ALCATEL
SPACE

HERSCHEL / PLANCK

MECHANICAL MATHEMATICAL MODEL SPECIFICATION

HP-1-ASPI-SP-0014

Product Code: 000000

	Function Name	Date	Signature
Rédigé par/ ***Written by***	**System Mechanical Analyses** **C. VEZAIN - P. FAILLET**	2/08/01 2/08/01	[signature]
Vérifié par/ ***Verified by***	**Mechanical Thermal Propulsion Architect** **P. CLAVEL**	2/08/01	[signature]
Vérifié par/ ***Verified by***	**System Engineering Manager** **P. RIDEAU**	02/08/01	[signature]
Approbation/ ***Approved by***	**Project Manager** **J.J. JUILLET**	02/08/01	[signature]

Entité Emettrice : Alcatel Space - Cannes
(détentrice de l'original) :

CLASSES DE CONFIDENTIALITE	SOCIETE	MILITAIRE	PROGRAMME
	Non Protégé : ☐ Réservé ASPI: ☒ Confidentiel ASPI : ☐ Secret ASPI : ☐	Non Protégé : ☒ Diff. Restreinte : ☐ Confid. Défense : ☐ Secret Défense : ☐	Classe 1/Grand Public: ☐ Classe 2/Industrie : ☐ Classe 3/Diff. Restr. : ☐ Classe 4/Confidentiel : ☐
CONTRAT ou MARCHE **(Note Externe)**	**Client** -	**n° contrat ou marché** -	**Programme** HERSCHEL/PLANCK
	Document Contractuel -	**Lot** -	**Poste** -

TITRE : MECHANICAL MATHEMATICAL MODEL SPECIFICATION

Catégorie de configuration
Configuré : ☐
Non configuré : ☒

AUTEUR :
Sigle : BO/PM/IA
Nom : Patrick FAILLET / Christine VEZAIN
Signature :

Date	Réf. Interne/Ed./Rev.	Réf. Programme	Nb pages	Nb d'annexes
07/06/01		**HP-1-ASPI-SP-0014**	36	**/**

RESUME D'AUTEUR :

Ce document présente les exigences relatives aux procédures de création et d'échanges des modèles éléments finis mécaniques de tous les sous-systèmes du programme HERSCHEL/PLANCK.

Références Informatiques	Réf. du fichier : SPEC HP FEM issue 1 Nom du logiciel : WINWORD	**Langue du document** :	Anglais
		Langue faisant foi :	Anglais

ETAT DES EDITIONS

N°	DATE	RAISON DE L'EVOLUTION

MOTS CLES
HERSCHEL PLANCK
MODELES ELEMENTS FINIS
SPECIFICATION
SOUS-SYSTEMES

Catégorie de diffusion

INTERET
Court terme : ☐
Moyen et long termes : ☒

DIFFUSION SUPPLEMENTAIRE
Autorisée : ☒
Contrôlée : ☐

Sigle : BO/PM/IA
Nom : J. BUFFE
Signature

Sigle : BO/PM/I
Nom : J.M.LEBLANC
Signature :

CHANGE RECORDS

ISSUE	DATE	§ : CHANGE RECORD	AUTHOR
01	07/06/01	First issue	FAILLET Patrick VEZAIN Christine

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

1 SCOPE

In this document are presented the rules for the definition of the mechanical mathematical models and the rules for these model exchanges.

The mechanical mathematical models of sub-systems will be used to:

- determine the dynamic behaviour,
- validate the mechanical strength under mechanical or thermoelastic environment,
- calculate the interface loads between each sub-structure.

Each supplier will provide physical and condensed models. Modal models are not applicable for Herschel and Planck satellites.

This document is applicable for all the finite elements models used during the program development, i.e. from the phase B with initial models up to the phase C/D with the correlated ones.

2 GENERAL REQUIREMENTS [GRA.]

2.1 SOFTWARE [GRAA]

Reference HP-1SP-GRAA-001 :

The mechanical simulation of the HERSCHEL/PLANCK must be performed with the finite element code MSC/NASTRAN, version 70.

2.2 AXES AND UNITS SYSTEMS [GRAB]

Reference HP-1SP-GRAB-001 :

The basic axis system of each model is the general axis system of the spacecraft:

- -the origin is at the basis of the launcher interface
- -the Z axis :
  - for **HERSCHEL** satellite : is in the plane normal to X-axis, such that nominally the Sun will lie in the (X,Z) plane (zero Roll angle with respect to Sun). Positive Z-axis is oriented towards the Sun.
  - for **PLANCK** satellite : is such that the Planck telescope line of sight is in the (X,Z) plane. The telescope is pointing in the + Z half-plane
- -the X axis is the launch vehicle longitudinal axis
- -the Y axis forms a direct orthonormed reference.

The basic axis systems are shown in figure 1 and figure 2.

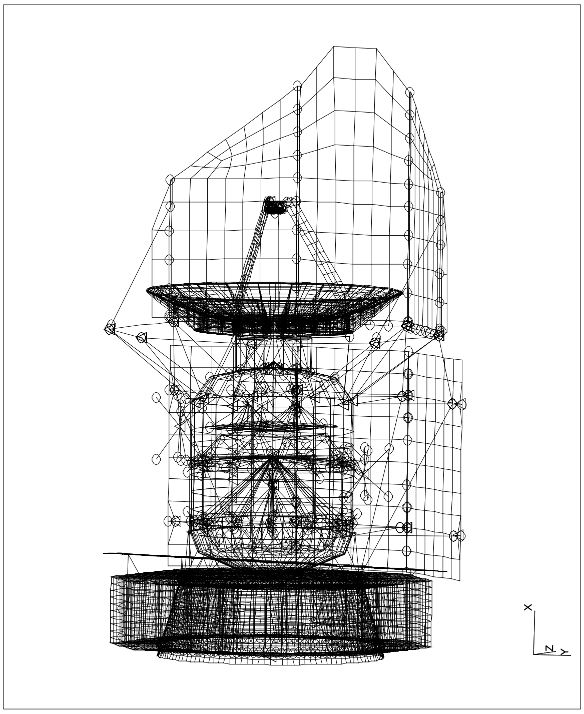

FIGURE 1 : basic axis system used for HERSCHEL

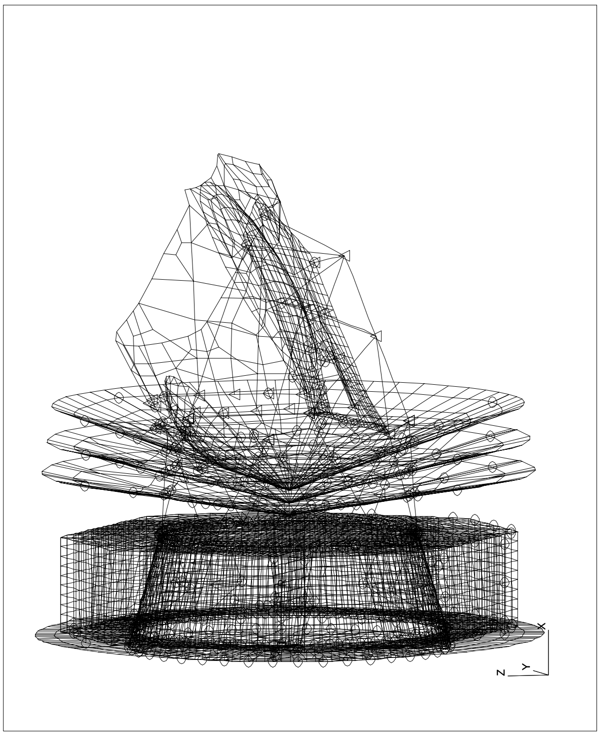

FIGURE 2 : basic axis system used for PLANCK

Reference HP-1SP-GRAB-002 :

The unit system is the International System (meter, kilogram, second, radian).

Reference HP-1SP-GRAB-003 :

Local axis systems are prohibited for the definition of the coordinates and the degrees of freedom (displacements) of all the conserved nodes (loaded and interface ones).

Reference HP-1SP-GRAB-004 :

All local axis systems must be defined with respect to the basic one, and their number limited to around 5.

2.3 RIGID BODIES [GRAC]

Reference HP-1SP-GRAC-001 :

Any rigid body or rigid element connecting **interface nodes between them** is prohibited.

Reference HP-1SP-GRAC-002 :

If an interface node is connected by a rigid element to non-interface nodes, the degrees of freedom of the interface node must be the independent (or non-constrained) ones.

2.4 DATA AND CAPABILITIES REQUESTED [GRAD]

Reference HP-1SP-GRAD-001 :

The physical finite element models shall contain all the data necessary to perform:

- static analysis
- dynamic analysis :
  - modal analysis (eigen modes calculation)
  - sine response analysis
- thermoelastic analysis (only CTE and reference temperature).

Reference HP-1SP-GRAD-002 :

According to the use of the finite element model (static, dynamic or thermoelastic) and its form (physical, condensed or modal ones), following models checking must be applied to the model:

		Finite element model purpose		
		Static analysis	Dynamic analysis	Thermoelastic analysis
Model checking	Static test	X	X	X
	Dynamic test		X	X
	Thermoelastic test			X

X means that the check must be performed by the supplier

TABLE 1: model checking according to the model purpose

Considered criteria for the model checking are considered in paragraph 2.11.

2.5 MASSES REPRESENTATION [GRAE]

Reference HP-1SP-GRAE-001 :

The masses representation (choice between concentrated masses and distributed masses) is to be defined by the supplier in order to fit as well as possible with the actual unit masses distribution. Comments cards will explain the meaning of the masses representation cards.

Reference HP-1SP-GRAE-002 :

In case of distributed mass, the following data must be detailled:

- structural mass value (kg/m, kg/m2 or kg/m3 depending on the element)
- non structural mass values (kg/m, kg/m2 or kg/m3 depending on the element)
- total mass value (kg/m, kg/m2 or kg/m3 depending on the element).

Reference HP-1SP-GRAE-003 :

The rigid mass of the model must have the same value along the 3 axes.

2.6 ASSOCIATED DATA FOR EACH STRUCTURAL CHARACTERISTIC [GRAF]

Reference HP-1SP-GRAF-001 :

The finite element models must be take into account the following standard characteristics:

- Geometrical characteristic : nominal values
- Material characteristic : average values
- Masses characteristic : maximal values
- Interface characteristic : closest to the reality
- Boundaries conditions : respect of the specification

2.7 RESULTS OF THE PHYSICAL MODEL [GRAG]

The following data are requested:

Reference HP-1SP-GRAG-001 :

- description of the model with plots of the mesh showing **clearly** the numbering of the most important nodes (conserved, loaded, interface, ...) and elements

Reference HP-1SP-GRAG-002 :

- results of the eigen mode analysis performed under free-free boundary conditions (frequencies of the 6 rigid modes + frequencies of the 3 first elastic modes)

Reference HP-1SP-GRAG-003 :

- results of the main eigen mode analysis performed with the specified boundaries conditions (frequencies, effective masses and inertia, participation factors, plots of mode shapes) for the significant modes (up to 150 Hz).

Reference HP-1SP-GRAG-004 :

- rigid masses, inertia, centre of gravity of the model compared with the data of the real mass breakdown

Reference HP-1SP-GRAG-005 :

- results of the tests defined § 2.11., allowing to state on the acceptability of the F.E.M. with regard to static, dynamic and thermoelastic analysis requirements.

2.8 RESULTS OF THE CONDENSED MODEL [GRAH]

The results considered in this paragraph correspond to the condensed model with the specified boundaries conditions. For the free-free condensed model, only the values of the 9 first frequencies are required.

The following data are requested:

Reference HP-1SP-GRAH-001 :

- same results and plots that for the uncondensed model, excepted thermoelastic test

Reference HP-1SP-GRAH-002 :

- a table of comparison to demonstrate the agreement between the condensed and the uncondensed models, containing for both models: the frequencies, the effective masses and inertia, the participation factors up to 200 Hz at least when rigidly mounted at its interface, or a larger range till the sum of participating masses reached 95 % of the Rigid Mass.

  For all degrees of freedom delivered in the condensed model, and for modes for which effective mass is more than 5% of the rigid mass of the equipment, the eigenvector shall not differ more than 5% with respect to the uncondensed model.
  The responses to typical excitations at the base shall not differ more than 10% for these degrees of freedom on peaks.

  In the frequencies range defined above, the requested representativity for the condensed model modal characteristics with regard to the physical model ones shall be:

  $\pm$5% on frequencies, effective masses and inertia (in these cases, the effective mass will be greater than 5% of the rigid mass of the equipment)
  $\pm$10% on responses to typical excitations at the base for the degrees of freedom on peaks.

Reference HP-1SP-GRAH-003 :

For the tanks, the model will describe the sloshing movement of the free surface of each tanks under 1g acceleration.

2.9 PLOTEL CARDS PACKAGE [GRAJ]

Reference HP-1SP-GRAJ-001 :

Plotel elements connecting the conserved nodes are required to plot the undeformed and deformed structures with sufficient level of representativity.

Reference HP-1SP-GRAJ-002 :

The identification number of the Plotel cards will be taken between the same limits that for the conserved nodes for each unit.

Reference HP-1SP-GRAJ-003 :

To make more representatives the plots of the substructure, some conserved nodes with the 6 degrees of freedom blocked may be used. They must fulfil all the requirements of the conserved nodes.

2.10 CHECK OF THE DELIVERED MODELS [GRAK]

Reference HP-1SP-GRAK-001 :

The supplier shall verify the following models checking according to the purpose of the finite element models as described in table 1.
The results of these tests shall be included in the official document delivered with the finite element model.

Reference HP-1SP-GRAK-002 :

All these tests must be performed on the condensed matrices or on the physical model **re-read on the delivered tape**.

All the models not in accordance with these tests will be rejected.

2.10.1 Static test

Reference HP-1SP-GRAK-003 :

The static test verification will be performed using a unitary acceleration (1 g) along each of the three axes. The interface boundaries conditions will be representative of the kind of fixation between the relevant sub-system and the primary structure. The resultant interface loads [Ri] must be equal to the sub-system mass [M] given by NASTRAN multiplied by 1g.

$$\sum_{i/f} \vec{R}_i = M \times \vec{g}$$

Reference HP-1SP-GRAK-004 :

The analysis of the reaction load must shown that these loads are presents only on nodes where are applied the boundaries conditions (use of SPCFORCES=ALL card).

Reference HP-1SP-GRAK-005 :

The verification of the sub-system mass budget present in the finite element model will show:

- the respect with the specified mass,
- the coherence of the centre of gravity coordinates computed by NASTRAN (the data of each coordinate provided two times by NASTRAN must be equal),
- the mass must be equal for each axis.

2.10.2 Dynamic test

Reference HP-1SP-GRAK-006 :

The unit supplier shall verify that the stiffness matrix is well conditioned by using the following combined tests {test A + test B }.

- TEST A

  The previous modal analysis with large mass can be replaced by a constraint check performed when the condensed model is in free-free configuration. The test to be performed is to calculate the strain energy as defined below for each rigid mode Φ_R:

$$S.E. = \frac{1}{2} . [\Phi_R]^T . [K] . [\Phi_R]$$

where	$[\Phi_R]$	is a vector for one of the six rigid modes
	$[\Phi_R]^T$	the transposed vector of $[\Phi_R]$,
	[K]	is the model stiffness matrix

  A unit rigid body displacement is applied on the whole structure on the 6 DOF (3 translations and 3 rotations).

  The strain energy computed for each of these rigid body motions shall be:

$$< 10^{-3} \text{ J}$$

  This last test shall be performed without SUPORT card.

  This Strain Energy Check is used to identify constrained or grounding problems in a FEM model and ensure that the model is free-free.

  This test can be performed using NASTRAN DMAP rigid body checks, or by a specific NASTRAN DMAP.

- TEST B

  A free-free modal analysis has to be performed without large mass, without SUPORT card and without rigid interface in order to extract the six rigid modes. The highest frequency of these rigid modes divided by the first elastic mode shall be lower than 10^{-4}.

  Moreover the 3 first elastic free-free frequencies will be provided.

2.10.3 Thermoelastic test

Reference HP-1SP-GRAK-007 :

A zero stress test must be applied on the finite elements model with the following hypotheses:

- unit with isostatic boundary conditions
- coefficient of thermoelastic expansion set to 20 10^{-6} °C^{-1} (only for this test) on all material of the model
- homogeneous increase of temperature of +100°C applied to the whole model.

The maximum Von Mises stress ,σ_{max}, and the maximum rotations, θ_{max}, due to the temperature increase must satisfy :

$$\theta_{max} < 1.10^{-7} \text{ rad}$$
$$\sigma_{max} < 10^{-3} \text{ MPa}$$

Reference HP-1SP-GRAK-008 :

One of the major condition necessary to fulfil this requirement is that there will be no rigid body with length > 0 connecting 2 nodes of the primary structure of the unit.

2.11 MAGNETIC TAPE CHARACTERISTICS [GRAL]

Reference HP-1SP-GRAL-001 :

The package defined here above will be provided on one of the following magnetic data storage:

- Streamer cartridge 150 Mbytes , SGI , UNIX , tar format
- DAT 4mm (60, 90 or 120 m length) SGI , UNIX , tar format
- Floppy disk 3 "1/2 DOS formatted ASCII code
- CD-ROM DOS formatted ASCII code

The command used for the creation of the tape archive is to be delivered with the tape delivery.
If data are compressed, the uncompress software must be provided on the magnetic tape .

2.12 ASSOCIATED DOCUMENTATION [GRAM]

The technical note delivered with the finite element model shall include all the items mentioned in chapter 2:

Reference HP-1SP-GRAM-001 :

- the results of the non condensed model (§ 2.7),
- the results of the condensed model (§ 2.8),
- the results of the model checking (§ 2.11),

Reference HP-1SP-GRAM-002 :

- plots of each substructure showing clearly the numbering of the main important nodes and elements of the physical model,
- plots performed with the PLOTEL cards showing the conserved nodes,
- a scheme showing the various local axis systems with respect to the basic one,

Reference HP-1SP-GRAM-003 :

- a description of each substructure and of the way used for the modelization,
- an explanation of the modelization hypotheses and of the equivalent representations,

Reference HP-1SP-GRAM-004 :

- the detailed mass breakdown of the model compared with the real one and indications of:
  - the representation of the masses: concentrated or distributed,
  - the considered offsets and rigid bodies between the masses and the structure,
  - a table summarising the main structural characteristics of each substructure according to the following example:

SUB - STRUCTURE							
Component	**NASTRAN Elements type**	**Material**				**Geometric Characteristics**	
		Name	PMAT	Young Modulus	Volumic mass	Property cards	t, h, S, Iy, Iz, J

* for the non isotropic material, the local axis systems will be plotted

TABLE 2: Summarise of the main substructures characteristics

Reference HP-1SP-GRAM-005 :

- a summary of the tape contents.

2.13 PLANNING OF FINITE ELEMENT MODELS DELIVERY [GRAN]

Reference HP-1SP-GRAN-001 :

The planning of the finite element models delivery proposed to the supplier is the following:

- Delivery of preliminary models as soon as possible during the subsystems phase B
- Delivery of more consolidated models with a due date at subsystem PDR [**TBC**]
- Delivery of detailed models with a due date 1 month before the subsystem CDR [**TBC**]

- Delivery of correlated models with a due date 2 months after the subsystem qualification test [**TBC**]

Each delivery will be composed of physical or condensed static models.

3. SPECIFIC REQUIREMENTS FOR THE HERSCHEL/PLANCK MODELS [RO..]

3.1 PHYSICAL MODELS [ROAA]

3.1.1 General

Reference HP-1SP-ROAA-001 :

The authorised NASTRAN elements are provided in the table hereunder.

Item	connectivity card	property card	material card
1D elements	CROD CBAR CBEAM	PROD PBAR PBEAM	MAT1
2D elements	CTRIA3 CQUAD4	PSHELL	MAT1 , MAT2 , MAT8
		PCOMP	MAT1 , MAT8
3D elements	CPENTA CTETRA CHEXA	PSOLID	MAT1 MAT9
Masses	CONM1 CONM2 CMASS2	-	-
Local stiffness and connections	CELAS1 CELAS2	PELAS	-
Rigid elements & constraints	RBAR RBE2 (*) MPC (*) RBE3 (*)	-	-
Miscellaneous	PLOTEL	-	-
NASTRAN parameter	PARAM AUTOSPC YES	-	-

(*) see additional requests, next page

TABLE 3: Authorised NASTRAN elements

Reference HP-1SP-ROAA-002 :

The following NASTRAN cards are to be prohibited:

NASTRAN prohibited cards	
NASTRAN parameters *	PARAM BAILOUT PARAM K6ROT PARAM MAXRATIO PARAM EPZERO
NASTRAN parameters **	PARAM WTMASS
NASTRAN cards	CQUAD8, CQUADR, CTRIAR CTRIA6, EGRID, DMIG

* parameters affecting model conditioning

** parameters affecting the other models during FEM assembly

TABLE 4: Prohibited NASTRAN elements

In case of necessity to use other cards than the authorized elements, the supplier will have to ask for the agreement of ALCATEL SPACE.

Additional requests concerning FEM rules:

Reference HP-1SP-ROAA-003 :

- For thermoelastic analyses, it is requested to use elements RBE2 with zero length and MPC with zero length

Reference HP-1SP-ROAA-004 :

- The use of MPC card is forbidden to link interface nodes between substructures. The interface shall be performed with zero length CELAS or RBE2.

Reference HP-1SP-ROAA-005 :

- The interface nodes do not have to be dependent nodes of rigid bodies.

Reference HP-1SP-ROAA-006 :

- The interface nodes do not have to be linked together by a rigid body.

Reference HP-1SP-ROAA-007 :

- Only RBE3 with simply supported independent nodes will be allowed (independent nodes : DOF 123 (456 forbidden) , reference node : no restriction on dependent DOF).
  RBE3 elements will be only used in order to obtain displacements or accelerations data, loads and masses will be not introduced by RBE3.

3.1.2 Size limitation

Reference HP-1SP-ROAA-008 :

The size limitation of each physical model is given in the following table:

- HERSCHEL

UNIT	SIZE LIMITATION	
	ELEMENTS	NODES
SVM	10000	10000
CRYOSTAT	3000	3000
STRUTS	50	50
TELESCOPE	5000	5000
SSH/SSD	2000	2000
HEAT SHIELD	500	500

- PLANCK

UNIT	SIZE LIMITATION	
	ELEMENTS	NODES
SVM	10000	10000
CRYOSTRUCTURE (BAFFLE)	1000	1000
CRYOSTRUCTURE (V-GROOVE)	1000	1000
CRYOSTRUCTURE (STRUTS SUPPORT)	100	100
SOLAR ARRAYS	1000	1000
TELESCOPE	3000	3000

TABLE 5: Size limitation of each physical model

These size limitations are **TBC**

3.1.3 List of the data to be supplied

Reference HP-1SP-ROAA-009 :

The following item will be provided on the magnetic tape characterised in § 2.12.:

- the complete Bulk Data Deck **gathered in one file**.

The Bulk Data Deck will be built in order that all the cards defining the same substructure will be gathered together in the file. Enough comments will be added to make easy the understanding of the file.

3.2 CONDENSED STATIC MODELS [ROAB]

3.2.1 General

The nodal points and degrees of freedom will be defined as follows:

Reference HP-1SP-ROAB-001 :

- the degrees of freedom will be related to nodal points (6 degrees of freedom maximum per nodal point ordered as follows: T_x, T_y, T_z, R_x, R_y, R_z - T for translation and R for rotation),

Reference HP-1SP-ROAB-002 :

- nodal points will be supplied in ascending numerical order,

Reference HP-1SP-ROAB-003 :

- nodal points coordinates will be supplied according to the HERSCHEL/PLANCK reference axes system (see § 2.1).
  Local coordinates systems are not acceptable.

Reference HP-1SP-ROAB-004 :

- nodal points must be kept at the location of the accelerometers foreseen for the sine test vibrations

Reference HP-1SP-ROAB-005 :

- nodal points must be chosen in order to plot the deformed shapes of the structure with a sufficient representativity .

These rules will determine the numbering of the rows and columns of the mass, stiffness and damping (if supplied) matrices.

3.2.2 Size limitation

Reference HP-1SP-ROAB-006 :

The maximum size of the stiffness, mass and damping (if supplied) matrices including the interface degrees of freedoms is given in following table:

- HERSCHEL

	SIZE LIMITATION	
UNIT	ELEMENTS	NODES
SVM	1000	1000
CRYOSTAT	300	300
TELESCOPE	500	500
SSH/SSD	200	200
HEAT SHIELD	50	50

- PLANCK

	SIZE LIMITATION	
UNIT	ELEMENTS	NODES
SVM	1000	1000
CRYOSTRUCTURE (BAFFLE)	100	100
CRYOSTRUCTURE (V-GROOVE)	100	100
CRYOSTRUCTURE (STRUTS SUPPORT)	50	50
SOLAR ARRAYS	100	100
TELESCOPE	300	300

TABLE 6: Size limitation of each condensed model

These maximum degrees of freedom are **TBC**

3.2.3 List of the data to be supplied

Reference HP-1SP-ROAB-007 :

The following items will be provided on the magnetic tape characterised in § 2.12.:

- the ASET1 cards package,
- the DMI partitioning vector,

  the partitioning vector to expanse the condensed matrices (size nc x nc) to the physical matrices (size nt x nt) under the form of a column vector (DMI NASTRAN vector) where:
  n_c is the number of conserved degrees of freedom
  n_t is the number of conserved nodes multiplied by 6.

- the conserved GRID cards,

  the conserved GRID cards package: each conserved node shall be present on a GRID card.
  All the degrees of freedoms of the conserved nodes that do not appear in the matrices are to be permanently constrained to zero directly on the GRID card.
  The conserved nodes (or degrees of freedom) mean the loaded nodes (or degrees of freedom) plus the interface ones.
  If the interface nodes are loaded by any mass or inertia, that shall be clearly indicated.

- the PLOTEL cards,
- the stiffness, mass and damping (if supplied) matrices in NASTRAN format OUTPUT4 option BCD non sparse, with D23.16 format for version 70 and following.

eventually

- if a resequencing process is used for the creation of the condensed matrices, it is required:
  - the SEQP cards
  - a table of correspondence between the nodes and the degrees of freedom after and before the resequencing.

3.3 MODAL MODELS [ROAC] (NOT APPLICABLE)

3.3.1 General

Reference HP-1SP-ROAC-001 :

The dynamic behaviour of the structure is described by the reduced stiffness and mass matrices, relative to the elastic cantilevered modes and rigid body modes of the structure.
The motion of the structure is represented as a superposition of the rigid body and elastic cantilevered motions.
The rigid body motion is represented by the six rigid body modes shapes referenced to unity at the structure interface. The elastic motion is represented by the elastic cantilevered structure interface modes shapes.

Thus:

$$(X) = \begin{pmatrix} \Phi_{e_{(p x n)}} & \Phi_{R_{(p x 6m)}} \\ 0_{(6m x n)} & I_{(6m x 6m)} \end{pmatrix} \begin{pmatrix} q \\ X_i \end{pmatrix} \quad \begin{matrix} \text{cantilevered modal coordinates} \\ \text{payload interface motion} \end{matrix} \qquad (1)$$

where:

n	is the number of elastic modes
m	is the number of nodes
p	is the number of degrees of freedom of the source model matrices
Φ_R	are the rigid body modes shapes
Φ_e	are the elastic cantilevered modes shapes (normalised such that $\Phi_e{}^t M \Phi_e = I^*$)
X	is the motion of the structure degrees of freedom

Using the above formulation, the modal equations of motion are:

$$(M_{GEN})\ddot{Q} + (B_{GEN})\dot{Q} + (K_{GEN})Q = F_{GEN}$$

where:

$$Q = \begin{pmatrix} q \\ X_i \end{pmatrix}$$

* A^t for transposed A and I for identity matrix.

or in partitioned formulation:

$$\begin{pmatrix} \mu & M_{eI} \\ M_{eI}^{t} & M_{I} \end{pmatrix} \ddot{Q} + \begin{pmatrix} 2\mu\xi\omega & 0 \\ 0 & B_{I} \end{pmatrix} \dot{Q} + \begin{pmatrix} \mu\omega^{2} & 0 \\ 0 & K_{I} \end{pmatrix} Q = \begin{pmatrix} 0 \\ F_{I} \end{pmatrix}$$

where:

K_I	is the condensed stiffness matrix at interface
M_I	is the condensed mass matrix at interface
B_I	is the condensed damping matrix at interface
M_{eI}	is the elastic coupling mass matrix
$(\mu) = \Phi_e^t M \Phi_e$	is the generalised mass matrix (normalised to unity)
$(2\mu\xi\omega) = (2\xi\omega)$	is the generalised damping matrix
$(\mu\omega^2) = (\omega^2)$	is the generalised stiffness matrix
(F_I)	are the structure interface loads.

Reference HP-1SP-ROAC-002 :

The size of M_{GEN}, B_{GEN} and K_{GEN} matrices is N rows by N columns where N is the number of elastic modes increased of the six interface rigid body degrees of freedom.
These matrices must be diagonal and elastic modes with no modal mass are forbidden.

3.3.2 Restitution matrices

For modal data delivery, displacement restitution matrix is necessary to provide structure internal responses. This matrix provides analytical relationship between the internal responses and the modal generalised parameters. This matrix must be a subset of the transformation matrix (see equation (1)) used to reduce the mass and stiffness matrices.

The equation is:

$$[X] = [DTM][Q]$$

Reference HP-1SP-ROAC-003 :

The restitution matrix must contain at least:

- Nodes at the location of the accelerometers foreseen for the sine test vibrations
- Nodes allowing to plot the deformed shapes of the structure with a sufficient representativity
- Any other nodes considered as important by the supplier.

3.3.3 Matrices size and output requirement limitations

Reference HP-1SP-ROAC-004 :

The number of restitution parameters must be less or equal to the number of degrees of freedom allowed for static condensed model and given in § 3.2.2.

Reference HP-1SP-ROAC-005 :

The delivered modal model will contain internal nodes representative to the main parts of the subsystem in order to allow the exploitation of dynamic responses inside of modal model (Notching possibilities).

Reference HP-1SP-ROAC-006 :

For modal model, only the modes having their effective mass (or inertia) greater than 0.5 % of the total subsystem mass will have to be retained in model delivery.

3.3.4 List of the data to be supplied

Reference HP-1SP-ROAC-007 :

The following items will be provided on the magnetic tape characterised in § 2.12:

- the list of dummy nodal points (see remark below),
- the list of dummy degrees of freedom (see remark below),
- the list of restitution degrees of freedom , if any (SECSET1,…)
- the DMI partitioning vector,

  the partitioning vector to expanse the condensed matrices (size nc x nc) to the physical matrices (size nt x nt) under the form of a column vector (DMI NASTRAN vector) where:
  n_c is the number of modes + the restituted degrees of freedom
  n_t is the number of global nodes multiplied by 6.

- the restitution GRID cards,

  the conserved GRID cards package: each conserved node shall be present on a GRID card.
  All the degrees of freedoms of the conserved nodes that do not appear in the matrices are to be permanently constrained to zero directly on the GRID card.
  The conserved nodes (or degrees of freedom) mean the loaded nodes (or degrees of freedom) plus the interface ones.
  If the interface nodes are loaded by any mass or inertia, that shall be clearly indicated.

- the PLOTEL cards,
- the stiffness, mass , damping (if supplied) and restitution [DTM] matrices in NASTRAN format OUTPUT4 option BCD non sparse with D23.16 format for version 70 and following,
- the displacement restitution matrix format OUTPUT4.

eventually

- if a resequencing process is used for the creation of the condensed matrices, it is required:
  - the SEQP cards
  - a table of correspondence between the nodes and the degrees of freedom after and before the resequencing.

<u>Remark:</u>

To allow provision of modal model in the same way as static condensed model, definition of dummy nodal points and dummy degrees of freedom is necessary.

Reference HP-1SP-ROAC-008 :

These points and degrees will de defined as follows:

- n nodal points (with 1 degree of freedom) associated to the n elastic modes (numbered in the range defined in § 5.1). For each nodal point, the coordinates are (1., 0., 0.).
- m nodal points (with 6 degrees of freedom) associated to each structure interface point. Their number have to be greater than n and must be chosen in the range given in § 5.1. Their coordinates are defined in chapter 5.

The defined number of degrees of freedom will be the same as the reduced mass and stiffness matrices size (N).

4. REQUIREMENTS FOR CORRELATED MODELS [RCM.]

4.1 PURPOSE [RCMA]

The purpose of the correlated models is to perform the System dynamic analyses to prepare the HERSCHEL/PLANCK satellite sine tests.

Reference HP-1SP-RCMA-001 :

The models must be representative of the last definition of the hardware and of the tests results. For the correlation with the tests the goals are for the significant modes and mainly for the first ones :

- ≤ ± 5 % on the frequencies
- ≤ ± 5 % on the accelerations

Reference HP-1SP-RCMA-002 :

For this delivery:

- the physical F.E.M. are requested,
- a comparison between the test results and the test predictions of the correlated model has to be provided.

4.2 COMPARISON BETWEEN PREDICTIONS AND TESTS [RCMB]

Reference HP-1SP-RCMB-001 :

The subcontractors of the different subsystems shall provide for each main mode and:

- for the point with the highest response in the correlated F.E.M.,
- for instrumented point with the highest response during the test of each substructure,

the comparison between the frequencies and amplifications measured during the low level runs and the ones predicted by the correlated model.

MODE	DESCRIPTION	MODEL predictions			TEST measures		
		frequency (Hz)	Q1 M	Q2 M	frequency (Hz)	Q1 T	Q2 T
N	Mode shape + location of G1 and G2						

TABLE 7: Comparison between model prediction and test measures

With:

N	: order of the model mode the closest (in term of frequency and mode shape) to the measured mode
Q1 M	: amplification predicted with the model on mode N at point G1
Q2 M	: amplification predicted with the model on mode N at point G2
Q1 T	: amplification measured during test on mode N at point G1
Q2 T	: amplification measured during test on mode N at point G2
G1	: node of the model having the maximum predicted amplification on mode N
G2	: point of the structure having the maximum amplification measured during test on mode N.

5. SPECIFIC REQUIREMENTS FOR EACH UNITS [RE..]

This chapter provides the requirements concerning

- the numbering for the physical and the condensed models,
- the interface points between the units of the satellite.

5.1 NUMBERING RANGE FOR PHYSICAL MODELS [REAA]

Reference HP-1SP-REAA-001 :

The physical finite element model of each unit shall comply with the following numbering requirement:

- HERSCHEL

Unit	Allowed Numbering Range for grids, elements, rigid bodies, MPC, properties, materials, coordinate systems		
SVM	1	-	20000
SSH/SSD	20001	-	40000
CRYOSTAT/STRUTS	40001	-	60000
HEAT SHIELD	60001	-	70000
TELESCOPE	70001	-	90000

- PLANCK

Unit	Allowed Numbering Range for grids, elements, rigid bodies, MPC, properties, materials, coordinate systems		
SVM	1	-	20000
CRYOSTRUCTURE (BAFFLE)	20001	-	30000
CRYOSTRUCTURE (V-GROOVE)	30001	-	40000
CRYOSTRUCTURE (STRUTS SUPPORT)	40001	-	50000
SOLAR ARRAYS	50001	-	60000
TELESCOPE	60001	-	70000

TABLE 8: Allowed numbering range for physical models

This allowed numbering ranges are **TBC**.

The numbering of two kinds of elements must be different. For example, a bar element and a mass element must have different identification number.
The continuation card must contain at least the identification number of the element. For example if the identification number of an element is **822499**, the continuation card can be written as **+822490A**, **+822490B**, etc…

5.2 NUMBERING RANGE FOR CONDENSED MODELS [REAB]

Reference HP-1SP-REAB-001 :

The condensed finite element model (static or modal one) of each unit shall comply with the following numbering requirement:

- HERSCHEL

Unit	Allowed Numbering Range for grids, elements, rigid bodies, MPC, properties, materials, coordinate systems		
SVM	1	-	2000
SSH/SSD	2001	-	4000
CRYOSTAT/STRUTS	4001	-	6000
HEAT SHIELD	6001	-	7000
TELESCOPE	7001	-	9000

- PLANCK

Unit	Allowed Numbering Range for grids, elements, rigid bodies, MPC, properties, materials, coordinate systems		
SVM	1	-	2000
CRYOSTRUCTURE (BAFFLE)	2001	-	3000
CRYOSTRUCTURE (V-GROOVE)	3001	-	4000
CRYOSTRUCTURE (STRUTS SUPPORT)	4001	-	5000
SOLAR ARRAYS	5001	-	6000
TELESCOPE	6001	-	7000

TABLE 9: Allowed numbering range for condensed model

This allowed numbering ranges are **TBC**.

5.3 LOCATIONS AND NUMBERING OF INTERFACE NODES

All interface coordinates must be expressed in the HERSCHEL/PLANCK coordinate system defined § 2.1.

5.3.1 First Stage Structure [REFS]

Reference HP-1SP-REFS-001 :

	INTERFACE WITH LAUNCHER					
	GRID NUMBER IN CASE OF		COORDINATES (m)			
I/F NODE NAME	PHYSICAL MODEL	CONDENSED OR MODAL MODEL	X	Y	Z
		/			
		/			

The number of interface nodes is TBD
The coordinates values are TBD.

Reference HP-1SP-REFS-002 :

- HERSCHEL

	INTERFACE HERSCHEL PLM/SVM					
	GRID NUMBER IN CASE OF		COORDINATES (m)			
I/F NODE NAME	PHYSICAL MODEL	CONDENSED OR MODAL MODEL	X	Y	Z
		/			
		/			

- PLANCK

	INTERFACE PLANCK PLM/SVM				
	GRID NUMBER IN CASE OF		COORDINATES (m)		
I/F NODE NAME	PHYSICAL MODEL	CONDENSED OR MODAL MODEL	X	Y	Z
		/			
		/			

The coordinates values are TBD.

Reference HP-1SP-REFS-003 :

- HERSCHEL

	INTERFACE TELESCOPE/HERSCHEL PLM				
	GRID NUMBER IN CASE OF		COORDINATES (m)		
I/F NODE NAME	PHYSICAL MODEL	CONDENSED OR MODAL MODEL	X	Y	Z
		/			
		/			

- PLANCK

	INTERFACE TELESCOPE/PLANCK PLM				
	GRID NUMBER IN CASE OF		COORDINATES (m)		
I/F NODE NAME	PHYSICAL MODEL	CONDENSED OR MODAL MODEL	X	Y	Z
		/			
		/			

The coordinates values are TBD.

LIST OF OBJECTS

END OF DOCUMENT