

**GUIDED INQUIRY FOR IMPROVING SOLIDWORKS CERTIFICATION
EXAM SCORES AND APPLYING SURFACING TECHNIQUES TO
SIMPLIFY THE DESIGN AND MANUFACTURING PROCESS OF
FUSEES**

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Abstract

Since 2013, the University of Idaho has used SOLIDWORKS certification exams, proctored by Dassault Systèmes, to help gauge the quality of the university's CAD curriculum. The course *Solid Modeling, Simulation and Manufacturing Capstone* prepares students to take seven SOLIDWORKS certification exams. This paper analyzes 140 Certified SOLIDWORKS Advanced Professional Surfacing exams taken by 100 University of Idaho students from Fall 2017 to Fall 2021 and 37 Certified SOLIDWORKS Expert exams taken by University of Idaho students from Fall 2019 to Fall 2021. The preparatory material from both exams was changed during the spring of 2020. The new preparatory material was designed around guided inquiry questions associated with models of challenging CAD parts to augment the previous tutorials. Impact of this new pedagogy was tracked in exams during the Spring 2021 and Fall 2021 semesters. Exam pass rates dramatically increased as did points acquired per minute during for those taking the surfacing and Expert exam. During the Spring 2021 and Fall 2021 semesters the ratio of guided inquiry to tutorial work increased progressively throughout the course, giving students greater and greater self-confidence as well as self-reliant in their CAD development. In retrospect, the previous use of only tutorials was associated with an implicit ceiling in CAD competency.

This thesis also explores how the techniques taught in the surfacing and expert sections of *the Solid Modeling, Simulation, and Manufacturing Capstone* course can be applied to the research and design of mechanical clocks. Specific attention was given to a mechanism called a fusee which normalizes the torque curve of a spiral spring to ensure that the clock keeps time correctly. Techniques from the surfacing and expert section of the *Solid Modeling, Simulation, and Manufacturing Capstone course* were used to develop a fusee design tool. A system was developed that allows the users to input variables into Microsoft Excel, then using TK Solver, Excel, and SOLIDWORKS the tool automatically generates a SOLIDWORKS model of a fusee and G-code for a Hass CNC lathe to manufacture the threads of a fusee for a specific spiral spring and torque output requirements. Fusee parts created using the methods in this thesis produced consistent thread depth and pitch $\pm .002$ for each revolution. These features help prevent the wire from slipping out of position and

promote constant torque output. This innovation significantly reduces the role of trial and error in obtaining an acceptable fusee, especially for hobby clockmakers.

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Dedication

First and foremost, I would like to thank my father Michel James McDonnell. In his room, there is a slip of yellow paper taped to the wall with scotch tape. It reads “Nothing in the world can take the place of persistence.” A quote by Calvin Coolidge. He exemplifies this quote. When he sets his mind to a thing, it gets done. Whenever I was up late working and thought about throwing in the towel, I would think “what would Iron Mike do?” Then I would get back to work. Dad, you are my direct inspiration. Thank you for everything.

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Chapter 1: Introduction

1.1: Introduction

Computer-aided design is systemic in mechanical engineering today. Thus, it is important to ensure a high quality of computer-aided design education. Since 2013, the University of Idaho has used SOLIDWORKS certification exams, proctored by Dassault Systèmes, to determine the quality of computer-aided design education at the university. The course *Solid Modeling, Simulation, and Manufacturing Capstone* taught at the University of Idaho prepares students to take seven SOLIDWORKS certification exams. This thesis will describe methods and examples used to prepare students to take the Certified SOLIDWORKS Advanced Professional Surfacing (CSWPA-SU) exam and the Certified SOLIDWORKS Expert in Mechanical Design exam (CSWE-MD). An analysis of 140 CSWPA-SU exams and 37 CSWE exams taken between Fall of 2017 to Fall of 2021 will also be conducted to help determine optimal computer-aided design pedagogy strategies.

Furthermore, this thesis will explore how the techniques taught in the surfacing and expert sections of the *Solid Modeling, Simulation, and Manufacturing Capstone course* can be applied to the research and design of mechanical clocks. Specifically, the design and manufacturing of a mechanism called a fusee. A fusee is a mechanism to normalize the torque curve of a spiral spring used in mechanical clocks to ensure that the clock keeps constant time. Techniques from the surfacing and expert section of the *Solid Modeling, Simulation, and Manufacturing Capstone course* were used to develop a fusee design tool. It allows the users to input variables into Microsoft Excel, then using TK Solver, Excel, and SOLIDWORKS the tool automatically generates a SOLIDWORKS model of a fusee and G-code for a Hass CNC lathe to manufacture the threads of a fusee for a specific spiral spring and torque output requirements.

1.2: Literature Review

When the *Solid Modeling, Simulation, and Manufacturing Capstone course* began teaching surfacing, the books for the class consisted of *Certified SOLIDWORKS Professional Advanced Preparation Material*, and *SOLIDWORKS 2017 Advanced Techniques* by Paul

Tran [1] [2]. In these books, Tran brings the student through a series of tutorials illustrating how to create various complex features in SOLIDWORKS. Another book written by Tran, *Certified SOLIDWORKS Expert Preparation Materials*, was later used to prepare students for the CSWE. These books are what inspired the university to create its own set of workbooks and were a pivotal step for the advanced computer-aided design community at the University of Idaho.

In the book *How to Make a Lyre Skeleton Clock*, master clock maker W.R. Smith depicts how to build a clock that he designed [3]. In it, he illustrates a method for manufacturing a mechanism called a fusee. A fusee is a nonlinear threaded cone used to normalize the torque output of a spiral spring in a barrel. This method for manufacturing a fusee is what inspired the G-code aspect of the fusee design tool.

In 1977 the Horological Journal published *Fusee Theory* [4]. In it, the mathematician Preisendorfer derives an equation to determine the profile of the fusee based on inputs such as the angle of twist, the outside radius of the barrel, required torque to turn the gear train of the clock, and pitch of the fusee. The inclusion of the pitch input variable is what makes his derivation unique and believed to be the most accurate. His equation is what is used in the fusee design tool.

In *An Analysis of the Spiral Spring* W. Swift presents a new method to analyze spiral springs [5]. His research determines that a free spiral spring forms a logarithmic spiral. His methods for determining the shape of a free spiral spring were modernized and used to extrapolate the spring constant of the spring driving the Lyre Skeleton Clock.

In *Finding Displacements Using Castigliano's Theorem, LaGrange Multipliers, Heaviside Step Functions, and Numerical Integration* Selso Gallegos creates a design tool using a generalized form of Castigliano's theorem to determine the stress of a flatbed semi-truck trailer and generate a 3D model. The design tool consisted of a system of programs including Microsoft Excel, TK Solver, and Autodesk Inventor. This design tool inspired the fusee design tool depicted in chapter 4.

Chapter 2: Methods for teaching Advanced SOLIDWORKS and Manufacturing

2.1: Introduction

Computer-aided Design (CAD) is the use of computers to generate 3D or 2D digital models to create, modify, analyze, and optimize designs [6]. SOLIDWORKS is a computer-aided design software owned by Dassault Systèmes with more than six million users [7]. Since 2013, the University of Idaho has used SOLIDWORKS certification exams, proctored by Dassault Systèmes, to help gauge the quality of the university's CAD curriculum. The course *Solid Modeling, Simulation and Manufacturing Capstone* prepares students to take seven SOLIDWORKS certification exams. The exams include the Certified SOLIDWORKS Professional – Mechanical Design Segments 1, 2, and 3, the Certified SOLIDWORKS Advanced Professional Drawing Tools, Sheet Metal, Weldments, and Surfacing exams, as well as the Certified SOLIDWORKS Expert in Mechanical Design exam. This chapter will describe methods and examples used to prepare students to take the Certified SOLIDWORKS Advanced Professional Surfacing (CSWPA-SU) exam and the Certified SOLIDWORKS Expert in Mechanical Design exam (CSWE-MD), as well as an analysis of data collected on the exam results. All the other exams covered in the class will be reviewed and analyzed by Ryan “Jack” Gonzalez in *Method to Accelerate Student Proficiency in CSWP/CSWPA Solidworks Certification Exams and Fusee Mechanism Profile Analysis* [8].

The study consists of 140 CSWPA-SU exams taken by 100 University of Idaho students from Fall 2017 to Fall 2021 and 37 CSWE exams taken by University of Idaho students from Fall 2019 to Fall 2021. The preparatory material from both exams was changed during the spring of 2020. The goal of this study is to analyze the effects of those changes.

2.2: History of Solid Modeling at the University of Idaho

In 2013 Dr. Edwin Odom wanted to gauge the quality of the CAD program at the University of Idaho by scheduling students to take the Certified SOLIDWORKS Associate Exam (CSWA-MD) exam. To give some background on the CSWA exam, it is the easiest SOLIDWORKS certification and tests students on basic SOLIDWORKS skills like Sketch entities, Boss Extrudes, and Cuts, as well as Mass properties [9]. It is a prerequisite for all the certification exams currently taken in *Solid Modeling, Simulation, and Manufacturing Capstone* in 2022. The exam results were positive, with a pass rate of 90% during the fall of 2013.

Over the past eight years, SOLIDWORKS certification exams have been used to analyze the quality of the CAD pedagogy at the University of Idaho. In the spring of 2015 students started to attempt the second and third segments of the CSWP and by the fall of 2015 students were becoming Certified SOLIDWORKS Professionals.

Figure 2.1 depicts the pass rate of each certification exam by semester. Note that the CSWA, all three segments of the CSWP, and all the CSWPA exams can be retaken after a 14-day waiting period between attempts [10]. Figure 2.1 counts all exam attempts equally, meaning first attempts and retakes affect the pass rate of the semester equally. The figure is also lacking in its analysis of the number of attempts per semester. For example, in the spring of 2018, only two students took the CSWPA-SU exam, meaning the pass rates of that exam in that semester gives a very low-resolution analysis of the teaching methodologies. Considering these issues, the CSWPA-SU and the CSWE exam results will be analyzed further in sections 2.5 and 2.7. Figure 2.1 does illustrate one thing quite concisely; over the past eight years, the University of Idaho's mechanical engineering department has steadily increased the number of certification exams that its students can pass in a semester. Thereby illustrating their increase in CAD ability and increasing their employment marketability.

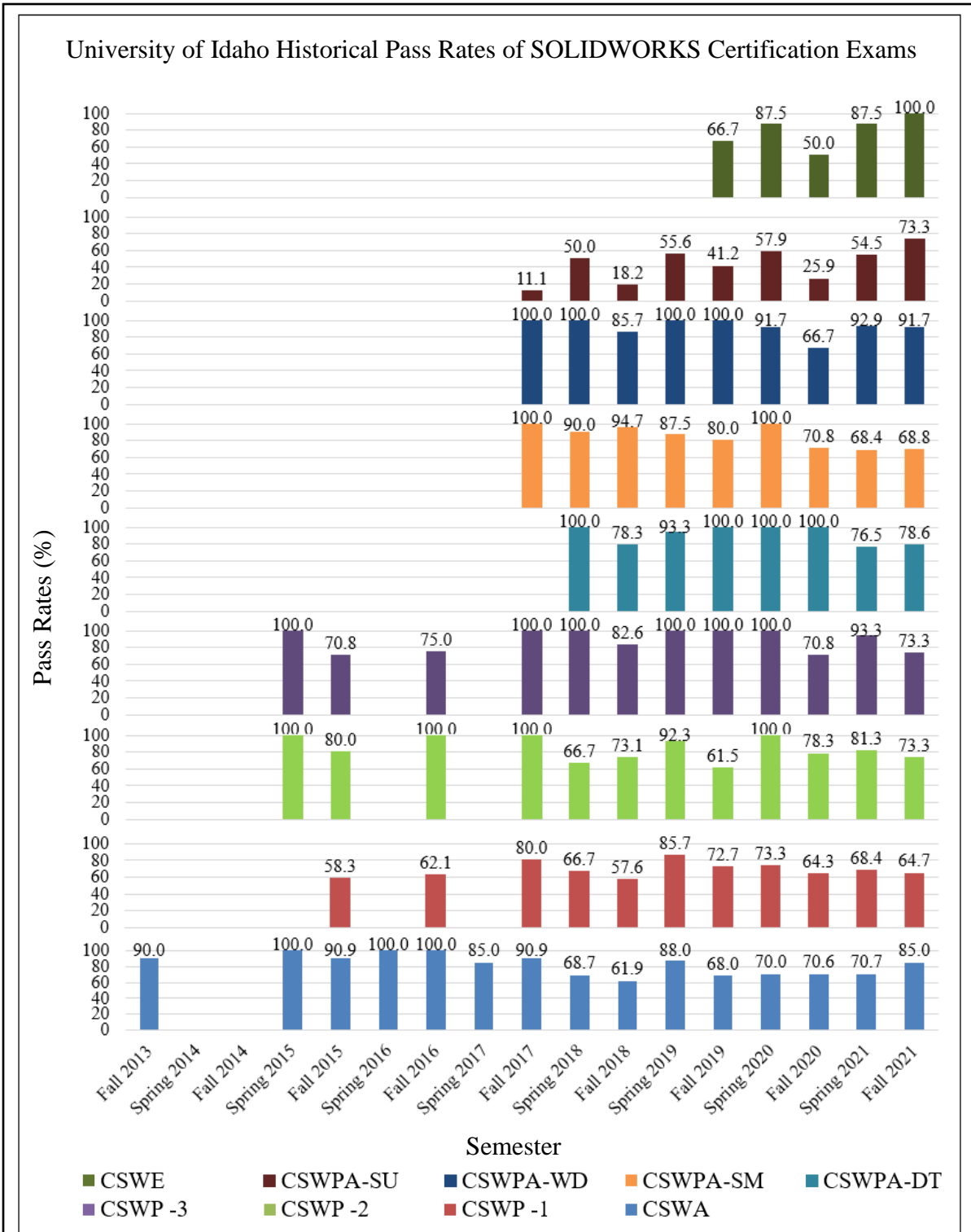
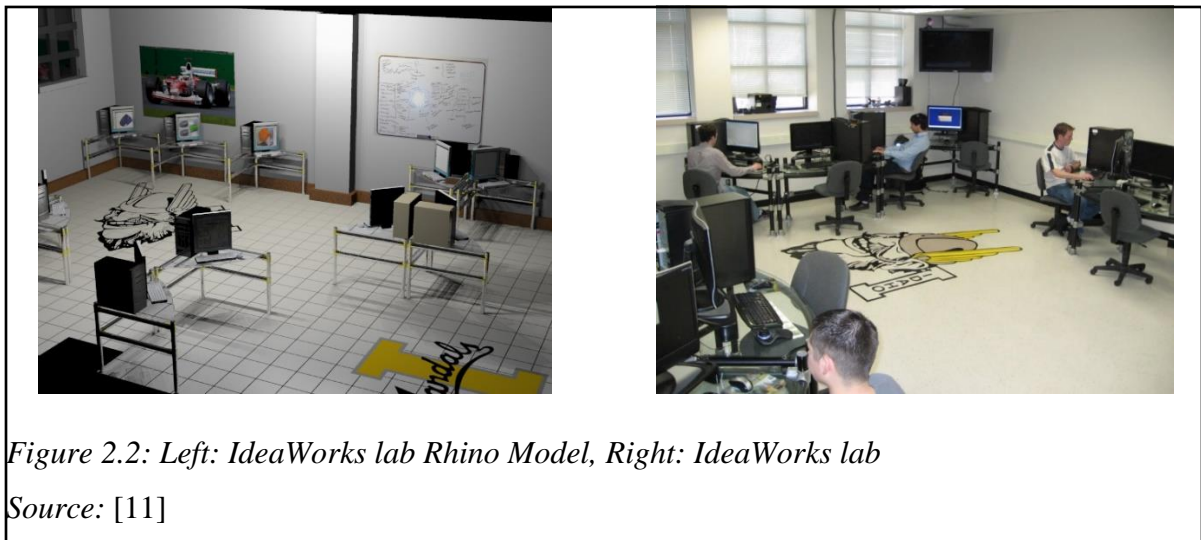


Figure 2.1: University of Idaho Pass Rates of SOLIDWORKS Certification Exams

2.3: Learning Environment, Teaching Methods, and Course Objective

Solid Modeling, Simulation, and Manufacturing Capstone is taught in the IdeaWorks Lab (Figure 2.2). Originally designed by Alexander Odom (left image of Figure 2.2), the lab was designed to increase the amount of contact that students have with each other. This worked by turning computer monitors towards the center of the room, meaning students would only have to turn around to see how everyone else was solving a problem. It also allows instructors to quickly assess the students which are struggling. The instructor can then proceed to help that student or prompt another student to help them.



Solid Modeling, Simulation and Manufacturing Capstone has a simple structure. On Mondays and Wednesdays, instructors give a lecture to students on CAD techniques. These lectures range anywhere from fifteen to twenty-five minutes. The rest of class is devoted to in-class exercises, homework, and quizzes. On Fridays, instructors run a machine shop lab where students learn basic skills on the vertical and horizontal bandsaws, drill press, lathe, vertical mill, and CNC lathe throughout the semester. The idea, especially early on is to give the students as much time to interact with each other as possible such that they can make use of the IdeaWorks lab and teach each other.

As the semester progresses students will be provided less and less direct instruction on how to solve a problem and be required to critically think to solve problems. This dichotomy is illustrated well in the differences between the Surfacing and Expert sections of

the class. During the CSWPA-SU section, students are provided with tutorials and example problems. The tutorials provide the students with the exact buttons they need to press to solve the problems and the example problems are very similar to the tutorials. This is done because students have no experience with SOLIDWORKS surfacing and do not even know what features exist. During the CSWE section students are only provided with example problems and video solutions. At this point in the semester, students are expected to be highly skilled in all mechanical design facets of the software and as such are expected to critically think and solve problems without instructor intervention.

The goal of *Solid Modeling, Simulation, and Manufacturing Capstone* is to provide students with a basic understanding of drafting and manufacturing techniques, improve their problem-solving capability, increase their employment marketability, and create an environment in the mechanical engineering department that perpetuates excellent CAD technique.

2.4: SOLIDWORKS Surfacing Pedagogy

Surfacing is a method for breaking up solid modeling into smaller steps to accomplish the same task. Imagine trying to model a box. In solid modeling one simply draws a square and extrude the square a distance equal to the side length of the square, thus creating all six faces in one step. In surface modeling, one must create every face individually and then knit them together to create a solid model. This, at first glance, seems like a disadvantageous form of modeling. In many cases, it is. Surfacing becomes advantageous in cases where the geometry of a model is so complicated that it is easier to break up the model into smaller steps. In mathematics, if a problem is complicated the mathematician will break up the problem into smaller steps and solve them one at a time. Much in the same way, in modeling, if the part is complicated the modeler will break up the model into faces and create them one at a time.

The topics covered in the SOLIDWORKS surfacing section of *Solid Modeling, Simulation and Manufacturing Capstone* include Swept Surface, Lofted Surface, Boundary Surface, Filled Surface, Planar Surface, Offset Surface, Ruled Surface, Delete Face, Extend Surface, Trim Surface, Untrim Surface, Knit Surface, Thicken, Thicken Cut, Cut with

Surface, Split Line, Projected Curve, and Curve Through Reference Points. At the end of this segment of the class, students are tasked with taking the ninety-minute, CSWPA-SU exam [12]. The next few pages will give a basic rundown of how these topics are currently covered in the class. For the complete *SOLIDWORKS Surfacing Workbook*, refer to Appendix A. Some of the models displayed in this thesis and the *SOLIDWORKS Surfacing Workbook* were originally found on GRABCAD and then heavily modified for this class [13].

The *SOLIDWORKS Surfacing Workbook* consists of two parts. One is the tutorials, and two are the example problems. At this point in the semester, the students have little to no experience with *SOLIDWORKS* surfacing. The tutorials aim to teach students where the buttons are in the *SOLIDWORKS* interface. This is important because it gives them a document that they can refer to, to remember where things are. The example problems give students a chance to critically think and apply the features illustrated to them in the tutorials.

The first thing covered in the *SOLIDWORKS Surfacing* Section is a classic example of when to use surface modeling. This example includes detailed instruction on how to make the part illustrated in the right image of Figure 2.3 using only the sketch geometry illustrated in the left image of Figure 2.3. It shows a surface modeling technique to create the part as well as illustrations of failed attempts using solid modeling techniques like Lofting and Sweeping. It introduces them to the idea that surfaces are construction geometry and teaches them to use the Boundary surface.

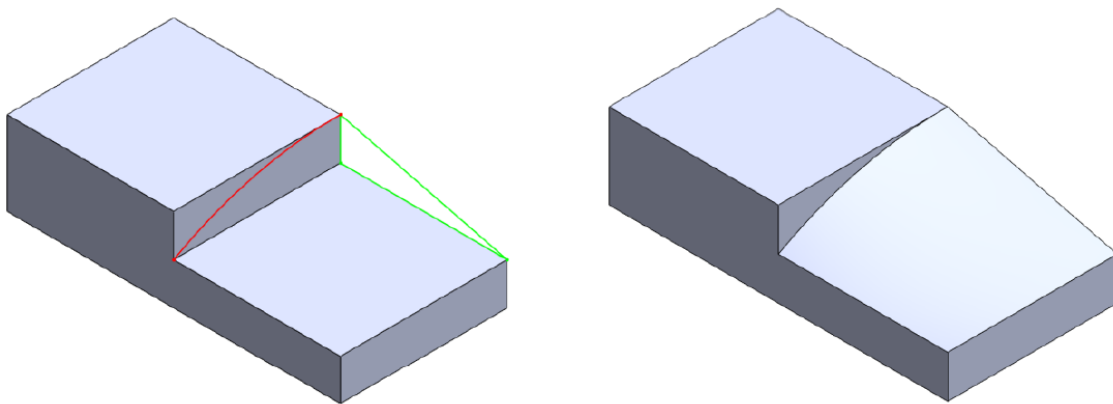


Figure 2.3: Left: Starting Model, Right: Complete Model in the Why Surfacing Tutorial in the SOLIDWORKS Surfacing Workbook (See Appendix A)

The example illustrated in Figure 2.4 is a basic introduction to Surface Lofting, Project Curve, Planar Surface, and Knit Surface. By the end of the tutorial, the student should be able to isolate faces of a model create the geometry and knit them together. The CAD file for this tutorial was adapted from a model originally found on GrabCAD [14].

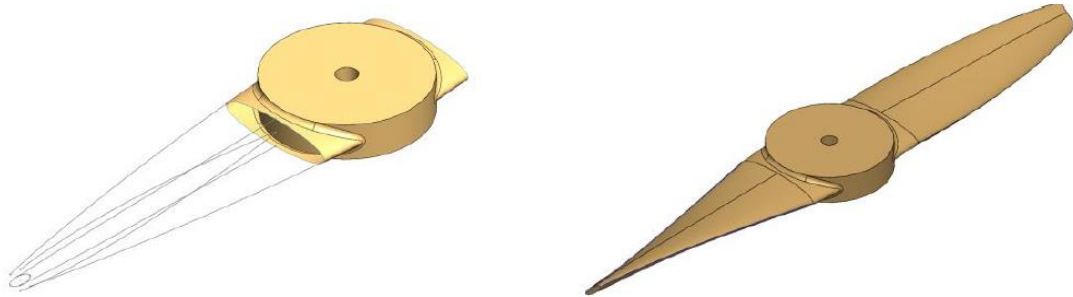


Figure 2.4: Left: Starting Model, Right: Complete Model in the Lofted Surfaces Tutorial in the SOLIDWORKS Surfacing Workbook (See Appendix A)

The next topic covered in the section is the difference between Boundary Surface and Lofted Surface. The example illustrated in Figure 2.5 clarifies the difference between Lofted Surface and Boundary Surface as well as introduces students to the Selection Manager and how to use it. It is important to illustrate these differences, so students know when it is advantageous to use one tool versus the other tool. The CAD file for this tutorial was adapted from a model originally found on GrabCAD [15].



Figure 2.5: Left: Starting Model, Right: Complete Model in the Boundary Surface vs. Lofted Tutorial in the SOLIDWORKS Surfacing Workbook (See Appendix A)

Next, an introduction to Swept Surfaces as well as the difference between Swept Surfaces and Centerline Lofted Surfaces is explored (Figure 2.6). This section is used to depict the difference between the tool interfaces as well as spark a discussion on the way the two tools generate surfaces.

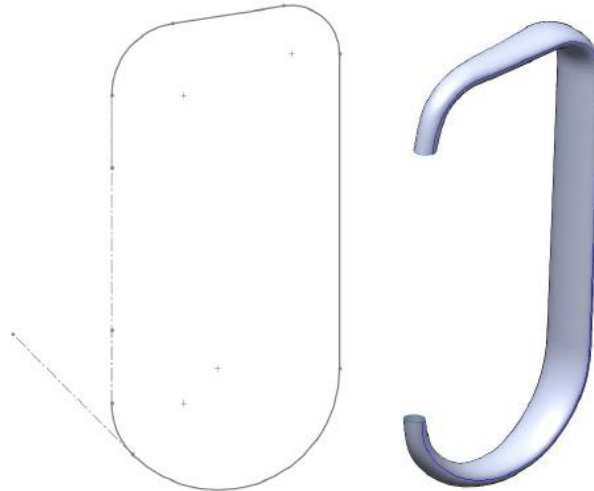


Figure 2.6: Left: Starting Model, Right: Complete Model in the Swept Surface Tutorial in the SOLIDWORKS Surfacing Workbook (See Appendix A)

Drafting is an operation that can be performed in a plethora of different ways in SOLIDWORKS, some of which include surfacing techniques. Through a couple of examples, including the example shown in Figure 2.7, different methods for creating draft angles are explored and illustrated. Some methods include Extruded Boss/Base Draft on, and Extruded Surface Draft on, DraftXpert, and Ruled Surface (Tapered to Vector). The CAD file for this question was adapted from a model originally found on GrabCAD [16].

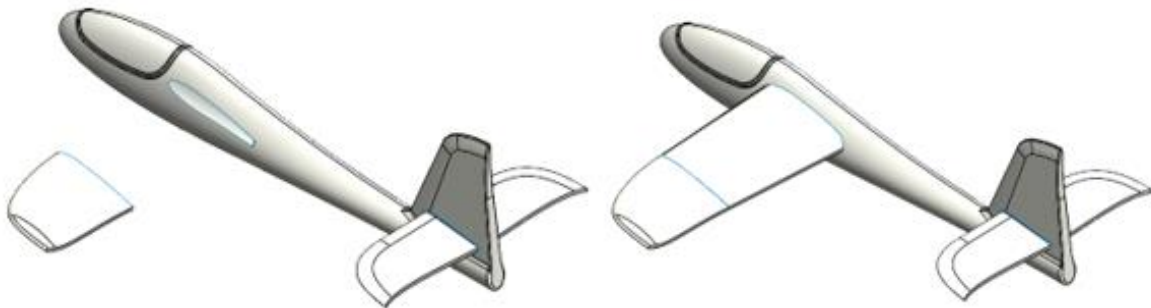


Figure 2.7: Left: Starting Model, Right: Complete Model of Practice Problem 3 in the SOLIDWORKS Surfacing Workbook (See Appendix A)

Creating inlays on a curved face in SOLIDWORKS is a complicated process without using surface modeling techniques. This section covers (example in Figure 2.8) a technique to create an inlay using a 2D sketch, Split line, Offset/Copy Surface, and Thicken Cut. The CAD file for this question was adapted from a model originally found on GrabCAD [17].

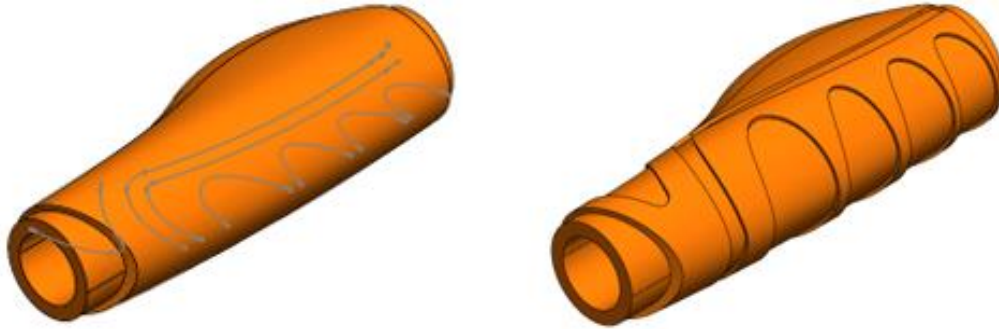


Figure 2.8: Left: Starting Model, Right: Complete Model of Practice Problem 4 in the SOLIDWORKS Surfacing Workbook (See Appendix A)

Mold Tools is a SOLIDWORKS Professional Advanced subject exam all on its own. This example (Figure 2.9) covers a couple of different methods for creating a cast from the core and cavity without using any Mold Tool features. One method to solve this problem is to copy the inside surface of the mold, knit the surfaces together, and create a solid body. The other method is to create an extrude then used the Combine tool to subtract the core and cavity from the extrusion. The surface modeling technique to solve this problem is tedious, as it requires the modeler to select a lot of faces, whereas the solid modeling technique is quite elegant. This example is used to illustrate the limits of modeling one face at a time and illustrate that solid modeling is advantageous most of the time.

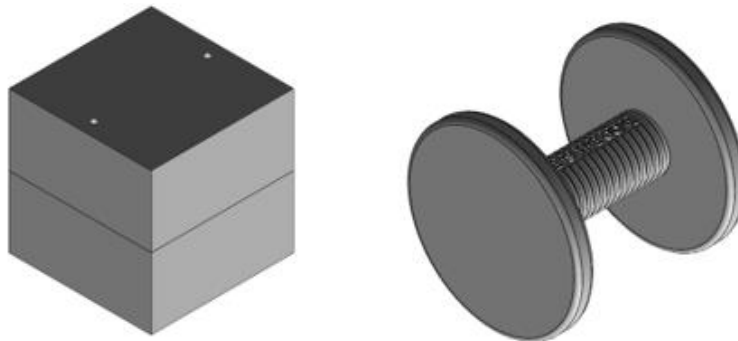


Figure 2.9: Left: Starting Model, Right: Complete Model in the Copy Surface and Combine Tutorial in the SOLIDWORKS Surfacing Workbook (See Appendix A)

Opening geometry in SOLIDWORKS that was created on a different modeling software SOLIDWORKS generates a pop-up menu asking if the modeler wants to use Feature Recognition. Feature Recognition will delete features that are not supported by SOLIDWORKS meaning that in some cases using Feature Recognition will delete critical features. If the modeler is unable to use feature recognition, for this reason, SOLIDWORKS instead generates what is called Imported Geometry. The feature history of this geometry cannot be modified. A situation like this is explored in the example below (Figure 2.10). To solve the problem students must use Delete Face and Untrim Surface. The CAD file for this question was adapted from a model originally found on GrabCAD [18].

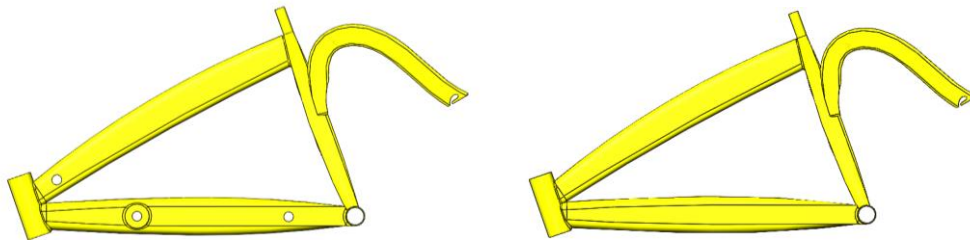


Figure 2.10: Left: Starting Model, Right: Complete Model of Practice Problem 7 in the SOLIDWORKS Surfacing Workbook (See Appendix A)

2.5: Certified SOLIDWORKS Advanced Professional Surfacing (CSWPA-SU) Exam Results

In the fall of 2017 the University of Idaho *Solid Modeling, Simulation and Manufacturing Capstone* class decided to attempt the CSWPA-SU. Out of the nine attempts that semester, one student passed (Figure 2.11). One thing to note about the Professional and Professional Advanced exams is that they can be retaken after a waiting period of 14 days. The single successful attempt that semester was on a retake of the exam. As our teaching methods and material have evolved the pass rates have gone up. The following study consists of 140 CSWPA-SU exams taken by 100 University of Idaho students from Fall 2017 to Fall 2021. Before each exam, the students were asked if they were willing to release their exam data to Dr. Edwin Odom. All students in this study agreed to do so. The data was collected by Dassault Systèmes, then made anonymous, accessed, and analyzed by the University of Idaho through the 3DEXPERIENCE Certification Center.

From Fall of 2017 to Fall of 2020 the class was using the surfacing sections from *Certified SOLIDWORKS Professional Advanced Preparation Materials* by Paul Tran to teach SOLIDWORKS surfacing [19]. From Fall of 2017 to Fall of 2020 pass rate and average scores increased as the instructors became more and more familiar with the subject matter.

Traditionally, the fall semester performs worse than the spring semester on the CSWPA-SU exam. From Fall of 2017 to Spring of 2020 the surfacing section of the class was two weeks. It is hypothesized that the lower scores in the fall were because Thanksgiving break fell in the middle of the two-week training period. From Fall of 2020 forward, the schedule was shifted such that the section took three weeks of the semester, and the Thanksgiving break did not fall in the middle of the training period.

During spring 2020 the surfacing section of *Solid Modeling, Simulation, and Manufacturing Capstone* was taught online due to COVID 19 restriction at the University of Idaho [20]. In the Fall of 2020, the class was taught both in-person and online. For this semester students spent part of the week in class and the other part of the week online. This is because the IDEAWORKS lab has a maximum capacity of thirteen including the instructor and by university policy that semester in-person classes were only allowed 50 percent maximum capacity [21]. Research indicates “that there is a positive relationship between the digital tools and academic performance during COVID-19” [22]. However, since students were no longer able to interact, the class dynamics shifted towards a more classical learning environment. This is hypothesized to have had a negative impact on the Spring 2020 class and be the primary reasoning behind the poor pass rates of both the CSWPA-SU and the CSWE in fall 2020.

During the spring of 2021, the *SOLIDWORKS Surfacing Workbook* depicted in 2.4 SOLIDWORKS Surfacing Pedagogy was implemented into the class. Due to the small sample size, it is hard to determine what effect the material has had on learning outcomes. However, all the data seems to imply that the material has positively impacted the class (see Figure 2.11, 2.12, 2.13). The main difference between this problem set and the set offered in *Certified SOLIDWORKS Professional Advanced Preparation Materials* is that the homework

problems are more diverse. This means that there are some tutorial-style homework and some question-answer-style homework.

Figure 2.11 shows the University of Idaho’s pass rates by semester for the CSWPA-SU. This plot does not differentiate between the first attempts and second attempts of the exam, which will be explored later in this chapter. A linear regression of this plot yields a trend line with a slope of 4.67 and an R-squared value of 38.42%. The R-squared value seems a little low however “Any study that attempts to predict human behavior will tend to have R-squared values less than 50%” [23]. This slope illustrates that over the past four years the CSWPA-SU scores at the University of Idaho have increased by about five percent per semester.

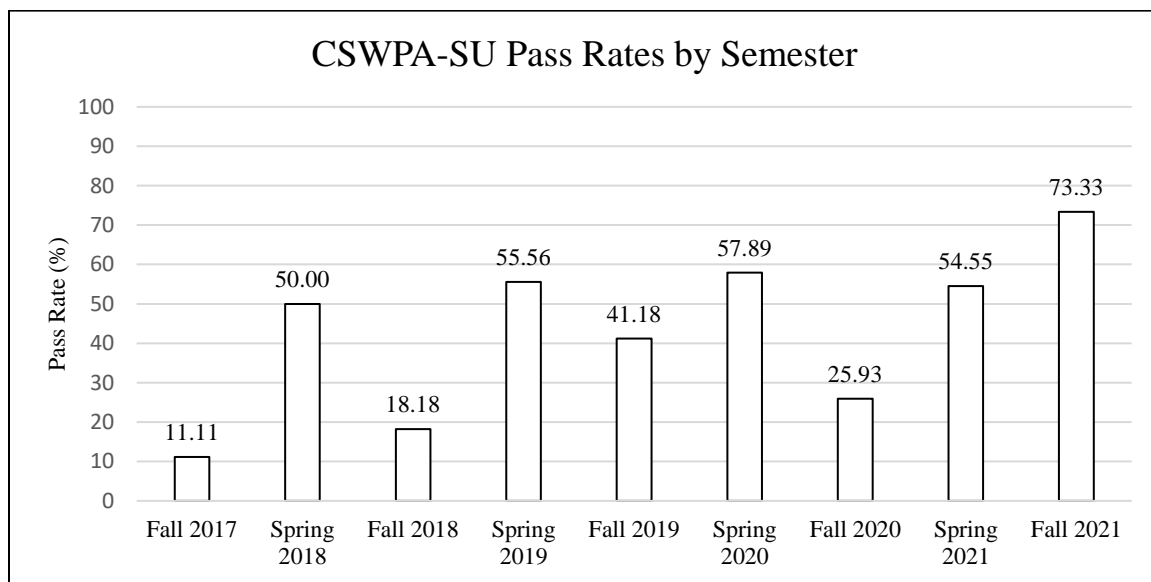


Figure 2.11: Certified SOLIDWORKS Professional Advanced Surfacing Exam Pass Rates (%) from Fall 2017 to Fall 2021

The number of passes (in white) and fails (in red) per semester is illustrated in Figure 2.12. This in many ways illustrates the same information as depicted in Figure 2.11, however, it also illustrates the number of attempts per semester. Note that the number of successful attempts is limited to the number of people enrolled in the class. For example, in the Fall of 2021, there were eleven students enrolled in the class and eleven people who passed.

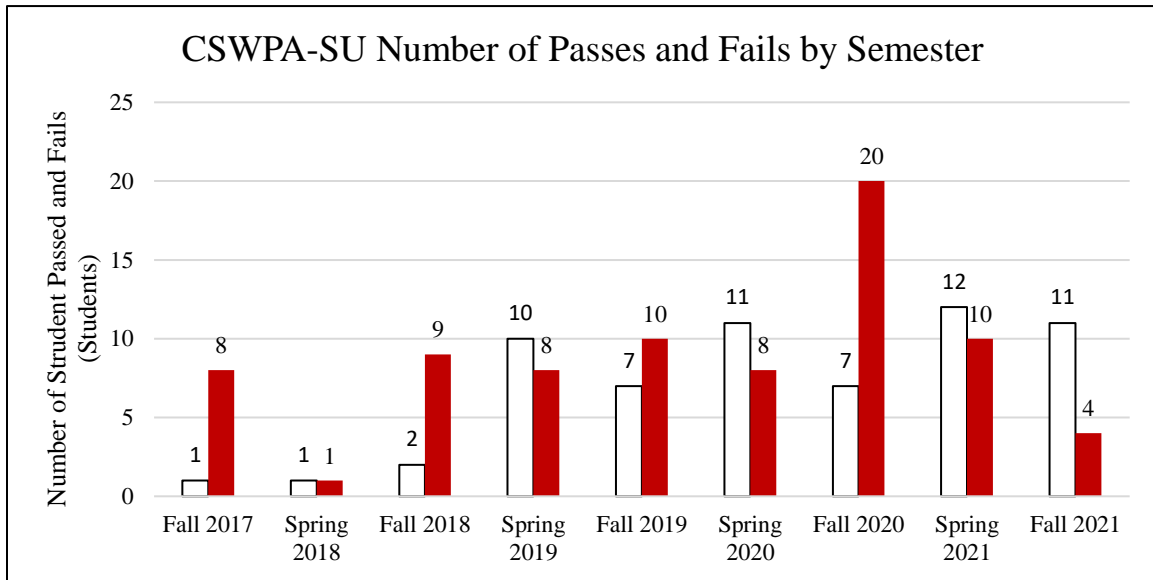


Figure 2.12: Certified SOLIDWORKS Professional Advanced Surfacing Exam Number of Passes and Fails per Semester from Fall 2017 to Fall 2021

Figure 2.13 is a box and whisker plot of the 140 CSWPA-SU exams sorted by academic semester. Note that in the spring of 2018 class only had two attempts throughout the whole semester. The red line indicates the minimum score needed to pass the exam.

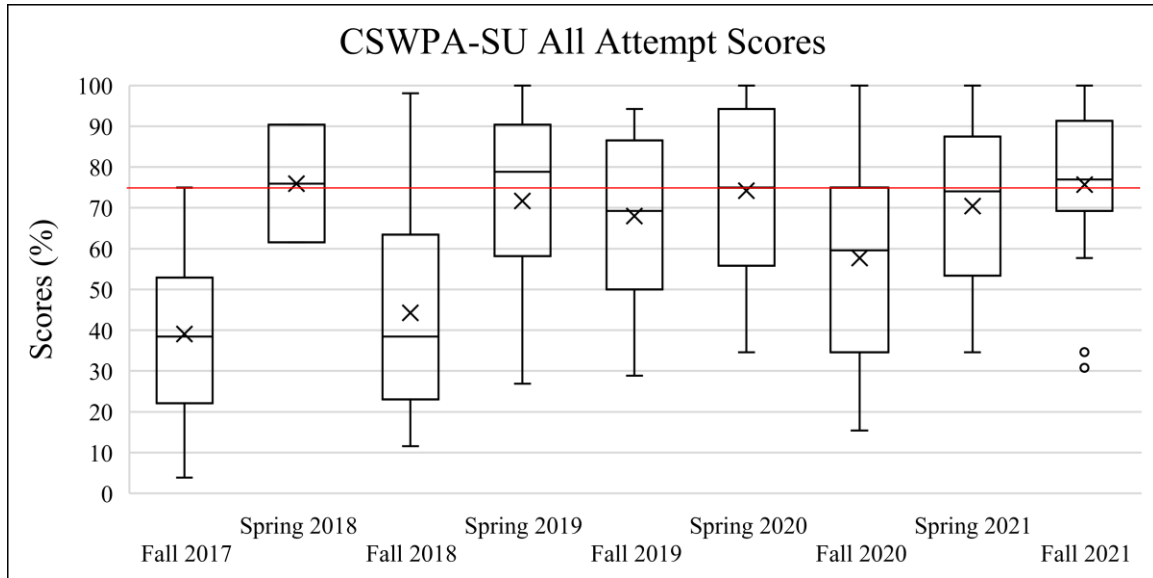


Figure 2.13: Box and Whisker Plot of All Certified SOLIDWORKS Professional Advanced Surfacing Exam Scores (%) from Fall 2017 to Fall 2021

Figure 2.14 was developed in the hopes of quantifying modeling speed. The box and whisker plot below shows the number of points the students received on the exam divided by the number of minutes the student took to complete the exam on the Y-axis and the academic semester along the X-axis. The plot is by no means an indicator of successful mastery of the subject as students could theoretically complete one easy question on the exam, then submit the exam, and receive a high score per minute.

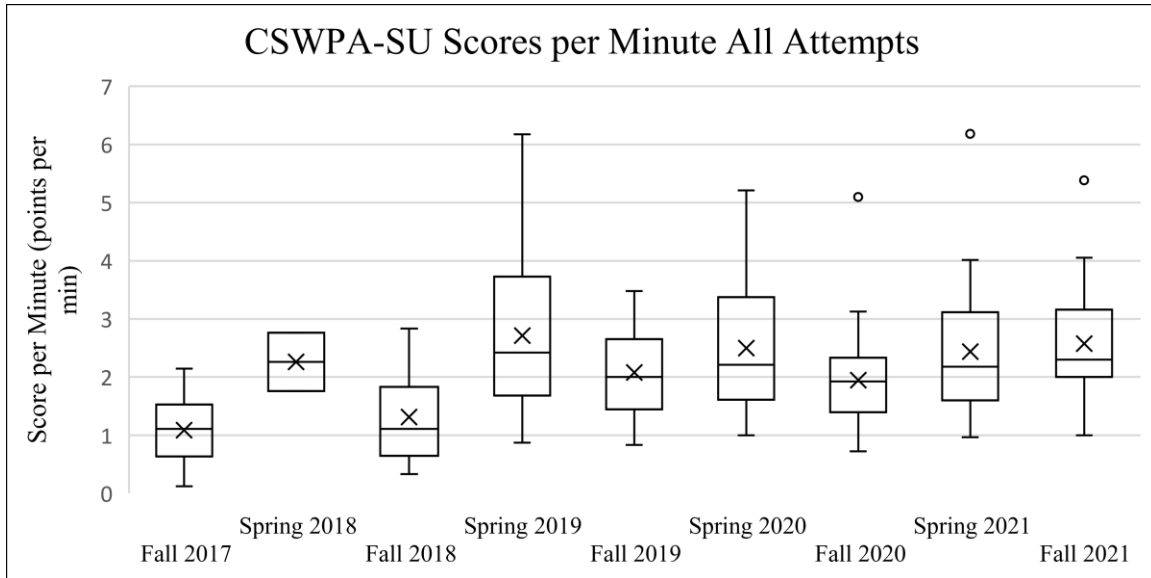


Figure 2.14: Box and Whisker Plot of All Attempts Certified SOLIDWORKS Professional Advanced Surfacing Exam Score per Minute from Fall 2017 to Fall 2021

If students do not pass the CSWPA-SU on the first try they are expected to retake it after a waiting period of 14 days. Since the retake is very similar to the original exam these students could perform well on the retake without using the course preparatory material. To isolate the impact of the preparatory material the first attempts of the exam were analyzed in Figures 2.15 and 2.16. Figure 2.15 is a box and whisker plot of the 100 CSWPA-SU first attempts sorted by academic semester. The red line indicates the minimum score needed to pass the exam.

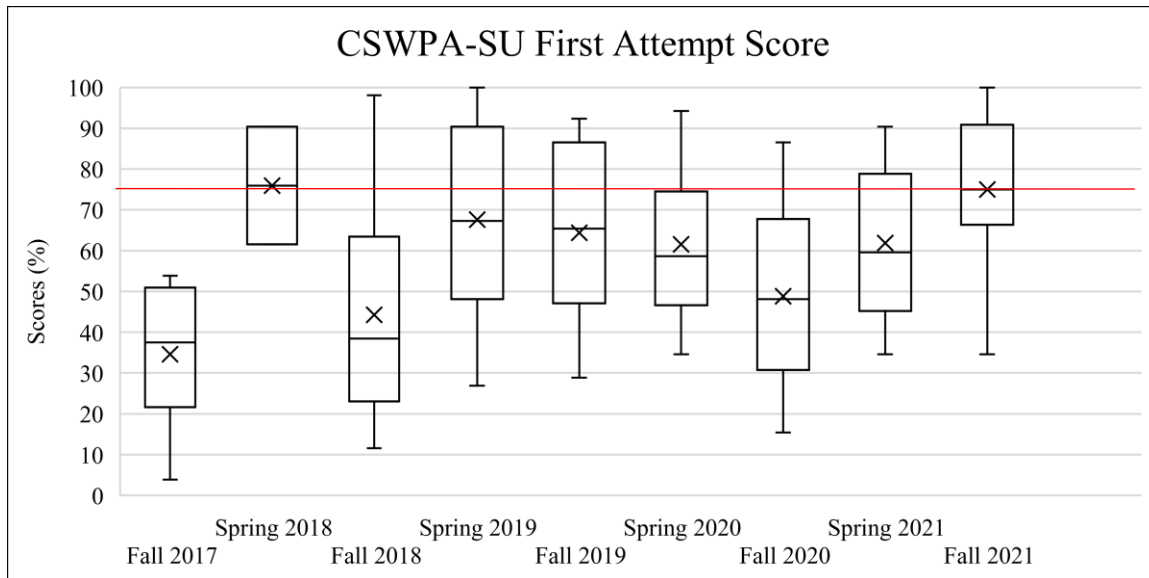


Figure 2.15: Box and Whisker Plot of First Attempts of the Certified SOLIDWORKS Professional Advanced Surfacing Exam Scores (%) from Fall 2017 to Fall 2021

Since 87% of the students who attempted the CSWPA-SU on the first try took 80 to 90 minutes to complete this exam the first attempt score per minute (Figure 2.16) is a very similar plot to the first attempt score plot [12]. This similarity illustrates that there has been no significant increase in modeling speed on this exam, only an increase in CSWPA-SU scores.

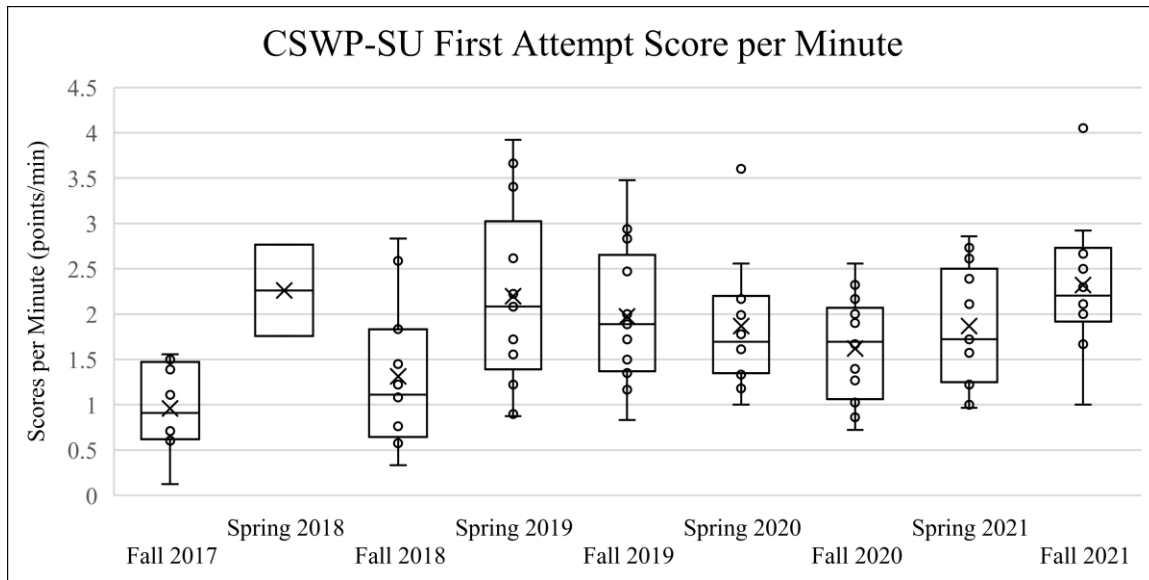


Figure 2.16: Box and Whisker Plot of First Attempts of the Certified SOLIDWORKS Professional Advanced Surfacing Exam Score per Minute from Fall 2017 to Fall 2021

2.6: SOLIDWORKS Expert Pedagogy

To qualify to take the CSWE Dassault Systèmes requires students must have passed the CSWP, and four out of five CSWPA exams [24]. Furthermore, if a student wishes to have the exam funded by the University of Idaho, they must meet more qualifications. Those qualifications include the student must have passed at least five of the other certification exams on the first try, they cannot have failed any exam twice, they must complete all the CSWE preparatory work before the Monday of dead week, and they must pass an exam proctored by all their instructors by the Monday of finals week. If the student fails the exam, they must wait ninety days to retake the exam. The CSWE covers a small section of all the other subject exams in the class as well as some small amount of additional material.

The topics covered in the CSWE section of *Solid Modeling, Simulation and Manufacturing Capstone* include Sketch Belt/Chain, Make Block, Draft, Combine, weldments features, 3D sketches, sheet metal features swept profile, Insert Part, in-context modeling, Copy Surface, Delete Face, Move/Copy Body, Mirror, Lofted Surface, Trim Surface, and Knit Surface, Assembly Configurations, Part Configurations, Bill of Materials, Boundary Surface, and Cam-follower mate. At the beginning of the CSWE segment of the

class, students are asked if they wish to take the CSWE. If they do not wish to, the class is over for them with no impact on their final grade. If they continue, they are tasked with completing the four-hour CSWE exam [24]. The next few pages will give a basic rundown of how these topics are covered in the class. For the complete *SOLIDWORKS Expert Workbook*, refer to Appendix B.

Unlike the *SOLIDWORKS Surfacing Workbook*, the *SOLIDWORKS Expert workbook* does not include any tutorials. At this point in the semester, students are familiar with all the tools they will need to pass this exam. Instead, the Expert Workbook consists of example problems and video solutions. The example problems are aimed to make them critically think and the video solutions are made so they can check their work.

The Belt/Chain sketch tool is used to determine belt length, rapidly calculate the ratio of angular displacement between pulleys/gears, and quickly revise designs by modifying belt thickness. In the example illustrated below (Figure 2.17), students are asked to create a Belt/Chain between the pulleys/gears and calculate the belt length as well as change in angle of one pulley/gear when rotating another. To solve this problem, the students need to know how to use the Make Block sketch tool. Make Block is a tool applied in chapter 4 to obtain XY coordinate data of a Spiral Spring.

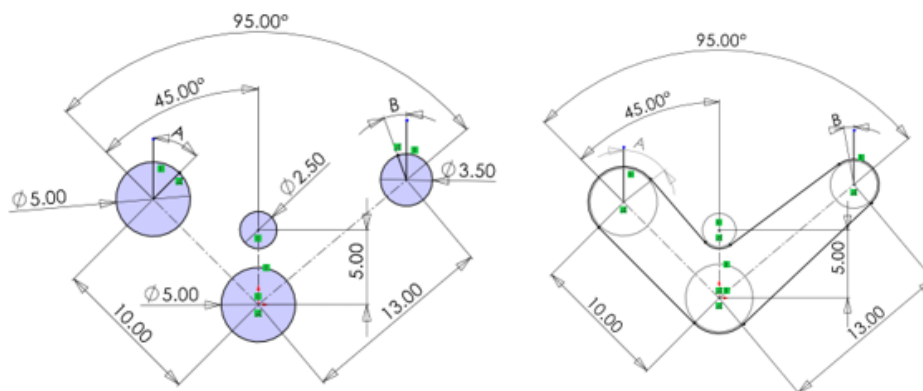


Figure 2.17: Left: Starting Model, Right: Complete Model in the Belt/Chain Problem Set in the *SOLIDWORKS Expert Workbook* (See Appendix B)

The chamfer creation shown in Figure 2.18 is surprisingly difficult to perform. If the student tries to use the Chamfer Feature the model will not build. Creating a revolved cut will not work because the face is oblong. Using a Swept Cut with a path following the perimeter

of the oval will not work because the profile orientation will change relative to the oval. To solve the problem, the students must split the part along the bottom edge of the chamfer and Draft the upper solid body.

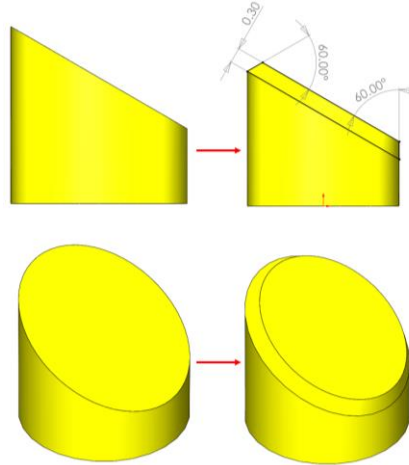


Figure 2.18: Left: Starting Model, Right: Complete Model in the Chamfer Problem in the SOLIDWORKS Expert Workbook (See Appendix B)

One of the major changes that should be made to the current edition of *Solid Modeling, Simulation, and Manufacturing Capstone* is that the Combine tool should be taught earlier in the semester. This example (Figure 2.19) requires students to create two hollow cylinders as separate solid bodies, then use the Combine tool, selecting the common option, to create the model shown below. They can also use a series of Extrudes and Cuts to create the part less efficiently.

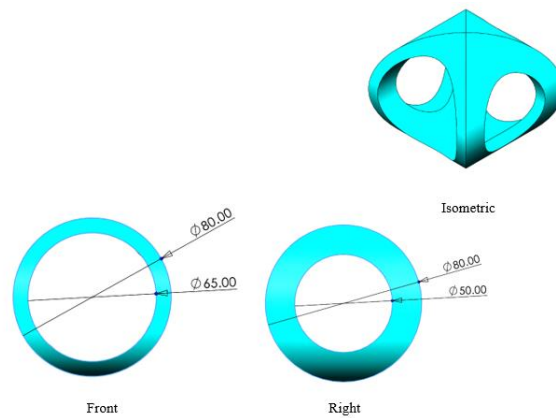


Figure 2.19: Complete Model in the Combine Problem in the SOLIDWORKS Expert Workbook (See Appendix B)

Some of the most difficult topics in the class include 3D sketching, identifying projected angles verse true angles, and 3D sketching projected angles. This example (Figure 2.20) covers basic weldment creation while dealing with all the topics discussed above as well as including sketch planes. The angles called out in the picture below are projected. Four methods for 3D sketching projected angles are taught at the University of Idaho.

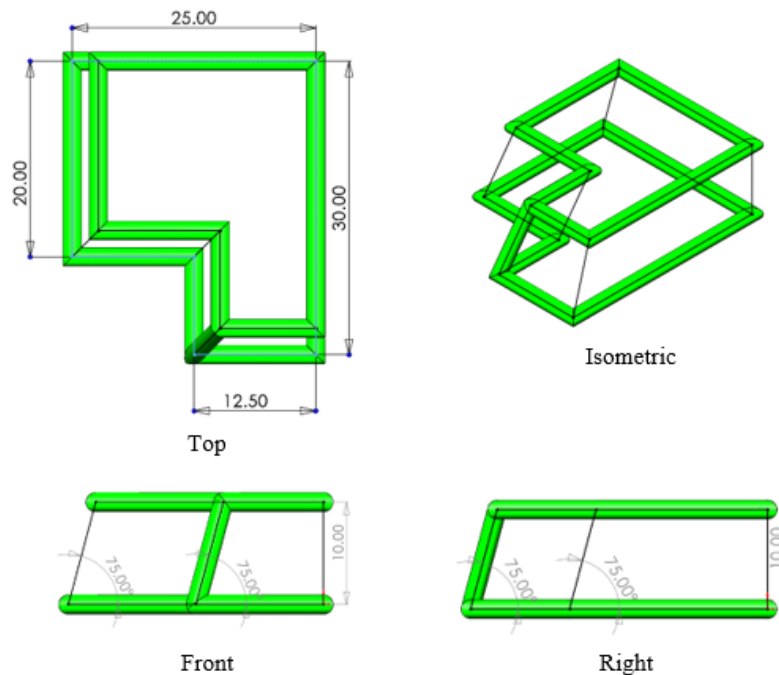


Figure 2.20: Complete Model in the Weldments Problem in the SOLIDWORKS Expert Workbook (See Appendix B)

The four methods include the Plane Method, Line Method, Trigonometry Method, and Convert Edge method (Figure 2.21). Imagine trying to create a line segment out of two projected angles and the length of the line segment. Using the Plane Method, the modeler would create two planes that create the projected angles between themselves and the top plane, one that is perpendicular to the right plane and the other perpendicular to the front plane. Then create a line segment with both endpoints coincident with the two planes and define the length. To use the Line Method, students must create the two projected lines as construction geometry then sketch the true line segment and select the Along X and Along Y sketch relation with the endpoints of the projected lines and the true line. The Trigonometry Method requires the student to use math to determine geometry. For example, if the student knows the change in position of the line segment along the Y-axis, they can select the

endpoints of the true line and hit the Tab key to define the change along the Y-Axis. They can do the same along the X and Z, however, it will require trigonometry if the line segment is defined with projected angles. The Convert Edges Method involves creating an Extruded Boss/Base with a draft angle equal to the complementary angle of the projected angle, then converting the edges of the solid body.

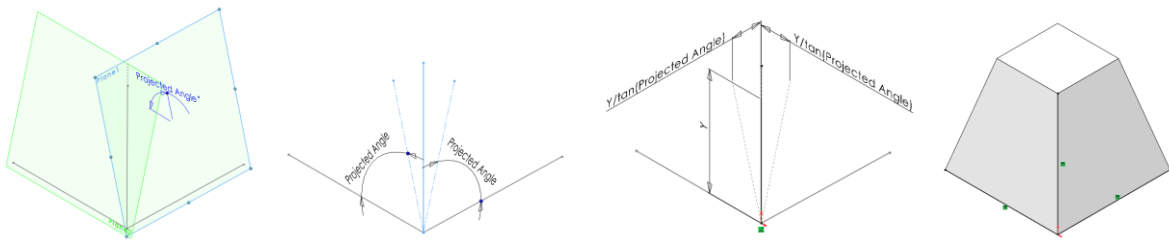


Figure 2.21: Illustration of the Four 3D Sketching Methods Taught In Solid Modeling, Simulation, and Manufacturing Capstone

Sheet Metal is an advanced subject exam covered earlier in Advanced SOLIDWORKS and Manufacturing. This example (Figure 2.22) covers importing SldWorks 2021 Application (.x_t) files into a SOLIDWORKS Part Document (.SDDLPT) using Import Diagnostics, as well as sheet metal features such as Convert to Sheet Metal, Fold, and Unfold.

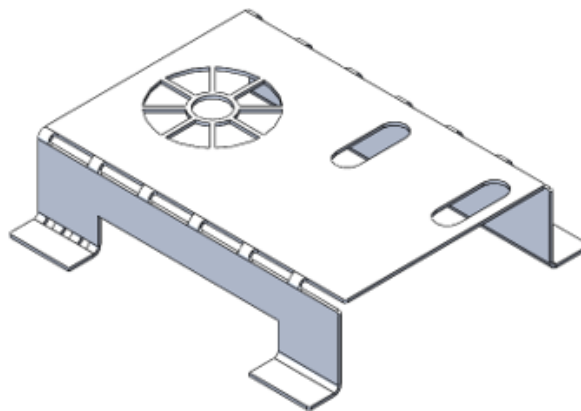


Figure 2.22: Complete Model in the Sheet Metal Problem Set in the SOLIDWORKS Expert Workbook (See Appendix B)

There are a variety of different methods for creating springs in SOLIDWORKS. The method depicted in this example (Figure 2.23) includes sweeping a circular profile along the length of the path and modifying parameters such as Profile Twist, Twist Control, and

Direction. A method very similar to this is applied in Chapter 4 to create the thread profile of a fusee in SOLIDWORKS.

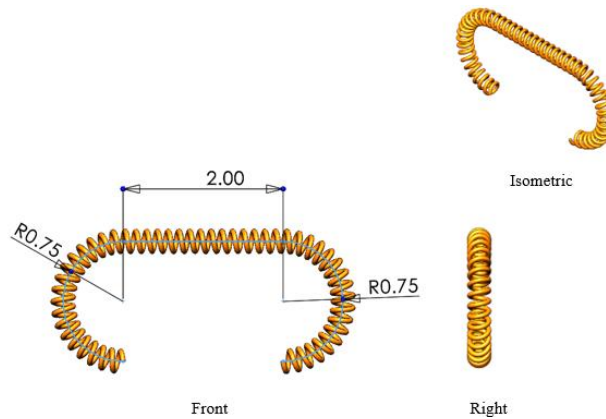


Figure 2.23: Complete Model in the Curved Spring Creation Problem in the SOLIDWORKS Expert Workbook (See Appendix B)

In-context modeling is a method to design parts in-context of an assembly. This example shows a method to combine two parts into one using a variety of different methods. In one method the student must mate the parts such that they are in line with each other, then edit one of the parts in-context of the assembly, convert edges of the other part into this part, and extrude cut or split the part with the converted edges. The other method includes opening one of the parts in a part file, inserting the other part into that part file, and using Combine<Common.

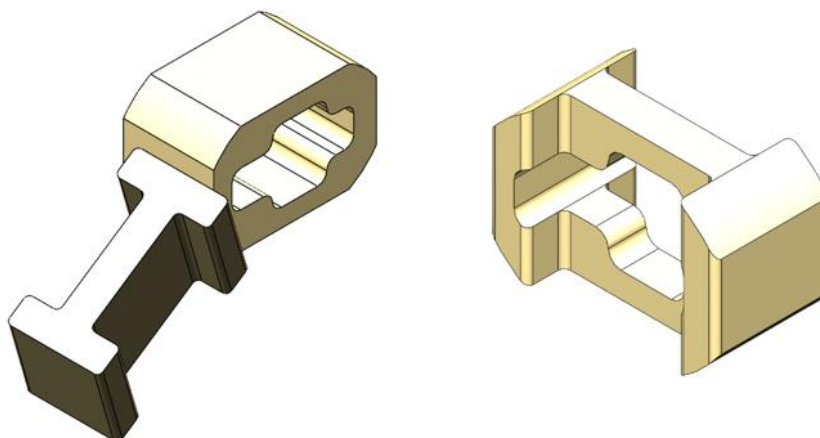


Figure 2.24: Left: Starting Model, Right: Complete Model in the Single Body Part from Assembly Problem in the SOLIDWORKS Expert Workbook (See Appendix B)

This problem (Figure 2.25) is another example of in-context modeling. The goal of this problem is to create the copper tube in-context of the assembly to determine the center of mass after adding the copper tube. This problem requires an advanced understanding of Spline creation, Spline definition, and the Fit Spline tool.

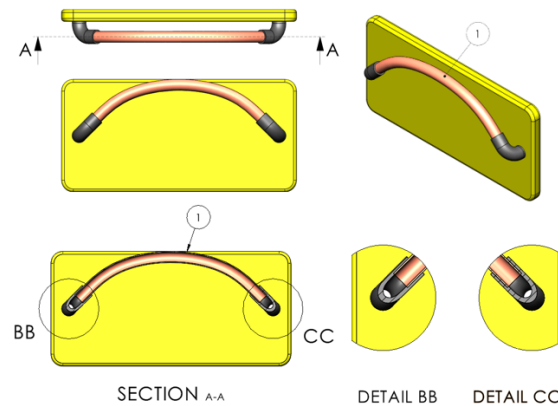


Figure 2.25: Complete Model in the Assembly Modification Problem in the SOLIDWORKS Expert Workbook (See Appendix B)

This example (Figure 2.26) is very similar to the one shown in Figure 2.24, except that instead of combining two parts in-context of an assembly, the bodies are combined in a part file. This changes the commands that the modeler must use to complete the task. There are multiple different methods to solve this problem. One method involves opening one of the parts, then using the Insert Part tool to import the other part file as a solid body, and using the Combine tool. Another method starts the same way (i.e. using the Insert Part tool) but instead of Combining the parts, the student must use the Surface Copy tool on the top surface of one of the parts, then use the Cut with Surface tool to cut the other body.



Figure 2.26: Complete Model in the Part Modification Problem Set in the SOLIDWORKS Expert Workbook (See Appendix B)

Modifying the model depicted in Figure 2.27 would be easy if it was a SOLIDWORKS assembly document, however, this model is a single body part file. To complete the part modifications, the student needs to enable View Temporary Axes, evaluate the angle between the centroid of the piston and the Right Plane using the measure tool, and use the features Delete Face, Move/Copy Body, Mirror, Lofted Surface, Trim Surface, and Knit Surface. The CAD file for this question was adapted from a model originally found on GrabCAD [25].

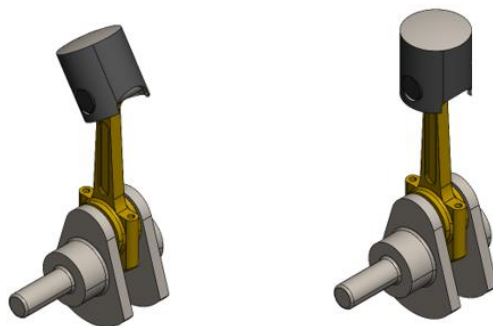
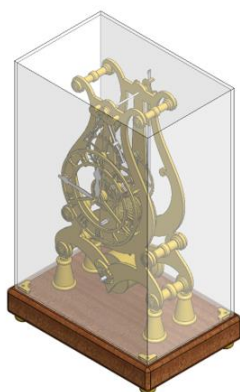


Figure 2.27: Left: Starting Model, Right: Complete Model in the Part Modification Problem Set in the SOLIDWORKS Expert Workbook (See Appendix B)

Assembly Configurations, Part Configurations, and Bill of Materials creation and modification are the topics covered in the example below (Figure 2.28). Applications of Part configurations are explored in sections 4.6 Modern Methods for Modeling a Fusee. The model used in this example is a modified version of the Lyre Skeleton Clock originally designed by W.R. Smith and reverse engineered in Chapter 4 of this paper [3].



ITEM NO.	PART NUMBER	DESCRIPTION	SW-Configuration Name (Configuration Name)	QTY.
1	01-01 Plates	Done	Default	1
2	01-01 Plates	Done	Back	1
3	01-02 Pillars	Done	Bottom	2
4	01-02 Pillars	Done	Middle	2
5	01-02 Pillars	Done	Top	2
6	01-03 Decorative Washers	Done	Bottom	4
7	01-03 Decorative Washers	Done	Top	4
8	01-03 Decorative Washers	Done	Middle	4
9	01-04 Feet		AAA	4
10	01-05 Base		Default	1
11	01-06 Chevron		Default	4
12	01-07 Bar Foot		AAA	4
13	01-08 Dust Cover		Default	1
14	02-01 Barrel		AAA	1
15	02-02 End Caps		With Cut	1
16	02-02 End Caps		Without Cut	1
17	02-03 Barrel Arbor		Default	1
18	02-04 Ratchet Wheel		AAA	1
19	02-05 Barrel Arbor Click		Default	1
20	03-01 Fusee		Default	1
21	03-02 Fusee Arbor		Default	1
22	03-03 Fusee Ratchet Wheel		Default	1
23	03-04 Fusee Click		Default	1

Figure 2.28: Left: Starting Model, Right: Complete Model in the Assembly Configurations and Bill of Materials Problem in the SOLIDWORKS Expert Workbook (See Appendix B)

“A cam-follower mate is a type of tangent or coincident mate. It allows you to mate a cylinder, plane, or point to a series of tangent extruded faces, such as you would find on a cam” [26]. This example (Figure 2.29) aims to teach students how to create a cam-follower mate. To complete the problem the student must make the cam face a continuous surface so that the follower can mate to it. The task is completed by converting the edge of the cam to a sketch, using the Fit Spline tool on the converted edge, making a Boundary Surface with the sketch, and creating a cam-follower mate between the follower and the Boundary Surface.

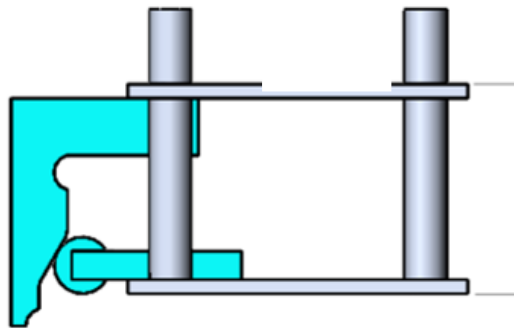


Figure 2.29: Complete Model in the Cam and Actuator Problem Set in the SOLIDWORKS Expert Workbook (See Appendix B)

The part modification illustrated in Figure 2.30 consists of Convert Entities, Extrude Up to Next, Revolve Surface, Extrude Up to Surface, Fillet Face, Shell. This is not a difficult modification to make, the biggest challenge of this problem is reading the drawing and understanding what the question is asking.

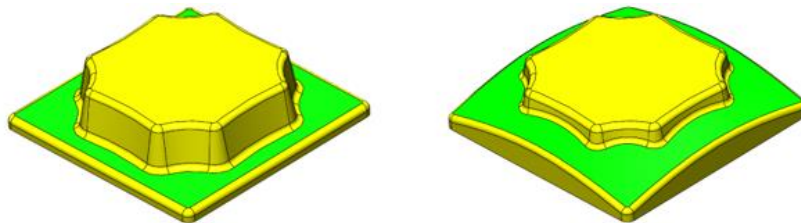


Figure 2.30: Left: Starting Model, Right: Complete Model in the Part Modification Problem in the SOLIDWORKS Expert Workbook (See Appendix B)

To complete the problem illustrated in figure 2.31 the student must first remove the square inlay by using the Delete Face tool with the option Delete and Patch selected. Then

sketch the projection of the slot inlay on the Top plane and Extrude Cut with the Offset from Surface end condition selected. The CAD file for this question was adapted from a part created in *Certified SOLIDWORKS Professional Advanced Preparation Material* [19].



Figure 2.31: Left: Starting Model, Right: Complete Model in the Part Modification Problem Set in the SOLIDWORKS Expert Workbook (See Appendix B)

At first glance, the problem illustrated in Figure 2.32 seems to be a simple Mirror operation. This is an incorrect conjecture since the surface on the top is not symmetric about the Right plane. To complete the problem students must project the inner edge of the swept path onto the Top plane using the Convert Entities sketch tool and mirror the sketch about the Right plane. Then use the Copy Surface feature to copy the surface highlighted in blue in the left image of Figure 2.32. Fill the hole in the copied surface with the Delete Hole feature. Use the Trim with Surface feature to cut away the old swept path. Finally, using the Projected Curve feature project the mirrored sketch onto the upper surface, create a swept profile, and sweep the profile about the projected curve.

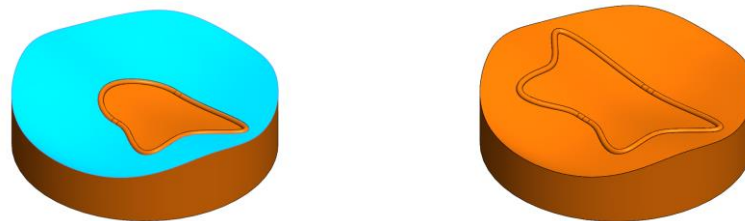


Figure 2.32: Left: Starting Model, Right: Complete Model in the Part Modification Problem Set in the SOLIDWORKS Expert Workbook (See Appendix B)

2.7: Certified SOLIDWORKS Advanced Professional Surfacing (CSWE-MD) Exam Results

In the fall of 2019 the University of Idaho *Solid Modeling, Simulation, and Manufacturing Capstone* class decided to attempt the CSWE (Cite the data). Two-thirds of the students that semester passed the exam, and the average score was around 83% (see Figures 2.33 and 2.34). That was regarded to be a spectacular outcome as compared to the first semester of the CSWPA-SU. Nevertheless, over the past two years, drastic changes have been made to the CSWE preparatory material culminating in the material presented in 2.6 SOLIDWORKS Expert Pedagogy. The following study consists of 37 CSWE exams taken by University of Idaho students from Fall 2019 to Fall 2021. Before each exam, the students were asked if they were willing to release their exam data to Dr. Edwin Odom. All students in this study agreed to do so. The data was collected by Dassault Systèmes, then made anonymous, accessed, and analyzed by the University of Idaho through the 3DEXPERIENCE Certification Center.

From fall of 2019 to fall of 2020 the class was taught using the *Certified SOLIDWORKS Expert Preparation Materials* by Paul Trans [2]. During the spring of 2021, the *SOLIDWORKS Expert Workbook* was implemented in the class. Due to the small sample size, it is hard to determine what effect the material has had on learning outcomes. However, all the data seems to imply that the material has positively impacted the class (see Figure 2.33, 2.34, 2.35).

Figure 2.33 shows the University of Idaho's pass rates by semester for the CSWE. Unlike the CSWP and CSWPA exams, the CSWE has a ninety-day waiting period between exam attempts [24]. This means that no student has ever had a retake of the CSWE funded by the University of Idaho and therefore all the data illustrated below is the student's first attempt at the exam. A linear regression of the data illustrated in Figure 2.33 yields a trend line with a slope of 6.66 and an R-squared value of 28.19%. This slope illustrates that over the past four years the CSWPA-SU scores at the University of Idaho have increased by about five percent per semester.

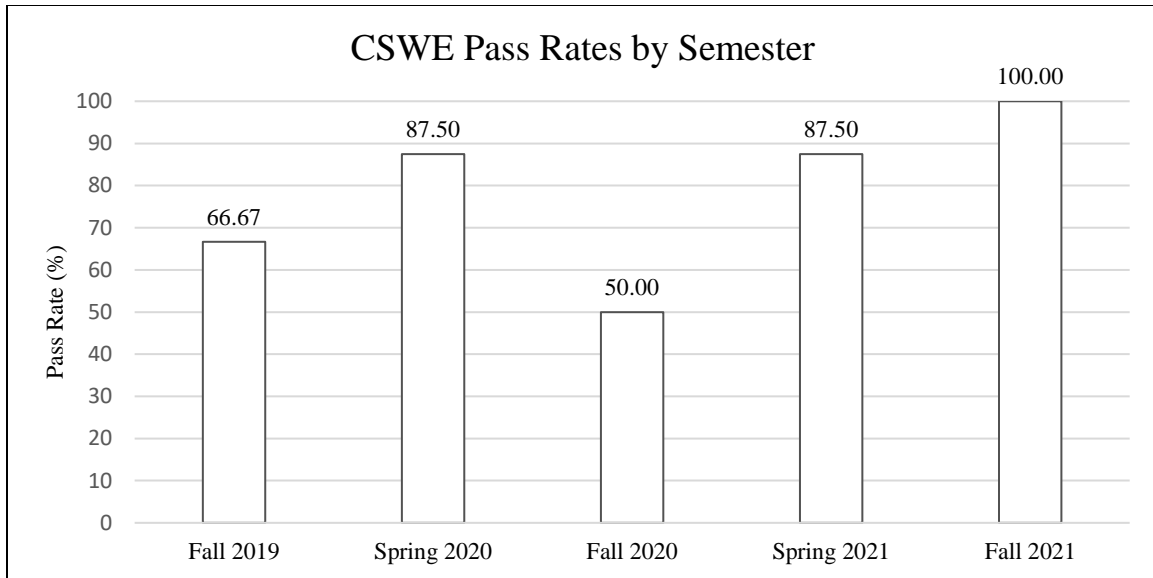


Figure 2.33: Certified SOLIDWORKS Expert Exam Pass Rates (%) from Fall 2019 to Fall 2021

Figure 2.34 illustrates the number of passes and fails per semester. While using the *SOLIDWORKS Expert Workbook* (from spring 2021 forward) sixteen out of seventeen students have passed the exam. This moves from a pass rate of 70% using other preparatory material to a pass rate of 94% using the SOLIDWORKS expert workbook.

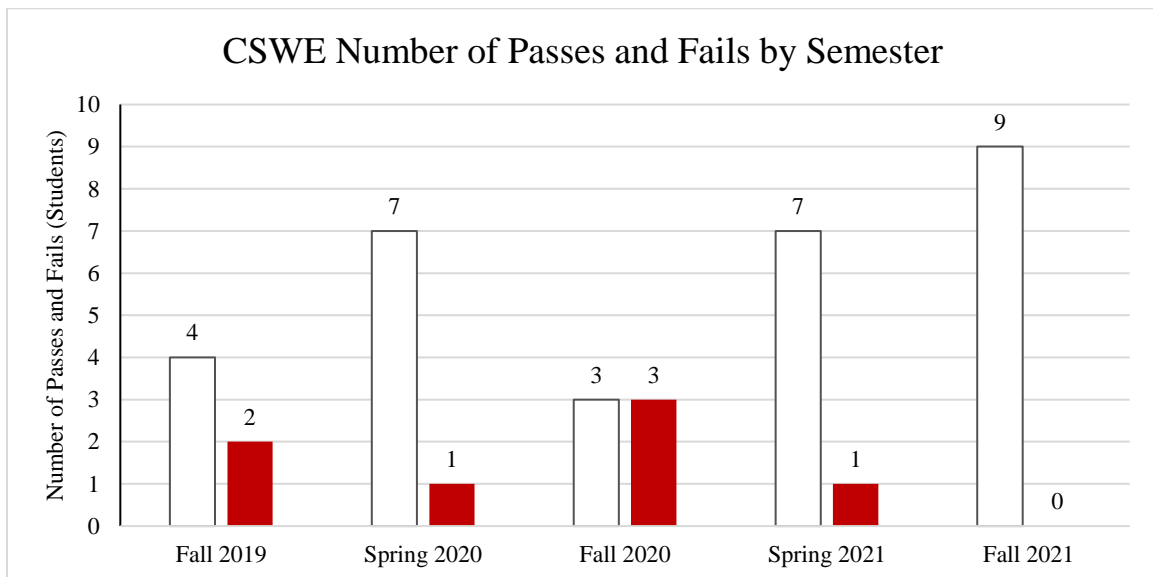


Figure 2.34: Certified SOLIDWORKS Expert Exam Number of Passes and Fails (%) from Fall 2019 to Fall 2021

The box and whisker plot below (Figure 2.35) illustrates the results of 37 CSWE exams sorted by academic semester. The red line indicates the minimum score needed to pass the exam. The average exam score differs by 13.33% from the highest averaging semester (Fall 2021) to the lowest averaging semester (Fall 2020). The most notable difference from semester to semester is the decrease in the interquartile range. This illustrates that the new training material does not drastically change the average score and learning outcome of the upper quartile, however, it does significantly increase the average score and learning outcome of the lower quartile.

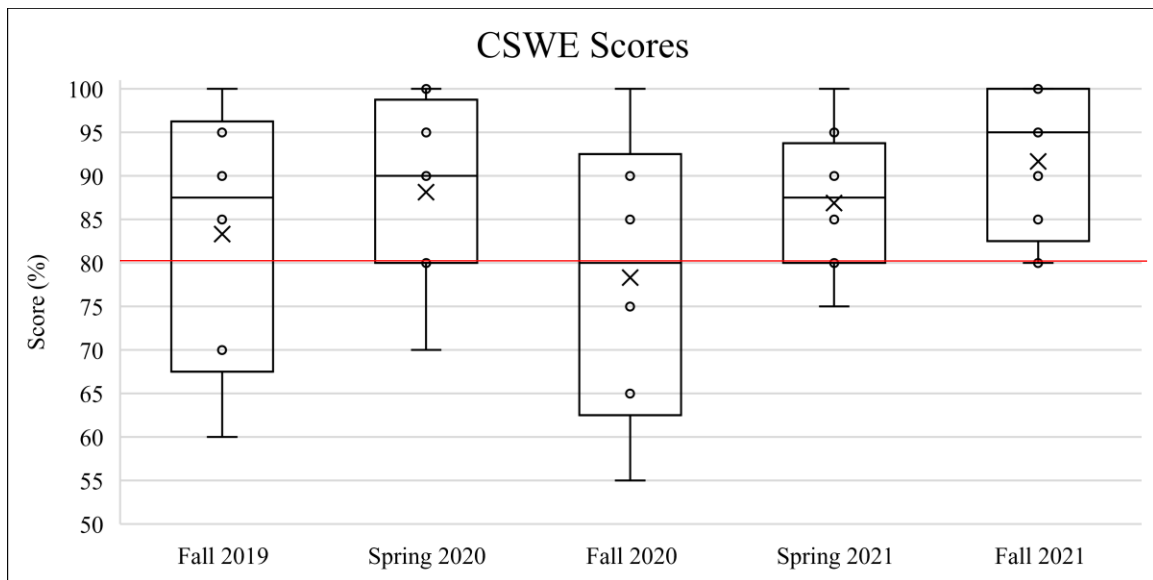


Figure 2.35: Box and Whisker Plot of All Certified SOLIDWORKS Expert Exam Scores (%) from Fall 2019 to Fall 2021

Figure 2.36 is a box and whisker plot below showing the number of points students received on the CSWE divided by the number of minutes the student took to complete the exam sorted by academic semester. This data does not correlate to any of the changes implemented in the class. Meaning the attempts to increase the modeling speed by showing SOLIDWORKS shortcuts and emphasizing them during lecture has had no effect.

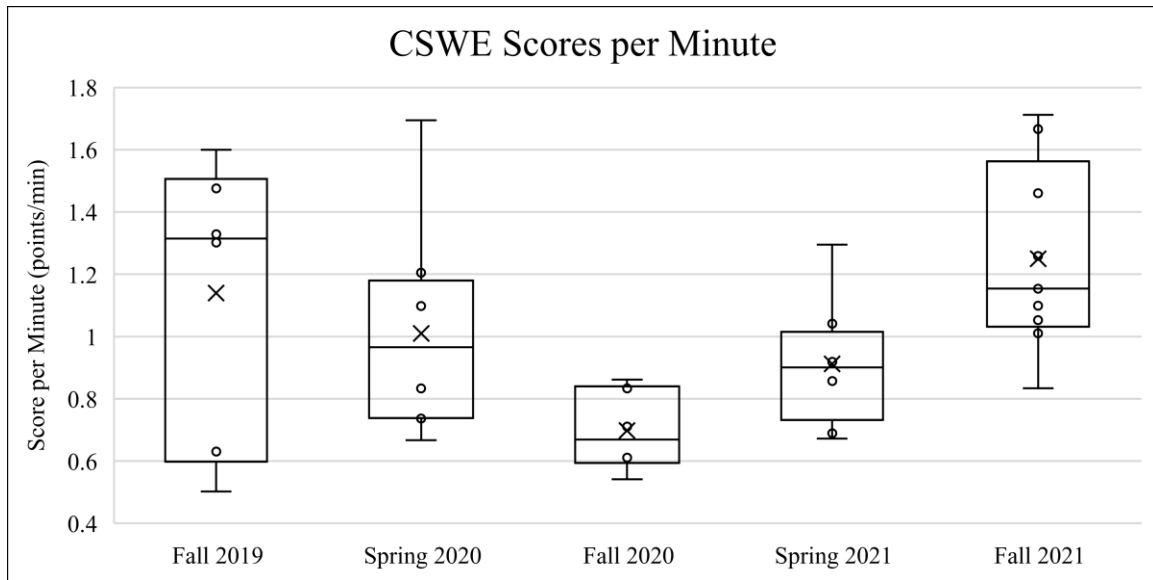


Figure 2.36: Box and Whisker Plot of All Attempts Certified SOLIDWORKS Expert Exam Score per Minute from Fall 2019 to Fall 2021

Chapter 3: Practical Application for Advanced Surfacing and Expert SOLIDWORKS Techniques

3.1: Introduction

This chapter illustrates how the techniques taught in the surfacing and expert sections of the *Solid Modeling, Simulation and Manufacturing Capstone course* can be applied to the research and design of a mechanical clock. SOLIDWORKS is a useful design tool, no one is debating that. But how useful are the surfacing and expert techniques?

3.2: Application of Advanced Surfacing and Expert SOLIDWORKS

Chapter four goes into more detail about the development of a tool to design and manufacture a mechanism called a fusee. Advanced SOLIDWORKS surfacing and expert techniques were instrumental in completing that project. In old mechanical clocks, the spiral spring powers the drive train thus keeping the clock running. To determine the spiral spring constant that spring was scanned, imported into SOLIDWORKS, traced using a spline tool, the center of the spiral was determined, the spline was made into a sketch block, repositioned such that the center of the spiral was the origin of the part, and from there data was collected and the equation for the position of the spiral was found. Sketching splines is taught in the surfacing section and making sketch blocks is taught in the expert section of the *Solid Modeling, Simulation, and Manufacturing Capstone course*. To create a solid model of the fusee the Move Face, Surface Sweep, and Trim Surface was used. For more detail on these parts refer to sections 4.5 and 4.6. Figure 3.1 shows the spiral spring sketch block and a surface sweep used to create the thread profile of a fusee.

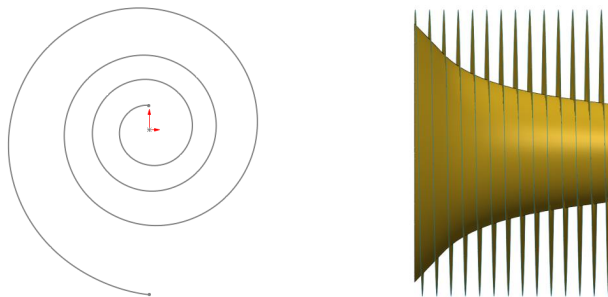


Figure 3.1: Left: Spiral Spring Sketch Block. Right: Fusee Surface Sweep

Chapter 4: Fusee Design Tool Applied to Redesign and Manufacture of the Fusee for Lyre Skeleton Clock

4.1: Introduction

In 1989 TIMECRAFT magazine released a series of articles by Master Clock Maker W.R. Smith depicting how to build a clock that he designed. In 1990 those articles, as well as additional unpublished articles, were gathered and published in *How to Make a Lyre Skeleton Clock* [3]. In it, he illustrates a method for manufacturing a mechanism called fusee. A fusee is a nonlinear threaded cone used to normalize the torque output of a spiral spring in a barrel (the main spring). Classically, the fusee is machined by moving the tool post to a specified location and rotating the cutter about the tool post. The turret is then rotated by hand into the fusee blank (left image of Figure 4.1). This creates a constant radius cut. The threads are then manufactured by applying finger pressure to a fusee grooving attachment for a lathe (right image of Figure 4.1). The fusee grooving attachment is dedicated tooling that is only useful for fusee manufacturing.

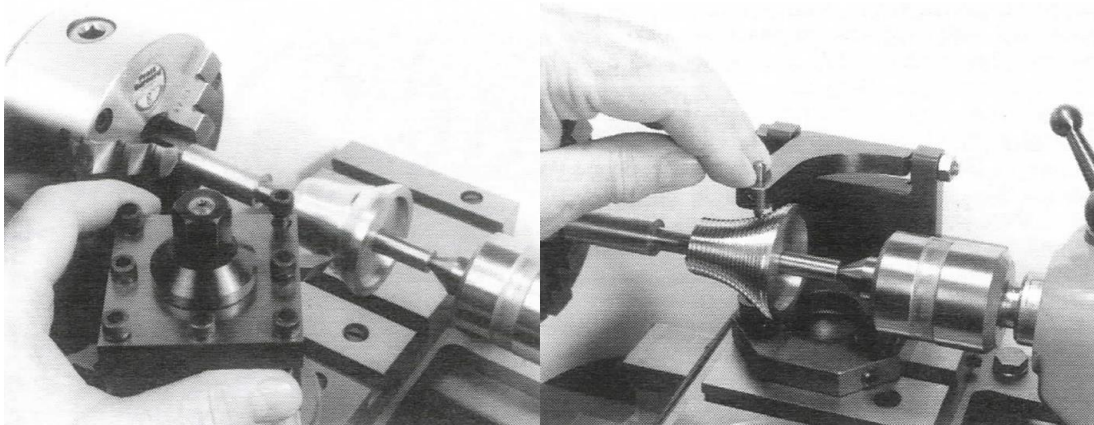


Figure 4.1: Left: Method for manufacturing the profile of a Fusee manually. Right: Method for Manufacturing the Threads of a Fusee manually

Source: [3]

The profile of a fusee is not a circular arc, it can be analytically defined by the equation

$$d\varphi = - \frac{RT}{r(\varphi)^2 k \sqrt{r(\varphi)^2 + \left(\frac{dr(\varphi)}{d\varphi}\right)^2 + \rho^2}} dr(\varphi), \quad (1)$$

where φ is the angle of twist of the fusee, R is the outside radius of the barrel, T the required torque to turn the gear train of the clock, r radius of the fusee a function of the angle twisted, and ρ pitch of the fusee [8].

Thus, the manufacturing methods illustrated above are inconsistent and limit the design capabilities of the horologist. The profile of a fusee does not necessarily have a constant radius, thus when a fusee is manufactured this way the torque curve generated by this fusee when attached to a spiral spring will not be constant. Furthermore, if the finger pressure, which isn't measured, is applied to the grooving tool unevenly then the depth of cut along the profile will be uneven which will generate an inaccurate torque curve.

A classical method to solve these problems is to manufacture a fusee with a constant radius profile, mount it to an arbor, attach it to a spiral spring in a barrel then test the torque output of the fusee arbor. Then increase the depth of cut in the regions that required more torque output. Figure 4.2 shows the torque output on the fusee before and after this method was implemented by W.R. Smith on his Grasshopper Skeleton Clock fusee [27].

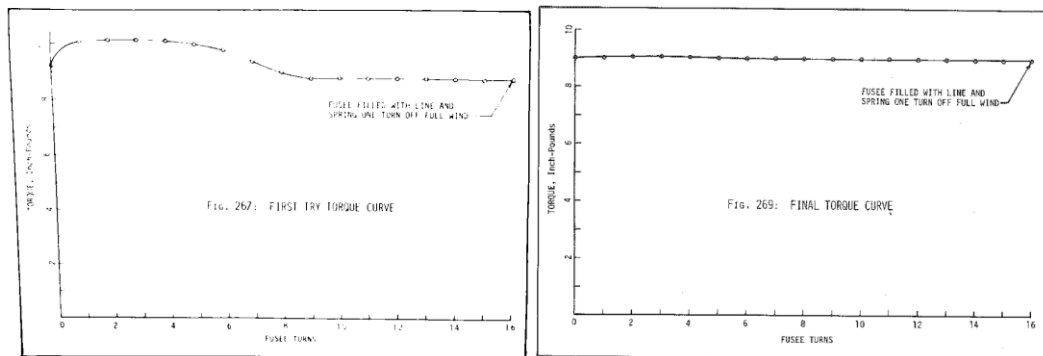


Figure 4.2: Left: the Torque Curve of a Fusee with a Constant Radius Profile. Right: the Torque Curve of a Fusee After Testing and Redesign

Source: [27]

The manufacturing techniques illustrated above require a high level of skill on the manual lathe, dedicated tooling, testing, and redesign. Computer Numerical Control (CNC) technology along with an equation for the profile of a fusee as defined by Preisendorfer in

1975, can help mitigate some of these drawbacks [4]. In *Method to Accelerate Student Proficiency in CSWP/CSWPA Solidworks Certification Exams and Fusee Mechanism Profile Analysis* Ryan Gonzalez and Edwin Odom develop a TK Solver model of Preisendorfer's equation for a profile of a fusee [8]. That model was used to develop a fusee design tool. The tool allows the user to input variables such as threads per radian, the outside radius of the barrel, spring constant, and required torque output into Microsoft Excel. Microsoft excel then sends the inputs to TK Solver where the profile of the fusee is solved using Preisendorfer's equation for the profile of a fusee. Then TK Solver Updates Excel and a list of radius and distance are used to create a parametric model of a fusee and G-code for a Hass CNC lathe to manufacture the threads of the fusee. The design, solid model, and manufacturing flow chart is shown in figure 4.3. The fusee design can be found in Appendix F.

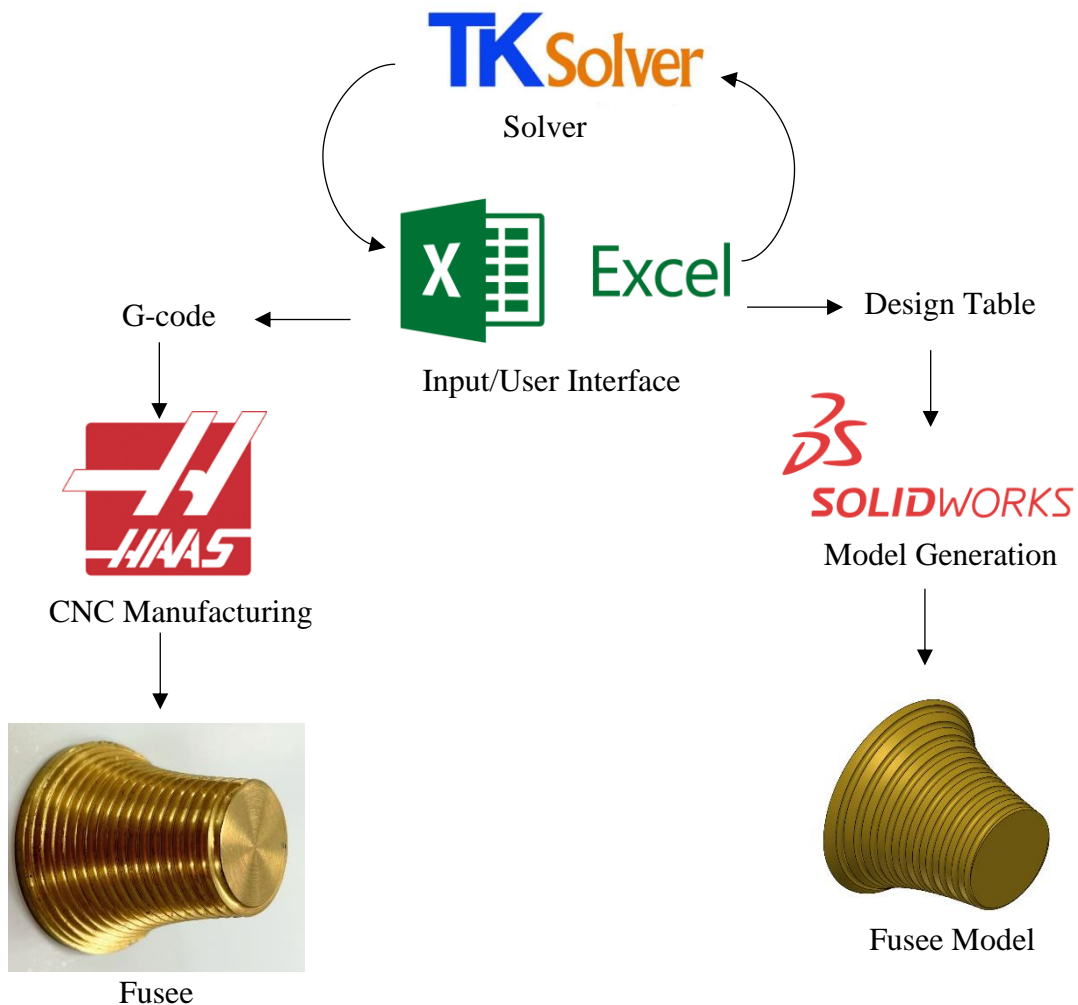


Figure 4.3: Fusee Design Tool Flow Chart

4.2: Fusee Design Tool Input Variables

The inputs required to determine the profile of the fusee include the number of threads, the radius of the barrel, the torque required to drive the gear train, the spiral spring spring constant, the pitch of the fusee, the beginning radius of the fusee, the diameter of the wire, and the length of the fusee. Variables required to machine the fusee include the final depth of cut, retreat distance in the X direction, spindle speed, and program number. Figure 4.4 shows some of the TK Solver code behind the fusee design tool and as well as the excel document where the input variables can be input by the user.

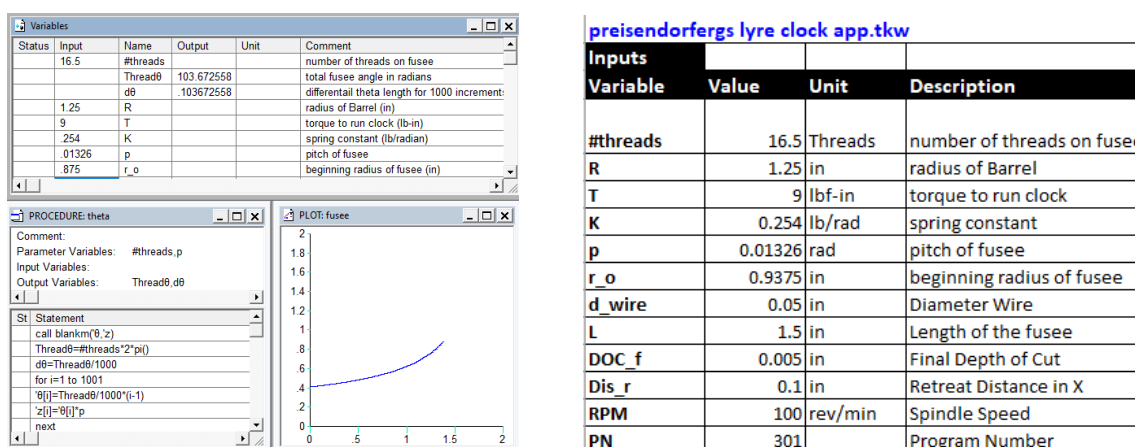


Figure 4.4: Left: TK solver Code in the Fusee Design and Manufacturing Tool Created by Ryan Gonzalez and Edwin Odom. Right: User Interface for the Fusee Design and Manufacturing Tool

Before going into detail on how the fusee design tool functions and was developed, it is important to understand how some of the more complex input variables can be determined. To do that it was decided that the fusee for the Lyre Skeleton Clock would be redesigned using the fusee design tool. This means most of the input variables have already been decided by Master Clock maker W.R. Smith. The only input variable left to determine are the torque required to drive the gear train, the spiral spring spring constant, and the variables required to machine the fusee. The next few sections of this chapter will describe the Lyre Skeleton Clock and cover how to determine these input variables.

4.3: The Lyre Skeleton Clock

To acquire a better understanding of how the Lyre Skeleton clock functions it was modeled in SOLIDWORKS using the plans from *How to Make a Lyre Skeleton Clock* [3]. Figure 4.5 shows the Lyre Skeleton Clock and an exploded view of the gear train for the Lyre Skeleton Clock. The colors indicate groups that rotate separately from one and other. Note that the hour wheel and the center wheel rotate about the same axis.

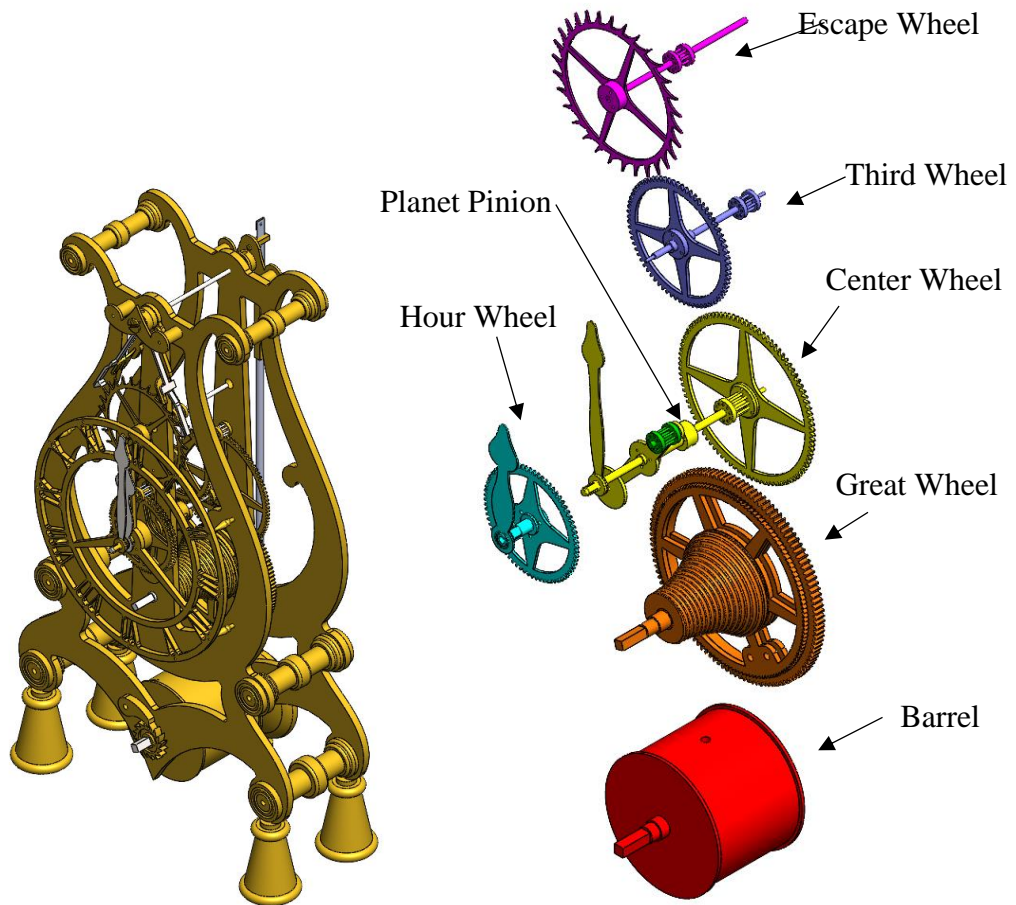


Figure 4.5: Left: Complete SOLIDWORKS Model of the Lyre Skeleton Clock. Right: Color Coded Exploded View of the Lyre Skeleton Clock Gear Train.

This section covers the more complex mechanisms of this clock to ensure that everyone reading this is on the same page. The description will begin at the bottom of the gear train where the torque driving the gear train is generated by a wound-up spiral spring, and end at the top of the gear train where the escapement controls the transfer of energy from the spiral spring to the gear train.

The Lyre Skeleton clock has six rotation axes and seven subassemblies that rotate separately from one and other. To start the fusee mechanism is located on the great wheel arbor and is connected by a wire to a spiral spring inside the barrel (Figure 4.6). The torque output of the spiral spring is nonlinear, however, the shape of the fusee mechanism counteracts this and generates constant torque on the great wheel arbor. This constant torque is transferred through the gear train and ensures that the clock keeps constant time.



Figure 4.6: Fusee Mechanism Attached to a Spiral Spring Inside a Barrel

Source: [8]

The energy is then transferred from the great wheel to the center wheel through to a set of gears. The center wheel is attached to the minute hand of the clock, therefore, must complete a full rotation once per hour. The planet pinion and the hour wheel rotate around the center arbor and operate using the Ferguson Mechanical Paradox. Figure 4.7, illustrates the Ferguson Mechanical paradox presented in the Lyre Skeleton clock.

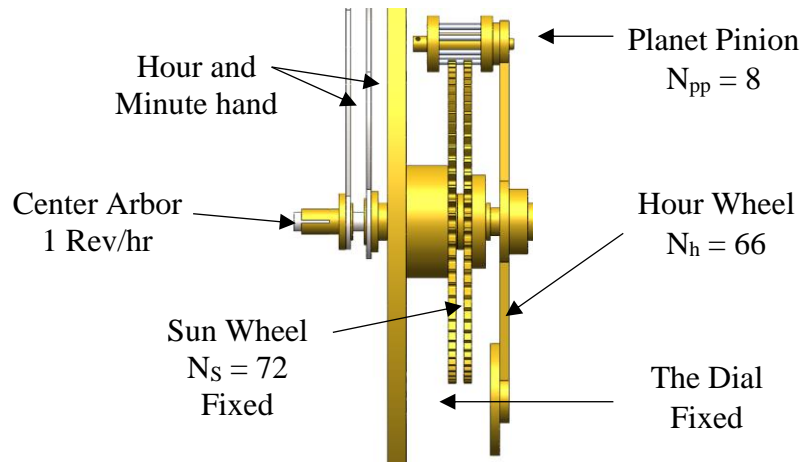


Figure 4.7: Diagram of the Interaction Between the Planet Pinion, Sun Wheel, and the Hour Wheel on the Lyre Skeleton Clock. Made in SOLIDWORKS

The sun wheel is fixed however the planet pinion is being rotated about the center arbor, in turn causing the planet pinion to rotate about its axis. The gear ratio between the planet pinion and the center arbor can then be determined using the equation:

$$GR_{c,pp} = 1 + \frac{N_s}{N_{pp}} \quad (2)$$

where $GR_{c,pp}$ is the gear ratio between the center wheel and the planet pinion, N_s is the number of teeth on the sun wheel, and N_{pp} is the number of teeth on the planet pinion [28].

Note that the sun wheel and the hour wheel have different numbers of teeth, however, both have the same pitch diameter and are meshed with the planet pinion. This is the aforementioned Ferguson Mechanical paradox. This works because there is slop between the planet pinion and the sun wheel. The following equation can be used to calculate the gear ratio between the center wheel and the hour wheel:

$$GR_{c,h} = \frac{N_h}{N_h - N_s} \quad (3)$$

where $GR_{c,h}$ is the gear ratio between the center wheel and the hour wheel, N_h is the number of teeth on the hour wheel, and N_s is the number of teeth on the sun wheel. With this equation, it was determined that the gear ratio between the center wheel and the hour wheel is 12:1. This makes sense as the center wheel is attached to the minute hand of the clock and the hour wheel is attached to the hour hand of the clock.

A set of gears transfers torque from the center wheel to the third wheel, then the third wheel the escape wheel. Torque is equal to force times distance. This means that even if the torque output of the fusee is constant the gear train accelerates. To control this acceleration the escape wheel is completely stopped by the escapement two times every period. That is what creates the iconic “Tick Tock” of old mechanical clocks. These sounds are called beats. This mechanism generates the same speed profile between every beat of the clock meaning the clock tells constant time. Figure 4.8 illustrates the escape wheel meeting the escapement in both positions that a beat can occur. Note that the escape wheel rotates half a tooth between beats. The energy transferred from the escape wheel to the escapement is transferred through the escape arbor to the pendulum where it counteracts the air friction on the pendulum to keep in moving.

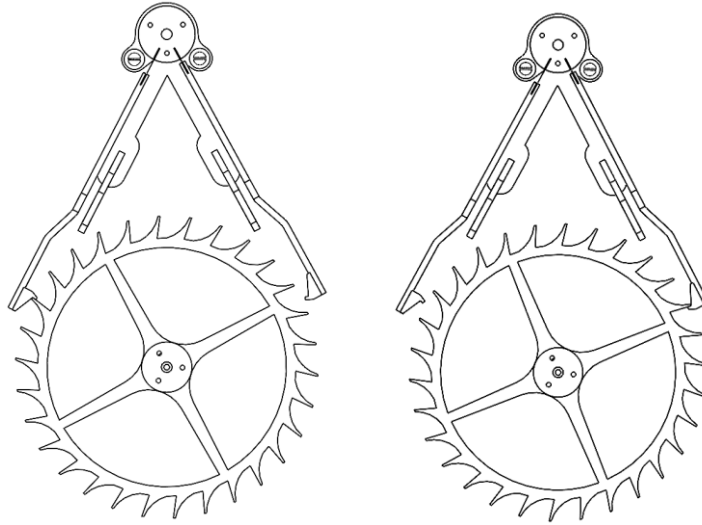


Figure 4.8: Escapement Wheel and Escapement Illustrating Both Positions of a Beat can Occur

4.4: Torque Required to Drive the Gear Train

The most practical way to determine the torque required to drive a clock is to attach an adjusting rod to the great wheel arbor, get it as close to horizontal as possible, and then adjust the counterweight along the rod until the gear train moves. From there the torque required to drive the gear train can be determined using the weight of the counterweight and the distance it was away from the center of the arbor. This is done before the fusee is made and the spiral spring is installed. Figure 4.9 is a figure from W.R. Smith's *How To Make A Strutt Epicyclic Train Clock*, illustrating this method for determining the torque required to drive a clock.

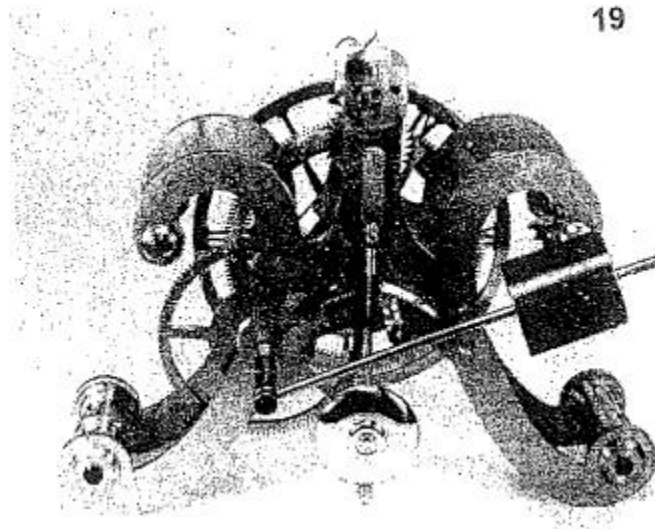


Figure 4.9: A Method to Determine the Torque Required to Drive a Clock

Source: [29]

The gear train of the Grasshopper Skeleton Clock is very similar to the Lyre Skeleton Clock [3] [27]. The torque required to drive the gear train of the Grasshopper skeleton clock was measured to be 9 lbf-in. Thus, for this thesis, the torque required to drive the Lyre Skeleton Clock will be assumed to be 9 lbf-in.

4.5: Spiral Spring Spring Constant

In July of 1971 W. Swift performed an analysis in which he determined the center point and equation of various spiral springs. He determined that free spiral springs follow the equation of a logarithmic spiral whereas fully wound spiral springs follow the equation of an Archimedes spiral [5]. The equation for a logarithmic spiral is given by [30]:

$$r = r_0 e^{b\theta}, \quad (4)$$

where r_0 is the distance from the origin, θ is the angle from the x-axis, and a and b are constants. Using modern techniques his methods were used to determine the spiral spring constant of the spiral spring used to power the Lyre Skeleton Clock.

All points on the curve of a logarithmic spiral have tangent lines that are perpendicular to a line that is coincident with the origin. This property is exploited and used to determine the center point of the spiral spring. First, the spring is photocopied, inserted into SOLIDWORKS, and scaled to size. It is scaled by photocopying the spring with a ruler in it.

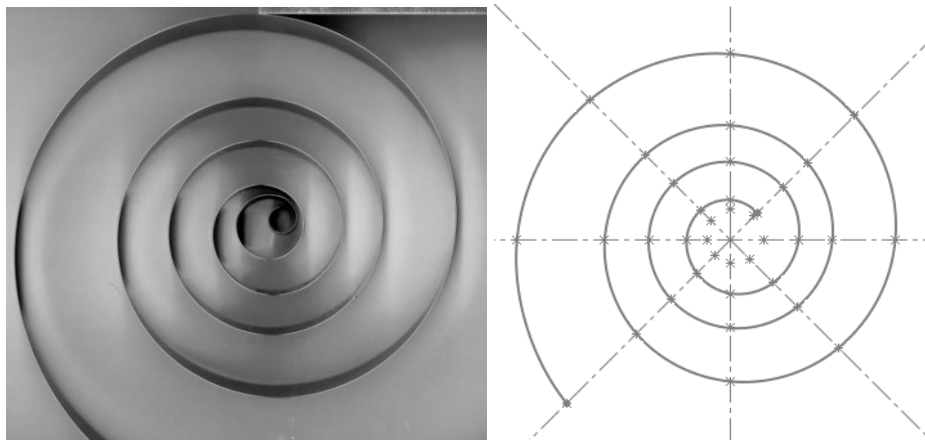


Figure 4.10: Left: Photocopied Spiral Spring Inserted into SOLIDWORKS and Traced with a Spline. Sketch Points Inserted Every $\pi/4$ Radians. Right: The Spline and Sketch Points Without the Photocopied Background

At this point in the process, the origin is assumed. Construction lines are produced to place the sketch points, one every $\frac{\pi}{4}$ radians starting at 0 radians. The construction lines and sketch points will be reevaluated after the center point is empirically determined.

Every revolution of the spiral is separated into separate sketches and two lines perpendicular to the tangent line are produced from each sketch point. The free endpoints of the lines are then merged with the free endpoints to adjacent lines to produce a total of eight approximated center points per revolution. For higher fidelity data increase the sketch points per revolution.

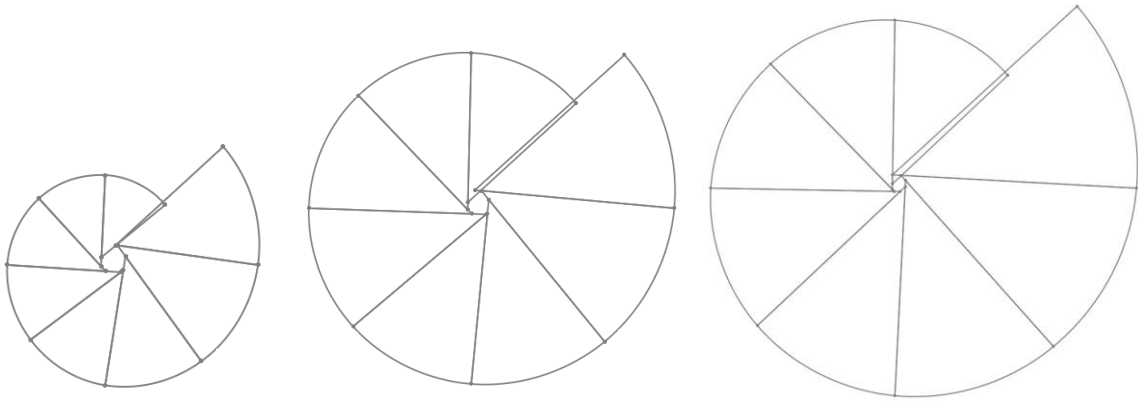


Figure 4.11: Left: Spiral from $\theta = 3.75\pi$ to 5.75π radians, Tangent Lines, and Approximated Centers. Middle: Spiral from $\theta = 5.75\pi$ to 7.75π radians, Tangent Lines, and Approximated Centers. Right: Spiral from $\theta = 7.75\pi$ to 9.75π radians, Tangent Lines, and Approximated Centers.

Determining which revolutions of the spiral spring to use when calculating the center point is key. By inspection, the spiral spring does not follow the equation of logarithmic spiral until about 3.75π radians. This is because of the manufacturing process of the spring [5]. Due to this, all data is excluded from 0 radians to when the spring starts following the equation of a logarithmic spiral. The error in the perpendicular line becomes magnified as the radius of the spiral increases. This is because the error is in the slope of the line, which has more distance to propagate as the radius increases. For springs larger than the one analyzed in this paper this error should be accounted for.

The approximated center points are then converted into a new sketch and the Visual Basic Macro PointtoExcel is used to export XY data to excel (see Appendix D) [31].

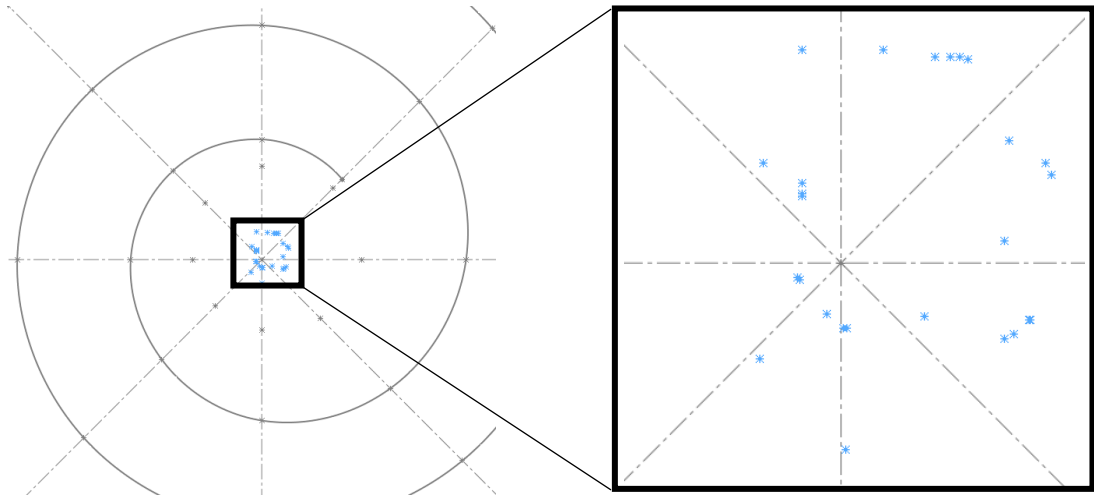


Figure 4.12: Approximated Center Points and Their Relation to the Assumed Center Point

The X values and Y values are then averaged to produce the calculated center point.

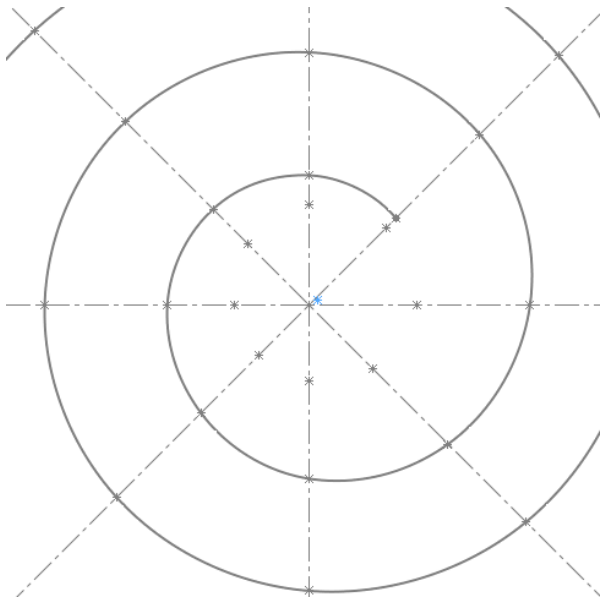


Figure 4.13: The Calculated Center Point and its Relation to the Assumed Center Point

In this instance, the calculated center point and the assumed center point are 0.0769 in away from each other. To give some context, the largest radius of the spring, calculated at 10.75π radians, was 5.8994 in. This is a very small difference in center points; however, this method should work for large differences as well.

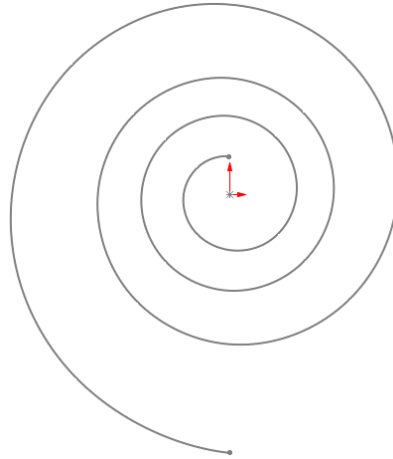


Figure 4.14: Spiral Spring Centered on the Origin (Origin Indicated by the Red Arrows)

The spiral is then converted into a new sketch and the center point is plotted on that sketch. The whole sketch is then highlighted and made into a Sketch Block, meaning it is free to move but fixed relative to itself. The center is then made coincident with the origin of the sketch and the outer endpoint of the spiral is made vertical with the origin/center point.

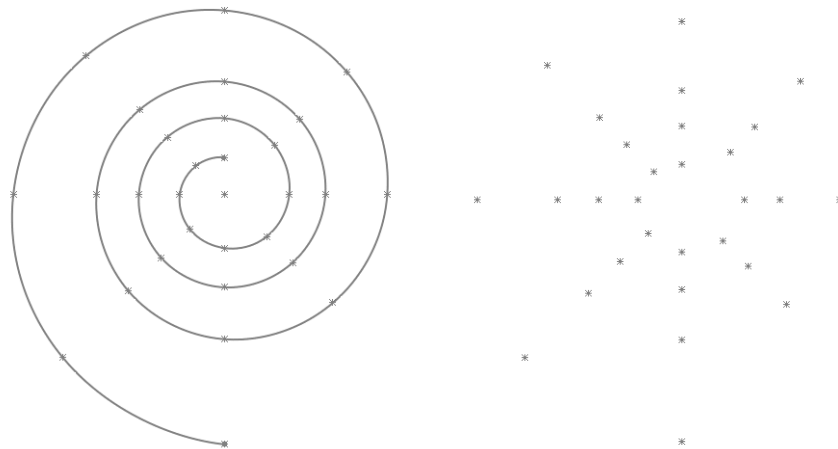


Figure 4.15: Spiral Spring Data Points Plotted in SOLIDWORKS

Then all points on the intersection between the spiral and XY axes are plotted and converted into a new sketch and the Visual Basic Macro PointtoExcel is used to export XY data to excel (see Appendix D) [31]. This data is then sorted and used to create the plots below. Figure 4.14 shows the radius of the spiral spring plotted against the angle. The equation for the trendline calculates the r_0 and b for this spiral spring to be 0.8775 and 0.0798 respectively. Figure 4.15 shows the measured versus predicted spiral spring position.

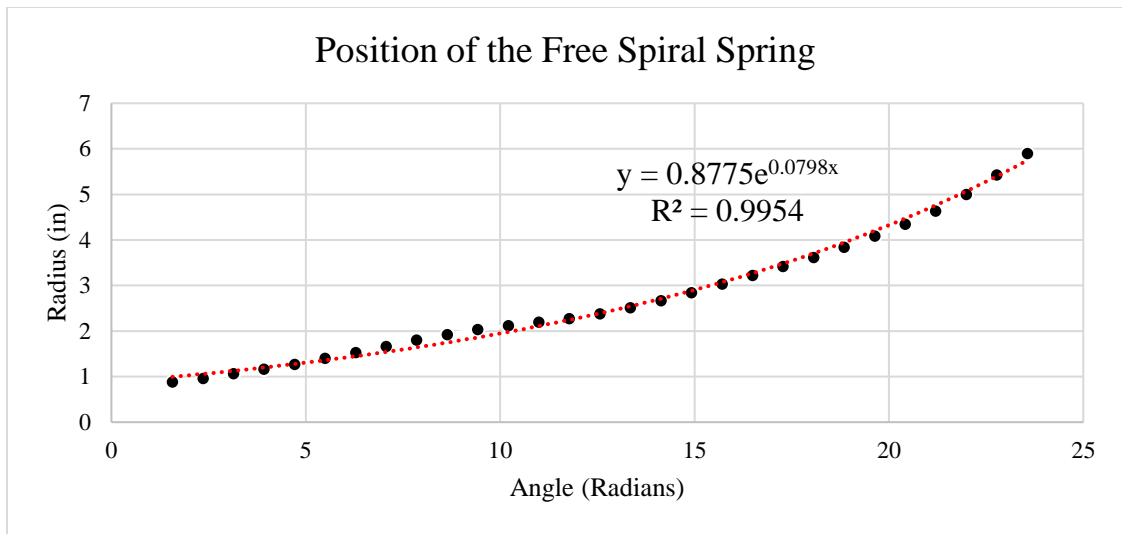


Figure 4.16: Radius vs Angle Position of the Spiral Spring

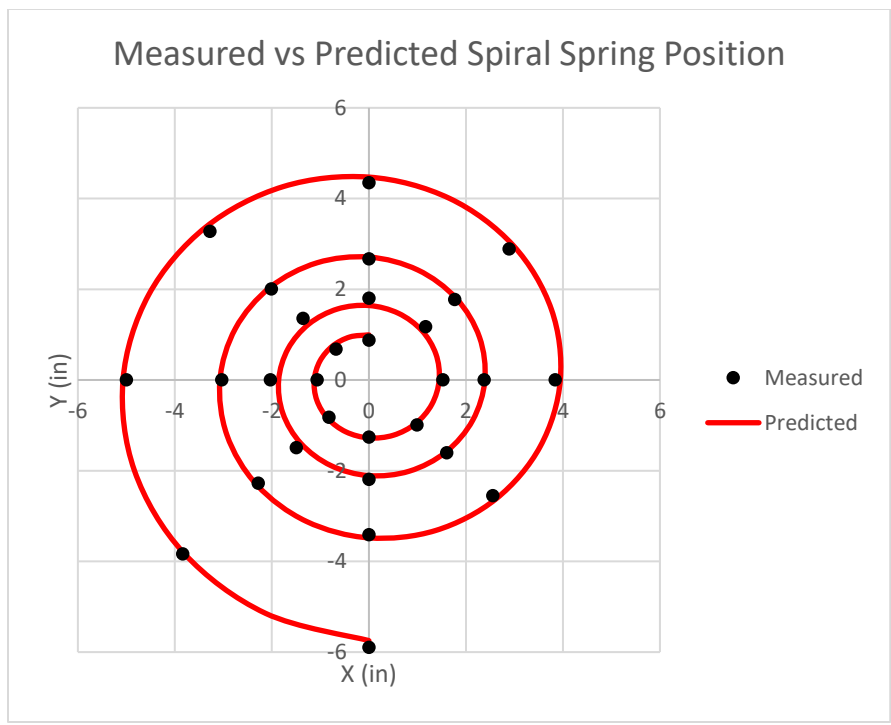


Figure 4.17: Predicted vs Measured Spiral Spring Position

After r_0 and b are determined they can be plugged into a system of equations originally derived by W. Swift in An Analysis of the Spiral Spring to determine the spring constant [5]. Edwin Odom recreated this analysis and determined the spring constant of this spiral spring to be 0.254 lbf/in.

4.6: Methods for Modeling a Fusee

This section illustrates the methods to create the parametric SOLIDWORKS model used in the fusee design tool described in Figure 4.3. In modern solid modeling, it is considered good practice to remove threads from a model to lower the number of calculations the modeling software is doing, thus decreasing loading time. This section will discuss adding threads to a concave cone to model a fusee accurately (see Figure 4.18). This is done for two reasons. First, if the user has a complete model, they can rapidly test the fusee design by 3D printing it, attaching it to a spiral spring in a barrel, and using a torque wrench to determine if it has normalized the torque curve. Secondly, if the user is so inclined, they can 3D print a fusee as a cheaper method of manufacturing a final part. This model only generates the profile and the threads of the fusee. Other features are such as the arbor hole vary depending on the implementation of the fusee and are simple features that can be created by the user.

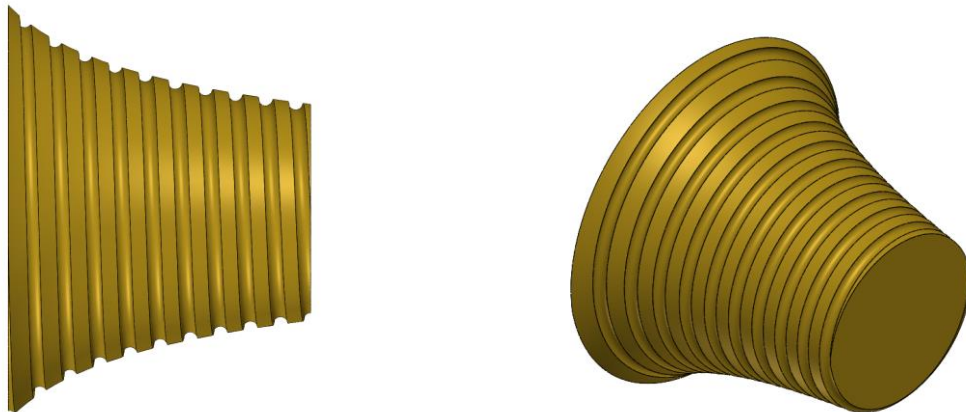


Figure 4.18: Left: Front View of a Solid Modeled Fusee. Right: Isometric View of a Solid Modeled Fusee

The parametric SOLIDWORKS model of a fusee was first made completely isolated from all other files. A design table was then used to link the SOLIDWORKS part file to the output of the Microsoft Excel file. To start, the sketch illustrated in Figure 4.19 was created by manually inputting the output of the Excel document. Then the spline tool was used to connect all the points.

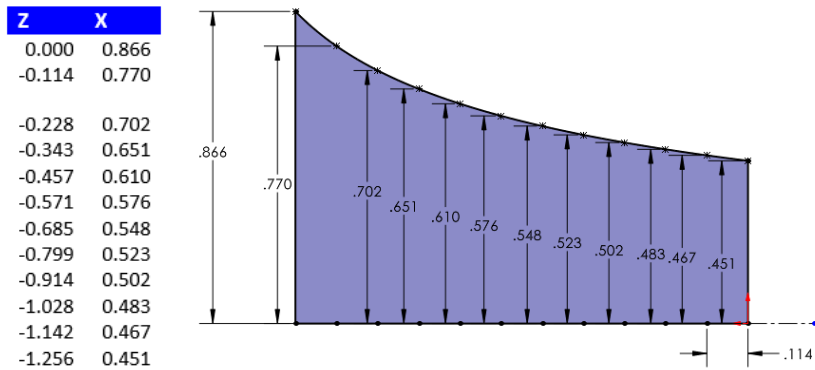


Figure 4.19: Left: Excel table generated from the TK solver code. Right: Corresponding SOLIDWORKS Sketch

The sketch was then revolved around the centroid of the fuselage as illustrated in Figure 4.20. Using this revolved profile and the number of threads per inch of a given fuselage one can create a path for the thread to follow in the form of a 3D sketch. Throughout the rest of this chapter, this path will be referred to as the thread path.

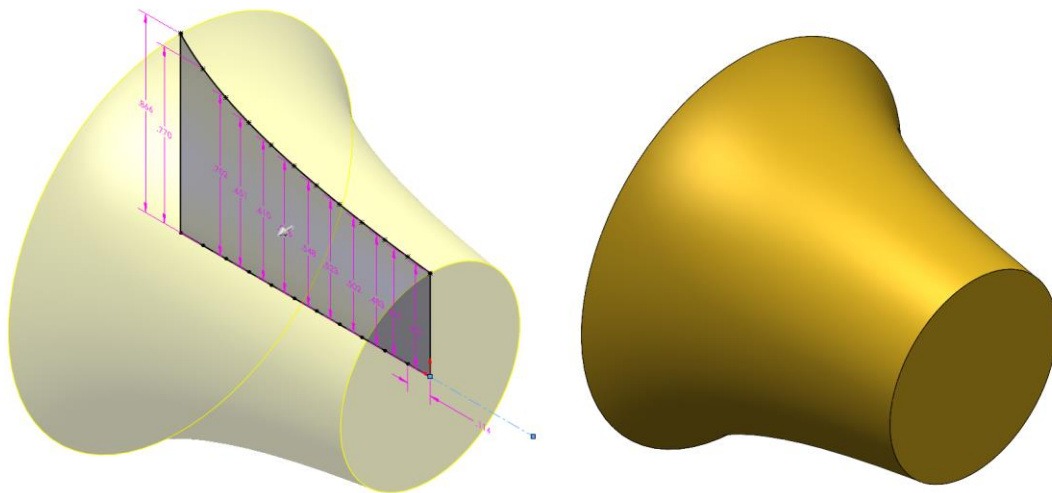


Figure 4.20: Left: Sketch Before Revolution About the Centroid of the Fuselage. Right: SOLIDWORKS Model After the Sketch is Revolved

The thread path needs to continue past the edge of the profile otherwise the thread will not continue through the entire solid body. The resulting thread will have an end condition like the one illustrated in Figure 4.21.

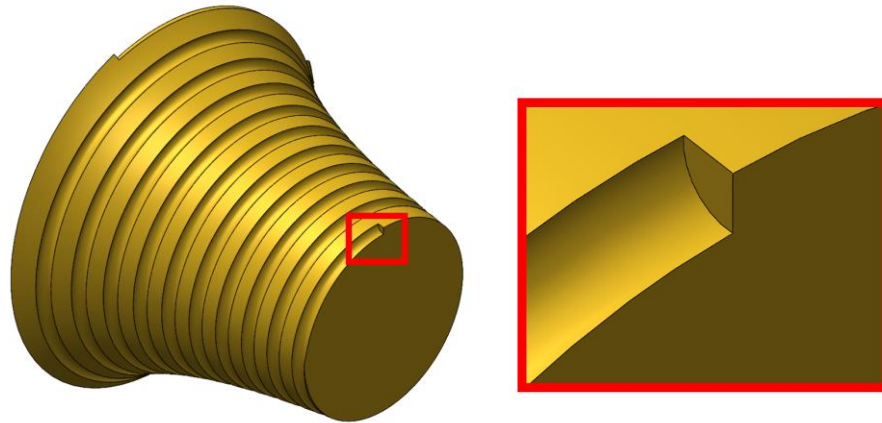


Figure 4.21: Left: Zoomed-out Picture of a Fusee Without Proper Thread Lead-in. Right: Detailed View of the End Condition of a Fusee without a Proper Lead-in

To resolve this problem, the profile is extended out past where it is calculated, then the thread path is created using the edge of the extended profile. Figure 4.22 shows the profile of the fusee extended out by 0.25 inches.

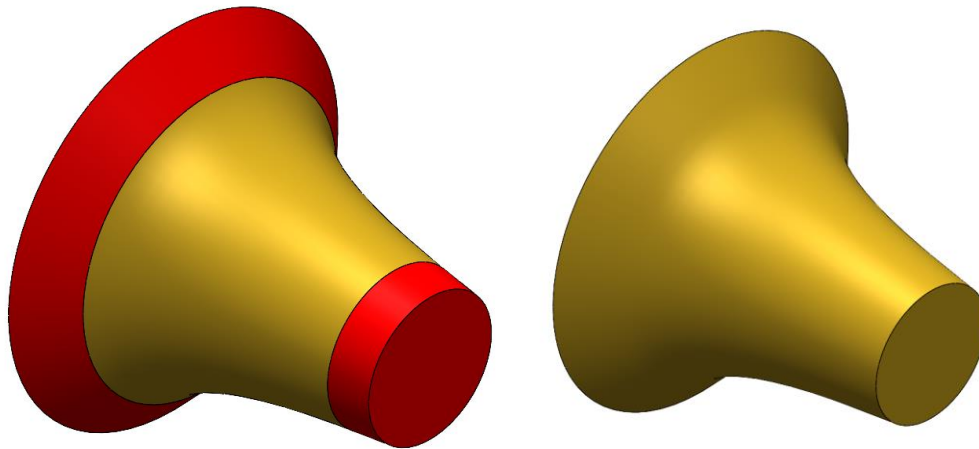


Figure 4.22: Extended Fusee Profile with Extensions Highlighted in Red. Right: Extended Fusee Profile

A line segment that is perpendicular to the centroid of the fusee was then sketched. One endpoint should be coincident with the centroid of the fusee as well as the leading edge of the extended profile. The line segment should be longer than the largest radii of the fusee. For the purposes of this paper, this line will be referred to as the Surface profile. Figure 4.23 shows an example, in this case, the line segment is defined as 0.125 inches

longer than the largest radii of the fusee. This makes the model parametric. For example, if one were to define the line segment as 1 inch long and generate the profile of a fusee that had radii longer than one inch, the model would break.

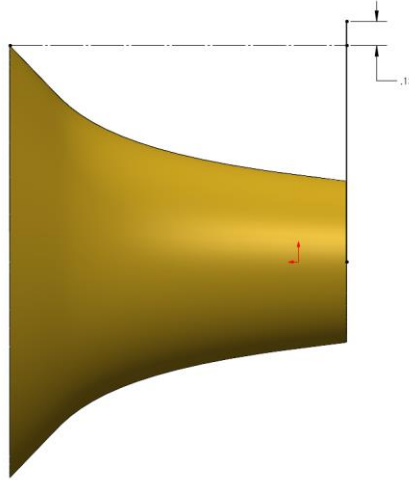


Figure 4.23: The Surface Profile

Next, a line segment that is in line with the centroid of the fusee was sketched. One endpoint should be coincident with the centroid of the fusee as well as the leading edge of the extended profile. The line segment should be one hundred times the pitch of the thread path. This dimension will later be linked to the TK Solver output through a design table as one-hundred times the pitch of the thread path. For the purposes of this paper, this line will be referred to as the surface path. Figure 4.24 shows an example, in this case, the line segment is defined as 12.5 inches long, meaning the pitch of the thread path is 0.125 and the fusee will have eight threads per inch.

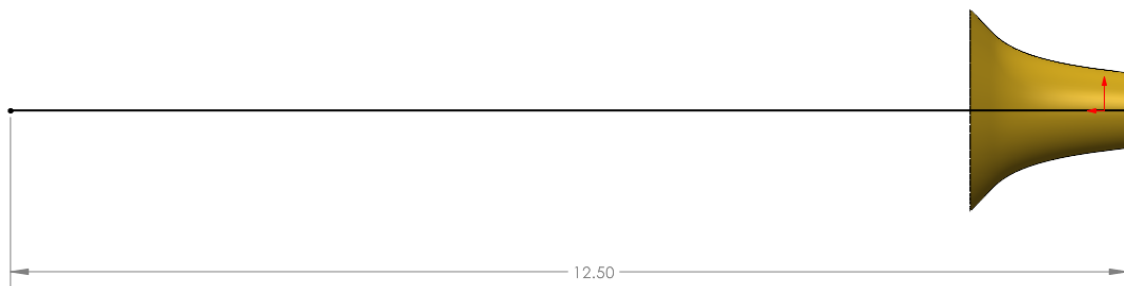


Figure 4.24: The Surface Path

The surface profile was then swept about the surface path revolving it one hundred times (illustrated in Figure 4.25). The length of this surface, in this example 12.5 inches, is the limiting factor for the length of the extended profile of the fusee. 12.5 inches is longer than most standard fusees. However, if the threads per inch are increased, the pitch decreased thereby decreasing the maximum limit for the length of the extended profile. Thus, there could be a scenario in which this model will fail. This is a fringe case, for example, this fusee would need to be 60% larger and the TPI would need to increase from 8 to 50 for the model to fail in this way.

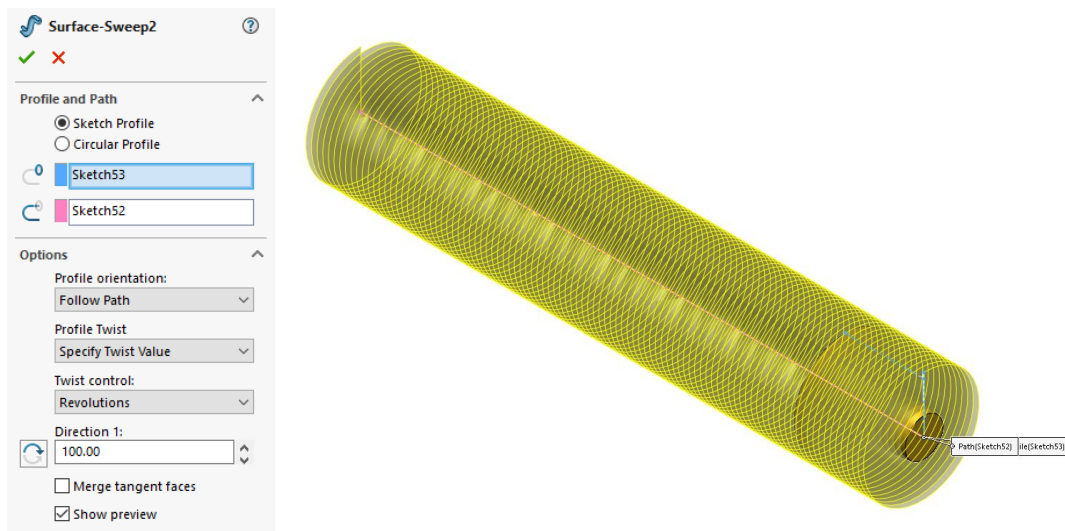


Figure 4.25: The Swept Surface Being Generated in SOLIDWORKS

When originally modeling this, the surface path was made coincident with both sides of the extended profile and the number of revolutions was a parametric parameter. This method generated rebuild errors every time the threads per inch were updated. The method illustrated here was used to work around this issue.

Next, the back face of the fusee and the extended profile were used to trim the swept surface. The outer edge of the trimmed surface is the thread profile (illustrated in the left image of Figure 4.26).

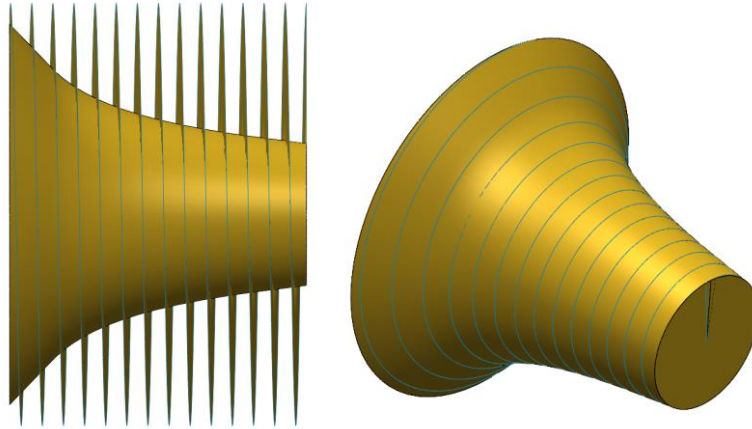


Figure 4.26: Left: Front View of the Fusee and Swept Surface After Trimming the Surface with the Back Face of the Fusee. Right: Isometric view of the Fusee and Swept Surface after Trimming with Both the Back Face and Extended Profile

The thread profile was then converted into a 3D sketch and the extended profile and the swept surface were deleted. Just the thread path of the fusee should be left (See Figure 4.27).

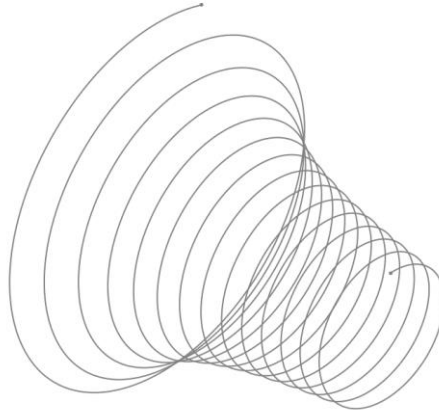


Figure 4.27: The Thread Profile of a Fusee in the Form of a 3D Sketch

The profile of the fusee was then recreated using the original sketch (no extended edges). Then a circular thread profile was created on a plane orthogonal to the thread path at one of the endpoints of the path (see Figure 4.28). The diameter of the profile was set to be equal to the diameter of the wire connecting the fusee to the spiral spring.

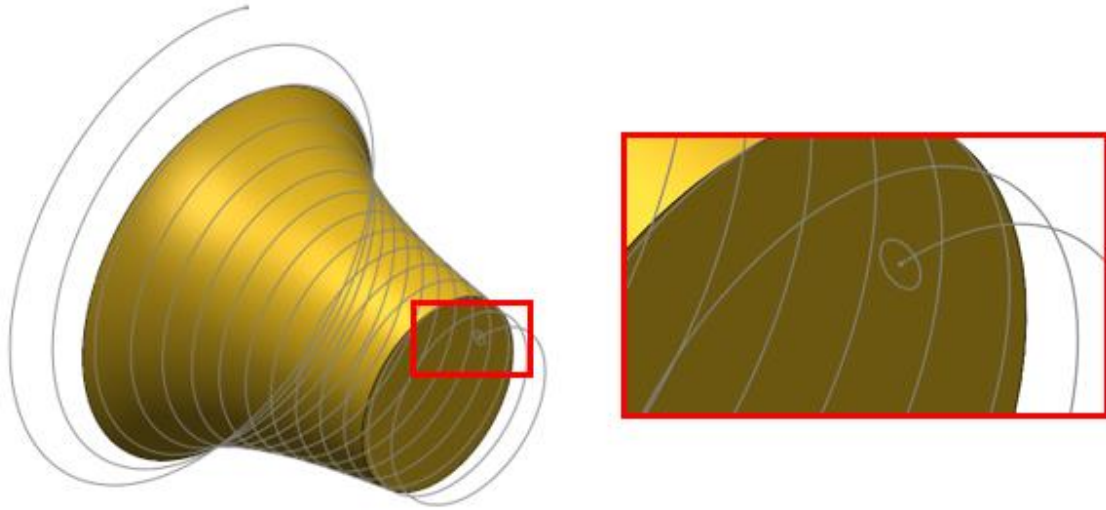


Figure 4.28: The Thread Profile of a Fusee

The thread profile was then swept cut along the thread path (illustrated in Figure 4.29). This is all the complicated geometry of the fusee. There are features such as the arbor hole that are not included in this model. As they are easy to produce and can be included later.

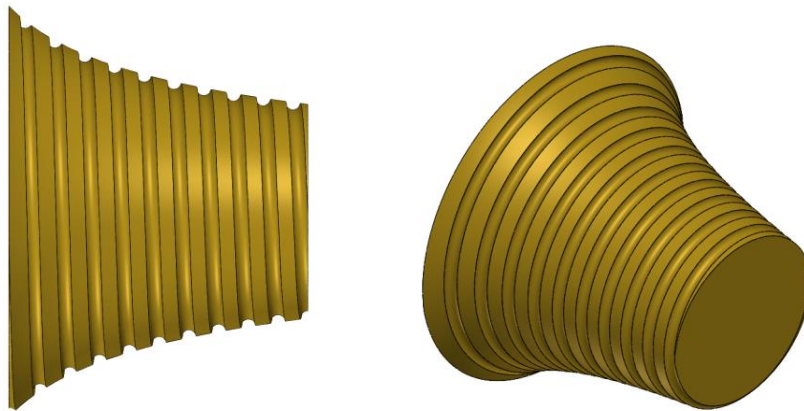


Figure 4.29: Left: Front View of a Solid Modeled Fusee. Right: Isometric View of a Solid Modeled Fusee

To make the model parametric all the parameters illustrated in Table 4.1 were linked to a design table. The design table was then be linked to the output of the Microsoft Excel file.

Table 4.1: SOLIDWORKS Generated Design Table Reformatted for use in this Paper

Design Table for: Fusee		Default
	xIncrement@Sketch1	0.114
	y1@Sketch1	0.866
	y2@Sketch1	0.770
	y3@Sketch1	0.702
	y4@Sketch1	0.651
	Y5@Sketch1	0.610
	Y6@Sketch1	0.576
	Y7@Sketch1	0.548
	Y8@Sketch1	0.523
	Y9@Sketch1	0.502
	Y10@Sketch1	0.483
	Y11@Sketch1	0.467
	Y12@Sketch1	0.451
	PitchX100@Sketch52	12.500
	ThreadDiameter@Sketch54	0.063

Figure 4.30 was included to illustrate what some of the values in Table 4.2 correlate to in the model. The values not shown in Figure 4.30 include PitchX100@Sketch52, which correlates to the length of the surface path, and ThreadDiameter@Sketch54 which correlates to the diameter of the thread profile.

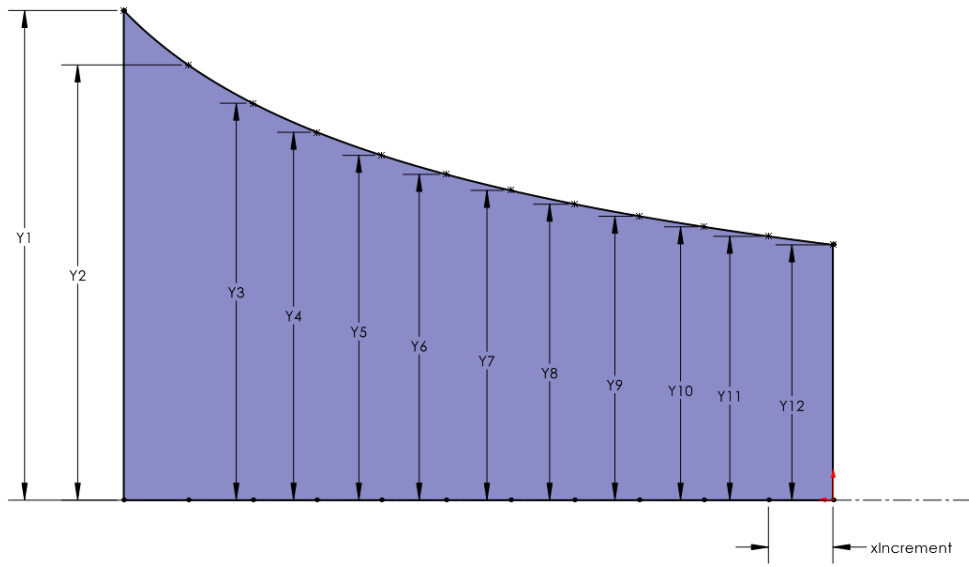


Figure 4.30: Sketch1 Annotated to Illustrate the Location of the Values in the Design Table

Note for fusee manufacturing: the coordinate system in the model does not match the coordinate system for manufacturing a fusee on a CNC lathe. The coordinate transformation illustrated in Figure 4.31 solves that problem.

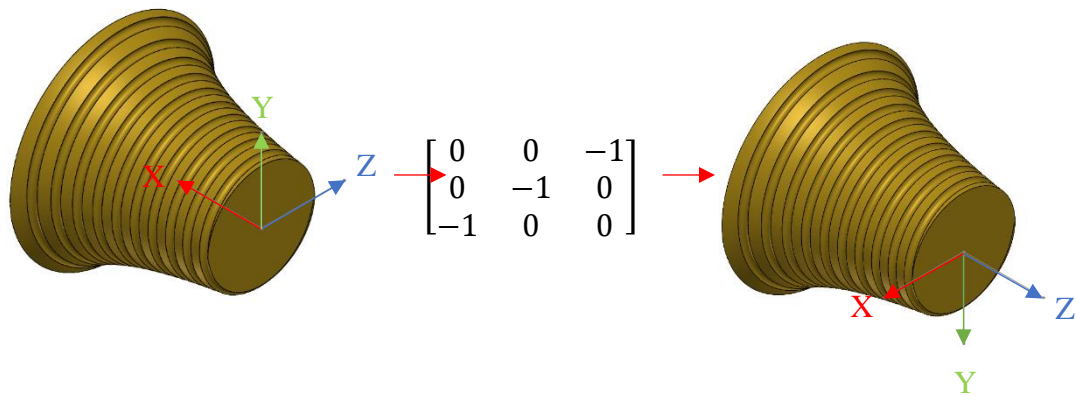


Figure 4.31: Left: the Coordinate System of the SOLIDWORKS Model, Middle: Transformation Matrix Between the SOLIDWORKS Model and the CNC Lathe, Right: the Coordinate System of the CNC Lathe

4.7: Modern Method for Fusee Manufacturing

This section describes the parametric G-code generated by the fusee design tool described in Figure 4.3. A G76 command can be used to create threads/tapered threads on a CNC lathe, it cannot however create nonlinear tapered threads [32]. This means that using the threading feature on a standard Computer-Aided Manufacturing (CAM) package will not generate the tool path required to manufacture a fusee. The G-code would instead need to be written manually. To make manufacturing easier for the everyday horologist the fusee design tool was set up to generate G-code for a Hass CNC lathe to machine a fusee automatically based on inputs such as radius of the barrel (in), required torque (in-lbs), spiral spring spring constant (lb/radian), and pitch. The CNC program below only manufactures the threads of the fusee. The profile of the fusee can be generated using a standard CAM package and the SOLIDWORKS model generated in section 5.2 Methods for Modeling a Fusee. Note that every time G-code is generated from the fusee design tool it should be treated as new code and should be watched throughout the whole program.

The coordinate system for this G-code is illustrated in Figure 4.32. Note that the tool offsets for the grooving tool are such that when it is at coordinate (0,0) the tip of the cutter will be coincident with the origin and will be centered about the x-axis.

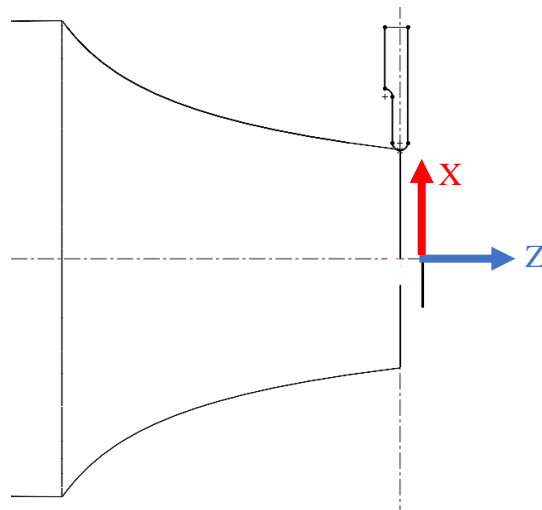


Figure 4.32: Coordinate System Used in the G-code Generated by the Fusee Tool

The G-code commands used in the Fusee Design Tool are translated in Table 4.2. This table is meant to aid in the understanding of figures presented later in this chapter.

Table 4.2: Translating the Turning G-Code Commands used in the Fusee Design Tool G-code

Command	Translation
%	Signifies the start and end of the code
O	Followed by the program number
N	Line indicator
T0101	Tool one offset one
G97	Constant Surface Speed off
G99	Feed per Revolution
G40	Tool Nose Compensation Cancel
G80	Cancel Canned Cycle
G00	Rapid Motion Positioning
M03 S100	Rotates the spindle at 100 revolutions per minute
Z	Followed by a number that brings the tool post to a specified position along the Z-axis
X	Followed by a number that brings the tool post to a specified position along the X-axis
M19 S45	Rotates the Spindle to 45 degrees and stops it
G01	Linear Interpolated Feed
F	Followed by a number that indicates the feed rate of the tool post
M05	Turns off the Spindle
M30	End of the Program

Figure 4.33 illustrates a reformatted version of the first couple lines of G-code generated by the fusee design tool for the Lyre Skeleton Clock Fusee. This code should be run in Single Block until the tool offsets are confirmed. For all the G-code presented in this paper the bold font indicates parametric values. The code starts by initializing the program, and the program number is input. The program number is an input variable defined by the user in the Excel document and must be five digits or less. The comment 0.05 FULL

ROUNDRADIUS GROOVING TOOL, indicates to the operator what grooving tool should be used for this operation. The parametric value 0.05 is defined by the specified wire diameter. The code selects tool one, offset one, which means the operator should set their grooving tool to be tool one, offset one before running the program. Constant Surface Speed is then turned off, this stops the programs from automatically increasing or decreasing spindle speed based on the X position to ensure that the spindle speed is constant, thus the pitch will not be modified by changes in spindle speed. The lathe is then set to feed rate per revolution as opposed to feed rate per minute. This makes visually checking the code easier because it means for linear threading operations, the feed rate per revolution is equal to the pitch of the fusee. Tool nose compensation is turned off to ensure that the tip of the cutter will follow the calculated fusee thread path, the ramification of this will be discussed further later in the chapter. The machine is then set to rapid movement mode, meaning it will travel as fast as possible to any position indicated.

```

%
O301 (Thread Fusee)
N1 (0.05 FULL ROUND RADIUS GROOVING TOOL)
N1 T0101 (0.05 FULL ROUND RADIUS GROOVING TOOL)
N2 G97 (Constant Spindle Speed)
N3 G99 (Feed Rate per revolution)
N4 G40 G80 G00

```

Figure 4.33: First Couple of Lines of G-code Generated by the Fusee Design Tool

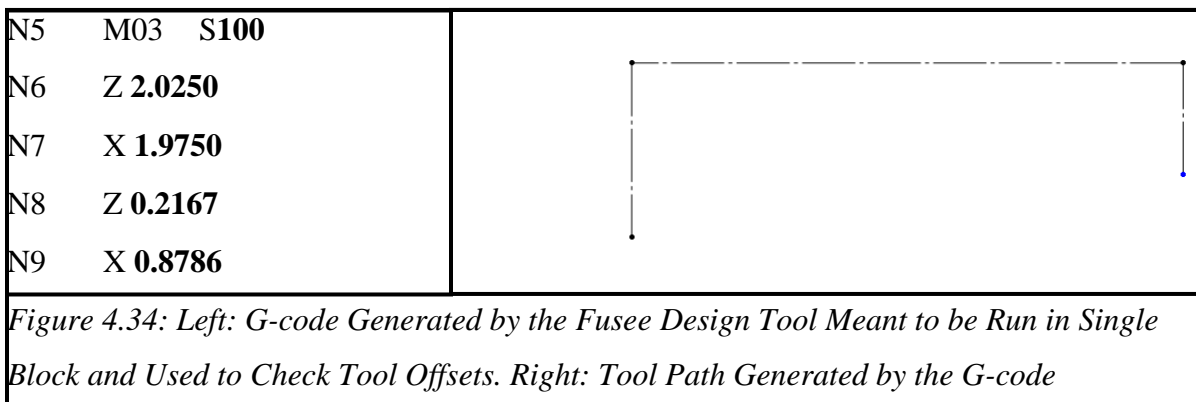
Figure 4.34 shows the g-code and the tool path used, to check tool offsets, and position the cutter for the first pass. To start, the spindle is rotated clockwise at 100 revolutions per minute. Then, the tool carriage is positioned such that the leading edge of the cutter is 2 inches away from the origin. This is calculated by adding the wire radius i.e., the radius of the grooving tool to 2 inches. This gives the operator a chance to stop the spindle and measure the distance between the cutter and the part to see if the tool offsets were set up correctly. The cutter is then moved in the X dimension to the largest diameter of the fusee plus the user input retreat distance. Throughout the rest of this chapter, this X position will be referred to as the safety distance. Then the cutter is moved such that it is two pitches plus the

diameter of the cutter away from the part plus in the Z. This distance is important and will be repeatedly used throughout the program to line up the thread between passes. For the purposes of this paper, this Z position will be referred to as the thread offset position.

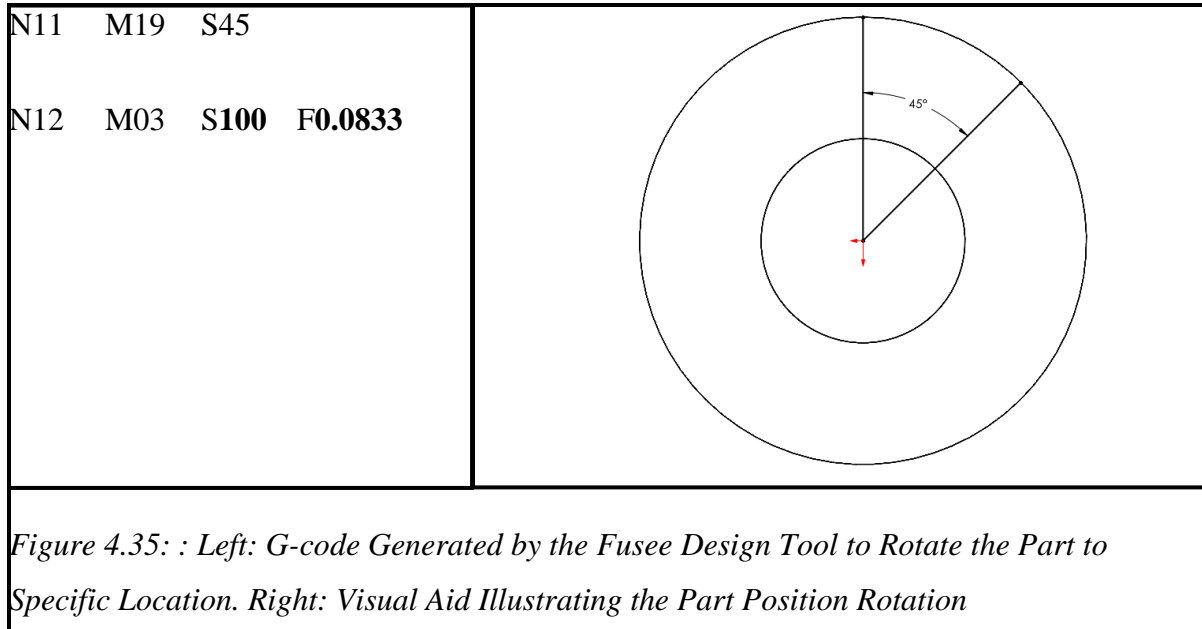
The output of the TK Solver code generates the radius at which the center point of the wire rests at any given point along the length of the Fusee. Haas CNC lathes operate in terms of diameter. Furthermore, through trial and error, it was also determined that the minimum thread depth to securely rest a wire in a fusee is equal to the diameter of the wire. Thus, to ensure that the center point of the wire is following the correct path and the wire does not dislodge from the fusee, the crest of the fusee threads, as input into a CNC lathe, must follow the equation

$$X_{crest} = (2 * r) + d_{wire}, \quad (5)$$

where X_{crest} is the X coordinate input in the CNC lathe, r is the calculated radius along with the profile of the fusee, and d_{wire} is the wire diameter. Using equation 5, the cutter is then moved to the X position of the first point along the crest profile. Note that the dashed lines in the tool path diagrams indicate the tool is in rapid movement mode. Solid lines indicate that the cutter is feeding using linear interpolation motion.



Now that the cutter is ready to make the first pass the workpiece must be rotated and lined up so that every cut follows the same thread path. Figure 4.35 illustrates the code to complete the maneuver. The spindle is stopped and then oriented such that it is at 45 degrees from 0. The spindle is then turned back on and the feed rate is set to the pitch of the fusee.



The first pass moves the tip of the cutter along the profile of the blank. Figure 4.37 shows a snippet of the G-code required to perform this action and the tool path that the G-code generates. The machine is set to linear interpolation motion. The cutter is then moved on to the part and follows a linear interpolated version of the crest profile. As the profile of the fusee becomes steeper the feed rate of the cutter increases. Normally, in threading operations, the feed rate is held constant to maintain a constant pitch. However, the threads per inch specified on a fusee refer to the number of threads per inch along the Z-axis. Note that when using linear interpolation motion “the speed of all axes is controlled so that the feed rate specified is achieved along the actual path” [33]. This means that if the cutter is traveling at an angle the feed rate must increase to project the correct feed rate along the Z-axis. The following equation illustrates a continuous solution to determine the feed rate of the cutter

$$F = \frac{p \cdot \sqrt{\left(\frac{dx}{2}\right)^2 + dz^2}}{|dz|} = \cos \left(\arctan \left(\frac{1}{2} \frac{dx}{dz} \right) \right), \quad (6)$$

where F is the feed rate of the cutter (inches/revolutions), p is the pitch of the fusee (inches/revolution), x is the diameter of the fusee (in), and z is the length along the fusee (in). Since this path uses linear interpolation, the equation simplifies to

$$F = \frac{p^* \sqrt{\left(\frac{\Delta x}{2}\right)^2 + \Delta z^2}}{|\Delta z|} = \cos\left(\arctan\left(\frac{1}{2} \frac{\Delta x}{\Delta z}\right)\right), \quad (7)$$

where all the variables are the same except instead of using an instantaneous position change to the distance between two points. Figure 4.36 shows the radius of a fusee and the feed rate of the cutter versus the length along the fusee.

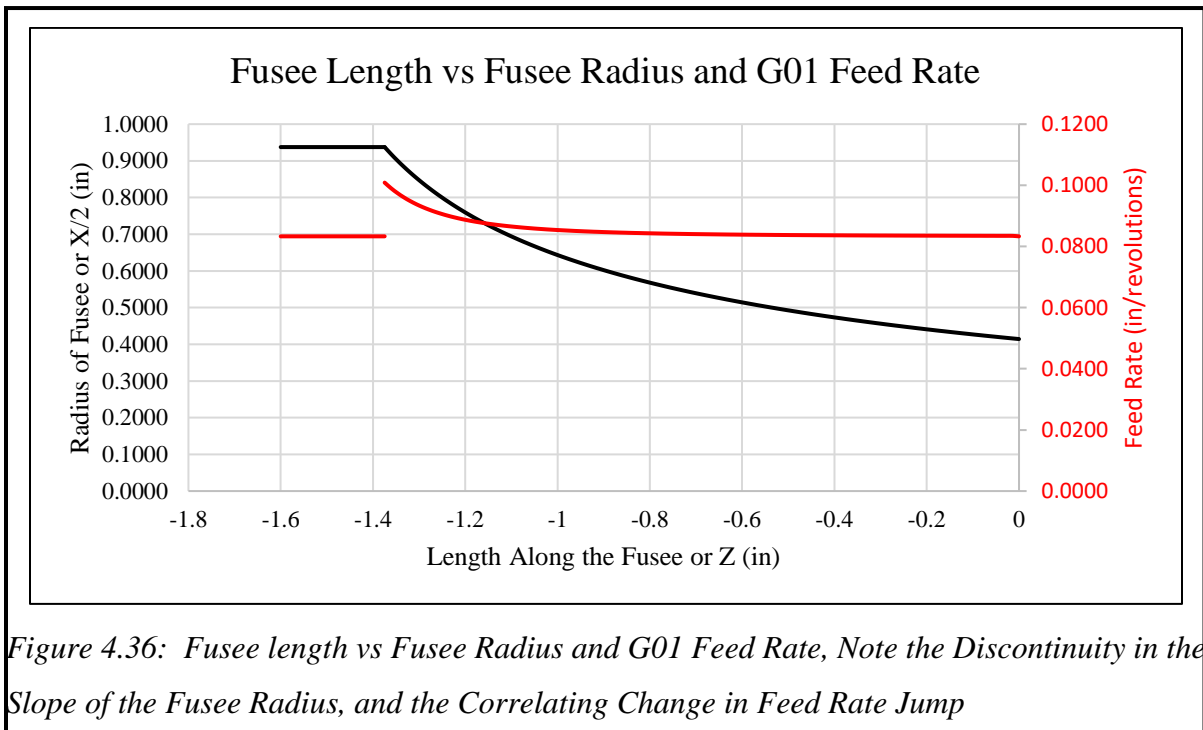
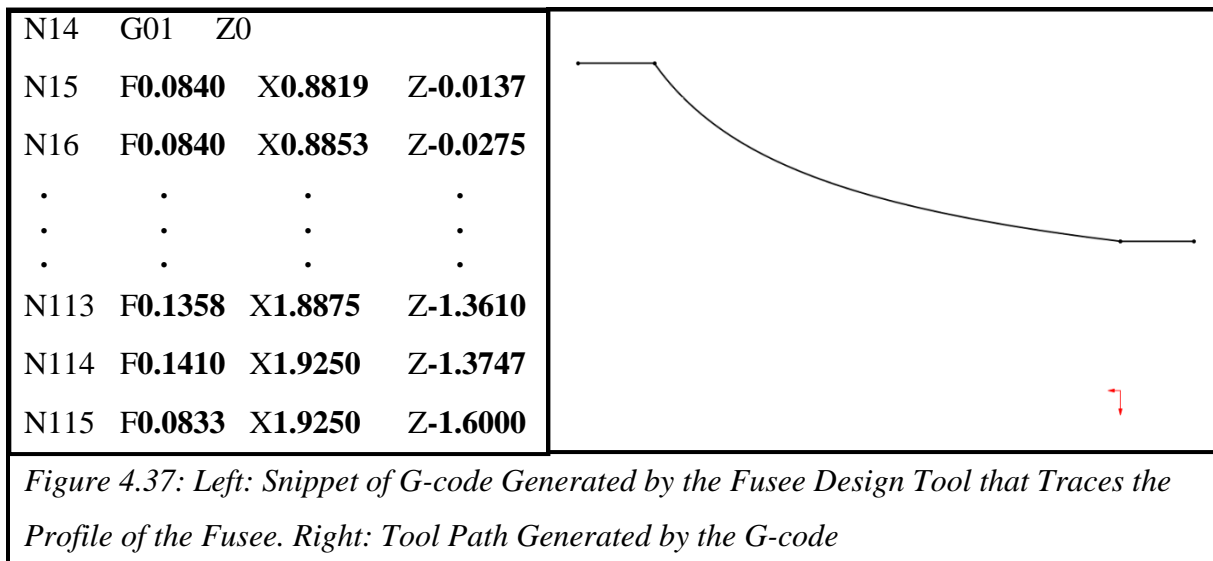
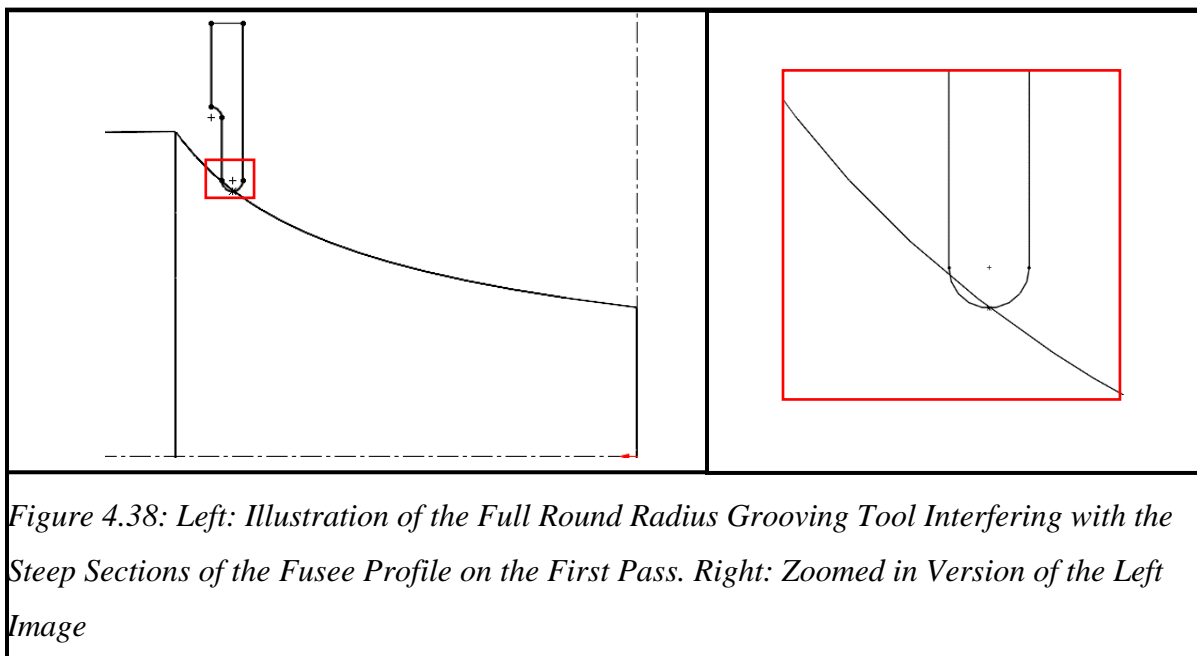


Figure 4.36: Fusee length vs Fusee Radius and G01 Feed Rate, Note the Discontinuity in the Slope of the Fusee Radius, and the Correlating Change in Feed Rate Jump

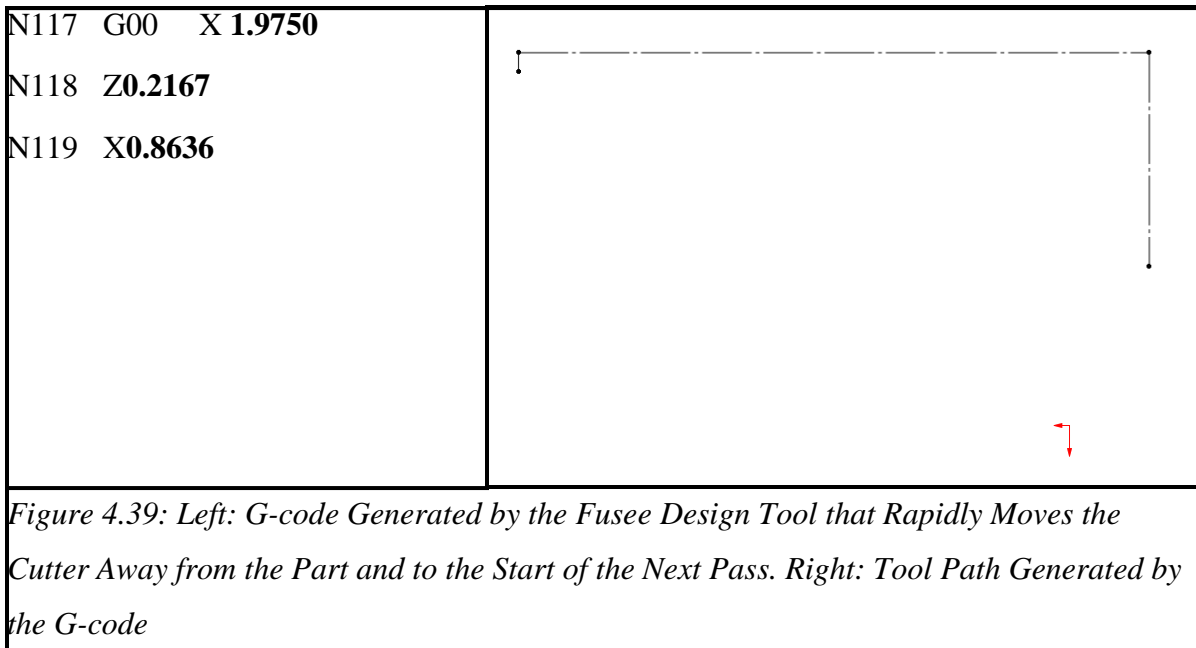
The X and Z coordinates were calculated using the TK Solver output, they will change based on inputs in Excel. This is true for all F, X, and Z feeds and coordinates in this G-code.



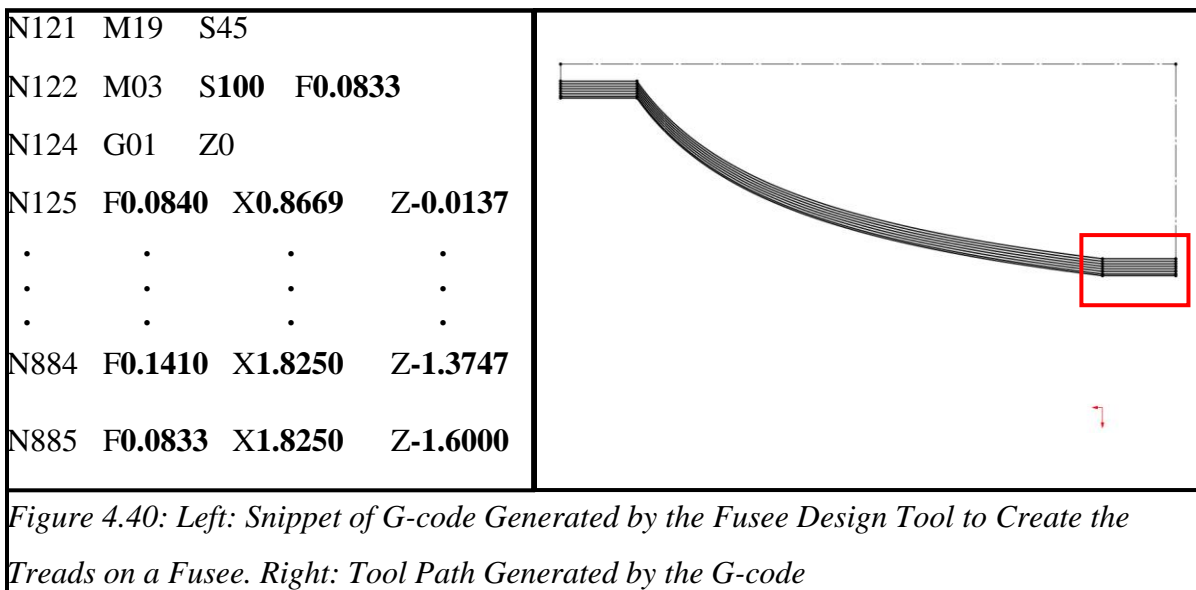
As the cutter moves along the profile of the fusee and the slope of the fusee profile increases the cutter will begin to remove some material. This is illustrated in Figure 4.38. This is an indicator of correct tool offsets. This occurs because tool nose compensation is turned off. This ensures the cutter is following the profile of the fusee, and not adjusting the thread depth based on profile slope.



The cutter then moves rapidly to the safety distance in X, moves to the thread offset position in Z, then moves the X position to the first point of the next pass. Figure 4.39 shows the G-code used to perform these maneuvers and the tool path generated by this G-code.



This cycle is repeated until the part is complete. Each pass moves in a specified depth of cut, until the final pass, which moves in only the user input final pass depth of cut. Figure 4.40 shows snippets of the G-code used to thread the part and the tool path generated by this G-code.



The depth of cut programmed for all the passes except the final pass is determined by the following equation:

$$DOC_{norm} = \frac{d_{wire} - DOC_{final}}{n}, \quad (8)$$

where DOC_{norm} is the depth of cut for all passes except the final pass, d_{wire} is the diameter of the wire, DOC_{final} is the depth of cut for the final pass, and n is the number of passes. Note that changes in the programmed X values between passes should be double the DOC calculated with this equation because the CNC lathe operates on diameter coordinates. Figure 4.41 shows a zoomed-in version of Figure 4.40 and illustrates the between the DOC_{norm} and DOC_{final} , as well as the total thread depth.

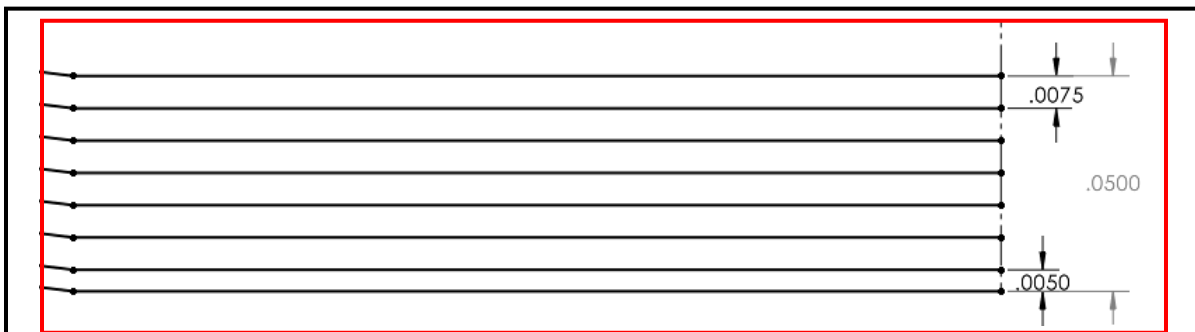


Figure 4.41: Illustration of the Difference Between the DOC_{norm} , DOC_{final} , and the Total Thread Depth

The cutter then moves rapidly to the safety distance in X, moves to the thread offset position in Z, turns off the spindle then rapidly moves off the part such that the leading edge of the cutter will be two inches away from the part in the Z. The machine is stopped, and the percentage sign marks the end of the code. Figure 4.42 shows this G-code and the tool path generated by it.

N887 G00 X 1.9750	
N888 Z 0.2167	
N890 M05 G00 Z 2.0250	
N891 M30	
%	

Figure 4.42: Left: G-code Generated by the Fusee Design Tool that Moves the Cutter Away from the Fusee After the Treads are Manufactured. Right: Tool Path Generated by the G-code

Figure 4.43 shows a Fusee manufactured with the fusee design tool.



Figure 4.43: A Fusee Designed and Manufactured by the Fusee Design Tool

Chapter 5: Discussion and Conclusion

Discussion and Conclusion

Over the past four years, the University of Idaho has seen a positive trend in pass rates for both the CSWPA-SU and the CSWE exams. It is hypothesized that this positive trend in the surfacing exam scores can be attributed to the diversification of homework types and more time to prepare for the exam. A diversification of homework types means there is some tutorial-style homework and some question-answer-style homework. The positive trend in CSWE pass rates can loosely be correlated to the change in homework type i.e., moving away from tutorial-style homework problems to exclusively question-answer-style homework. In total 62 students earned their CSWPA-SU certification, and 30 students earned their CSWE certification during the *Solid Modeling, Simulation, and Manufacturing Capstone* course at the University of Idaho.

The techniques taught in the surfacing and expert sections of the *Solid Modeling, Simulation and Manufacturing Capstone* course were applied to create a fusee design tool. It allows the users to input variables into Microsoft Excel, then using TK Solver, Excel, and SOLIDWORKS the tool automatically generates a SOLIDWORKS model of a fusee and G-code for a Hass CNC lathe to manufacture the threads of a fusee for a specific spiral spring and torque output requirements. During this project, the surfacing and expert techniques proved invaluable. They however were not enough on their own. CAD is not a replacement for fundamental engineering principles, it can however be a good place to start.

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Appendix

The appendix for this thesis can be found at:

S:\Storage-Engineering>Documents>Senior Design>-Course Folders>ME 490>Ian Glasgow
Thesis Appendix

It includes the following sub-section:

Appendix A - SOLIDWORKS Surfacing Workbook

Appendix B - SOLIDWORKS Expert Workbook

Appendix C - SOLIDWORKS Certification Exam Results

Appendix D - Spiral Spring Analysis

Appendix E - Lyre Skeleton Clock CAD and CAM

Appendix F - Fusee Design Tool