DYNAMIC SUBSTRUCTURE ANALYSIS OF THE INTERNATIONAL ULTRAVIOLET EXPLORER (IUE) SPACECRAFT

William R. Case
NASA Goddard Space Flight Center

13

SUMMARY

This paper presents the results of a steady state vibration analysis of the IUE spacecraft simulating its sinusoidal vibration test. The model of the spacecraft, including solar arrays and the scientific instrument consisted of three separate substructure models; one of which was used to represent the two identical solar arrays. Substructuring techniques were used since the large overall size of the problem precluded solving it utilizing a single model. The paper discusses the models used for each substructure, including reduction to an acceptable size for the combined dynamic analysis. Also discussed are the DMAP alters needed for performing the modal analysis and the subsequent modal frequency response and substructure data recovery. Comparison of the results with data obtained during vibration tests of the spacecraft are also included.

INTRODUCTION

The International Ultraviolet Explorer (IUE) is a 6670 N (1500 lb) Explorer class satellite designed to make astronomical observations in the ultraviolet spectrum. It is to be placed in synchronous orbit from which it will be in continuous contact with the control center at the Goddard Space Flight Center (GSFC). Guest observers, from this country as well as others, will come to GSFC to use the observatory, consisting of the ground control center and the orbiting satellite.

The IUE satellite, in its orbital configuration, is shown in figure 1. Figure 2 shows an exploded view of the satellite with only one of the two solar arrays indicated. The satellite consists of the basic spacecraft (S/C) structure, two solar arrays and the scientific instrument (SI) which is the heart of the IUE.

In the launch configuration the solar arrays are folded around the S/C upper body as indicated in figure 2. The arrays are latched to the top deck of the upper body structure at this time. The satellite is attached to the Delta launch vehicle through the use of a conical adapter which is clamped to the satellite at station 0.0 of figure 1. The adapter is not shown in the figure but is part of the structure that is vibrated during the design qualification vibration testing of the satellite. The analysis described herein was performed to estimate the vibration levels and dynamic loads that would be experienced by the IUE during the sinusoidal vibration tests.

IUE STRUCTURE DESCRIPTION

Spacecraft

The IUE S/C structure is shown in figure 2 (along with a solar array and the SI). It consists of the upper body structure, two equipment decks, an upper cone structure to which an apogee motor connects, a propulsion bay and a lower cone. The base of the lower cone, which is S/C station 0.0, is attached to a Delta launch vehicle conical adapter via a Marmon clamp. The S/C structure was designed and built at GSFC and is made mainly of aluminum.

The upper body structure is basically a truss with shear panels around the outer, octagonally shaped, periphery. The two equipment decks, which attach to the top and bottom of the truss, are aluminum skin honeycomb.

Solar Arrays

A schematic of one of the solar arrays is shown in figure 2; the other array attaches to the S/C 180° from the one shown. Each array is connected to the S/C during launch at five locations. At the upper S/C deck the arrays connect with ball joints that take all of the thrust axis loads from the arrays. The deployment mechanism is built to take shear in a plane perpendicular to the thrust axis. These are also swivel pins that partially restrain the motion at the lower corners of the arrays. The array structure is honeycomb panels stiffened by lightweight beams (not shown in figure 2). They are fabricated in Cannes, France by Societe Nationale Industrielle Aerospatiale (SNIAS) under subcontract to the European Space Research Organization (ESRO). Each solar array weighs approximately 89 N (20 lb).

Scientific Instrument

A cutaway view of the SI is shown in figure 3. It consists of a 45 cm Ritchey-Chretien telescope and an echelle spectrograph. The telescope structure consists mainly of an aluminum cylinder with stiffening rings at either end (strong ring and secondary mirror support spider). The SI is attached to the S/C upper body truss at three locations around the strong ring. The spectrometer structure is three main decks (camera, echelle and collimator decks) supported by three pairs of legs spaced 120° apart. The support legs are made of graphite fibre reinforced epoxy (GFRP) and the decks for the engineering test unit S/C were made of aluminum. Two smaller decks are mounted to the uppermost main deck (camera deck). Around the spectrograph is a non-structural dust cover. The spectrograph is designed and built at GSFC and weighs approximately 870 N (250 lb).

FINITE ELEMENT MODELS

Spacecraft

The original S/C structure finite element model was generated by Avco Corp. under a contract to the Mechanical Systems Division at GSFC. This model, following some modifications and updates was used in the combined dynamic analysis due to its availability and the fact that it appeared to be of sufficient detail to obtain the first few modes (in each axis) adequately. Figure 4 shows this model with the upper body truss shown separate for clarity. The main structural members are the two decks, the upper body truss, the shear panels (around the octagonal outside faces of the upper body truss), the upper and lower cones and the upper cone truss. The model contains 200 grid points with 1062 unconstrained degrees of freedom (DOF) and employs 426 elements (mainly CBAR, CQUAD, CTRIA). For the dynamic analysis, the 1062 DOF were reduced to 157 DOF using the Guyan reduction.

Solar Array

The solar array model used in the combined dynamic analysis was generated by the Mechanical Systems Division at GSFC from data supplied by ESTEC, the technical monitors for ESRO on the solar array contract. ESTEC supplied NASA with their finite element model and it was then converted to a NASTRAN model. Figure 5 shows the mesh used. The model consists mainly of CQUAD and CBAR elements. There are 165 grid points with 916 unconstrained DOF in the model for one array. For the dynamic analysis, the 916 DOF were reduced to 126 DOF.

In order to verify the solar array model with ESTEC results, an eigenvalue analysis of the cantilevered array was run. The array was constrained in all degrees of freedom to which it connects to the S/C. The modes of the NASTRAN model, compared to the ESTEC results (using the ASKA program) are shown below for the first few modes. Also shown are some preliminary test data identifying the first few modes of the array. In order to obtain better agreement with the test data so that a better representation of any S/C-array coupling problems could be assessed, the array model stiffness was lowered by 10% for the coupled dynamic analysis. The resulting frequency comparisons are also shown below.

Solar Array Mode	Frequencies (Hz)			
	ESTEC ASKA model	NASA NASTRAN model	Test Results	NASA NASTRAN model (10% reduced stiffness)
1st Sym.	59.8	55.5	53-55	52.9
2nd Sym.	77.5	73.5	65-68	70.2
1st Antisym.	50.5	50.8	46-48	48.4
2nd Antisym.	73.0	78.4	74-76	74.7

Some more recent test data indicate slightly lower frequencies but these have not as yet been documented by ESTEC.

Scientific Instrument

A detailed finite element model of the SI was formulated by the Test and Evaluation Division at GSFC for performing combined structural-thermal-optical analyses to assess the performance of the SI while in orbit. Figures 6 and 7 show the telescope and spectrometer portions of this model. For the dynamic analysis this model was reduced to only a few grid points for each of the three main decks plus several degrees of freedom for the telescope, which is quite stiff. In all, the original model contained 3006 unconstrained DOF with a reduction to 72 DOF by the Guyan reduction.

Verification of the spectrometer portion of the model was accomplished by comparison with test data on the SI with the telescope removed. This comparison is shown below for the modes of the spectrometer cantilevered at the strong ring with the NASTRAN results from reference 1.

Spectrometer Mode	Frequency (Hz) NASTRAN model	Test
1st Y bending	16.2	15
1st Z bending	16.8	
1st torsion	23.8	25
2nd Y bending	37.3	37
2nd Z bending	37.3	

These theoretical modes are essentially the same with the telescope connected to the strong ring due to the high stiffness of the strong ring and the telescope tube.

DYNAMIC ANALYSIS

Formulation

From the preceeding discussions it can be seen that the complete model, if run as a single finite element model without substructuring, would be very large. Due to this fact, as well as the fact that there are a relatively few DOF in which the solar arrays and the scientific instrument connect to the S/C, it was decided to use substructing techniques to solve for the modes and steady state vibration response of the complete satellite. Another factor which makes this approach attractive is that the two solar arrays are identical and can be represented through the use of a single finite element model data deck if substructing techniques are used. The table below shows the size of the models for each of the substructures together with the computer time (CPU seconds on the IBM 360/95) required to generate the reduced stiffness and mass matrices for each substructure.

Substructure	Unconstrained DOF "F"	Reduced DOF "A"	Phase I CPU Time (seconds)
S/C	1200	157	478
Solar Array	990	126	245
SI	3006	72	1078
Totals (1)	6132	427	-

(1) Total DOF numbers reflect the fact that there are two solar arrays but do not include duplicated DOF at points where substructures attach to one another.

It can be seen that if the problem were solved as one structure it would contain 6132 DOF which is too large, even for the IBM 360/95. Taken as three separate substructures, the total time to generate all of the matrices needed to perform the eigenvalue analysis is 1801 CPU seconds indicated in the table. This includes the two solar arrays since only one set of matrices is needed due to the fact that they are identical. The grid point locations and the stiffness and mass matrices for the solar array DOF were generated in a local coordinate system. For the DOF on the S/C where the arrays attach, two coordinate systems were defined (one for each array) such that they would match the local system for that array. This coordinate system was then used for the global system for the DOF for the particular array and its S/C attach points.

In order to obtain the vibration modes and steady state response using substructure techniques, the problem is run in several phases as indicated below.

Phase	Description	No. of Runs
I	Formulate K_{AA}, M_{AA} for each distinct substructure model	3: S/C, solar array (once), SI
II-a	Combine substructures and do modal analysis	1
II-b	Restart II-a and do modal frequency response	1
III-a	Restart I with II-a tape of "A" set mode shapes to recover mode shapes for other DOF	4: S/C, solar array (twice), SI
III-b	Restart I with II-b tape of "A" set frequency responses to recover frequency response for other DOF	4: S/C, solar array (twice), SI

Notice that only one solar array Phase I run is needed although there are two solar arrays. When Phase II-a is run, the solar array stiffness and mass matrices are expanded and added to the S/C matrices twice (in the appropriate rows and columns) so that each array is represented in the combined model. When responses are calculated in Phase III (for the points reduced by Guyan reduction), the "A" set points from Phase II are used for the DOF corresponding to the appropriate array.

In order to implement the substructuring procedure described, Direct Matrix Abstraction Programming (DMAP) alters to NASTRAN Rigid Formats 3 and 11 were used. The basic DMAP alters for a similar substructuring problem were set up by Universal Analytics, Inc. under contract to GSFC and are described in reference 2. Those procedures were modified slightly to take advantage of the fact that the two solar arrays can be represented by one model, as already described. Appendix A lists the DMAP alters to NASTRAN level 15.5.1 that were used for the IUE analysis. The partitioning vectors (PV1, PV2, PV3, PV4) required in Phase II-a when the reduced substructures are combined are used to expand the matrices to the combined "A" set size and add rows and columns of substructure matrices at the DOF where substructures connect. The use and construction of these are discussed in reference 2 and in the NASTRAN User's Manual (ref. 3).

Modal Analysis

As indicated in the previous section, the three substructures were reduced to 157, 126 and 72 DOF for the S/C, one solar array and the SI respectively. When combined into the complete model (with two arrays and accounting for duplicate connection points) the model contained 427 DOF. From this model the modes of the structure cantilevered at the base of the Delta adapter, as in the sine vibration tests, were calculated. The particulars for this Phase II-a run are:

No. of DOF: "A"	Eigenvalue Method	No. Eigenvectors Computed	Computer Core Required (bytes)	Phase II-a CPU Time (seconds)
427	Givens	25	750K	665

The results of this Phase II-a run give the eigenvalues and the eigenvectors for the "A" set DOF. To recover the mode shapes for the other DOF for each substructure, the Phase III-a runs are executed. These use a restart tape from the Phase I set up run along with the OUTPUT1 tape (from Phase II-a) with the partitioned "A" set mode shapes for the particular substructure. As seen from Appendix I, the Phase II-a run partitions the "A" set mode shapes into four data blocks; one for each substructure. These go on one physical tape and the appropriate data block is read from this tape during each Phase III-a run. In this manner, the mode shapes for all DOF in the problem are obtained and can be plotted, one substructure at a time. The Phase III-a mode shape recovery solution times are given below.

Substructure	Phase III-a CPU Time (seconds)
S/C	102
Solar Array No. 1	126
Solar Array No. 2	126
SI	342

Frequency Response

Once the eigenvalues and "A" set mode shapes are obtained, a modal frequency (or steady state) response analysis can be run. As mentioned previously, this is accomplished with a restart of Phase II-a and a switch from NASTRAN rigid format 3 to rigid format 11. The output from this phase, as with Phase II-a are solutions for the "A" set DOF. In this case, they are the frequency response solutions to whatever steady state loads were input. For the IUE vibration test simulation, the input "loads" are actually accelerations at the base of the Delta adapter. Since NASTRAN does not allow direct specification of base motions, this was accomplished by including, in the original model, a large mass at the base of the Delta adapter. During the Phase II-a modal analysis, the DOF to which the large mass was attached were not constrained but included in a SUPORT Bulk Data card. The resulting analysis gave rigid body "shaker" modes plus elastic modes that, due to the presence of the large mass, were essentially cantilevered from the large mass. The rigid body modes are then included in the frequency response analysis and the structure is excited with a load at the large mass whose value is the large mass multiplied by the desired acceleration at the large mass DOF (the shaker/structure interface). For this analysis, the desired acceleration at the base of the Delta adapter was 1.0g in order to obtain transmissibilities of the structure for base acceleration. The Phase II-b solution time is given below.

No. of Modes	No. of Frequencies	Phase II-b CPU Time
25	51	203

Once Phase II-b is run, the transmissibilities (in this case) for the "A" set DOF are available and the Phase III-b runs can be executed to obtain frequency responses for any desired output for any substructure. This is very similar to the restart to obtain mode shape data (Phase III-a) after completing Phase II-a. The

Phase III-b solution time to recover the frequency response output for each substructure is given below.

Substructure	Phase III-b CPU Time (seconds)
S/C	477
Solar Array No. 1	337
Solar Array No. 2	337
SI	858

Some of the results of the analysis for Z axis vibration are shown in figures 8 to 13. Figures 8 to 11 show transmissibilities representing the acceleration at each of four locations per unit acceleration at the base of the Delta adapter. Also included in these figures are test results. The test results were obtained from vibration tests at a low, 0.2g, input level as well as from the design qualification tests which were run at up to 2.0g. It can be seen that there is an appreciable difference in the high and low input test data due to nonlinearities in the structure. This is particularily evident on figure 9 where the 16 Hz spectrometer mode did not show up in the low level tests but did in the high level. Considering the spread in the high and low level test data, the theoretical results appear to give good apreement with the test data particularily below about 25 to 30 Hz. Above this frequency range the theoretical results follow the trend of the test data but is shifted higher in frequency. It therefore appears that the model is too stiff, as one would expect, as frequency increases.

From the frequency response plots the lowest mode is at 10.5 Hz which is the first S/C bending mode in the Z axis. The Y axis mode is also at 10.5 Hz. The next mode is at 15 Hz which is the first spectrometer mode. Modes in the range 30 to 40 Hz are due to S/C and SI second bending. Modal damping values of 10% critical were used for all lateral modes and was obtained from early tests on a structure using only a mass representation of the SI.

Figures 12 and 13 show data that are used to set limits on the test inputs. The design qualification bending moment is required not to exceed 26.2 kN-m (232 000 in-lb) during test. From figure 12 an input acceleration value of 0.23g must not be exceeded in order to prevent the bending moment from exceeding 26.2 kN-m (232 000 in-lb).

Figure 13 shows the relative deflection between the lowest spectrometer deck (at grid point 5007 in figure 7) and the S/C upper body truss leg which is nominally 2.2 cm (0.85 inches) away. If the input acceleration gets high enough the clearance will be exceeded resulting in banging of the deck on the S/C structure and possibly damaging of the sensitive alinement of the SI. At the 10 Hz fundamental mode the clearance will not be exceeded since the input has to be notched to 0.23g to avoid exceeding the bending moment at the separation plane. However, at 16 Hz it can be seen that the relative deflection will be exceeded if the input is above about 2.0g. This is the level which is to be input during the design qualification tests. In fact, the tests did indicate that the IUE could just barely take 2.0g input and not have banging of the spectrometer on the S/C support legs.

CONCLUSIONS

Substructuring techniques have been used to solve a problem that would probably have been too large to solve as one computer submittal on the GSFC IBM 360/95. The total CPU time required for 3 Phase I, 1 Phase II-a and II-b, and 4 Phase III-a and III-b runs was 1 1/2 hr. The results of the analysis compare favorably with test data.

APPENDIX I

DMAP ALTERS FOR MODAL AND FREQUENCY RESPONSE SUBSTRUCTURE ANALYSIS (NASTRAN LEVEL 15.5.1)

PHASE I : Create KAA,MAA for each distinct substructure (three runs) i=1 (S/C),i=2 (SI),i=3 (Solar Array)

```
SOL 3,0

ALTER 74

OUTPUT1  KAA,MAA,,,/C,N,-1/C,N,i/C,N,INPi

EXIT

ENDALTER
```

These are conventional rigid format 3 runs except that they terminate after the Guyan reduction. Any DOF that will connect to another substructure must be left in the "A" set in these runs,must be sequenced the same in all substructures and must have compatible coordinate systems.

PHASE II-a : Combine substructures and solve for eigenvalues and "A" set eigenvalues.

```
SOL 3,0

ALTER 47

PARAM    //C,N,NOP/V,N,TRUE=-1
$
$Read S/C matrices in and incorporate into overall K,M
$
INPUTT1  /K1,M1,,,/C,N,-1/C,N,1/C,N,INP1

MERGE,   ,,,K1,PV1,/KG1

ADD      KGG,KG1/KGG1

EQUIV    KGG1,KGG/TRUE

MERGE,   ,,,M1,PV1,/MG1

ADD      MGG,MG1/MGG1

EQUIV    MGG1,MGG/TRUE
```

```
$
$Read SI matrices in and incorporate into overall K,M
$
INPUTT1   /K2,M2,,,/C,N,-1/C,N,2/C,N,INP2

MERGE,    ,,,K2,PV2,/KG2

ADD       KGG,KG2/KGG2

EQUIV     KGG2,KGG/TRUE

MERGE,    ,,,M2,PV2,/MG2

ADD       MGG,MG2/MGG2

EQUIV     MGG2,MGG/TRUE
$
$Read matrices for one Solar Array and incorporate
$into overall K,M for both arrays using PV3,PV4
$
INPUTT1   /K3,M3,,,/C,N,-1/C,N,3/C,N,INP3

MERGE,    ,,,K3,PV3,/KG3

ADD       KGG,KG3/KGG3

EQUIV     KGG3,KGG/TRUE                       Solar Array No.1
                                              (use K3,M3,PV3)
MERGE,    ,,,M3,PV3,/MG3

ADD       MGG,MG3/MGG3

EQUIV     MGG3,MGG/TRUE

MERGE,    ,,,K3,PV4,/KG4

ADD       KGG,KG4/KGG4

EQUIV     KGG4,KGG/TRUE                       Solar Array No.2
                                              (use K3,M3,PV4)
MERGE,    ,,,M3,PV4,/MG4

ADD       MGG,MG4/MGG4

EQUIV     MGG4,MGG/TRUE

CHKPNT    KGG,MGG

ALTER 50,54
```

```
$
$Write partitioned modal data on tape for Phase III-a
$
OUTPUT1   LAMA,,,,//C,N,-1/C,N,4/C,N,INP4

PARTN     PHIG,,PV1/,PHIA1,,/C,N,1/C,N,1

PARTN     PHIG,,PV2/,PHIA2,,/C,N,1/C,N,1

PARTN     PHIG,,PV3/,PHIA3,,/C,N,1/C,N,1

PARTN     PHIG,,PV4/,PHIA4,,/C,N,1/C,N,1

OUTPUT1   PHIA1,PHIA2,PHIA3,PHIA4,//C,N,0/C,N,4/C,N,INP4

ENDALTER
```

The Bulk Data deck should include a dummy scalar mass and stiffness just to get SMA1 and SMA2 to generate KGG and MGG data blocks to get started. It should also contain the direct matrix input (DMI) cards for the partitioning vectors for each substructure (including the second Solar Array) and an SPOINT card specifying as many scalar points as there are "A" set DOF.

<u>PHASE II-b</u> : Restart Phase II-a for modal frequency response

```
SOL 11,0

ALTER 146,146

ALTER 156

OUTPUT1   PPF,,,,//C,N,-1/C,N,4/C,N,INP4

PARTN     UPVC,,PV1/,UPVS1,,/C,N,1/C,N,3

PARTN     UPVC,,PV2/,UPVS2,,/C,N,1/C,N,3

PARTN     UPVC,,PV3/,UPVS3,,/C,N,1/C,N,3

PARTN     UPVC,,PV4/,UPVS4,,/C,N,1/C,N,3

OUTPUT1   UPVS1,UPVS2,UPVS3,UPVS4,//C,N,0/C,N,4/C,N,INP4

CØND      LB14,NØP

ENDALTER
```

The data deck for this run is the same as any rigid format 11 restart of a rigid format 3 run. It should include all of the dynamic load data,damping and the excitation frequency list.

PHASE III-a : Recover mode shapes for each substructure (four runs) i=1 (S/C),i=2 (SI),i=3 (Solar Array No.1),i=4 (Solar Array No.2). These runs are restarts of the Phase I runs with two restarts of the Solar Array model.

```
SOL 3,0

ALTER 74

PARAM     //C,N,NOP/V,N,TRUE=-1

JUMP      LBPH3

ALTER 94

LABEL     LBPH3

INPUTT1   /LAMAS,,,,/C,N,-1/C,N,4/C,N,INP4

EQUIV     LAMAS,LAMA/TRUE

INPUTT1   /UVSUB,,,,/C,N,i-1/C,N,4

EQUIV     UVSUB,PHIA/TRUE

ENDALTER
```

The Case Control deck requests output desired and there is no additional Bulk Data.

PHASE III-b : Recover frequency response data for each substructure (four runs);i=1 (S/C),i=2 (SI),i=3 (Solar Array No.1), i=4 (Solar Array No.2). These runs are also restarts of Phase I runs

```
SOL 11,0

ALTER 46

PARAM     //C,N,NOP/V,N,TRUE=-1

ALTER 94,158

INPUTT1   /PPF,,,,/C,N,-1/C,N,4/C,N,INP4

INPUTT1   /UDV1f,,,,/C,N,i-1/C,N,4

SDR1      USETD,,UDV1F,,,GO,GM,,KFS,,/UPVC,,QPC/C,N,1/
          C,N,DYNAMICS
```

```
CHKPNT     UPVC,QPC

SDR2       CASECC,CSTM,MPT,DIT,EQEDYN,SILD,,,,PPF,QPC,UPVC,
           EST,XYCDB/OPPC1,OQPC1,OUPVC1,OESC1,OEFC1,/C,N,
           FREQ/V,N,NOSORT2

ALTER 169,170

RANDOM     XYCDB,DIT,PSDL,OUPVC2,OPPC2,OQPC2,OESC2,OEFC2,
           CASECC/PSDF,AUTO/V,N,NORD

SAVE       NORD

ALTER 181,183

ENDALTER
```

The Case Control deck requests output desired. Dummy dynamic loads damping,frequency response list should be included in the Bulk Data deck.

REFERENCES

1. Bruck, L. R.: Development of the IUE Spectrometer Finite Element Models, NASA-GSFC Memorandum Report No. 741-30.

2. Universal Analytics, Inc.: NASTRAN Substructuring Analysis Report, (to NASA-GSFC - no report number), July 9, 1974.

3. McCormick, Caleb W., ed.: The NASTRAN User's Manual (Level 15.0). NASA SP-222(01), 1973.

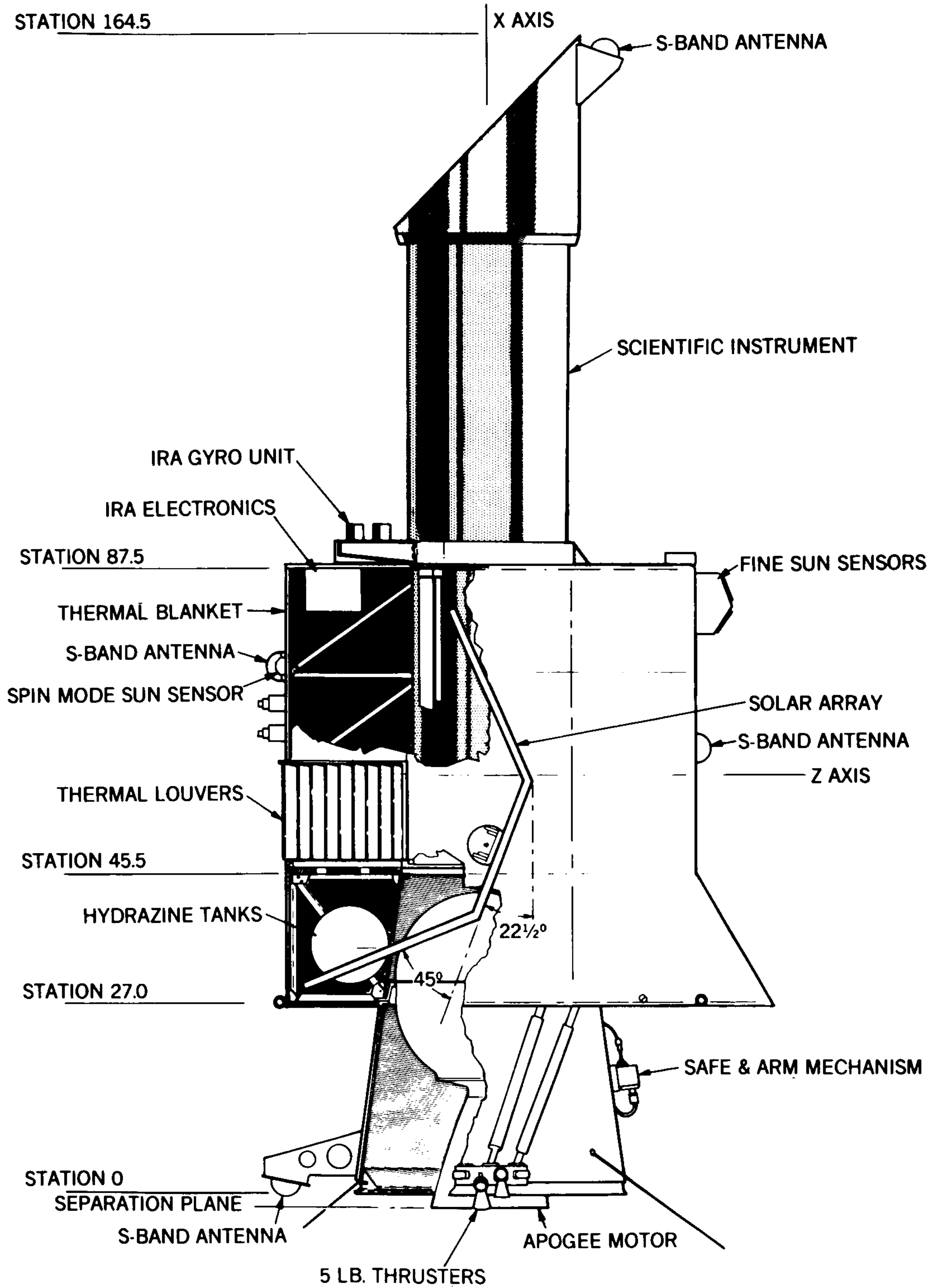

Figure 1.- IUE spacecraft with solar arrays deployed.
1 lb = 4.448 N.

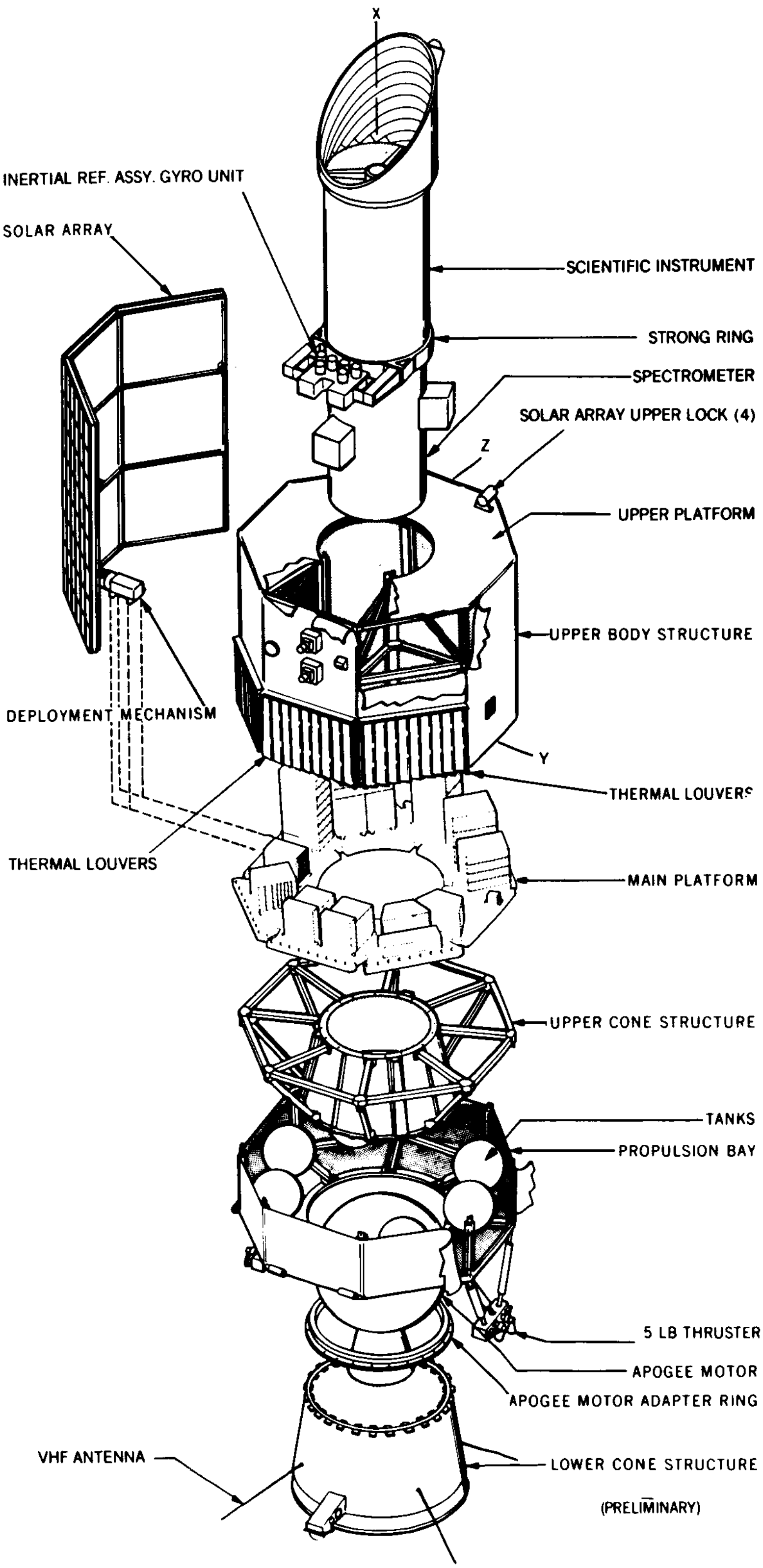

Figure 2.- Exploded view of IUE spacecraft. 1 lb = 4.448 N.

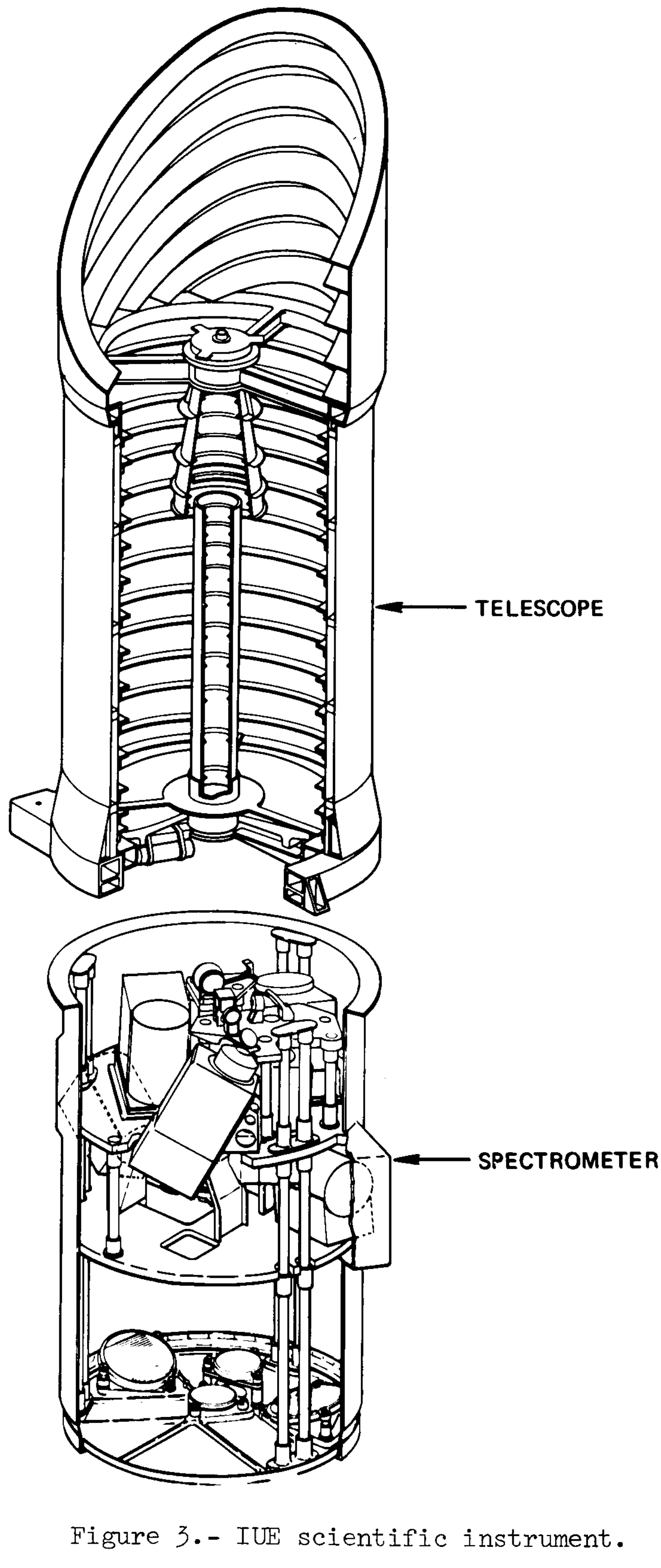

Figure 3.- IUE scientific instrument.

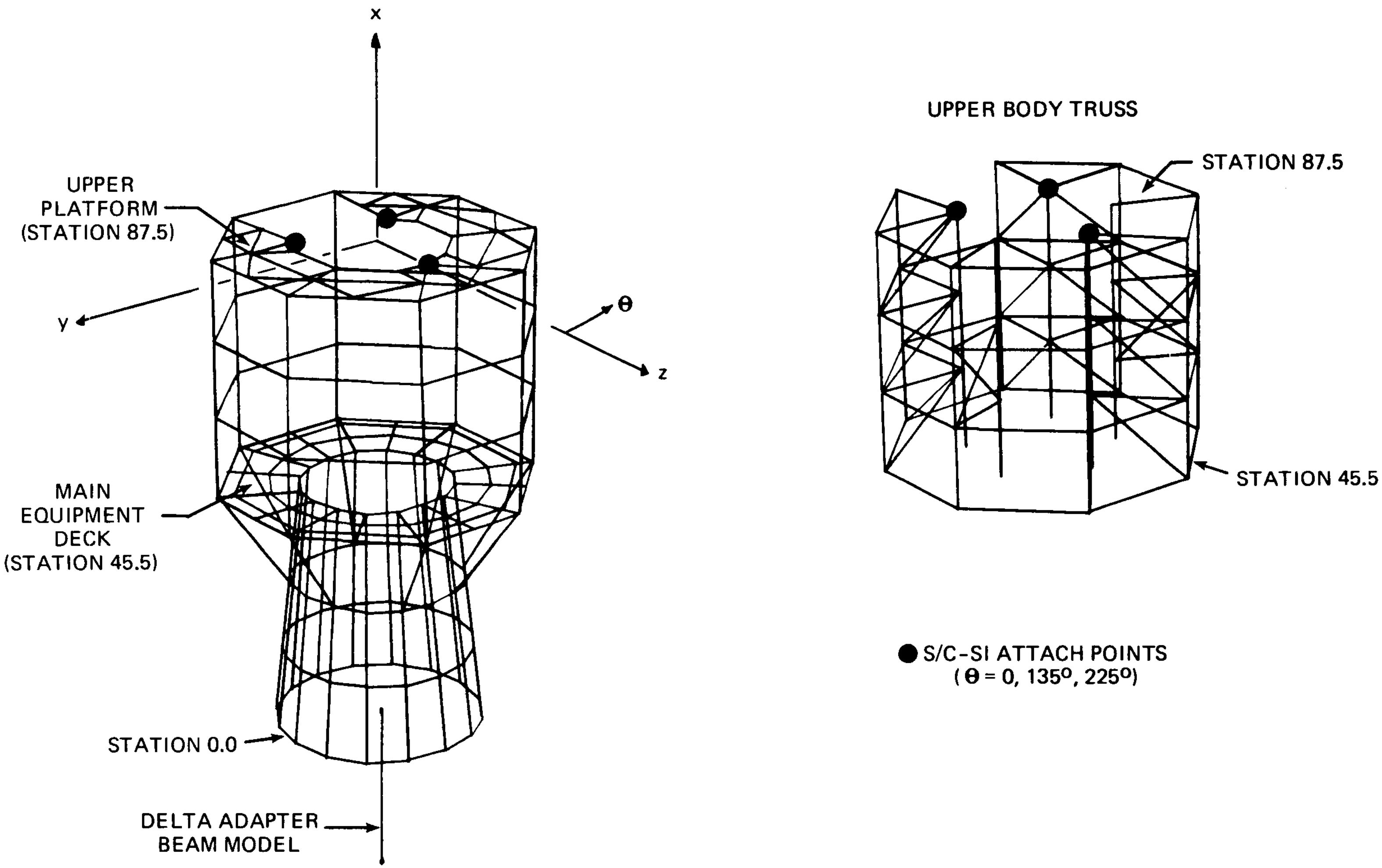

Figure 4.- Finite element model of S/C.

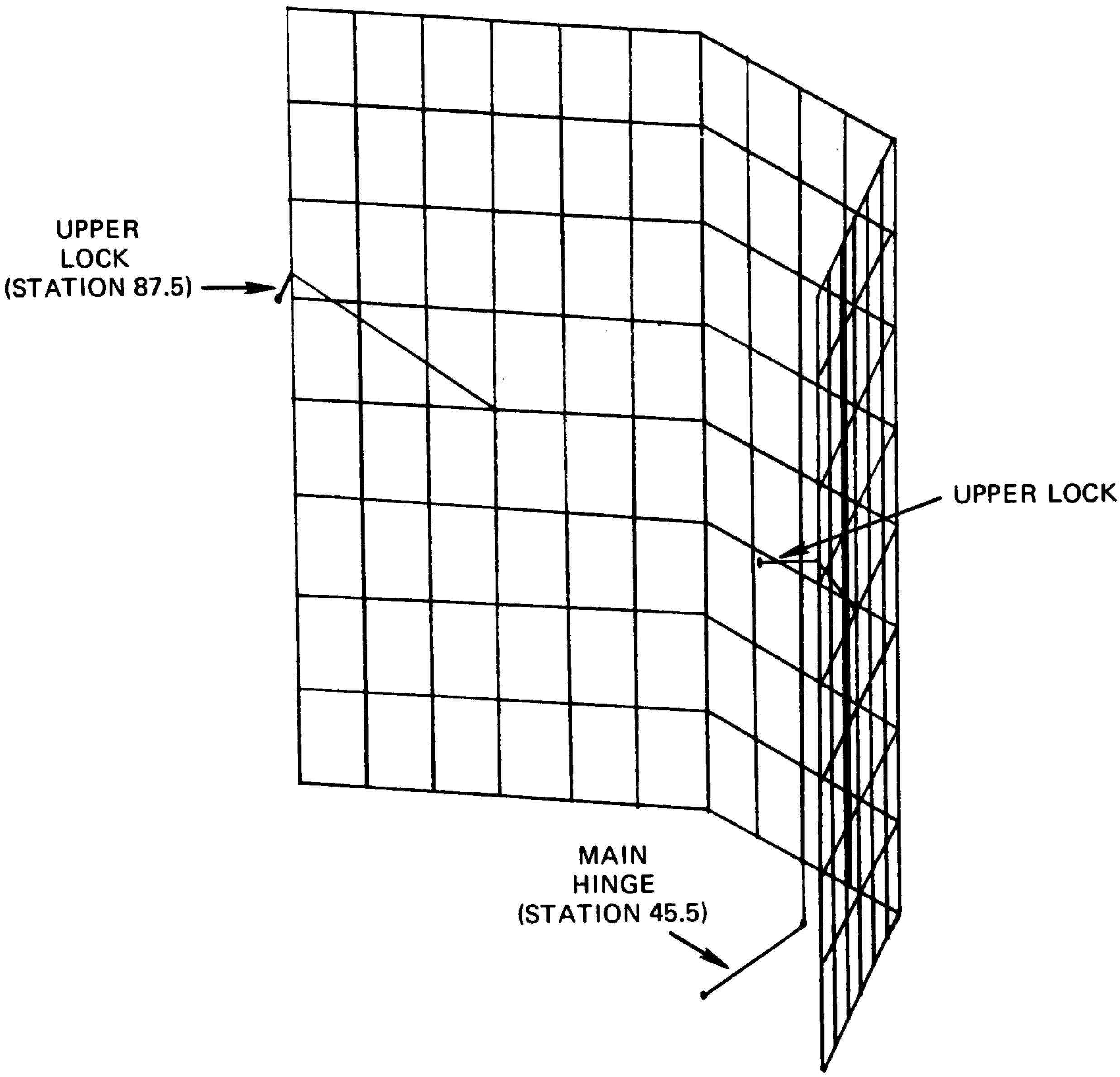

Figure 5.- Finite element model of one solar array.

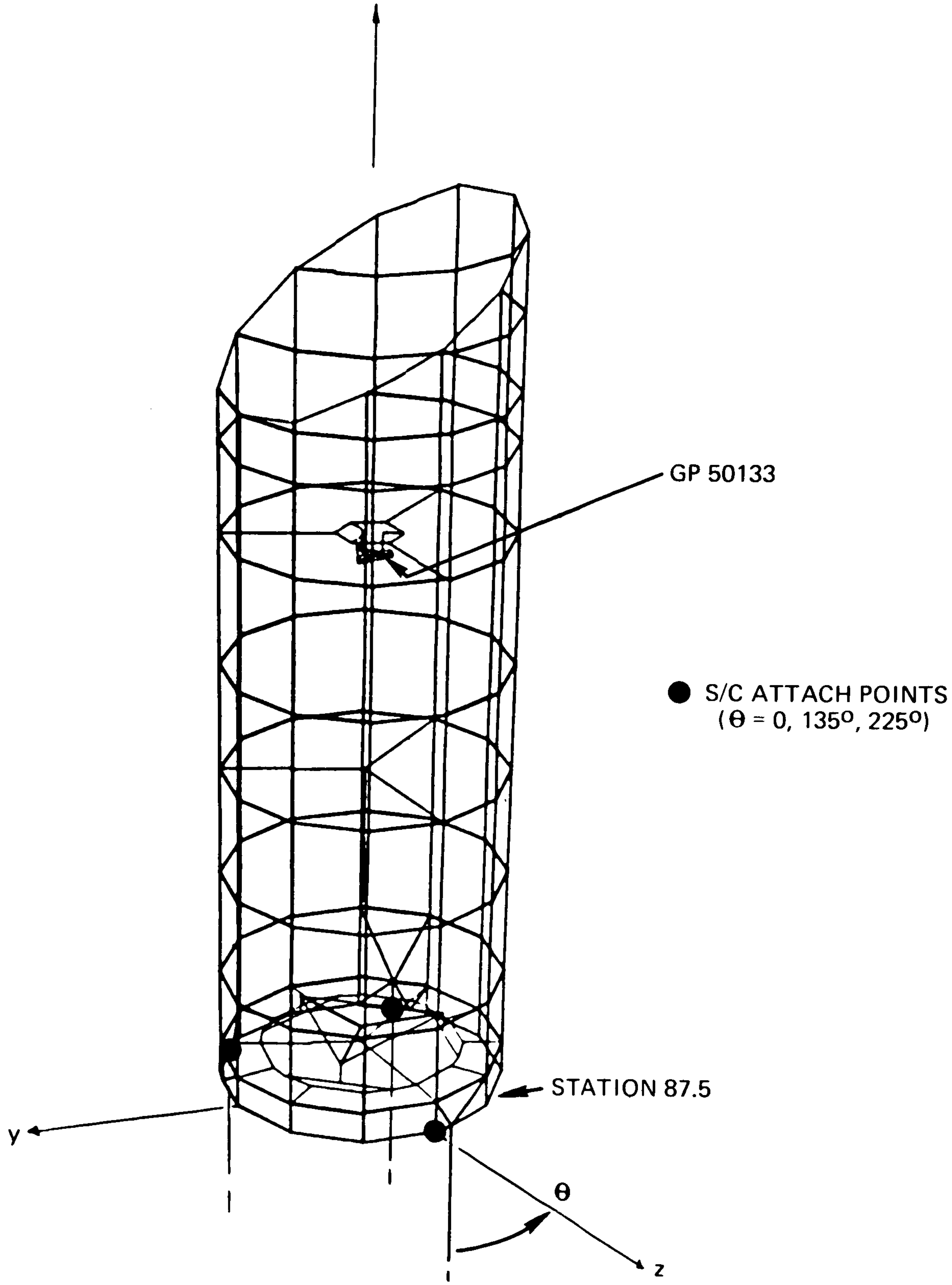

Figure 6.- Finite element model of telescope.

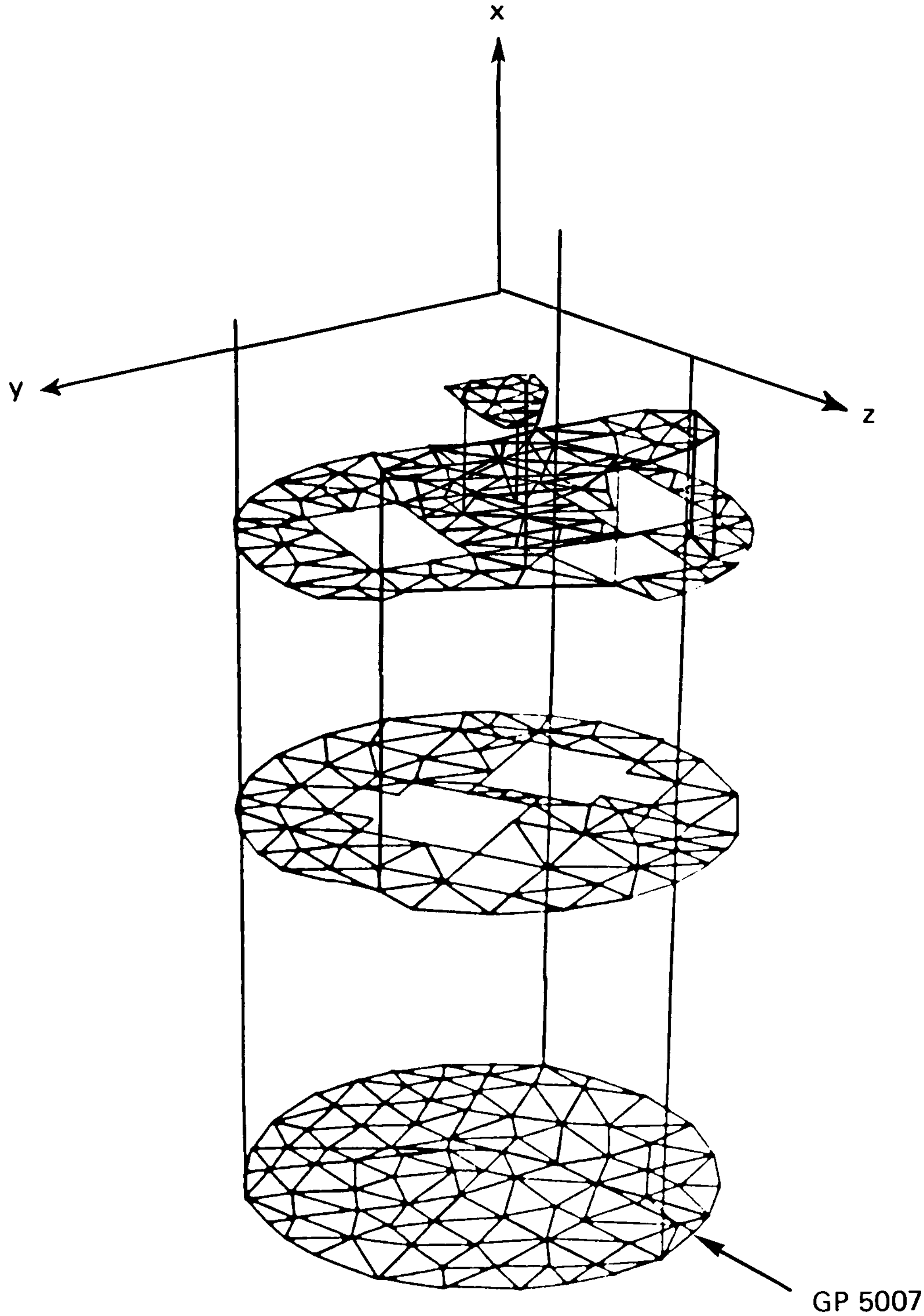

Figure 7.- Finite element model of spectrometer.

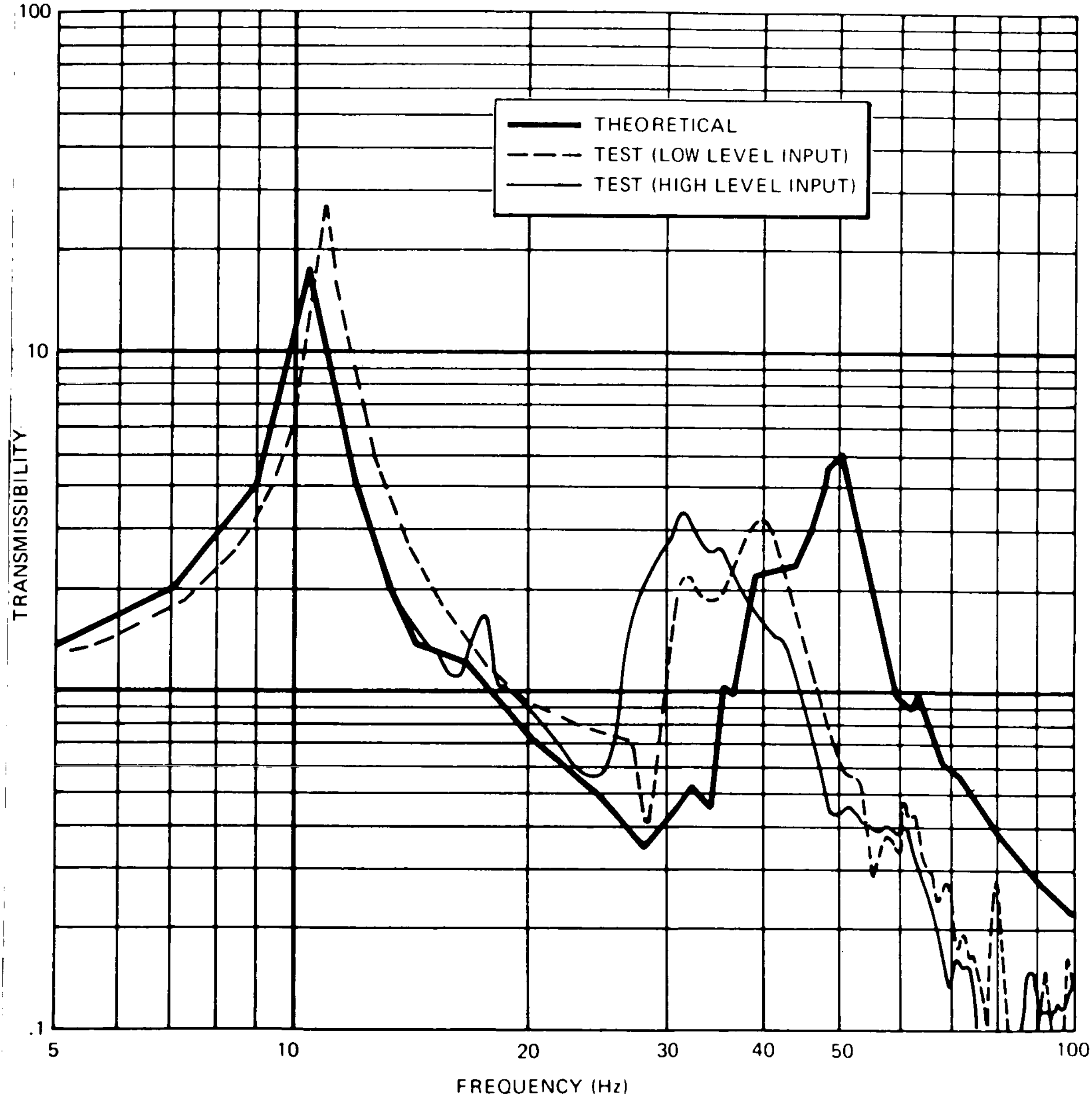

Figure 8.- Transmissibility at strong ring ($\theta = 135^{\circ}$) for Z axis vibration input to base of Delta adapter.

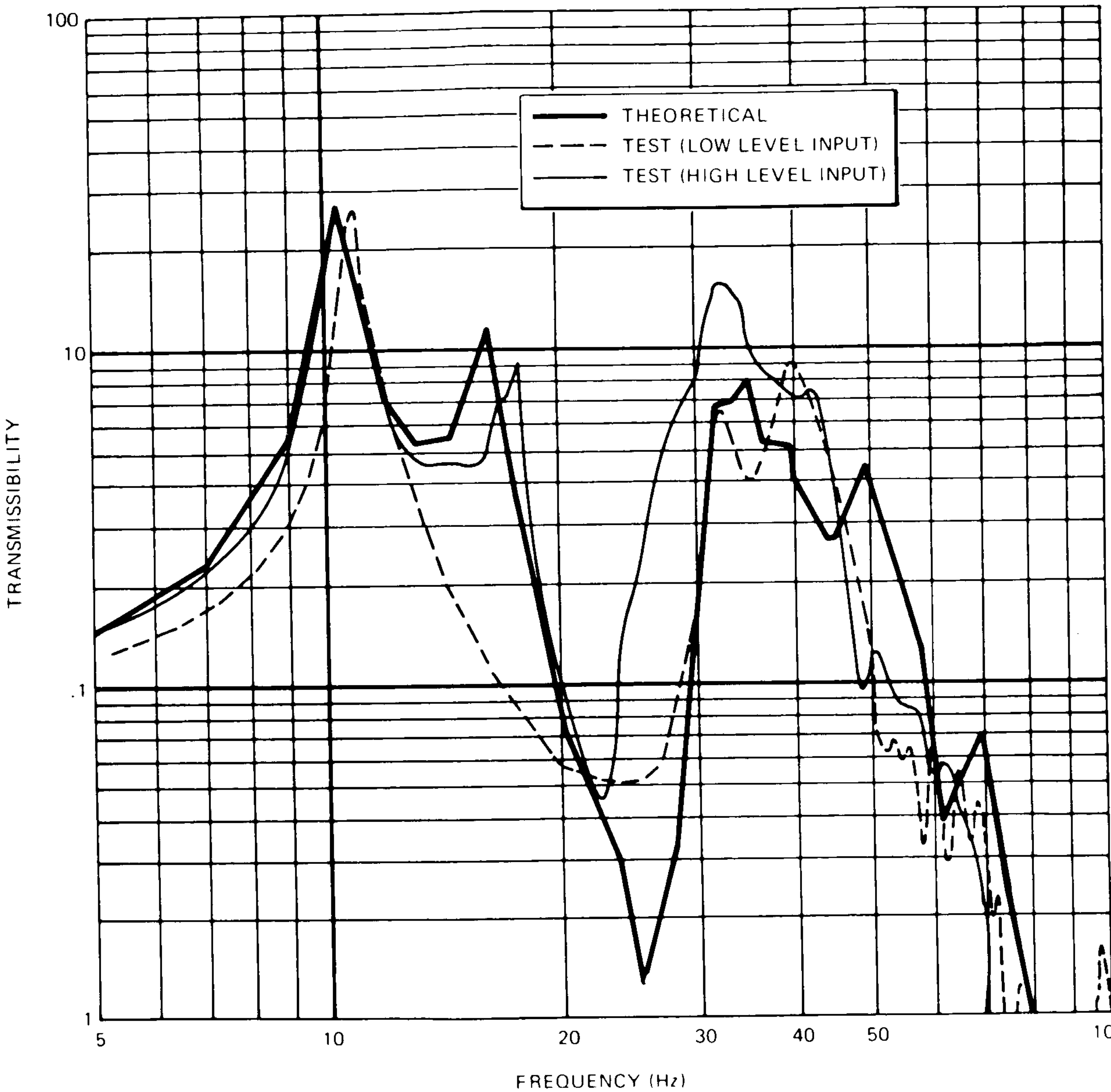

Figure 9.- Transmissibility at collimator deck (grid point 5007) for Z axis vibration input to base of Delta adapter.

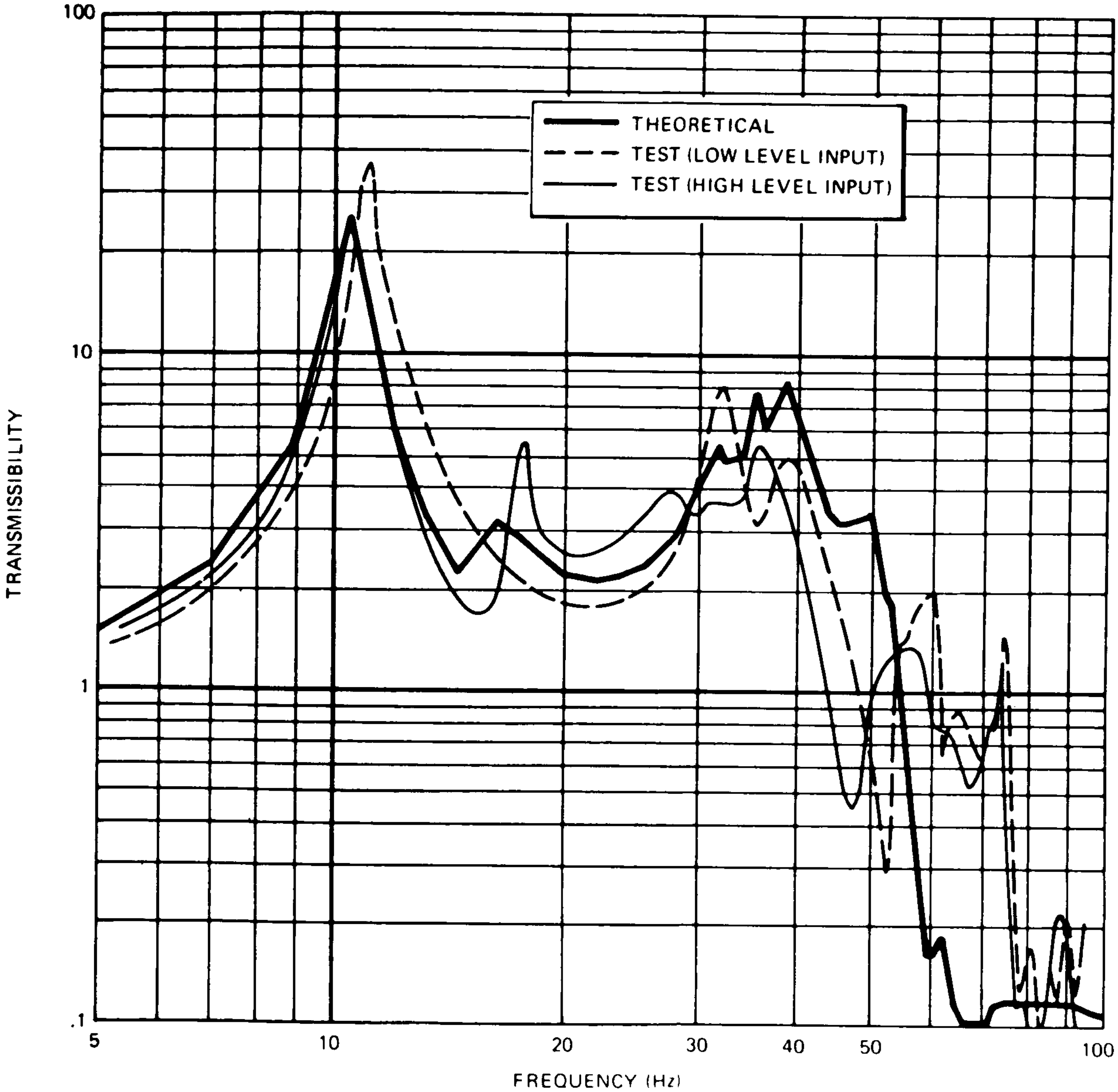

Figure 10.- Transmissibility at telescope secondary mirror (grid point 50133) for Z axis vibration input to base of Delta adapter.

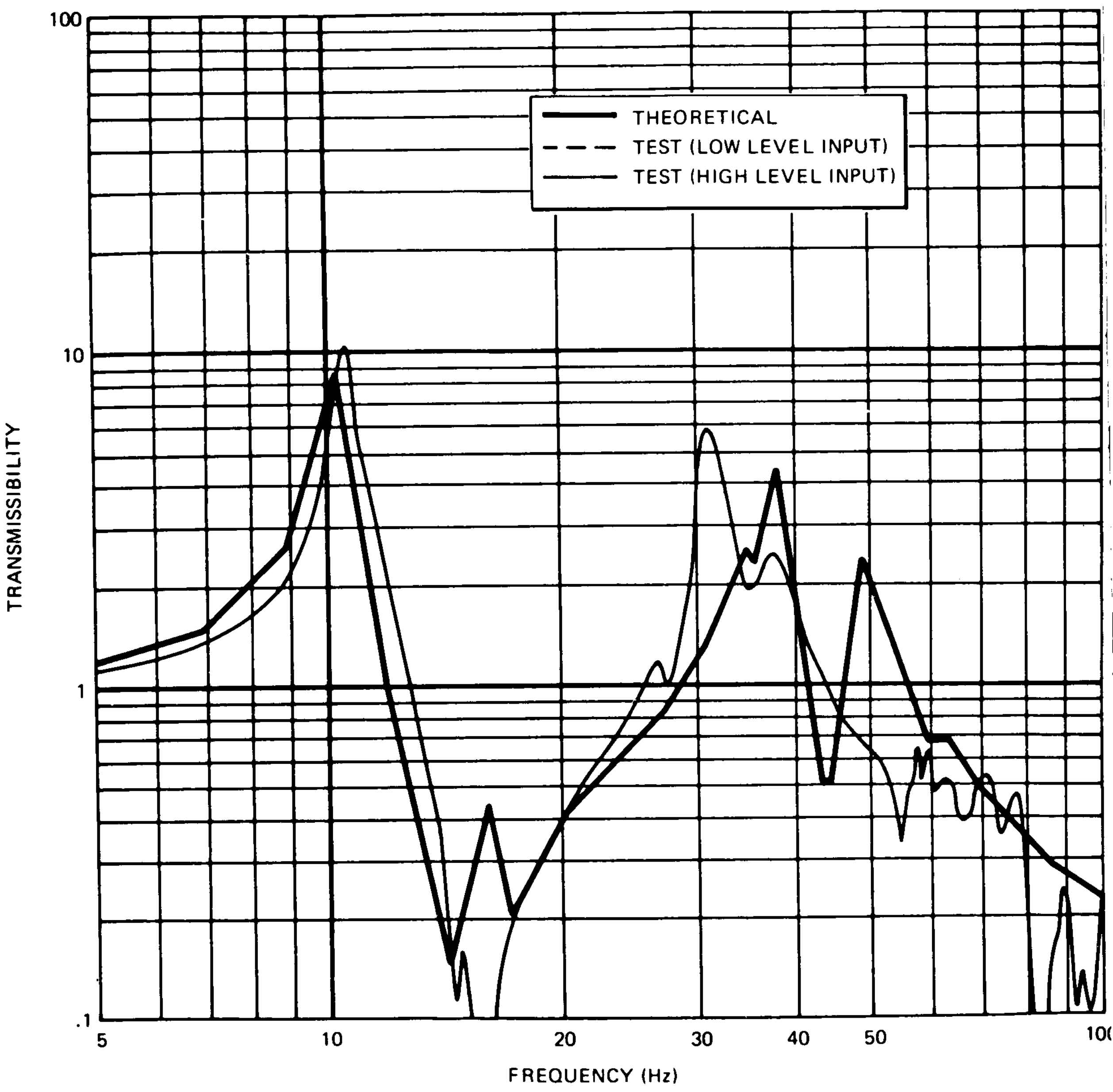

Figure 11.- Transmissibility at main deck (station 45.5) for Z axis vibration input to base of Delta adapter.

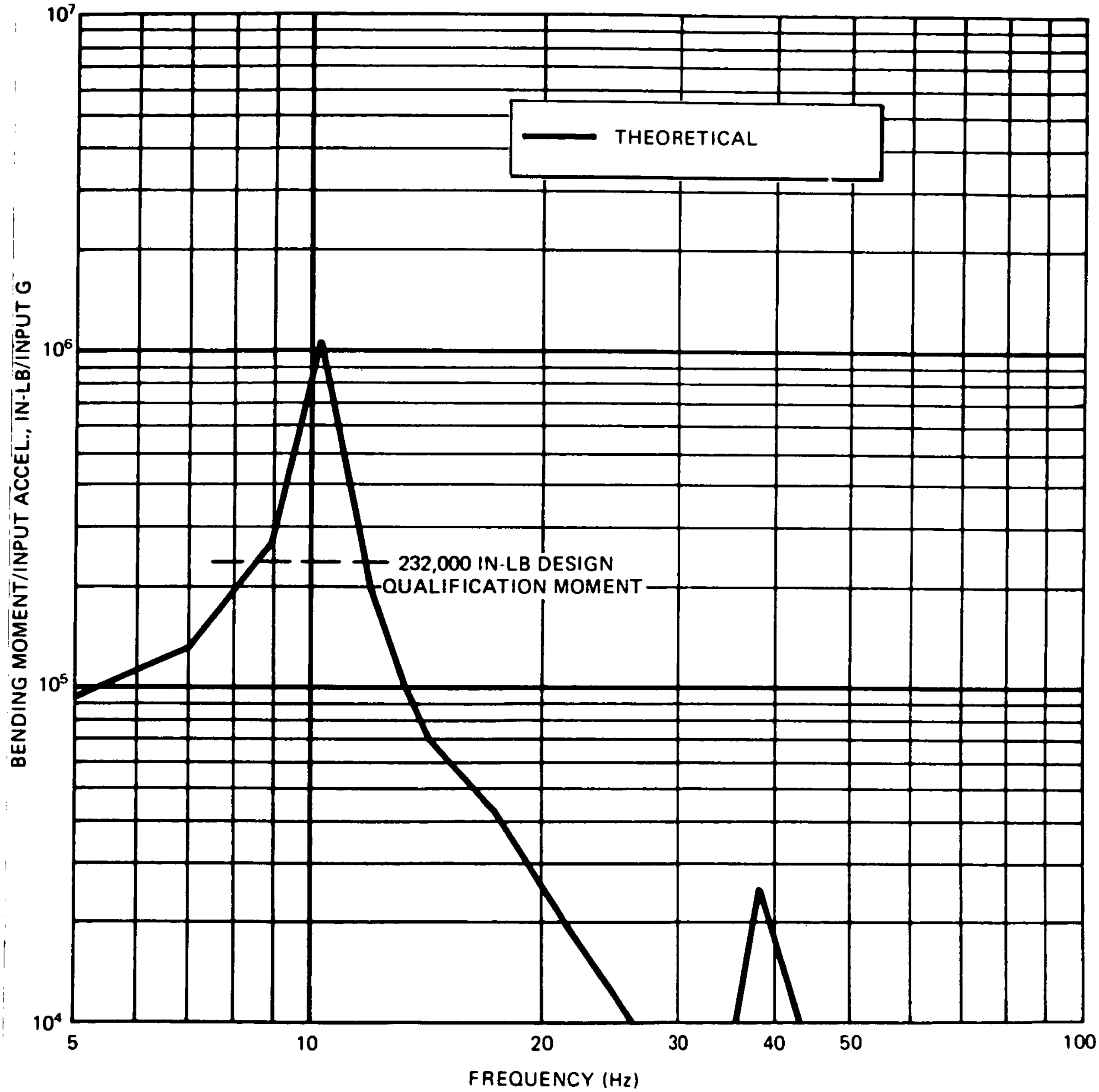

Figure 12.- Bending moment at separation plane (station 0.0) for Z axis vibration input to base of Delta adapter. 1 in-lb = 0.11298 N-m.

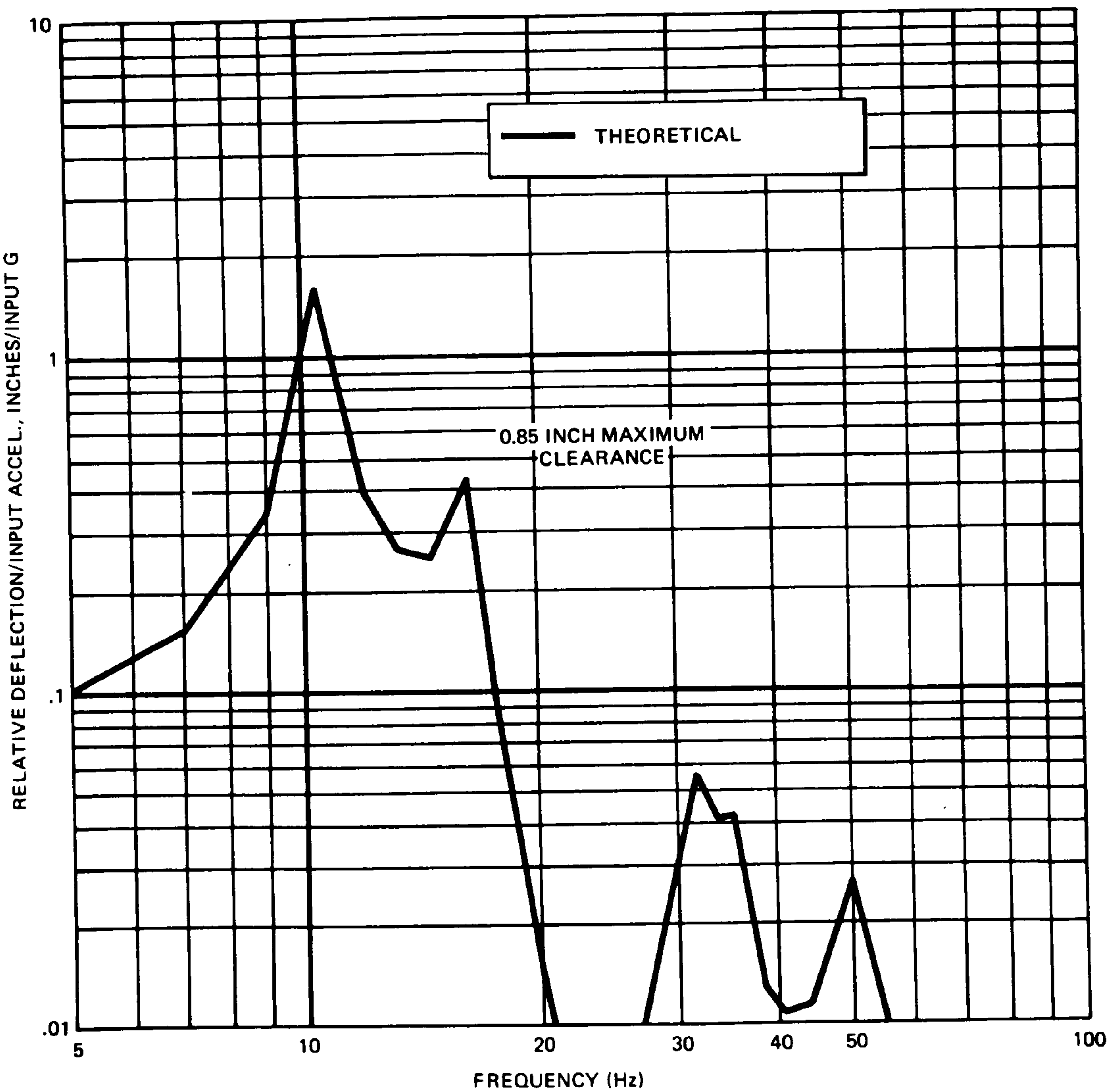

Figure 13.- Relative deflection between collimator deck and S/C support leg for Z axis vibration input to base of Delta adapter. 1 in. = 2.54 cm.