2004

Sensitivity analysis of the C-130 sensor deployment system arm using finite element methods

Lawrence Anthony Feragotti
West Virginia University

Follow this and additional works at: <https://researchrepository.wvu.edu/etd>

Recommended Citation

Feragotti, Lawrence Anthony, "Sensitivity analysis of the C-130 sensor deployment system arm using finite element methods" (2004). *Graduate Theses, Dissertations, and Problem Reports*. 1428.
<https://researchrepository.wvu.edu/etd/1428>

This Thesis is protected by copyright and/or related rights. It has been brought to you by the The Research Repository @ WVU with permission from the rights-holder(s). You are free to use this Thesis in any way that is permitted by the copyright and related rights legislation that applies to your use. For other uses you must obtain permission from the rights-holder(s) directly, unless additional rights are indicated by a Creative Commons license in the record and/ or on the work itself. This Thesis has been accepted for inclusion in WVU Graduate Theses, Dissertations, and Problem Reports collection by an authorized administrator of The Research Repository @ WVU. For more information, please contact researchrepository@mail.wvu.edu.

**Sensitivity Analysis of the C-130 Sensor
Deployment System Arm using Finite Element Methods**

Lawrence Anthony Feragotti

A thesis submitted to the College of Engineering and Mineral Resources at
West Virginia University

In partial fulfillment of the requirements for the degree of

Master of Science

in

Aerospace Engineering

James Smith, PhD., Chair
Kenneth Means, PhD.
Victor Mucino, PhD.

Department of Mechanical and Aerospace Engineering

Morgantown, West Virginia
2004

Keywords: Pro/ENGINEER, Pro/MECHANICA, RoRo
Copyright Lawrence Anthony Feragotti, 2004. All rights Reserved

ABSTRACT

Sensitivity Analysis of the C-130 Sensor Deployment System Arm using Finite Element Methods

Lawrence Feragotti

Almost anything designed today can be made better in some aspect. Whether it is made faster, stronger, lighter, or cheaper, there are ways to optimize many of the products used in today's world. This is especially true when dealing with mechanical systems. Engineers compensate for calculated forces by over-designing the mechanisms they create using high safety factors along with extra fasteners and supports. This is why 'version two' of most products is smaller than the first, but retains at least the same performance, if not more. Sensitivity studies are the tests done to ensure maximum performance with minimal weight and cost. These are the studies used to optimize designs and are typically done using Finite Element Methods.

The Finite Element Method may be the most powerful tool in the engineer's toolbox today, and can be used in many different applications. The purpose of this thesis is to utilize this tool to optimize the current design of the roll-on roll-off sensor deployment system support arm for the C-130 Hercules. The Department of Defense (DOD) and the National Guard (NG) will be using these sensor pallet systems in a variety of command and control configurations for counter narco-terrorism applications. The original design for the sensor deployment arm will be drawn using CAD, and then a Finite Element Analysis will be run using Pro/MECHANICA. This will show the stress concentrations and the areas where weight can be saved. The most concerning variable will be the height of the mechanical arm attachment. By decreasing that height, and shortening the mechanical arm, the moments will decrease, and the required torque will be less. To summarize, this study uses a Finite Element Analysis of the current deployment system design of the C-130 Hercules within Pro/ENGINEER and Pro/MECHANICA; then re-designs the deployment system to decrease the weight and the power requirements of the initial layout. The contribution of this study is a lighter and safer deployment system that performs as well if not better than the original.

ACKNOWLEDGEMENTS

I started taking graduate classes in July of 2003, and now I am preparing to graduate with my Master's Degree in May of 2004. This could not have been possible without the guidance of my advisor, Dr. James Smith. He was always confident in me and my abilities, and he constantly pushed me causing me to accomplish so much in such a short time. He truly feels that anything is possible, and he opened my eyes to that very same concept. Along with Dr. Smith, some other administration that I would like to thank includes Dr. Wade Heubsch, Chuck Coleman, and all of the MAE secretaries.

Outside of the University was also a good place for me to find support. I have spent almost my entire collegiate career with the same roommate, Tony Paletta, and even a few years of high school together. Tony has always been supportive of my work and is one of my closest friends. His brother Richard Paletta is also a very close friend, and I am very thankful for the time I spent in college with him. Out of all of the people I have had in my life in the last 6 years, not one of them had the impact that Cortney Collins has had on my life. I want to thank her for making the last 4 years so wonderful.

As for the people not living in Morgantown, I would like to thank my mother, father, and sister. They are always there when I need them. However, the one person I do have to thank for this is my mother, Sharon Feragotti. There was never a time that she did not want to talk to me or see me. She has devoted her life to her children and I am forever grateful for that. Without her constant love and support I am confident that I would not be where I am today.

LIST OF CONTENTS

ABSTRACT.....	ii
ACKNOWLEDGEMENTS.....	iii
LIST OF CONTENTS.....	iv
LIST OF TABLES.....	v
LIST OF FIGURES.....	vi
NOMENCLATURE.....	xii
CHAPTER 1: INTRODUCTION.....	1
CHAPTER 2: LITERATURE REVIEW.....	19
CHAPTER 3: PROCEDURE.....	28
CHAPTER 4: RESULTS.....	32
CHAPTER 5: CONCLUSIONS.....	42
REFERENCES.....	43
APPENDIX A.....	45
APPENDIX B.....	59
APPENDIX C.....	71
APPENDIX D.....	83
APPENDIX E.....	95
APPENDIX F.....	107
APPENDIX G.....	119
APPENDIX H.....	123

LIST OF TABLES

Table 1. Decrease in required torque as a result of the arm attachment height.....	<u>32</u>
Table 2. Sensitivity study results for Arm A @ 13.25 inches.....	<u>33</u>
Table 3. Sensitivity study results for Arm B @ 11 inches.....	<u>34</u>
Table 4. Sensitivity study results for Arm C @ 8.25 inches.....	<u>35</u>
Table 5. Sensitivity study results for Arm D @ 5.5 inches.....	<u>36</u>
Table 6. Sensitivity study results for Arm E @ 2.75 inches.....	<u>37</u>
Table 7. Sensitivity study results for Arm F @ 0 inches.....	<u>38</u>
Table 8. Final stresses in the arm at a height of $2\frac{3}{4}$ inches and a cross-sectional area of $3\times\frac{1}{2}$ inches including required torque from gear reducer.....	<u>40</u>

LIST OF FIGURES

Figure 1. Overall layout showing the four arms at their most extended position.....	3
Figure 2. Overall layout showing the backside of the plate and linear driveshaft.....	4
Figure 3. Sensor pallet which holds the deployment system.....	5
Figure 4. Linear drive rail with bearing used to connect the pallet to the plate.....	6
Figure 5. Underside of plate showing stiffeners and bolt holes for the gear reducer...	7
Figure 6. Top of plate with stiffening channels and bearing spacers attached.....	8
Figure 7. Gear reducer used for rotational deployment of mechanical arms.....	9
Figure 8. Falk Coupling used to connect the 3.25” output shaft to the 2” shaft.....	10
Figure 9. Hub with keyway used to connect the arms to the 2”shaft.....	11
Figure 10. Rotational arm used to deploy the sensor pallet below the bay door.....	12
Figure 11. Stands used to prevent the rotational arm from extending too far back....	13
Figure 12. Top view of the rotational-drive showing all bearings & coupling.....	14
Figure 13. Side view showing the height from the plate to the rotary arm shaft.....	15
Figure 14. Example of a disc brake mesh (not actual used).....	21
Figure 15. Constraints in the x, y, and z directions at each bolt hole	29
Figure 16. Static loads applied at the end of the mechanical arm	31
Figure 17. Rear isometric view of the optimal design.....	40
Figure 18. Von Mises stress distribution throughout 3x0.5 inch rotational arm under critical loading.....	51

APPENDIX A: Arm Sensitivity Study at 13.25 inches above the plate

Figure 19. Von Mises stress distribution throughout 4x0.75 inch rotational arm under critical loading.....	47
Figure 20. Convergence plot showing that the FEA did converge on its fifth polynomial pass with 5% error or less.....	48

Figure 21. Von Mises stress distribution throughout 3x0.75 inch rotational arm under critical loading.....	<u>49</u>
Figure 22. Convergence plot showing that the FEA did converge on its fifth polynomial pass with 5% error or less.....	<u>50</u>
Figure 23. Von Mises stress distribution throughout 2x0.75 inch rotational arm under critical loading.....	<u>51</u>
Figure 24. Convergence plot showing that the FEA did converge on its fifth polynomial pass with 5% error or less.....	<u>52</u>
Figure 25. Von Mises stress distribution throughout 1x0.75 inch rotational arm under critical loading.....	<u>53</u>
Figure 26. Convergence plot showing that the FEA did converge on its seventh polynomial pass with 5% error or less.....	<u>54</u>
Figure 27. Von Mises stress distribution throughout 3x0.5 inch rotational arm under critical loading.....	<u>55</u>
Figure 28. Convergence plot showing that the FEA did converge on its fifth polynomial pass with 5% error or less.....	<u>56</u>
Figure 29. Von Mises stress distribution throughout 2x0.5 inch rotational arm under critical loading.....	<u>57</u>
Figure 30. Convergence plot showing that the FEA did converge on its fourth polynomial pass with 5% error or less.....	<u>58</u>
APPENDIX B: Arm Sensitivity Study at 11 inches above the plate	
Figure 31. Von Mises stress distribution throughout 4x0.75 inch rotational arm under critical loading.....	<u>61</u>
Figure 32. Convergence plot showing that the FEA did converge on its fifth polynomial pass with 5% error or less.....	<u>62</u>
Figure 33. Von Mises stress distribution throughout 3x0.75 inch rotational arm under critical loading.....	<u>63</u>
Figure 34. Convergence plot showing that the FEA did converge on its fifth polynomial pass with 5% error or less.....	<u>64</u>
Figure 35. Von Mises stress distribution throughout 2x0.75 inch rotational arm under critical loading.....	<u>65</u>

Figure 36. Convergence plot showing that the FEA did converge on its fifth polynomial pass with 5% error or less.....	<u>66</u>
Figure 37. Von Mises stress distribution throughout 3x0.5 inch rotational arm under critical loading.....	<u>67</u>
Figure 38. Convergence plot showing that the FEA did converge on its fifth polynomial pass with 5% error or less.....	<u>68</u>
Figure 39. Von Mises stress distribution throughout 2x0.5 inch rotational arm under critical loading.....	<u>69</u>
Figure 40. Convergence plot showing that the FEA did converge on its fourth polynomial pass with 5% error or less.....	<u>70</u>
APPENDIX C: Arm Sensitivity Study at 8.25 inches above the plate	
Figure 41. Von Mises stress distribution throughout 4x0.75 inch rotational arm under critical loading.....	<u>73</u>
Figure 42. Convergence plot showing that the FEA did converge on its fifth polynomial pass with 5% error or less.....	<u>74</u>
Figure 43. Von Mises stress distribution throughout 3x0.75 inch rotational arm under critical loading.....	<u>75</u>
Figure 44. Convergence plot showing that the FEA did converge on its fifth polynomial pass with 5% error or less.....	<u>76</u>
Figure 45. Von Mises stress distribution throughout 2x0.75 inch rotational arm under critical loading.....	<u>77</u>
Figure 46. Convergence plot showing that the FEA did converge on its fifth polynomial pass with 5% error or less.....	<u>78</u>
Figure 47. Von Mises stress distribution throughout 3x0.5 inch rotational arm under critical loading.....	<u>79</u>
Figure 48. Convergence plot showing that the FEA did converge on its fifth polynomial pass with 5% error or less.....	<u>80</u>
Figure 49. Von Mises stress distribution throughout 2x0.5 inch rotational arm under critical loading.....	<u>81</u>

Figure 50. Convergence plot showing that the FEA did converge on its fourth polynomial pass with 5% error or less.....	<u>82</u>
--	-----------

APPENDIX D: Arm Sensitivity Study at 5.5 inches above the plate

Figure 51. Von Mises stress distribution throughout 4x0.75 inch rotational arm under critical loading.....	<u>85</u>
--	-----------

Figure 52. Convergence plot showing that the FEA did converge on its fifth polynomial pass with 5% error or less.....	<u>86</u>
---	-----------

Figure 53. Von Mises stress distribution throughout 3x0.75 inch rotational arm under critical loading.....	<u>87</u>
--	-----------

Figure 54. Convergence plot showing that the FEA did converge on its fifth polynomial pass with 5% error or less.....	<u>88</u>
---	-----------

Figure 55. Von Mises stress distribution throughout 2x0.75 inch rotational arm under critical loading.....	<u>89</u>
--	-----------

Figure 56. Convergence plot showing that the FEA did converge on its fifth polynomial pass with 5% error or less.....	<u>90</u>
---	-----------

Figure 57. Von Mises stress distribution throughout 3x0.5 inch rotational arm under critical loading.....	<u>91</u>
---	-----------

Figure 58. Convergence plot showing that the FEA did converge on its fifth polynomial pass with 5% error or less.....	<u>92</u>
---	-----------

Figure 59. Von Mises stress distribution throughout 2x0.5 inch rotational arm under critical loading.....	<u>93</u>
---	-----------

Figure 60. Convergence plot showing that the FEA did converge on its fourth polynomial pass with 5% error or less.....	<u>94</u>
--	-----------

APPENDIX E: Arm Sensitivity Study at 2.75 inches above the plate

Figure 61. Von Mises stress distribution throughout 4x0.75 inch rotational arm under critical loading.....	<u>97</u>
--	-----------

Figure 62. Convergence plot showing that the FEA did converge on its fifth polynomial pass with 5% error or less.....	<u>98</u>
---	-----------

Figure 63. Von Mises stress distribution throughout 3x0.75 inch rotational arm under critical loading.....	<u>99</u>
--	-----------

Figure 64. Convergence plot showing that the FEA did converge on its fifth polynomial pass with 5% error or less.....	<u>100</u>
Figure 65. Von Mises stress distribution throughout 2x0.75 inch rotational arm under critical loading.....	<u>101</u>
Figure 66. Convergence plot showing that the FEA did converge on its fifth polynomial pass with 5% error or less.....	<u>102</u>
Figure 67. Von Mises stress distribution throughout 3x0.5 inch rotational arm under critical loading.....	<u>103</u>
Figure 68. Convergence plot showing that the FEA did converge on its fifth polynomial pass with 5% error or less.....	<u>104</u>
Figure 69. Von Mises stress distribution throughout 2x0.5 inch rotational arm under critical loading.....	<u>105</u>
Figure 70. Convergence plot showing that the FEA did converge on its fourth polynomial pass with 5% error or less.....	<u>106</u>
 APPENDIX F: Arm Sensitivity Study at 0 inches above the plate	
Figure 71. Von Mises stress distribution throughout 4x0.75 inch rotational arm under critical loading.....	<u>109</u>
Figure 72. Convergence plot showing that the FEA did converge on its fifth polynomial pass with 5% error or less.....	<u>110</u>
Figure 73. Von Mises stress distribution throughout 3x0.75 inch rotational arm under critical loading.....	<u>111</u>
Figure 74. Convergence plot showing that the FEA did converge on its fifth polynomial pass with 5% error or less.....	<u>112</u>
Figure 75. Von Mises stress distribution throughout 2x0.75 inch rotational arm under critical loading.....	<u>113</u>
Figure 76. Convergence plot showing that the FEA did converge on its fifth polynomial pass with 5% error or less.....	<u>114</u>
Figure 77. Von Mises stress distribution throughout 3x0.5 inch rotational arm under critical loading.....	<u>115</u>
Figure 78. Convergence plot showing that the FEA did converge on its fifth polynomial pass with 5% error or less.....	<u>116</u>

Figure 79. Von Mises stress distribution throughout 2x0.5 inch rotational arm under critical loading.....	<u>117</u>
Figure 80. Convergence plot showing that the FEA did converge on its fourth polynomial pass with 5% error or less.....	<u>118</u>
Figure 81. Fatigue log life fringe plot showing that the arm can withstand 10^6 cycles	<u>120</u>
Figure 82. Fatigue confidence of life fringe plot showing adequate confidence.....	<u>121</u>
Figure 83. Goodman diagram for optimized arm with cross-sectional area of $3 \times \frac{1}{2}$ inches.....	<u>122</u>
Figure 84. Simplified arm with constraints and loading	<u>123</u>
Figure 85. Von Mises fringe plot of the simplified arm geometry	<u>124</u>

NOMENCLATURE

ADINA	Automatic Dynamic Incremental Nonlinear Analysis
CAD	Computer Aided Design
D	Hole Diameter
DOD	Department of Defense
FEA	Finite Element Analysis
FEM	Finite Element Method
fos	Factor of Safety
g	Gravity
H	Height from Plate to Rotary Arm
in	Inches
ksi	Kilo Pounds per Square Inch
lbs	Pounds
NASA	National Aeronautics and Space Administration
NG	National Guard
P-Loop	Polynomial Loop
PSI	Pounds per Square Inch
RoRo	Roll-on Roll-off
δy	Yield Strength
WCS	World Coordinate System

Chapter 1: INTRODUCTION

1.1 Sensitivity Study Background

Sensitivity studies are often used to find the overall effect on a system by varying one or more design parameters, such as the dimensions in a mechanical system. The purpose of such a study is that it can be used to optimize the system being considered. There are many different applications for sensitivity studies in the world today, and they are used to make products better, cheaper, and safer. Sensitivity studies have been used on linear programming, cross country growth regressions, mechanical systems, electrical circuitry, and many other areas of study. Each of these applications has different design parameters that can be changed to affect the final outcome, and each of them has different tests that are run after the parameter has been modified.

1.2 Benefits of Sensitivity Studies

In large-scale production, cost is directly proportional to weight, size, and performance of the final product. For example, in the aircraft industry it is much more expensive to use composite materials to decrease the weight of an aircraft. However, the performance of the aircraft increases as the weight decreases, therefore the money is spent to increase the performance of the aircraft. This shows that by changing the material used, parameters such as cost and weight are affected. Another benefit to sensitivity studies is shown in the automotive industry. By using these studies to remove the areas of vehicles which are over-designed for safety, the automotive industry can save a great deal of money when manufacturing vehicles. The goal of these studies is to find the trade-off point where all minimum requirements are met, while the safety of the product has not been compromised. When this trade-off is found the cost and weight of

the final product will be optimized, thus saving the manufacturer time and money while increasing performance.

1.3 Purpose of this Sensitivity Study

The current work involves the design of a sensor pallet deployment system deployment arm for a C-130 Hercules. This sensor pallet is used to deploy a sensor pod that contains various types of sensor devices used by the National Guard and the Department of Defense in the war on drugs. The basic idea is to linearly extend a plate containing a rotary motion arm out of the back of the C-130 during flight. At the end of the rotating arm is a sensor pod which can be used to hold various cameras and sensors. The initial layout was over-designed to compensate for the aerodynamic drag on the pod, safety, and for the forces needed to counter the weight of the system during deployment. This “over-design” has raised questions as to the weight constraints on the ramp of the aircraft, and with the large size of the mechanical arms it will be difficult to seal the door of the C-130 when flying at altitudes above 10,000 feet. This becomes a problem for the crew as they will not be able to breath without an oxygen mask at these higher altitudes. While the initial design has been decided, and the prototype is being built, this study will look at the design of the deployment arm system from the linear motion to the rotary arm that extends the sensor pod out of the aircraft, to reduce the weight and power requirements of the mechanical system.

Within this study all of the components of the mechanical system were drawn in a CAD program call Pro/ENGINEER. These components were chosen by the team of engineers currently designing the sensor pallet, and the first configuration to be modeled is the currently existing design. Using an assembler within Pro/ENGINEER, the exact

design of the current deployment system can be created virtually. Upon completion of the model a Finite Element Analysis (FEA), also know as the Finite Element Method (FEM) could be computed using a Pro/MECHANICA package found in the Pro/ENGINEER software. This analysis shows the stresses, strains, and displacements within the system and the critical points can now be addressed. Upon completion of this FEA, different arrangements of the components and different geometries can be simulated to complete the parametric study and find the best arrangement for the sensor pallet system. The following figures are 3-dimensional renderings of the current deployment system (Figures 1-13).

Figure 1: Overall layout showing the four arms at their most extended position

Figure 1 shows an isometric view of the front of the deployment system with the arms at their most critical loading position (this is where the largest moment will occur). This view includes the entire mechanical system from the pallet up to the arms, but not including the sensor pod. The pod will not be optimized in this study.

Figure 2: Overall layout showing the backside of the plate and linear driveshaft

Figure 2 shows a side view of the system, again with the arms extended to the critical loading position. It is also easy to see the arm supports on the back of the plate for stowing, and the linear driveshaft extending from under the plate. The linear driveshaft is what moves the entire system outward before the arms rotate around the door. Also note the spacers and the green bearings between each set of arms and on the door.

outside of each set. These are to support the rotary driveshaft and the loads from the sensor pod

Figure 3: Sensor pallet which holds the deployment system

Figure 3 shows an isometric view of the aluminum pallet. This is a large piece of metal frame and weighs over 1100 pounds. The only modification made to the pallet was a $\frac{3}{4}$ inch plate bolted to the top. This was to ensure that the weight of the sensor deployment system would not damage the pallet. Everything within the system will be mounted on this plate. The channels shown in the picture are used by fork lifts when these pallets are being transported and positioned.

Figure 4: Linear drive rail with bearing used to connect the pallet to the plate

Figure 4 shows the key to the linear movement of the plate. Eight of these rails will be bolted to the pallet. They will then be connected to the linear plate using one bearing on each of the rails. It takes eight bolts to secure one rail to the pallet, and four more to secure the bearing to the linear plate. After it has been completely assembled the eight rails and their bearings will account for 96 bolts. All of the bolts used are SAE grade 5.

Figure 5: Underside of plate showing stiffeners and bolt holes for the gear reducer

Figure 5 shows the linear plate with all the welded stiffeners. Each of the channels shown is welded to the underside of the plate to prevent buckling. The bolt pattern for the gear reducer can also be seen, and there is a slot in the channels for another stiffener plate that will share the bolt holes of the reducer. More stiffeners will also be present on top of the linear plate.

Figure 6: Top of plate with stiffening channels and bearing spacers attached

Figure 6 shows the top of the linear plate along with stiffening channels and the bearing mounts. The bearing mounts are more towards the front and sit higher than the channels, as the channels run the length of the plate. Not only can the bolt pattern for the reducer be seen here, but the bolt patterns for the bearings and the channels are shown. This gives the reader an idea of the amount of fasteners that are used in creating just one system.

Figure 7: Gear reducer used for rotational deployment of mechanical arms

Figure 7 shows the Textron gear reducer used for the rotary motion. This reducer has a 600:1 reduction, and contains two input and output shafts. Each of the two output shafts are used as there will be a rotary driveshaft on each side with two arms. The output shafts are 3.25 inches in diameter, and are reduced by a coupling before the 2 inch rotary shafts are attached. As for the input shafts, one will be powered by an electric motor, and the other will be used manually in the event of a motor or power failure. This system with the motor weighs over 600 pounds, and is a primary concern when reducing weight. Textron was selected because of their innovative technologies, such as Conex Helicoidal and Double Enveloping Geometries, and their cooperativeness in helping us in this design process [14].

Figure 8: Falk Coupling used to connect the 3.25” output shaft to the 2” shaft

Figure 8 shows the coupling used to connect the 3.25 inch output shaft, to the 2 inch shaft of the rotary system. This coupling is designed by Falk Corp, and through the application of precision technology used in the aerospace industry, the Falk Corporation has developed its unique line of Steelflex® grid couplings, a coupling that provides a 10 percent increase in some instances [14]. In addition, Steelflex combines the high-load capacities of a gear coupling with the flexibility of an elastomer design which protects expensive driving and driven equipment from misalignment, vibration, and shock and thrust loads [15].

Figure 9: Hub with keyway used to connect the arms to the 2”shaft

Figure 9 shows the hub used to attach the rotary arms to the 2 inch shaft. These are machined in house and are made entirely of stainless steel. Each of these has a six bolt pattern that matches the end of the arm and a keyway to prevent it from spinning on the shaft.

Figure 10: Rotational arm used to deploy the sensor pod below the cargo door

Figure 10 shows the rotation arm used to deploy the pod below the cargo door. This arm is made of aluminum and has a cross sectional area of $4 \times \frac{3}{4}$ inches. This is a primary concern in the sensitivity study because that four inch distance will be hard to seal at altitudes above 10,000 feet. Figure 10 also shows the six bolt pattern and the hole for the rotary driveshaft. Also note that all of the bends in the arms are filleted to reduce stress concentrations. This is because the distribution of stresses in the critical cross sections is dependent only upon the geometry of the member [17].

Figure 11: Stands used to prevent the rotational arm from extending too far back

Figure 11 shows the stands that are mounted on the back of the linear plate. These stands were also shown in Figures 1 and 2, and are used to prevent the arms from rotating too far back when they are stowed. The primary concern is that the arms could rotate too far back and interfere with some of the other components on the plate, including the rotary motor. These stands were machined in house and are made completely of aluminum.

Figure 12: Top view of the rotational-drive showing all bearings & coupling

Figure 12 shows a top view of the rotary shaft including all components. The reducer can be seen in blue, and the Falk Couplings are directly attached to the reducer. The bearings are shown in green, and the hubs can be seen attached to the rotary arms. These are the components responsible for rotating the sensor pod out of the cargo door, and safely back in when the mission is complete.

Figure 13: Side view showing the height from the plate to the rotary arm shaft

This sensitivity study will key on one particular variable in achieving both goals of reducing weight and reducing the required power. The variable in question is the distance (H) from the linear drive plate to the rotary arm shaft, as is shown in Figure 13. It is anticipated that as this distance is decreased the goal of this study will be met.

1.4 Introduction to Finite Element Analysis (FEA)

The FE method is based on being able to find a field in what is being analyzed. For example, in a stress analysis it is known as the displacement field, while in a thermal analysis it is called the heat flux, or in a fluid flow it would be the velocity potential function, and so on. This method was developed using physical insight rather than abstract methods, and was first applied to problems of stress analysis, but recently has been used in many other applications [2]. The fundamental concept behind the Finite

Element Method is that it breaks down the object being analyzed and looks at several smaller elements of the original piece. After it looks at the behavior of these smaller elements the FEM reconnects them at locations known as nodes to see the effect on the entire shape. These nodes are like the glue that hold the entire shape together. When the system has been broken into elements, a set of simultaneous algebraic equations are formed. These are known as the interpolating polynomial equations. In stress analysis, these equations are equilibrium equations of the nodes, and there may be several hundred or thousand such equations. This is why computers must be implemented to solve these sets of equations.

K. J. Bathe identified two possible and different objectives for studying Finite Element Analysis and methods: to learn the proper use of the method to solve complex problems (the practitioner's goal), and to understand the methods themselves in depth so as to pursue further development of the theory (the researcher's goal) [1]. It is increasingly important to understand this statement when working with software packages so that the engineer does not use the software incorrectly. Finite Element Analysis is quite possibly the most important tool added to the mechanical design engineer's toolbox in the last 20 years, and can be used to obtain more accurate design computations in complex situations. This allows for improvements in both the design procedure and products. However users of any FEA software must also be aware of the GIGO ("Garbage In = Garbage Out") principle, in that users can be easily misled into blind acceptance of answers produced by the programs. Users of FEA software should have a good understanding of the Finite Element Method before performing simulations within the software.

The software package chosen to run the finite element analysis for this study is Pro/MECHANICA, also referred to as Pro/M. What sets Pro/M apart from other FEA programs is that it uses the convergence of P-Elements instead of the classical H-Elements. The difference between these is that the H stands for the size of the element in the mesh, and the P stands for the order of polynomial in the interpolating equation.

When using the H-element approach, the FEA program uses low order polynomials as the governing equations mentioned earlier, and for accurate results this requires that there be many elements. For example, strain is obtained by taking the derivatives of the displacement field and the stress is computed from the material strain. For a first order interpolating polynomial within the element, this means that the strain and stress components within the element are constant everywhere. This also means that if the elements are not small enough, errors will occur within the FEA. Also, areas with larger gradients will be very inaccurate because of the constant values through the elements. The means for solving this in the past was to use finer meshes so that the elements would be smaller. However, the finer the mesh the harder it was for the computer to calculate the FEA. This is due to the increased amount of equations from the increased amount of elements in the finer mesh. By using the method of mesh refinement, a convergence test could be done, but if the computing power is lacking the user will never truly know if the test converged.

The alternative to the H-element method is the P-element method, and this uses higher order interpolating polynomial equations. Therefore, instead of continually refining the mesh, the degree of the interpolating polynomial is increased until convergence is reached. This allows for the mesh to stay the same for each test, and uses

far less interpolating equations. Pro/M allows for the order of the interpolating polynomial to reach as high as a ninth order. In most FEA software the mesh refinement method does work to reduce the errors. However, the amount of mesh refinements it would take to match the accuracy of the higher order P-element would involve an incredible amount of elements. This is why the P-element method is much easier and faster to use.

1.5 Summary

To summarize, what has been done in this study is the Finite Element Analysis of the current deployment system arm design of the C-130 Hercules within Pro/ENGINEER and Pro/MECHANICA; then using this FEA to re-design the deployment system to decrease the weight and the power requirements of the initial layout. The contribution of this study is a lighter and safer deployment system that performs as well if not better than the original.

Chapter 2: LITERATURE REVIEW

2.1 History of the Finite Element Method

In 1943, a mathematician by the name of Courant used triangular elements to solve a torsion problem [2]. However his work was not noticed by engineers of his time because his piecewise polynomial solution was impractical due to the lack of computer power in the early 1940's. Although Courant's methods were non-applicable when he discovered them, they would prove to be very useful to engineers down the road. It became apparent in the following years that this method described by Courant could be used in the aircraft industry to analyze delta wings. The problem with delta wings is that they are too short for the traditional beam theory to be reliable. The use of this method to break down the wings to small elements led to the coining of the term "finite element", and the finite element method was born. The name "finite element" was coined in a paper by Clough, in which the technique was presented for plane stress analysis [3]. Within the next five years the finite element method had become recognized as a valid method, and its application expanded to heat transfer, groundwater flow, magnetic fields, and other areas of engineering. The early 1970's showed the availability of software packages and in the 1980's the software was available on microcomputers. To date there is over 40,000 papers and books about the FE method and its applications [4].

2.2 FEM within the Automotive Industry

J. A. Tomas of the Royal Melbourne Institute of Technology in Australia was one of the earliest users of the Finite Element Method in the automotive industry. Using the FE Method, Tomas successfully re-designed a disc brake, and a gas turbine wheel. The objective of his study with the disc brake was to design a brake having a zero

deformation angle in the disc and minimum stress. His method to do this was that of M.J. Box. Tomas chose the Box method because of its ease to understand, it does not require the calculation of gradients, it accommodates different types of constraints, and it finds the global optimum more frequently than most other methods [5]. Tomas also used CDC-MARC and ADINA to compute the finite element analysis. CDC-MARC being French FEA software, while ADINA stands for Automatic Dynamic Incremental Nonlinear Analysis, and was founded in 1986 by Dr. K. J. Bathe and associates. The exclusive mission of the company is the development of the ADINA System for the analysis of solids, structures, fluids and fluid flow with structural interactions [18]. It should also be noted that some changes were necessary in the original program code in order to increase the efficiency of the procedure.

The first step to re-designing the brake was to create the mesh of the disc brake. Tomas used a mesh containing 114 axisymmetrical rectangular elements, and to simplify the computations, some of the elements in the mesh were restricted. This meant that only the elements in the critical areas were analyzed for deformation, and the non-critical areas were constrained. The figure below shows where the critical elements (grey) in relevance to the non-critical elements (white) [5].

Figure 14: Example of a disc brake mesh (not actually used)

The optimum design was calculated for the case of braking from 87 mph to stop during a six second period. Six time steps were sufficient for reliable results, and a significant reduction in the size of the disc brake was noted. This experiment proved to be a success, and lead to more extensive use of the Finite Element Method in the automotive industry.

Another example within the automotive industry was a Finite Element Analysis on steel and forged-aluminum medium- and heavy-duty truck wheels by a Kentucky-based Accuride Corp. They used the Finite Element Method to prove that there was a significant amount of flexibility in the weight of the wheels, and this led to lighter better wheels for the heavy-duty trucks. “There were four wheels involved, and we were able to cut weight between 4 and 14%,” Simms says. “The lighter the wheels, the more payload they can carry” [6].

2.3 FEM within the Appliance Industry

The automotive industry is not the only workforce of engineers using the Finite Element Method to solve problems and optimize their products. The following is an example of an FEA in the appliance industry conducted by engineers at Whirl-pool. “Our big engineering issues are reducing costs--especially materials costs--and reducing design time,” says Lead Engineer Curtis Niemier. Reducing the number of physical prototypes is critical for the latter, he says. Software enables cutting months off product development time by making it possible for engineers to design in new features and test them out on a computer [7]. This is true in any industry today, and Whirl-pool currently uses Pro/ENGINEER for their computer aided design while they conduct their FEA in ANSYS. In a recent project, they have used FEA studies to decrease the thickness of the walls of a new refrigerator and maintained the efficiency of the original product. They also found in their FEA that the wall thickness compromised the stiffness so they compensated for this using various brackets and other fasteners. “Without the software, we would have had to build physical prototypes to be sure that strategy would work,” Niemier says [7]. Engineers have also learned alternative ways to attach the front rail across the bottom of the unit using software iterations. Maytag has also used FEA studies to optimize their refrigerators. Ron Anderson, manager of advanced technology for Maytag quoted, “We have run thousands of analyses just for optimizing refrigerator cabinets”. “We share solid model files with vendors who make the tools for parts,” says Anderson [7]. By not having to build the prototypes for these models the respective companies save time and money, and in the world of industry time is money.

One appliance that would not appear to need a Finite Element Analysis to improve is the hair dryer. However, Johnson Electric investigated why many hair dryers

tend to emit a thin, high-pitched noise when turned on and off, and used the Finite Element Method to solve the problem. After inspection, they determined that it was the carbon brush that was causing the noise, and ran a Finite Element Analysis to investigate the mode shapes and frequencies of the brush leaf. The software they used to create the mesh was Solidworks, and is not much different from Pro/ENGINEER. The analysis was then conducted and it showed that the maximum amplitude of the vibration occurs mainly on the ends of the flanges of the brush leaf. The new design developed by the engineers involved reinforcing certain parts of the brush leaf, and changing bend angles and the stiffness of the material [8]. The results of the new design completely eliminated the high-pitched noise, and reduced the running noise by about 4 dB at low speeds, and 3 dB at high speeds.

2.4 FEM within Bio-medics

The goal of one study in bio-medics was to create an explicit finite element model that would offer insight to the kinematics and stresses generated by a total knee replacement. This model would be subjected to the average usage of the knee and then analyzed to see how it would hold up. The simulation measured the effects of vertical loads with medial offsets of up to 15mm, as well as unicondylar loading (also known as edge loading). “Only minor variations were observed in the kinematics with a medial offset of the vertical load of as much as 15 mm (representing a medial:lateral loading ratio of 86:14), although the polyethylene stresses did increase by approximately 3 MPa throughout the stance phase of gait,” [9]. However, “There was a significant change in the kinematics and stresses when unicondylar loading occurred (95:5 medial:lateral loading ratio). Even for the unicondylar load case, contact always was maintained within

the lateral compartment,” [9]. The model was also used to predict regions of plastic deformation that was found in previous patients with extreme wear. This study proved that the explicit finite element model offered a considerable amount of insight into the kinematics and stresses generated by total knee replacement during different and varied loading conditions that occur during normal usage. The results of this FEA proved to be useful and the study was a success.

Another area of the body where the Finite Element Method has proven to be useful is in the hip. Finite Element Analysis has showed areas of wear in artificial hips, and the duration it takes for this wear to appear. Using a pin-on-disk experiment for wear, researchers at the National Chung Hsing University in Taiwan have used computer simulation to model wear behavior appearing in total hip prostheses. The researchers stated, “Through the successful verification of wear depth and volume loss of the pin-on-disk plate as well as the artificial hip joint, the current algorithms provide significant agreement with experiments, clinical measurements, and numerical calculations, and are shown to be both valid and feasible.” [10]. Their experiments have led to better alternatives to hip replacement designs and a better understanding of wear behavior in hip replacements.

2.5 FEM within the Aerospace Industry

Some of the most precise Finite Element Analysis can be found in the aerospace industry. The FEM is used to determine reliable structures as they will be subject to aeronautical loading during flight, and using computational fluid dynamics to determine flow characteristics from one airfoil to another. With the effects of failure in aircraft being so catastrophic, engineers must be sure that their calculations are correct. For

example, flight 587 into Belle Harbor, New York, is believed to have crashed because of a simple lug in the rear of the plane. The lug was subjected to a load condition derived from the accident aircraft's flight data recorder and subsequent structural Finite Element Analysis. For certification, design ultimate load must be at least 1.5 times the highest load expected to be encountered in service, and the FEA showed that the loads exceeded the ultimate allowable. The test failure appeared consistent with failure load analyses by both Airbus and NASA, says the NTSB [11]. Had the FEA been run earlier and this mistake was caught, 260 lives would have been saved.

A second example of an aerospace application in Finite Element Analysis was present at the United States Air Force Research Laboratory's Propulsion Directorate at Edwards Air Force Base, California where engineers are developing a low-cost method for transferring satellites from lower-Earth orbits to a higher energy, geosynchronous orbit. In order to obtain the energy to ignite the thrusters and send the satellites to the higher energy orbit, scientists created inflatable solar concentrators made almost entirely of a new polyimide material. These solar concentrators are pre-molded and very light weight, but must be designed so that their 9 x 13-foot reflectors achieve a precise surface slope when inflated in orbit. In order to do this without running multiple experiments with expensive prototypes, a Finite Element Analysis was done to test the solar concentrators on a computer. The software package chosen to run the analysis was ALGOR, computer software created by ALGOR, Inc, which provides services for mechanical and civil engineers in industries such as automotive, aerospace, medical, consumer products, military, electric power, petroleum, large structures, micro electro mechanical systems (MEMS) and more [19].

SRS Technologies conducted most of the work on the solar concentrator models. SRS technologies, since its founding in 1970, has become a dynamic engineering company providing diversified Information Technology Services and Products to Government and commercial customers on a worldwide basis. SRS' mission is: a) to provide the very best services in System Engineering, System Integration and Information Technologies; b) to foster continuous innovation and performance improvement; c) to achieve annual growth of 20%; and d) to provide maximum opportunity, growth and satisfaction for its stockholders and employees [20]. Before using ALGOR software to optimize the design of the solar concentrators, SRS first performed a physical test on a solar concentrator prototype. The prototype was created based on specifications provided by SRS optical engineers that dictated the shape required to collect the optimal amount of solar energy for propulsion. By applying internal pressures and static loads to the prototype and comparing them to the ALGOR software, SRS engineers verified the accuracy of the software. "We learned the value of verifying the software's accuracy first with a physical test," said Jim Moore, program manager at SRS' Aerospace Directorate [12]. The results between the experimental tests and software correlated 94 %. Jim Moore again stated "It gave us the confidence that ALGOR's software analysis results were indicative of real-world results. It also enabled us to save thousands of dollars on tooling for solar concentrator prototypes that would have been necessary to optimize the design and many months building and testing those prototypes." [12]. Because of the Finite Element Analysis run within ALGOR, this project was a success. The software package solved for the required pressures to deploy the solar concentrators at the required angle so that they would collect enough sunlight to

ignite the propellant. The introduction of these highly accurate solar concentrators to the aerospace industry will drastically reduce the cost of transferring satellites to a geosynchronous orbit while using a clean, abundant and safe power source - the Sun. [12]

2.6 FEM within Modern Sports

Solo Enterprises is an aerospace subcontractor to Boeing, but they also have a sub-division called Solo Golf. This division of Solo has taken their expertise of the aerospace industry and used it to create a series of golf clubs. By the use of computer aided design and the Finite Element Method Solo has reduced development time 75%, while cutting design costs up to 60%. “We designed a club with CAD for the first time in the late 1980s, and found that the distribution of weight and mass in the right places was the key to performance. With that knowledge, we designed a club that looked good to golfers, and made it perform better.” [13]. As the years went on, Solo realized that when dealing with golf clubs, they did not need the accuracy that they were accustomed to in the aerospace industry. With this in mind, and knowing that the golfer wanted the club to look good, they used a simple FEA with a program called CATIA for their design. Ed Mugica, vice president and designer for Solo Golf states "We can use fairly low-level FEA to look at mass properties and fatigue factors, wall thickness, and distribution of mass and weight. CATIA has an analysis menu with a mass properties function and it tells us the overall mass, volume, and moments of inertia. You want to distribute weight as far away from the center of gravity as possible, so that the club won't twist and will be more forgiving," [13].

Chapter 3: PROCEDURE

3.1 Creating the Virtual Environment

The first step in this study was to create the current configuration of the sensor pallet in a virtual environment within Pro/ENGINEER. This allows for the weight of the entire system to be calculated within seconds by simply applying densities to the components, and it also enables the user to calculate distances from any two points within the deployment system. With these two tools the study will be able to prove that weight has been deducted by changes in geometry, and moments will be easily calculated using dimensions from the program when geometries are changed or moved. Upon completion of the geometry, the Finite Element Analysis can be started.

3.2 Preparing the Finite Element Analysis

When preparing a finite element analysis, the first step is to apply constraints to the object. In this case the constraints will be applied to the inner surfaces of the six bolt-hole pattern that connects to the hub on the rotary driveshaft. This can be done because the shear loading for these bolts is well over the required amount before failure will occur in this application [15]. This will also show a more accurate analysis because the stress distribution around the bolt holes will be accounted for. This then leaves the arms as the main concern for failure. These constraints are shown in Figure 15 below.

Figure 15: Constraints in the x, y, and z directions at each bolt hole

Once the constraints are fixed, applying the loading is the next step. There will be two different forms of loading in this analysis. The first will be the obvious weights of the pod, sensors, and the arms themselves, and the second will be the aerodynamic forces that the system will see during deployment. The weight of the arm will be accounted for by applying gravity to the FEA. The gravity vector can be seen in Figure 15. For this study a static analysis will be done to show the critical stresses at the worst case scenario. These occur when the arm has deployed to an angle of 162 degrees from the plate. These weights were included with the total pod weight of 900 pounds to come up with the loading at the end of the moment arm. To simplify the computing work, the maximum forces calculated were divided by four, and applied to one arm. This is possible because there are four arms and it is assumed that they will each receive an equal distribution of the loading. This process will be done at intervals less than three inches until the zero

mark is reached. The loading applied at the end of the arm can be seen in Figure 16 below.

Figure 16: Static loads applied at the end of the mechanical arm

3.3 Conducting the Finite Element Analysis

When conducting the Finite Element Analysis, a multi-pass adaptive check was used with a maximum polynomial order of 9. Pro/MECHANICA uses high order polynomial equations instead of mesh refinement as mentioned in Chapter 1. By using the multi-pass adaptive check the program will continually increase the polynomial order and assess the error from each step to the next. Typically it does not take more than 5 passes to reach convergence, and if it does take all nine steps it is recommended that the mesh be manually refined so that it will converge earlier. This is because it may not have

converged even though it has stopped at the ninth order. When the error is below a certain value the program will have converged and the final answer is then given, but if it makes it to the ninth order it will stop regardless of convergence. This was not an issue for the study. For this study the convergence was set to 5% of the local displacement, local strain energy. This means that the test will reach convergence when the local displacement and local strain energy errors between p-levels is less than or equal to 5%. Once these parameters are set the test can be run and the results can be plotted as graphs or fringe plots in the results section of Pro/MECHANICA. The tests will be run on different cross-sectional areas of the arm. These dimensions are shown in Figure 10 on page 12.

Chapter 4: RESULTS

4.1 Initial Calculations

Before running any sensitivity studies, each arm position had to be created. Upon creating each new arm, a new moment arm was produced. This was due to the arm becoming shorter in length each time the height of the arm was lowered. Table 1 shows the different arm configurations and the required torque due to the new moment arm.

Table 1: Decrease in required torque as a result of the arm attachment height

Arm	Height in	Moment Arm in	Required Torque Inch-lbs	% decrease in Torque %
A	13.25	51	-60914.7	0
B	11	48.4	-58612.1	3.8
C	8.25	45.9	-56398.1	7.4
D	5.5	43.4	-54184.1	11
E	2.75	40.8	-51881.5	14.8
F	0	38.3	-49667.5	18.5

These numbers were calculated using a static analysis of the system with the arms extended to the most critical position. By using static equilibrium, Newton's first two laws of motion are applied. These being, if a body is at rest, the sum of all forces acting on the body must be zero [21]. At that position, the weight of the arms, pod, and the aerodynamic forces apply a moment that is translated to the gear reducer to show the required torque of the reducer. By knowing that the reducer must counter the moment of the forces on the pod, the required torque was calculated using the following equation.

$$\text{Torque Required (in*lbs)} = \text{moment arm (in)} * \text{Net Force on CG of Pod (lbs)} \quad (1)$$

4.2 Arm A Sensitivity Study

The first step in the sensitivity study of the arm was to conduct a study at the original height. This is needed so that alternate positions could be compared to the original

configuration. It also marks the beginning point of the sensitivity study, and from this point each study conducted will have a height lower than the starting point of 13.25 inches. Within each study, the arm height will be held constant as the cross-sectional area is affected. After running through various cross-sectional areas, the arm height will be decreased and a new study will be conducted. This first study will be labeled Arm A, and the results for the sensitivity study of Arm A can be seen in Table 1 below.

Table 2: Sensitivity study results for Arm A @ 13.25 inches

Arm Thickness Held Constant At 0.75 Inches					
Arm Width	Von Mises Stress	AL 7075 T6 Yield Stress	factor of safety	Al 6061 T6 Yield Stress	factor of safety
In	PSI	PSI		PSI	
4	34110	78600	2.3	40000	1.17
3	35190	78600	2.23	40000	1.14
2	59250	78600	1.33	40000	0.68
1	351900	78600	0.22	40000	0.11
Arm Width Held Constant At 3 Inches					
Arm Thickness	Von Mises Stress	AL 7075 T6 Yield Stress	factor of safety	Al 6061 T6 Yield Stress	factor of safety
In	PSI	PSI		PSI	
0.75	35190	78600	2.23	40000	1.14
0.5	44770	78600	1.76	40000	0.89
Arm Width Held Constant At 2 Inches					
Arm Thickness	Von Mises Stress	AL 7075 T6 Yield Stress	factor of safety	Al 6061 T6 Yield Stress	factor of safety
In	PSI	PSI		PSI	
0.75	59250	78600	1.33	40000	0.68
0.5	79690	78600	0.99	40000	0.5

This table shows the maximum Von Mises Stress in the arm, and compares two different types of aluminum materials. As is seen in the table, the Al 7075 T6 has a much higher yield stress, and is capable of being much smaller without failure. However, The Al 7075 T6 is a much more expensive material and is more difficult to machine. This is why the current arms use Al 6061 T6. The configurations with failure are noted

by having a factor of safety being less than one. This shows that the Al 6061 T6 can only be reduced to a cross-sectional area of $3 \times \frac{3}{4}$ inch, while the Al 7075 T6 can be reduced to either a $3 \times \frac{1}{2}$ or $2 \times \frac{3}{4}$ inch cross-sectional area. In either case the cross-sectional area would be the same, but it is more desirable to reduce the width so that the door can be closed further. For a more accurate display of the location of the Von Mises Stress, and a convergence plot showing that the FEA of Arm A is accurate, refer to Appendix A. Appendix A contains fringe plots of the Von Mises Stress for each configuration in Table 2, and a corresponding convergence plots.

4.3 Arm B Sensitivity Study

The next study looks at the arm being mounted 11 inches above the linear plate. This position will be labeled Arm B. The same FEA was conducted, and the results of the Arm B study are shown below in Table 3. Note that there was no study with an arm width of 1 inch this time due to the drastic increase of the Von Mises Stress in the Arm A study.

Table 3: Sensitivity study results for Arm B @ 11 inches

Arm Thickness Held Constant At 0.75 Inches					
Arm Width	Von Mises Stress	AL 7075 T6 Yield Stress	factor of safety	Al 6061 T6 Yield Stress	factor of safety
In	PSI	PSI		PSI	
4	28390	78600	2.77	40000	1.41
3	30490	78600	2.58	40000	1.31
2	58110	78600	1.35	40000	0.69
Arm Width Held Constant At 3 Inches					
Arm Thickness	Von Mises Stress	AL 7075 T6 Yield Stress	factor of safety	Al 6061 T6 Yield Stress	factor of safety
In	PSI	PSI		PSI	
0.75	30490	78600	2.58	40000	1.31
0.5	37690	78600	2.09	40000	1.06
Arm Width Held Constant At 2 Inches					
Arm Thickness	Von Mises Stress	AL 7075 T6 Yield Stress	factor of safety	Al 6061 T6 Yield Stress	factor of safety
In	PSI	PSI		PSI	
0.75	58110	78600	1.35	40000	0.69
0.5	79620	78600	0.99	40000	0.5

As the table shows, the safety factors have increased in each of the configurations. This is due to the shorter moment arm. As the moment arm decreases the arm will be able to withstand larger forces. Table 3 shows that the Al7075 T6 can almost be reduced to a cross-sectional area of 2x½ inch, and the Al 6061 T6 can now be made with a cross-sectional area of 3x½ inch. This is an improvement from Arm A, and the distance H has only been reduced by 2.25 inches. Fringe plots of the Von Mises Stress for each configuration of the Arm B study, along with the corresponding convergence check can be found in Appendix B.

4.4 Arm C Sensitivity Study

The Arm C sensitivity study was conducted with the arm attached 8.25 inches from the linear plate. This study proved to open a new option for the arm configurations with the Al 7076 T6, and the results are shown in Table 4.

Table 4: Sensitivity study results for Arm C @ 8.25 inches

Arm Thickness Held Constant At 0.75 Inches					
Arm Width	Von Mises Stress	AL 7075 T6 Yield Stress	factor of safety	Al 6061 T6 Yield Stress	factor of safety
in	PSI	PSI		PSI	
4	23000	78600	3.42	40000	1.74
3	23010	78600	3.42	40000	1.74
2	40760	78600	1.93	40000	0.98
Arm Width Held Constant At 3 Inches					
Arm Thickness	Von Mises Stress	AL 7075 T6 Yield Stress	factor of safety	Al 6061 T6 Yield Stress	factor of safety
in	PSI	PSI		PSI	
0.75	23010	78600	3.42	40000	1.74
0.5	28520	78600	2.76	40000	1.40
Arm Width Held Constant At 2 Inches					
Arm Thickness	Von Mises Stress	AL 7075 T6 Yield Stress	factor of safety	Al 6061 T6 Yield Stress	factor of safety
in	PSI	PSI		PSI	
0.75	40760	78600	1.93	40000	0.98
0.5	50280	78600	1.56	40000	0.80

As Table 4 shows, the safety factors have increased in each of the configurations. This is again due to the shorter moment arm. The results show that the Al7075 T6 can now be safely reduced to a cross-sectional area of 2x½ inch, and the Al 6061 T6 can still be made with a cross-sectional area of 3x½ inch. This is again an improvement from the previous arm, and there is now another configuration option. Fringe plots of the Von

Mises Stress for each configuration of the Arm C study, along with the corresponding convergence check can be found in Appendix C.

4.5 Arm D Sensitivity Study

The Arm D sensitivity study was conducted with the arm attached 5.5 inches from the linear plate. This study did not allow for any new configurations, but did show an increase in the safety factors. The results are shown below in Table 5.

Table 5: Sensitivity study results for Arm D @ 5.5 inches

Arm Thickness Held Constant At 0.75 Inches					
Arm Width	Von Mises Stress	AL 7075 T6 Yield Stress	factor of safety	Al 6061 T6 Yield Stress	factor of safety
in	PSI	PSI		PSI	
4	20740	78600	3.79	40000	1.93
3	20940	78600	3.75	40000	1.91
2	41510	78600	1.89	40000	0.96
Arm Width Held Constant At 3 Inches					
Arm Thickness	Von Mises Stress	AL 7075 T6 Yield Stress	factor of safety	Al 6061 T6 Yield Stress	factor of safety
in	PSI	PSI		PSI	
0.75	20940	78600	3.75	40000	1.91
0.5	26740	78600	2.94	40000	1.50
Arm Width Held Constant At 2 Inches					
Arm Thickness	Von Mises Stress	AL 7075 T6 Yield Stress	factor of safety	Al 6061 T6 Yield Stress	factor of safety
in	PSI	PSI		PSI	
0.75	40210	78600	1.96	40000	0.99
0.5	49980	78600	1.57	40000	0.81

The results from sensitivity studies Arm A and Arm D yield the same allowable configurations, but since the safety factors increased there is still room for improvement as the arm height is reduced. Fringe plots of the Von Mises Stress for each configuration of the Arm D study, along with the corresponding convergence check can be found in Appendix D.

4.6 Arm E Sensitivity Study

The Arm E sensitivity study was conducted with the arm attached 2.75 inches from the linear plate. This study showed that all configurations are now possible. The reason for the large increase from Arm D to Arm E came from the geometry of the first bend in the arm. With the arm height at 2.75 inches, Arm E has a 90 degree bend for its first bend instead an angle greater than 90 degrees as in the first four studies. This change in geometry showed a large decrease in the Von Mises Stress throughout the arm. The results are shown below in Table 6.

Table 6: Sensitivity study results for Arm E @ 2.75 inches

Arm Thickness Held Constant At 0.75 Inches					
Arm Width	Von Mises Stress	AL 7075 T6 Yield Stress	factor of safety	Al 6061 T6 Yield Stress	factor of safety
in	PSI	PSI		PSI	
4	20090	78600	3.91	40000	1.99
3	21770	78600	3.61	40000	1.84
2	26540	78600	2.96	40000	1.51
Arm Width Held Constant At 3 Inches					
Arm Thickness	Von Mises Stress	AL 7075 T6 Yield Stress	factor of safety	Al 6061 T6 Yield Stress	factor of safety
in	PSI	PSI		PSI	
0.75	21770	78600	3.61	40000	1.84
0.5	23610	78600	3.33	40000	1.69
Arm Width Held Constant At 2 Inches					
Arm Thickness	Von Mises Stress	AL 7075 T6 Yield Stress	factor of safety	Al 6061 T6 Yield Stress	factor of safety
in	PSI	PSI		PSI	
0.75	26540	78600	2.96	40000	1.51
0.5	33040	78600	2.38	40000	1.21

As Table 6 shows, the safety factors have increased in each of the configurations. This is again due to the shorter moment arm, and the angle of the first bend in the arm. The results show that the Al7075 T6 and the Al 6061 T6 can now be safely reduced to a

cross-sectional area of 2x½ inch. Fringe plots of the Von Mises Stress for each configuration of the Arm E study, along with the corresponding convergence check can be found in Appendix E.

4.7 Arm F Sensitivity Study

The Arm F sensitivity study was conducted with the arm attached 0 inches from the linear plate. This study did not allow for any new configurations, but did show an increase in the safety factors. The results are shown below in Table 7.

Table 7: Sensitivity study results for Arm F @ 0 inches

Arm Thickness Held Constant At 0.75 Inches					
Arm Width	Von Mises Stress	AL 7075 T6 Yield Stress	factor of safety	Al 6061 T6 Yield Stress	factor of safety
in	PSI	PSI		PSI	
4	18290	78600	4.30	40000	2.19
3	15970	78600	4.92	40000	2.50
2	24860	78600	3.16	40000	1.61
Arm Width Held Constant At 3 Inches					
Arm Thickness	Von Mises Stress	AL 7075 T6 Yield Stress	factor of safety	Al 6061 T6 Yield Stress	factor of safety
in	PSI	PSI		PSI	
0.75	15970	78600	4.92	40000	2.50
0.5	17770	78600	4.42	40000	2.25
Arm Width Held Constant At 2 Inches					
Arm Thickness	Von Mises Stress	AL 7075 T6 Yield Stress	factor of safety	Al 6061 T6 Yield Stress	factor of safety
in	PSI	PSI		PSI	
0.75	24860	78600	3.16	40000	1.61
0.5	27840	78600	2.82	40000	1.44

From these results it is evident that the arm dimensions can be reduced from 4x¾ inches to 2x½ inches. In doing this the weight of each arm can be reduced from 22.1 lbs to just under 7 lbs. When considering all four arms this will save over 60 lbs in arm

weight. This does not include the weight saved from using less bearings and eliminating the spacers, or the smaller gear reducer.

4.8 Optimum Design

Realistically, the height above the plate will be determined by the torque motor design and the diameter of the output shaft. The following table shows a design for a reasonable torque motor with a shaft location of $2\frac{3}{4}$ inches above the plate. For this design, Figure 17, the arm design will use a cross-sectional area of $3 \times \frac{1}{2}$ inches. The final stresses in the arm can be seen using the Von Mises Fringe plot in Figure 18.

Table 8: Final stresses in the arm at a height of $2\frac{3}{4}$ inches and a cross-sectional area of $3 \times \frac{1}{2}$ inches including required torque from gear reducer

Arm Width Held Constant At 3 Inches					
Arm Thickness	Von Mises Stress	Al 6061 T6 Yield Stress	factor of safety	Moment Arm	Required Torque
in	PSI	PSI		Inches	Inch-Pounds
0.5	23610	40000	1.69	40.8	52000

Figure 17: Rear isometric view of the optimal design

Figure 18: Von Mises stress distribution throughout 3x0.5 inch rotational arm under critical loading

To ensure the reliability of the data, some hand calculations were also done and compared to that of the sensitivity study. These calculations can be seen in Appendix H.

CHAPTER 5: CONCLUSIONS

By decreasing the height of the rotary drive shaft the arm length was reduced and the moment was successfully decreased. This lowered the required torque of the gear reducer and allows for a smaller reducer to be used. Also, by decreasing the height of the rotary arm to the level of the plate, fewer bearings are needed and the mounts for the bearings can be removed. Decreasing the height of the rotary arm also plays an important role in lowering the stress in the mechanical arm, allowing for the arm to be optimized. It is seen from the sensitivity study of the mechanical arm that the dimensions can be reduced and a smaller arm will withstand the most extreme loading scenario. This will allow for improvements in the sensor pod that could accommodate a greater variety of sensors. A fatigue analysis was also conducted alternating the load from zero to its maximum value. This study showed that the arm could withstand greater than 100000 cycles, or essentially infinite life. The data for this analysis is shown in Appendix G.

Recommendations for future assemblies include a much smaller gear reducer arranged so that the output shaft is as low as it can possibly be. This will eliminate the need for the spacers, and reduce the amount of bearings needed. It is also suggested that the arm dimensions be reduced, and that the aluminum used is 7075 instead of 6061. This is due to the much higher yield strength of the 7075 aluminum. All of these variables together will form a lighter, more efficient design for the C-130 sensor deployment system. This optimal design has the arm placed level with the plate with a cross sectional area of $3 \times \frac{1}{2}$ inches, and a smaller gear reducer. Before making any of these changes a modal analysis should be conducted on the system to ensure that it will not have resonance problems after it is optimized.

REFERENCES

1. Toogood, R., "Pro/MECHANICA Structure" Student Edition Tutorial, University of Alberta, Alberta, Canada 2001
2. Bathe, K. J., "Finite Element Procedures" Prentice-Hall, Inc. Englewood Cliffs, New Jersey 1995
3. Bathe, K. J. and Wilson, Edward L. "Numerical Methods in Finite Element Analysis" Prentice-Hall, Inc. Englewood Cliffs, New Jersey 1976
4. Cook, Robert D., "Finite Element Modeling For Stress Analysis" University of Wisconsin – Madison, John Wiley & Sons, Inc. July 1994
5. Kabaila, A. P. and Pulmano, V. A., "Finite Element Methods in Engineering" The University of New South Wales, Australia, Sydney, Australia, July 1979
6. Reed Business Information, "No need to re-invent this wheel: encoding its history of design rules, wheel manufacturer saves design time. (Knowledge-Based Engineering)." Design News, May 05, 2003
7. Teague, Paul E., "Software Sparks Cool Ideas In Appliances; CAD and FEA yield ideas, fewer prototypes" Design News, October 20, 2003
8. Dana Chase Publications Inc., "A technology solution for a quieter hair dryer motor" Appliance, March 2003
9. University of South Hampton, "Explicit finite element model offers insight into kinematics generated by TKA" Biotech Week, November 12, 2003
10. National Chung Hsing University, "Computer simulates wear behavior appearing in total hip prostheses" Biotech Week, March 19, 2003
11. Warwick, Graham, "Overload led to New York crash; Airbus's structural testing shows that forces on fin of American A300-600R exceeded design limits on attachment lugs" News; Air Transport, November 18, 2003
12. ALGOR, Center for Mechanical Design Technology,
http://www.algor.com/news_pub/cust_app/srs/srstech.asp
13. Cahners Business Information, "Solo applies aerospace lessons to golf club design" Design News, August 06, 2001
14. Textron Power Transmissions, Worm Gear Reducers,
<http://www.textronpt.com/usa/worm.htm>

15. Falk Corporation, Falk's Steelflex® Grid Coupling Line,
<http://www.falkcorp.com/news/steelflexrating.asp>
16. SAE Bolt Strength, http://dodgeram.org/tech/specs/bolts/SAE_bolt_strength.html
17. Beer, Ferdinand P. and Johnston, E. Russell JR., “Mechanics of Materials”,
McGraw Hill, Inc. 1981
18. ADINA R & D, Inc., <http://www.adina.com/>
19. ALGOR Inc, Center for Mechanical Design Technology, Finite Element Analysis,
Simulation and Optimization Methods, <http://www.algor.com/>
20. SRS Technologies, <http://www.srs.com/profile.asp>
21. Waldron, Kenneth J. and Kinzel, Gary L., “Kinematics, Dynamics, and Design of
Machinery”, John Wiley & Sons, Inc. 1999

Appendix A

Arm A Sensitivity Study at 13.25 inches above the plate

Appendix A

The following appendix shows Von Mises Stress fringe plots and convergence checks for each geometry of the arm. The fringe plots use a color scheme to show where the maximum Von Mises Stress occurs so that it can be compared to the yield stress of the aluminum used. Each of the convergence checks show the amount of P-Loops it took for Pro/MECHANICA to obtain the results of the FEA. Each P-Loop is a higher order polynomial equation that is used to determine the gradients between nodes in the mesh. Therefore, with the convergence check of 5 % that was used, and a convergence at the fifth P-Loop, it is understood that the program calculated an error of less than 5 % when going from the fourth to the fifth P-Loop. This shows that the program stopped due to the accuracy of the solution.

Figure 19: Von Mises stress distribution throughout 4x0.75 inch rotational arm under critical loading

Figure 20: Convergence plot showing that the FEA did converge on its fifth polynomial pass with 5% error or less

Stress von Mises (WCS)
 (in lbm / sec² / in²)
 Loadset:LoadSetI

Figure 21: Von Mises stress distribution throughout 3x0.75 inch rotational arm under critical loading

Figure 22: Convergence plot showing that the FEA did converge on its fifth polynomial pass with 5% error or less

Figure 23: Von Mises stress distribution throughout 2x0.75 inch rotational arm under critical loading

Figure 24: Convergence plot showing that the FEA did converge on its fifth polynomial pass with 5% error or less

Figure 25: Von Mises stress distribution throughout 1x0.75 inch rotational arm under critical loading

Figure 26: Convergence plot showing that the FEA did converge on its seventh polynomial pass with 5% error or less

Figure 27: Von Mises stress distribution throughout 3x0.5 inch rotational arm under critical loading

Figure 28: Convergence plot showing that the FEA did converge on its fifth polynomial pass with 5% error or less

Figure 29: Von Mises stress distribution throughout 2x0.5 inch rotational arm under critical loading

Figure 30: Convergence plot showing that the FEA did converge on its fourth polynomial pass with 5% error or less

Appendix B

Arm B Sensitivity Study at 11 inches above the plate

Appendix B

The following appendix shows Von Mises Stress fringe plots and convergence checks for each geometry of the arm. The fringe plots use a color scheme to show where the maximum Von Mises Stress occurs so that it can be compared to the yield stress of the aluminum used. Each of the convergence checks show the amount of P-Loops it took for Pro/MECHANICA to obtain the results of the FEA. Each P-Loop is a higher order polynomial equation that is used to determine the gradients between nodes in the mesh. Therefore, with the convergence check of 5 % that was used, and a convergence at the fifth P-Loop, it is understood that the program calculated an error of less than 5 % when going from the fourth to the fifth P-Loop. This shows that the program stopped due to the accuracy of the solution.

Figure 31: Von Mises stress distribution throughout 4x0.75 inch rotational arm under critical loading

Figure 32: Convergence plot showing that the FEA did converge on its sixth polynomial pass with 5% error or less

Stress von Mises (WCS)
(in lbrn / sec^2 / in^2)
Loadset:LoadSetI

Figure 33: Von Mises stress distribution throughout 3x0.75 inch rotational arm under critical loading

Figure 34: Convergence plot showing that the FEA did converge on its fifth polynomial pass with 5% error or less

Stress von Mises (WCS)
(in lbm / sec² / in²)
Loadset: LoadSet1

Figure 35: Von Mises stress distribution throughout 2x0.75 inch rotational arm under critical loading

Figure 36: Convergence plot showing that the FEA did converge on its fifth polynomial pass with 5% error or less

Stress von Mises (WCS)
(in lbrn / sec^2 / in^2)
Loadset:LoadSet1

Figure 37: Von Mises stress distribution throughout 3x0.5 inch rotational arm under critical loading

Figure 38: Convergence plot showing that the FEA did converge on its fifth polynomial pass with 5% error or less

Stress von Mises (WCS)
(in lbm / sec² / in²)
Loadset: LoadSet1

Figure 39: Von Mises stress distribution throughout 2x0.5 inch rotational arm under critical loading

Figure 40: Convergence plot showing that the FEA did converge on its fourth polynomial pass with 5% error or less

Appendix C

Arm C Sensitivity Study at 8.25 inches above the plate

Appendix C

The following appendix shows Von Mises Stress fringe plots and convergence checks for each geometry of the arm. The fringe plots use a color scheme to show where the maximum Von Mises Stress occurs so that it can be compared to the yield stress of the aluminum used. Each of the convergence checks show the amount of P-Loops it took for Pro/MECHANICA to obtain the results of the FEA. Each P-Loop is a higher order polynomial equation that is used to determine the gradients between nodes in the mesh. Therefore, with the convergence check of 5 % that was used, and a convergence at the fifth P-Loop, it is understood that the program calculated an error of less than 5 % when going from the fourth to the fifth P-Loop. This shows that the program stopped due to the accuracy of the solution.

Figure 41: Von Mises stress distribution throughout 4x0.75 inch rotational arm under critical loading

Figure 42: Convergence plot showing that the FEA did converge on its sixth polynomial pass with 5% error or less

Figure 43: Von Mises stress distribution throughout 3x0.75 inch rotational arm under critical loading

Figure 44: Convergence plot showing that the FEA did converge on its fifth polynomial pass with 5% error or less

Figure 45: Von Mises stress distribution throughout 2x0.75 inch rotational arm under critical loading

Figure 46: Convergence plot showing that the FEA did converge on its fifth polynomial pass with 5% error or less

Stress von Mises (WCS)
(in lbm / sec^2 / in^2)
Loadset:LoadSetI

Figure 47: Von Mises stress distribution throughout 3x0.5 inch rotational arm under critical loading

Figure 48: Convergence plot showing that the FEA did converge on its fifth polynomial pass with 5% error or less

Stress von Mises (WCS)
 (in lbm / sec^2 / in^2)
 Loadset:LoadSet1

Figure 49: Von Mises stress distribution throughout 2x0.5 inch rotational arm under critical loading

Figure 50: Convergence plot showing that the FEA did converge on its fourth polynomial pass with 5% error or less

Appendix D

Arm D Sensitivity Study at 5.5 inches above the plate

Appendix D

The following appendix shows Von Mises Stress fringe plots and convergence checks for each geometry of the arm. The fringe plots use a color scheme to show where the maximum Von Mises Stress occurs so that it can be compared to the yield stress of the aluminum used. Each of the convergence checks show the amount of P-Loops it took for Pro/MECHANICA to obtain the results of the FEA. Each P-Loop is a higher order polynomial equation that is used to determine the gradients between nodes in the mesh. Therefore, with the convergence check of 5 % that was used, and a convergence at the fifth P-Loop, it is understood that the program calculated an error of less than 5 % when going from the fourth to the fifth P-Loop. This shows that the program stopped due to the accuracy of the solution.

Stress von Mises (WCS)
(in lbm / sec² / in²)
Loadset:LoadSet1

Figure 51: Von Mises stress distribution throughout 4x0.75 inch rotational arm under critical loading

Figure 52: Convergence plot showing that the FEA did converge on its fifth polynomial pass with 5% error or less

Figure 53: Von Mises stress distribution throughout 3x0.75 inch rotational arm under critical loading

Figure 54: Convergence plot showing that the FEA did converge on its fifth polynomial pass with 5% error or less

Figure 55: Von Mises stress distribution throughout 2x0.75 inch rotational arm under critical loading

Figure 56: Convergence plot showing that the FEA did converge on its fifth polynomial pass with 5% error or less

Stress von Mises (WCS)
(in lbm / sec^2 / in^2)
Loadset:LoadSet1

Figure 57: Von Mises stress distribution throughout 3x0.5 inch rotational arm under critical loading

Figure 58: Convergence plot showing that the FEA did converge on its fifth polynomial pass with 5% error or less

Figure 59: Von Mises stress distribution throughout 2x0.5 inch rotational arm under critical loading

Figure 60: Convergence plot showing that the FEA did converge on its fifth polynomial pass with 5% error or less

Appendix E

Arm E Sensitivity Study at 2.75 inches above the plate

Appendix E

The following appendix shows Von Mises Stress fringe plots and convergence checks for each geometry of the arm. The fringe plots use a color scheme to show where the maximum Von Mises Stress occurs so that it can be compared to the yield stress of the aluminum used. Each of the convergence checks show the amount of P-Loops it took for Pro/MECHANICA to obtain the results of the FEA. Each P-Loop is a higher order polynomial equation that is used to determine the gradients between nodes in the mesh. Therefore, with the convergence check of 5 % that was used, and a convergence at the fifth P-Loop, it is understood that the program calculated an error of less than 5 % when going from the fourth to the fifth P-Loop. This shows that the program stopped due to the accuracy of the solution.

Figure 61: Von Mises stress distribution throughout 4x0.75 inch rotational arm under critical loading

Figure 62: Convergence plot showing that the FEA did converge on its sixth polynomial pass with 5% error or less

Stress von Mises (WCS)
(in lbm / sec² / in²)
Loadset:LoadSetI

Figure 63: Von Mises stress distribution throughout 3x0.75 inch rotational arm under critical loading

Figure 64: Convergence plot showing that the FEA did converge on its fifth polynomial pass with 5% error or less

Figure 65: Von Mises stress distribution throughout 2x0.75 inch rotational arm under critical loading

Figure 66: Convergence plot showing that the FEA did converge on its fifth polynomial pass with 5% error or less

Figure 67: Von Mises stress distribution throughout 3x0.5 inch rotational arm under critical loading

Figure 68: Convergence plot showing that the FEA did converge on its fifth polynomial pass with 5% error or less

Figure 69: Von Mises stress distribution throughout 2x0.5 inch rotational arm under critical loading

Figure 70: Convergence plot showing that the FEA did converge on its fifth polynomial pass with 5% error or less

Appendix F

Arm F Sensitivity Study at 0 inches above the plate

Appendix F

The following appendix shows Von Mises Stress fringe plots and convergence checks for each geometry of the arm. The fringe plots use a color scheme to show where the maximum Von Mises Stress occurs so that it can be compared to the yield stress of the aluminum used. Each of the convergence checks show the amount of P-Loops it took for Pro/MECHANICA to obtain the results of the FEA. Each P-Loop is a higher order polynomial equation that is used to determine the gradients between nodes in the mesh. Therefore, with the convergence check of 5 % that was used, and a convergence at the fifth P-Loop, it is understood that the program calculated an error of less than 5 % when going from the fourth to the fifth P-Loop. This shows that the program stopped due to the accuracy of the solution.

Stress von Mises (WCS)
(in lbm / sec² / in²)
Loadset:LoadSetI

Figure 71: Von Mises stress distribution throughout 4x0.75 inch rotational arm under critical loading

Figure 72: Convergence plot showing that the FEA did converge on its sixth polynomial pass with 5% error or less

Figure 73: Von Mises stress distribution throughout 3x0.75 inch rotational arm under critical loading

Figure 74: Convergence plot showing that the FEA did converge on its fifth polynomial pass with 5% error or less

Stress von Mises (WCS)
(in lbf / sec² / in²)
Loadset:LoadSet1

Figure 75: Von Mises stress distribution throughout 2x0.75 inch rotational arm under critical loading

Figure 76: Convergence plot showing that the FEA did converge on its fifth polynomial pass with 5% error or less

Figure 77: Von Mises stress distribution throughout 3x0.5 inch rotational arm under critical loading

Figure 78: Convergence plot showing that the FEA did converge on its fifth polynomial pass with 5% error or less

Stress von Mises (WCS)
(in lbm / sec^2 / in^2)
_loadset:LoadSetI

Figure 79: Von Mises stress distribution throughout 2x0.5 inch rotational arm under critical loading

Figure 80: Convergence plot showing that the FEA did converge on its fifth polynomial pass with 5% error or less

Appendix G

This appendix consists of fatigue analysis plots showing log life, life confidence, and a fatigue factor of safety fringe plot. The life confidence was calculated using a desired life of 100000 cycles, and a level 3 confidence is considered adequate confidence in Pro/MECHANICA. The level comes from a ratio between the desired life and the expected life from the FEA. A Goodman diagram is also shown in this appendix. The Goodman Diagram shows nearly infinite life which is comparable to that of the Pro/MECHANICA analysis showing just over 10^6 life cycles. The Goodman diagram actually shows the load line reaching the modified Goodman line, denoting a value of one for the fatigue factor of safety. This shows that it is safe to assume infinite life for the optimized arm.

Fatigue Log Life (WCS)
Loadset:Mode Combination

Figure 81: Fatigue log life fringe plot showing that the arm can withstand 10^6 cycles

Fatigue Confidence of Life (WCS)
Loadset: Mode Combination

Figure 82: Fatigue confidence of life fringe plot showing adequate confidence

Figure 83: Goodman diagram for optimized arm with cross-sectional area of $3 \times \frac{1}{2}$ inches

Appendix H

To ensure confidence in the software's calculations, a simplified arm was created and compared to hand calculated results. This simplified arm maintained the same loading and moment arm of the original but without the complicated geometry. The cross-sectional area of the simplified arm is also equal to that of the recommended optimized arm. The following figures show the simplified arm with the forces and constraints, and a fringe plot of the Von Mises Stress throughout the arm.

Figure 84: Simplified arm with constraints and loading

Figure 85: Von Mises fringe plot of the simplified arm geometry

The finite element method showed a maximum stress of 26290 psi. This value was lower than that of the original arm, and was expected because there was no fillet radius in the simplified geometry. Using the following equations, the maximum stress within the arm was also calculated by hand and compared to that of the FEA:

Moment arm = 40 inches

$F_x = 320 \text{ lbs}$

$F_z = 160 \text{ lbs}$

$c = 1.5 \text{ inches}$

$M_{y1} = \text{Moment arm} * F_x = 40 * 320 \text{ in*lb} = 12800 \text{ in*lb}$

$M_{y2} = \text{Moment arm} * F_z = 40 * 160 \text{ in*lb} = 6400 \text{ in*lb}$

$$I = 1/12 * b * h^3 = 1.125 \text{ in}^4$$

$$\sigma = M * c / I = M_{y1} * c / I + M_{y2} * c / I = [(12800 * 1.5 / 1.125) + (6400 * 1.5 / 1.125)] = 25600 \text{ psi}$$

$$\% \text{ difference} = 100 * (\sigma_{\text{FEA}} - \sigma_{\text{hand}}) / \sigma_{\text{FEA}} = 2.63\%$$