Analysis of Chamber Lid Seismic Study 2

Author:	cdkemp
Analysis Created:	Thursday, February 19, 2009 11:30:42 AM
Analysis Last Modified:	Thursday, February 19, 2009 11:30:42 AM
Report Created:	Thursday, February 19, 2009 1:52:40 PM
Database:	*C:\Drew_Working_Folder\Special Projects\COLTRIMS\Chamber Lid Seismic Study 2.ipa*
Software:	Autodesk Inventor Professional 2009 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
Chamber Lid Seismic Study 2.ipt Statistics

Bounding Box Dimensions	26.5 in 26.5 in 20.0 in
Part Mass	242.5 lbm
Part Volume	2475 in^3
Mesh Relevance Setting	0
Nodes	5464
Elements	2403

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^3
Tensile Yield Strength	3.989e+004 psi
Tensile Ultimate Strength	4.496e+004 psi

Loads and Constraints

The following body loads act on the part. The Location column applies only to rotational velocity. Location represents a point on the axis of rotation.

TABLE 3
Body Load Definitions

Name	Magnitude	Vector	Location
Standard Earth Gravity	386.1 in/s^2	0.0 in/s^2 0.0 in/s^2 386.1 in/s^2	N/A

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

TABLE 4
Load and Constraint Definitions

Name	Type	Magnitude	Vector
Force 1	Surface Force	165.0 lbf	-2.021e-014 lbf -165.0 lbf 0.0 lbf
Fixed Constraint 1	Surface Fixed Constraint	0.0 in	0.0 in 0.0 in 0.0 in

TABLE 5
Constraint Reactions

Name	Force	Vector	Moment	Moment Vector
Fixed Constraint 1	293.1 lbf	5.245e-008 lbf 165.0 lbf 242.3 lbf	2888 lbf·in	-2888 lbf·in -2.697e-003 lbf·in 5.098e-005 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 6
Structural Results

Name	Minimum	Maximum
Equivalent Stress	1.165e-002 psi	3922 psi
Maximum Principal Stress	-649.5 psi	3707 psi
Minimum Principal Stress	-4000 psi	621.1 psi
Deformation	0.0 in	2.743e-002 in
Safety Factor	10.17	N/A

Figures

FIGURE 1
Equivalent Stress

FIGURE 2
Maximum Principal Stress

FIGURE 3
Minimum Principal Stress

FIGURE 4
Deformation

FIGURE 5
Safety Factor