ARMY RESEARCH LABORATORY

Design of a Three-Axis Machine Tool Module

Marshal A. Childers

ARL-MR-0561 **JUNE 2003**

Approved for public release; distribution is unlimited.

NOTICES

Disclaimers

Army Research Laboratory
Aberdeen Proving Ground, MD 21005-5066

ARL-MR-0561 June 2003

Design of a Three-Axis Machine Tool Module

Marshal A. Childers
Weapons & Materials Research Directorate

Approved for public release; distribution is unlimited.

INTENTIONALLY LEFT BLANK

REPORT DOCUMENTATION PAGE

Form Approved
OMB No. 0704-0188

1. REPORT DATE *(DD-MM-YYYY)* June 2003	2. REPORT DATE Final	3. DATES COVERED (From - To)

4. TITLE AND SUBTITLE Design of a Three-Axis Machine Tool Module	5a. CONTRACT NUMBER
	5b. GRANT NUMBER
	5c. PROGRAM ELEMENT NUMBER
6. AUTHOR(S) Childers, M.A. (ARL)	5d. PROJECT NUMBER 1L162618.H80
	5e. TASK NUMBER
	5f. WORK UNIT NUMBER
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) U.S. Army Research Laboratory Weapons & Materials Research Directorate Aberdeen Proving Ground, MD 21005-5066	8. PERFORMING ORGANIZATION REPORT NUMBER ARL-MR-0561
9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)	10. SPONSOR/MONITOR'S ACRONYM(S)
	11. SPONSOR/MONITOR'S REPORT NUMBER(S)

12. DISTRIBUTION/AVAILABILITY STATEMENT

Approved for public release; distribution is unlimited.

13. SUPPLEMENTARY NOTES

14. ABSTRACT

This report documents the design improvement process of the components in a tool module for a three-axis machine tool, which occurred during the period of March-April 2002 in support of a critical U.S. Army Materiel Command manufacturing technology (ManTech) program. Given a preliminary physical design, the objective was to apply finite element analysis (FEA) to predict stresses in the module and apply necessary design changes to increase the survivability and expected life of the device. The module is a component in a complex three-axis machine tool system. With a rigid structure assumption, an FEA model was developed, based on a preliminary design, and was used to predict stresses in the device when it is exposed to an expected loading condition. The effect of using contact elements in the FEA to calculate stresses of specific components was evaluated. Large calculated stresses that were concentrated in the angle brackets prompted model improvements and structural design changes, including grid refinement of the FEA mesh and increased angle bracket thickness. Based on the results of the analyses, recommendations were made for changes from the original design of the angle brackets.

15. SUBJECT TERMS

contact elements FEA structural analysis 3-D stress analysis

16. SECURITY CLASSIFICATION OF			17. LIMITATION OF ABSTRACT	18. NUMBER OF PAGES	19a. NAME OF RESPONSIBLE PERSON Marshal A. Childers
a. REPORT	b. ABSTRACT	c. THIS PAGE			19b. TELEPHONE NUMBER *(Include area code)*
Unclassified	Unclassified	Unclassified	UL	25	410-306-1951

INTENTIONALLY LEFT BLANK

Contents

List of Figures

INTENTIONALLY LEFT BLANK

Acknowledgments

The author recognizes Mr. Robert Kaste of the Composites and Lightweight Structures Branch, Weapons and Materials Research Directorate, U.S. Army Research Laboratory, for his support during the performance of this analysis and preparation of the report. His advice about the structural analysis and his careful technical review of this document are greatly appreciated.

INTENTIONALLY LEFT BLANK

1. Introduction

The purpose of this report is to document the design improvement process of the components in a tool module for a three-axis machine, which occurred during the period of March-April 2002 in support of a critical U.S. Army Materiel Command manufacturing technology (ManTech) program. Given a preliminary physical design, the objective was to apply finite element analysis (FEA) to predict stresses in the module, hereafter called "the device," and to apply necessary design changes to increase the survivability and expected life of the device. This device is a component in a complex system that is referred to as "the machine" in this report.

2. Problem Description

The device (see Figure 1) consisted of a linkage that was designed to transfer an applied load in the z-direction on the bottom bracket shaft interface to a linear translation in the y-direction of the front piece. In addition to the resultant y-direction load on the front piece, an additional load in the x-direction is applied by the operation of the machine. The proposed material for each component of the device was 0.25-inch thick A36 mild steel. Engineering drawings for each component of the device are provided in Appendix A of this report.

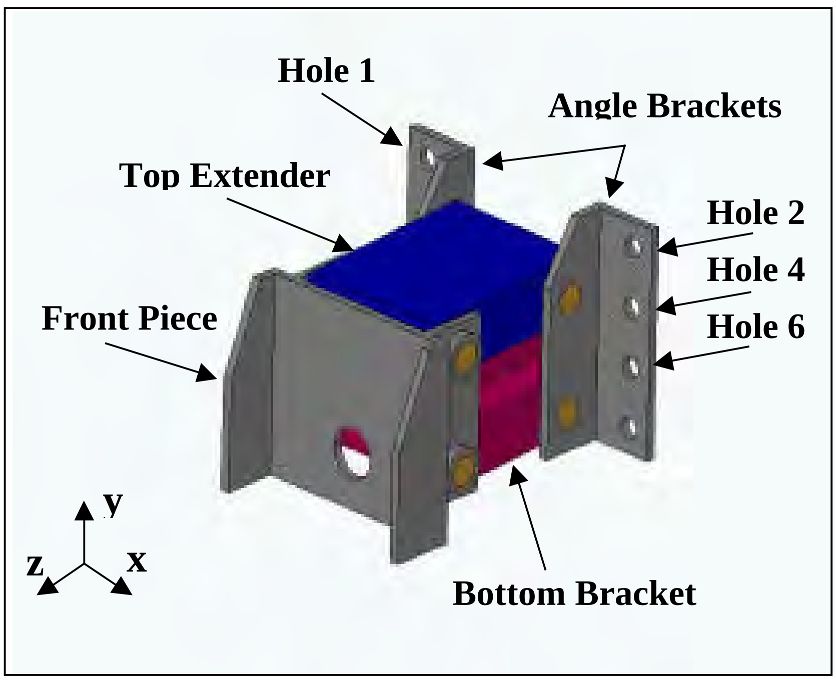

Figure 1. Tool module.

3. Analysis

ANSYS[1] structural analysis software was used to provide a three-dimensional (3-D) static model of the resultant stresses in the device. The free body diagram provided in Figure 2 shows the boundary conditions for the problem. The angle brackets were restricted from translation in the z-direction by the displacement boundary conditions on the back surface from the top down to half-way between the 1-2 and 3-4 mounting hole pairs. To simulate the restriction imposed by mounting bolts, the mounting hole surfaces of the first three pairs were restricted from displacement in the x-direction and y-direction, and nodes on the front surface around the hole edge were restricted in the z-direction.

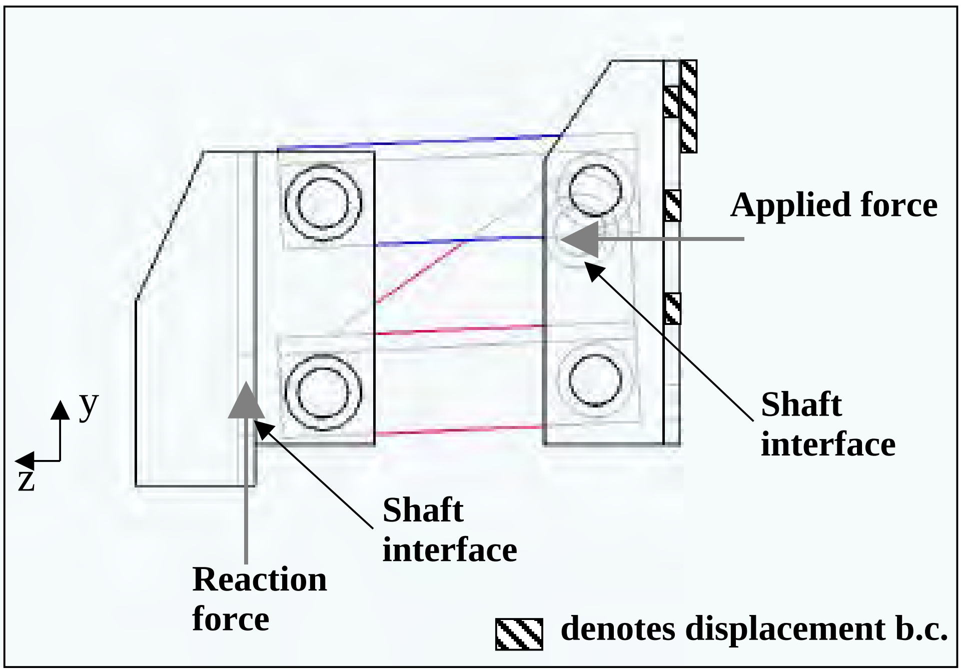

Figure 2. Free body diagram.

Rather than apply a load at the bottom bracket shaft interface and allow a full body rotation, the free body diagram was used to solve for the reaction forces on the front piece. The calculated reaction forces were applied to the hole face in the front piece, and the bottom bracket shaft hole interface was restricted from displacement in the z-direction and y-direction.

To expedite a solution, the device was modeled as a rigid structure. To encompass the effects of a 3-D linkage assembly connected by shafts requires the model to include the effects of surface

[1]ANSYS (not an acronym) is a software of Swanson Analysis Systems, Inc., P.O. Box 65, Johnson Road, Houston PA 15342-0065.

interaction between the links and shafts. To justify the use of a rigid structure approach, the simplified linkage shown in Figure 3 was modeled in two ways with ANSYS: first, as an assembly with contact elements prescribed at the shaft interface, and second, as a rigid structure. In both cases, the load and boundary conditions were as shown in Figure 3.

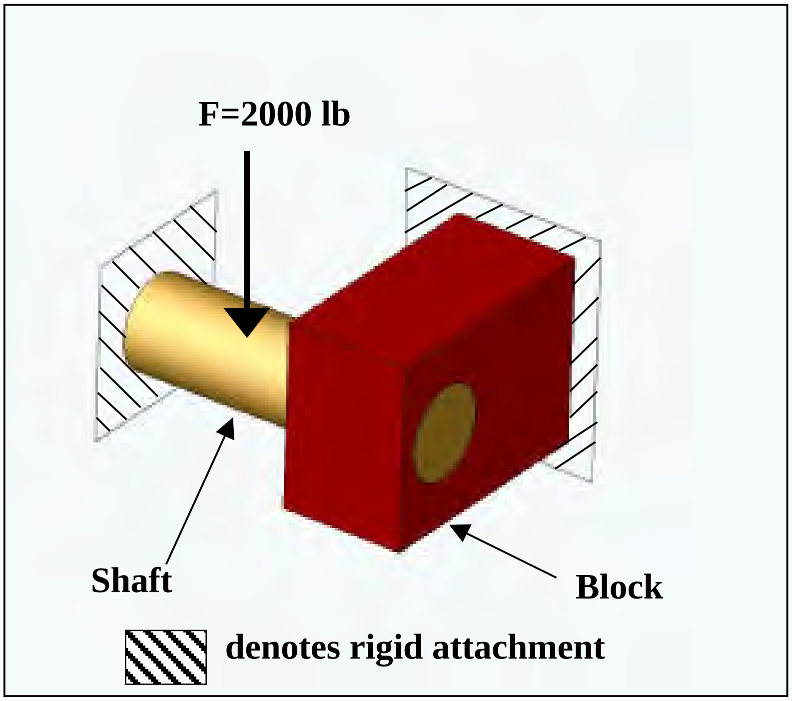

Figure 3. Simplified linkage.

Given that the shaft and block had similar material stiffness, the contact problem was classified as flexible to flexible. First, the shaft and block volumes were meshed with Solid45[2] type elements; second, Conta174[3] contact elements were allocated on the hole face in the block, and third, Targe170[3] target elements were applied to the shaft face. To minimize the amount of surface penetration, the normal penalty stiffness was prescribed as a maximum available value of 1. A coefficient of friction of 0.1 was applied at the material interface. All remaining ANSYS contact element options remained at the default values. Figure 4 shows that the computed maximum Von Mises stress in the assembly contact model was valued at 6.7 kilo-pounds per square inch (ksi). It is observed that the maximum stress appeared in the shaft interface region and that stresses are also concentrated at the fixed end of the shaft. The computed von Mises stresses for the rigid structure model are depicted in Figure 5. The maximum computed stress value for the rigid model was approximately 13 ksi and was situated

[2] ANSYS solid element type
[3] ANSYS contact element type

at the fixed end of the shaft. The maximum calculated stress at the shaft interface for the rigid model was approximately 6 ksi.

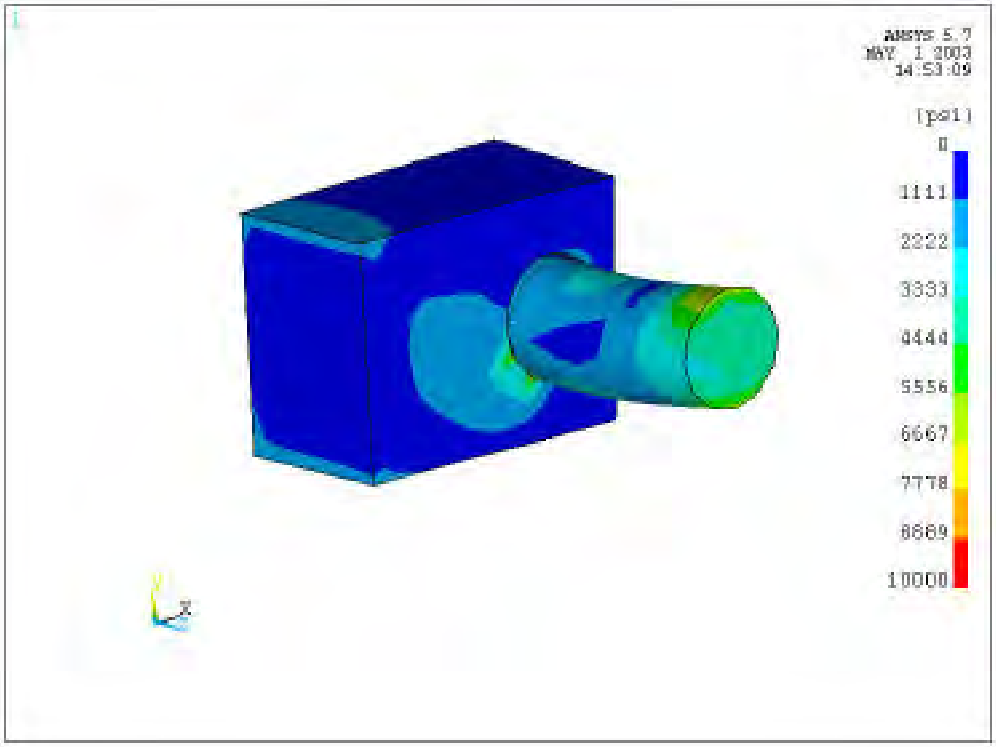

Figure 4. Calculated von Mises stresses for simplified linkage contact model.

The results obtained from the comparison between a rigid simplified linkage and an assembly with contact elements show a significant difference in computed stresses at the region of applied boundary conditions. Thus, the validity of the applied boundary conditions must be considered when one is interpreting the calculated results. Given that the calculated von Mises stresses at the shaft interface for both models had similar magnitudes, the rigid structure assumption was acceptable for considering the stresses that passed through these interfaces.

The preliminary solid model geometry of the device was created with SolidWorks[4] software, and a rigid continuous model was exported in Initial Graphics Exchange Specification (IGES) format for use in ANSYS. When the IGES file was imported into ANSYS, the resultant volume was assigned material properties for mild steel and meshed with Solid92[5] tetrahedral elements. Approximately 100,000 elements were used to describe this structure. The aforementioned boundary conditions were applied, and the applied loads at the front piece shaft interface were 7500 pounds in the +y-direction and 105 pounds in the +x-direction.

---

[4] SolidWorks Corporation, 300 Baker Avenue, Concord MA 01742.

[5] ANSYS solid element type

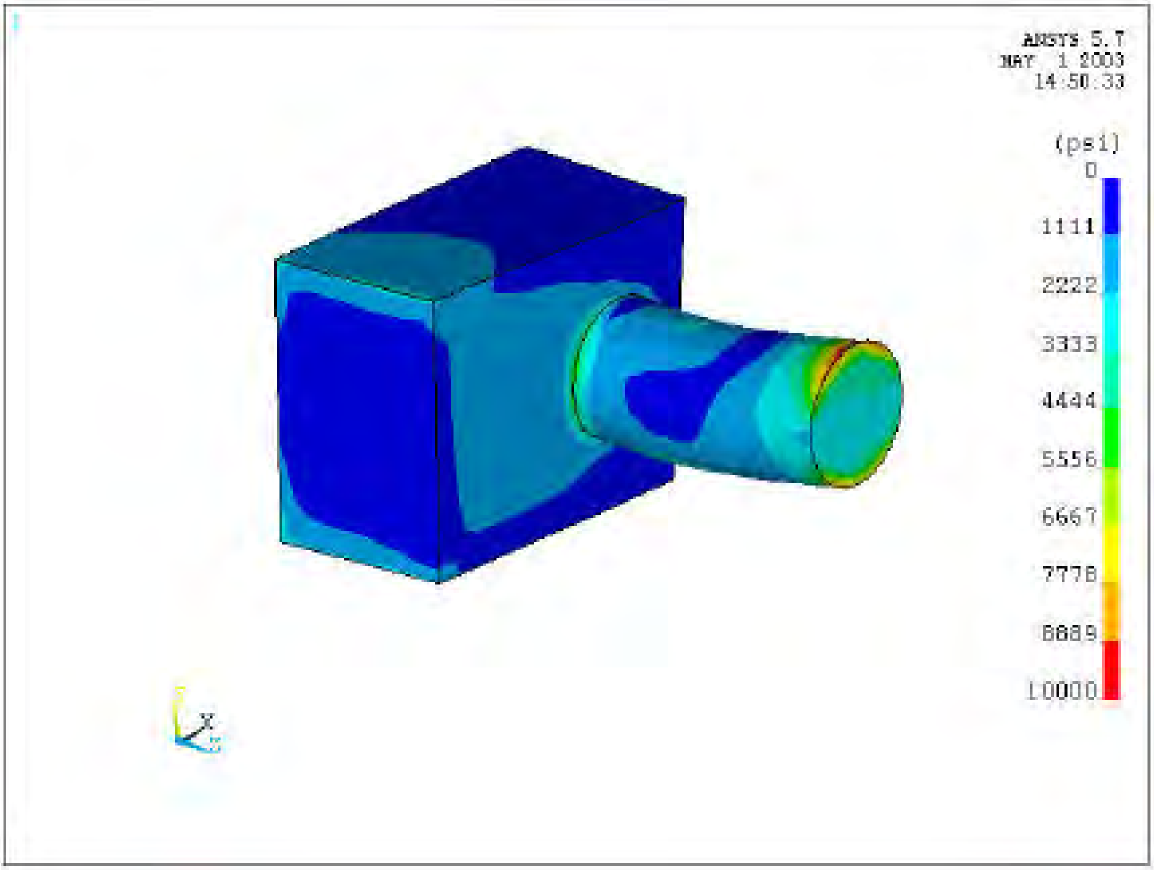

Figure 5. Calculated von Mises stresses for simplified linkage rigid model.

4. Results

The computed von Mises stresses for the preliminary design are shown in Figure 6. Regions of high stress concentration were found in the angle brackets on the edges of the fifth and sixth mounting holes, in the inside corners, and where the angle bracket connects to the top extender. Given that the design did not intend for the angle bracket and top extender to be rigidly attached, these stresses will not appear in the actual device and therefore can be ignored. The large calculated stresses that appeared on the edges of the fifth and sixth mounting holes may be attributable to the applied boundary conditions but at this point in the analysis, these stresses were assumed to be legitimate and to require further attention. To decrease the computed stresses in the angle bracket mounting holes and corners, an increase in the angle bracket thickness and/or an application of radii on the appropriate edges is recommended.

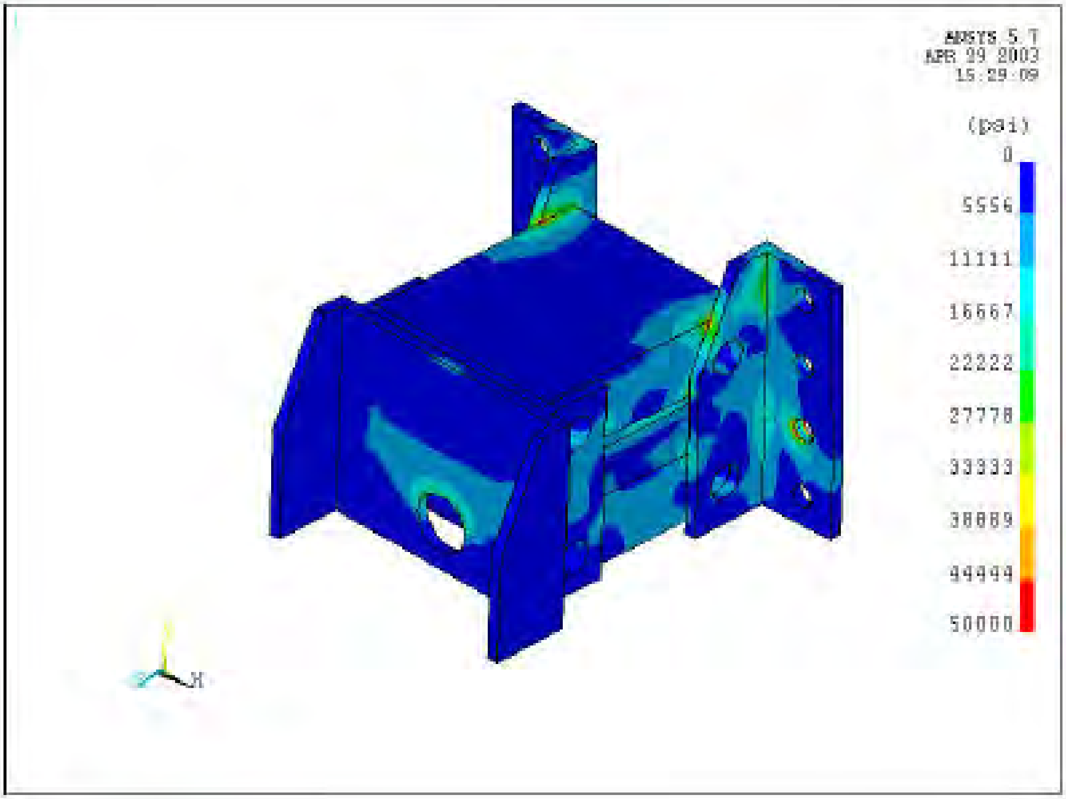

Figure 6. Calculated von Mises stresses for preliminary device.

Modifications of the design were made, based on the calculated stresses in the preliminary model. The first modification was an increase in the angle bracket thickness from 0.25 to 0.375 inch. The remaining model dimensions were unchanged and the boundary conditions were as in the preliminary model. Figure 7 shows the calculated von Mises stresses in the 0.375-inch thick bracket. It is observed that the stresses in the corner are reduced in comparison to those of the preliminary model and are acceptable with respect to the yield stress of A36 steel (approximately 36 ksi). However, the regions near the edges of the fifth and sixth holes still exhibit large stress values. If the stresses on the hole edges were realistic, then the stresses in the thicker angle brackets would have smaller values in comparison to those of the 0.25-inch angle brackets. Since the hole edge stresses are approximately the same for both angle bracket thicknesses, it is suspected that the displacement boundary condition imposed on the inner wall and front edges of the holes contributed to the calculated stress values in this region.

Based on the results from the second model, it was assumed that the displacement boundary condition imposed on the nodes situated on the hole edge was insufficient to describe the contact between a bolt head and the front surface of the angle bracket. Therefore, a third model, which contained 0.25-inch thick angle brackets was developed so that a larger area of nodes on the front surface was restricted from motion in the z-direction. The stress results from this model showed similar stresses on the hole edges as those of the first and second models. Furthermore, it was observed that the high stress values occurred in only a few elements on the hole edge, which would suggest that mesh refinement near the holes may improve the integrity of the results.

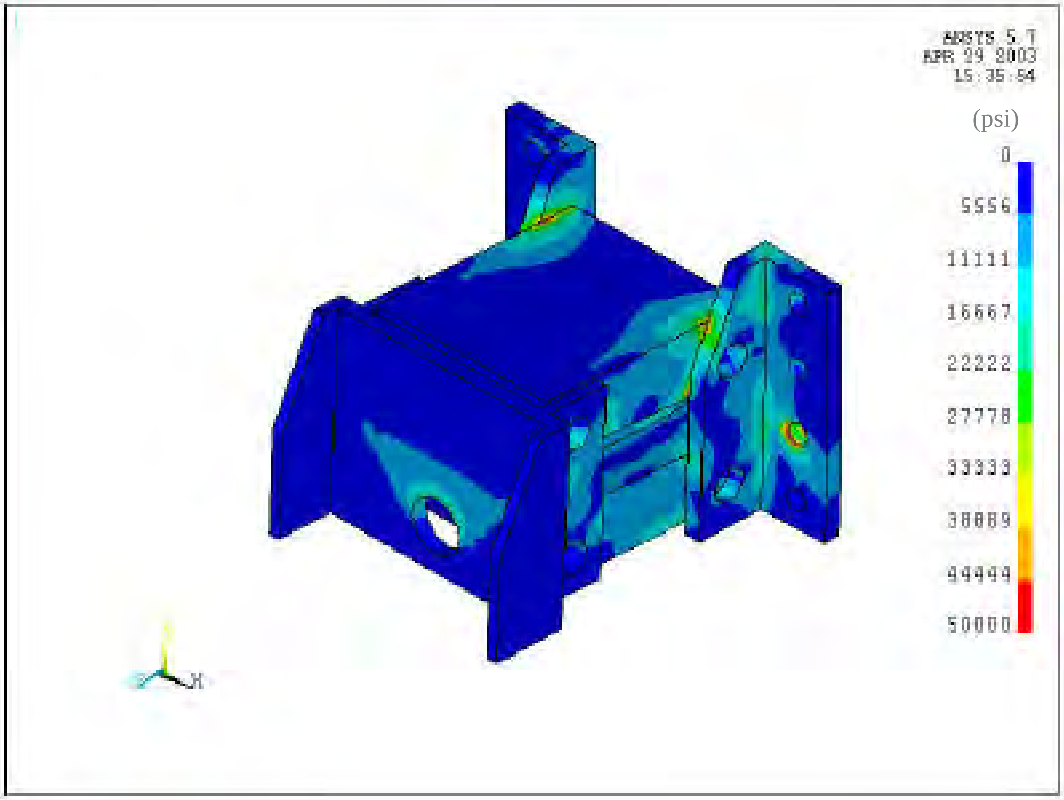

Figure 7. Calculated von Mises stresses for 0.375-inch thick angle brackets.

A fourth model was developed so that grid refinement was performed on elements in the region surrounding the fifth and sixth mounting holes of the third model. This modification increased the number of mesh elements to approximately 140,000 elements. The fourth model was subjected to the same loading and boundary conditions as the third model. Figure 8 shows that the calculated von Mises stresses on the hole edges are reduced to an acceptable level with respect to the yield stress of the material.

To further determine if 0.375-inch thick angle brackets are sufficient for this design, the free body diagram was used to solve for the resultant forces (3,750 pounds in the +y-direction and 6,700 pounds in the +z-direction) on each angle bracket. These forces, in conjunction with stress concentration factor theory, were then used to calculate an approximate maximum stress (12 ksi) in a cross section at the mounting holes, which was smaller than the yield stress of A36 steel.

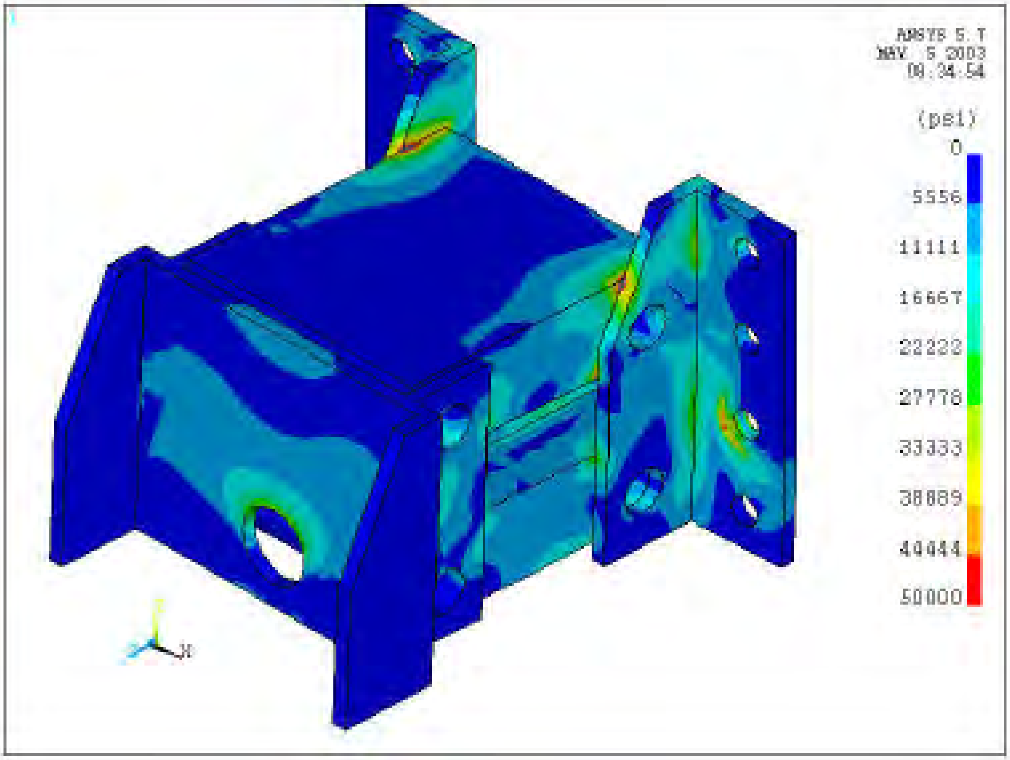

Figure 8. Calculated von Mises stresses for grid-refined 0.25-inch thick angle brackets.

5. Conclusions and Recommendations

FEA was applied to the machine tool module to predict stresses that will occur when the device is exposed to an expected loading condition. A rigid structure assumption was made, and design changes based on the FEA results were explored. Initial structural analyses indicated large calculated stress values on some mounting hole edges and in the corners of the angle brackets. The stresses in the corners of the angle bracket were reduced in a subsequent model by an increase in the angle bracket thickness to 0.375 inch. Additional models that included improved boundary conditions and grid refinement were investigated, and it was determined that the observed stresses on the mounting hole edges were an artifact of the initial model boundary conditions. Stress concentration theory was used to calculate an approximate maximum stress at the mounting holes. Given that the calculated maximum stress in the hole region was approximately 12 ksi and that the stresses in the corners of the 0.375-inch thick angle brackets were smaller than the yield stress of the material (approximately 36 ksi), it is recommended that the device be constructed with 0.375-inch thick angle brackets. Furthermore, these angle brackets are to be constructed from standard "L" channel that has a substantial radius between the webs. This feature will further reduce the stresses in these components and thus increase their expected life. All other components should be constructed of 0.25-inch thick A36 steel.

Appendix A. Three-Axis Machine Tool Module Components: Engineering Drawings

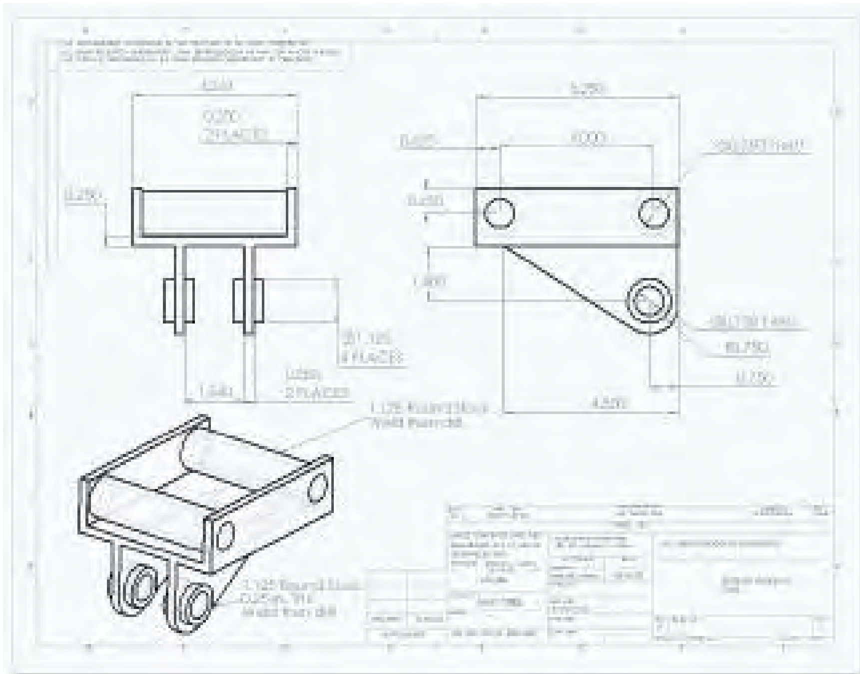

Figure A-1. Bottom bracket.

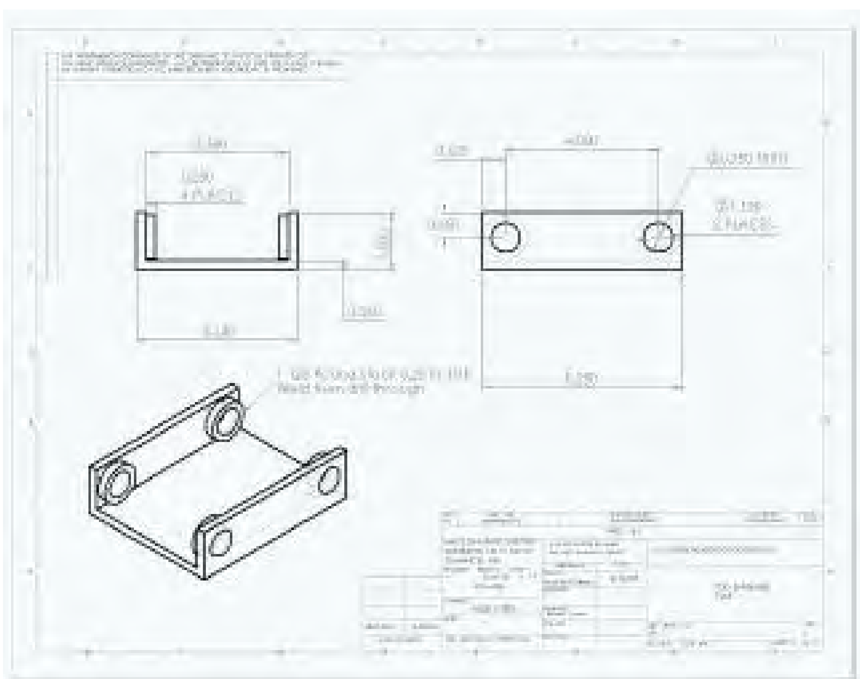

Figure A-2. Top extender.

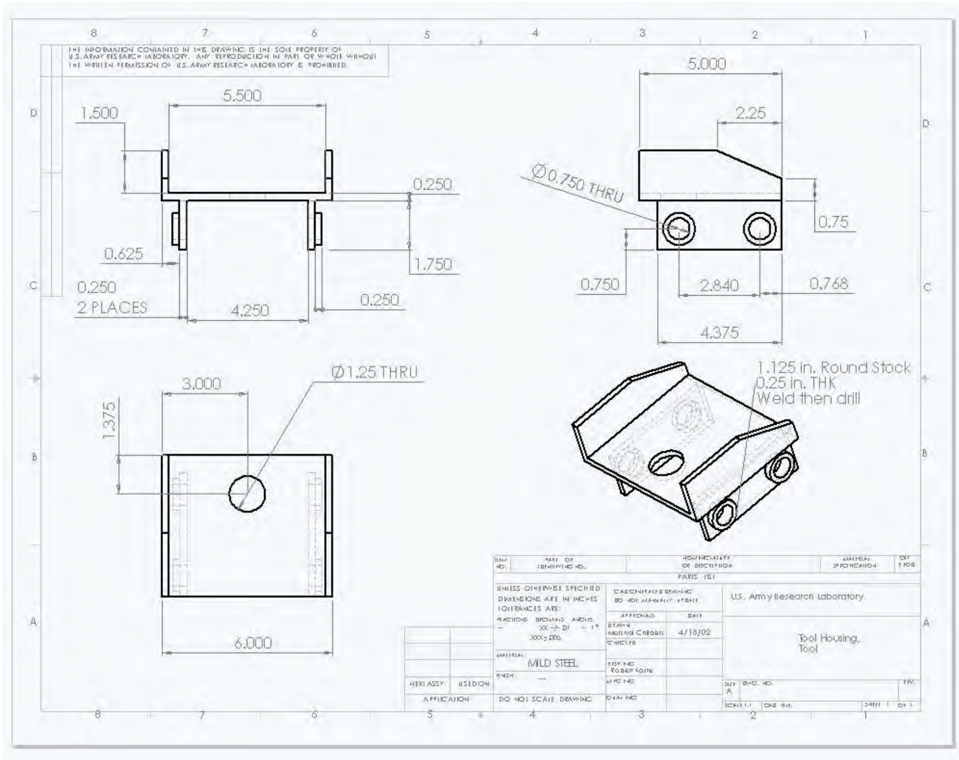

Figure A-3. Tool housing.

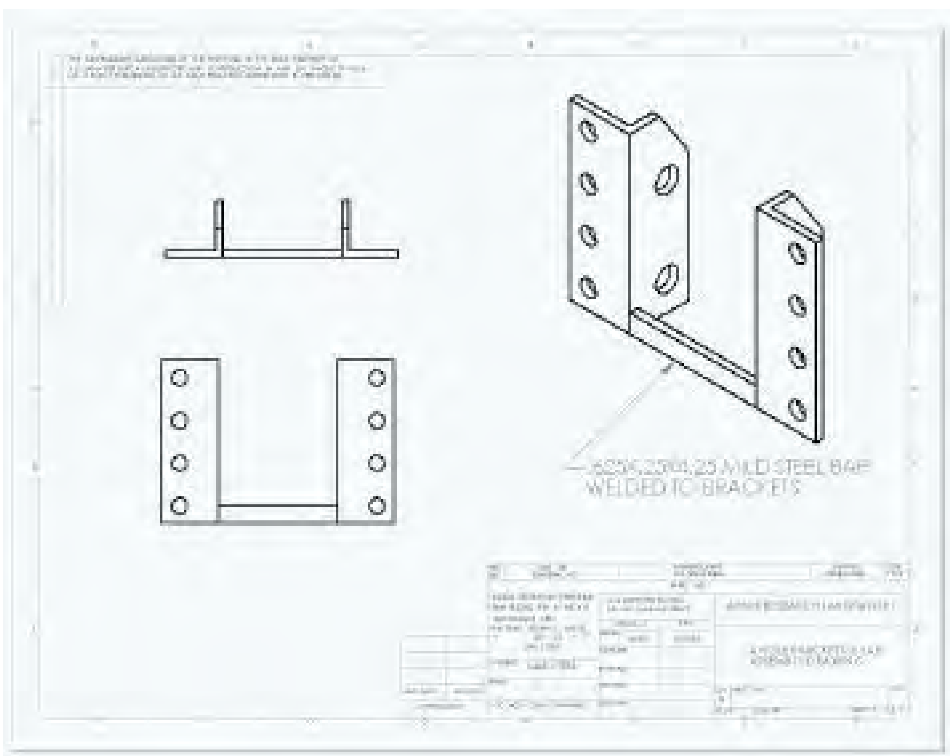

Figure A-4. Bracket assembly drawing.

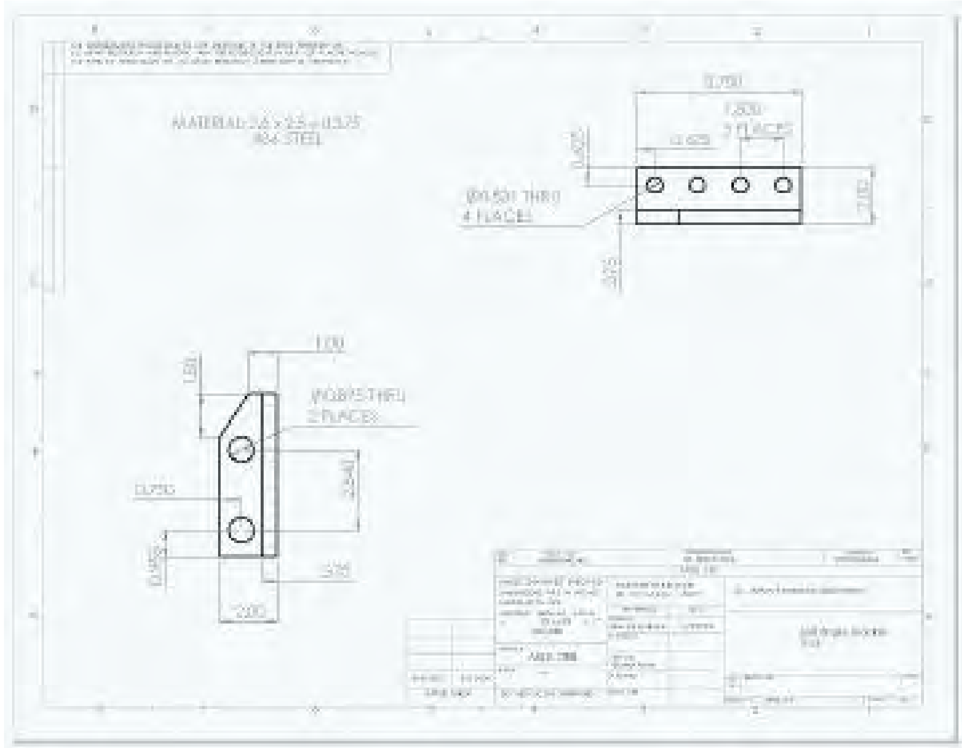

Figure A-5. Left angle bracket.

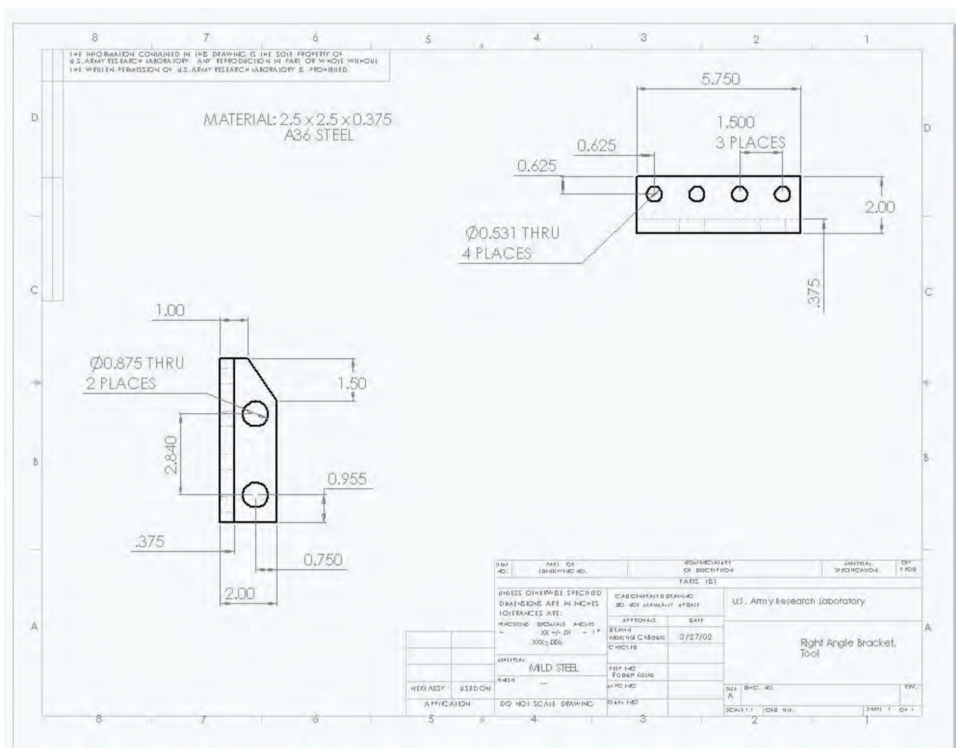

Figure A-6. Right angle bracket.

NO. OF
COPIES ORGANIZATION

1 ADMINISTRATOR
DEFENSE TECHNICAL INFO CTR
ATTN DTIC OCA
8725 JOHN J KINGMAN RD STE 0944
FT BELVOIR VA 22060-6218

1 DIRECTOR
US ARMY RSCH LABORATORY
ATTN AMSRL CI AI R REC MGMT
2800 POWDER MILL RD
ADELPHI MD 20783-1197

1 DIRECTOR
US ARMY RSCH LABORATORY
ATTN AMSRL CI LL TECH LIB
2800 POWDER MILL RD
ADELPHI MD 20783-1197

1 DIRECTOR
US ARMY RSCH LABORATORY
ATTN AMSRL D D SMITH
2800 POWDER MILL RD
ADELPHI MD 20783-1197

2 DIRECTOR
US ARMY RSCH LABORATORY
ATTN AMSRL SE RL M DUBEY
B PIEKARSKI
2800 POWDER MILL RD
ADELPHI MD 20783-1197

2 DIRECTOR
US ARMY RSCH LABORATORY
ATTN AMSRL SE RM H WALLACE
AMSRL SE SS LADAS
2800 POWDER MILL RD
ADELPHI MD 20783-1197

1 CDR US ARMY TACOM ARDEC
ATTN AMSTA AR FSP M HOLLIS
PICATINNY ARSENAL NJ 07806-5000

1 CDR NAVAL SURF WARFARE CTR
ATTN G33 D HAGEN
17320 DAHLGREN ROAD
DAHLGREN VA 22448-5100

3 PROGRAM MGR ITTS
PEO STRI
ATTN AMFTI EL D SCHNEIDER
12350 RESEARCH PKWY
ORLANDO FL 32826-3276

NO. OF
COPIES ORGANIZATION

1 US ARMY CECOM RDEC
ATTN AMSEL RD NV AS UEA
G KLAUBER
10221 BURBECK ROAD
FT BELVOIR VA 22060-5806

1 APPLIED PHYSICS LABORATORY
ATTN W D'AMICO
11100 JOHNS HOPKINS ROAD
LAUREL MD 20723-6099

ABERDEEN PROVING GROUND

2 DIRECTOR
US ARMY RSCH LABORATORY
ATTN AMSRL CI LP (TECH LIB)
BLDG 305 APG AA

2 CDR USA AMSAA
ATTN AMXSY EV G CASTLEBURY
AMXSY EF S MCKEY
BLDG 392

1 US ARMY EVALUATION CMD
ATTN USA CSTE EAC AZ R MIRABELLE
BLDG 4120

2 CDR US ARMAMENT RD&E CTR
ATTN AMSTA AR FST T J MATTS
J WHITESIDE
BLDG 120

1 CDR USA DTC
ATTN CSTE DTC TTM J SCHNELL
RYAN BLDG

2 DIR USARL
ATTN AMSRL WM B A HORST
W CIEPIELLA
BLDG 4600

18 DIR USARL
ATTN AMSRL WM BA T BROWN (2 CYS)
F BRANDON T BROSSEAU
B DAVIS T HARKINS
D HEPNER G KATULKA
D LYON R MCGEE
P MULLER P PEREGINO
A THOMPSON M CHILDERS (5 CYS)
BLDG 4600

2 DIR USARL
ATTN AMSRL WM BF H EDGE
T HAUG
BLDG 390

NO. OF COPIES	ORGANIZATION
6	DIR USARL ATTN AMSRL WM BC P PLOSTINS M BUNDY J GARNER B GUIDOS J NEWILL P PLOSTINS BLDG 390
2	DIR USARL ATTN AMSRL WM RP J BORNSTEIN C SHOEMAKER BLDG 1121
1	DIR USARL ATTN AMSRL WM MB R KASTE BLDG 4600