Mechanical FEM Specification
ExoMars Rover Module Sub-Systems

CI CODE: E131000000

DTI EXPORT CONTROL RATING: 9E001, 9E002, 9E004
Rated by: M. Roe

This document is produced under ESA contract 19234/05/NL/CB, ESA export exemptions may therefore apply.
These Technologies may require an export licence if exported from the EU

Prepared by: J. Michelin

Checked by: A. Kiley

Approved by: J. Clemmet

Authorised by: M. Roe

© Astrium Limited 2010

Astrium Limited owns the copyright of this document which is supplied in confidence and which shall not be used for any purpose other than that for which it is supplied and shall not in whole or in part be reproduced, copied, or communicated to any person without written permission from the owner.

Astrium Limited
Gunnels Wood Road, Stevenage, Hertfordshire, SG1 2AS, England
# CONTENTS

1. INTRODUCTION .......................................................................................................................... 6

2. ABBREVIATIONS AND ACRONYMS ........................................................................................................ 6

3. MODEL DATA AND IDENTIFICATION ........................................................................................................ 7

   3.1 GENERAL INFORMATION .................................................................................................................... 7

       3.1.1 Deployed/Stowed configuration .................................................................................................. 7

       3.1.2 Model identification .................................................................................................................. 7

       3.1.3 Linearity of the model ............................................................................................................... 7

       3.1.4 Version of NASTRAN software ............................................................................................... 7

       3.1.5 Model units ............................................................................................................................... 7

   3.2 BOUNDARY CONDITIONS .................................................................................................................. 7

       3.2.1 Interface node label ................................................................................................................... 7

       3.2.2 Internal interfaces/joints ......................................................................................................... 8

   3.3 NUMBERING RANGE ......................................................................................................................... 8

       3.3.1 Bulk data card numbering range ............................................................................................. 8

       3.3.2 Bulk data duplicate card ......................................................................................................... 8

       3.3.3 Continuation card .................................................................................................................... 8

   3.4 COORDINATE SYSTEM ...................................................................................................................... 9

       3.4.1 Reference system ...................................................................................................................... 9

       3.4.2 Local reference system .......................................................................................................... 9

       3.4.3 Deployable item systems ....................................................................................................... 10

       3.4.4 Cards referring to reference system ....................................................................................... 10

   3.5 NASTRAN PARAMETERS ...................................................................................................................... 10

       3.5.1 Prohibited NASTRAN parameters ....................................................................................... 10

       3.5.2 Required NASTRAN parameters .......................................................................................... 10

       3.5.3 AUTOSPC parameter ............................................................................................................. 10

       3.5.4 Parameters affecting a set of data ........................................................................................... 10

   3.6 PERMITTED AND PROHIBITED CARDS .......................................................................................... 11

       3.6.1 Permitted NASTRAN card ..................................................................................................... 11

       3.6.2 Prohibited NASTRAN card .................................................................................................. 11

   3.7 NODES AND ELEMENTS ..................................................................................................................... 11

       3.7.1 Free nodes .............................................................................................................................. 11

       3.7.2 Nodes connecting CELAS .................................................................................................... 11

       3.7.3 Spring elements ..................................................................................................................... 12

   3.8 PROPERTIES ......................................................................................................................................... 12

       3.8.1 Material orientation ................................................................................................................. 12

       3.8.2 E, G and v relation ................................................................................................................... 12

       3.8.3 Bar element orientation ....................................................................................................... 12

       3.8.4 Model mass definition .......................................................................................................... 12

       3.8.5 Structural mass distribution ................................................................................................ 12

       3.8.6 Non-structural mass distribution .......................................................................................... 12

   3.9 THERMAL ........................................................................................................................................... 13

       3.9.1 Model thermo elastic properties .......................................................................................... 13

   3.10 DAMPING ........................................................................................................................................... 13

       3.10.1 Prohibited damping definition ............................................................................................ 13

       3.10.2 Recommended damping definition ..................................................................................... 13

       3.10.3 Symmetry of the damping matrix ....................................................................................... 13

4. CHECKS .................................................................................................................................................. 14

   4.1 GEOMETRY CHECKS .......................................................................................................................... 14

       4.1.1 Free boundaries ....................................................................................................................... 14
4.1.2 Coincident nodes..........................................................................................................................14
4.1.3 Element geometry............................................................................................................................14
4.1.4 Skew..................................................................................................................................................14
4.1.5 Taper ...............................................................................................................................................14
4.1.6 Aspect ratio .....................................................................................................................................14
4.1.7 Warp ...............................................................................................................................................14
4.1.8 Element co-ordinate systems...........................................................................................................14
4.1.9 Coincident elements..........................................................................................................................15

4.2 BOUNDARY CONDITIONS..................................................................................................................15
4.2.1 Magnitude of seismic mass............................................................................................................15
4.2.2 Magnitude of interface load CELAS.............................................................................................16

4.3 MODEL CONDITIONING CHECKS....................................................................................................16
4.3.1 Over constraint check.......................................................................................................................16
4.3.2 Diagonal terms of stiffness matrix ...................................................................................................17
4.3.3 DFLR check.....................................................................................................................................17
4.3.4 Warning messages.............................................................................................................................17
4.3.5 Maxratio.........................................................................................................................................17

4.4 FREE-FREE MODES............................................................................................................................18
4.4.1 Number of free/free modes .............................................................................................................18
4.4.2 Frequencies of rigid body modes.....................................................................................................18
4.4.3 SUPORT strain energy.........................................................................................................................18

4.5 MASS PROPERTIES..........................................................................................................................18
4.5.1 Mass according to axes......................................................................................................................18
4.5.2 Mass matrix at G, N, F and A set size...............................................................................................18
4.5.3 Centre of gravity................................................................................................................................18

4.6 STATIC LOAD CHECKS......................................................................................................................19
4.6.1 Force balance..................................................................................................................................19
4.6.2 Enforced unit displacement and rotation.........................................................................................19
4.6.3 Epsilon error ratio............................................................................................................................19
4.6.4 Elements with zero load...................................................................................................................19
4.6.5 Magnitude of SPCFORCE at grids constrained by AUTOSPC..........................................................19

4.7 DYNAMIC CHECKS..........................................................................................................................20
4.7.1 Low frequency acceleration............................................................................................................20

4.8 SIMPLIFIED MODEL.............................................................................................................................20
4.8.1 Simplified model accuracy...............................................................................................................20

4.9 THERMAL CHECKS............................................................................................................................20
4.9.1 Reference temperature check.........................................................................................................20
4.9.2 Thermo elastic analysis suitability checks.......................................................................................20

5. ACCURACY OF FEM VERSUS HARDWARE.......................................................................................21
5.1.1 Mass, inertia and centre of gravity accuracy......................................................................................21
5.1.2 Dynamic accuracy............................................................................................................................21

6. DELIVERABLES......................................................................................................................................22
6.1 FILES DELIVERED...............................................................................................................................22
6.1.1 Type of delivered files.......................................................................................................................22
6.1.2 Format of delivered files...................................................................................................................22
6.1.3 Files identification.............................................................................................................................22
6.2 NASTRAN INPUT FILES.....................................................................................................................22
6.2.1 NASTRAN input files.......................................................................................................................22
6.2.2 Use of ALTER..................................................................................................................................22
6.2.3 NASTRAN version..............................................................................................................................22
6.3 NASTRAN BULK DATA.........................................................................................................................22
6.3.1 Use of PARAM.................................................................................................................................22
6.3.2 NASTRAN files for thermo elastic model.......................................................................................22
6.4 ANALYSIS RESULTS............................................................................................................................22
6.4.1 Model check results...........................................................................................................................22
6.4.2 Normal mode summary ..............................................................................................................22
6.5 MODEL DESCRIPTION REPORT ....................................................................................................23
   6.5.1 FEM description .......................................................................................................................23
   6.5.2 Report outline ..........................................................................................................................23

TABLES

Table 1  Rover Module FEM Numbering .............................................................................................8
Table 2  Permitted model inputs .........................................................................................................11
Table 3  Prohibited model inputs ......................................................................................................11

FIGURES

Figure 1  Rover Module Co-ordinate System Alignment ....................................................................9
Figure 2  Shell element normal checking coordinate systems ............................................................15
Figure 3  Suggested Model Description Report Table of Contents .....................................................23
1. INTRODUCTION

This document describes a set of requirements for Finite Element Models (FEM) delivered by suppliers to form part of the Rover Module FEM. Defined are a set of generic FEM requirements for static, dynamic and thermo elastic analyses and specific requirements for Rover Module sub-systems. The requirements are based primarily on Astrium internal specifications, with additional ExoMars ESA FEM specifications and Thales-Alenia FEM requirements taken into account where applicable.

It is expected that sub-system FEMs be supplied as simplified ‘non reduced’ FEMs (i.e. not Craig Bampton or Guyan reduction models). This document does not therefore include requirements for reduced models. If a reduced model is agreed then a further reduced model requirements document will be issued.

2. ABBREVIATIONS AND ACRONYMS

ASU  Astrium Stevenage (Rover Vehicle sub-contractor)
CID  Co-ordinate System ID
CTE  Coefficient of Thermal Expansion
DM  Descent Module
DOF  Degrees of Freedom
EIDP  End Item Data Pack
FE(A/M)  Finite Element (Analysis/Model)
RBE  Rigid Body Element
RM  Rover Module
TAS-I  Thales Alenia (prime contractor)
TED  Thermo Elastic Distortion
3. MODEL DATA AND IDENTIFICATION

3.1 GENERAL INFORMATION

3.1.1 Deployed/Stowed configuration
Deployable systems shall be supplied in their stowed and nominal deployed configuration.

3.1.2 Model identification
A finite element model shall be assigned a unique identification name or number and refer to a configuration of issue where possible. For example this can be a mechanical interface control document of the equipment including mass, inertia and centre of gravity or an issue of the system mass budget at key phases of the design, PDR, CDR...

3.1.3 Linearity of the model
A full structural model shall be delivered using only linear elastic elements and properties.

3.1.4 Version of NASTRAN software
All data in any delivered finite element definition shall be compatible with MSC MD NASTRAN Version 2006r1 unless an alternative is agreed with ASU. When an alternative analysis code has been used the results must be correlated with the delivered NASTRAN model.

3.1.5 Model units
NASTRAN solver and associated pre/post processors have no preference for units, it is the responsibility of the analyst to ensure consistent units are used in the model. All models shall be defined using S.I. units:
- Newton (N) for force
- Kilograms (kg) for mass
- Metres (m) for length
- Seconds (s) for time
- Degrees Celsius (°C) for temperature
with the derived units:
- E-modulus [Newton/meter²]
- density [kg/meter³]

3.2 BOUNDARY CONDITIONS

3.2.1 Interface node label
All model interfaces shall be representative of the physical attachment points of the items and shall be modelled as independent points with appropriate degrees of freedom, typically six degrees of freedom. The label of each interface node shall be clearly identified.

The following comments shall be included within the model description document:
- Interface grid points, identification and location,
- Boundary condition, clamped degrees of freedom,
- Grids included in the ASET or CSUPER when appropriate.
3.2.2 Internal interfaces/joints

Any internal sub-system interfaces/joints (for example deployment hinges) shall be modelled with CELAS2 elements to allow force recovery.

3.3 NUMBERING RANGE

3.3.1 Bulk data card numbering range

Identification numbers of all NASTRAN cards (e.g. nodes, elements, coordinate systems, material properties...) shall be in accordance with the numbering range set out in Table 1. Note that some sub-systems may be the subject of further numbering range sub-division in subsequent issues of this document (for example service module into frame and PCB boards). A record of identities, quantities of elements and grids associated with all parts of the FEM shall be kept. This will assist in error tracing during the completion of constraint and conditioning checks.

<table>
<thead>
<tr>
<th>ITEM</th>
<th>start ID</th>
<th>end ID</th>
</tr>
</thead>
<tbody>
<tr>
<td>ROVER MODULE</td>
<td>500000</td>
<td>699999</td>
</tr>
<tr>
<td>MAIN TUB STRUCTURE</td>
<td>510000</td>
<td>549999</td>
</tr>
<tr>
<td>SOLAR ARRAYS (fixed and deployed)</td>
<td>550000</td>
<td>559999</td>
</tr>
<tr>
<td>DRILL BOX ASSEMBLY</td>
<td>560000</td>
<td>569999</td>
</tr>
<tr>
<td>CAMERA MAST ASSEMBLY</td>
<td>570000</td>
<td>579999</td>
</tr>
<tr>
<td>INSTRUMENT ARM ASSEMBLY</td>
<td>580000</td>
<td>589999</td>
</tr>
<tr>
<td>WISDOM RADAR ASSEMBLY (including support bracket)</td>
<td>590000</td>
<td>599999</td>
</tr>
<tr>
<td>ANALYTICAL LABORATORY DRAWER</td>
<td>600000</td>
<td>609999</td>
</tr>
<tr>
<td>SERVICE MODULE</td>
<td>610000</td>
<td>629999</td>
</tr>
<tr>
<td>RADIATORS AND THERMAL</td>
<td>630000</td>
<td>639999</td>
</tr>
<tr>
<td>STEERABLE ANTENNA ASSEMBLY</td>
<td>640000</td>
<td>649999</td>
</tr>
<tr>
<td>RESERVED (not currently used)</td>
<td>650000</td>
<td>669999</td>
</tr>
<tr>
<td>MISC ITEMS</td>
<td>670000</td>
<td>679999</td>
</tr>
<tr>
<td>LOCOMATION ASSEMBLY (including wheels)</td>
<td>680000</td>
<td>699999</td>
</tr>
</tbody>
</table>

Table 1 Rover Module FEM Numbering

3.3.2 Bulk data duplicate card

No duplicate node or element identification numbers shall exist in the model.

3.3.3 Continuation card

The continuation card shall be written with “+” or “*” symbol. If a number is used in the continuation card, it must be the card ID number. For instance “+E120000” continuation card is only authorised for element n° 120000.

3.3.4 FEM size

Sub-system FEMs supplied for integration into the RM shall use only the minimum number of DOFs required to describe dynamic behaviour of the component at system level analysis; for example all modes likely to impart significant force on the structure or affect RM behaviour and allow output of acceleration responses at key equipment locations. Where a detailed model is reduced in complexity for delivery to ASU a comparative check shall be made as described in section 4.8.
3.4 COORDINATE SYSTEM

3.4.1 Reference system

One primary parent rectangular right handed reference system shall be defined by the supplier for the sub model. This parent system shall be aligned to the RM system ($X_{RB}$, $Y_{RB}$, $Z_{RB}$) illustrated in Figure 1 with positive X-axis in the direction of forward motion and positive Z-axis normal to (and upward from) the RM ground plane. Where practical it is preferred that the origin of this system and all child systems be within the physical boundary of the sub-system.

![Figure 1 Rover Module Co-ordinate System Alignment](image)

3.4.2 Local reference system

All local reference systems shall be related to this primary reference system and in general aligned as closely as possible to the primary RM system.

3.4.3 Deployable item systems

A separate co-ordinate system hierarchy shall be used for deployable items to allow deployment by re-orientation of a single system. Where deployment requires orientation at more than one location in stages, each stage of deployment shall be possible by re-orientation of a single system.

3.4.4 Cards referring to reference system

All NASTRAN cards using reference systems (e.g. GRID, CONM2, CBUSH etc.) shall be defined with respect to the primary reference system or a local reference system i.e. the use of the default identification number "0" is not allowed for any card. Where possible it is recommended on the GRID card to use the same co-ordinate system for reference and displacement.
3.5 NASTRAN PARAMETERS

NASTRAN has available a number of parameters to provide some options on how the model is processed.

3.5.1 Prohibited NASTRAN parameters

The following Parameters shall not to be used:

- BAILOUT, -1
- MAXRATIO
- EPZERO
- WTMASS
- MECHFIX, YES

The use of other default values in version 2006 shall be agreed with ASU.

3.5.2 Required NASTRAN parameters

The following parameters shall be used with the specified value:

- BAILOUT, 0
- K6ROT, 1.0
- SNORM, 20.0
- MECHFIX, NO
- AUTOSPC, YES

To verify the use of K6ROT the following normal modes sensitivity analyses shall be performed:

<table>
<thead>
<tr>
<th></th>
<th>K6ROT</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Sensitivity using K6ROT *10^-1</td>
</tr>
<tr>
<td>2</td>
<td>Baseline value</td>
</tr>
<tr>
<td>3</td>
<td>Sensitivity using K6ROT *10</td>
</tr>
</tbody>
</table>

Acceptance criterion: frequency shift and modal mass of paired modes ≤ 0.1% compared to baseline value. This means that the shift in frequency when modifying the K6ROT value should be lower than 0.1% taking into account possible mode re-ordering.

The use of other default values in version 2006 shall be agreed with ASU.

3.5.3 AUTOSPC parameter

When a permanent single point constraint set has been generated from AUTOSPC, YES this set must be kept separate from the model boundary constraint set (i.e. different SPC card, same SPC set allowed) and is part of the definition of the model so must be provided to ASU.

3.5.4 Parameters affecting a set of data

Any input specification, which can affect the data of other models when merged, shall not be used. For example assigning a value of greater than 0 for PARAM, COUPMASS invokes the creation of the coupled mass matrix which can lead to different frequencies compared to the use of the lumped mass matrix.
3.6 PERMITTED AND PROHIBITED CARDS

3.6.1 Permitted NASTRAN card
The following NASTRAN cards are permitted:

<table>
<thead>
<tr>
<th>COORDINATE SYSTEMS</th>
<th>CORD2R, CORD2C, CORD2S</th>
</tr>
</thead>
<tbody>
<tr>
<td>NODES</td>
<td>GRID, SPOINT</td>
</tr>
<tr>
<td>1D ELEMENTS</td>
<td>CROD, CBAR, CBEAM</td>
</tr>
<tr>
<td>2D ELEMENTS</td>
<td>CTRIA3, CQUAD4</td>
</tr>
<tr>
<td>3D ELEMENTS</td>
<td>CPENTA, CTETRA, CHEXA</td>
</tr>
<tr>
<td>CONCENTRATED MASSES</td>
<td>CONM2</td>
</tr>
<tr>
<td>INTERFACES SPRINGS</td>
<td>CELAS1, CELAS2, CBUSH</td>
</tr>
<tr>
<td>RIGID ELEMENTS</td>
<td>MPC, RBAR &amp; RBE2, RBE3</td>
</tr>
<tr>
<td>OTHERS</td>
<td>PLOTEL, DMIG, GENEL</td>
</tr>
<tr>
<td>BOUNDARY CONDITIONS</td>
<td>SPC, SPC1</td>
</tr>
<tr>
<td>PROPERTIES</td>
<td>Any linear properties</td>
</tr>
</tbody>
</table>

Table 2  Permitted model inputs

3.6.2 Prohibited NASTRAN card
The following bulk data input is prohibited:

<table>
<thead>
<tr>
<th>MSGMESH commands</th>
<th>MSGMESH commands used to create model bulk data input.</th>
</tr>
</thead>
<tbody>
<tr>
<td>COORDINATE SYSTEMS</td>
<td>CORD1x</td>
</tr>
<tr>
<td>NODES</td>
<td>EPOINT</td>
</tr>
<tr>
<td>1D ELEMENTS</td>
<td>BAROR</td>
</tr>
<tr>
<td>2D ELEMENTS</td>
<td>CTRIA6, CQUAD8</td>
</tr>
<tr>
<td>RIGID ELEMENTS*</td>
<td>Non zero length RBAR, RBE2 in models for thermal distortion analysis unless using a CTE value</td>
</tr>
<tr>
<td>PROPERTIES</td>
<td>Non linear properties, structural damping</td>
</tr>
<tr>
<td>OTHERS</td>
<td>GRDSET</td>
</tr>
</tbody>
</table>

Table 3  Prohibited model inputs

3.7 NODES AND ELEMENTS

3.7.1 Free nodes
Free nodes (nodes not associated to a particular element) are prohibited.

3.7.2 Nodes connecting CELAS
To avoid possible over constraint, ill conditioning and erroneous load paths, grids connected by CELAS elements shall:

- be defined in the same co-ordinate system
- reference the same output co-ordinate system
3.7.3 Spring elements

CELAS elements shall be labelled in accordance with the associated dof of grids e.g. CELAS connecting dof 1 of two grids shall be labelled as xxxxx1. CELAS connecting dof 2 of two grids shall be labelled as xxxxx2...

If the CELAS is connecting two SPOINT, it shall be labelled xxxxx0.

3.8 PROPERTIES

3.8.1 Material orientation

Material orientation and its consistency shall be taken into account if the following conditions apply to FE models:

- composite materials are modelled with anisotropic material properties,
- loads are to be applied to the FEM with respect to the material axes,
- element forces or stresses are to be extracted with respect to material axes.

3.8.2 E, G and ν relation

For MAT1, only two of the three values of E, G and ν shall be filled except if this is coherent with the relation

\[ G = \frac{E}{2(1+\nu)} \]

This should eliminate warning message 2251 associated with inconsistent values for E, G and ν.

3.8.3 Bar element orientation

The orientation of linear elements, bars and beams, require the definition of an orientation vector. The orientation shall be defined by the specification of components of the orientation vector. The use of a grid point to define the element orientation is not permitted.

3.8.4 Model mass definition

The FEM mass and mass distribution shall be consistent with the predicted mass \( M_p \) of the sub-system or equipment where:

\[ M_p = M_{bee} + M_{mm} \]

- \( M_{bee} \) = mass, best engineering estimate
- \( M_{mm} \) = maturity margin

Any deviations from the predicted mass \( M_p \) should be documented.

3.8.5 Structural mass distribution

Where practical the density field of the appropriate MAT card shall be used to define structural mass only. This shall be representative of the material considered.

3.8.6 Non-structural mass distribution

Non-structural masses shall be spread over the structure via NSM field of property cards or as specific lumped mass using CONM2 elements. Variation of material mass density to simulate distributed non-structural masses shall be avoided. If used this shall be clearly documented.
3.9 THERMAL

3.9.1 Model thermo elastic properties
All properties necessary for thermo-elastic analysis applications shall be specified, i.e. Coefficient of Thermal Expansion and reference temperature. A reference temperature of 20°C shall be used. The use of non-zero length rigid elements (excluding RBE3s) is not permitted for thermal analysis unless RBE2s are specified with the correct CTE value (i.e. matching the component to which the RBE2 is attached). A single RBE2 shall not be used to connect two items of differing CTE. Where material CTE differs along each material axis (for example carbon fibre lay-ups) an average CTE value shall be used.

3.10 DAMPING

3.10.1 Prohibited damping definition
Any viscous damping elements shall be avoided unless they are inherent in the design. Specification of damping on CELAS, CBUSH and MAT cards is generally prohibited. In exceptional circumstances (with the agreement of ASU), where material/element based structural damping needs to be specified, the supplier shall clearly identify and justify the specification of damping on any element or material input data.

3.10.2 Recommended damping definition
When the FEM has been used for dynamic response analysis the supplier shall provide the critical modal damping versus frequency (NASTRAN TABDMP1 input data), in the technical description document.

3.10.3 Symmetry of the damping matrix
If a damping matrix is provided it shall be a symmetric.
4. CHECKS

4.1 GEOMETRY CHECKS

4.1.1 Free boundaries
Element free boundaries or edges of the model shall be checked to ensure there are no unintentional “splits” in the mesh or missing elements.

4.1.2 Coincident nodes
A check shall be made for non-intentional node coincidence. Node coincidence in an FEM is acceptable and often intentional, a common example is when spring elements are used to extract loads, and such springs must be of zero length.

4.1.3 Element geometry
Element geometry checks shall be performed to determine taper, skew angle, warping and aspect ratios. Elements that violate the generally accepted conditions may not necessarily be incorrect. Generally accepted conditions for quality checks are as follows,

4.1.4 Skew
Skew is the angle between lines that join opposite mid sides of an element it is recommended that this angle be greater than 30 degrees. An angle of 90 degrees represents no skew.

4.1.5 Taper
Taper is the ratio of the areas on the two sides of a diagonal. This ratio shall be less than 0.5.

4.1.6 Aspect ratio
Aspect ratio is the ratio of the length of any two sides on a CQUAD4 element. It is recommended that this value is less than 4.

4.1.7 Warp
Warping of shell elements occurs when the connected grids are not in the same plane. For contoured surfaces warping shall be generally limited to 5° and not exceed 15°. Flat surfaces shall be checked for zero warpage.

4.1.8 Element co-ordinate systems
Element co-ordinate systems shall be consistent where possible including the element normal. Consistent element co-ordinate systems for elements on the same elementary structure ensure:

- Consistent application of the direction of element pressure.
- Consistent interpretation of element forces.
- Consistent interpretation of element stresses.

Most pre processors have facilities for showing element co-ordinate systems and normals.
4.1.9 Coincident elements
Element coincidence shall be checked in all pre processors. Such errors will not be highlighted by constraint and conditioning checks and typically occur when creating multiple meshes. A clear inventory of element usage, numbering and quantity, as well as the use of groups can minimise the occurrence of these errors.

4.2 BOUNDARY CONDITIONS

4.2.1 Magnitude of seismic mass
For sine and random analysis the large (seismic) mass method is often used. A mass much greater than the mass of the structure being excited is used to achieve fixed base modes of vibration and to provide a scaling factor for base force input.

The magnitude of the seismic mass required to represent the fixed base condition shall be verified by comparing the frequencies and effective modal mass obtained from processing the model with fixed base boundary conditions constrained defined by SPC’s and with the seismic mass with SUPORT specification.

The differences between the eigenfrequencies ($f_1$, $f_2$) of the two models shall be less than 0.1%. As a guide, the seismic mass should be of the order of $10^5$ to $10^7$ times the model mass and inertias about the excitation reference point.
4.2.2 Magnitude of interface load CELAS

Supplementary “stiff” CELAS elements are required at interface points in order to recover interface loads.

The magnitude of the stiffness coefficients of CELAS elements shall be determined by comparing the frequencies and effective modal mass obtained from processing the model with or without this supplementary stiffness:

The difference between the eigenfrequencies \( f_1, f_2 \) of the two models shall be less than 0.1%.
It can be generally achieved with a stiffness \( K \) of \( 10^{10} \) N/m for translational dof and \( 10^8 \) N.m/rad for rotational dof. Model conditioning checks shall consider these elements.

4.3 MODEL CONDITIONING CHECKS

4.3.1 Over constraint check

The purpose of the constraint check is to verify the model includes no automatically created or unintentional constraints that will have an effect on the internal loads or rigid body behaviour.
This verification can be made with the NASTRAN GROUNDCHECK parameter (available since version 2001 of MSC/NASTRAN). The model is constrained by specification of sufficient degrees of freedom to eliminate rigid body motion via a SUPORT bulk data input. Where possible the constraint conditions should be representative of those to be used in subsequent analyses.

The results of this check reported in the f06 file include four matrices, called KRBI. These are calculated from the stiffness matrix and a rigid body vector based on the geometry of the model:

\[
[K_{RB}]= [\mathbf{e}]^T [K] [\mathbf{e}] \text{ (Joules)}
\]

Where \([K]\) is the stiffness matrix and \([\mathbf{e}]\) a rigid body mode vector calculated from the model geometry. This product is equivalent to twice the strain energy.
Matrix KRBI are performed at various model sizes:
- G set (KRBG matrix): all structural degrees of freedom,
- N set (KRBN matrix): all structural degrees of freedom after implementation of multi-point constraint relationships,
- F set (KRBF matrix): all unconstrained (free) degrees of freedom, after applications of all constraints,
- A set (KRBA matrix): analysis set.

The resultant matrices KRBI are equal to twice the strain energy in the structure following its motion as a rigid body. With SI units the six terms on the diagonal of the respective energy matrix, KRBi, shall be:
- G set: less than \(1.0 \times 10^5\) J
- N set: less than \(1.0 \times 10^5\) J
- F set: less than \(1.0 \times 10^5\) J for translational dof and \(5.0 \times 10^2\) J for rotational dof
- A set: less than \(1.0 \times 10^5\) J for translational dof and \(5.0 \times 10^2\) J for rotational dof
In the event the supplier is unable to meet these limits, ASU will require a justification for the reasons why these limits are not met with a sound rationale for the acceptability of the FE model. The scope of justification depends primarily on the magnitude of deviation between the achieved 2*Strain Energy levels versus the above limits.

4.3.2 Diagonal terms of stiffness matrix

Warning messages associated with inconsistent material properties shall be investigated. The material properties are used to compute the stiffness properties associated with the degrees of freedom of the element. NASTRAN for example uses only Young’s Modulus for axial and bending stiffness and shear modulus for the torsional stiffness of linear elements; Poisson’s Ratio is not used. For shell elements Young’s Modulus and Poisson’s Ratio is used for in plane stiffness properties and shear modulus only for transverse shear.

Printing the diagonal items of the stiffness matrix may be useful to assess the potential ill conditioning of inconsistent material properties.

4.3.3 DFLR check

The NASTRAN procedure for this conditioning check is based on this method of determining the existence of singularities. The F-set size stiffness matrix is partitioned by use of a SUPORT card as follows:

\[
K_{FF} = \begin{bmatrix}
K_{FRFR} & K_{FRR} \\
K_{RFR} & K_{RR}
\end{bmatrix}
\]

It is the sub-matrix \( K_{FRFR} \) which is decomposed and so any singularities in this matrix are associated with ‘soft’ areas within the structure. The degrees of freedom chosen for the R-set by entering them on the SUPORT card should be those which most closely represent the true support conditions of the structure. However only six degrees of freedom should be used in order to give a statically determinate support.

There are two matrices to be studied as a result of this check designated DLFR and MECHFR (equivalent to MAXRATIO). Firstly the matrix DLFR outputs the diagonal terms of the decomposed stiffness matrix if they are less than unity, as a small value indicates a point of low stiffness. Secondly the matrix MECHFR outputs the ratio between the terms on the leading diagonal of the matrix \( K_{FRFR} \) to those on the leading diagonal of the decomposed stiffness matrix, if the value exceeds 1000. The DOFs identified by MECHFR indicate the presence of potential mechanisms in the structure.

The accuracy to be achieved from this check shall be as follows:

**DLFR Values : > 1.0 E-3 (positive)**

4.3.4 Warning messages

Any warning messages occurring in the NASTRAN f06 file shall be reviewed to confirm they do not influence the results obtained.

4.3.5 Maxratio

The maximum ratio represents the ratio between the highest and the lowest term of the stiffness matrix. If the value of MAXRATIO is greater than the NASTRAN default value (10\(^7\) in version 2001) an error message is printed. If the value is less than the default value, this is not reported within the NASTRAN solution sequence.

The value of MAXRATIO shall be less than 10\(^7\). In the event the supplier is unable to meet this criterion, a justification shall be provided.
4.4 FREE-FREE MODES
The purpose of the "free-free" check is to verify the rigid body modes of the model. This verification is made with a NASTRAN SOL 103 dynamic analysis with the model in free-free conditions, i.e. no constraint specified by SPC or SUPORT input.

4.4.1 Number of free/free modes
It shall be verified that the number of rigid body modes is 6. Supplementary rigid body modes (such as mechanisms) shall be justified on a case-by-case basis.

4.4.2 Frequencies of rigid body modes
The ratio of the highest rigid body frequency to the frequency of the first elastic mode shall be less than 10^{-4}. Furthermore, all rigid body frequencies shall be <0.005Hz.

4.4.3 SUPORT strain energy
This check uses the strain energy to confirm acceptable mathematical conditioning of internal loads of the model.

This check is made with a classical NASTRAN dynamic analysis (SOL 103) with the model constrained using SUPORT input data. The strain energy is calculated and given in the "*.f06" file at every support point. The value of the strain energy shall be less than or equal to 5.0E^{-2} J.

4.5 MASS PROPERTIES

4.5.1 Mass according to axes
The mass distribution check performed with respect to the NASTRAN “Grid Point Weight Generator (GPWG)” provides:

- mass matrix of the structure (M0) at a reference point defined by PARAM GRDPNT,
- mass of the structure and position of the centre of gravity with respect to the reference point in the primary local coordinate system,
- inertia matrix at the centre of gravity and in the primary local coordinate system of the reference point [I(S)] or in the principal axis of inertia [I(Q)],
- the reference frame transformation matrix (Q), between the local coordinate system of the reference point and the principal axis of inertia, the matrix of direction cosines.

Mass figures shall be the same for axes X, Y and Z. The centre of gravity location shall correspond with design mass estimates or the appropriate ICD to within 5% of the overall linear dimension in each axis. If this is not the case differences shall be justified.

4.5.2 Mass matrix at G, N, F and A set size
The integrity of the mass matrix shall be checked at the NASTRAN G, N, F and A set size. The rigid body mass matrix shall be determined:


Where [M] is the mass matrix and [\phi] a rigid body vector calculated from the model geometry.

4.5.3 Centre of gravity
Diagonal terms of the centre of gravity matrix shall be less than 10^{-6} m. The two other terms (successively for each direction) must be equal.
4.6 STATIC LOAD CHECKS

4.6.1 Force balance

The purpose of the static load check is to confirm that total forces at the interface of the model balance with the applied load. The reaction forces for all loads analysed, gravity vector, point load or thermal load, shall be checked to confirm they are at expected locations and that the forces and moments balance the applied load.

The application of unit acceleration vector \(1 \text{ m.s}^{-2}\) independently in each axis shall be used to confirm that the sum of the constraint forces divided by the applied acceleration equals the model mass.

Interface reaction forces should be computed using the NASTRAN command SPCFORCES=ALL. NASTRAN provides in the f06 output file the RESULTANT LOAD, the sum of the applied loads at the GPWG reference point; the default is the origin of the basic coordinate system. The RESULTANT SPCFORCES, the sum of the SPCFORCES is also provided. For unit acceleration vectors \(1 \text{ m.s}^{-2}\) the force must be equal to the mass.

4.6.2 Enforced unit displacement and rotation

A unit enforced displacement in all six degrees of freedom shall be applied at the base of the structure (i.e. at approximately the centroid of the structure interface points) to verify no over constraint is present in the model. With the model only constrained at the position of applied enforced displacement the model should move as a rigid body when it is translated one unit of displacement or one radian of rotation. For a condition of rigid body motion the element forces, stresses and grid point forces should all be “zero”.

Displacement results from the three translational subcases shall be 1.0 in the input direction and zero in the other five directions. For the rotational cases the rotation in the input direction shall be 1.0 and 0.0 in the two rotational directions. The translations from the rotational cases will not be zero.

4.6.3 Epsilon error ratio

The ratio of work done by residual loads vs. work done by external applied loads shall be less than 1.0E-8 (reported as Epsilon by Nastran).

4.6.4 Elements with zero load

The model constraint checks, conditioning checks and free-free modes checks do not confirm correct connectivity of all load paths. To verify that valid load paths have not been omitted an applied load representative of that to be used in analysis (e.g. gravity vector, point load or thermal load) shall be applied to the model. Appropriate Element output data, loads or stresses, shall be obtained and processed to identify elements (CELAS, RBE, CTRIA, CQUAD, BAR, BEAM…) with zero load. Elements with zero load or stress shall be reviewed to assess potential loss of connectivity.

4.6.5 Magnitude of SPCFORCE at grids constrained by AUTOSPC.

Forces associated with degrees of freedom constrained by AUTOSPC should tend to zero. In practice some non-zero constraint forces will exist. The magnitude however shall be reduced to a quantity that produces negligible errors both to the overall solution and to the model locally.
4.7 DYNAMIC CHECKS

4.7.1 Low frequency acceleration

For dynamic analyses using modal superposition, it shall be verified that the behaviour at low frequencies is correct. The application of unit acceleration vector (1 m/s²) at low frequencies (near 0 Hz) independently in each axis shall be used to confirm that the sum of the constraint forces divided by the model mass is at least equal to 95 % of the applied acceleration.

Values less than 95 % indicate insufficient modes in the analysis. It can be corrected with the PARAM, RESVEC or with more modes.

4.8 SIMPLIFIED MODEL

4.8.1 Simplified model accuracy

In cases where the FEM supplied to ASU for RM integration is a simplified (reduced number of elements) version of a more detailed sub-system FEM the following additional check shall be performed to ensure it is sufficiently representative. The results from a fixed base normal modes analysis of the simplified model shall be compared against the detail model and the following criteria met:

- Major mode frequencies (effective mass >5% rigid item mass) shall be within 3%
- Effective masses of major modes shall be within 5%
- All other mode frequencies up to and including the tenth flexible mode shall be within 5%
- Effective masses of all other modes up to and including the tenth flexible mode shall be within 10%

Mass and other model checking criteria shall conform to the requirements outlined throughout section 4.

4.9 THERMAL CHECKS

To check that the model is suitable for thermal distortion analysis the following assessments shall be performed.

4.9.1 Reference temperature check

To check for correct reference temperature definitions a uniform temperature of 20°C shall be applied to the model and results checked to ensure all element stresses/loads and nodal displacements are numerically zero.

4.9.2 Thermo elastic analysis suitability checks

The linear expansion coefficient for all the material properties in the model shall be assigned a value of 10⁻⁵ m/m°C and a reference temperature of 20°C. A uniform temperature change, ΔT=100°C, shall be applied to the model with the model iso-statically constrained.

The maximum rotation in the model shall be less than or equal to 10⁻⁷ rad. The maximum Von Mises stresses in the model shall be less then or equal to 100 Pa.

Furthermore with the model material thermal properties set to their correct (model delivery) values a uniform temperature change, ΔT=100°C, shall be applied to the model as described above. An inspection of the FEM thermo elastic distortion and stresses shall be made to ensure no unexpected behaviour occurs resulting from incorrect CTE assignments.
5. ACCURACY OF FEM VERSUS HARDWARE

5.1.1 Mass, inertia and centre of gravity accuracy
The accuracy of the model versus the equipment hardware (*) shall be:

- 1% for the mass,
- 5% for the moments of inertia,
- 1% of the maximum distance for the centre of gravity versus substructure I/F.

(*) The reference for accuracy should be the appropriate ICD relevant to the project or a delivered system budget. The latter should be taken as the reference in the absence of weighed data e.g. early study work, PDR stage etc. On the production of hardware the reference for accuracy should be the weighed item.

5.1.2 Dynamic accuracy
Dynamic accuracy of the FEM versus the hardware shall be computed when experimental modal testing has been performed. Error shall be lower than:

- 20 % for static or thermo-elastic displacements,
- 5 % for main frequencies(1) of the item in test configuration,
- 20 % for significant responses performed during tests.

(1) Main frequencies means that the model effective mass has more than 5 % of the rigid mass of the item involved.
6. DELIVERABLES

6.1 FILES DELIVERED

6.1.1 Type of delivered files
The model files to be delivered shall be:
- NAStRAN input files,
- NAStRAN bulk data,
- modal analysis with representative boundary condition (.f06 summary of eigen-frequency listing etc),

6.1.2 Format of delivered files
All data files shall be in ASCII format.

6.1.3 Files identification
The model name, issue and date shall be clearly specified.
A brief summary of all delivered files shall be provided.

6.2 NAStRAN INPUT FILES

6.2.1 NAStRAN input files
NAStRAN input files (containing the executive and case control section) shall be supplied for all analysis types. These shall be supplied separate to the accompanying bulk data files.

6.2.2 Use of ALTER
If NAStRAN written DMAP ALTERS are used the name and all used parameters shall be supplied and described.

6.2.3 NAStRAN version
The version and vendor of NAStRAN used shall be declared in the header of the file.

6.3 NAStRAN BULK DATA

6.3.1 Use of PARAM
All the PARAM cards required to run the model shall be included.

6.3.2 NAStRAN files for thermo elastic model
For the thermo elastic model the complete set of bulk data cards for the zero stress test (section 4.9.2) shall be delivered.

6.4 ANALYSIS RESULTS

6.4.1 Model check results
Extracts from the NAStRAN f06 output files appropriate to the model checks shall be reported in the model description report.

6.4.2 Normal mode summary
A table showing at least the first ten elastic modes (from a fixed base normal modes analysis) shall be provided including frequencies, effective masses and brief mode shape descriptions.
6.5 MODEL DESCRIPTION REPORT

6.5.1 FEM description
The finite element model description report shall be written in English.

6.5.2 Report outline
The FEM report shall define as a minimum the model designation, properties, mass distribution, modelling of all parts that the integrator needs to reference, attach to, adapt on integration to system level and results of the model checks performed.

Figure 3 provides a suggested list of the subjects in the model description report.

A sketch of the co-ordinate system(s) associated with the primary appendages should be provided. In general, all input data used for the model shall be clearly documented. Detailed plots of the model clearly showing all nodes, elements, and connectivity of primary interfaces (and relevant numbering and types) shall be supplied.

Compliance with all requirements specified in this document shall be declared.

<table>
<thead>
<tr>
<th>LIST OF ACRONYMS</th>
</tr>
</thead>
<tbody>
<tr>
<td>LIST OF SYMBOLS</td>
</tr>
</tbody>
</table>

1. SCOPE

2. DOCUMENTS
2.1 APPLICABLE DOCUMENTS
2.2 REFERENCE DOCUMENTS

3. DESCRIPTION
3.1 OVERALL DESCRIPTION
3.2 DESIGN LEVEL
3.3 MASS AND COM BUDGET

4. FE MODEL
4.1 FEM CODE AND PRE/POST-PROCESSORS
4.2 FEM UNITS
4.3 FEM CO-ORDINATE SYSTEMS
4.4 FE MODEL DESCRIPTION
4.4.1 Assumption and idealisations
4.4.2 Geometry (node and element numbering)
4.4.3 Interfaces
4.4.4 Boundary conditions
4.4.5 Properties and materials
4.4.6 Mass distribution summary
4.4.7 Other special modelling features

5. FE/ MODEL CHECKS
5.1 Gravity load check
5.2 Strain energy check
5.3 Rigid-body frequency check
5.4 Enforced displacement check
5.5 Thermo-elastic check
5.6 Modal Behaviour Summary

6. COMPLIANCE WITH FEM REQUIREMENTS

---

Figure 3  Suggested Model Description Report Table of Contents
DOCUMENT CHANGE DETAILS

<table>
<thead>
<tr>
<th>ISSUE</th>
<th>CHANGE AUTHORITY</th>
<th>CLASS</th>
<th>RELEVANT INFORMATION/INSTRUCTIONS</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>-</td>
<td>-</td>
<td>Initial Issue</td>
</tr>
</tbody>
</table>

DISTRIBUTION LIST

INTERNAL

- A. Kiley
- R. Slade
- M. Roe
- J. Clemmet
- J. Michelin
- Configuration Management

EXTERNAL

- G. Giordano (TAS-I)