Analysis of MASM-0002c.ipt

Author:	Bobby
Analysis Created:	Thursday, March 22, 2007 4:46:43 AM
Analysis Last Modified:	Thursday, March 22, 2007 4:46:43 AM
Report Created:	Thursday, March 22, 2007 4:47:01 AM
Database:	*C:\Documents and Settings\Bobby\Desktop\BSME\2007 Spring\Sr.Design\inventor_files\real_deal_v10\MASM-0002c.ipa*
Software:	Autodesk Inventor Professional 11.0 ANSYS Technology

Autodesk ANSYS

Introduction

Autodesk Inventor Professional Stress Analysis was used to simulate the behavior of a mechanical part under structural loading conditions. ANSYS technology generated the results presented in this report.

Do not accept or reject a design based solely on the data presented in this report. Evaluate designs by considering this information in conjunction with experimental test data and the practical experience of design engineers and analysts. A quality approach to engineering design usually mandates physical testing as the final means of validating structural integrity to a measured precision.

Additional information on AIP Stress Analysis and ANSYS products for Autodesk Inventor is available at http://www.ansys.com/autodesk.

Geometry and Mesh

The Relevance setting listed below controlled the fineness of the mesh used in this analysis. For reference, a setting of -100 produces a coarse mesh, fast solutions and results that may include significant uncertainty. A setting of +100 generates a fine mesh, longer solution times and the least uncertainty in results. Zero is the default Relevance setting.

TABLE 1
MASM-0002c.ipt Statistics

Bounding Box Dimensions	6.75 in 0.625 in 0.375 in
Part Mass	0.1462 lbm
Part Volume	1.492 in^3
Mesh Relevance Setting	0
Nodes	3959
Elements	2204

Bounding box dimensions represent lengths in the global X, Y and Z directions.

Material Data

The following material behavior assumptions apply to this analysis:

- Linear - stress is directly proportional to strain.
- Constant - all properties temperature-independent.
- Homogeneous - properties do not change throughout the volume of the part.
- Isotropic - material properties are identical in all directions.

TABLE 2
Aluminum-6061

Young's Modulus	9.993e+006 psi
Poisson's Ratio	0.33
Mass Density	9.798e-002 lbm/in³
Tensile Yield Strength	3.989e+004 psi
Tensile Ultimate Strength	4.496e+004 psi

Loads and Constraints

The following loads and constraints act on specific regions of the part. Regions were defined by selecting surfaces, cylinders, edges or vertices.

TABLE 3
Load and Constraint Definitions

Name	Type	Magnitude	Vector
Force 1	Surface Force	12.5 lbf	0. lbf 12.5 lbf 0. lbf
Fixed Constraint 1	Surface Fixed Constraint	0. in	0. in 0. in 0. in

TABLE 4
Constraint Reactions

Name	Force	Vector	Moment	Moment Vector
Fixed Constraint 1	12.5 lbf	-6.034e-009 lbf -12.5 lbf -5.966e-009 lbf	0.6843 lbf·in	0.6843 lbf·in -6.279e-008 lbf·in 8.788e-003 lbf·in

Note: vector data corresponds to global X, Y and Z components.

Results

The table below lists all structural results generated by the analysis. The following section provides figures showing each result contoured over the surface of the part.

Safety factor was calculated by using the maximum equivalent stress failure theory for ductile materials. The stress limit was specified by the tensile yield strength of the material.

TABLE 5
Structural Results

Name	Minimum	Maximum
Equivalent Stress	1.669 psi	1081 psi
Maximum Principal Stress	-37.38 psi	711.5 psi
Minimum Principal Stress	-681.6 psi	40.42 psi
Deformation	0. in	1.199e-004 in
Safety Factor	15.	N/A

Figures

FIGURE 1
Equivalent Stress

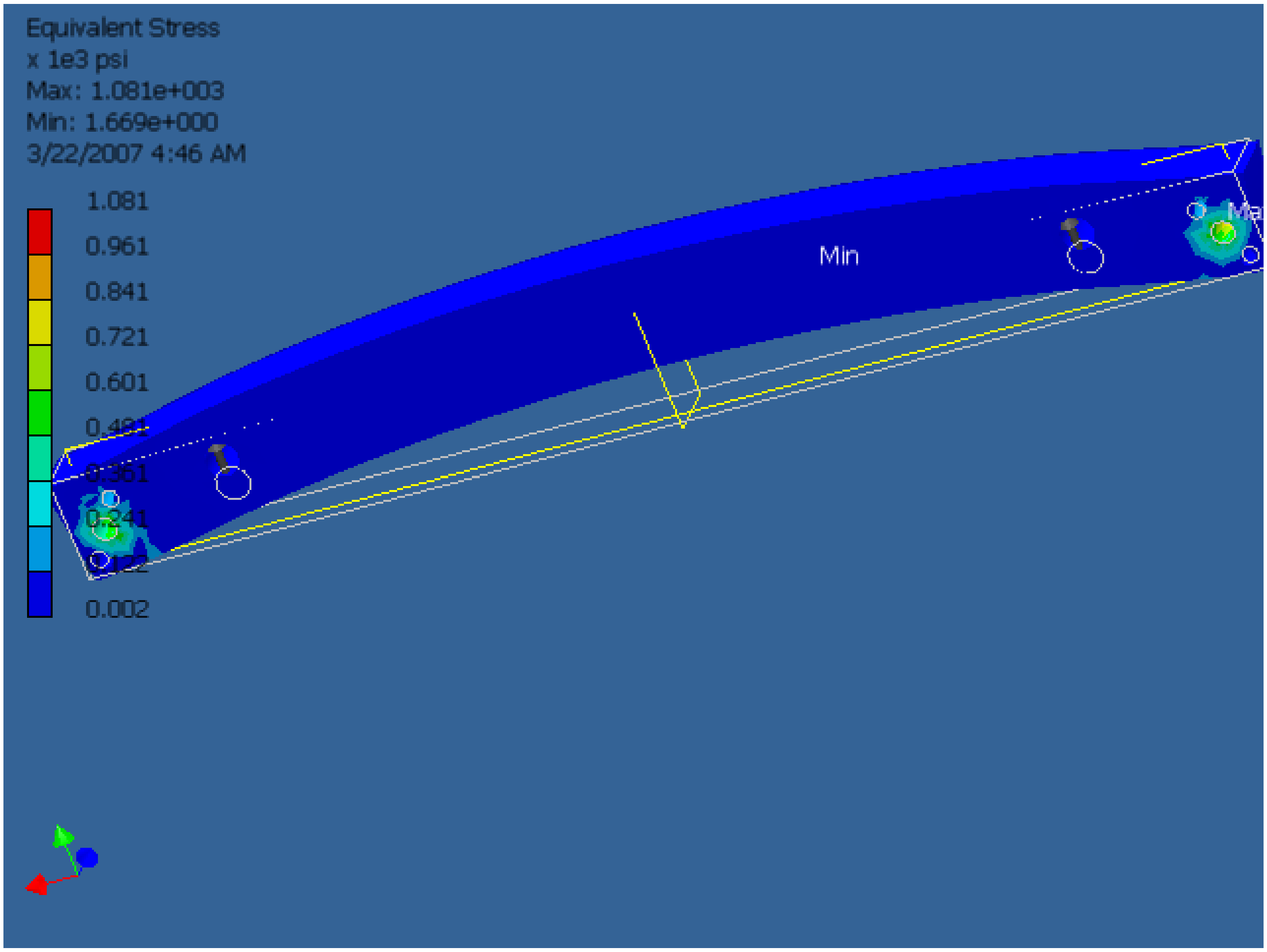

FIGURE 2
Maximum Principal Stress

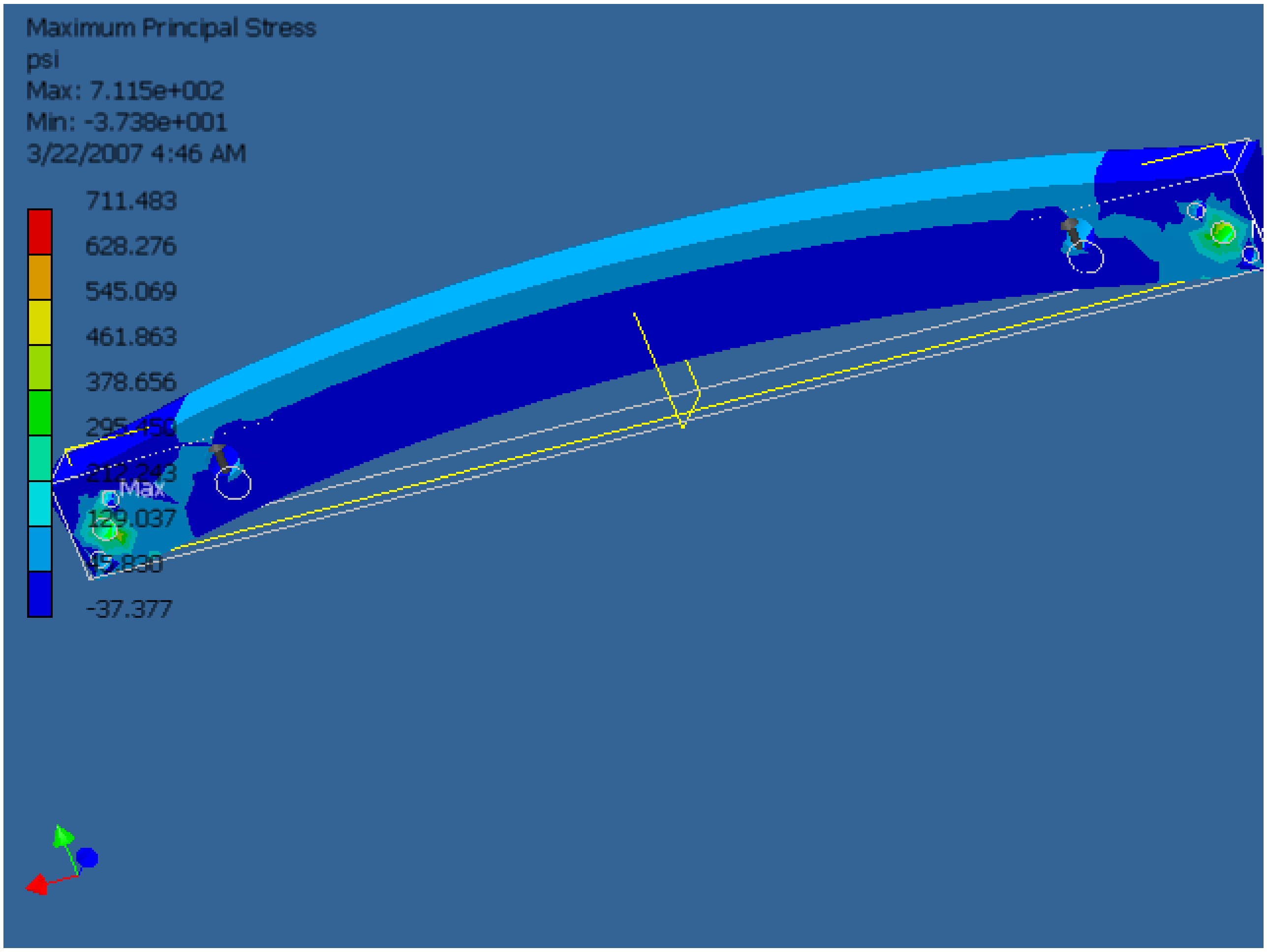

FIGURE 3
Minimum Principal Stress

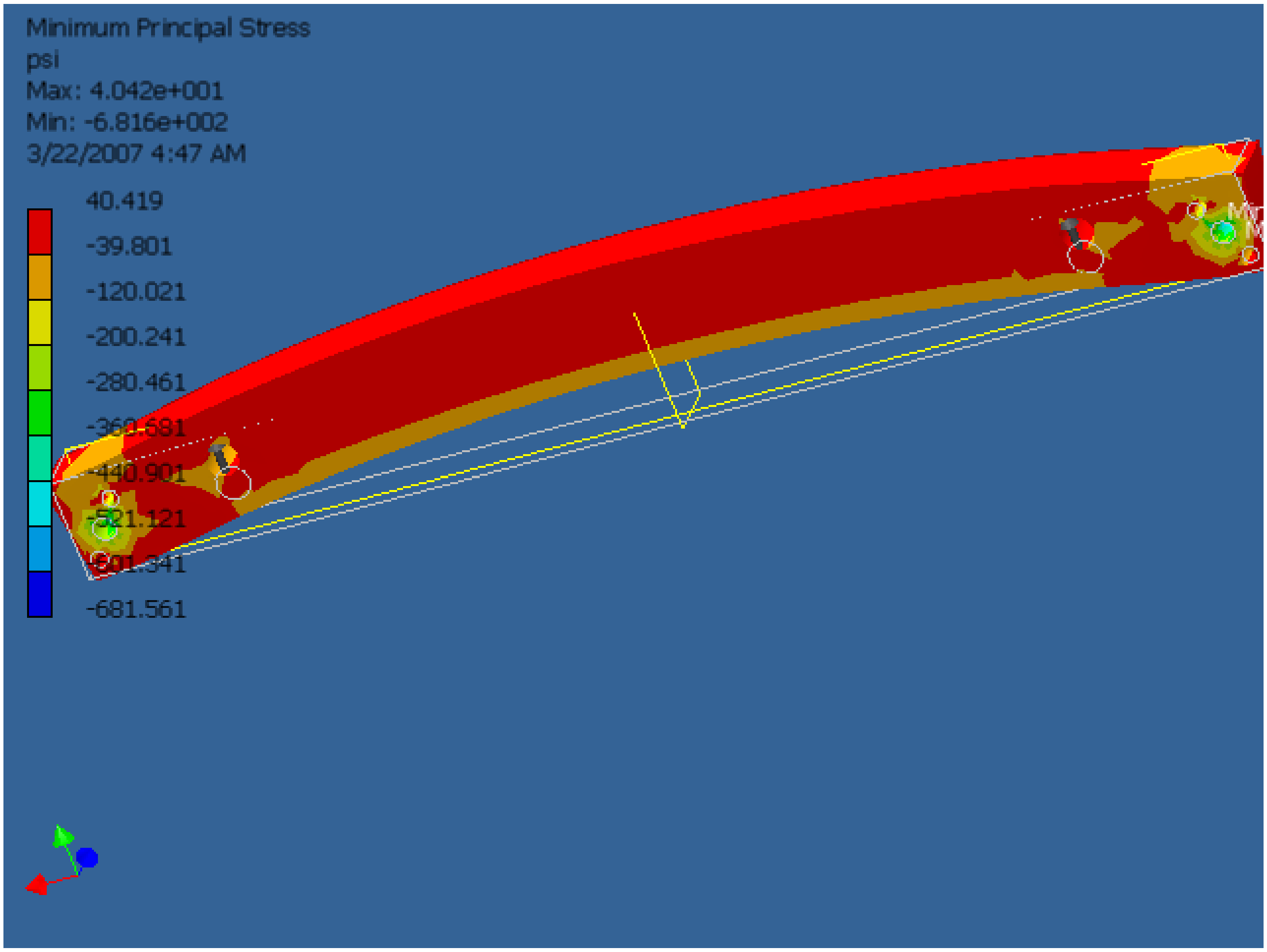

FIGURE 4
Deformation

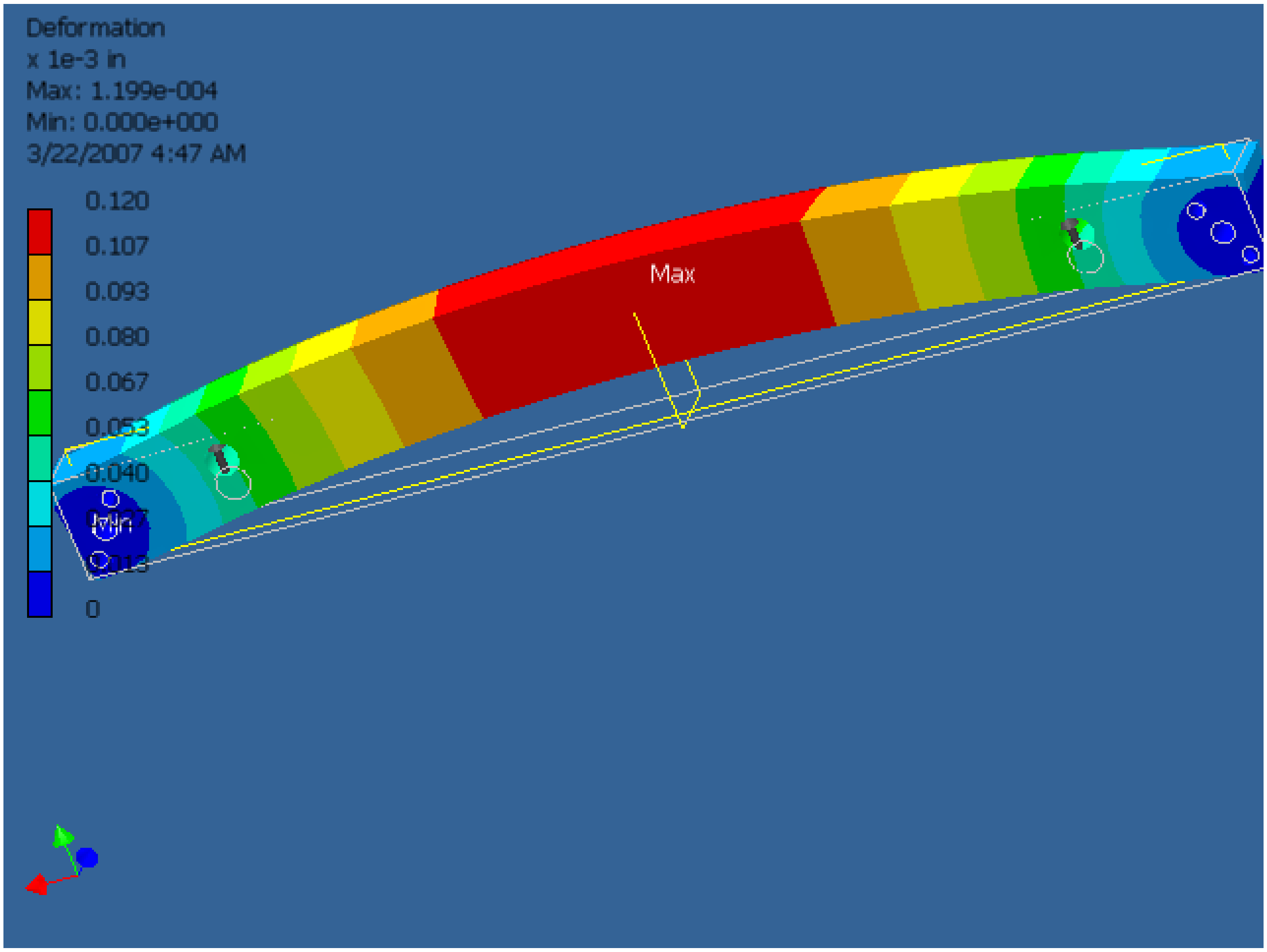

FIGURE 5
Safety Factor