

# Analysis of MASM-0005.ipt

---

**Author:** Bobby  
**Analysis Created:** Thursday, March 22, 2007 4:57:40 AM  
**Analysis Last Modified:** Thursday, March 22, 2007 4:57:40 AM  
**Report Created:** Thursday, March 22, 2007 4:59:29 AM  
**Database:** *C:\Documents and Settings\Bobby\Desktop\BSME\2007 Spring\Sr.Design\inventor\_files\real\_deal\_v10\MASM-0005.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**  
**MASM-0005.ipt Statistics**

Bounding Box Dimensions	6.75 in 0.25 in 5.5 in
Part Mass	0.747 lbm
Part Volume	7.624 in <sup>3</sup>
Mesh Relevance Setting	0
Nodes	7670
Elements	3847

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/in <sup>3</sup>
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
Fixed Constraint 1	Surface Fixed Constraint	0. in	0. in
			0. in
			0. in
Bearing Load 1	Cylindrical Bearing Load	50. lbf	0. lbf
			-50. lbf
			0. lbf

**TABLE 4**  
**Constraint Reactions**

Name	Force	Vector	Moment	Moment Vector
Fixed Constraint 1	50. lbf	-4.795e-007 lbf	3.01e-002 lbf-in	2.98e-002 lbf-in
		50. lbf		-1.586e-006 lbf-in
		-1.285e-006 lbf		4.252e-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.

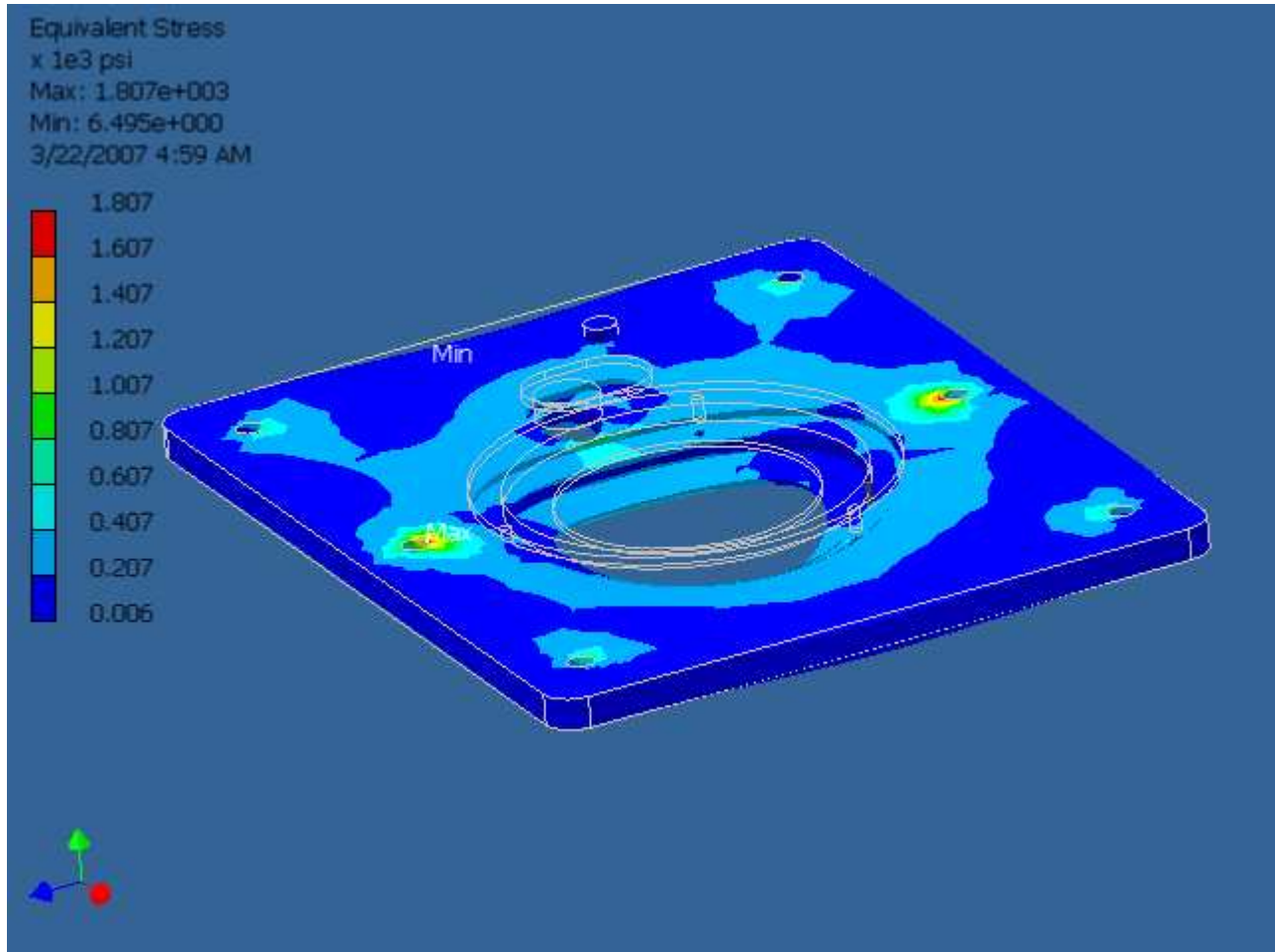
Safety factor was calculated by using the maximum equivalent stress failure theory for ductile materials. The stress limit was specified by the tensile yield strength of the material.

**TABLE 5**  
**Structural Results**

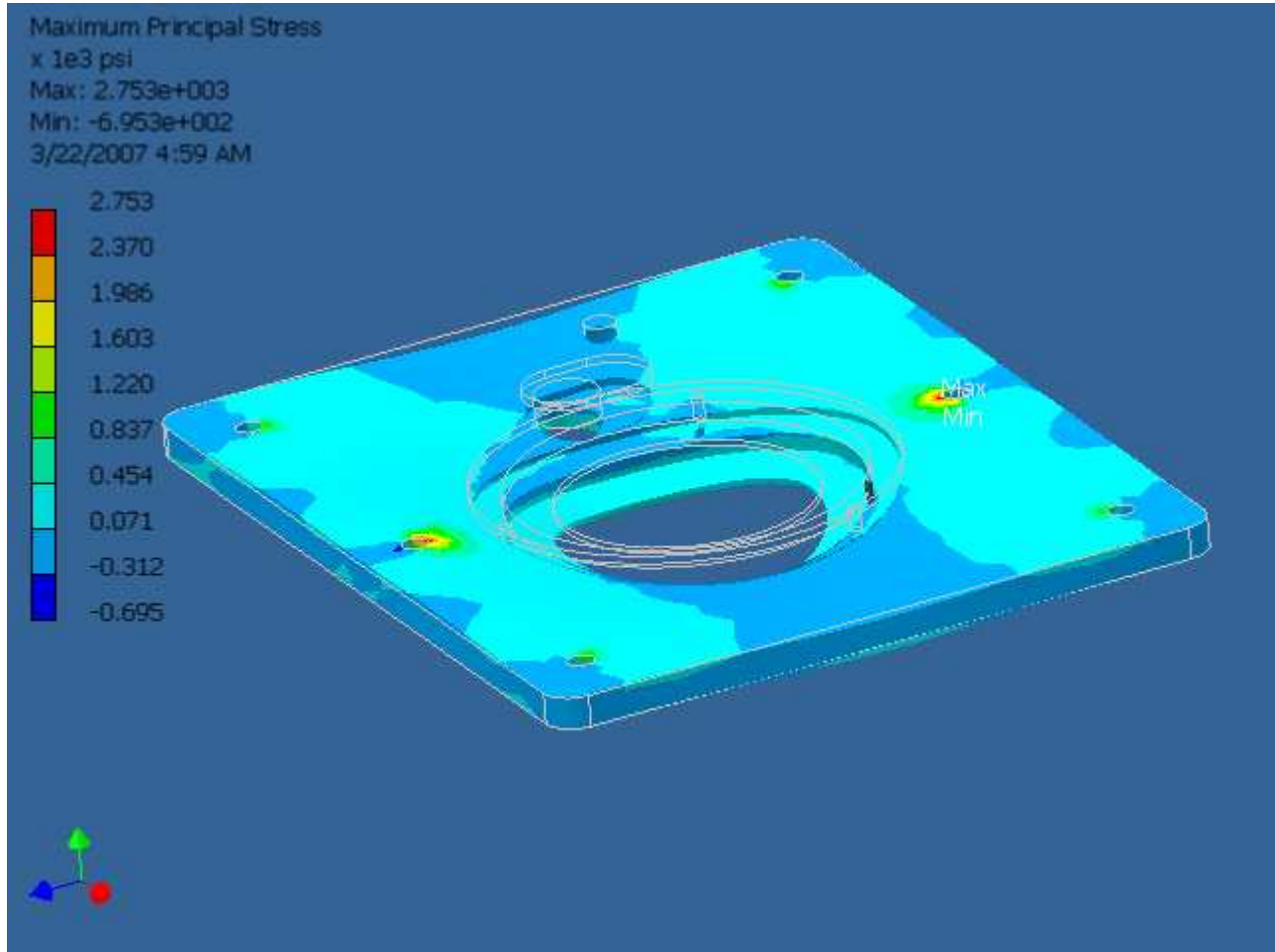
<b>Name</b>	<b>Minimum</b>	<b>Maximum</b>
Equivalent Stress	6.495 psi	1807 psi
Maximum Principal Stress	-695.3 psi	2753 psi
Minimum Principal Stress	-2596 psi	835.3 psi
Deformation	0. in	6.42e-004 in
Safety Factor	15.	N/A

# Figures

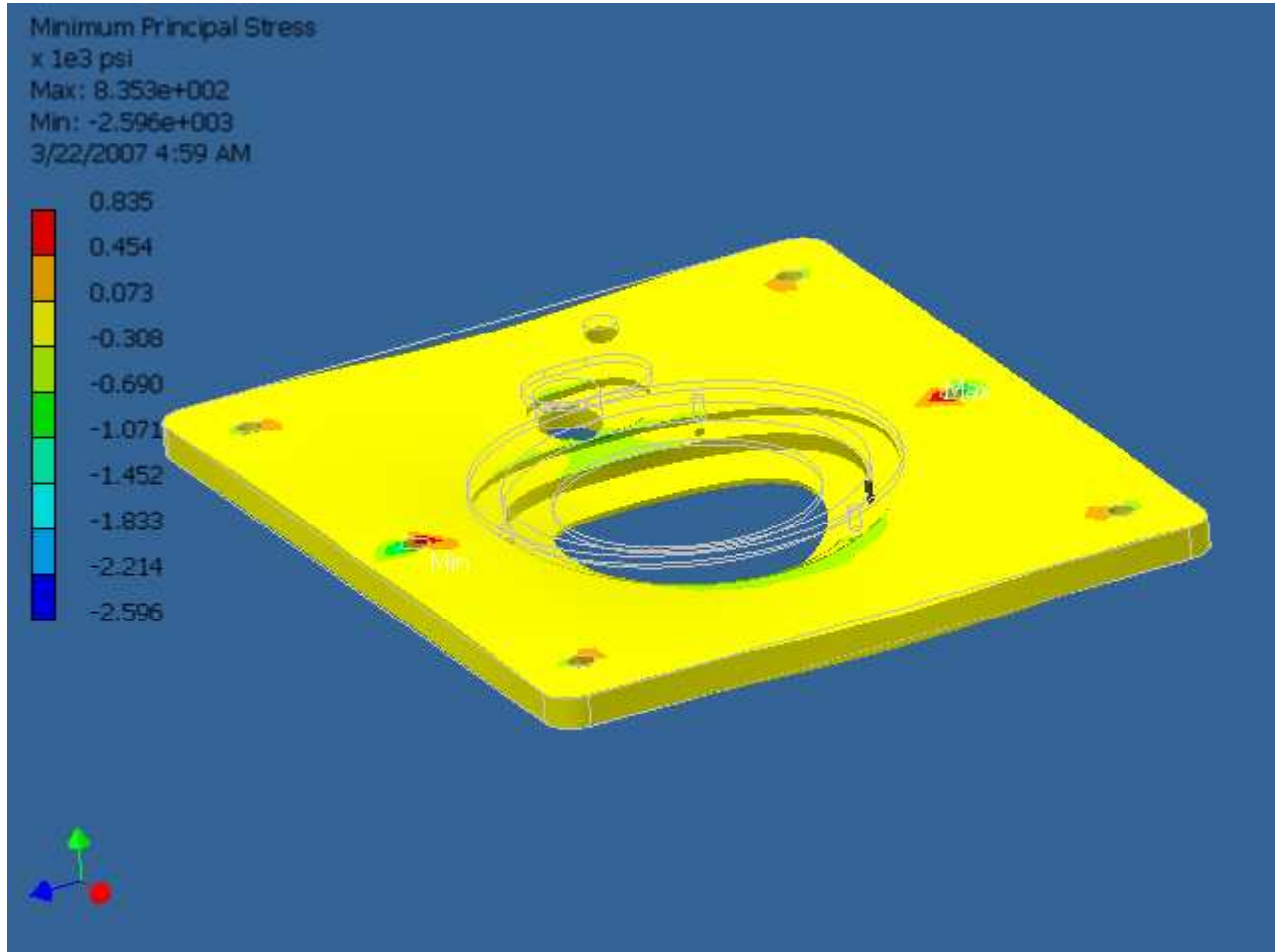
**FIGURE 1**  
**Equivalent Stress**



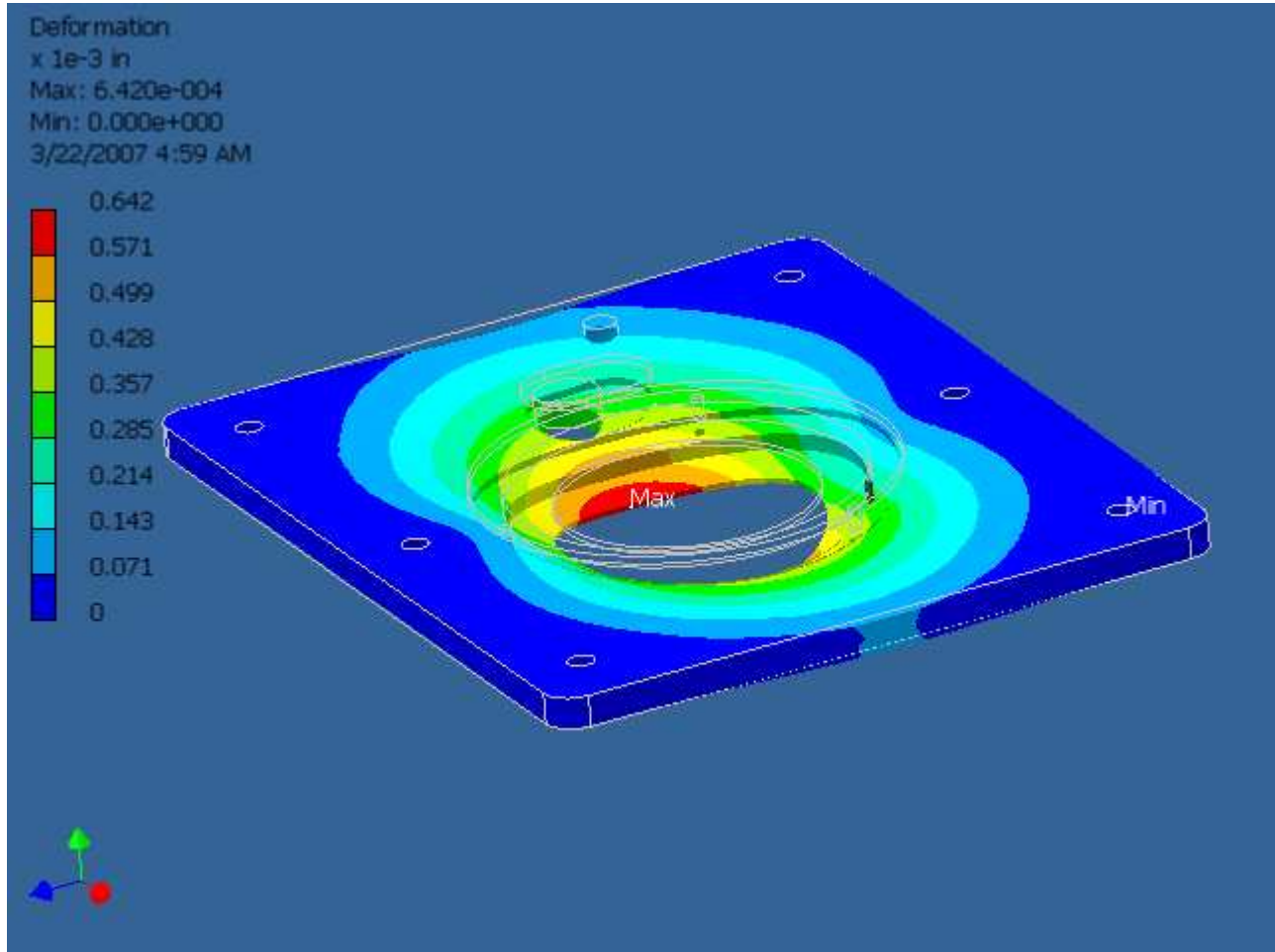
**FIGURE 2**  
**Maximum Principal Stress**



**FIGURE 3**  
**Minimum Principal Stress**



**FIGURE 4**  
**Deformation**



**FIGURE 5**  
**Safety Factor**

