MEMORANDUM

Research Division - Engineering

To: Lowell Klaisner, Dick Horn

From: Tim Thurston **Ref:** TST03-1(Draft A)

Date: 28 October 2002

CC:

Attachments: Figures 1-13

SUBJECT: Anomaly report: Assessment of Tracker Module Closeout Panel FEA Model

---

Summary:

The overall assessment of the FEA model and analysis of the tracker module is as follows –

- The FEA model in the area of the failure is quite coarse and provides no detail of the metal insert or bolting interaction.
- The FEA model did place bolting restraints in the model that centered the restraint in the location of the insert/bolt. The applied interaction of the bolt only provided restraint on the adjacent exterior element nodes.
- The interaction of the bolt/insert and the compressive interaction for preload were not modeled.
- The distortion results of the first five modes are not surprising. This indicates no major errors in the model.
- The RMS stress profile for random vibration indicates high stress points at the base of the closeout, directly below the metal inserts for the corner flexure attachments. In test #2, cracks presented in all eight-corner inserts.
- The RMS stress levels of the FEA were slightly below (~30%) the material tensile strength. Since these stresses are RMS, the peak stresses may exceed the material strength. (This assessment was made without consideration to insert/bolting interaction with the closeout and modeling accuracy)
- The model showed that once a crack presented at a corner insert, redistribution of loads and stresses would accelerate the failure at each of the other corner inserts.

- The modeling of the module interface to the LAT grid and test fixtures has not been performed. The influence of these interfaces on the module analysis is not well understood.

Recommendation:

The recommendations for the FEA analysis of the tracker closeout come in two flavors, the model detail, and the use of the current results. Design recommendations are not addressed in this memo.

- Model recommendations:
    - Increase granularity of the model in the area corners of the bottom closeout, particularly in the area of the metal insert for the corner flexure. Increase granularity in the area of other inserts would also be beneficial.
    - Include insert detail in the model. Include bonded interfaces and insert detail at the interface.
    - Include effects of bolt pre-load in the model. The current model does not allow compressive and shear stresses to be accounted in the overall FEA results.
    - Eliminate the use of 2-D elements in the lower closeout.
- Results recommendations:
    - The minimum design margin of safety should be at least 2.0 for the resulting FEA RMS stresses (von Mises). Additional margin may be necessary based on materials and modeling detail.
    - The results of the current coarse FEA model should use design margins of safety of at lease 3.0 since the details of bolting stresses have not been included in the model and the details of the interface structure has not been accurately modeled.
- Project recommendations:
    - All complex analyses should be given a level of review beyond what was done in this case. A lesson-learned in the resolution of this anomaly resolution is one of review. Complex analyses especially with critical hardware should always be independently review or checked. This analysis, once reviewed in detail, shows that there were area needing additional investigation but it was overlooked.

Purpose of Investigation:

This investigation was prompted as a result of tracker module vibration test failure. The failure was identified after a second vibration test conducted in the summer of 2002. The post-mortem of the failure identified cracks in the corners of the lower tray assembly of the tracker module. The cracks occurred in composite material at each of the inserts attached to the corner flexures. It was also revealed that a hairline crack had presented after the first vibration test

In an effort to clearly understand the root cause of this anomaly, a couple of subordinate issues arose:

- First, what influence did the first crack have on the cracks generated in the second test.
- Secondly, do we have the analysis tools necessary to validate design modification intended to ensure failure free implementation of the tracker modules.

The purpose of this investigation is to respond to these issues and to make recommendation for the continued or enhancement of the current analytical/modeling tools.

Investigations/Findings:

The investigations/findings are mostly related to the finite element model correlation to the actual hardware, the fidelity of the FEA, the relationship of the FEA results to the failure, and the analysis substantiation of the failure propagation theory. These are as follows:

<u>Failure locations</u>:

The FEM was reviewed, particularly in the area of the corner fastener inserts. The attached figure 1 shows the base trays of the tracker module. Figure 1 has been annotated to show the corner locations that manifested crack during the module vibration testing. The first vibration test manifested a hairline crack in one locate only. It was not noticed until the test module was being prepared for the second round of tests. During the second round of tests, the remaining locations manifest much more significant.

Detail drawings of the lower tray walls are shown in figures 5 through 8. The wall of the lower closeout tray is made up of two MCM and two Structural Walls bonded at the corners. Refer to figures 1 for a prospective of the orientation of the MCM and Structural Walls. Figures 6 and 8 display the wall cross-sections and indicate crack locations.

<u>Correlation of FEM/Hardware</u>:

Because of the complexity of the entire tracker module, the fastener areas were not modeled in detail. The model detail is shown in figures 2, 3 and 4. As can be seen in figure 4, no details of the inserts are modeled. The walls were simply modeled as solid members using 3-D elements and 2-D shell elements.

The node layout of the structural closeout wall is shown in figure 9 superimposed on the cross-section of the wall at the crack location. The model geometry is matches the design cross-section quite well. However, the model is coarse and does not account for the insert.

The model and insert attach points were modeled using 2-D shell elements to interconnect the flexure fixation to the closeout wall. This distribute the insert reactions to the neighboring element nodes. A roughly sketch of this is shown on figure 12. This crude simulation was unable to account for insert bonding, material cutouts or fastener pre-loading. Since the interaction of the insert, fastener and closeout wall was not modeled, the resulting FEA stresses in the area of the insert should vary greatly from the actual stress conditions.

Validation of the Finite Element Model

Validation of the model is not as clear-cut as one would wish it to be. Without instrumenting the test module for measuring localized strains in critical areas, the model validated must rely on model/analysis symmetry, stiffness and deflection measurement, and successful testing. It should be noted that none of these shed a great deal of light on the correlation of stresses.

Based on the analysis and test results currently available, the most available methods to check the validity of the FEM and FEM are:

- Geometry/properties validation with test article.
- Check the symmetry in deflections, stresses, mode frequencies, and mode shapes with expectations and preliminary calculation.
- Correlation of mode frequencies, and deflections to test measurements.
- Correlation of FEA stress to failure points on the test article.

Even with good correlation between the analysis and test results, the validation of the current model at best is fair. The model geometry assembly, and fixation can be checked sufficiently as can the correlation of stiffness and mode frequencies. However, without localized stress/strain measurements, design margins and error bands cannot be determined.

Model Geometry - Since the focus of this investigation is in the area of the failure (cracks at the corner inserts), a layout of the closeout sidewall and model geometry was made and is shown in figure 9. The model geometry and cross-section are very similar, only minor dimensional discrepancies. One area of concern was noted in the closeout wall where both 2- and 3-D elements were used. Care must be taken to ensure proper coupling of these elements. The use of 3-D elements exclusively in this area would have been preferred.

Analyzed Behavior - The FEA output behaved very much as one would expect. The observation are:

a. Static load deflection and reactions are symmetrical and as expected.

b. Resonance mode shapes are symmetrical (see figure 10) and fairly intuitive.

c. Torsion in the closeout walls appears to be a factor. A closer investigation of the element deflections at the center of the sidewall and in the area of the crack is shown in figure 11. The relative contribution of torsion was not extracted from the analysis but it does indicate that the interaction of the module attachment is more significant that simple bending and shearing at the inserts.

d. Resonance frequencies for the first few modes compared favorably to hand calculations.

e. Resonance frequencies of the test article was consistent the FEA predictions. The comparison of test and FEA was presented in the HTN-102070-0005, HTN-102070-0002, and HTN-102070-0008.

This behavior tend to support the validation of the FEM as being modeled correctly. The FEA should be able to identify deflection patterns and stress distributions within the closeout walls and identify weaknesses in the design. However, without better correlation to test measurements, the FEA generated stress should be used with caution since the modeling error is well established.

Stress Level at Insert Locations – When subjected to random vibration, the FEA produced high RMS stress level in the area of the corner inserts. These stresses were over 70% of the tensile strength of the closeout wall material. Figure 12 displays stress plot resulting from the analysis (HTN-102070-0005). The high stress areas are in the closeout wall at the start location of the cracks.

Mode/Analysis Considerations – Five principle considerations should be given to the results of the FEA. These are:

- The coarseness of the model will not provide good resolution or accuracy of the stress concentrations. The model should however provide trends in the profile of stresses, strains, and deflections. To gain good resolution and accuracy the model must be much finer in the area of stress concentrations. Based on the stress levels of the surrounding elements an error bar of ±30% could be assumed.
- Lack of detail modeling for inserts, bonding and bolt preloads introduces additional uncertainty. The insert, glue joint, and bolting interact with load transfers, strain redistributions and stress concentrations not accounted for in the FEA. The actual stresses in the closeout wall could easily to be greater and more significant that the results of this FEA.
- Fastener preloads will introduce compressive and shear stresses in the closeout material that will be in the range of 20 to 40% of the material tensile strength (assuming a 800 nt preload applied to the bearing and shear area of the insert). Since this loading was not considered in the model, the FEA results in the area of the inserts are low by a similar factor.
- The non-isentropic material properties of the closeout material will have an impact on the stress representation of the FEA. It appears that the material properties were included in the model. Von Mises stress plots do consider material properties and therefore the stress relationship is not an accurate measure for determining margin. The stress levels indicated in figure 12 may in fact be in fact be closer to failure conditions than it would appear.
- RMS values from the FEA do not predict peak stress. The RMS stress levels are a factor of 2-3 times lower stress levels anticipated at the 2-and 3- sigma levels. Taking this in to account, the FEA values are certain to exceed the material strength and allowables.

Crack Propagation – In an August meeting of the ART, the results of a FEA was presented for a model having the constraints released from one of the inserts. This analysis was performed to show the impact of a single failure at an insert. The results shown in figure 13 clearly indicate a significant redistribution of loads into the other inserts location, of order 3 times greater. Once a failure occurs at one insert location, it is most likely to propagation the failure to other corner inserts locations.

Conclusions:

The FEM/FEA strengths are:

- The correlation of the model and dynamic mode shape shows good correlation to the test results.
- The stress profiles and trends are of value and can be used to attain points of focus for further design and analysis.

The FEM/FEA weaknesses are:

- The coarseness of the current model prohibits the use or validation of stresses in the insert area.
- Modeling detail diminished the understanding failure mechanisms in the closeout material.

The initial presentation of cracking of the first test most likely led to propagation of the failure at the other insert locations during the second test.

The FEA indicated that failure in any one of the insert locations is likely to propagated to the other insert locations.

The FEM/FEA provided a good analysis of the Tracker module, however, the model did not have the resolution or detail to investigate or build an understanding of the failure mechanism of this anomaly. A more detailed model was needed and more in-depth testing would have been beneficial in resolving this problem.

The stress levels in the area of the closeout material failure (at the corner flexure attachment inserts) were high and should have triggered further refinement of the model and design investigation. Typically independent review or checking is performed on these types of analyses. If this was not performed for this analysis, it should be noted as a lesson-learned and applied through the other portions of the project.

I

Attachments:

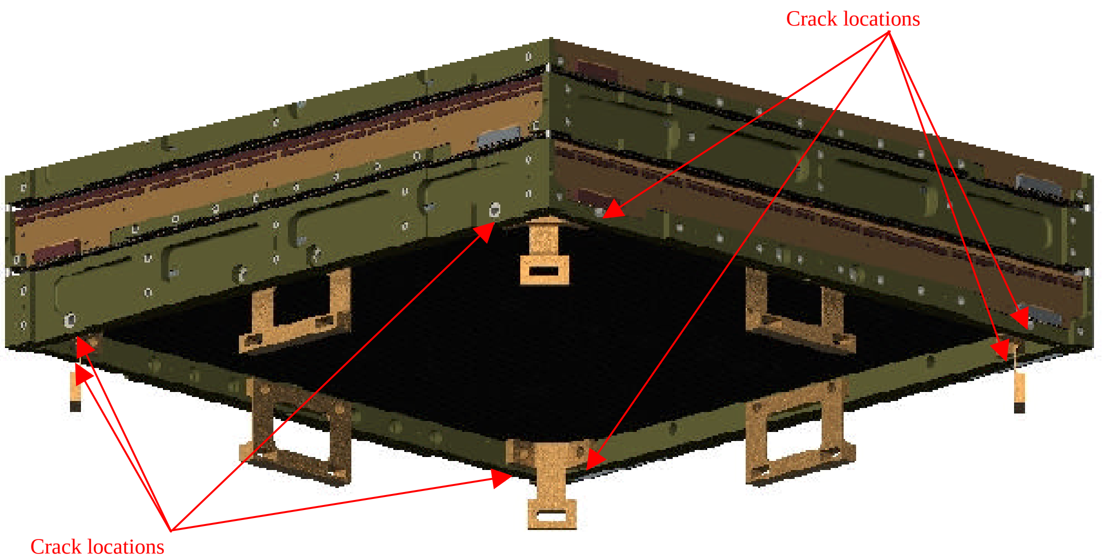


Figure 1. Bottom tray-to-grid

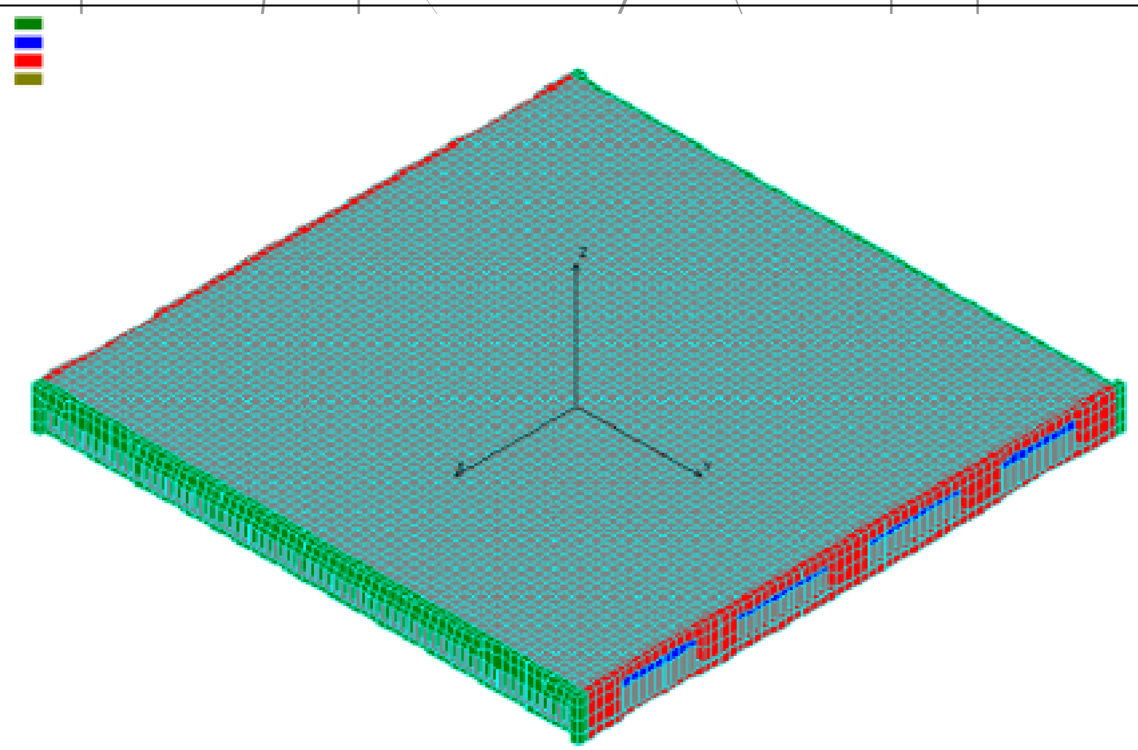


Figure 2. Tray FEM detail

(HTN-102070-0005, figure 4)

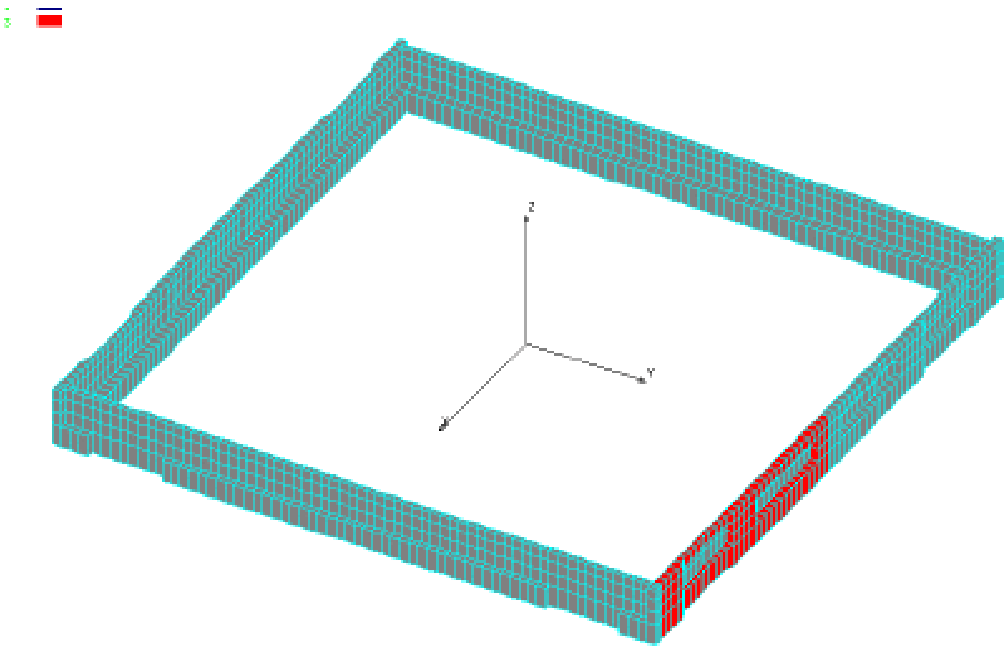


Figure 3 Base tray closeout model

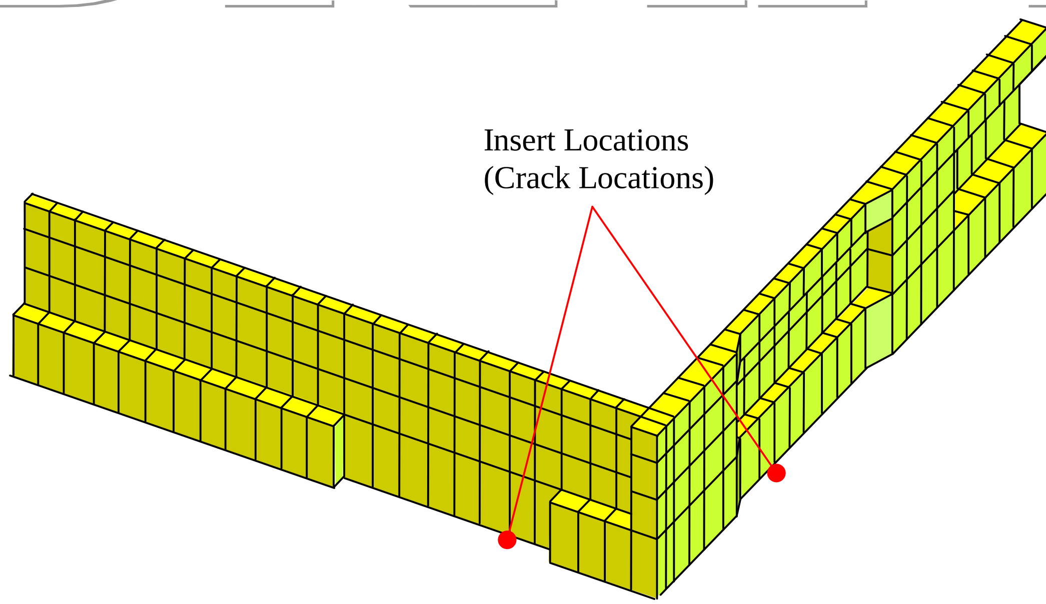
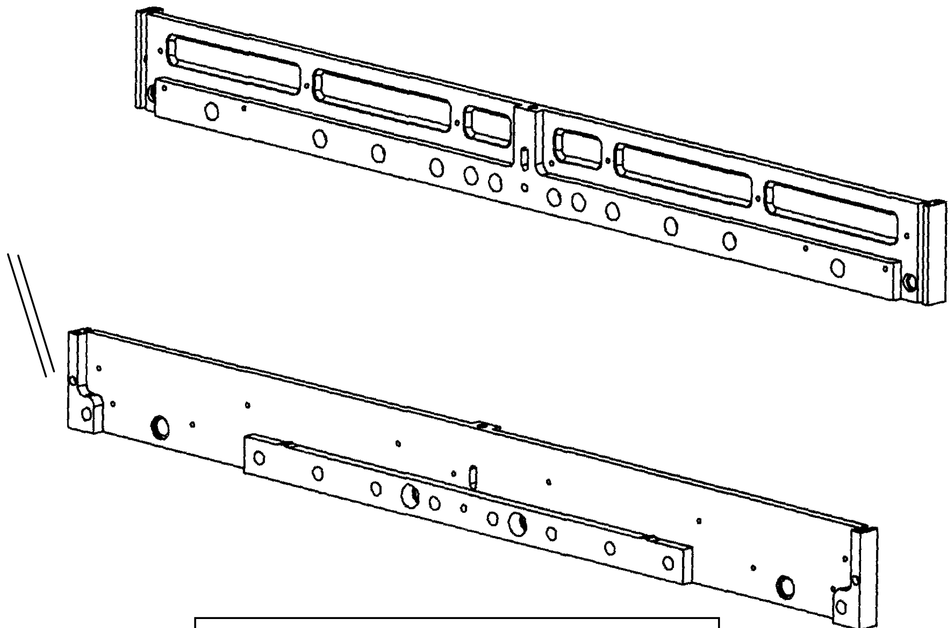
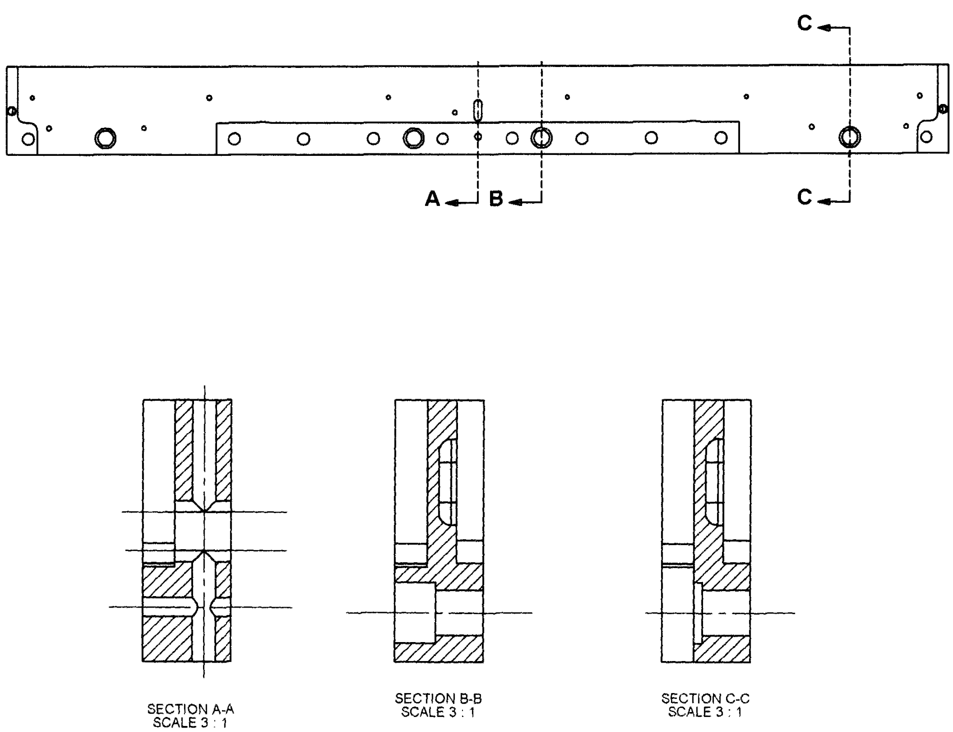


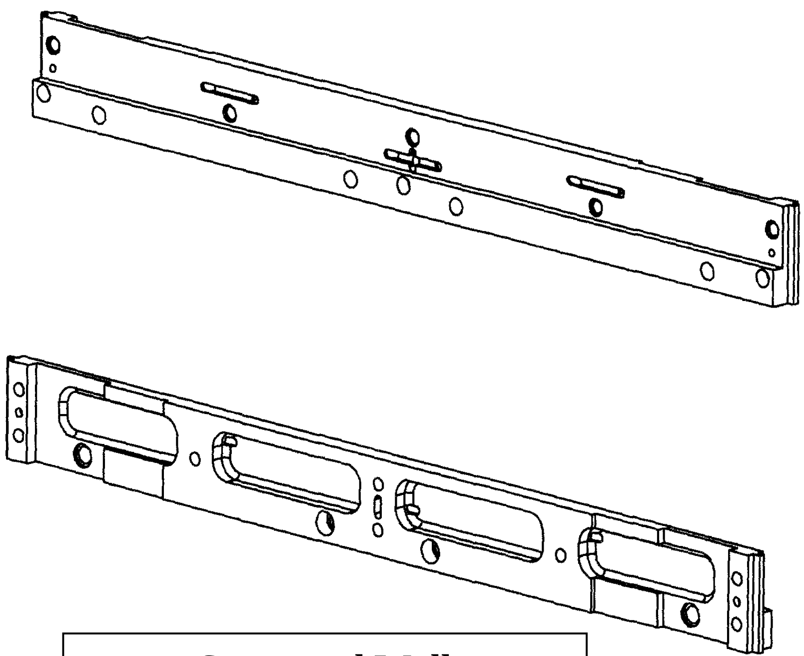
Figure 4 - Base tray closeout model - corner



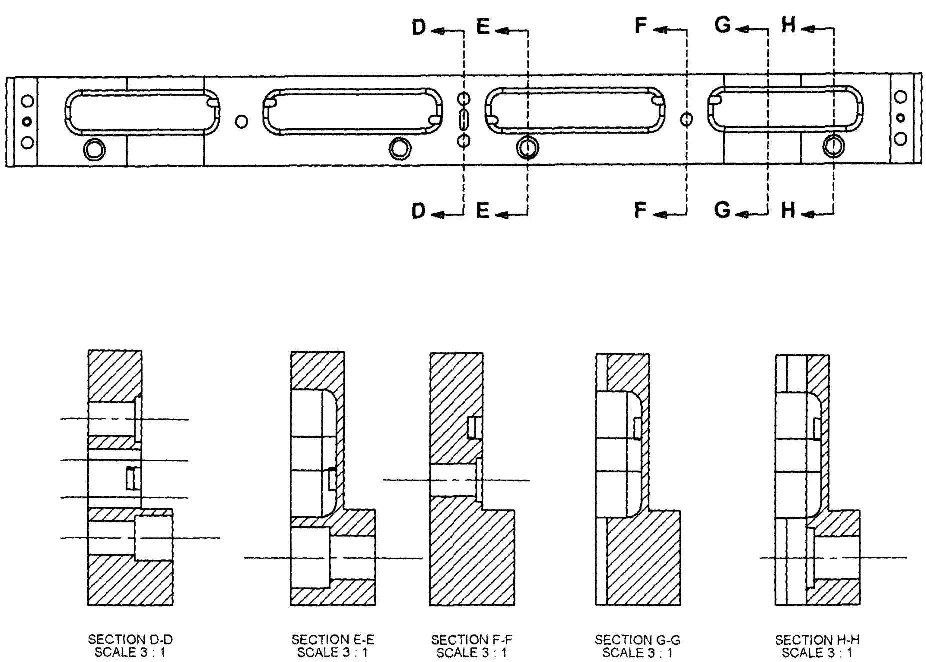
MCM Closeout Wall
Figure 5



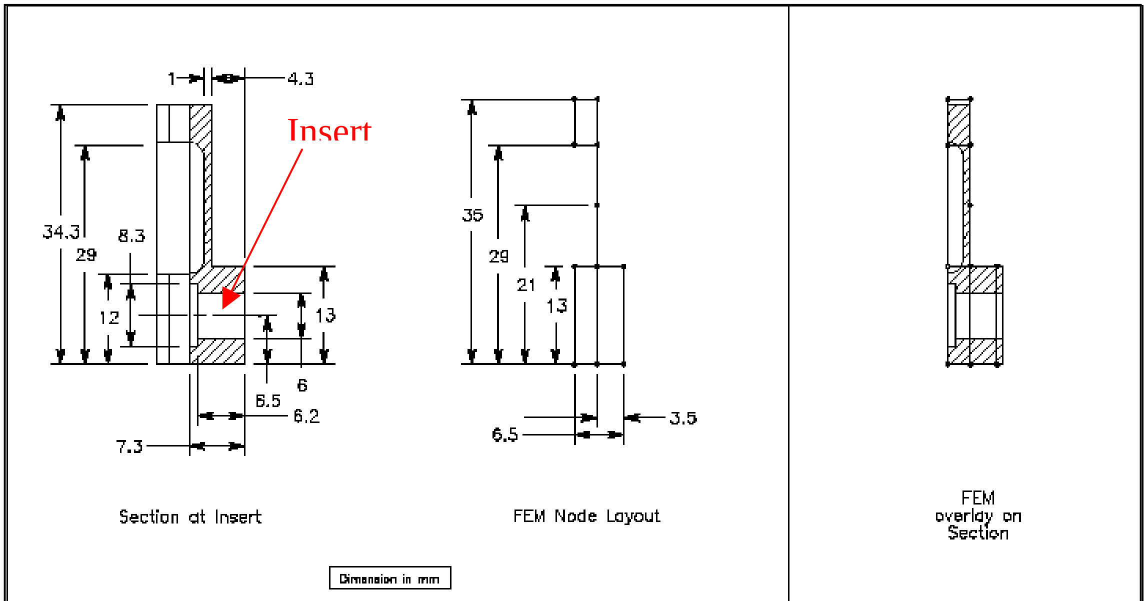
MCM Closeout Wall with Cross Section



Structural Wall
Figure 7



Structural Closeout with
Cross Section



FEM Node overlay at Structural Wall Corner Insert

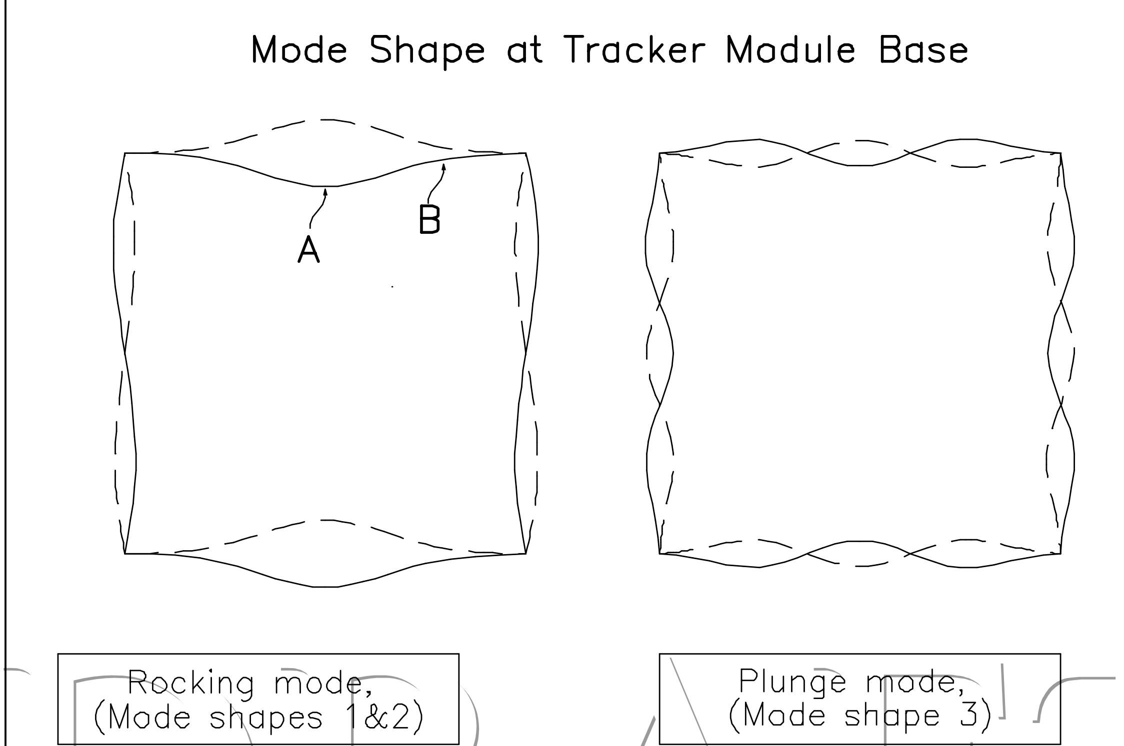
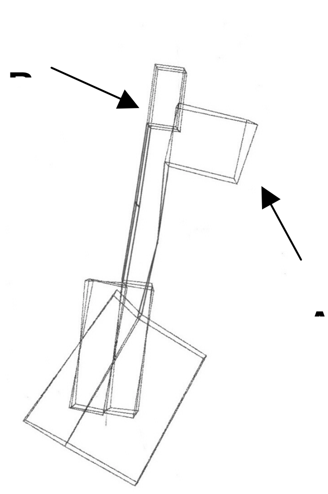


Figure 10 – Mode shapes

Distortion at TKR Module Closeout Wall Base



Note:
Node amplified displacements from the vertical of structural

Figure 11 – Deformed shape of closeout wall elements

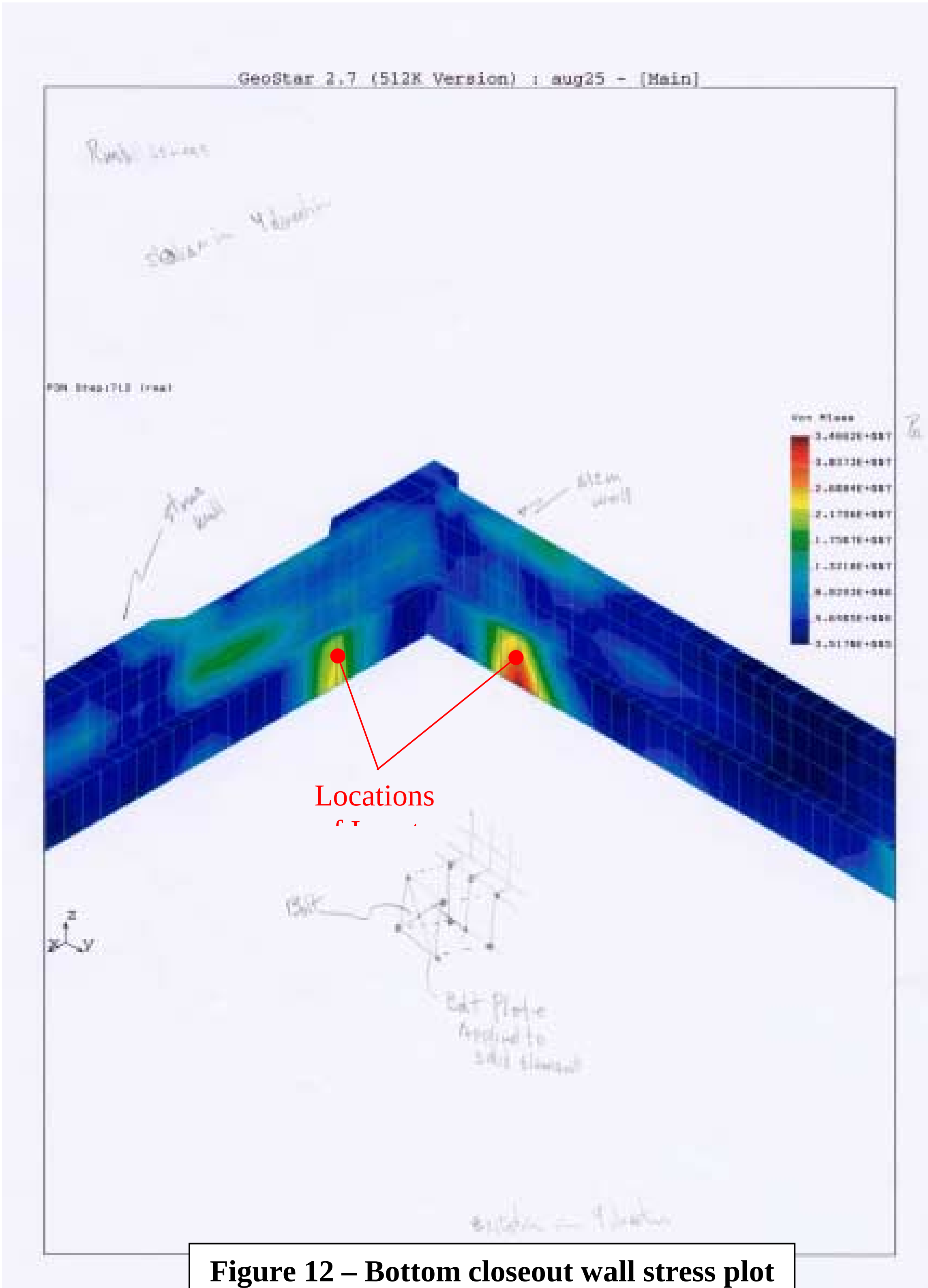
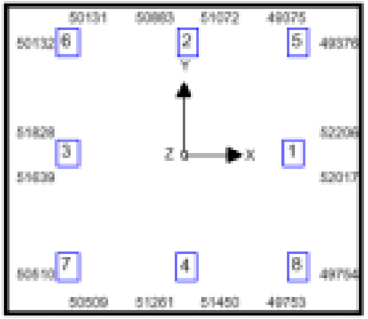


Figure 12 – Bottom closeout wall stress plot

******* Flexure Fastener Loads *******

Element Number and Location



Static Load Cases

Load Description	# of Fasteners Removed	Flexure Type	Maximum Load on Fastener								
			Flex Loc	Axial (N)	Axial (lbs)	Flex Loc	Lateral Shear (N)	Lateral Shear (lbs)	Flex Loc	Vertical Shear (N)	Vertical Shear (lbs)
Liftoff & Transonic	0	Side	1	-167.7	-37.7	4	226.8	51.0	2	-142.1	-31.9
		Corner	8	71.2	16.0	8	115.0	25.9	5	-383.8	-86.3
MECO	0	Side	2	102.6	23.1	4	40.1	9.0	2	-93.1	-20.9
		Corner	7	40.7	9.2	8	40.4	9.1	6	-253.1	-56.9
Liftoff & Transonic	1 (#8)	Side	1	-241.0	-54.2	2	220.3	49.5	1	-194.8	-43.8
		Corner	8	249.2	56.0	8	217.0	48.4	5	-439.6	-98.8
MECO	1 (#8)	Side	4	-153.0	-34.4	4	-80.6	-18.1	4	-150.0	-33.7
		Corner	8	98.6	22.2	8	85.0	19.1	7	-234.4	-52.7

Random Vibration Load Cases

Load Description	# of Fasteners Removed	Flexure Type	Maximum Load on Fastener								
			Flex Loc	Axial (N)	Axial (lbs)	Flex Loc	Lateral Shear (N)	Lateral Shear (lbs)	Flex Loc	Vertical Shear (N)	Vertical Shear (lbs)
X Direction to GEVS (12.3 grms)	0	Side	3	481.8	108.3	4	665.9	149.7	3	403.8	90.8
		Corner	7	261.1	58.7	8	310.4	69.8	6	1416.4	318.4
Y Direction to GEVS (12.3 grms)	0	Side	2	566.8	127.4	3	666.4	149.8	2	490.3	110.2
		Corner	7	305.7	68.7	7	354.8	79.8	8	1312.6	295.1
Z Direction to GEVS (12.3 grms)	0	Side	4	375.3	84.4	4	138.9	31.2	4	325.1	73.1
		Corner	7	149.0	33.5	7	143.3	32.2	6	890.8	200.3
X Direction to GEVS Delta II (12.3 grms)	0	Side	1	222.2	50.0	4	417.7	93.9	4	230.5	51.8
		Corner	7	125.8	28.3	8	157.5	35.4	6	632.4	142.2
Y Direction to GEVS Delta II (12.3 grms)	0	Side	2	264.9	59.5	3	411.8	92.6	2	230.1	51.7
		Corner	7	154.5	34.7	7	183.9	41.3	8	598.2	134.5
Z Direction to GEVS Delta II (12.3 grms)	0	Side	4	304.2	68.4	4	112.6	25.3	4	263.4	59.2
		Corner	7	120.7	27.1	7	116.2	26.1	6	721.9	162.3
X Direction to GEVS (12.3 grms)	1 (#8)	Side	1	625.0	140.5	2	588.4	132.3	4	537.7	120.9
		Corner	8	514.5	115.7	8	449.5	101.1	5	1382.5	310.8
Y Direction to GEVS (12.3 grms)	1 (#8)	Side	4	710.2	159.7	1	710.3	159.7	4	634.4	142.6
		Corner	8	471.4	106.0	8	412.9	92.8	6	1461.9	328.6
Z Direction to GEVS (12.3 grms)	1 (#8)	Side	4	485.3	109.1	4	370.0	83.2	4	484.8	109.0
		Corner	7	204.6	46.0	7	227.1	51.1	5	983.1	221.0
X Direction to GEVS (12.3 grms)	2 (#8)	Side	4	885.4	199.1	4	576.4	129.6	4	687.5	154.6
		Corner	7	235.0	52.8	7	260.2	58.5	6	1664.0	374.1
Y Direction to GEVS (12.3 grms)	2 (#8)	Side	4	849.4	191.0	4	542.8	122.0	4	697.8	156.9
		Corner	7	233.2	52.4	6	266.9	60.0	6	1479.3	332.6
Z Direction to GEVS (12.3 grms)	2 (#8)	Side	4	518.8	116.6	4	303.3	68.2	4	482.8	108.5
		Corner	7	208.2	46.8	7	225.2	50.6	7	1060.1	238.3

* Loads are highly dependent on Q (using Q = 12.5 in FEA - matched to white noise response in test)

Figure 13 – Flexure Fastener loads
(with 0, 1, & 2 fasteners removed)