Copy Number **Report number :** MDC-00K9115

MD-83

ASA 261 HS Jackscrew Torque Tube Static Load Test

Revision date April 2002 **Revision letter** F

Issue date July 2000 **Contract number**

Prepared by [redacted] / [redacted]
Sudhir Sheth / Timothy Young

Reviewed by [redacted] [redacted]
Kenneth Umeda / Kenneth Orpitelli

Approved by [redacted] 5/22/00 [redacted] 5/22/00
Jerry Smets / Phong Le

MCDONNELL DOUGLAS

INDEX OF PAGE CHANGES

REVISION LETTER	PAGES AFFECTED REVISED	PAGES AFFECTED ADDED	PAGES AFFECTED REMOVED	REMARKS	REVISED BY	DATE	APPROVED
A	ii,iii	16 thru 22		Added test plan for Test No.2.	SS/TY	9/00	KU
B	Pages ii,iii Sections 2 and. 12.1	iv thru vi Sections 13,14,15		Added Test numbers 1 and 2 results and torque tube fracture surface analyses	SS/TY	11/17/00	KU
C	Pages iii,v,vi	Page vii Appendix A		Added Finite Element Analysis of Jackscrew Assembly	JL	1/10/01	KU
D	Pages iii,vi, ,viii	Page vii, Appendix B		Added Enhanced Finite Element Analysis of Jackscrew Assembly	DSR	10/01	KU
E	Page B-8			Added detail to description	DSR	1/02	KU
F	Pages iii,vii, ,viii	Appendix C		Added Finite Element Results For accident scenario	DSR	4/02	KU

TABLE OF CONTENTS

TABLE OF CONTENTS (Contd)

LIST OF FIGURES

LIST OF FIGURES (CONTINUED)

LIST OF FIGURES(CONTINUED)

LIST OF FIGURES(CONTINUED)

LIST OF TABLES

APPENDIX C

FINITE ELEMENT MODEL RESULTS FOR PREDICTED ACCIDENT SCENARIO

JACKSCREW ASSEMBLY FINITE ELEMENT MODEL

C.1. INTRODUCTION

The finite element model reported in Appendix B was revised to predict the load capability of the jackscrew assembly with the ACME nut and the stop collar aligned consistent with the marks on the recovered components from ASA 261. Additionally, the material properties of the titanium torque tube have been changed to reflect those obtained from the actual material of the accident component.

The orientation of the ACME nut is shown in Figure C1, (see Figure 8 of this report). The NTSB early in the investigation developed this orientation. The nut orientation is shown in outline over the picture of the accident stop collar.

The model elements of the ACME nut was rotated 55.5 degrees from the position reported in Appendix B to align with the position shown in Figure C1.

The model torque tube material properties have been adjusted to match those obtained from tests on a sample exercised from the accident torque tube. The following table notes these properties:

Engineering			
strain	**stress**	**true stress**	**Eln plast**
0	0	0	0.000
0.0081	130000	131056	0.000
0.0110	152000	153672	0.001
0.0160	156000	158496	0.006
0.0276	158000	162359	0.017
0.0401	160000	166420	0.029
0.0577	162000	171345	0.045
0.0815	163000	176286	0.067
0.0953	162000	177439	0.080
0.1041	160000	176653	0.088
0.1229	154000	172925	0.105
0.1379	138000	157036	0.119
0.1630	96000	111650	0.144

Table C.1 Ti-6Al-4V Material Input for accident configuration

JACKSCREW ASSEMBLY FINITE ELEMENT MODEL

C.1. INTRODUCTION (continued)

The model elements near the contact area between the ACME nut and the stop collar are shown in Figures C2 through C6.

In order to reflect the actual conditions of the accident as close as possible, modifications were made to the Appendix B model configuration. In the actual part, contact between the ACME nut and the stop collar begins as only point contact. As the load increases, the interface surface of the stop collar begins to locally yield in bearing and the points become small “pads”. As the stop collar continues to locally yield, the bearing area increases until the bearing stress drops to a level that the aluminum stop collar “pads” can hold. The crushing of the stop collar forms bearing “pads” of sufficient size to allow higher loads to be carried. The modeling of this forming process (i.e. – creating the bearing “pads”) is a difficult challenge using the Abaqus standard program. Since the formation of these “pads” for bearing are of secondary interest in transferring the load into the torque tube, the initial geometry was modified to have these “pads” as the initial condition.

The following modifications were made to the model geometry:

- The indentation marked ‘A’ in figure C1 was added to the collar.
- The geometry was modified to match the bearing surface of the ACME nut.
- The height of the ACME nut was adjusted to provide a bearing area consistent with the indentation shown in Figure C1.

JACKSCREW ASSEMBLY FINITE ELEMENT MODEL

C.1. INTRODUCTION (continued)

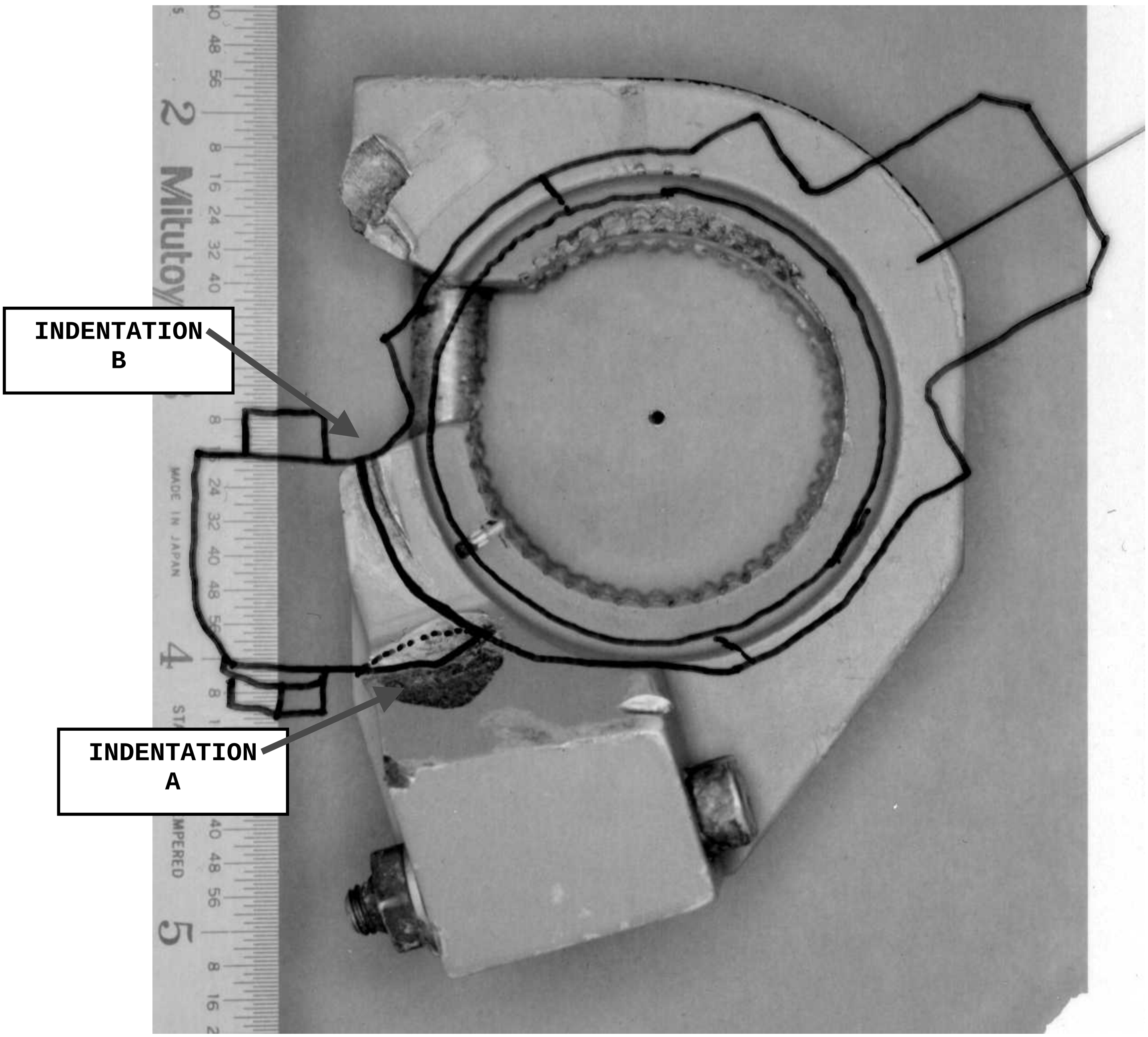

Figure C1 Orientation of ACME nut over lower stop collar (Figure 8, page 14 of MDC00K9115)

JACKSCREW ASSEMBLY FINITE ELEMENT MODEL

C.1. INTRODUCTION (continued)

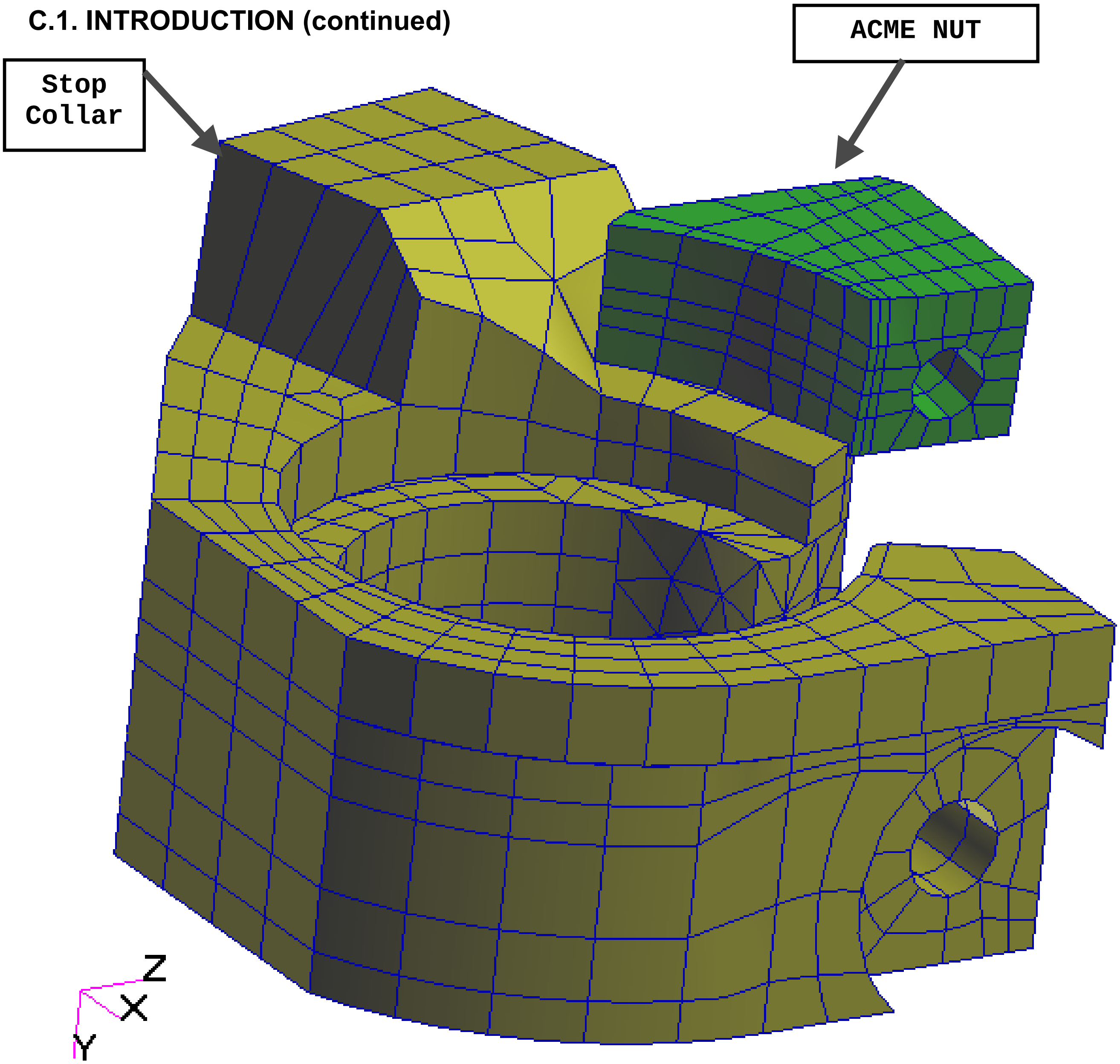

Figure C2 Elements at ACME nut to Stop Collar Contact

JACKSCREW ASSEMBLY FINITE ELEMENT MODEL

C.1. INTRODUCTION (continued)

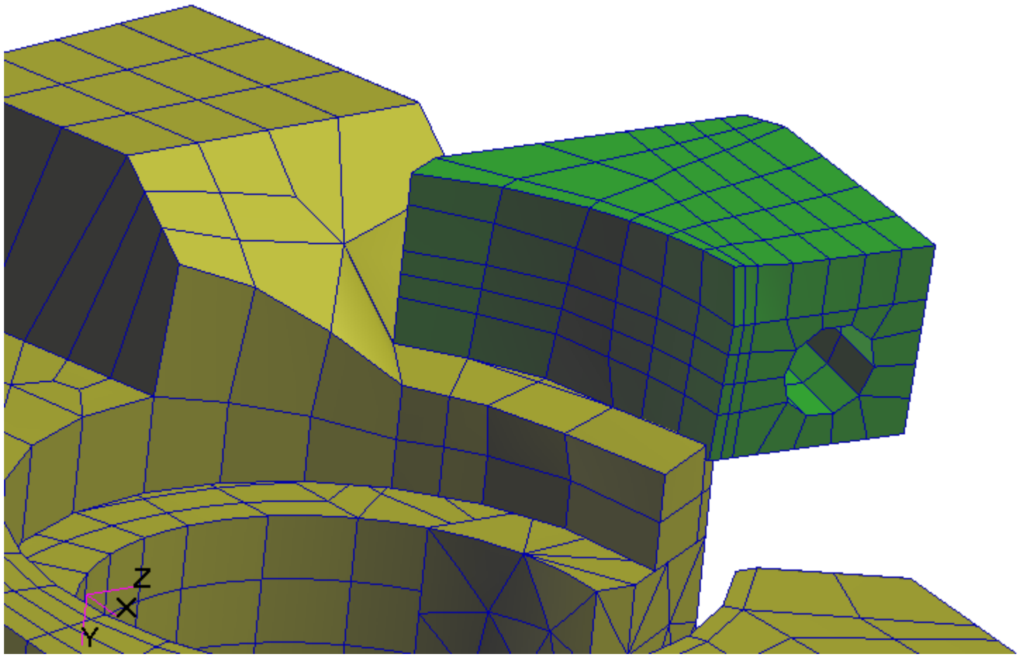

Figure C3 Elements at ACME nut to Stop Collar Contact

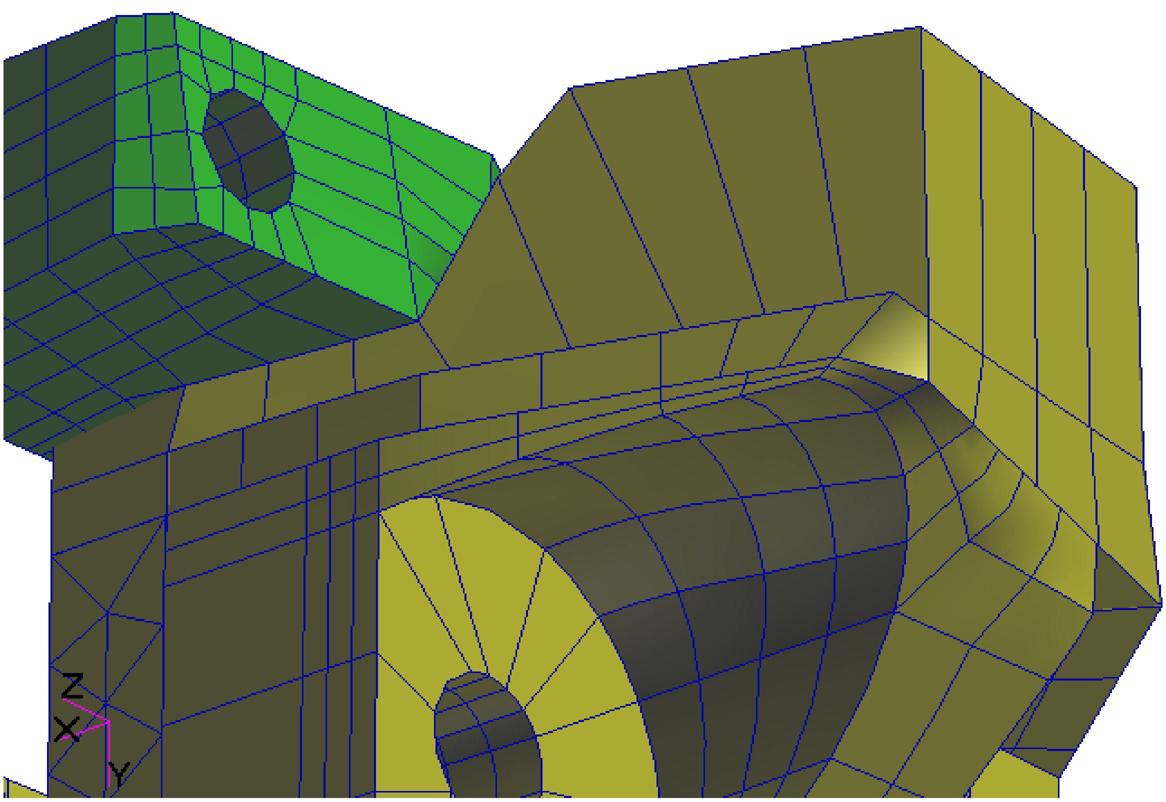

Figure C4 Elements at ACME nut to Stop Collar Contact

JACKSCREW ASSEMBLY FINITE ELEMENT MODEL

C.2 DISCUSSION

Based on the configuration noted above, the model shows excessive strain on the surfaces in contact and predicts fracture of the lower stop prior to the torque tube.

The model indicated that the rotation of the stop collar concentrates the load from the ACME nut in the region of indentation B shown in Figure C1. The metallurgical examination of the accident quill shaft indicted a low cycle fatigue crack was present at the quill shaft fracture site prior to the final fracture (Reference NTSB report no. 00-145). This model does not have that fatigue crack present, so the quill shaft has a higher strength. Without the consideration of the presence of this fatigue crack, the model shows that the stop collar fractures prior to the torque tube based on strain criterion. In order to establish torque tube to fracture consistent with the accident, the model was modified so that the contact in the region of indentation B was removed from the model.

When the model was run with contact only at the ACME nut to stop collar contact shown by indentation A in Figure C1, the model still shows stop collar fracture due to excessive strain at 24,400 pounds. The plastic strain in the quill shaft was only 0.10 in/in, which is below the fracture criteria of 0.144 in/in.

A workable solution was to convert the elements directly involved in contact to linear material properties. This change altered the pressure distribution by preventing material yielding at the contact surface. In turn, this focused the contact pressure at a single point. This increased the apparent moment arm of the applied load, thus reducing the torque tube fracture load to 24,300 pounds.

The model results with non-linear properties at the contact were extrapolated in the following way:
The model with linear properties in the contact region

P= 23,200 pounds e_{pl}=0.10
P=24,300 pounds e_{pl} = 0.144
P= 1,100 pounds is delta load

Model with non-linear properties in the contact region

P=24,400 pounds e_{pl} = 0.10
For e_{pl} = .144 P=24,400 + 1,100 = 25,500 pounds

The linear elements at the contact area are shown in Figures C5 and C6.

JACKSCREW ASSEMBLY FINITE ELEMENT MODEL

C.2 DISCUSSION (continued)

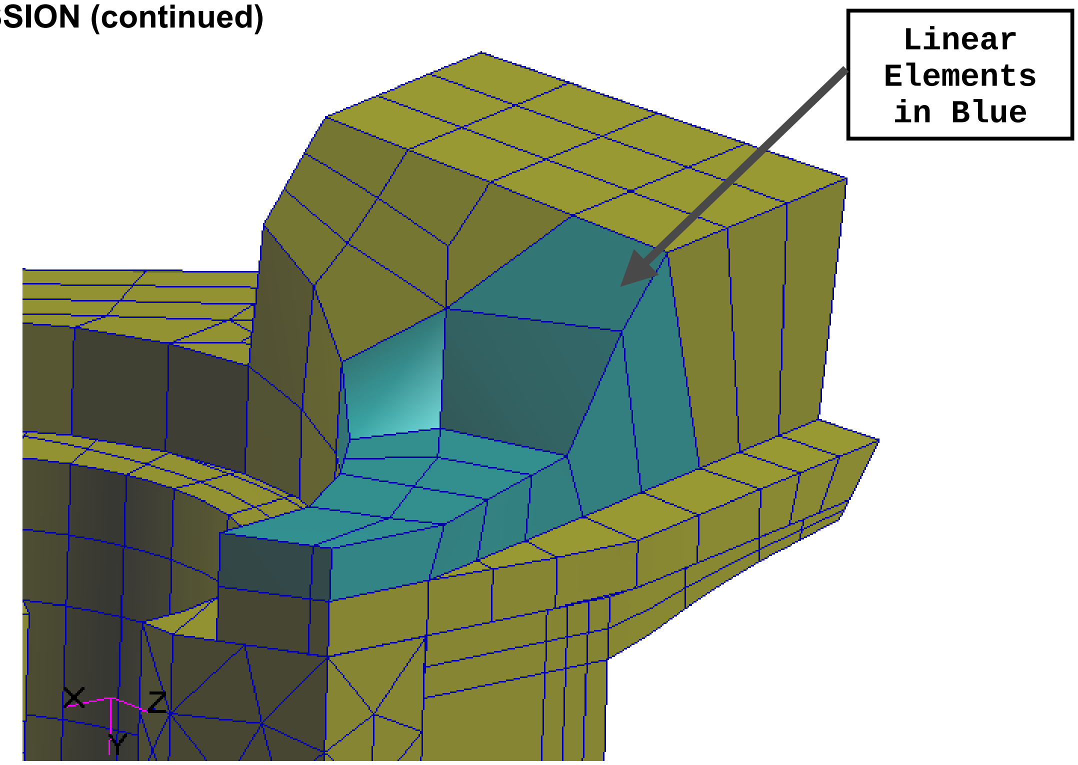

Figure C5 Stop Collar Elements at ACME nut to Stop Collar Contact

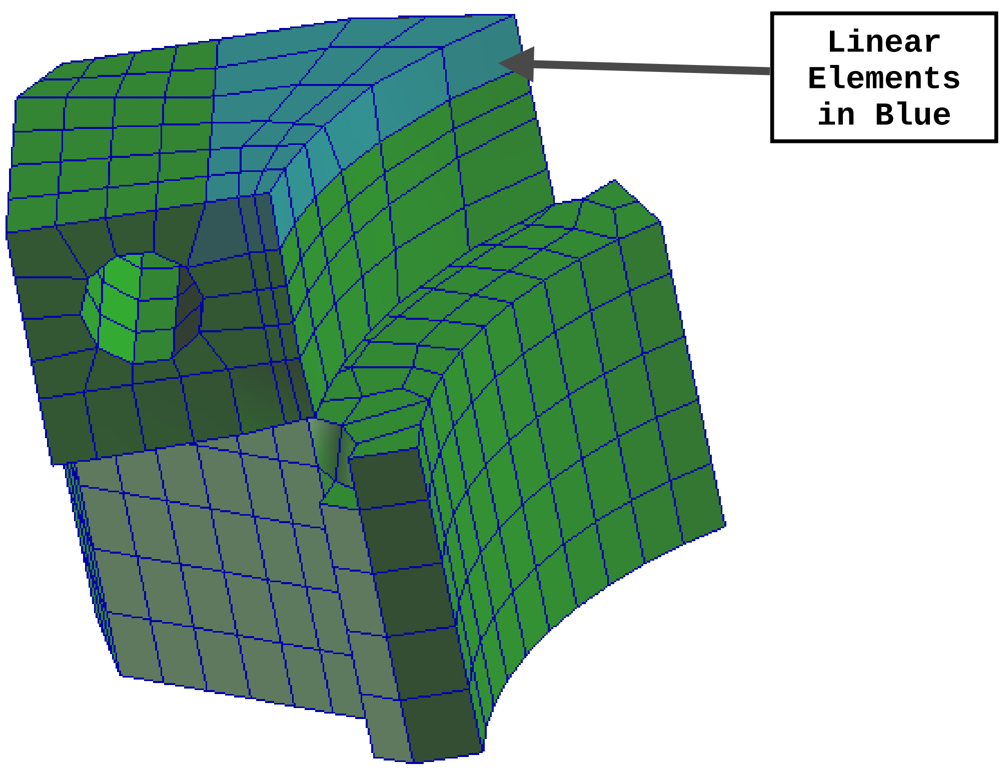

Figure C6 ACME Nut Elements at ACME nut to Stop Collar Contact

JACKSCREW ASSEMBLY FINITE ELEMENT MODEL

C.3 Accident Scenario Configuration Results

The model was run with increasing downward displacement on the ACME nut until the fracture criterion was met. The fracture criterion was an equivalent plastic strain in the thread submodel of 0.144, which is equivalent to 0.163 in/in engineering strain. A coefficient of friction of 0.15 that was considered a typical value for lubricated surfaces was used.

Fracture load accident scenario orientation:

Coefficient of Friction = 0.15

Linear elements at contact — Fracture load = 24,300 pounds

Non-linear elements at contact (extrapolated see pg C-7) — Fracture load = 25,500 pounds

Test orientation (reference page B-58)

Coefficient of Friction = 0.15 — Fracture load = 24,000 pounds

The model files for this analysis are archived on:

Boeing archive system **ufs2.ca.boeing.com**

In directory:

/user8/longbch-struct/c388057/asa261/report-k9115-appc

C.3 Accident Scenario Configuration Results (continued)

Overall model

The deformed shape for the overall model is shown in Figures C7 to C10.

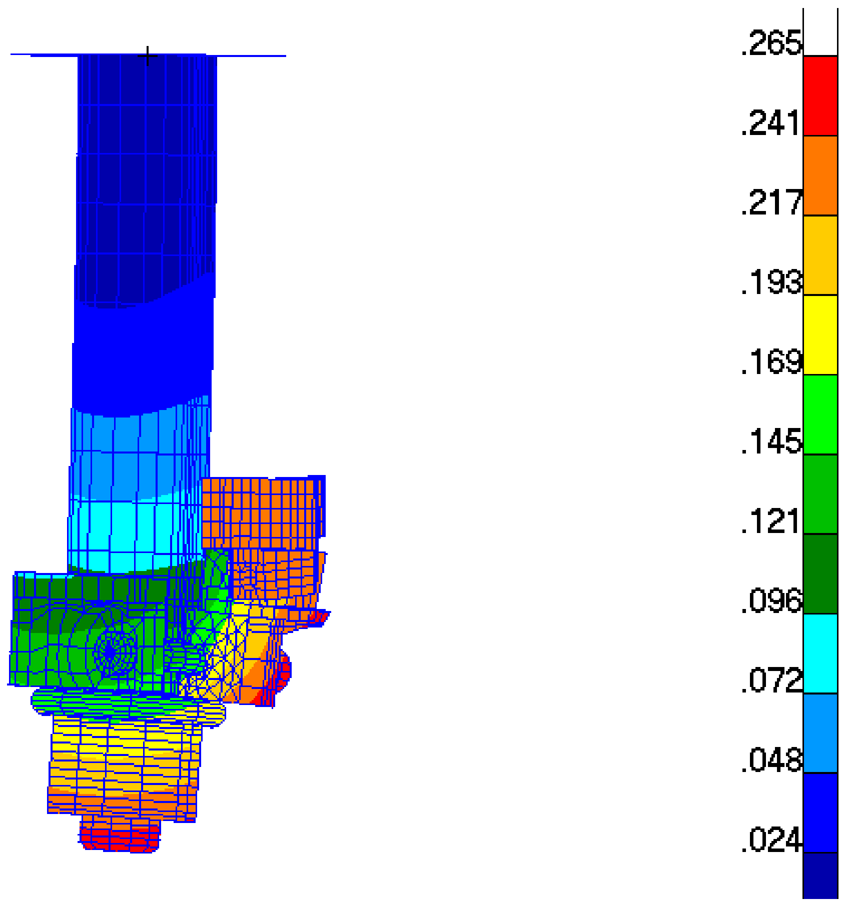

Figure C7 Overall Model Deformed shape

C.3 Accident Scenario Configuration Results (continued)
Overall model

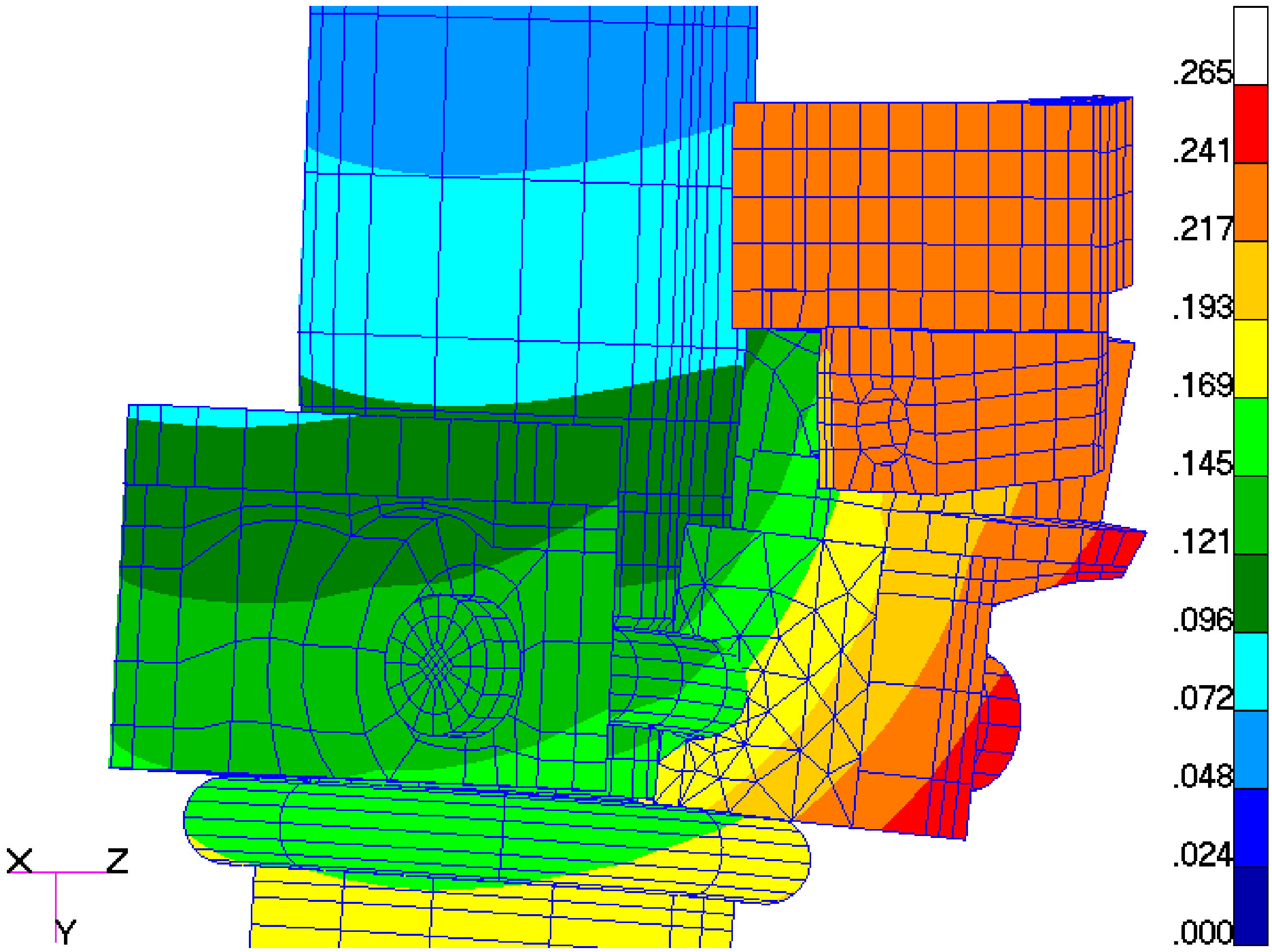

Figure C8 Model Deformation – View 1

C.3 Accident Scenario Configuration Results (continued)

Overall model

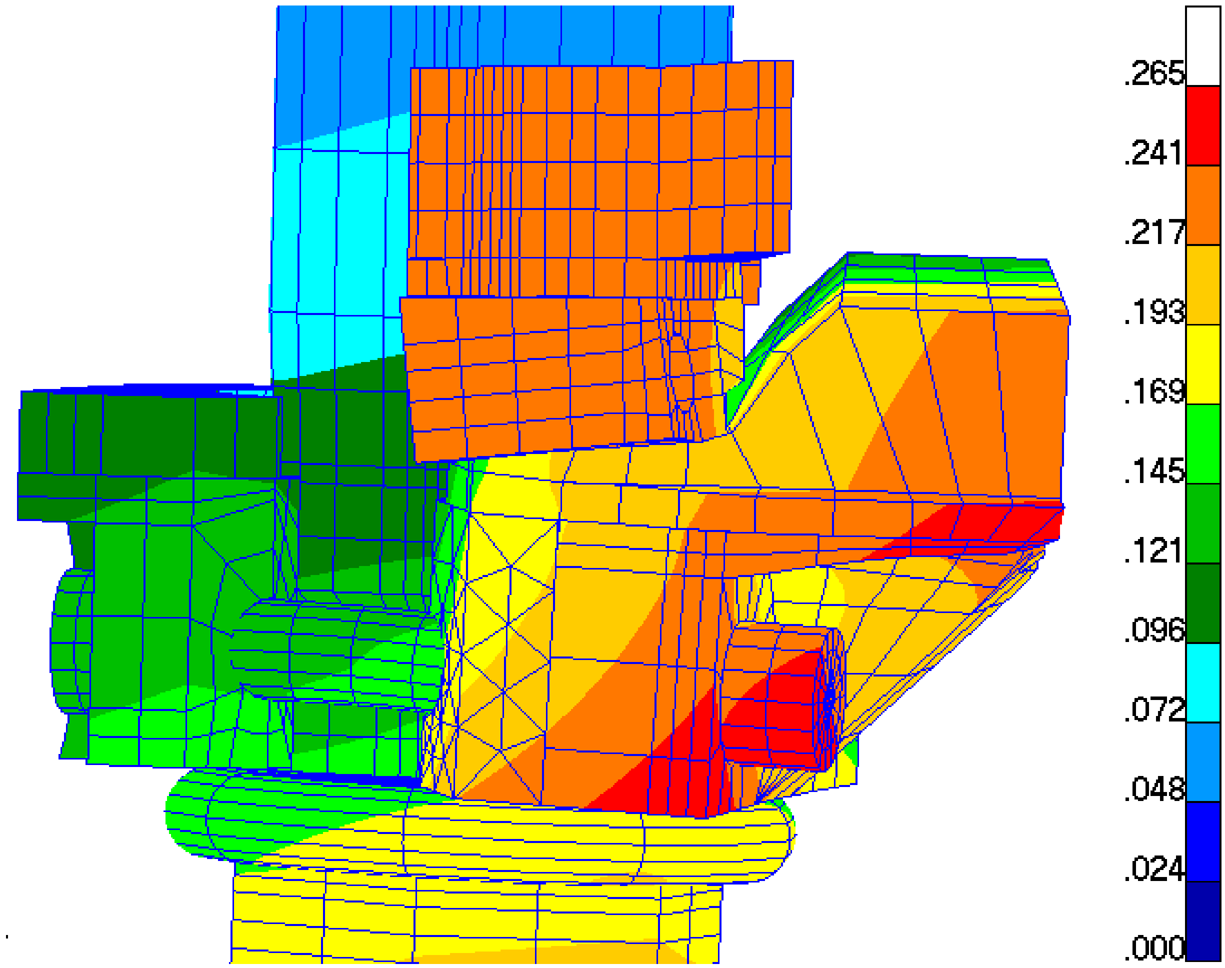

Figure C9 Model Deformation – View 2

C.3 Accident Configuration Results (continued)

Overall model

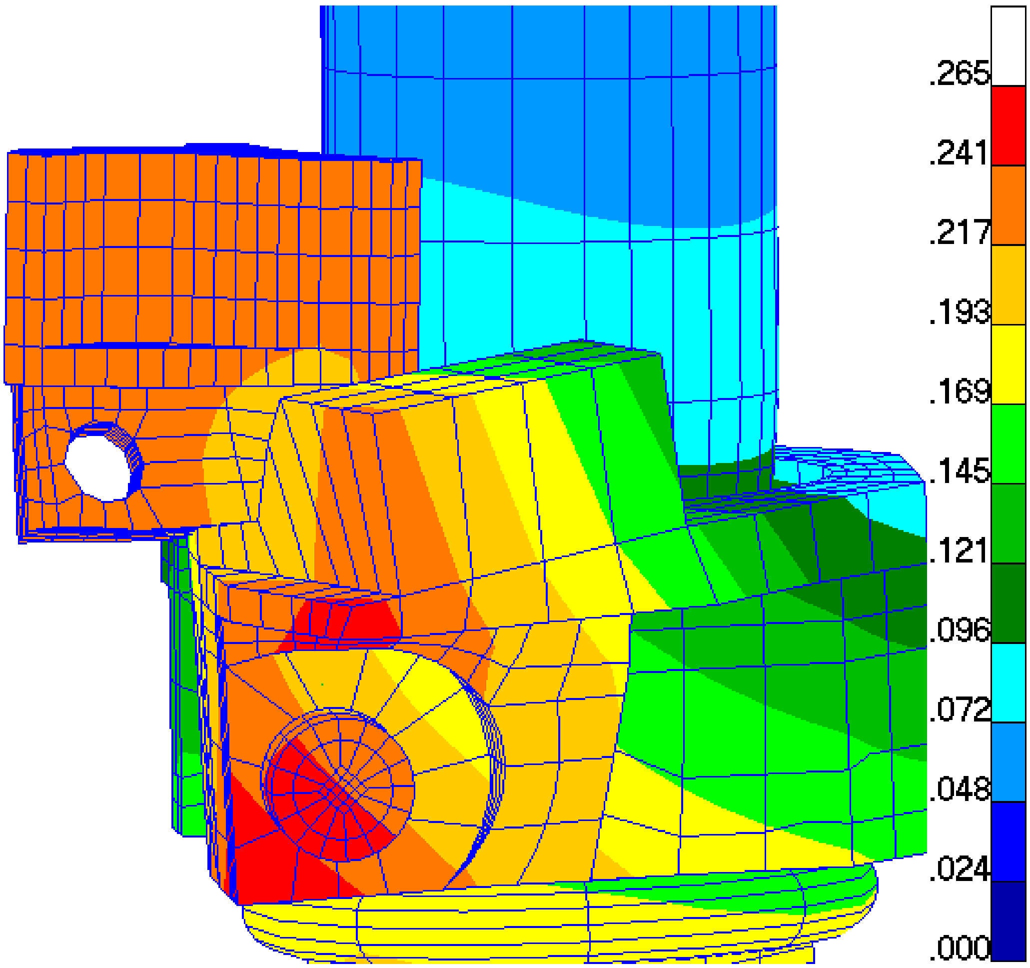

Figure C10 Model Deformation – View 3

C.3 Accident Scenario Configuration Results (continued)
ACME NUT and Stop Collar Contact Area

The deformation of the ACME nut and stop collar are shown in Figures C11 to C13. The overlap due to the removal of the contact at location A is shown in Figure C8. The overlap matches fairly well with the depth and size of the compressed area shown as Indentation B in Figure C1.

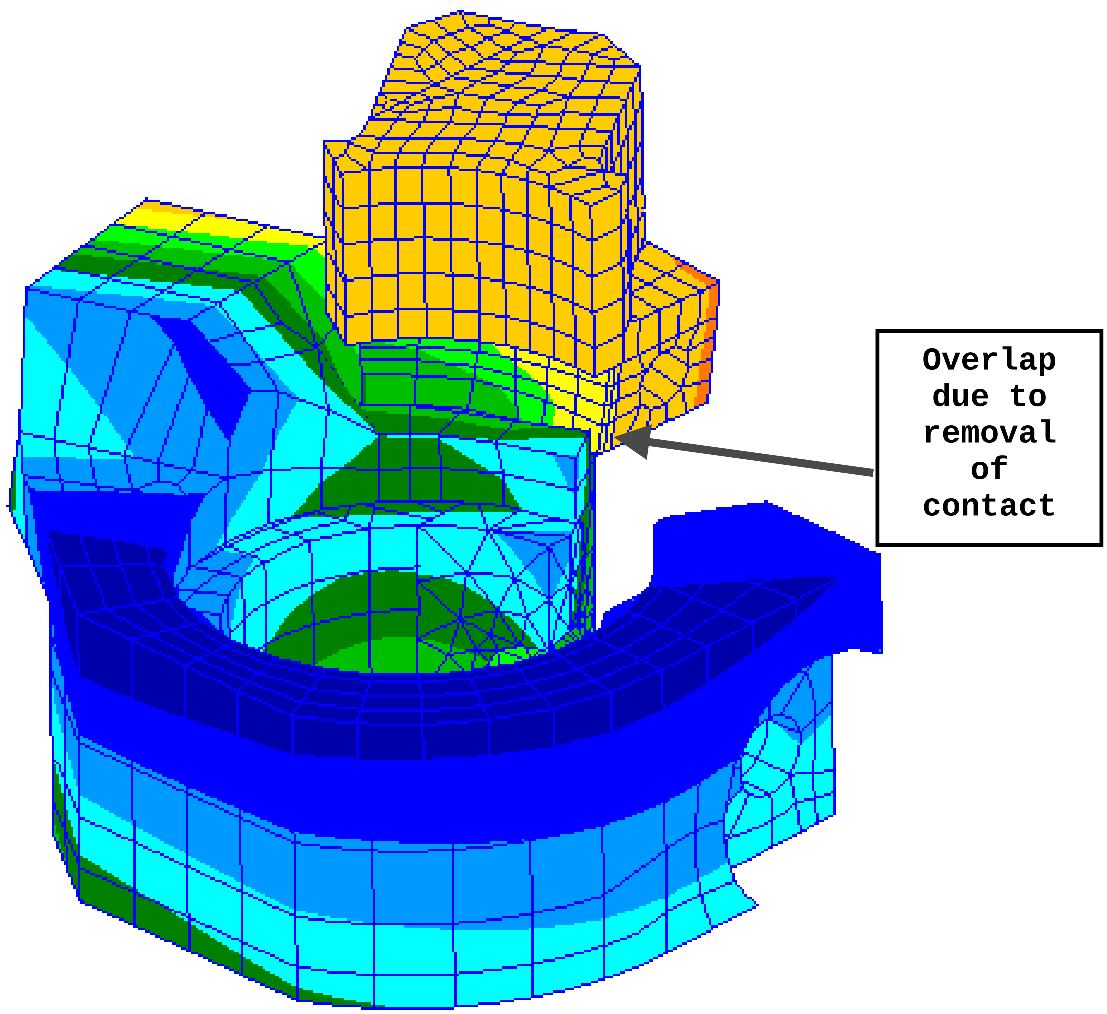

Figure C11 ACME Nut and Stop Collar Deformation

C.3 Accident Scenario Configuration Results (continued)
ACME NUT and Stop Collar Contact Area

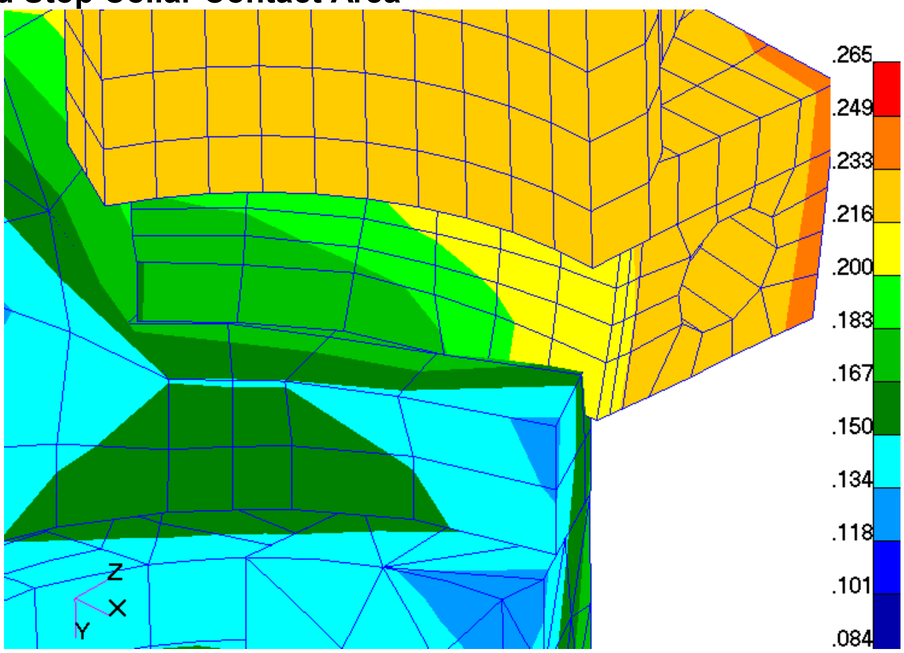

Figure C12 ACME Nut and Stop Collar Deformation

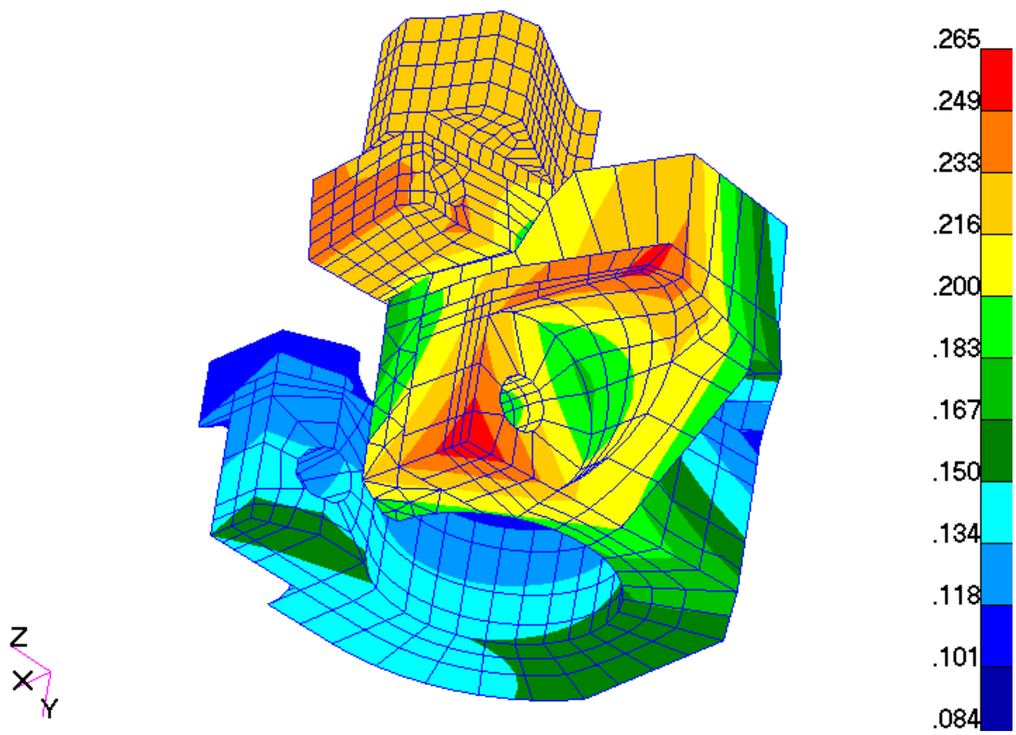

Figure C13 ACME Nut and Stop Collar Deformation

C.3 Accident Scenario Configuration Results (continued)
ACME NUT

The equivalent plastic strain in the ACME nut is shown in Figure C14. The significant plastic deformation in the nut is as expected for the small contact area involved. High strain does not occur at the contact surface because the elements were made linear (no plastic deformation) to get a solution. The contact pressure between the ACME nut and the Stop Collar is shown in Figure C15. The extremely high contact pressure is due to the linear material properties used for the contacting elements.

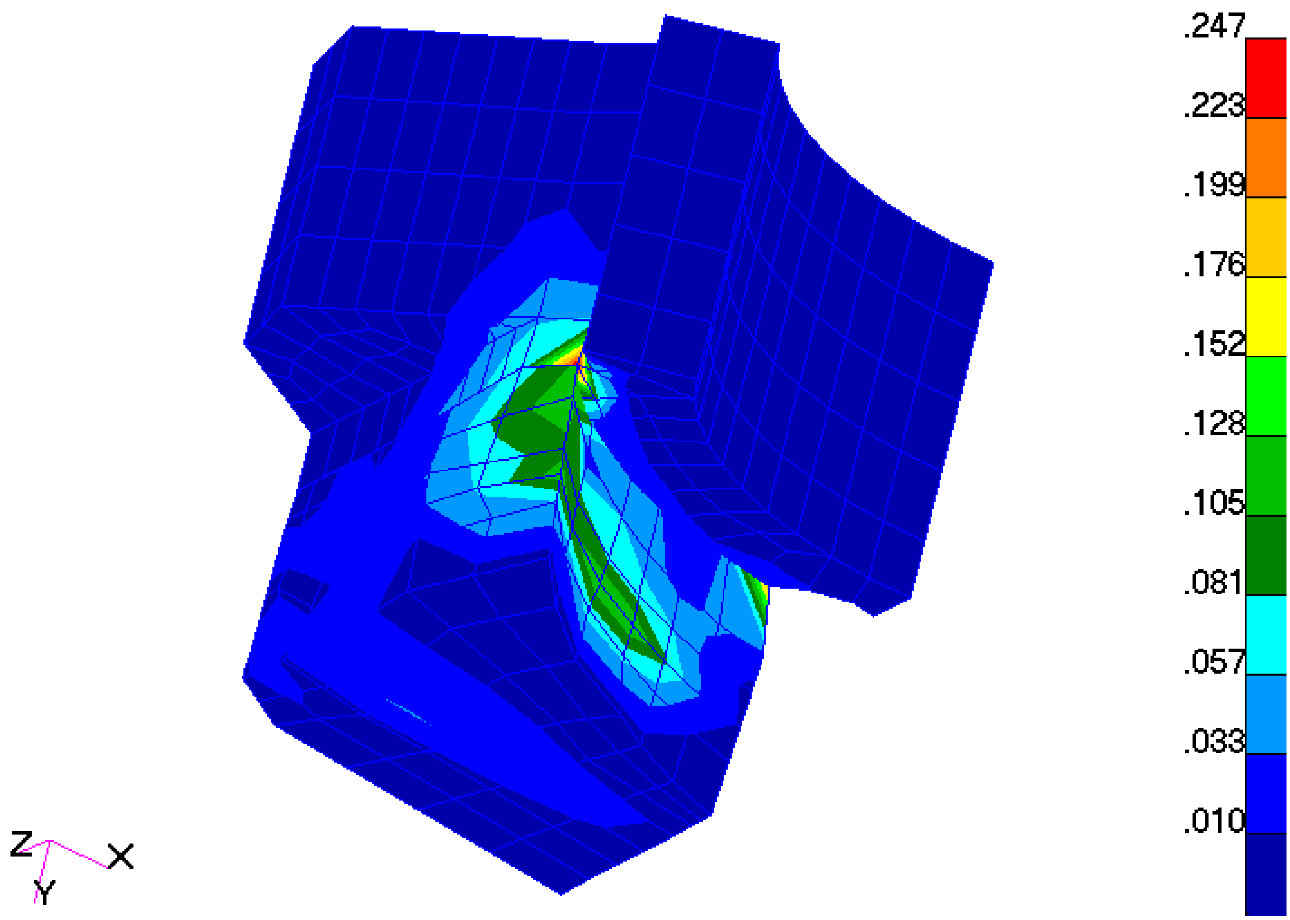

Figure C14 ACME Nut Equivalent Plastic Strain

C.3 Accident Scenario Configuration Results (continued)
ACME NUT and Stop Collar Contact Area (continued)

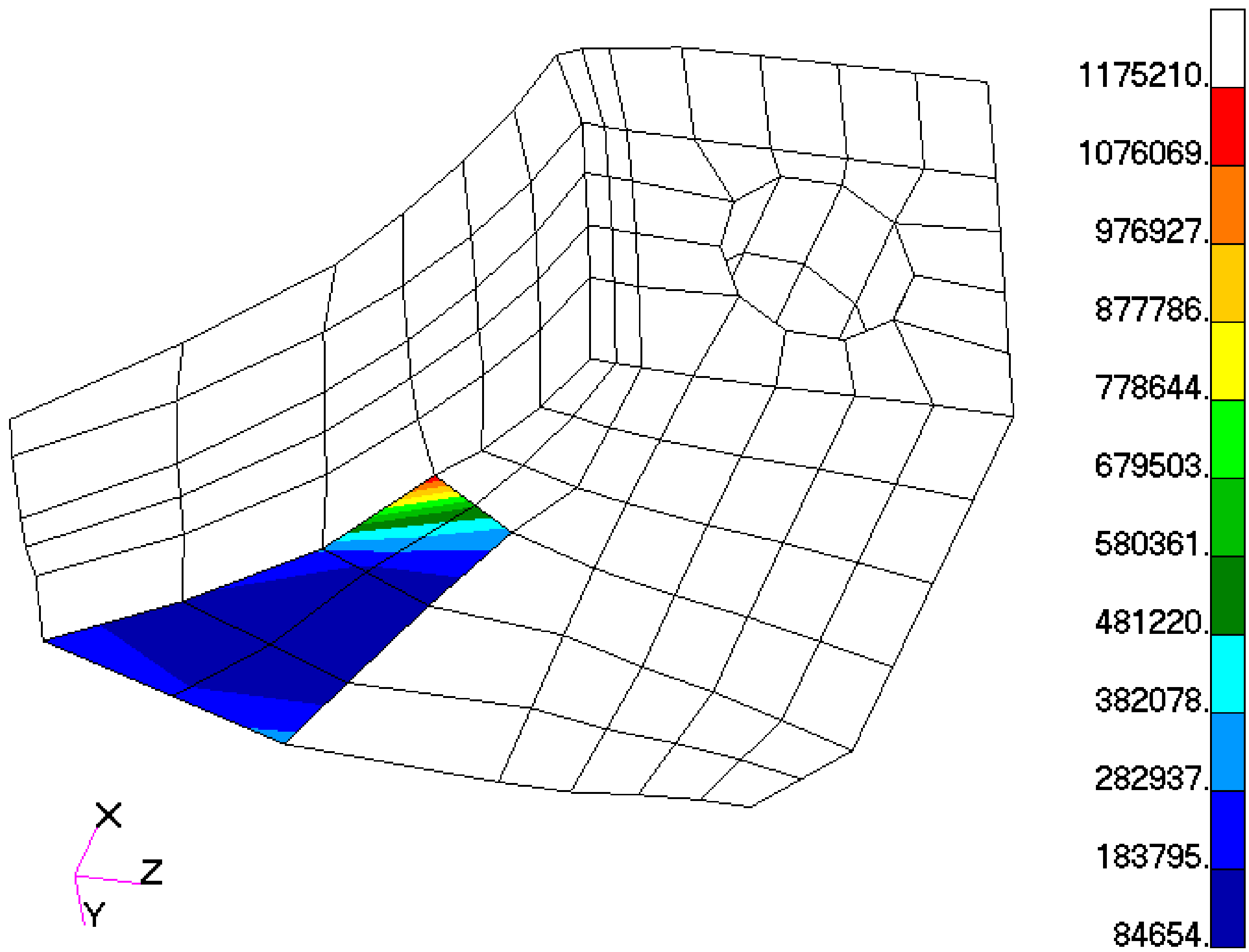

Figure C15 ACME Nut Contact Pressure (psi) with Stop Collar

C.3 Accident Scenario Configuration Results (continued)

Stop Collar

The equivalent plastic strain of the stop collar is shown in Figures C16. The strains are consistent with the applied loading. A high strain occurs under the contact region. The contact region itself is not yielded because the nonlinear properties were not used in the contact zone.

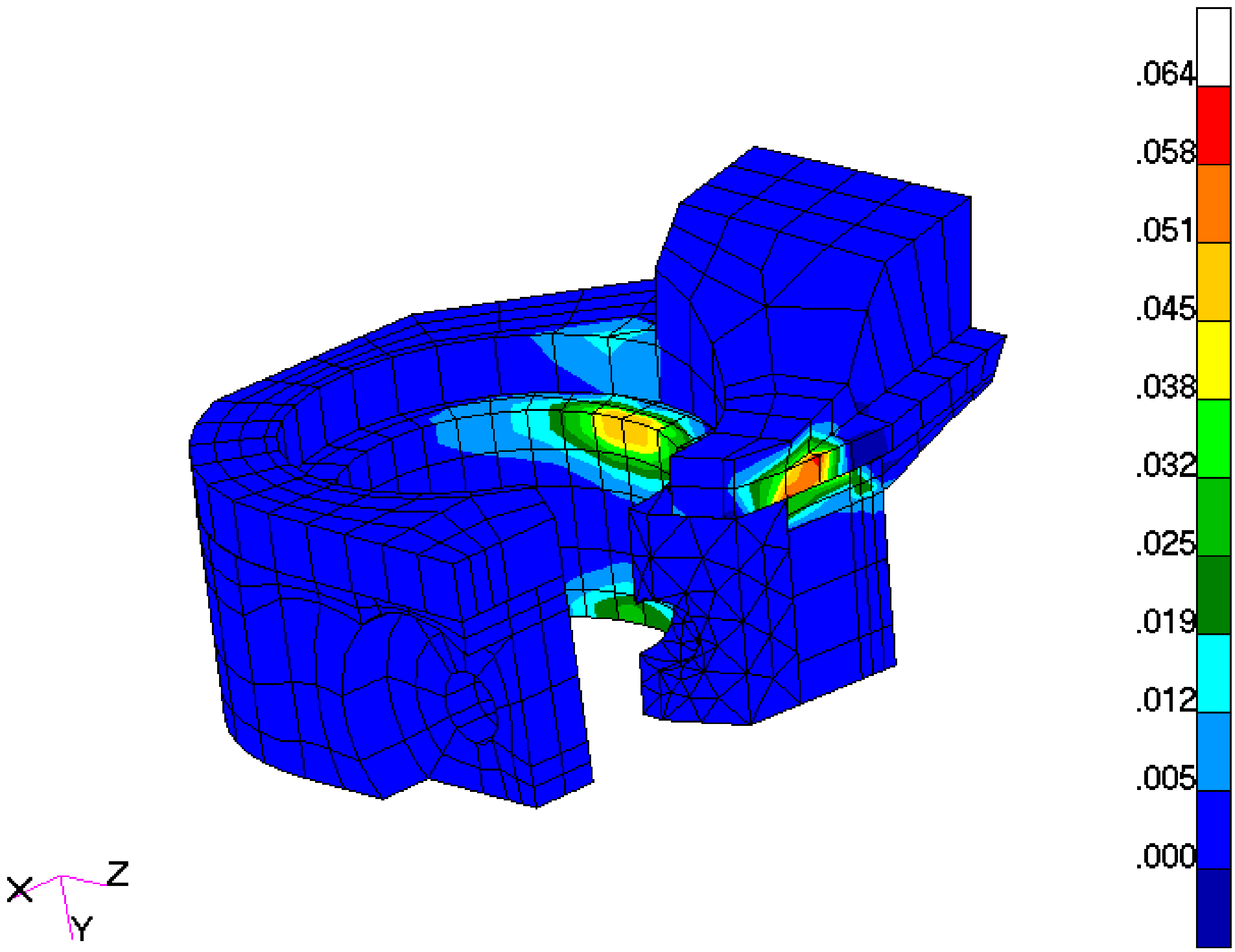

Figure C16 Stop Collar Equivalent Plastic Strain

C.3 Accident Scenario Configuration Results (continued)

Torque Tube

The equivalent plastic strain of the torque tube is shown in Figure C17. As expected, the peak strain occurs in the first thread.

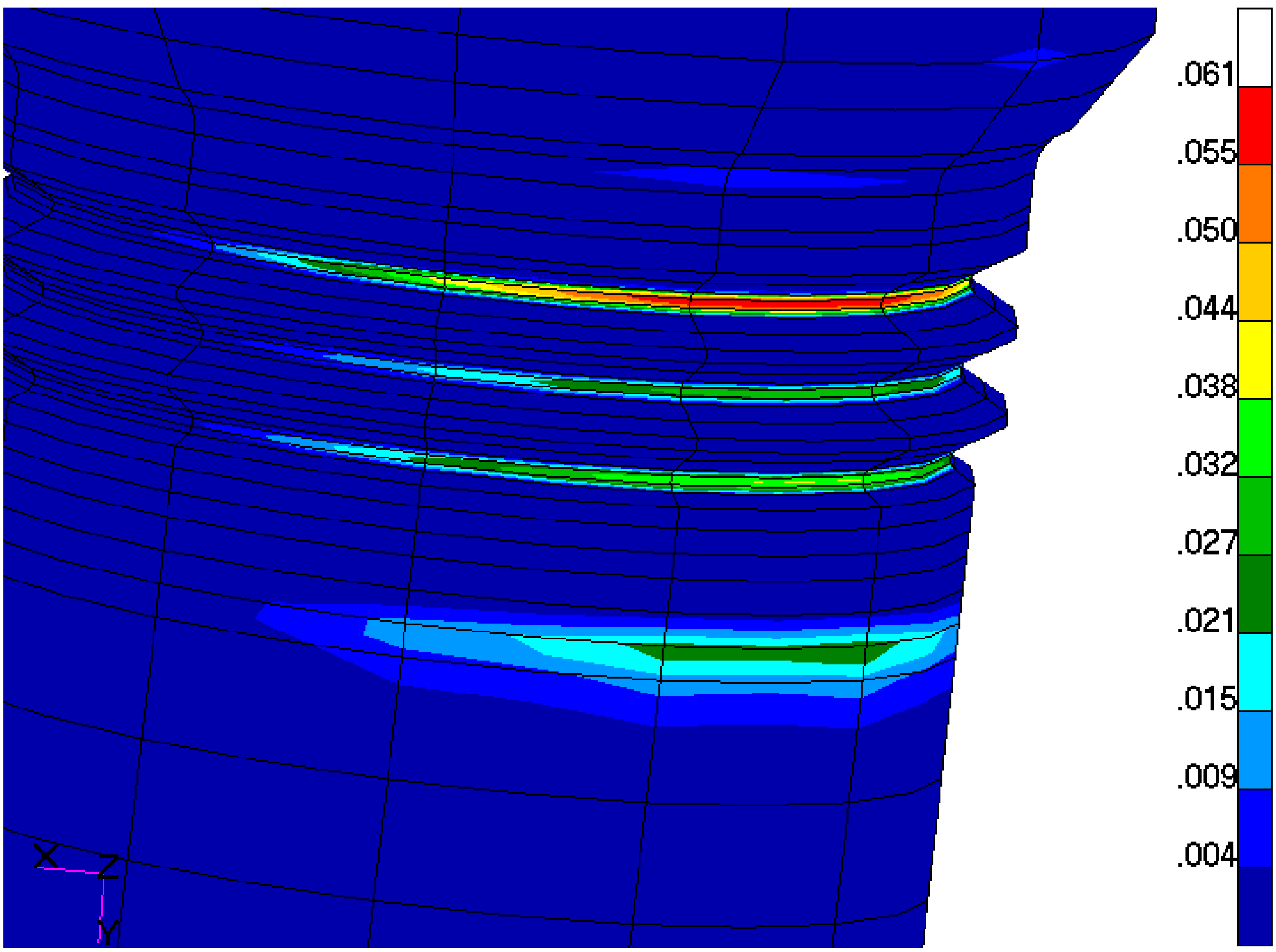

Figure C17 Torque Tube Equivalent Plastic Strain

C.3 Accident Scenario Configuration Results (continued)

Torque Tube Submodel

The equivalent plastic strain of the torque tube thread submodel is shown in Figures C18. The load at which the equivalent plastic strain was equal to 0.144 in/in determined the fracture load.

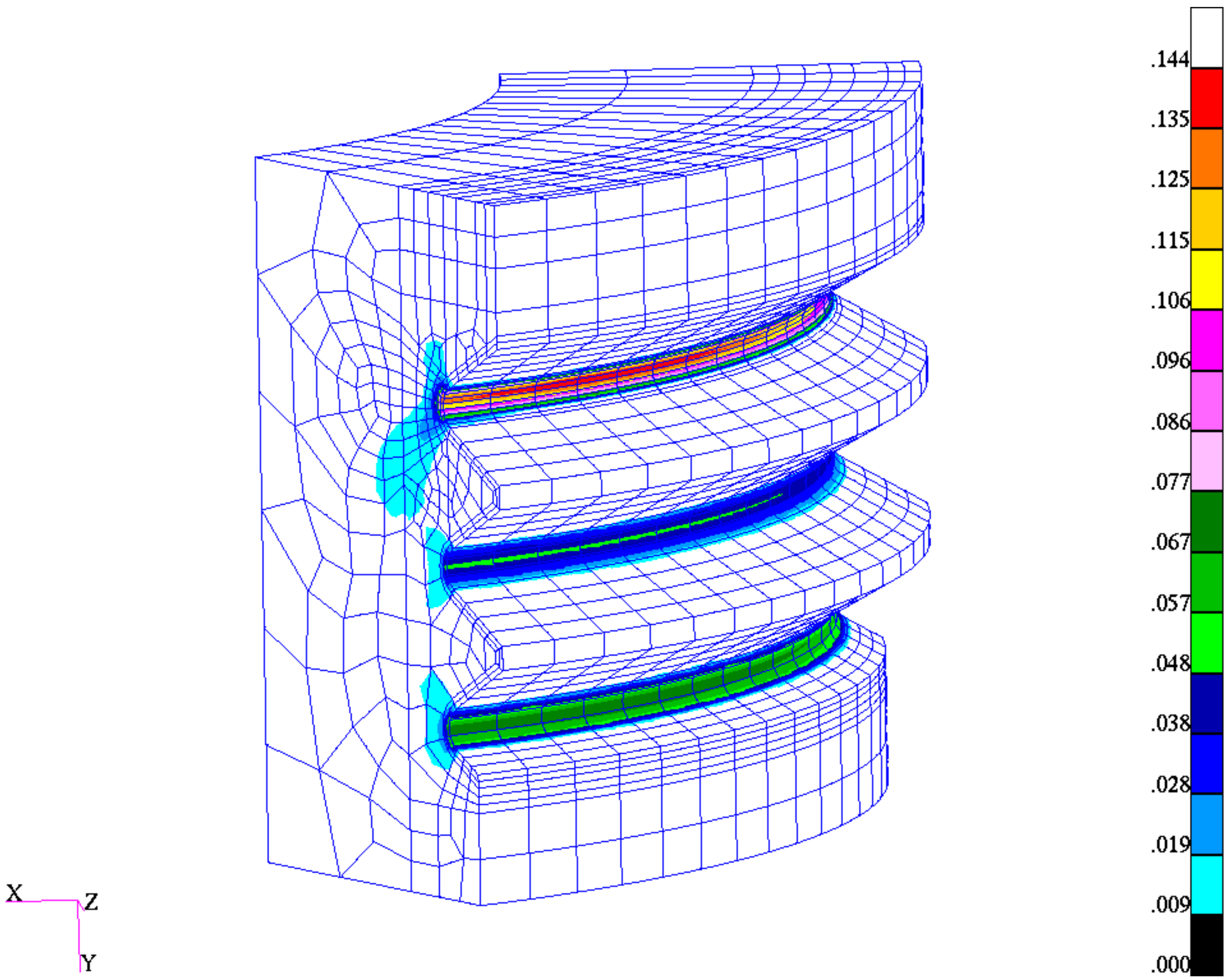

Figure C18 Torque Tube Submodel Equivalent Plastic Strain

C.3 Accident Scenario Configuration Results (continued)

MODEL SUMMARY

The original MD83 jackscrew model showed a -6% error with respect to the test. The purpose of the analysis presented here was to model a configuration considered by the NTSB to be consistent with the recovered parts from the accident airplane. Attempts to directly simulate this contact between the ACME nut and the stop collar results in fracture of the stop collar prior to the fracture of the torque tube. This fracture scenario in the model can be attributed to the absence of low cycle fatigue damage observed in the accident torque tube fracture face. The presence of this crack prior to the application of the high loads considered for this simulation would significantly reduce the structural capability of the torque tube. To mitigate the effects of the presence of the fatigue damage, the elements directly involved in contact were not allowed to plastically deform. With this consideration, the model predicted fracture of the first thread of the torque tube at 24,300 pounds. However, this configuration reduced the area of contact to a single point. If an extrapolation of these results was applied to the model results with yielding in the contact area, the predicted fracture load increases 5% to 25,500 pounds.