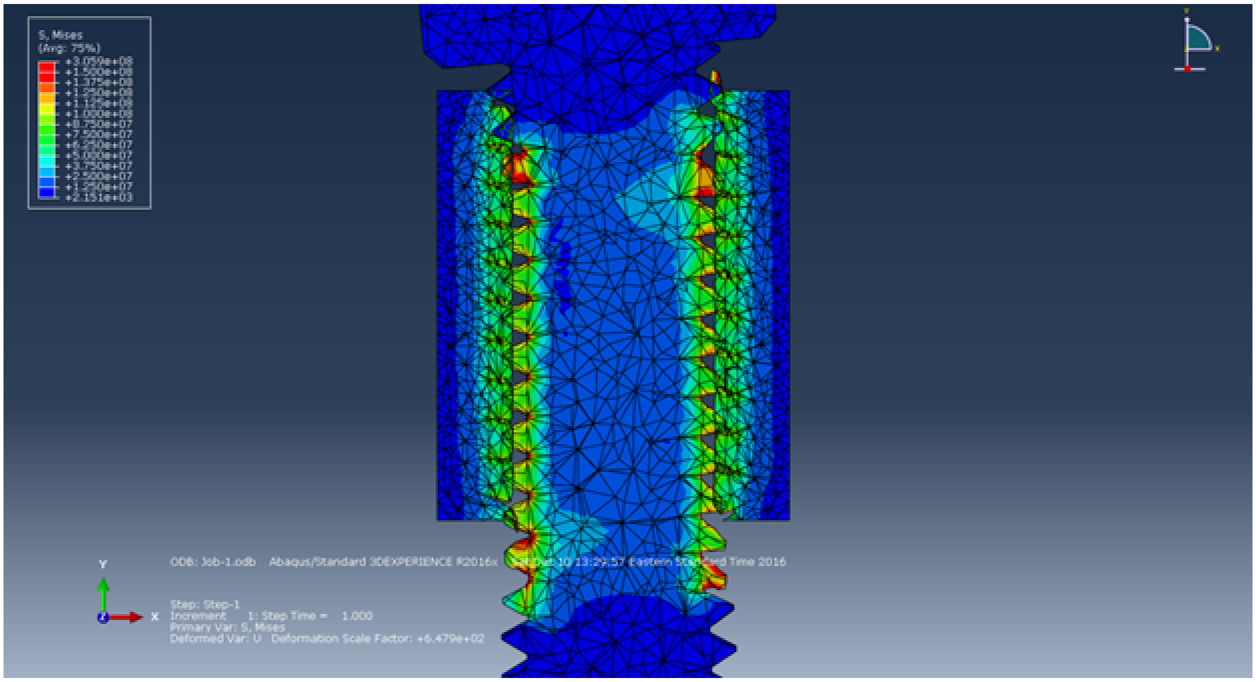


Figure 16: Tertiary Results: Section View 2

## 12.4 Final Results for Pullout Resistance

After presenting the tertiary results to Groov-Pin, it was requested that the team create simulations for pullout resistance that show the earliest steps of the interaction in which the displacement shown is limited to just the first couple threads of the models. The team redesigned the simulation to limit displacement and show more explicitly the steps of the simulation of pertinence. The results from running the final simulation can be seen in figure 17 and 18. The results for force required for deformation is comparable to known tensile strength data for the given materials.

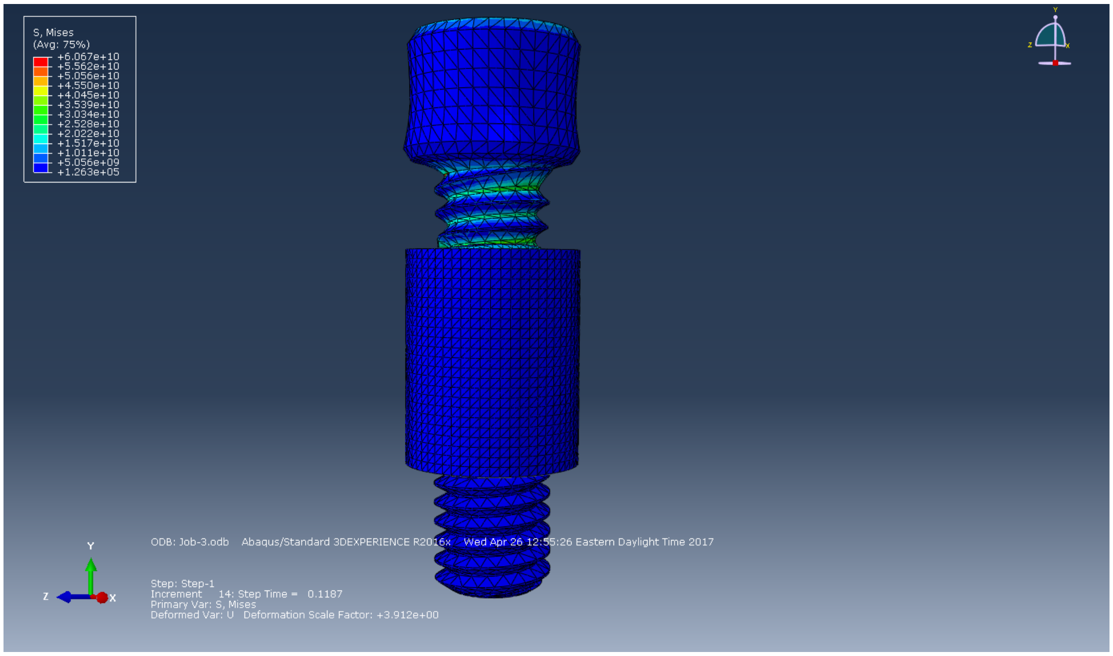


Figure 17: Final Results Full View

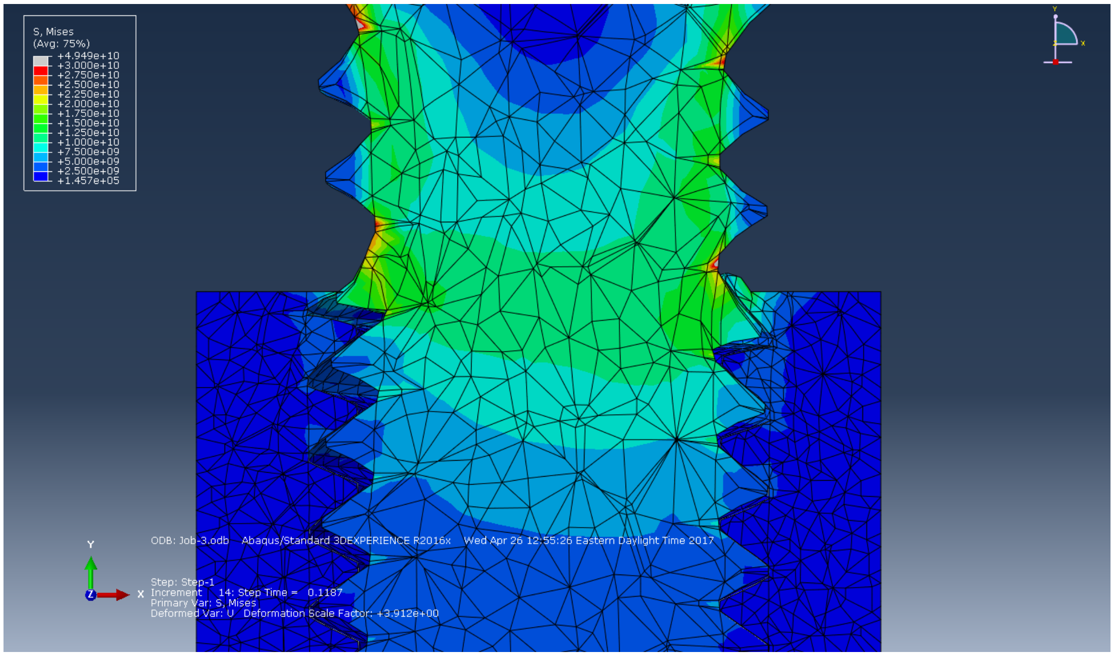


Figure 18: Tertiary Results: Section View Close Up

## 12.5 Results for Deformation of Insert

The team created a deformation of insert simulation following the procedures shown in the Detailed Product Design and Operations section of the report. Operations section of the report. The following results were produced as shown in figures 19 and 20.

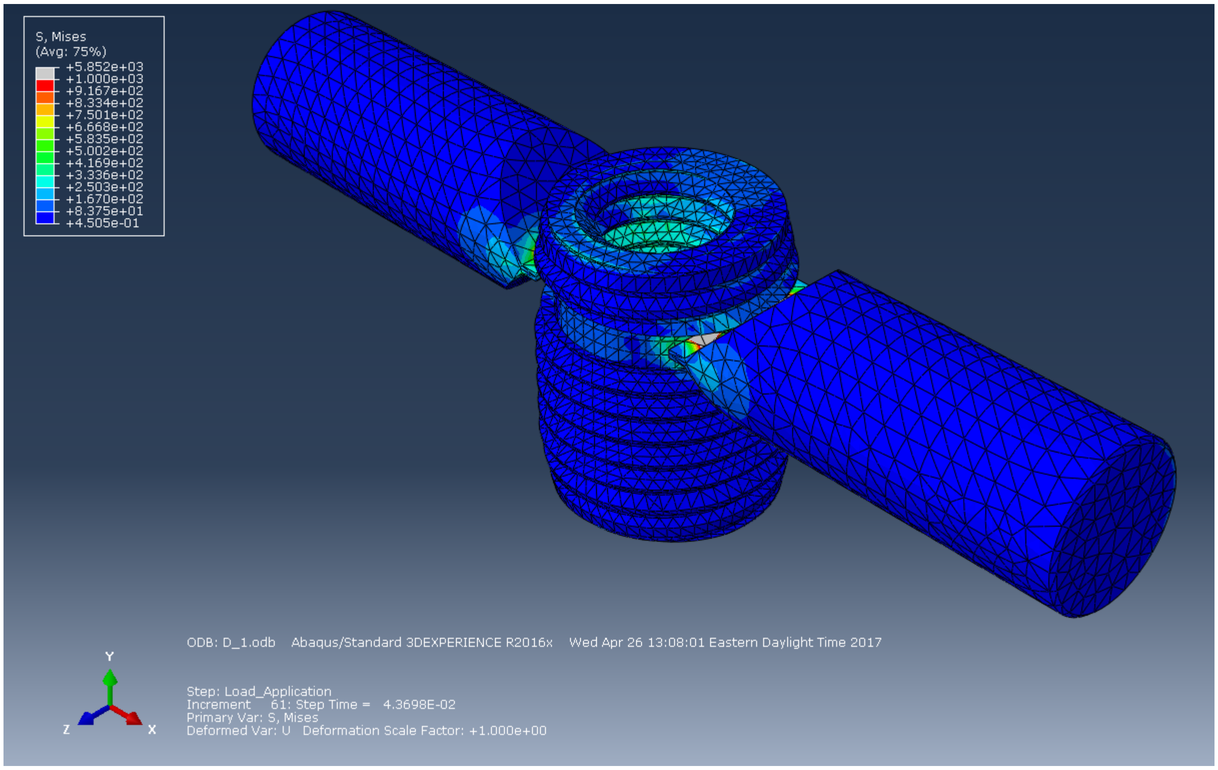


Figure 19: Results for the outer surface of the insert when deforming using punching tools.

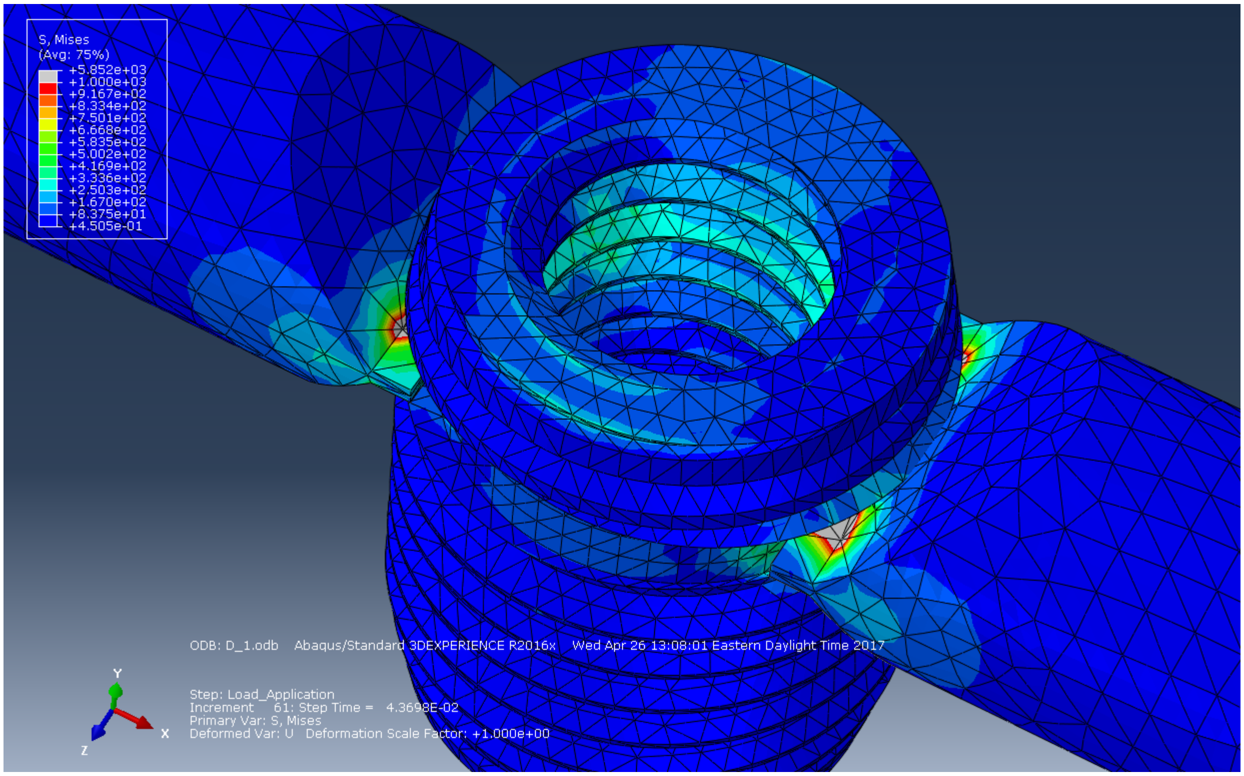


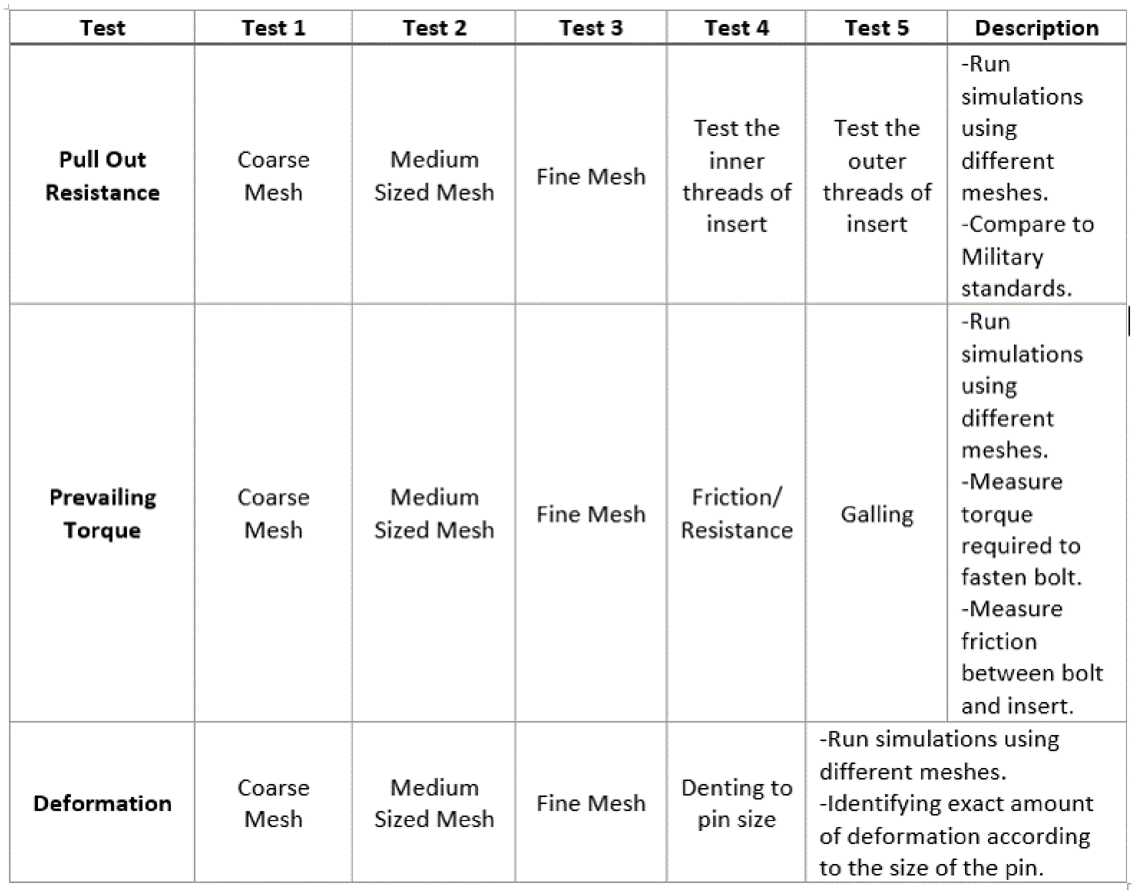
Figure 20: Results for the inner surface of the insert when deforming using punching tools.

## 13 Build / Manufacturability

Team Groovy-Pin only created Abaqus models during the coarse of the Capstone experience. Groov-Pin asked the team to tests the inserts that are currently being manufactured already and the team was not to adjust or redesign the threaded insert. The build process for team Groovy-Pin was strictly the digital Abaqus models that the team created and no physical tests were performed.

# 14 Testing

In the testing phase of the project, the team created a testing matrix in order to decide how to test each simulation. The test matrix can be seen in figure 21. As mentioned in the design for reliability and repeatability sections, the teams deemed it necessary to perform mesh convergence tests by using different meshes. After verifying convergence, the team compared the given results to research and known experimental data pertaining to each simulation.



| Test | Test 1 | Test 2 | Test 3 | Test 4 | Test 5 | Description |
|---|---|---|---|---|---|---|
| **Pull Out Resistance** | Coarse Mesh | Medium Sized Mesh | Fine Mesh | Test the inner threads of insert | Test the outer threads of insert | -Run simulations using different meshes.<br>-Compare to Military standards. |
| **Prevailing Torque** | Coarse Mesh | Medium Sized Mesh | Fine Mesh | Friction/ Resistance | Galling | -Run simulations using different meshes.<br>-Measure torque required to fasten bolt.<br>-Measure friction between bolt and insert. |
| **Deformation** | Coarse Mesh | Medium Sized Mesh | Fine Mesh | Denting to pin size | -Run simulations using different meshes.<br>-Identifying exact amount of deformation according to the size of the pin. | |

Figure 21: A test matrix created by the team to test each simulation.

# 15 Redesign

The redesign process for Groovy-Pin included a series of trial and error in the Abaqus program for different models. This included meshing the parts differently, changing the geometry of parts in Solidworks, creating partitions, applying different types of loads, and changing boundary conditions. The team ran into multiple types of errors in Abaqus that did not allow the team to run the model. Common errors that the team ran into had to do with boundary conditions and contact surfaces. During the pull-out test the team noticed that when the bolt was being pulled-out the insert was sliding down the bolt more then one thread pitch and the threads of the base material were completely distorted. The team was only concerned about the bolt being pulled out one thread pitch and had to change the boundary conditions on the bolt to only move upward one thread pitch. This gave the team a more accurate test of the pull-out and analyze the threads of the base material before failure. The denting simulation on the insert with the crimping tools was also redesigned to give accurate results. At first the team tried performing the test on the 3/8-16 insert with out any modifications. However the team was informed that the inserts that Groov-Pin dents has a small cutout section around the insert where the crimping tools come in contact. This enables the crimping tools to dent the insert at four points around the insert. The Solidworks model of the 3/8-16 insert was modified to take out a strip of threads along the insert and allow the crimping tool to work correctly. The team also initially attempted to apply a force on the crimping tools but revised this model with boundary conditions that forced the tools to dent the insert inward the specified distance.

# 16 Operation

One of the goals for this project is to design the simulations so that they are modular and can easily be recreated. The team decided that it would be appropriate to create manuals that will serve as tutorials for the creation of the pullout resistance simulation and the deformation simulation. Using the steps and procedures given by the following sections, materials and parts can easily be changed to meet the requirements of the user.

## 16.1 Pull-Out Resistance Manual

Step 1: Open Abaqus and select File¿Import¿Part. Locate the STEP file containing the assembly created in Solidworks and select OK.

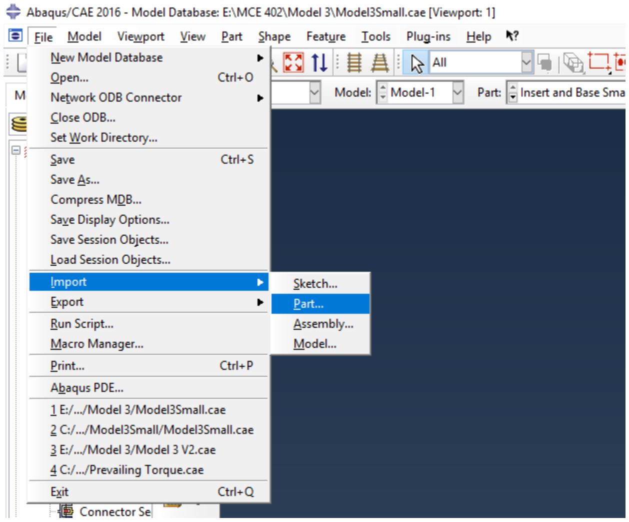


Figure 22: How to import model into Abaqus

Step 2: Once the model has been imported, rename each individual part by expanding Parts under the drop down menu, right click each part and select Rename.

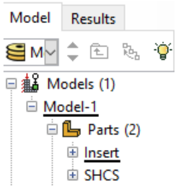
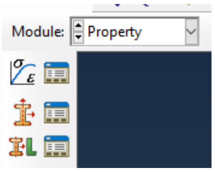


Figure 23: Part description

Step 3: Now it is time to define the desired materials and their corresponding properties. Select Property from the drop down module and then select the icon to create a material.



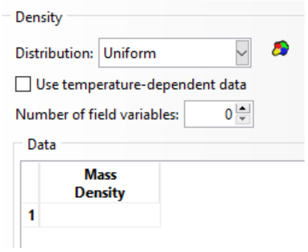
Module: Property

Figure 24: Property module

Step 4: It is important to create two separate materials in order to define which part will retain the plastic deformation properties. Locate the desired material properties, preferably ones derived experimentally and input them into Abaqus.

Material 1

Input the density, Youngs Modulus and Poissons Ratio. (Abaqus is a unitless program so it is important to note that the units need to be consistent in order to obtain accurate results).



Density

Distribution: Uniform

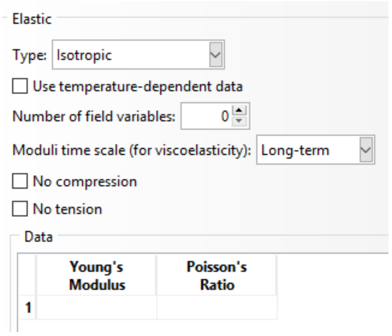
☐ Use temperature-dependent data

Number of field variables: 0

Data

| | Mass Density |
|---|---|
| 1 | |

Figure 25: Density property



Elastic

Type: Isotropic

☐ Use temperature-dependent data

Number of field variables: 0

Moduli time scale (for viscoelasticity): Long-term

☐ No compression

☐ No tension

Data

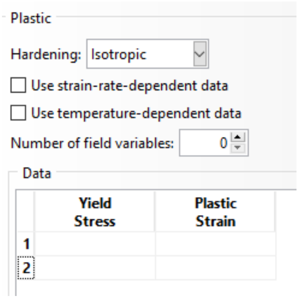
| | Young's Modulus | Poisson's Ratio |
|---|---|---|
| 1 | | |

Figure 26: Elasticity properties

This material will be used to define the Lock Punch parts. Save this property as the material name.

Material 2

To enable plastic deformation, the second material has an additional property that needs to be defined. Once the density and elastic properties have been defined, select Mechanical >Plasticity >Plastic. Two rows need to be defined, the first being the values determined experimentally or by a stress-strain curve at the yield point and the second being the values at a point after the initial yield point. This will allow Abaqus to determine the remaining yield points.



Plastic

Hardening: Isotropic

☐ Use strain-rate-dependent data

☐ Use temperature-dependent data

Number of field variables: 0

Data

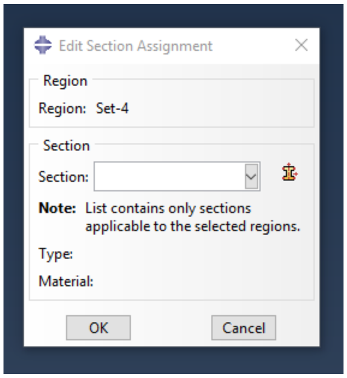
|  | Yield Stress | Plastic Strain |
|---|---|---|
| 1 |  |  |
| 2 |  |  |

Figure 27: Plasticity properties

Save this material as the material name. This will differentiate the two materials when a section is added to the individual parts.

Step 5: Create two sections, both homogeneous and give them the same name as the material.

Step 6: Once the sections have been created, select Assign Section and assign the sections to the corresponding parts.



Region: Set-4

Section:

**Note:** List contains only sections applicable to the selected regions.

Type:

Material:

Figure 28: Edit section assignment

Step 7: Under the module drop down list, select Assembly and then select the icon to create an instance.

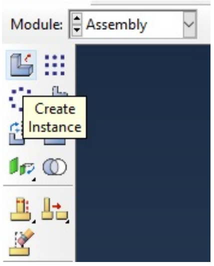
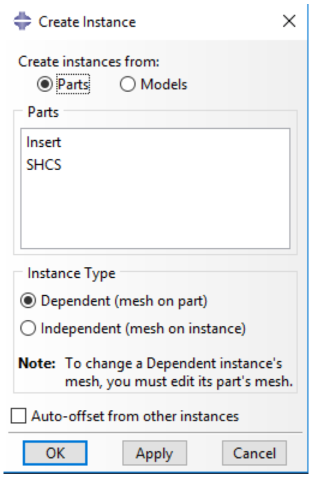


Figure 29: Assembly module

A dialog will appear with the two parts created by the STEP file. Select each listed part and then select Apply. The parts will appear in their appropriate place as they were created in Solidworks.


Create Instance

Create instances from:
● Parts ○ Models

Parts
Insert
SHCS

Instance Type
● Dependent (mesh on part)
○ Independent (mesh on instance)

**Note:** To change a Dependent instance's mesh, you must edit its part's mesh.

☐ Auto-offset from other instances

Figure 30: Create instance

After the two parts have been assigned to the assembly, click cancel to proceed.

In order to limit the areas of interaction, it is necessary to create partitions on the bolt. In the Part module, select the bolt and create a datum plane.

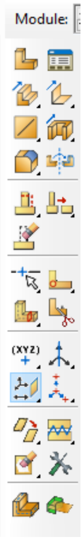


Figure 31: Create Datum Plane

Choose the XZ plane and create a datum plane by entering an offset distance from the center. The distance specified will depend on the size of the insert. Two datum planes need to be created for the top and bottom edge of the insert.

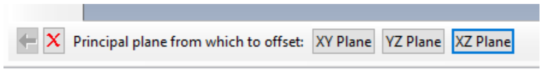


Figure 32: Choose XZ plane.

Create a partition by choosing the Partition Face by Datum Plane tool.

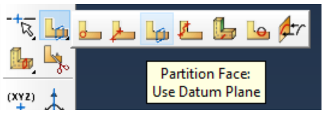


Figure 33: Choose the partition Face tool.

Select the surface to partition. Create a partition for each datum plane for a total of 2 partitions.

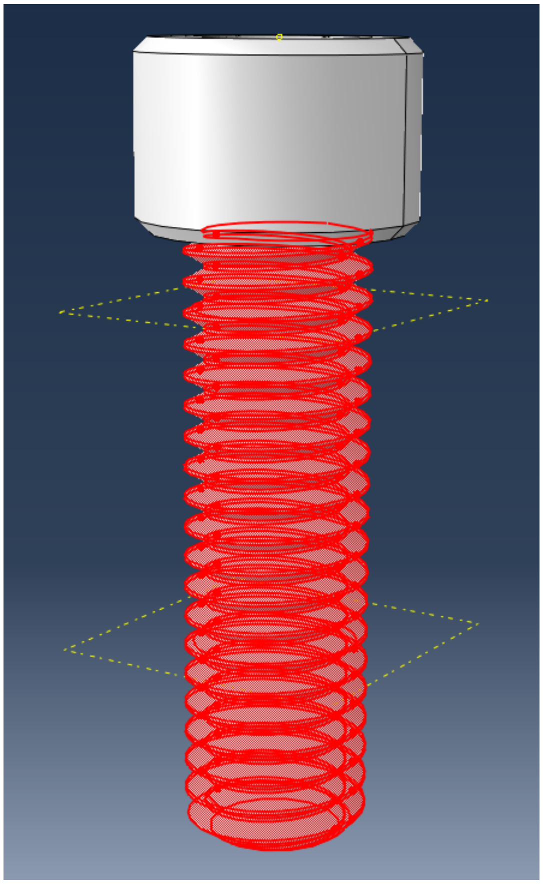


Figure 34: Choose the surface to partition.

Step 8: Create a step by selecting the icon shown.

Name the step something relevant, select Static, General and then Continue

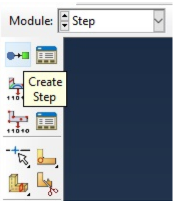
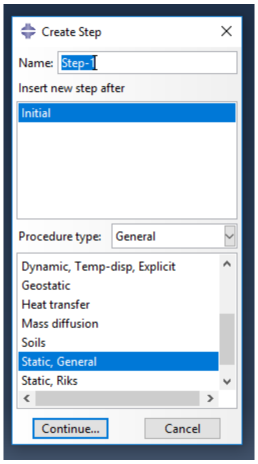


Figure 35: Step module



Name: Step-1

Insert new step after

Initial

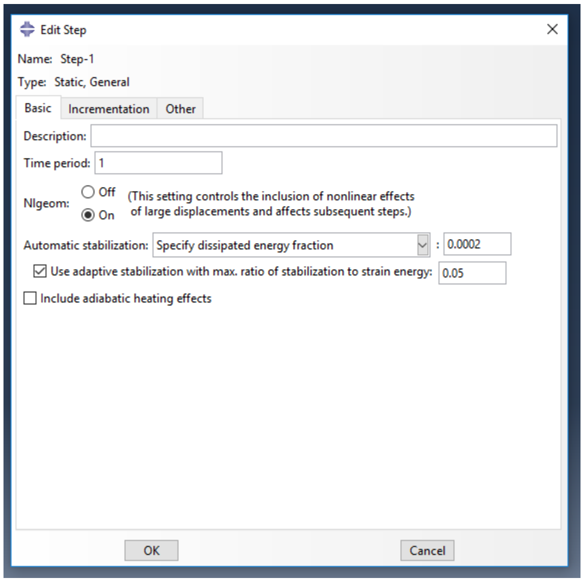
Procedure type: General

Dynamic, Temp-disp, Explicit
Geostatic
Heat transfer
Mass diffusion
Soils
Static, General
Static, Riks

Figure 36: Create a step

A dialog will appear with some options. Ensure that Nlgeom and Specify dissipated energy fraction under Automatic Stabilization are selected. Then go to Incrementation and change the initial and maximum increment size appropriately.

Step 9: Now it is time to create contact between the threaded insert and bolt. Begin by selecting Interaction under the module and then Create Interaction.



Edit Step

Name: Step-1

Type: Static, General

Basic | Incrementation | Other

Description:

Time period: 1

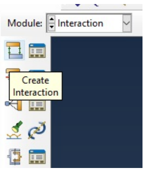
Nlgeom: ○ Off ● On (This setting controls the inclusion of nonlinear effects of large displacements and affects subsequent steps.)

Automatic stabilization: Specify dissipated energy fraction : 0.0002

☑ Use adaptive stabilization with max. ratio of stabilization to strain energy: 0.05

☐ Include adiabatic heating effects

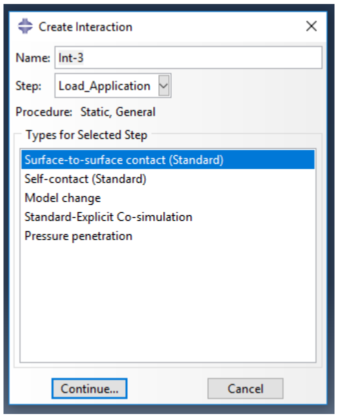
Figure 37: Edit step



Module: Interaction

Figure 38: Interaction module

Once the dialog appears, select Surface-to-surface contact (standard) and then Continue



Create Interaction

Name: Int-3

Step: Load_Application

Procedure: Static, General

Types for Selected Step

- Surface-to-surface contact (Standard)
- Self-contact (Standard)
- Model change
- Standard-Explicit Co-simulation
- Pressure penetration

Figure 39: Create an interaction

In order to properly select surfaces, it is necessary to use the assembly display options to hide parts. In order to choose which parts to hide, go to View and select Assembly Display Options. In the Instance tab, check or uncheck the check boxes to control which parts are visible.

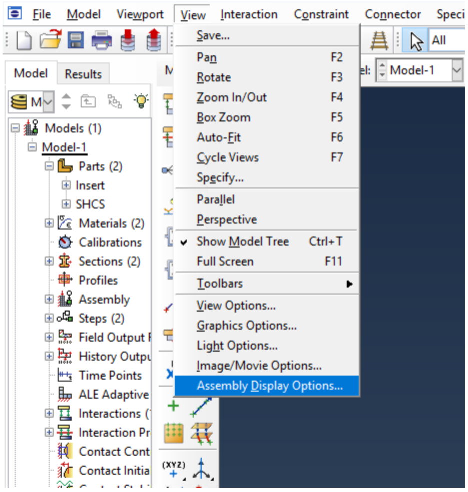


Figure 40: Assembly Display Options

Abaqus will ask for a Master surface. Use the view cut tool in conjuction with the assembly display options in order to select the inner surface of the insert.

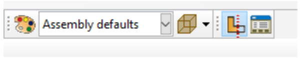
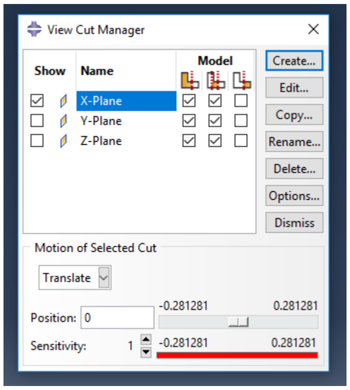


Figure 41: Viewcut Tool



| Show | Name | Model | | |
|---|---|---|---|---|
| ☑ | X-Plane | ☑ | ☑ | ☐ |
| ☐ | Y-Plane | ☑ | ☑ | ☐ |
| ☐ | Z-Plane | ☑ | ☑ | ☐ |

Motion of Selected Cut

Translate

Position: 0 -0.281281 0.281281

Sensitivity: 1 -0.281281 0.281281

Figure 42: Viewcut Manager

Be sure to select the inner surfaces and rotate to ensure the entire inner surface is selected as the Master surface.

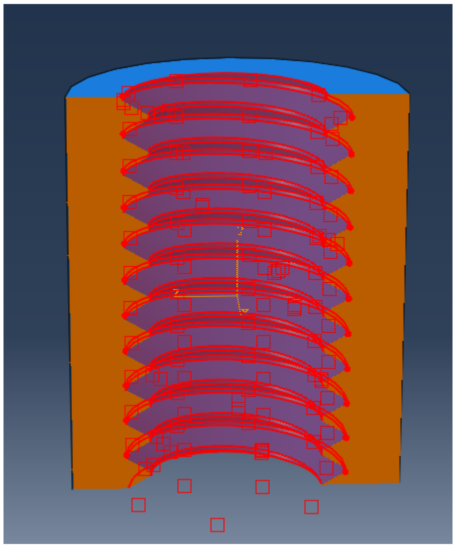


Figure 43: Select the master surface.

Then, Abaqus will ask for a slave surface. Select the exterior face of the bolt as the Slave Surface. Press Done. Abaqus will ask for the slave type, select Surface and then select the previously partitioned surface on the bolt.

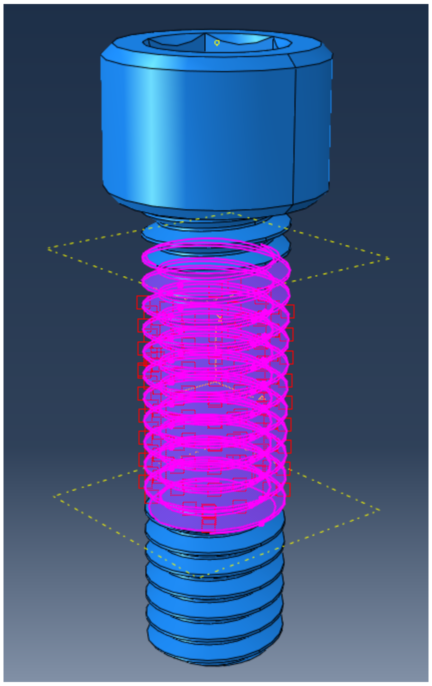
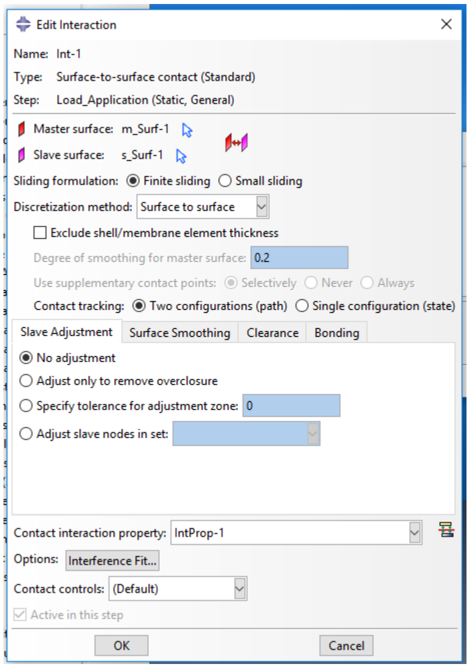


Figure 44: Choose the slave surface.

Once the dialog appears, find the icon to create an interaction property and select it.



Edit Interaction

Name: Int-1

Type: Surface-to-surface contact (Standard)

Step: Load_Application (Static, General)

Master surface: m_Surf-1

Slave surface: s_Surf-1

Sliding formulation: ● Finite sliding ○ Small sliding

Discretization method: Surface to surface

☐ Exclude shell/membrane element thickness

Degree of smoothing for master surface: 0.2

Use supplementary contact points: ● Selectively ○ Never ○ Always

Contact tracking: ● Two configurations (path) ○ Single configuration (state)

● No adjustment

○ Adjust only to remove overclosure

○ Specify tolerance for adjustment zone: 0

○ Adjust slave nodes in set:

Contact interaction property: IntProp-1

Options: Interference Fit...

Contact controls: (Default)

☑ Active in this step

Figure 45: Edit interaction

Select Contact. Then select Mechanical >Tangential Behavior and ensure it is frictionless. Select OK.

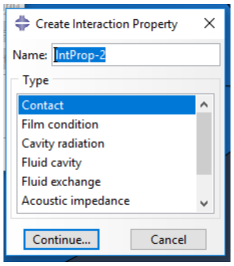
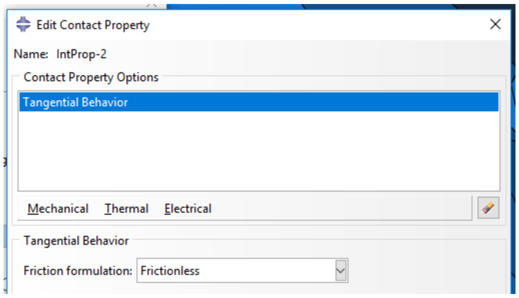


Figure 46: Create interaction property



Friction formulation: Frictionless

Figure 47: Edit contact property

Step 10: Under the module drop down list, select Load and then Create a Boundary Condition.

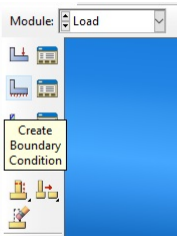
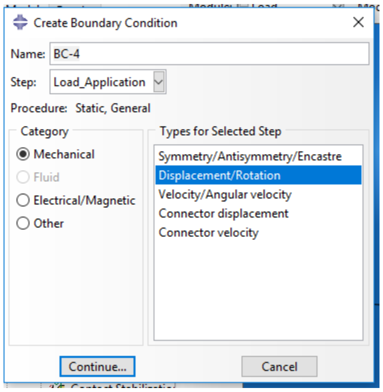


Figure 48: Load module

Ensure the step previously created is selected under Step and then select Displacement/Rotation.



Create Boundary Condition

Name: BC-4

Step: Load_Application

Procedure: Static, General

Category

● Mechanical

○ Fluid

○ Electrical/Magnetic

○ Other

Types for Selected Step

Symmetry/Antisymmetry/Encastre

Displacement/Rotation

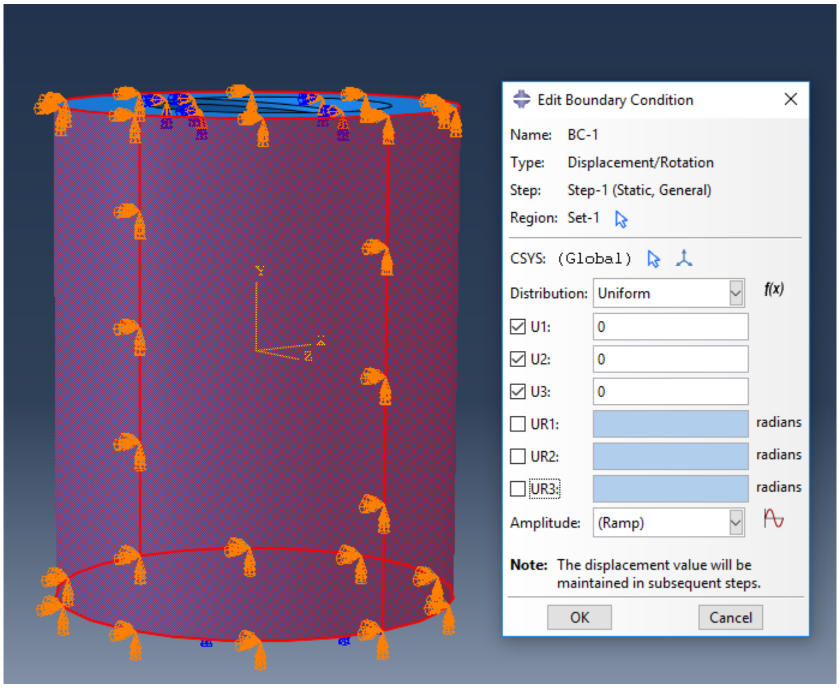
Velocity/Angular velocity

Connector displacement

Connector velocity

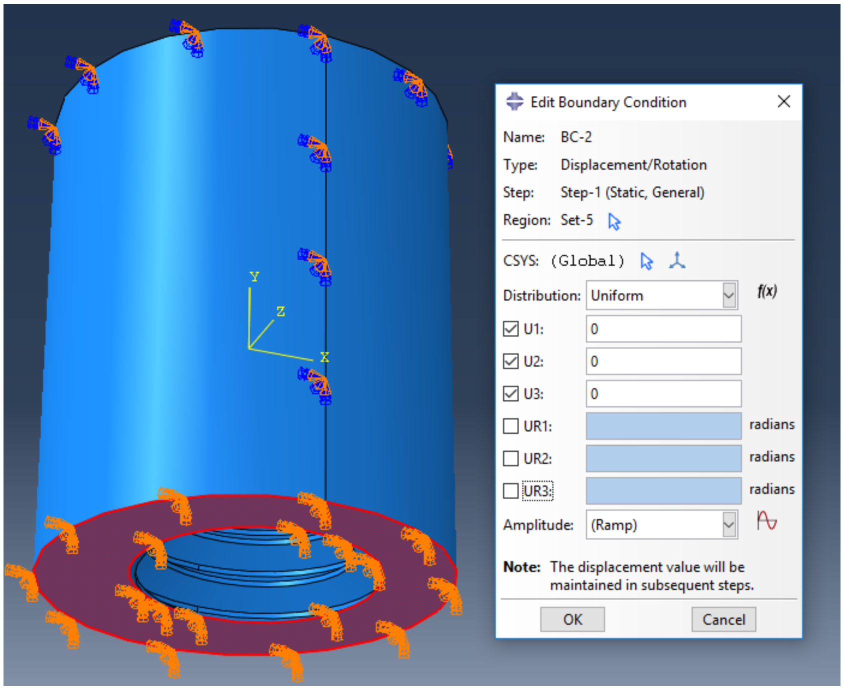
Figure 49: Create boundary condition

Input the following boundary conditions:



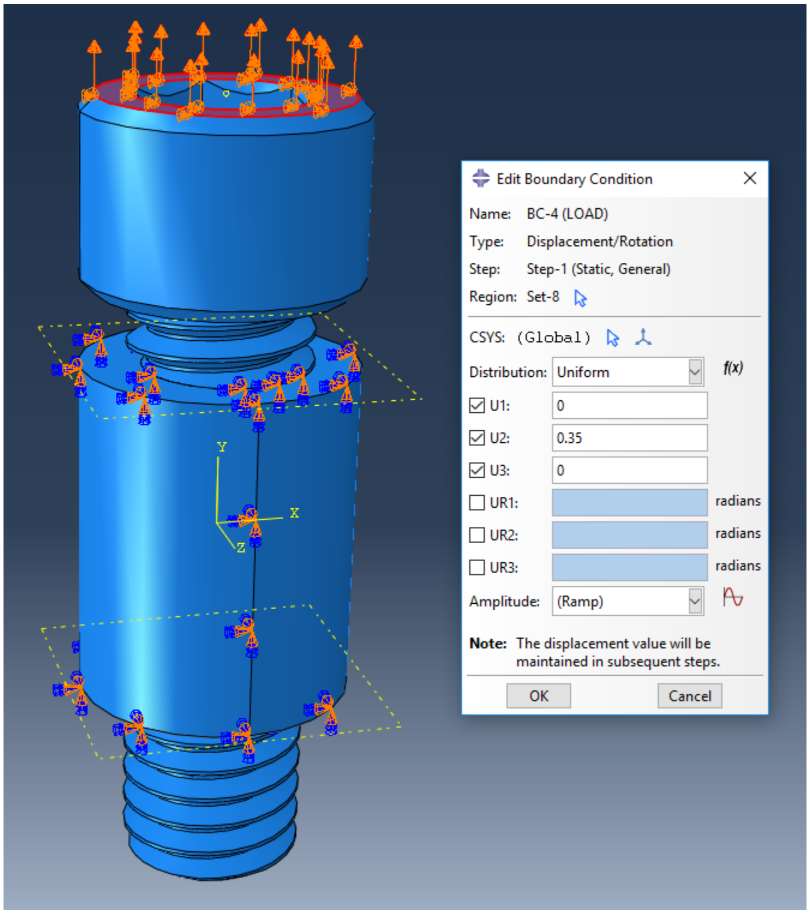
Name: BC-1
Type: Displacement/Rotation
Step: Step-1 (Static, General)
Region: Set-1
CSYS: (Global)
Distribution: Uniform
☑ U1: 0
☑ U2: 0
☑ U3: 0
☐ UR1: radians
☐ UR2: radians
☐ UR3: radians
Amplitude: (Ramp)
**Note:** The displacement value will be maintained in subsequent steps.

Figure 50: Create boundary condition on insert.



Name: BC-2
Type: Displacement/Rotation
Step: Step-1 (Static, General)
Region: Set-5
CSYS: (Global)
Distribution: Uniform
☑ U1: 0
☑ U2: 0
☑ U3: 0
☐ UR1: radians
☐ UR2: radians
☐ UR3: radians
Amplitude: (Ramp)
**Note:** The displacement value will be maintained in subsequent steps.

Figure 51: Create boundary conditions on the top and bottom surface of the insert.



Edit Boundary Condition

Name: BC-4 (LOAD)

Type: Displacement/Rotation

Step: Step-1 (Static, General)

Region: Set-8

CSYS: (Global)

Distribution: Uniform f(x)

☑ U1: 0

☑ U2: 0.35

☑ U3: 0

☐ UR1: radians

☐ UR2: radians

☐ UR3: radians

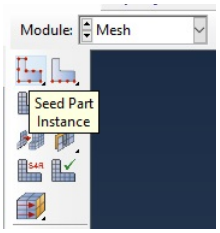
Amplitude: (Ramp)

**Note:** The displacement value will be maintained in subsequent steps.

OK Cancel

Figure 52: Create a displacement on the bolt.

Step 11: Under the module drop down, select Mesh. Ensure Part is selected, so separate parts may be meshed. For each part, select the icon to seed the part instance.



Module: Mesh

Seed Part Instance

Figure 53: Seed part instances

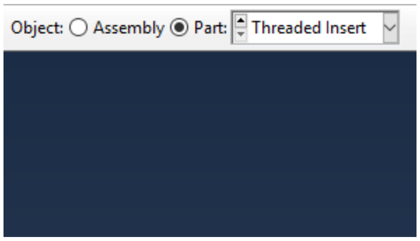
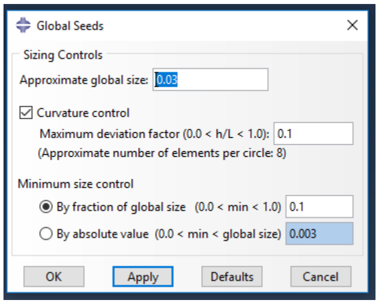

Object: ○ Assembly ● Part: Threaded Insert

Figure 54: Mesh object options

Assign the approximate global size for each part appropriately. Click OK.


Global Seeds

Sizing Controls

Approximate global size: 0.03

☑ Curvature control

Maximum deviation factor (0.0 < h/L < 1.0): 0.1

(Approximate number of elements per circle: 8)

Minimum size control

● By fraction of global size (0.0 < min < 1.0) 0.1

○ By absolute value (0.0 < min < global size) 0.003

Figure 55: Global seeds

Select the icon to assign the mesh controls.

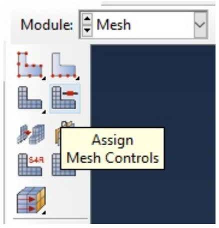
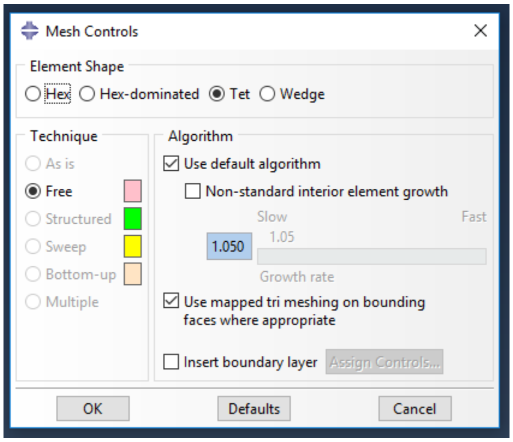

Module: Mesh

Figure 56: Assign mesh controls

Highlight the entire part and click Done. Change the element shape to Tet and select OK.


Mesh Controls

Element Shape

○ Hex ○ Hex-dominated ● Tet ○ Wedge

Technique

○ As is

● Free 🟥

○ Structured 🟩

○ Sweep 🟨

○ Bottom-up

○ Multiple

Algorithm

☑ Use default algorithm

☐ Non-standard interior element growth

Slow Fast

1.050 1.05

Growth rate

☑ Use mapped tri meshing on bounding faces where appropriate

☐ Insert boundary layer Assign Controls...

OK Defaults Cancel

Figure 57: Mesh controls

Once the mesh controls have been set, select the icon to mesh the part and select Done. Ensure this is done for both parts.

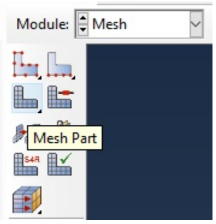


Figure 58: Mesh part

Step 12: Under the drop down module, select Job. Select the icon to create a job.

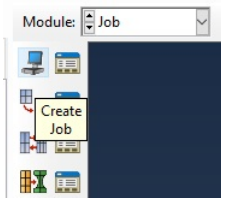
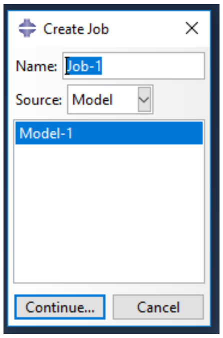


Figure 59: Job module


Create Job

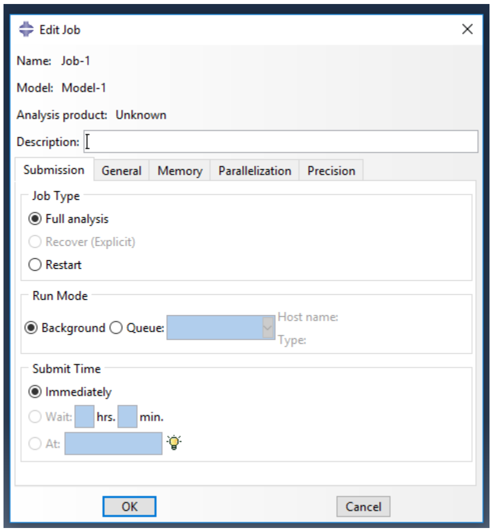
Name: Job-1

Source: Model

Model-1

Figure 60: Create a job

Press Continue, leave everything as default and select OK.


Edit Job

Name: Job-1

Model: Model-1

Analysis product: Unknown

Description:

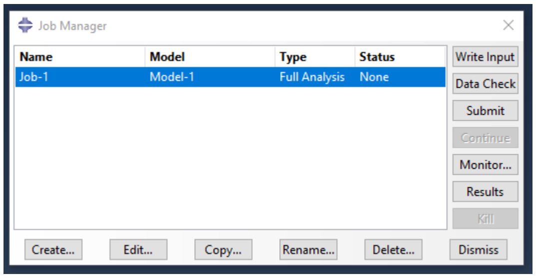
Job Type: ● Full analysis ○ Recover (Explicit) ○ Restart

Run Mode: ● Background ○ Queue:

Submit Time: ● Immediately ○ Wait: hrs. min. ○ At:

Figure 61: Edit job

Once the job is created, submit the job for analysis.

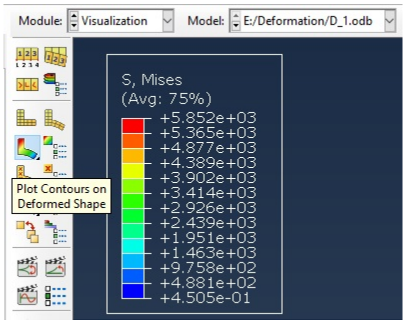

Job Manager

| Name | Model | Type | Status |
|---|---|---|---|
| Job-1 | Model-1 | Full Analysis | None |

Figure 62: Job manager

Step 13: During or after the job has finished, select Results under the job manager.

Select the icon to plot the contours on the deformed shape. You will immediately see the stresses on the deformed shape.

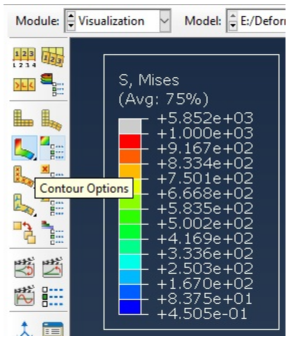


Module: Visualization

Model: E:/Deformation/D_1.odb

Figure 63: Plot contours function

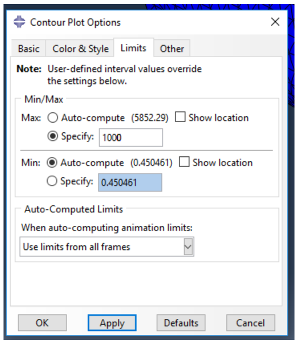
Since the stresses are much lower in some areas, it would be wise to adjust the limits of the stresses to get a better representation of the stresses on the threaded insert. Select the contour options icon to change the limits appropriately. Then click OK.



Module: Visualization

Model: E:/Defor

Figure 64: Contour options function



Contour Plot Options

**Note:** User-defined interval values override the settings below.

Min/Max

Max: ○ Auto-compute (5852.29) ☐ Show location

● Specify: 1000

Min: ● Auto-compute (0.450461) ☐ Show location

○ Specify: 0.450461

Auto-Computed Limits

When auto-computing animation limits: Use limits from all frames

Figure 65: Contour plot limits

Once the limits have been adjusted, you may animate the deformation by selecting Animate >Time History.

## 16.2 Deformation Manual

PART I

Step 1: Obtain the following files and save them into a folder with the name of your choosing: "37516 ALTER.SLDPRT," "Lock Punch.SLDPRT" and "Deformation Assembly.SLDASM." Then open the assembly file in Solidworks. (If a different computer aided design program is being used, open the "Deformation Assembly.STEP" file and follow the subsequent instructions.)

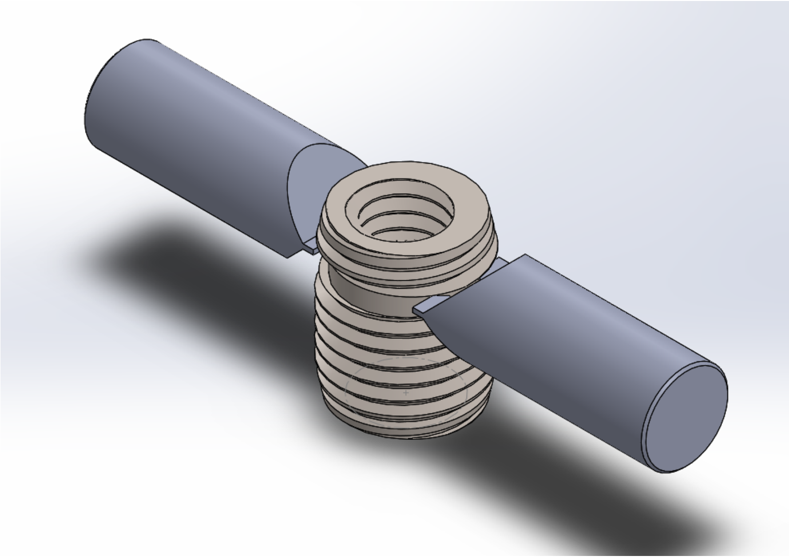


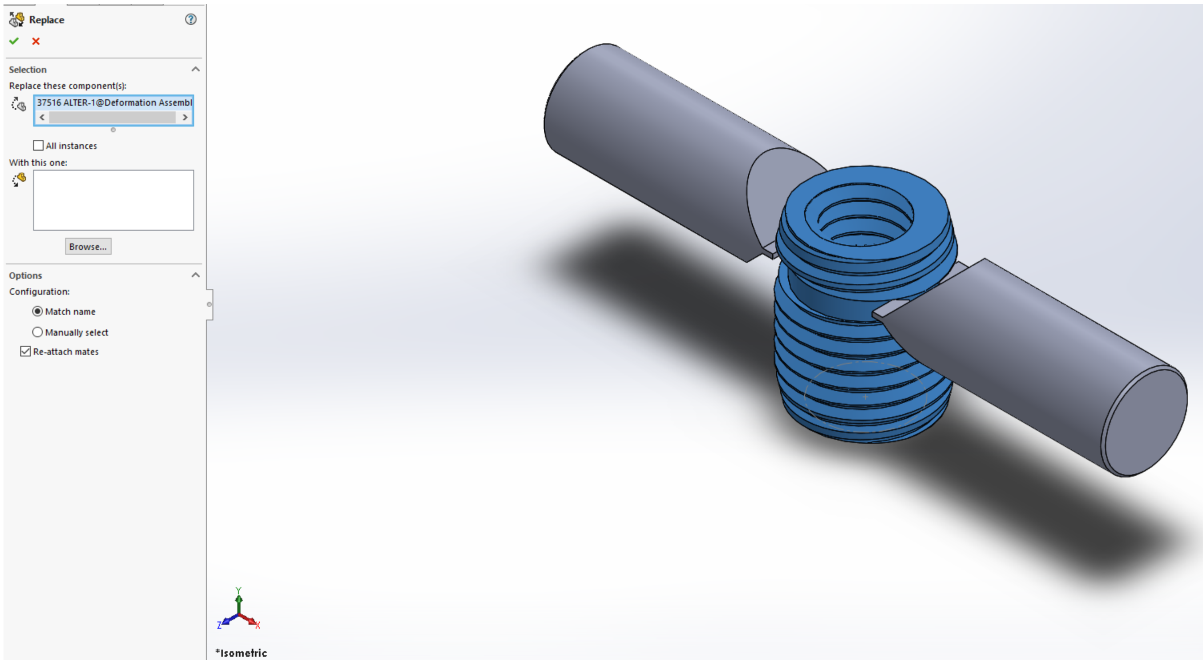
Figure 66: 3D model of deformation

Step 2: To allow for modularity of this digital experiment, the assembly file that is opened in SolidWorks has been mated at the origin using the Front, Right and Top planes. The Lock Punches are mated at equal distances from the center Right plane. If a different threaded insert is desired, right click the threaded insert and scroll down to the option Replace Components. Once this has been selected, select Browse and locate the desired part, and select the green checkmark.

Step 3: To reiterate, the original threaded insert has been mated by its Front, Right, and Top plane accordingly at the origin. This should allow the new part to be changed without any issue. If the new part has been modeled differently and/or modeled outside of the origin, a dialog will appear asking you to select the appropriate planes and/or faces to finalize the replacement.

Step 4: Once the three-dimensional model has been finalized, select File¿Save As and from the drop down menu for File Type, select STEP AP214 (*.step,*.stp) and save to the existing folder.

Once this has been completed, the STEP file may be imported into Abaqus for analysis.



Replace these component(s): 37516 ALTER-1@Deformation Assembl

☐ All instances

With this one:

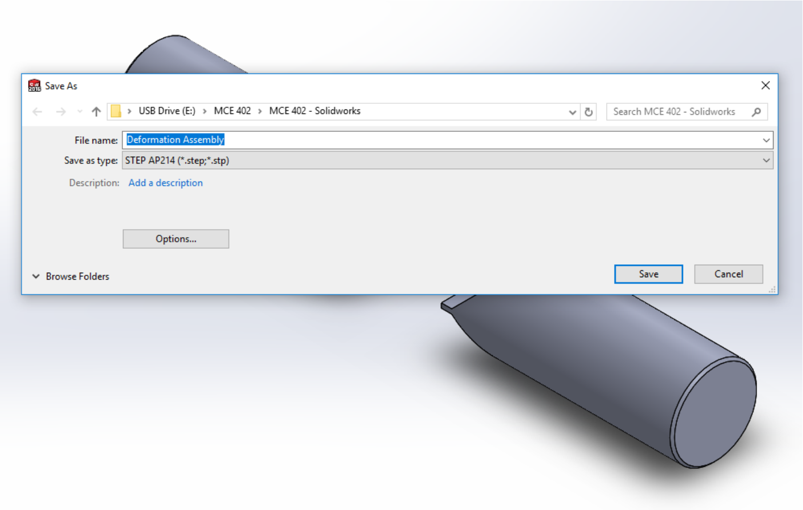
Configuration:

● Match name

○ Manually select

☑ Re-attach mates

Figure 67: Replace components feature



File name: Deformation Assembly

Save as type: STEP AP214 (*.step;*.stp)

Figure 68: How to save a STEP file

Part II

Step 1: Open Abaqus and select File¿Import¿Part. Locate the STEP file created in Part I and select OK.

Step 2: Once the model has been imported, rename each individual part by expanding Parts under the drop down menu, right click each part and select Rename.

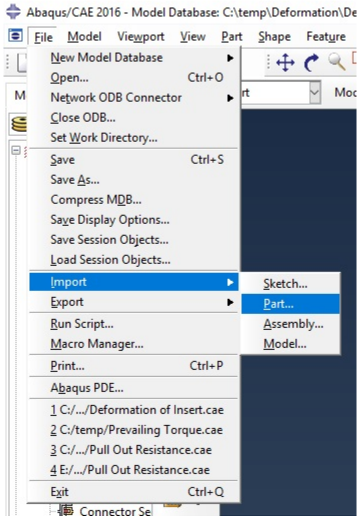


Figure 69: How to import model into Abaqus

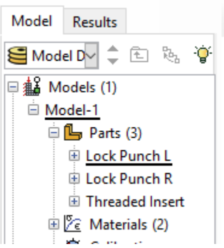


Figure 70: Part description

Step 3: Now it is time to define the desired materials and their corresponding properties. Select Property from the drop down module and then select the icon to create a material.

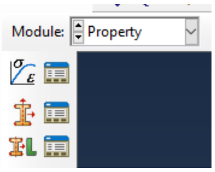
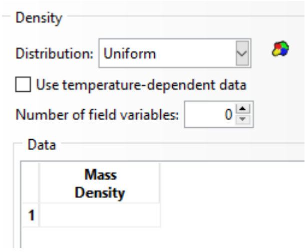


Figure 71: Property module

Step 4: It is important to create two separate materials in order to define which part will retain the plastic deformation properties. Locate the desired material properties, preferably ones derived experimentally and input them into Abaqus.

Material 1

Input the density, Youngs Modulus and Poissons Ratio. (Abaqus is a unitless program so it is important to note that the units need to be consistent in order to obtain accurate results).


Density

Distribution: Uniform

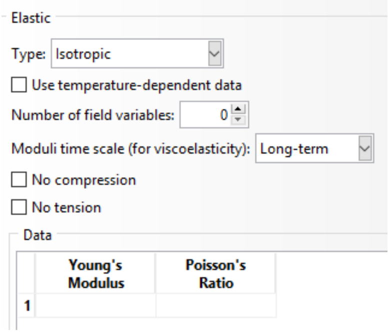
☐ Use temperature-dependent data

Number of field variables: 0

Data

| | Mass Density |
|---|---|
| 1 | |

Figure 72: Density property


Elastic

Type: Isotropic

☐ Use temperature-dependent data

Number of field variables: 0

Moduli time scale (for viscoelasticity): Long-term

☐ No compression

☐ No tension

Data

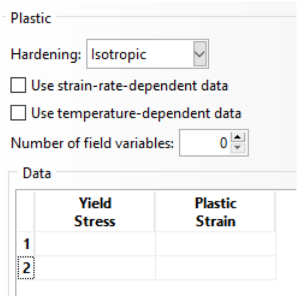
| | Young's Modulus | Poisson's Ratio |
|---|---|---|
| 1 | | |

Figure 73: Elasticity properties

This material will be used to define the Lock Punch parts. Save this property as the material name.

Material 2

To enable plastic deformation, the second material has an additional property that needs to be defined. Once the density and elastic properties have been defined, select Mechanical >Plasticity >Plastic. Two rows need to be defined, the first being the values determined experimentally or by a stress-strain curve at the yield point and the second being the values at a point after the initial yield point. This will allow Abaqus to determine the remaining yield points.



Plastic

Hardening: Isotropic

☐ Use strain-rate-dependent data

☐ Use temperature-dependent data

Number of field variables: 0

Data

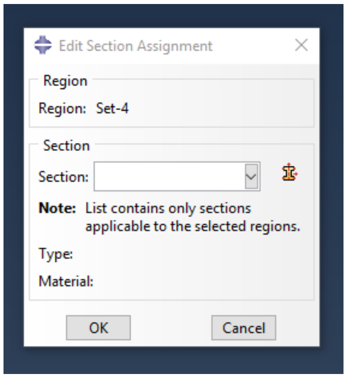
|  | Yield Stress | Plastic Strain |
|---|---|---|
| 1 |  |  |
| 2 |  |  |

Figure 74: Plasticity properties

Save this material as the material name followed by _plastic. This will differentiate the two materials when a section is added to the individual parts.

Step 5: Create two sections, both homogeneous and give them the same name as the material.

Step 6: Once the sections have been created, select Assign Section and assign the sections to the corresponding parts. Only the threaded insert should be assigned the material with plasticity. This will allow only the threaded insert to deform.

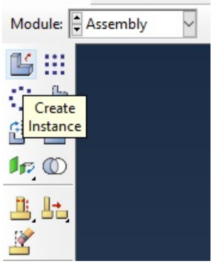


Region: Set-4

**Note:** List contains only sections applicable to the selected regions.

Figure 75: Edit section assignment

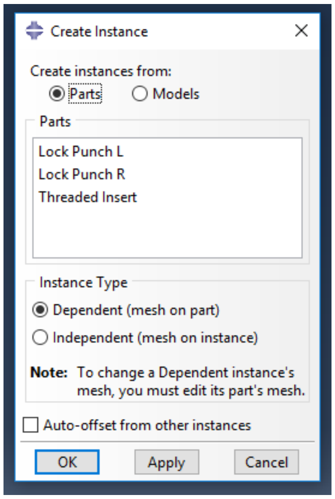
Step 7: Under the module drop down list, select Assembly and then select the icon to create an instance.



Module: Assembly

Figure 76: Assembly module

A dialog will appear with the three parts created by the STEP file. Select each listed part and then select Apply. The parts will appear in their appropriate place as they were created in Solidworks.


Create Instance

Create instances from: ● Parts ○ Models

Parts

Lock Punch L
Lock Punch R
Threaded Insert

Instance Type

● Dependent (mesh on part)
○ Independent (mesh on instance)

**Note:** To change a Dependent instance's mesh, you must edit its part's mesh.

☐ Auto-offset from other instances

Figure 77: Create instance

After all three parts have been assigned to the assembly, click cancel to proceed.

Step 8: Create a step by selecting the icon shown.

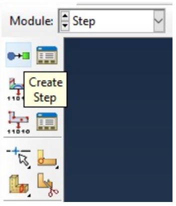
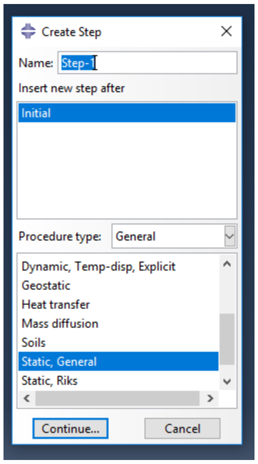

Module: Step

Figure 78: Step module

Name the step something relevant, select Static, General and then Continue


Create Step

Name: Step-1

Insert new step after

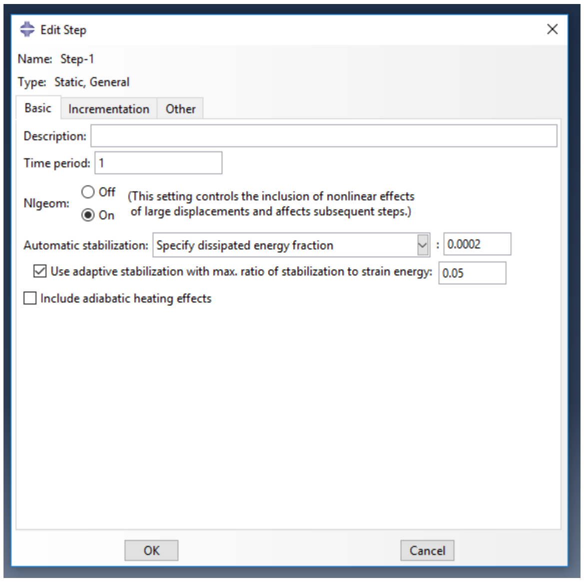
Initial

Procedure type: General

Dynamic, Temp-disp, Explicit
Geostatic
Heat transfer
Mass diffusion
Soils
Static, General
Static, Riks

Figure 79: Create a step

A dialog will appear with some options. Ensure that Nlgeom and Specify dissipated energy fraction under Automatic Stabilization are selected. Then go to Incrementation and change the initial and maximum increment size appropriately.



Edit Step

Name: Step-1

Type: Static, General

Description:

Time period: 1

Nlgeom: ○ Off ● On (This setting controls the inclusion of nonlinear effects of large displacements and affects subsequent steps.)

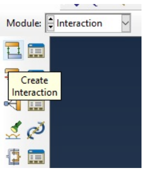
Automatic stabilization: Specify dissipated energy fraction : 0.0002

☑ Use adaptive stabilization with max. ratio of stabilization to strain energy: 0.05

☐ Include adiabatic heating effects

Figure 80: Edit step

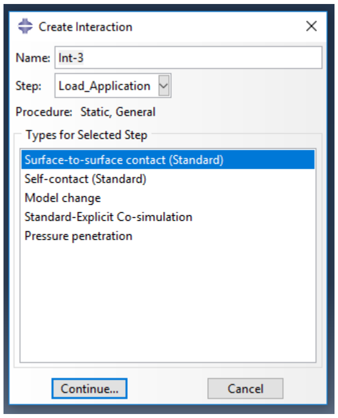
Step 9: Now it is time to create contact between the threaded insert and lock punches. Begin by selecting Interaction under the module and then Create Interaction.



Module: Interaction

Figure 81: Interaction module

Once the dialog appears, select Surface-to-surface contact (standard) and then Continue



Name: Int-3

Step: Load_Application

Procedure: Static, General

Figure 82: Create an interaction

Select the exterior face of the threaded insert that will make contact with the lock punch as the Master Surface. Press Done. Abaqus will ask for the slave type, select Surface and then select the four edges on the punch as the Slave Surface. Press Done.

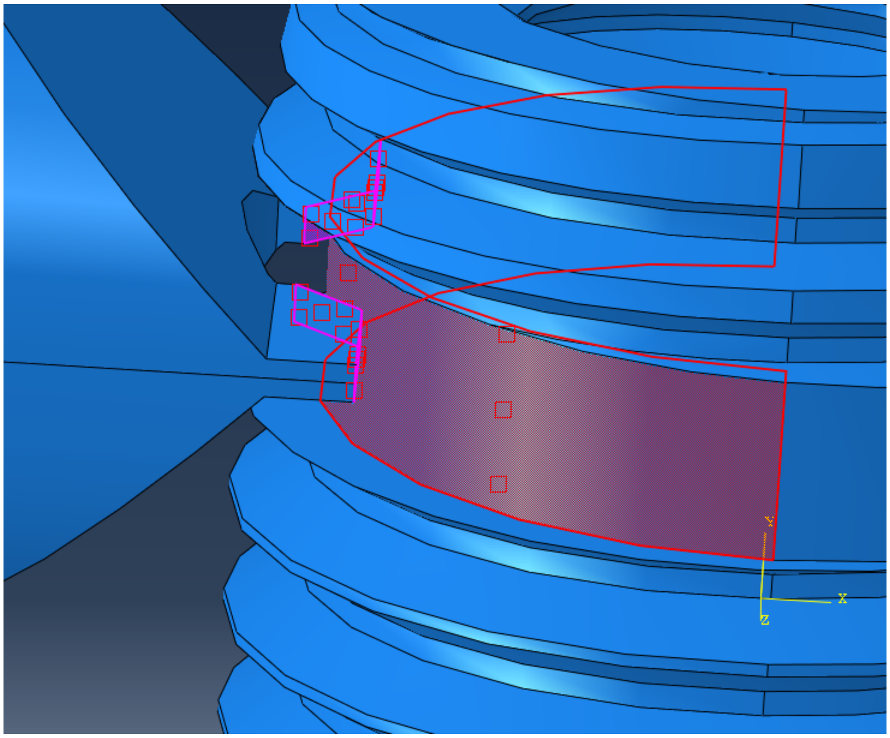
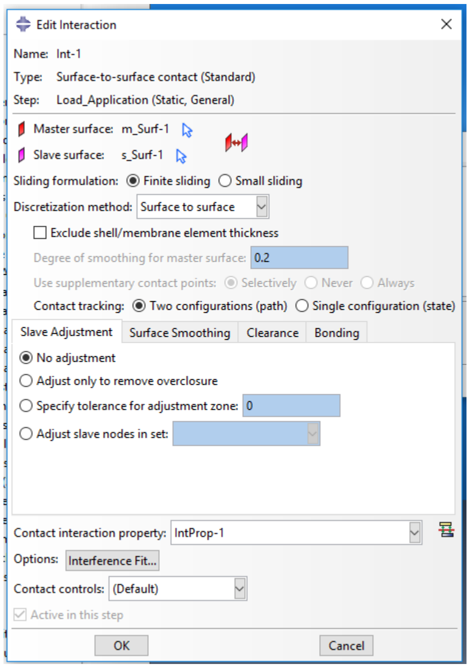


Figure 83: Master and slave surfaces

Once the dialog appears, find the icon to create an interaction property and select it.


Edit Interaction

Name: Int-1
Type: Surface-to-surface contact (Standard)
Step: Load_Application (Static, General)

Master surface: m_Surf-1
Slave surface: s_Surf-1

Sliding formulation: ● Finite sliding ○ Small sliding
Discretization method: Surface to surface

☐ Exclude shell/membrane element thickness
Degree of smoothing for master surface: 0.2
Use supplementary contact points: ● Selectively ○ Never ○ Always
Contact tracking: ● Two configurations (path) ○ Single configuration (state)

Slave Adjustment | Surface Smoothing | Clearance | Bonding

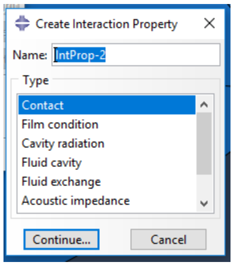
● No adjustment
○ Adjust only to remove overclosure
○ Specify tolerance for adjustment zone: 0
○ Adjust slave nodes in set:

Contact interaction property: IntProp-1
Options: Interference Fit...
Contact controls: (Default)
☑ Active in this step

OK | Cancel

Figure 84: Edit interaction

Select Contact. Then select Mechanical >Tangential Behavior and ensure it is frictionless. Select OK.


Create Interaction Property

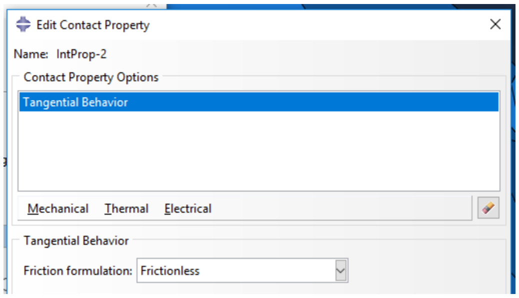
Name: IntProp-2

Type

- Contact
- Film condition
- Cavity radiation
- Fluid cavity
- Fluid exchange
- Acoustic impedance

Continue... | Cancel

Figure 85: Create interaction property


Edit Contact Property

Name: IntProp-2

Contact Property Options

Tangential Behavior

Mechanical | Thermal | Electrical

Tangential Behavior

Friction formulation: Frictionless

Figure 86: Edit contact property

Step 10: Under the module drop down list, select Load and then Create a Boundary Condition.

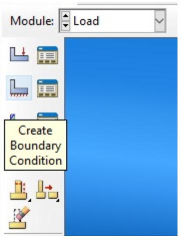
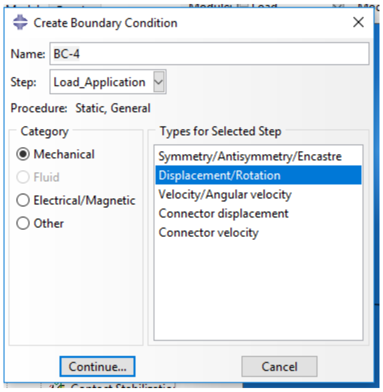


Figure 87: Load module

Ensure the step previously created is selected under Step and then select Displacement/Rotation.


Create Boundary Condition

Name: BC-4

Step: Load_Application

Procedure: Static, General

Category

● Mechanical

○ Fluid

○ Electrical/Magnetic

○ Other

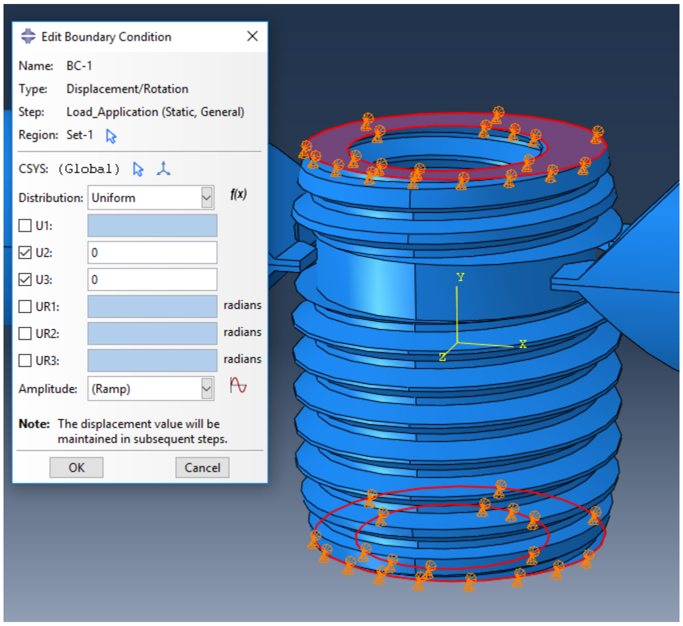
Types for Selected Step

- Symmetry/Antisymmetry/Encastre
- Displacement/Rotation
- Velocity/Angular velocity
- Connector displacement
- Connector velocity

Figure 88: Create boundary condition

Input the following boundary conditions:

Threaded Insert

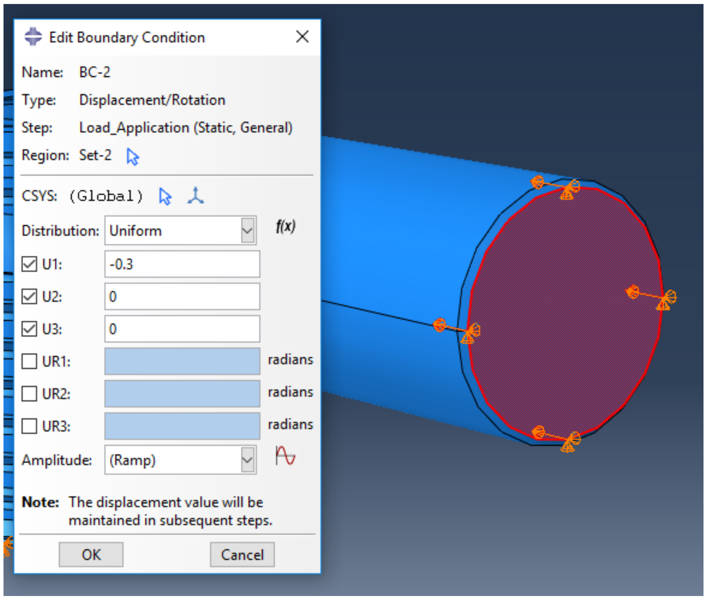


Name: BC-1
Type: Displacement/Rotation
Step: Load_Application (Static, General)
Region: Set-1
CSYS: (Global)
Distribution: Uniform
☐ U1:
☑ U2: 0
☑ U3: 0
☐ UR1: radians
☐ UR2: radians
☐ UR3: radians
Amplitude: (Ramp)

**Note:** The displacement value will be maintained in subsequent steps.

Figure 89: Threaded insert boundary conditions

Right Lock Punch

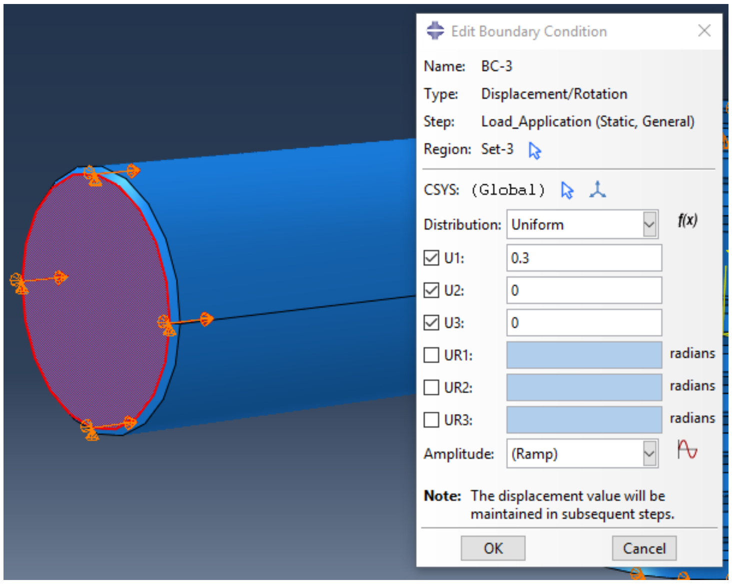


Name: BC-2
Type: Displacement/Rotation
Step: Load_Application (Static, General)
Region: Set-2
CSYS: (Global)
Distribution: Uniform
☑ U1: -0.3
☑ U2: 0
☑ U3: 0
☐ UR1: radians
☐ UR2: radians
☐ UR3: radians
Amplitude: (Ramp)

**Note:** The displacement value will be maintained in subsequent steps.

Figure 90: Right lock punch boundary conditions

Left Lock Punch



Name: BC-3
Type: Displacement/Rotation
Step: Load_Application (Static, General)
Region: Set-3
CSYS: (Global)
Distribution: Uniform
☑ U1: 0.3
☑ U2: 0
☑ U3: 0
☐ UR1: radians
☐ UR2: radians
☐ UR3: radians
Amplitude: (Ramp)

**Note:** The displacement value will be maintained in subsequent steps.

Figure 91: Left lock punch boundary conditions

Step 11: Under the module drop down, select Mesh. Ensure Part is selected, so separate parts may be meshed. For each part, select the icon to seed the part instance.

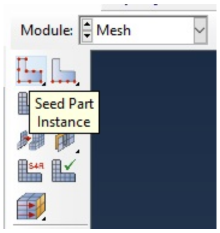

Module: Mesh

Figure 92: Seed part instances

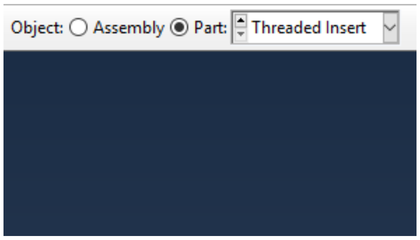
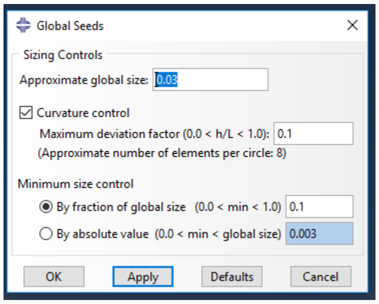

Object: ○ Assembly ● Part: Threaded Insert

Figure 93: Mesh object options

Assign the approximate global size for each part appropriately. Click OK.



Global Seeds

Sizing Controls

Approximate global size: 0.03

☑ Curvature control

Maximum deviation factor (0.0 < h/L < 1.0): 0.1

(Approximate number of elements per circle: 8)

Minimum size control

● By fraction of global size (0.0 < min < 1.0) 0.1

○ By absolute value (0.0 < min < global size) 0.003

Figure 94: Global seeds

Select the icon to assign the mesh controls.

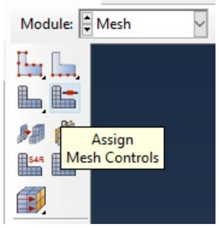
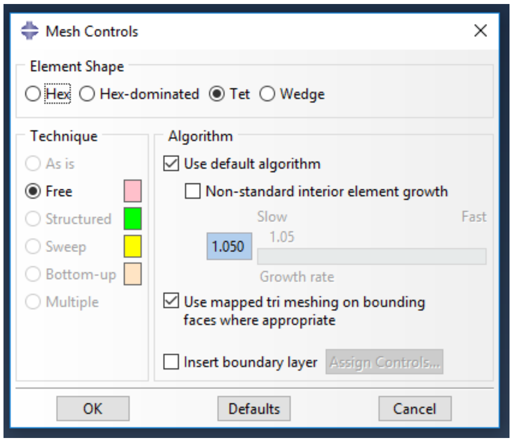


Figure 95: Assign mesh controls

Highlight the entire part and click Done. Change the element shape to Tet and select OK.



Mesh Controls

Element Shape

○ Hex ○ Hex-dominated ● Tet ○ Wedge

Technique

○ As is

● Free

○ Structured

○ Sweep

○ Bottom-up

○ Multiple

Algorithm

☑ Use default algorithm

☐ Non-standard interior element growth

Slow Fast

1.050 1.05

Growth rate

☑ Use mapped tri meshing on bounding faces where appropriate

☐ Insert boundary layer

Figure 96: Mesh controls

Once the mesh controls have been set, select the icon to mesh the part and select Done. Ensure this is done for all three parts.

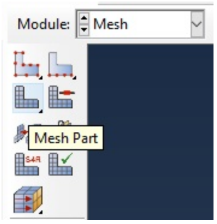


Figure 97: Mesh part

Step 12: Under the drop down module, select Job. Select the icon to create a job.

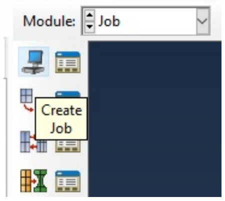
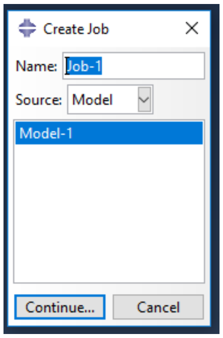


Figure 98: Job module



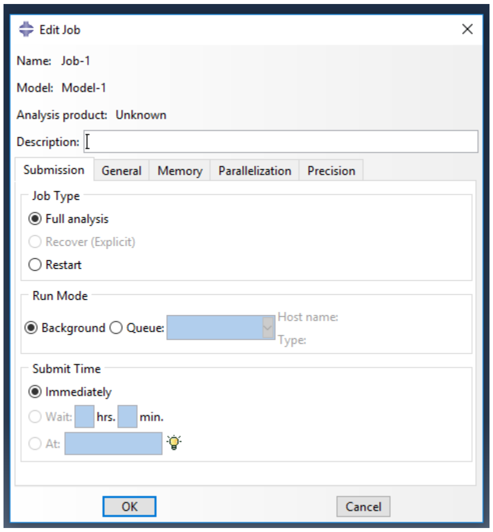
Name: Job-1

Source: Model

Model-1

Figure 99: Create a job

Press Continue, leave everything as default and select OK.



Name: Job-1

Model: Model-1

Analysis product: Unknown

Description:

Job Type

● Full analysis

○ Recover (Explicit)

○ Restart

Run Mode

● Background ○ Queue: Host name: Type:

Submit Time

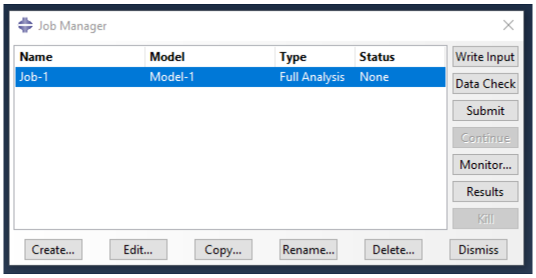
● Immediately

○ Wait: hrs. min.

○ At:

Figure 100: Edit job

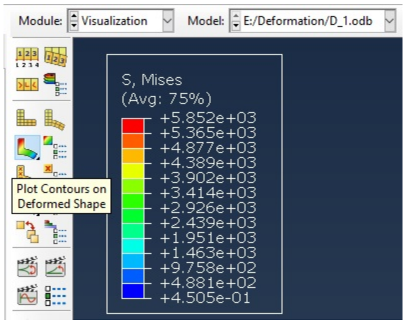
Once the job is created, submit the job for analysis.



| Name | Model | Type | Status |
|---|---|---|---|
| Job-1 | Model-1 | Full Analysis | None |

Figure 101: Job manager

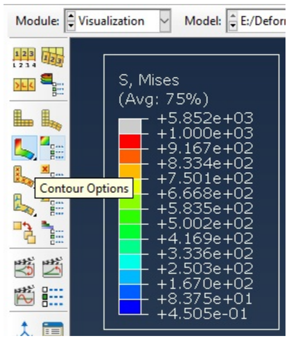
Step 13: During or after the job has finished, select Results under the job manager. Select the icon to plot the contours on the deformed shape. You will immediately see the stresses on the deformed shape.



S, Mises
(Avg: 75%)
+5.852e+03
+5.365e+03
+4.877e+03
+4.389e+03
+3.902e+03
+3.414e+03
+2.926e+03
+2.439e+03
+1.951e+03
+1.463e+03
+9.758e+02
+4.881e+02
+4.505e-01

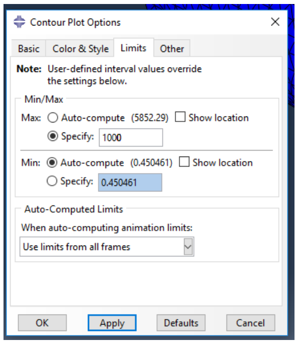
Figure 102: Plot contours function

Since the stresses are much higher on the punch tool, it would be wise to adjust the limits of the stresses to get a better representation of the stresses on the threaded insert. Select the contour options icon to change the limits appropriately. Then click OK.



S, Mises
(Avg: 75%)
+5.852e+03
+1.000e+03
+9.167e+02
+8.334e+02
+7.501e+02
+6.668e+02
+5.835e+02
+5.002e+02
+4.169e+02
+3.336e+02
+2.503e+02
+1.670e+02
+8.375e+01
+4.505e-01

Figure 103: Contour options function



**Note:** User-defined interval values override the settings below.

Min/Max

Max: ○ Auto-compute (5852.29) ☐ Show location

● Specify: 1000

Min: ● Auto-compute (0.450461) ☐ Show location

○ Specify: 0.450461

Auto-Computed Limits

When auto-computing animation limits:

Use limits from all frames

Figure 104: Contour plot limits

Once the limits have been adjusted, you may animate the deformation by selecting Animate >Time History.

# 17 Maintenance

Due to the nature of the project being mostly digital, maintenance is generally not needed. However, when working with digital files, there are steps that can be taken to ensure that they are still relevant in the future. The Abaqus software is constantly undergoing changes and updates. Ensuring the the latest version of Abaqus is installed is recommended so that the user can stay up-to-date on the latest features and tutorials. Abaqus automatically converts older models to the current version so that all older models are backwards compatible.

# 18 Additional Considerations

## 18.1 Economic Impact

The work that Groovy-Pin completed over the past year could have a huge economical impact on Groov-Pin. If Groov-Pin decides to utilize the digital experiments that were designed in place of physical experiments, they could save a tremendous amount of money. These digital experiments would save them the cost of performing destructive tests to determine failure modes.

Utilizing these digital experiments could also increase sales as Groov-Pin will be able to visualize the failure modes for their customers. Customers would not only enjoy the benefits of Groov-Pin's quality products but they would also be able to design their products according to the stresses identified in Abaqus. This would save the customer money and time by eliminating their own destructive testing for their specific applications.

## 18.2 Environmental Impact

Groovy-Pin was able to successfully design digital experiments using finite element analysis to simulate and identify peak stressed and failure modes of Groov-Pin's threaded inserts. As previously stated, these tests will eliminate many of the destructive tests performed by Groov-Pin to determine these failure modes. This ultimately saves material and energy required to manufacture these threaded inserts used in the destructive tests.

## 18.3 Societal Impact

There is not much societal impact from Groovy-Pin's results. However Groov-Pin does not currently use Abaqus and if the company where to begin using the software it would open up job positions for an FEA engineer. The program Abaqus could also change the way

Groov-Pin tests their inserts and this would have an impact on the culture at Groov-Pin by changing from physical testing to digital. This type of testing not only would effect Groov-Pin but would also effect the whole fastener industry.

## 18.4 Political Impact

The models that Groovy-Pin were able to simulate may provide a slight political impact due to the industries that Groov-Pin is involved in. Groov-Pin helps supply the US military with their Tap-Lok series threaded insert that resist vibrations and are used to take on the impact and hold together supplies that are dropped from planes or helicopters for the military when there is no safe area to land. Groovy-Pin worked to supply the company with models that would test the strength of the insert. The results will give Groov-Pin a better understanding of the interaction between the threads so the insert can be redesigned and improved. With stronger inserts Groov-Pin can continue to support the US military and help the troops access supplies without damage.

## 18.5 Ethical Impact

Groov-Pin products are used in industries that count on their products for reliability. Although digital experiments and simulations themselves have relatively little impact on ethical considerations, ethics must always be considered in the engineering process as a whole in which they involved. Groov-Pin products need to be engineered to be reliable so that when they are used in products they perform as expected. The company has an ethical responsibility to their customers and the public to deliver products that perform as specified so that they can be held accountable when failure occurs.

## 18.6 Health, Ergonomics, Safety Impact

Due to the nature of the project being mostly digital, there are few immediate health, safety and ergonomic concerns involved with the project. However, as mentioned before, the Abaqus simulations are just a small step in a larger engineering process. Of greatest relevance to Groov-Pin products is the aspect of safety associated with their designs. Groov-Pin products are used in industries that count on Groov-Pin products for reliability. For example, if a part were to fail in a commercial airplane, the occupants in the plane would be at risk. The failure of the product can be traced back to the engineering design process which includes Abaqus simulations. In this respect, it is apparent that safety must always be considered at any point in the engineering process.

## 18.7 Sustainability Impact

When considering sustainability in regards to the project, the most influential factors are related to the actual physical products that Groov-Pin makes rather than digital experiments and simulations. However, it may be the case that the simulations help Groov-Pin discover that their products may use more material than necessary for their function. If a product can function just as well with less material, the product can be redesigned to be more efficient. If a percentage of material can be saved in one unit of a product, due to the large volumes of products manufactured as a whole.

# 19 Conclusions

Team Groovy-Pin successfully completed both the deformation and pull-out resistance FEA models. The results from these experiments were found to be accurate to similar physical experiments. The deformation simulation was able to deform the threaded insert at the four specific points of the punch tools to match what Groov-Pin does during their manufacturing process. These four deformations create a tighter fit between the exterior threads of a bolt and the interior threads of the threaded insert. The pull-out resistance simulation demonstrated that the top three threads of the threaded insert received the most amount of stress and deformation. Although this experiment was not completed with a threaded insert in a base material, the materials of the screw and the threaded insert demonstrate what would happen at the interface of a threaded insert and a base material.

If Groovy-Pin had additional time for this project, the prevailing torque experiment would be explored. This particular experiment involved a series of complex functions that the team did not have time to complete. The research that was done to complete this test showed that the threaded insert would first need to be deformed and then a screw would need to be rotated into the threaded insert while it is still deformed. This is the only way that accurate torques could have been measured. The model that was completed has all of the necessary components to complete the prevailing torque experiment so that Groov-Pin can explore this further.

# 20 References

[1] TeDesco, Jim. "How Much Does SOLIDWORKS Cost?" CATI Tech Notes. Computer Aided Technology, INC, 10 Apr. 2015. Web. 18 Dec. 2016.

[2] Joly Concept. "Unified National Imperial Screw Thread Calculator." Theoretical Machinist. Web. 13 Feb. 2015.

[3] Marks, L., 2017, Tutorial 5: Simple plastic deformation with unloading, Simuleon, Netherlands.

[4] Mississippi State University, 316 Stainless Steel Stress Strain Data, n.d., from https://icme.hpc.msstate.edu/ mediawiki/index.php/316_Stainless_Steel

# 21 Appendix

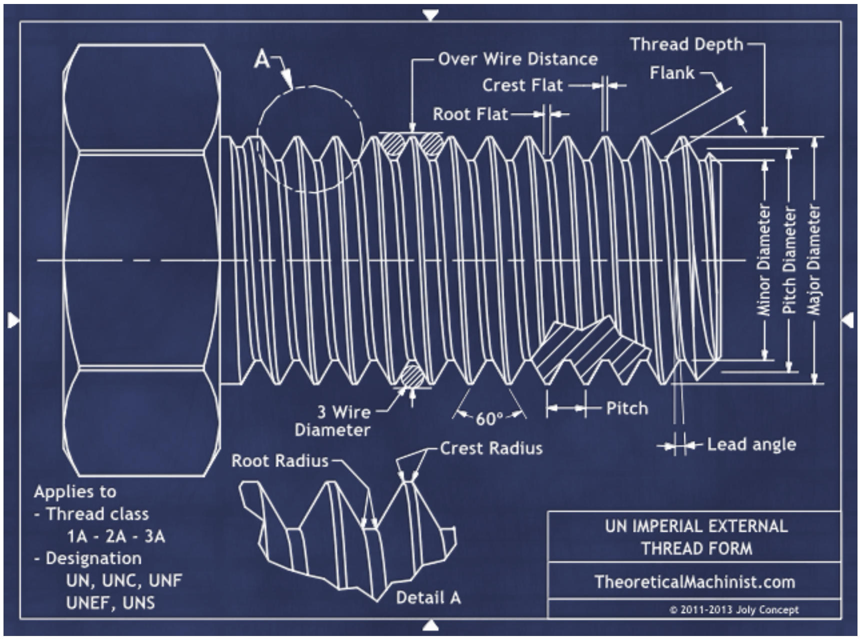


Figure 105: External Threads Reference

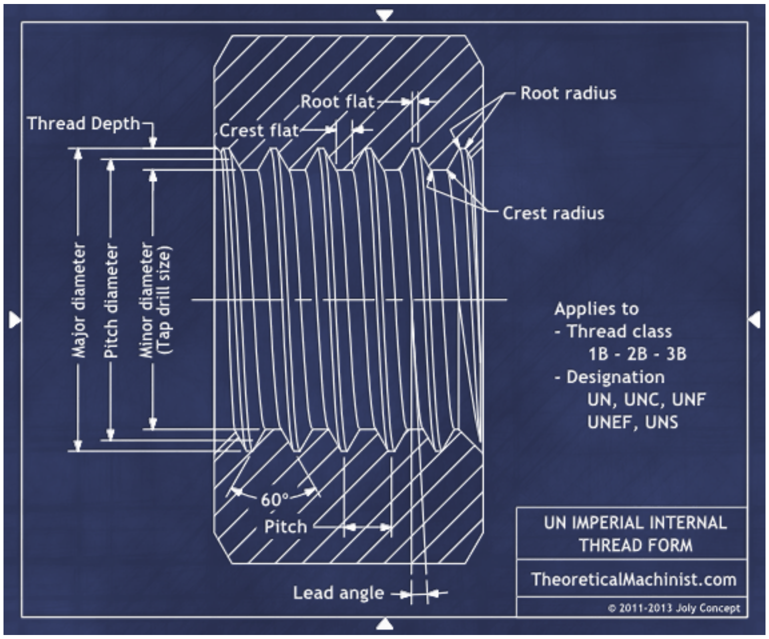
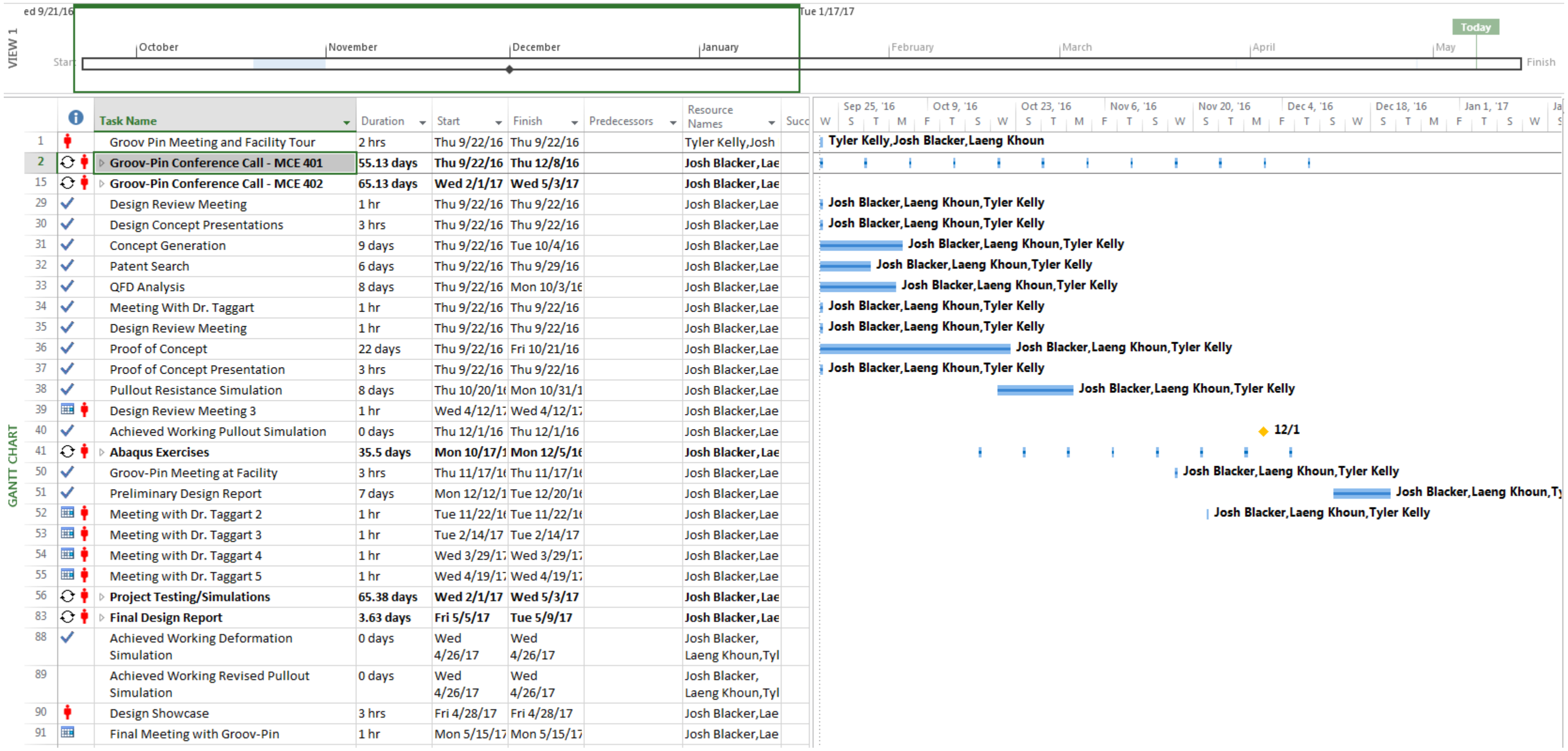
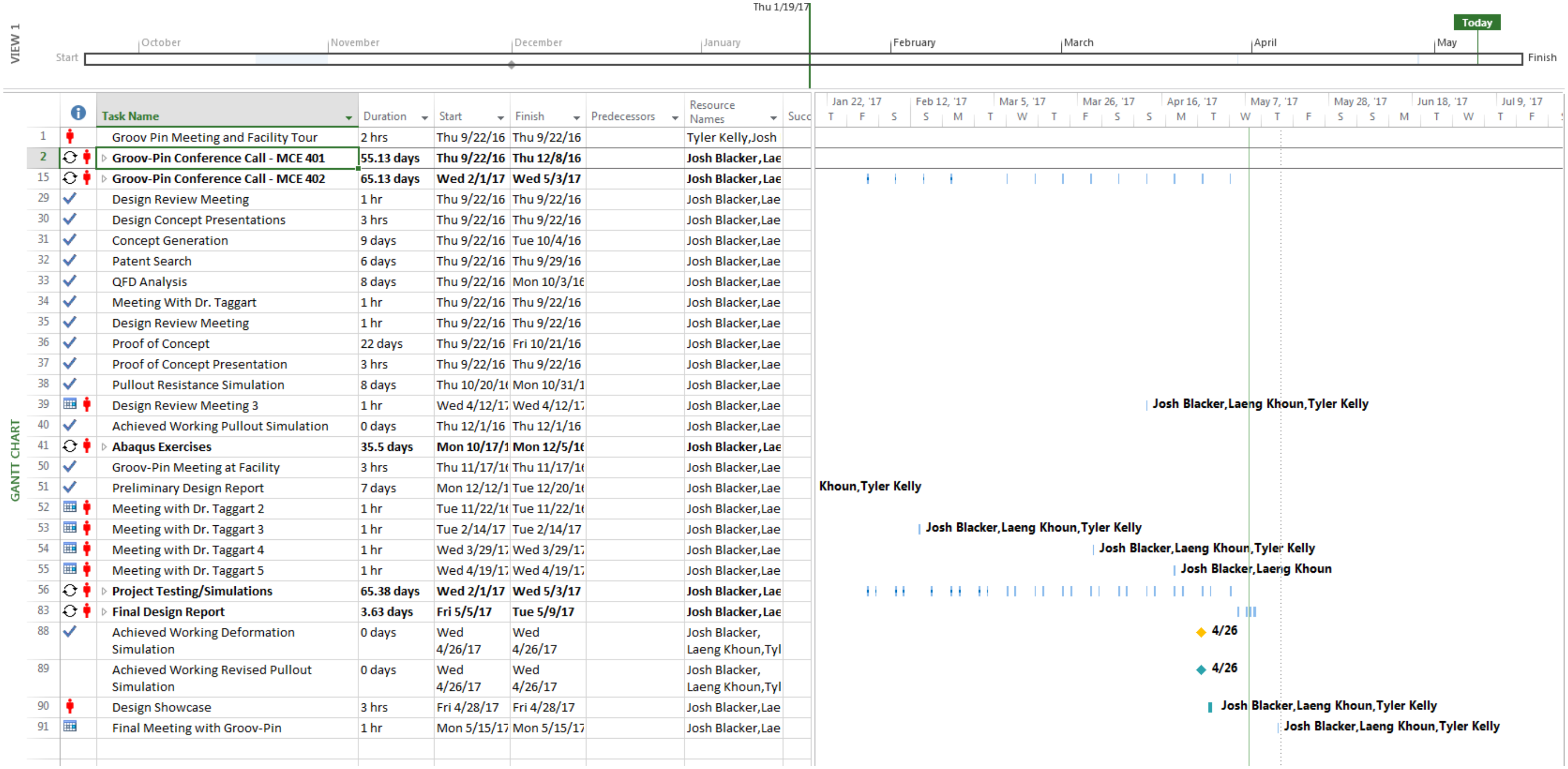


Figure 106: Internal Threads Reference



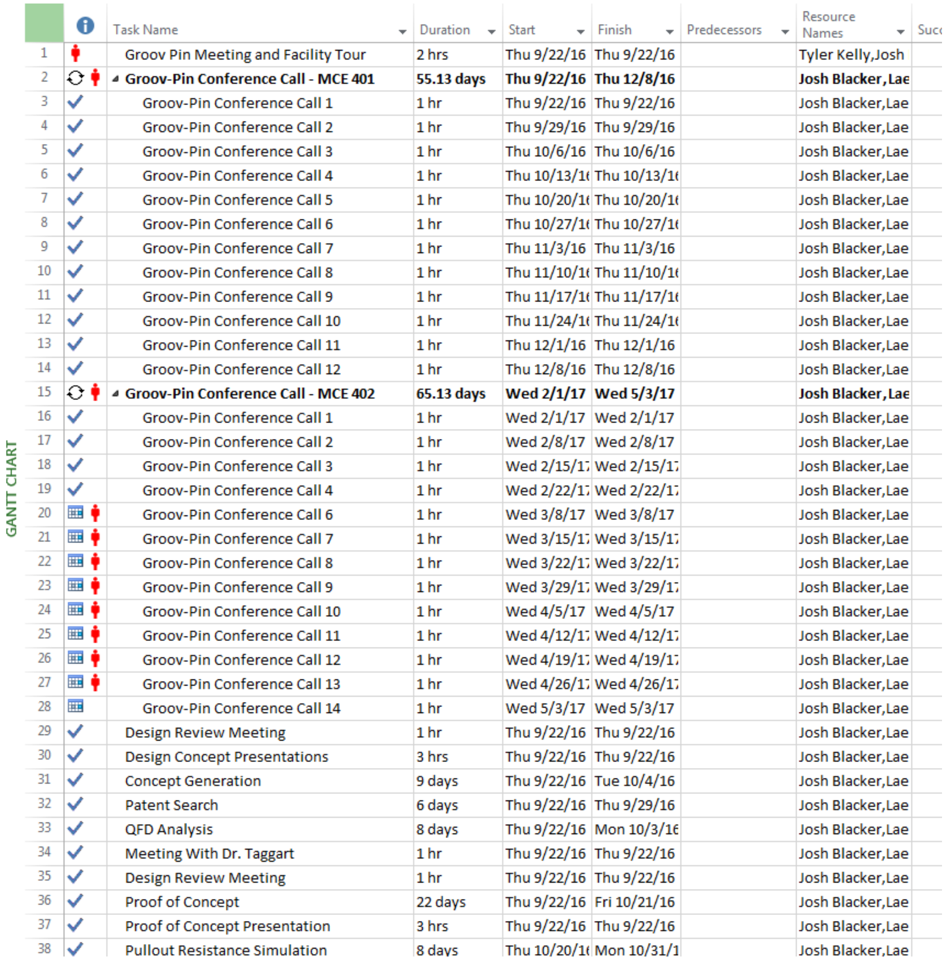
| | ⓘ | Task Name | Duration | Start | Finish | Predecessors | Resource Names | Succ |
|---|---|---|---|---|---|---|---|---|
| 1 | 👤 | Groov Pin Meeting and Facility Tour | 2 hrs | Thu 9/22/16 | Thu 9/22/16 | | Tyler Kelly,Josh | |
| 2 | ⟳ 👤 | **Groov-Pin Conference Call - MCE 401** | **55.13 days** | **Thu 9/22/16** | **Thu 12/8/16** | | **Josh Blacker,Lae** | |
| 15 | ⟳ 👤 | **Groov-Pin Conference Call - MCE 402** | **65.13 days** | **Wed 2/1/17** | **Wed 5/3/17** | | **Josh Blacker,Lae** | |
| 29 | ✓ | Design Review Meeting | 1 hr | Thu 9/22/16 | Thu 9/22/16 | | Josh Blacker,Lae | |
| 30 | ✓ | Design Concept Presentations | 3 hrs | Thu 9/22/16 | Thu 9/22/16 | | Josh Blacker,Lae | |
| 31 | ✓ | Concept Generation | 9 days | Thu 9/22/16 | Tue 10/4/16 | | Josh Blacker,Lae | |
| 32 | ✓ | Patent Search | 6 days | Thu 9/22/16 | Thu 9/29/16 | | Josh Blacker,Lae | |
| 33 | ✓ | QFD Analysis | 8 days | Thu 9/22/16 | Mon 10/3/16 | | Josh Blacker,Lae | |
| 34 | ✓ | Meeting With Dr. Taggart | 1 hr | Thu 9/22/16 | Thu 9/22/16 | | Josh Blacker,Lae | |
| 35 | ✓ | Design Review Meeting | 1 hr | Thu 9/22/16 | Thu 9/22/16 | | Josh Blacker,Lae | |
| 36 | ✓ | Proof of Concept | 22 days | Thu 9/22/16 | Fri 10/21/16 | | Josh Blacker,Lae | |
| 37 | ✓ | Proof of Concept Presentation | 3 hrs | Thu 9/22/16 | Thu 9/22/16 | | Josh Blacker,Lae | |
| 38 | ✓ | Pullout Resistance Simulation | 8 days | Thu 10/20/1 | Mon 10/31/1 | | Josh Blacker,Lae | |
| 39 | ▦ 👤 | Design Review Meeting 3 | 1 hr | Wed 4/12/1 | Wed 4/12/1 | | Josh Blacker,Lae | |
| 40 | ✓ | Achieved Working Pullout Simulation | 0 days | Thu 12/1/16 | Thu 12/1/16 | | Josh Blacker,Lae | |
| 41 | ⟳ 👤 | **Abaqus Exercises** | **35.5 days** | **Mon 10/17/1** | **Mon 12/5/1** | | **Josh Blacker,Lae** | |
| 50 | ✓ | Groov-Pin Meeting at Facility | 3 hrs | Thu 11/17/1 | Thu 11/17/1 | | Josh Blacker,Lae | |
| 51 | ✓ | Preliminary Design Report | 7 days | Mon 12/12/1 | Tue 12/20/1 | | Josh Blacker,Lae | |
| 52 | ▦ 👤 | Meeting with Dr. Taggart 2 | 1 hr | Tue 11/22/1 | Tue 11/22/1 | | Josh Blacker,Lae | |
| 53 | ▦ 👤 | Meeting with Dr. Taggart 3 | 1 hr | Tue 2/14/17 | Tue 2/14/17 | | Josh Blacker,Lae | |
| 54 | ▦ 👤 | Meeting with Dr. Taggart 4 | 1 hr | Wed 3/29/1 | Wed 3/29/1 | | Josh Blacker,Lae | |
| 55 | ▦ 👤 | Meeting with Dr. Taggart 5 | 1 hr | Wed 4/19/1 | Wed 4/19/1 | | Josh Blacker,Lae | |
| 56 | ⟳ 👤 | **Project Testing/Simulations** | **65.38 days** | **Wed 2/1/17** | **Wed 5/3/17** | | **Josh Blacker,Lae** | |
| 83 | ⟳ 👤 | **Final Design Report** | **3.63 days** | **Fri 5/5/17** | **Tue 5/9/17** | | **Josh Blacker,Lae** | |
| 88 | ✓ | Achieved Working Deformation Simulation | 0 days | Wed 4/26/17 | Wed 4/26/17 | | Josh Blacker, Laeng Khoun,Tyl | |
| 89 | | Achieved Working Revised Pullout Simulation | 0 days | Wed 4/26/17 | Wed 4/26/17 | | Josh Blacker, Laeng Khoun,Tyl | |
| 90 | 👤 | Design Showcase | 3 hrs | Fri 4/28/17 | Fri 4/28/17 | | Josh Blacker,Lae | |
| 91 | ▦ | Final Meeting with Groov-Pin | 1 hr | Mon 5/15/1 | Mon 5/15/1 | | Josh Blacker,Lae | |

Figure 107: Project Plan Gantt Chart Page 1



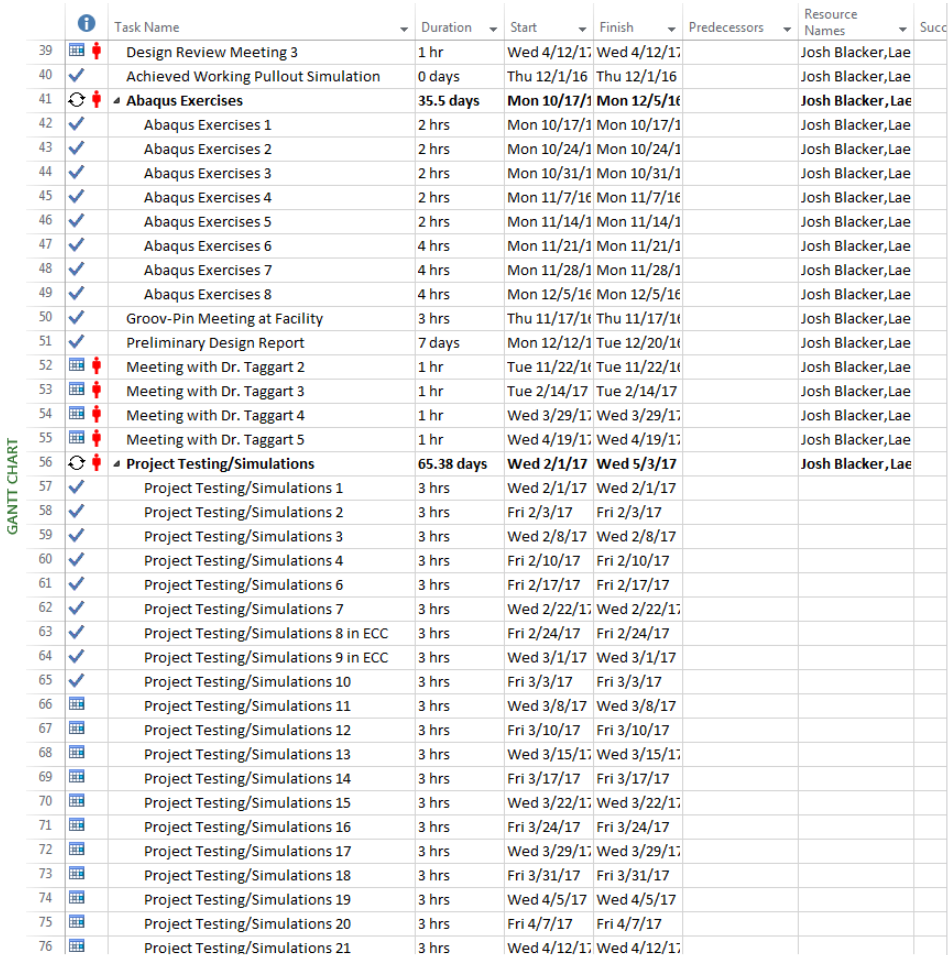
| | ℹ | Task Name | Duration | Start | Finish | Predecessors | Resource Names | Succ |
|---|---|---|---|---|---|---|---|---|
| 1 | | Groov Pin Meeting and Facility Tour | 2 hrs | Thu 9/22/16 | Thu 9/22/16 | | Tyler Kelly,Josh | |
| 2 | ⟳ | **Groov-Pin Conference Call - MCE 401** | **55.13 days** | **Thu 9/22/16** | **Thu 12/8/16** | | **Josh Blacker,Lae** | |
| 15 | ⟳ | **Groov-Pin Conference Call - MCE 402** | **65.13 days** | **Wed 2/1/17** | **Wed 5/3/17** | | **Josh Blacker,Lae** | |
| 29 | ✓ | Design Review Meeting | 1 hr | Thu 9/22/16 | Thu 9/22/16 | | Josh Blacker,Lae | |
| 30 | ✓ | Design Concept Presentations | 3 hrs | Thu 9/22/16 | Thu 9/22/16 | | Josh Blacker,Lae | |
| 31 | ✓ | Concept Generation | 9 days | Thu 9/22/16 | Tue 10/4/16 | | Josh Blacker,Lae | |
| 32 | ✓ | Patent Search | 6 days | Thu 9/22/16 | Thu 9/29/16 | | Josh Blacker,Lae | |
| 33 | ✓ | QFD Analysis | 8 days | Thu 9/22/16 | Mon 10/3/1 | | Josh Blacker,Lae | |
| 34 | ✓ | Meeting With Dr. Taggart | 1 hr | Thu 9/22/16 | Thu 9/22/16 | | Josh Blacker,Lae | |
| 35 | ✓ | Design Review Meeting | 1 hr | Thu 9/22/16 | Thu 9/22/16 | | Josh Blacker,Lae | |
| 36 | ✓ | Proof of Concept | 22 days | Thu 9/22/16 | Fri 10/21/16 | | Josh Blacker,Lae | |
| 37 | ✓ | Proof of Concept Presentation | 3 hrs | Thu 9/22/16 | Thu 9/22/16 | | Josh Blacker,Lae | |
| 38 | ✓ | Pullout Resistance Simulation | 8 days | Thu 10/20/1 | Mon 10/31/1 | | Josh Blacker,Lae | |
| 39 | ▦ | Design Review Meeting 3 | 1 hr | Wed 4/12/1 | Wed 4/12/1 | | Josh Blacker,Lae | |
| 40 | ✓ | Achieved Working Pullout Simulation | 0 days | Thu 12/1/16 | Thu 12/1/16 | | Josh Blacker,Lae | |
| 41 | ⟳ | **Abaqus Exercises** | **35.5 days** | **Mon 10/17/1** | **Mon 12/5/1** | | **Josh Blacker,Lae** | |
| 50 | ✓ | Groov-Pin Meeting at Facility | 3 hrs | Thu 11/17/1 | Thu 11/17/1 | | Josh Blacker,Lae | |
| 51 | ✓ | Preliminary Design Report | 7 days | Mon 12/12/1 | Tue 12/20/1 | | Josh Blacker,Lae | |
| 52 | ▦ | Meeting with Dr. Taggart 2 | 1 hr | Tue 11/22/1 | Tue 11/22/1 | | Josh Blacker,Lae | |
| 53 | ▦ | Meeting with Dr. Taggart 3 | 1 hr | Tue 2/14/17 | Tue 2/14/17 | | Josh Blacker,Lae | |
| 54 | ▦ | Meeting with Dr. Taggart 4 | 1 hr | Wed 3/29/1 | Wed 3/29/1 | | Josh Blacker,Lae | |
| 55 | ▦ | Meeting with Dr. Taggart 5 | 1 hr | Wed 4/19/1 | Wed 4/19/1 | | Josh Blacker,Lae | |
| 56 | ⟳ | **Project Testing/Simulations** | **65.38 days** | **Wed 2/1/17** | **Wed 5/3/17** | | **Josh Blacker,Lae** | |
| 83 | ⟳ | **Final Design Report** | **3.63 days** | **Fri 5/5/17** | **Tue 5/9/17** | | **Josh Blacker,Lae** | |
| 88 | ✓ | Achieved Working Deformation Simulation | 0 days | Wed 4/26/17 | Wed 4/26/17 | | Josh Blacker, Laeng Khoun,Tyl | |
| 89 | | Achieved Working Revised Pullout Simulation | 0 days | Wed 4/26/17 | Wed 4/26/17 | | Josh Blacker, Laeng Khoun,Tyl | |
| 90 | | Design Showcase | 3 hrs | Fri 4/28/17 | Fri 4/28/17 | | Josh Blacker,Lae | |
| 91 | ▦ | Final Meeting with Groov-Pin | 1 hr | Mon 5/15/17 | Mon 5/15/17 | | Josh Blacker,Lae | |

Figure 108: Project Plan Gantt Chart Page 1



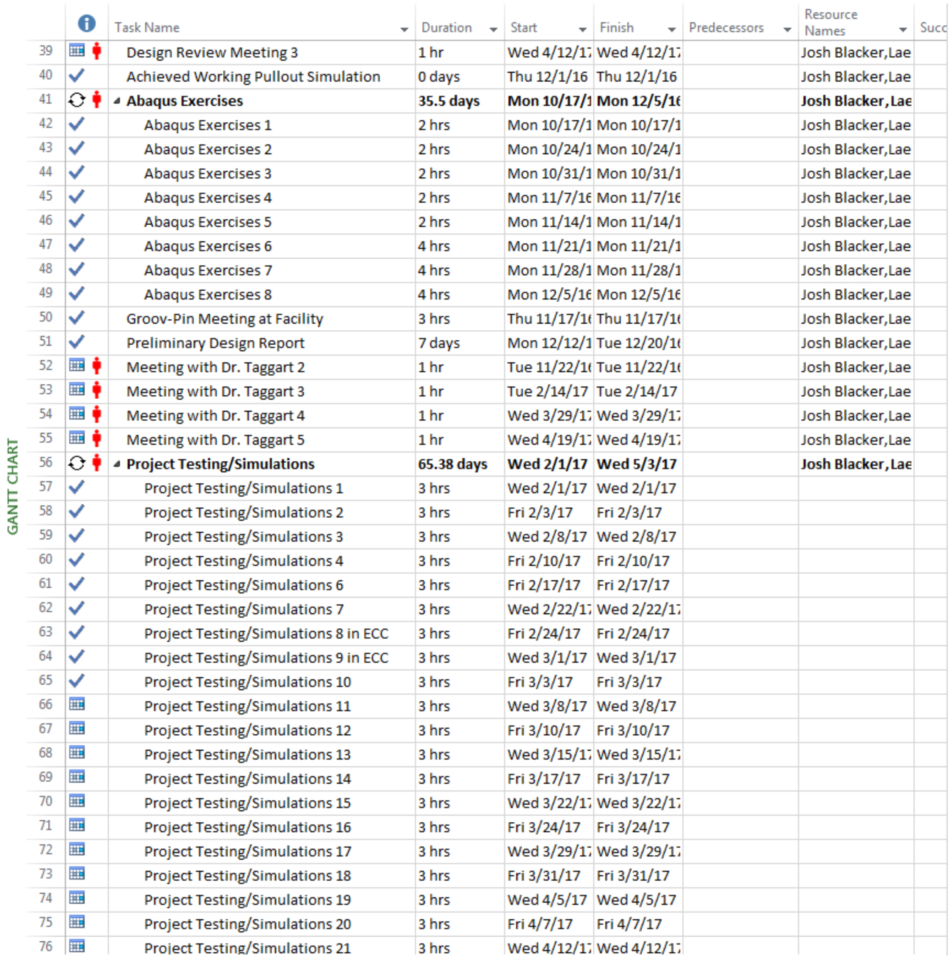
| | ℹ | Task Name | Duration | Start | Finish | Predecessors | Resource Names | Succ |
|---|---|---|---|---|---|---|---|---|
| 1 | 👤 | Groov Pin Meeting and Facility Tour | 2 hrs | Thu 9/22/16 | Thu 9/22/16 | | Tyler Kelly,Josh | |
| 2 | ↻ 👤 | **◢ Groov-Pin Conference Call - MCE 401** | **55.13 days** | **Thu 9/22/16** | **Thu 12/8/16** | | **Josh Blacker,Lae** | |
| 3 | ✓ | Groov-Pin Conference Call 1 | 1 hr | Thu 9/22/16 | Thu 9/22/16 | | Josh Blacker,Lae | |
| 4 | ✓ | Groov-Pin Conference Call 2 | 1 hr | Thu 9/29/16 | Thu 9/29/16 | | Josh Blacker,Lae | |
| 5 | ✓ | Groov-Pin Conference Call 3 | 1 hr | Thu 10/6/16 | Thu 10/6/16 | | Josh Blacker,Lae | |
| 6 | ✓ | Groov-Pin Conference Call 4 | 1 hr | Thu 10/13/1 | Thu 10/13/1 | | Josh Blacker,Lae | |
| 7 | ✓ | Groov-Pin Conference Call 5 | 1 hr | Thu 10/20/1 | Thu 10/20/1 | | Josh Blacker,Lae | |
| 8 | ✓ | Groov-Pin Conference Call 6 | 1 hr | Thu 10/27/1 | Thu 10/27/1 | | Josh Blacker,Lae | |
| 9 | ✓ | Groov-Pin Conference Call 7 | 1 hr | Thu 11/3/16 | Thu 11/3/16 | | Josh Blacker,Lae | |
| 10 | ✓ | Groov-Pin Conference Call 8 | 1 hr | Thu 11/10/1 | Thu 11/10/1 | | Josh Blacker,Lae | |
| 11 | ✓ | Groov-Pin Conference Call 9 | 1 hr | Thu 11/17/1 | Thu 11/17/1 | | Josh Blacker,Lae | |
| 12 | ✓ | Groov-Pin Conference Call 10 | 1 hr | Thu 11/24/1 | Thu 11/24/1 | | Josh Blacker,Lae | |
| 13 | ✓ | Groov-Pin Conference Call 11 | 1 hr | Thu 12/1/16 | Thu 12/1/16 | | Josh Blacker,Lae | |
| 14 | ✓ | Groov-Pin Conference Call 12 | 1 hr | Thu 12/8/16 | Thu 12/8/16 | | Josh Blacker,Lae | |
| 15 | ↻ 👤 | **◢ Groov-Pin Conference Call - MCE 402** | **65.13 days** | **Wed 2/1/17** | **Wed 5/3/17** | | **Josh Blacker,Lae** | |
| 16 | ✓ | Groov-Pin Conference Call 1 | 1 hr | Wed 2/1/17 | Wed 2/1/17 | | Josh Blacker,Lae | |
| 17 | ✓ | Groov-Pin Conference Call 2 | 1 hr | Wed 2/8/17 | Wed 2/8/17 | | Josh Blacker,Lae | |
| 18 | ✓ | Groov-Pin Conference Call 3 | 1 hr | Wed 2/15/1 | Wed 2/15/1 | | Josh Blacker,Lae | |
| 19 | ✓ | Groov-Pin Conference Call 4 | 1 hr | Wed 2/22/1 | Wed 2/22/1 | | Josh Blacker,Lae | |
| 20 | ▦ 👤 | Groov-Pin Conference Call 6 | 1 hr | Wed 3/8/17 | Wed 3/8/17 | | Josh Blacker,Lae | |
| 21 | ▦ 👤 | Groov-Pin Conference Call 7 | 1 hr | Wed 3/15/1 | Wed 3/15/1 | | Josh Blacker,Lae | |
| 22 | ▦ 👤 | Groov-Pin Conference Call 8 | 1 hr | Wed 3/22/1 | Wed 3/22/1 | | Josh Blacker,Lae | |
| 23 | ▦ 👤 | Groov-Pin Conference Call 9 | 1 hr | Wed 3/29/1 | Wed 3/29/1 | | Josh Blacker,Lae | |
| 24 | ▦ 👤 | Groov-Pin Conference Call 10 | 1 hr | Wed 4/5/17 | Wed 4/5/17 | | Josh Blacker,Lae | |
| 25 | ▦ 👤 | Groov-Pin Conference Call 11 | 1 hr | Wed 4/12/1 | Wed 4/12/1 | | Josh Blacker,Lae | |
| 26 | ▦ 👤 | Groov-Pin Conference Call 12 | 1 hr | Wed 4/19/1 | Wed 4/19/1 | | Josh Blacker,Lae | |
| 27 | ▦ 👤 | Groov-Pin Conference Call 13 | 1 hr | Wed 4/26/1 | Wed 4/26/1 | | Josh Blacker,Lae | |
| 28 | ▦ | Groov-Pin Conference Call 14 | 1 hr | Wed 5/3/17 | Wed 5/3/17 | | Josh Blacker,Lae | |
| 29 | ✓ | Design Review Meeting | 1 hr | Thu 9/22/16 | Thu 9/22/16 | | Josh Blacker,Lae | |
| 30 | ✓ | Design Concept Presentations | 3 hrs | Thu 9/22/16 | Thu 9/22/16 | | Josh Blacker,Lae | |
| 31 | ✓ | Concept Generation | 9 days | Thu 9/22/16 | Tue 10/4/16 | | Josh Blacker,Lae | |
| 32 | ✓ | Patent Search | 6 days | Thu 9/22/16 | Thu 9/29/16 | | Josh Blacker,Lae | |
| 33 | ✓ | QFD Analysis | 8 days | Thu 9/22/16 | Mon 10/3/16 | | Josh Blacker,Lae | |
| 34 | ✓ | Meeting With Dr. Taggart | 1 hr | Thu 9/22/16 | Thu 9/22/16 | | Josh Blacker,Lae | |
| 35 | ✓ | Design Review Meeting | 1 hr | Thu 9/22/16 | Thu 9/22/16 | | Josh Blacker,Lae | |
| 36 | ✓ | Proof of Concept | 22 days | Thu 9/22/16 | Fri 10/21/16 | | Josh Blacker,Lae | |
| 37 | ✓ | Proof of Concept Presentation | 3 hrs | Thu 9/22/16 | Thu 9/22/16 | | Josh Blacker,Lae | |
| 38 | ✓ | Pullout Resistance Simulation | 8 days | Thu 10/20/1 | Mon 10/31/1 | | Josh Blacker,Lae | |

Figure 109: Project Plan Table Expanded Page 1



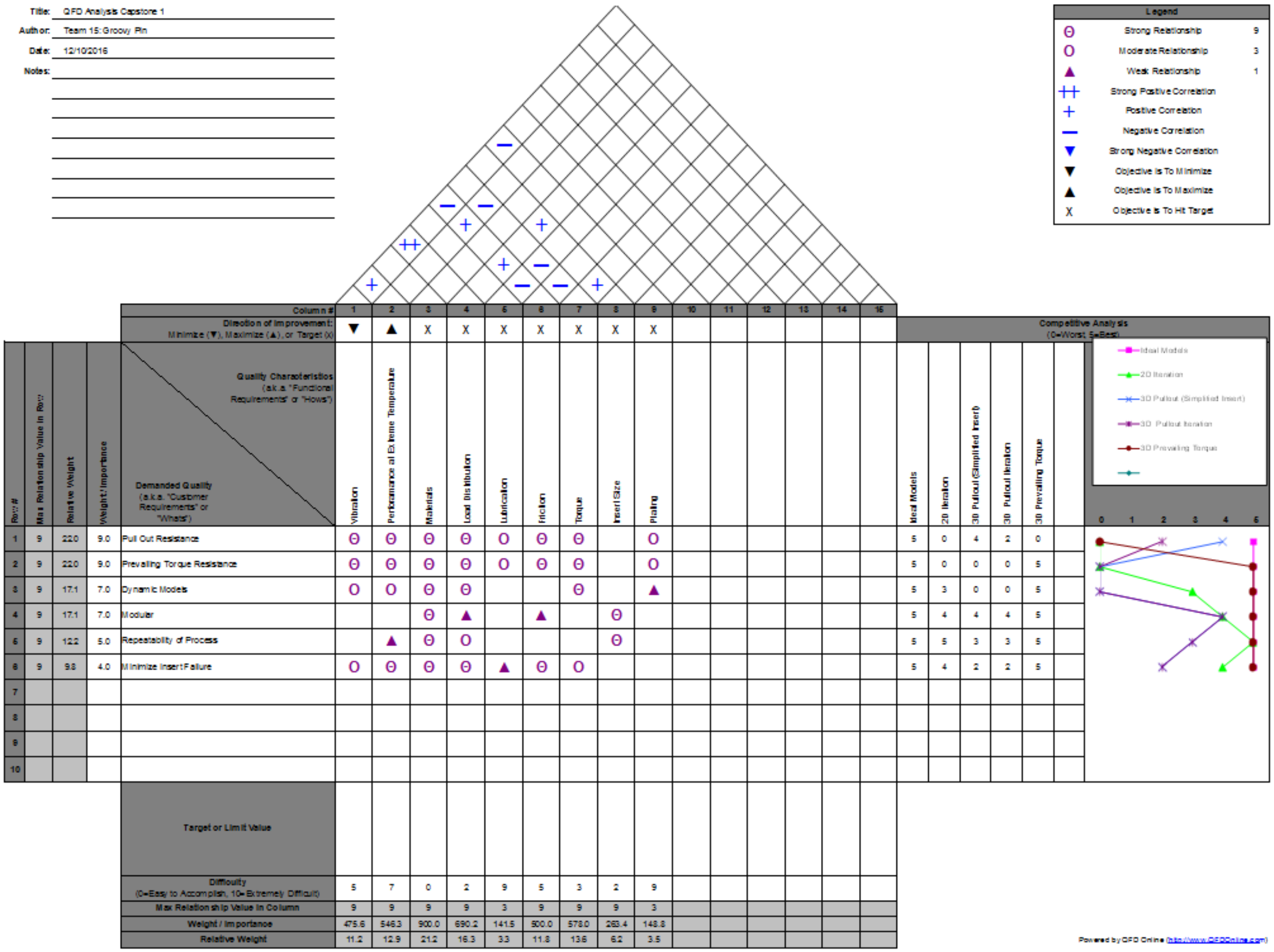
| | ℹ | Task Name | Duration | Start | Finish | Predecessors | Resource Names | Succ |
|---|---|---|---|---|---|---|---|---|
| 39 | ▦ 👤 | Design Review Meeting 3 | 1 hr | Wed 4/12/1 | Wed 4/12/1 | | Josh Blacker,Lae | |
| 40 | ✓ | Achieved Working Pullout Simulation | 0 days | Thu 12/1/16 | Thu 12/1/16 | | Josh Blacker,Lae | |
| 41 | ⟳ 👤 | **◢ Abaqus Exercises** | **35.5 days** | **Mon 10/17/1** | **Mon 12/5/1** | | **Josh Blacker,Lae** | |
| 42 | ✓ | Abaqus Exercises 1 | 2 hrs | Mon 10/17/1 | Mon 10/17/1 | | Josh Blacker,Lae | |
| 43 | ✓ | Abaqus Exercises 2 | 2 hrs | Mon 10/24/1 | Mon 10/24/1 | | Josh Blacker,Lae | |
| 44 | ✓ | Abaqus Exercises 3 | 2 hrs | Mon 10/31/1 | Mon 10/31/1 | | Josh Blacker,Lae | |
| 45 | ✓ | Abaqus Exercises 4 | 2 hrs | Mon 11/7/1 | Mon 11/7/1 | | Josh Blacker,Lae | |
| 46 | ✓ | Abaqus Exercises 5 | 2 hrs | Mon 11/14/1 | Mon 11/14/1 | | Josh Blacker,Lae | |
| 47 | ✓ | Abaqus Exercises 6 | 4 hrs | Mon 11/21/1 | Mon 11/21/1 | | Josh Blacker,Lae | |
| 48 | ✓ | Abaqus Exercises 7 | 4 hrs | Mon 11/28/1 | Mon 11/28/1 | | Josh Blacker,Lae | |
| 49 | ✓ | Abaqus Exercises 8 | 4 hrs | Mon 12/5/1 | Mon 12/5/1 | | Josh Blacker,Lae | |
| 50 | ✓ | Groov-Pin Meeting at Facility | 3 hrs | Thu 11/17/1 | Thu 11/17/1 | | Josh Blacker,Lae | |
| 51 | ✓ | Preliminary Design Report | 7 days | Mon 12/12/1 | Tue 12/20/1 | | Josh Blacker,Lae | |
| 52 | ▦ 👤 | Meeting with Dr. Taggart 2 | 1 hr | Tue 11/22/1 | Tue 11/22/1 | | Josh Blacker,Lae | |
| 53 | ▦ 👤 | Meeting with Dr. Taggart 3 | 1 hr | Tue 2/14/17 | Tue 2/14/17 | | Josh Blacker,Lae | |
| 54 | ▦ 👤 | Meeting with Dr. Taggart 4 | 1 hr | Wed 3/29/1 | Wed 3/29/1 | | Josh Blacker,Lae | |
| 55 | ▦ 👤 | Meeting with Dr. Taggart 5 | 1 hr | Wed 4/19/1 | Wed 4/19/1 | | Josh Blacker,Lae | |
| 56 | ⟳ 👤 | **◢ Project Testing/Simulations** | **65.38 days** | **Wed 2/1/17** | **Wed 5/3/17** | | **Josh Blacker,Lae** | |
| 57 | ✓ | Project Testing/Simulations 1 | 3 hrs | Wed 2/1/17 | Wed 2/1/17 | | | |
| 58 | ✓ | Project Testing/Simulations 2 | 3 hrs | Fri 2/3/17 | Fri 2/3/17 | | | |
| 59 | ✓ | Project Testing/Simulations 3 | 3 hrs | Wed 2/8/17 | Wed 2/8/17 | | | |
| 60 | ✓ | Project Testing/Simulations 4 | 3 hrs | Fri 2/10/17 | Fri 2/10/17 | | | |
| 61 | ✓ | Project Testing/Simulations 6 | 3 hrs | Fri 2/17/17 | Fri 2/17/17 | | | |
| 62 | ✓ | Project Testing/Simulations 7 | 3 hrs | Wed 2/22/1 | Wed 2/22/1 | | | |
| 63 | ✓ | Project Testing/Simulations 8 in ECC | 3 hrs | Fri 2/24/17 | Fri 2/24/17 | | | |
| 64 | ✓ | Project Testing/Simulations 9 in ECC | 3 hrs | Wed 3/1/17 | Wed 3/1/17 | | | |
| 65 | ✓ | Project Testing/Simulations 10 | 3 hrs | Fri 3/3/17 | Fri 3/3/17 | | | |
| 66 | ▦ | Project Testing/Simulations 11 | 3 hrs | Wed 3/8/17 | Wed 3/8/17 | | | |
| 67 | ▦ | Project Testing/Simulations 12 | 3 hrs | Fri 3/10/17 | Fri 3/10/17 | | | |
| 68 | ▦ | Project Testing/Simulations 13 | 3 hrs | Wed 3/15/1 | Wed 3/15/1 | | | |
| 69 | ▦ | Project Testing/Simulations 14 | 3 hrs | Fri 3/17/17 | Fri 3/17/17 | | | |
| 70 | ▦ | Project Testing/Simulations 15 | 3 hrs | Wed 3/22/1 | Wed 3/22/1 | | | |
| 71 | ▦ | Project Testing/Simulations 16 | 3 hrs | Fri 3/24/17 | Fri 3/24/17 | | | |
| 72 | ▦ | Project Testing/Simulations 17 | 3 hrs | Wed 3/29/1 | Wed 3/29/1 | | | |
| 73 | ▦ | Project Testing/Simulations 18 | 3 hrs | Fri 3/31/17 | Fri 3/31/17 | | | |
| 74 | ▦ | Project Testing/Simulations 19 | 3 hrs | Wed 4/5/17 | Wed 4/5/17 | | | |
| 75 | ▦ | Project Testing/Simulations 20 | 3 hrs | Fri 4/7/17 | Fri 4/7/17 | | | |
| 76 | ▦ | Project Testing/Simulations 21 | 3 hrs | Wed 4/12/1 | Wed 4/12/1 | | | |

Figure 110: Project Plan Table Expanded Page 2



| | ℹ | Task Name | Duration | Start | Finish | Predecessors | Resource Names | Succ |
|---|---|---|---|---|---|---|---|---|
| 39 | ▦ 👤 | Design Review Meeting 3 | 1 hr | Wed 4/12/1 | Wed 4/12/1 | | Josh Blacker,Lae | |
| 40 | ✓ | Achieved Working Pullout Simulation | 0 days | Thu 12/1/16 | Thu 12/1/16 | | Josh Blacker,Lae | |
| 41 | ⟳ 👤 | **◢ Abaqus Exercises** | **35.5 days** | **Mon 10/17/1** | **Mon 12/5/1** | | **Josh Blacker,Lae** | |
| 42 | ✓ | Abaqus Exercises 1 | 2 hrs | Mon 10/17/1 | Mon 10/17/1 | | Josh Blacker,Lae | |
| 43 | ✓ | Abaqus Exercises 2 | 2 hrs | Mon 10/24/1 | Mon 10/24/1 | | Josh Blacker,Lae | |
| 44 | ✓ | Abaqus Exercises 3 | 2 hrs | Mon 10/31/1 | Mon 10/31/1 | | Josh Blacker,Lae | |
| 45 | ✓ | Abaqus Exercises 4 | 2 hrs | Mon 11/7/1 | Mon 11/7/1 | | Josh Blacker,Lae | |
| 46 | ✓ | Abaqus Exercises 5 | 2 hrs | Mon 11/14/1 | Mon 11/14/1 | | Josh Blacker,Lae | |
| 47 | ✓ | Abaqus Exercises 6 | 4 hrs | Mon 11/21/1 | Mon 11/21/1 | | Josh Blacker,Lae | |
| 48 | ✓ | Abaqus Exercises 7 | 4 hrs | Mon 11/28/1 | Mon 11/28/1 | | Josh Blacker,Lae | |
| 49 | ✓ | Abaqus Exercises 8 | 4 hrs | Mon 12/5/1 | Mon 12/5/1 | | Josh Blacker,Lae | |
| 50 | ✓ | Groov-Pin Meeting at Facility | 3 hrs | Thu 11/17/1 | Thu 11/17/1 | | Josh Blacker,Lae | |
| 51 | ✓ | Preliminary Design Report | 7 days | Mon 12/12/1 | Tue 12/20/1 | | Josh Blacker,Lae | |
| 52 | ▦ 👤 | Meeting with Dr. Taggart 2 | 1 hr | Tue 11/22/1 | Tue 11/22/1 | | Josh Blacker,Lae | |
| 53 | ▦ 👤 | Meeting with Dr. Taggart 3 | 1 hr | Tue 2/14/17 | Tue 2/14/17 | | Josh Blacker,Lae | |
| 54 | ▦ 👤 | Meeting with Dr. Taggart 4 | 1 hr | Wed 3/29/1 | Wed 3/29/1 | | Josh Blacker,Lae | |
| 55 | ▦ 👤 | Meeting with Dr. Taggart 5 | 1 hr | Wed 4/19/1 | Wed 4/19/1 | | Josh Blacker,Lae | |
| 56 | ⟳ 👤 | **◢ Project Testing/Simulations** | **65.38 days** | **Wed 2/1/17** | **Wed 5/3/17** | | **Josh Blacker,Lae** | |
| 57 | ✓ | Project Testing/Simulations 1 | 3 hrs | Wed 2/1/17 | Wed 2/1/17 | | | |
| 58 | ✓ | Project Testing/Simulations 2 | 3 hrs | Fri 2/3/17 | Fri 2/3/17 | | | |
| 59 | ✓ | Project Testing/Simulations 3 | 3 hrs | Wed 2/8/17 | Wed 2/8/17 | | | |
| 60 | ✓ | Project Testing/Simulations 4 | 3 hrs | Fri 2/10/17 | Fri 2/10/17 | | | |
| 61 | ✓ | Project Testing/Simulations 6 | 3 hrs | Fri 2/17/17 | Fri 2/17/17 | | | |
| 62 | ✓ | Project Testing/Simulations 7 | 3 hrs | Wed 2/22/1 | Wed 2/22/1 | | | |
| 63 | ✓ | Project Testing/Simulations 8 in ECC | 3 hrs | Fri 2/24/17 | Fri 2/24/17 | | | |
| 64 | ✓ | Project Testing/Simulations 9 in ECC | 3 hrs | Wed 3/1/17 | Wed 3/1/17 | | | |
| 65 | ✓ | Project Testing/Simulations 10 | 3 hrs | Fri 3/3/17 | Fri 3/3/17 | | | |
| 66 | ▦ | Project Testing/Simulations 11 | 3 hrs | Wed 3/8/17 | Wed 3/8/17 | | | |
| 67 | ▦ | Project Testing/Simulations 12 | 3 hrs | Fri 3/10/17 | Fri 3/10/17 | | | |
| 68 | ▦ | Project Testing/Simulations 13 | 3 hrs | Wed 3/15/1 | Wed 3/15/1 | | | |
| 69 | ▦ | Project Testing/Simulations 14 | 3 hrs | Fri 3/17/17 | Fri 3/17/17 | | | |
| 70 | ▦ | Project Testing/Simulations 15 | 3 hrs | Wed 3/22/1 | Wed 3/22/1 | | | |
| 71 | ▦ | Project Testing/Simulations 16 | 3 hrs | Fri 3/24/17 | Fri 3/24/17 | | | |
| 72 | ▦ | Project Testing/Simulations 17 | 3 hrs | Wed 3/29/1 | Wed 3/29/1 | | | |
| 73 | ▦ | Project Testing/Simulations 18 | 3 hrs | Fri 3/31/17 | Fri 3/31/17 | | | |
| 74 | ▦ | Project Testing/Simulations 19 | 3 hrs | Wed 4/5/17 | Wed 4/5/17 | | | |
| 75 | ▦ | Project Testing/Simulations 20 | 3 hrs | Fri 4/7/17 | Fri 4/7/17 | | | |
| 76 | ▦ | Project Testing/Simulations 21 | 3 hrs | Wed 4/12/1 | Wed 4/12/1 | | | |

Figure 111: Project Plan Table Expanded Page 2



Title: QFD Analysis Capstone 1

Author: Team 15: Groovy Pin

Date: 12/10/2016

Notes:

| Legend | | |
|---|---|---|
| Θ | Strong Relationship | 9 |
| O | Moderate Relationship | 3 |
| ▲ | Weak Relationship | 1 |
| ++ | Strong Positive Correlation | |
| + | Positive Correlation | |
| − | Negative Correlation | |
| ▼ | Strong Negative Correlation | |
| ▼ | Objective Is To Minimize | |
| ▲ | Objective Is To Maximize | |
| X | Objective Is To Hit Target | |

| | | | | Column # | 1 | 2 | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 | 11 | 12 | 13 | 14 | 15 |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| | | | | Direction of Improvement: Minimize (▼), Maximize (▲), or Target (x) | ▼ | ▲ | X | X | X | X | X | X | X | | | | | | |
| Row # | Max Relationship Value in Row | Relative Weight | Weight / Importance | Quality Characteristics (a.k.a. "Functional Requirements" or "Hows") / Demanded Quality (a.k.a. "Customer Requirements" or "Whats") | Vibration | Performance at Extreme Temperature | Materials | Load Distribution | Lubrication | Friction | Torque | Insert Size | Plating | | | | | | |
| 1 | 9 | 22.0 | 9.0 | Pull Out Resistance | Θ | Θ | Θ | Θ | O | Θ | Θ | | O | | | | | | |
| 2 | 9 | 22.0 | 9.0 | Prevailing Torque Resistance | Θ | Θ | Θ | Θ | O | Θ | Θ | | O | | | | | | |
| 3 | 9 | 17.1 | 7.0 | Dynamic Models | O | O | Θ | Θ | | | Θ | | ▲ | | | | | | |
| 4 | 9 | 17.1 | 7.0 | Modular | | | Θ | ▲ | | ▲ | | Θ | | | | | | | |
| 5 | 9 | 12.2 | 5.0 | Repeatability of Process | | ▲ | Θ | O | | | | Θ | | | | | | | |
| 6 | 9 | 9.8 | 4.0 | Minimize Insert Failure | O | Θ | Θ | Θ | ▲ | Θ | O | | | | | | | | |
| 7 | | | | | | | | | | | | | | | | | | | |
| 8 | | | | | | | | | | | | | | | | | | | |
| 9 | | | | | | | | | | | | | | | | | | | |
| 10 | | | | | | | | | | | | | | | | | | | |
| | | | | Target or Limit Value | | | | | | | | | | | | | | | |
| | | | | Difficulty (0=Easy to Accomplish, 10=Extremely Difficult) | 5 | 7 | 0 | 2 | 9 | 5 | 3 | 2 | 9 | | | | | | |
| | | | | Max Relationship Value in Column | 9 | 9 | 9 | 9 | 3 | 9 | 9 | 9 | 3 | | | | | | |
| | | | | Weight / Importance | 475.6 | 546.3 | 900.0 | 690.2 | 141.5 | 500.0 | 578.0 | 263.4 | 148.8 | | | | | | |
| | | | | Relative Weight | 11.2 | 12.9 | 21.2 | 16.3 | 3.3 | 11.8 | 13.6 | 6.2 | 3.5 | | | | | | |

Competitive Analysis (0=Worst; 5=Best)

| Row # | Ideal Models | 2D Iteration | 3D Pullout (Simplified Insert) | 3D Pullout Iteration | 3D Prevailing Torque | |
|---|---|---|---|---|---|---|
| 1 | 5 | 0 | 4 | 2 | 0 | |
| 2 | 5 | 0 | 0 | 0 | 5 | |
| 3 | 5 | 3 | 0 | 0 | 5 | |
| 4 | 5 | 4 | 4 | 4 | 5 | |
| 5 | 5 | 5 | 3 | 3 | 5 | |
| 6 | 5 | 4 | 2 | 2 | 5 | |
| 7 | | | | | | |
| 8 | | | | | | |
| 9 | | | | | | |
| 10 | | | | | | |

Powered by QFD Online (http://www.QFDOnline.com)

Figure 112: Quality Function Deployment Chart