Analysis of Attachment Piece

Author:	Nathan
Analysis Created:	Thursday, October 25, 2007 5:01:24 PM
Analysis Last Modified:	Thursday, October 25, 2007 5:01:24 PM
Report Created:	Thursday, October 25, 2007 5:18:23 PM
Database:	*I:\Senior Design II\Brackets\Attachment Piece.ipa*
Software:	Autodesk Inventor Professional 11.0 ANSYS Technology

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
Attachment Piece.ipt Statistics

Bounding Box Dimensions	0.7501 in 1.875 in 2.5 in
Part Mass	9.056e-002 lbm
Part Volume	0.9242 in^3
Mesh Relevance Setting	0
Nodes	3046
Elements	1535

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	Edge Force	150. lbf	150. lbf 0. lbf -1.049e-014 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	150. lbf	-150. lbf 3.05e-006 lbf -4.494e-006 lbf	228.8 lbf·in	-1.214e-006 lbf·in -187.6 lbf·in -131. 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	7.284 psi	9282 psi
Maximum Principal Stress	-1237 psi	1.102e+004 psi
Minimum Principal Stress	-9466 psi	1157 psi
Deformation	0. in	4.413e-003 in
Safety Factor	4.297	N/A

Figures

FIGURE 1
Equivalent Stress

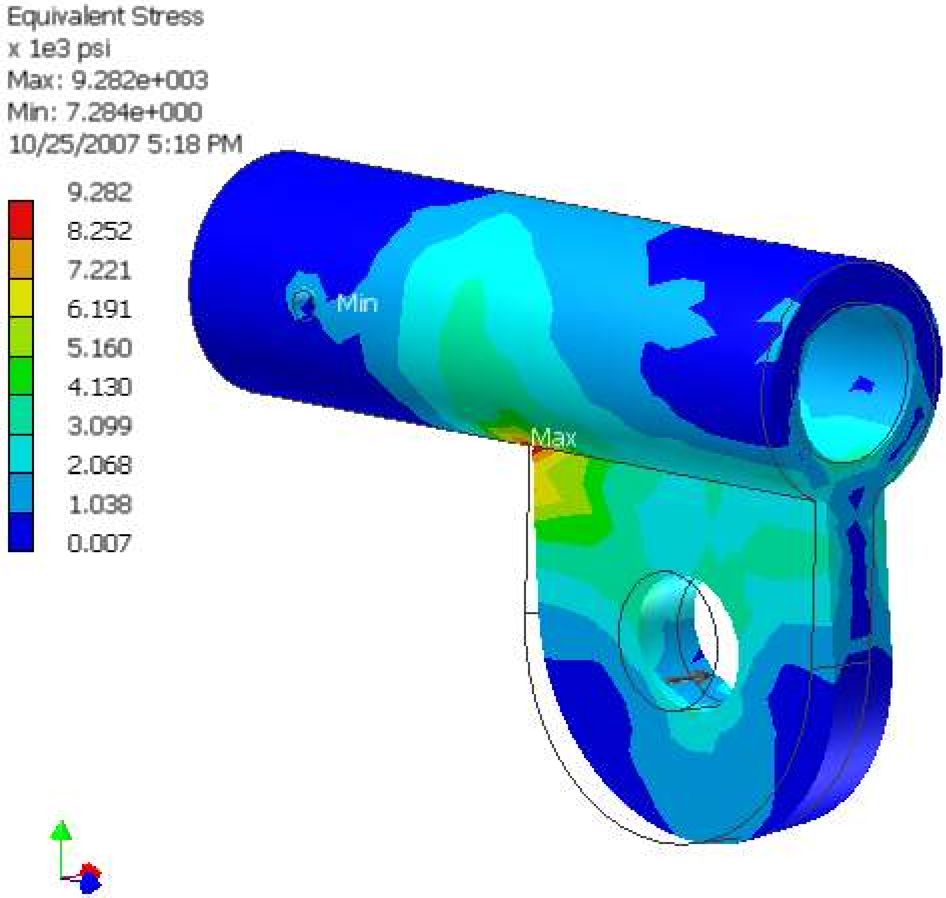


FIGURE 2
Maximum Principal Stress

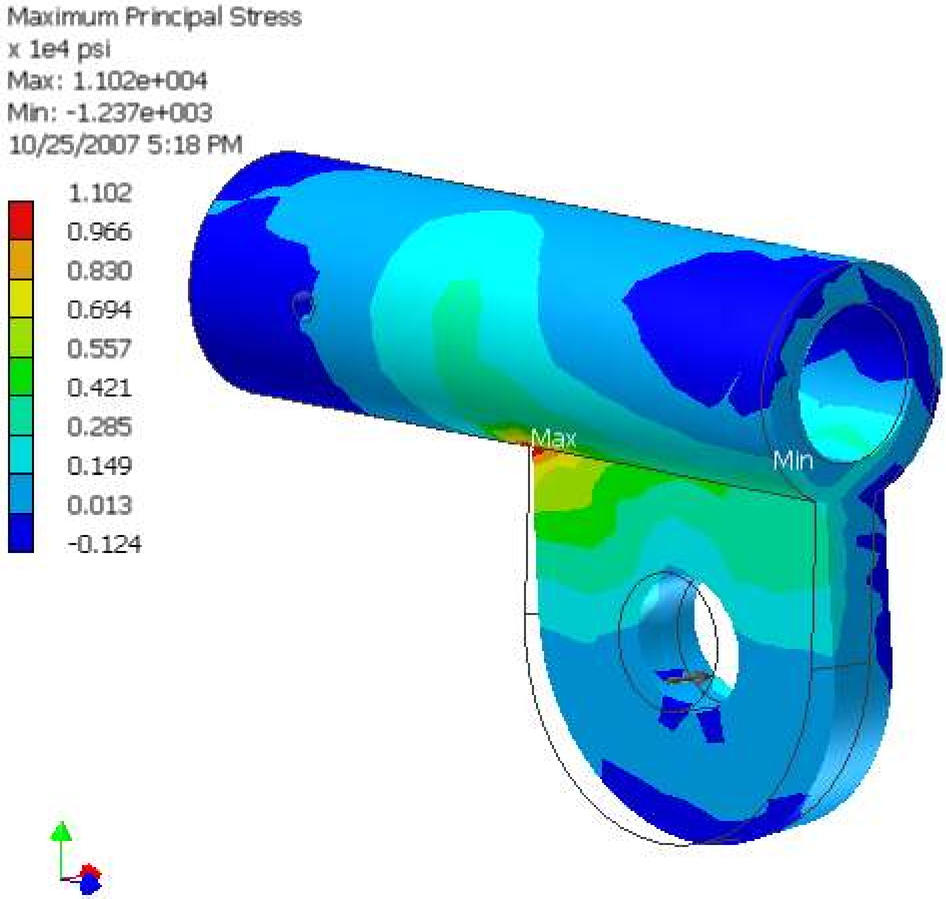


FIGURE 3
Minimum Principal Stress

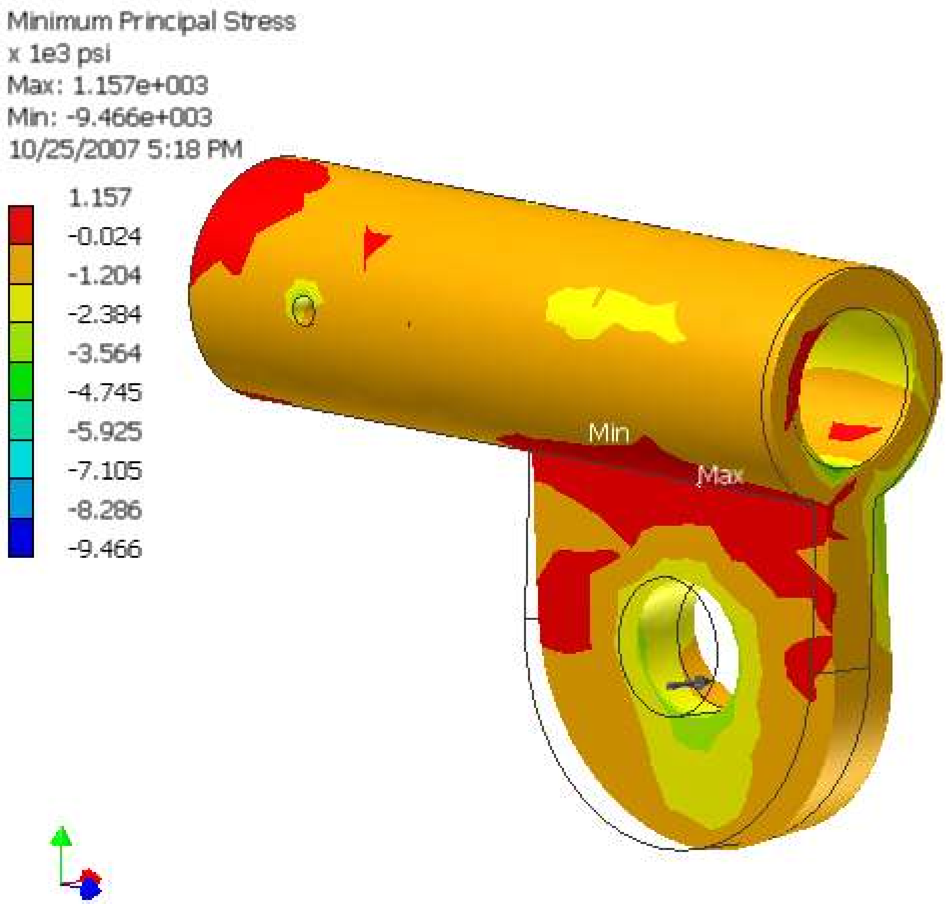


FIGURE 4
Deformation

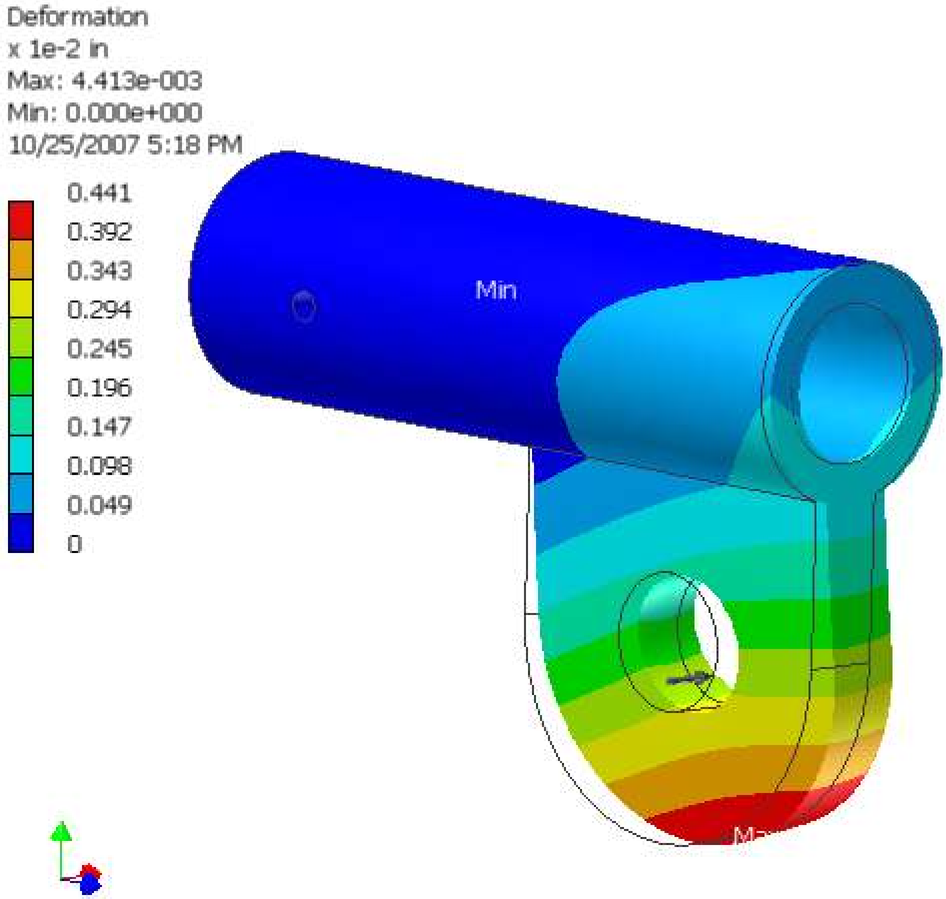
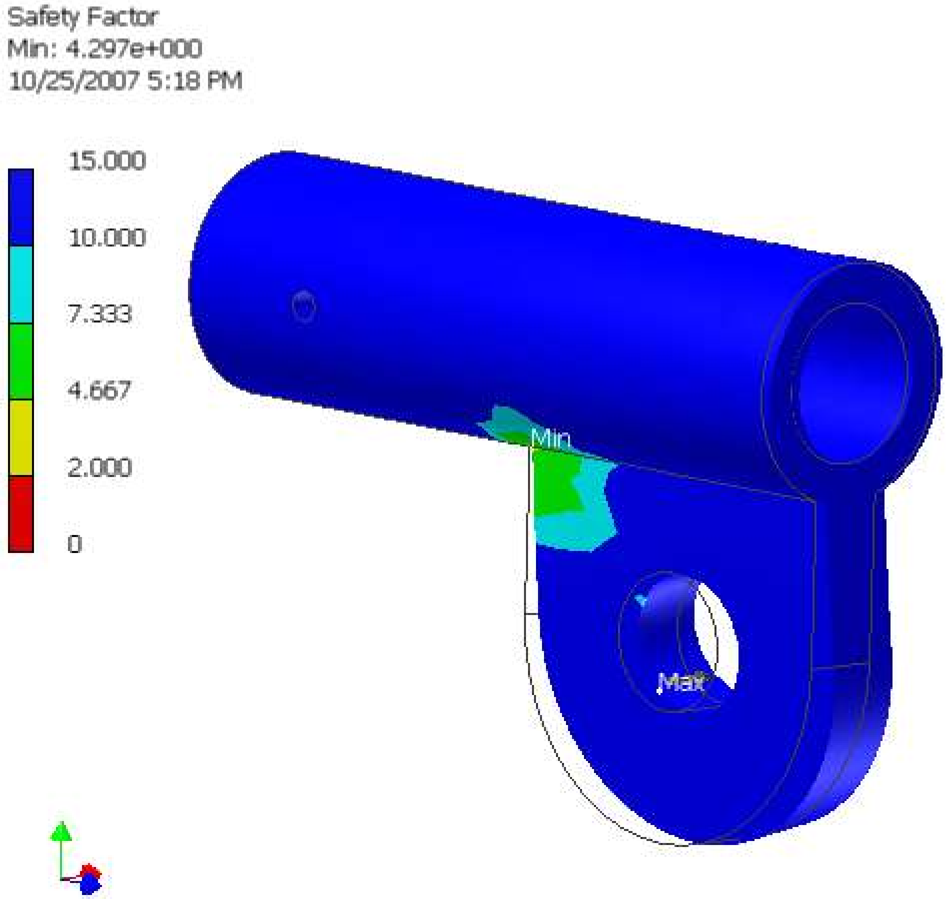


FIGURE 5
Safety Factor


Safety Factor
Min: 4.297e+000
10/25/2007 5:18 PM
15.000
10.000
7.333
4.667
2.000
0
Min
Max