OrbiNas at a Glance

- General Description
- Flyer
- Launcher
- OrbiNas Modules
- OrbiNas 'Set-Limits' Sheet
- FEM Quality Check / Modal Analysis
- Condensed Matricial Models Generation
- Frequency Response Analysis & Model Comparison

General Description

- OrbiNas is a software helping the FEM Analyst, when using MSC.Nastran code.
- The development is based upon more than 35 years of experience on FEM analysis, using Nastran, mainly in Space and Aeronautics Sectors.
- Programming languages used are mainly Fortran, MS-Dos Script and VBA for Excel & PowerPoint. Punctually, Python for Word Report Generation.
- Currently the Available & Supported Software for OrbiNas V-2025.5 are:
  - Microsoft Versions
    - Windows7-64 bits / Windows10 / Windows11
    - Office 2016 & Later 365 (Excel, PowerPoint, Word)
  - MSC.Nastran Versions
    - MSC.Nastran 2017 and later
    - For some options, using specific OrbiNas DMAP (mainly DynCond & StatCond modules): 2017.1 / 2018.2 / 2019.0
  - MSC.Patran Versions
    - From 2016 to 2018 versions have been tested. Later should be compatible (session files are used).

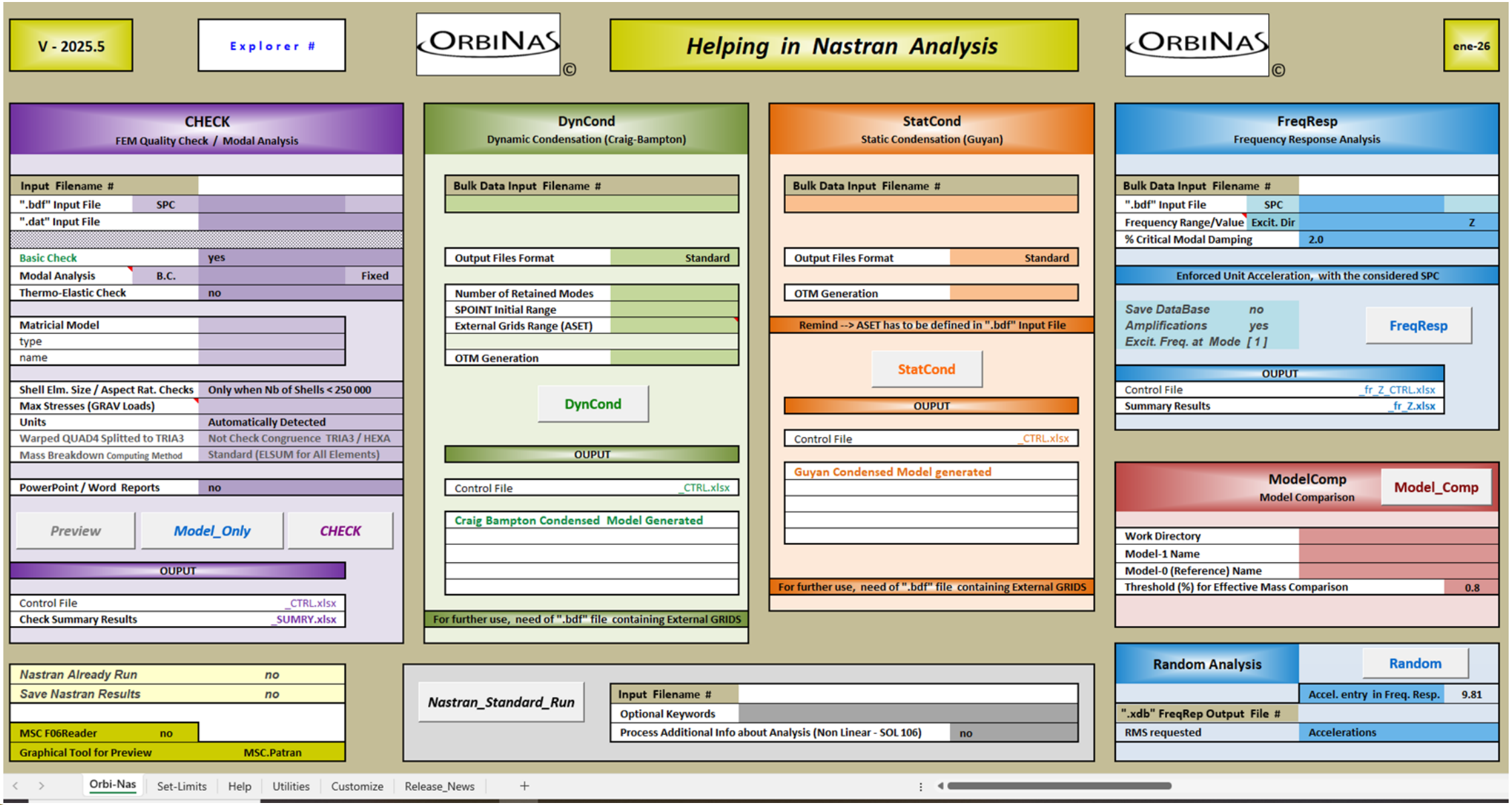
Flyer

- Aims
  - Ensure the Quality and Reliability of the FEMs handled and delivered
  - Increase Productivity

- Capabilities, helping in the following FEM Analysis Areas and Tasks
  - Finite Element Model Checks and Validation (these checks are the Standard ones requested in Space and Aeronautics Sectors)
  - Modal Analysis with Effective Mass Evaluation
  - Frequency Response and Random Analyses
  - Generation/Integration of Condensed Matricial Models
  - FEM Post-Processing with MSC.Patran
  - Documentation

- Time Saving and Increased Reliability in the Areas/Tasks listed above
  - FEM Standard Checks are automatically performed
  - Relevant Information about the Model (Input Data & Check Results) is collected, easily accessible and logically related to each other
  - Generation of Documentation that can be directly integrated into FEM Description/Validation documents and Analysis Reports

- Time Saving in the Model Tuning Phase
  - Rapid Detection of Suspicious/Erroneous Inputs in the Model
  - In the case of Non-Compliance with Specifications
    - Problem identification
    - Proposed Solution (or at least indications to guide in its resolution)


V - 2025.5
Explorer #
ORBINAS ©
Helping in Nastran Analysis
ORBINAS ©
ene-26
CHECK
FEM Quality Check / Modal Analysis
Input Filename #
".bdf" Input File SPC
".dat" Input File
Basic Check yes
Modal Analysis B.C. Fixed
Thermo-Elastic Check no
Matricial Model
type
name
Shell Elm. Size / Aspect Rat. Checks Only when Nb of Shells < 250 000
Max Stresses (GRAV Loads)
Units Automatically Detected
Warped QUAD4 Splitted to TRIA3 Not Check Congruence TRIA3 / HEXA
Mass Breakdown Computing Method Standard (ELSUM for All Elements)
PowerPoint / Word Reports no
Preview
Model_Only
CHECK
OUPUT
Control File _CTRL.xlsx
Check Summary Results _SUMRY.xlsx
DynCond
Dynamic Condensation (Craig-Bampton)
Bulk Data Input Filename #
Output Files Format Standard
Number of Retained Modes
SPOINT Initial Range
External Grids Range (ASET)
OTM Generation
DynCond
OUPUT
Control File _CTRL.xlsx
Craig Bampton Condensed Model Generated
For further use, need of ".bdf" file containing External GRIDS
StatCond
Static Condensation (Guyan)
Bulk Data Input Filename #
Output Files Format Standard
OTM Generation
Remind --> ASET has to be defined in ".bdf" Input File
StatCond
OUPUT
Control File _CTRL.xlsx
Guyan Condensed Model generated
For further use, need of ".bdf" file containing External GRIDS
FreqResp
Frequency Response Analysis
Bulk Data Input Filename #
".bdf" Input File SPC
Frequency Range/Value Excit. Dir Z
% Critical Modal Damping 2.0
Enforced Unit Acceleration, with the considered SPC
Save DataBase no
Amplifications yes
Excit. Freq. at Mode [1]
FreqResp
OUPUT
Control File _fr_Z_CTRL.xlsx
Summary Results _fr_Z.xlsx
ModelComp
Model Comparison
Model_Comp
Work Directory
Model-1 Name
Model-0 (Reference) Name
Threshold (%) for Effective Mass Comparison 0.8
Nastran Already Run no
Save Nastran Results no
MSC F06Reader no
Graphical Tool for Preview MSC.Patran
Nastran_Standard_Run
Input Filename #
Optional Keywords
Process Additional Info about Analysis (Non Linear - SOL 106) no
Random Analysis
Random
Accel. entry in Freq. Resp. 9.81
".xdb" FreqRep Output File #
RMS requested Accelerations
Orbi-Nas
Set-Limits
Help
Utilities
Customize
Release_News

Launcher

OrbiNas Modules

- **OrbiNas is composed of 7 Modules:**
  - **Check**: FEM Quality Check & Modal Analysis (the main module – relevant percentage of the whole software).
  - **DynCond/StatCond**: 2 modules allowing a quick, easy, automatic and reliable generation of condensed matricial models, dynamically (Craig-Bampton) and statically (Guyan) respectively.
  - **FreqRep**: Module that should be used as complementary to "Check" module, when further dynamic analyses as Sine, Random or Time-history are foreseen.
  - **ModelComp**: Module comparing 2 models in terms of Dynamic Behaviour: frequencies and effective masses of normal modes are compared. Useful to validate a Condensed Matricial Model which has to be analogue to the original Physical one.
  - **Nastran_Standard_Run**: Easy way to run Nastran from OrbiNas, including Standard Model Check (*Model_Only*), quick access to Patran to visualise Model & Results, and giving in addition some interesting information about Nastran SOL Process, useful for debugging and DMAP developments. Additional Run information for Non-Linear Statics (SOL 106)
  - **Random Analysis**: Easy way and quick access into MSC.Prandom, to obtain Random Analysis Results from Frequency Response Analysis.

- **OrbiNas launcher description**
  - Once Inputs are given, filling the coloured cells, process is launched clicking the corresponding Button of the module.
  - For **Check** module, 3 buttons are available:
    - ***Preview*** : FEM visualisation in PATRAN
    - ***Model_Only*** : FEM Description, Input Data Information, bounds check, coherences, suspicious values...
    - ***Check*** : Model_Only + Check [5 run options: Basic; Basic + Modal; Thermo-elastic; Basic + Thermo-elastic; Complete Check] / Modal Analysis Run Alone
  - Other 4 sheets of OrbiNas Launcher EXCEL:
    - ***Set-Limits*** : Allows change threshold values considered as acceptable limits in the Checks (see next slide)
    - ***Help*** : Gives Information about Requirements for Use, Current Limitations, Input options and Navigation rules inside Check Results Output File (_SUMRY.xlsx)
    - ***Utilities*** : Complementary ToolKit (2) → [Generate Model By Property / Grid Relation]
    - ***Customize*** : Sheet to be updated by Customer, after installation, indicating OrbiNas and Referring Software versions and locations

OrbiNas 'Set-Limits' Sheet

Quality Shell Mesh Check					
Element Distorsion	Minimun Limit	Maximum limit	Units	Defaults	
Aspect Ratio (only QUAD4)		5	Dimensionless		5
Taper: QUAD4		0.5	Dimensionless		0.5
Warping Factor: QUAD4		5.0E-02	Dimensionless		0.05
Skew Angle: QUAD4	30		degrees	30	
Skew Angle: TRIA3	10		degrees	10	
Internal Angle: QUAD4	30	150	degrees	30	150
Internal Angle: TRIA3		160	degrees		160

FEM Numerical Quality Check - Free-free conditions / Stabilised conditions					
	Minimum Limit	Maximum Limit	Units	Defaults	
Free-free conditions					
Max - 3 Levels Strain Energy (PARAM,GRDPNT at CoG)		1.0E-03	N * m		1.0E-03
Rigid Body Modes - Highest Frequency Value		5.0E-03	Hz		5.0E-03
Lowest Elastic / Highest Rigid Frequency Ratio	10000		Dimensionless	10000	
Restrained conditions (with SPC)					
Stiffness Matrix Conditioning (Max. Ratio)		1.0E+07	Dimensionless		1.0E+07
Epsilon		1.0E-09	Dimensionless		1.0E-09
Balance between Applied Loads & Resulting Reactions (Ratio Residual/Applied)		1.0E-04	Dimensionless		1.0E-04

Thermo-Elastic Check - Homogeneous Heating - Alpha Constant & Iso-Static Conditions					
		Maximum Limit	Units	Defaults	
Rotations		1.0E-07	rad		1.0E-07
Stresses		100.	Pa		100.

Shell Elements - Mesh Size					
	Minimun Limit	Maximum limit	Units	Defaults	
Element Edge Length	0.001	1.000	m	0.001	1.000

Setting Limits (Units: N, m, kg, s)

Procedure to Change Limits Values

1 - Modify desired Limits (Minimum and/or Maximum)
(values have to be put in "SI" units, even when other units are used)

2 - Push "Set-Limits" Button to confirm the new limits

3 - Go Back to "Orbi-Nas" Sheet to run the Check

Remark

"OrbiNas-settings-BAK.txt" file will be created after the first run:

For further uses with the specified limits,
rename this file to "OrbiNas-settings.txt" before further runnings

Set-Limits

Model Comparison (ModelComp Module) - Allowed Maximum Deviations		
	Maximum Limit	Defaults
Frequencies	0.10%	0.10%
Cumulative Effective Masses	0.05%	0.05%
Effective Masses	0.50%	0.50%

Orbi-Nas | Set-Limits | Help | Utilities | Customize | Release_News | +

FEM Quality Check/Modal Analysis – Input

- **Input Nastran File** (for Model Only & Check)
  - ".bdf" Nastran Input file (Bulk Data Only) or ".dat" Nastran Input file (Complete, Run-able)

- **Main Inputs** (for Check)
  - SPC Set only in case of ".bdf" Input (for ".dat" Input, first encountered Case Control entries will be considered, for SPC, MPC, NSM, K2GG...)
  - If Modal Analysis is desired: Number of modes or Searching Frequency Range
  - If Thermo-Elastic check is desired: "yes" for this option
  - If Matricial Model is integrated: Type ("Craig-Bampton" or "Guyan") and Name of this model

- **Other Inputs** (Options)
  - Shell Element Size Check
  - Units
  - Mass Breakdown Computing Method
  - Max Stresses computed under Gravity Loads
  - Congruence check TRIA3/HEXA when Warped QUAD4 are splitted into 2 TRIA3 (this process could be performed in Thermo-elastic check)
  - Output request: PowerPoint Report Generation

- **Specific Switch** (for Development and Demos)
  - Nastran Already Run
  - Save Nastran Results

- **Inputs considered via External File** (for Check, in case of ".bdf" file)
  - If MPC, or other Case Control entries as NSM, K2GG, are considered in the Analysis, these Inputs must be previously put in a file named ".casex"

FEM Quality Check/Modal Analysis – Output – Aim

- **Output 1 – "_SUMRY.xlsx"** Excel File – This file integrates the whole information obtained from CHECK Results
  - When file is opened, the shown sheet is the first one: **"DIAG"**. This sheet summarizes the Result of the Checks:
    - Diagnostic and Main Potential Problems Found
  - It is composed of several sheets, being "Summary" sheet the one that summarizes main results and, if necessary, guides us in the analysis of it through Links leading to Other Sheets or External files
  - The number of these other sheets is variable, depending on: The number and type of checks/analyses requested – The obtained results
  - Navigation Rules inside this file are given in the Sheet '**Help**' of OrbiNas Launcher EXCEL
- **Output 2 – "_check_results.pptx"** PowerPoint File
  - This PowerPoint file extracts, from "_SUMRY.xlsx" Excel file, the relevant information useful for its integration into a Presentation or Report
- **Output 3 – "_modal_results.pptx"** PowerPoint File
  - This PowerPoint file extracts, from "_SUMRY.xlsx" Excel file, the Modal Results Information for its integration into a Presentation or Report
- **Output 4 – "_FEM_Descrip_Valid.docx"** Word File
  - This Word file is a FEM Description & Validation Report, summarising a quick description of the model and the results of the standard quality checks
  - It is a full structured report, which would only to be completed, adding some pictures of several FEM views
- **Output 5 – "_fem_images.pptx"** PowerPoint File
  - This PowerPoint is automatically generated when Users access to Patran from "_SUMRY.xlsx", clicking "**New .db**" Cell to visualize the model
  - It contains FEM Images on several views: Mesh, Materials, 2D Thicknesses (discrete ranges)
- **Aim of CHECK Tool**
  - Generate automatically Information in a format easy to integrate in a "FEM Description & Validation" and Analysis Report Documents
  - Help the analyst synthetizing relevant information about the model, advertising on input errors or potential problems, and then reducing tuning time
  - Save time in Post-Processing tasks
  - In case of Non-Compliance with Check Specifications, Highlight the problem and Guide the Analyst to Solve it

FEM Quality Check/Modal Analysis – "Summary" Sheet (1)

- **"Summary" sheet – General Description**
  - ***Summarises the results obtained for all requested checks***
    - If all is **Green** :
      - No Potential Problem has been found
      - Information in Other Sheets can be used for Documentation
      - Links (symbol #) driving to complementary information about the item
    - If Something appears in **Orange** or **Red** :
      - Potential Problem has been found (only the Red is a confirmed real problem)
      - Links (symbol #) to information related to the problem are activated (info can be situated in some other sheet or in external file)
      - Information given can be: Problem localisation, possible reasons of non-compliance (or at least help to find these causes)
  - ***"Summary" sheet content is organised in 4 Blocks,*** *summarising the Results of the 4 Check Types performed:*
    - **Block 1 – [FEM Information Summary; Elements Summary; Properties, Frequencies for the 1st mode free-free & constrained]**
      FEM General Information, including Properties Check, and for Shell Elements, Material Orientation and Off-Set Checks, 1st mode frequency
    - **Block 2 – [Mesh Quality Check]**
      Checks Element Distortion, but also Existence of Elements with Duplicate Topology, CELAS, CBUSH & RBE2 with non-zero length
    - **Block 3 – [FEM Numerical Quality Check]**
      Standard Checks considering, first a modal analysis in free-free conditions, and second a linear static with unitary gravity loads in constrained conditions
    - **Block 4 – [Thermo-Elastic Check] – If further Thermo-Elastic analysis is foreseen**
      Checks that when the model stands under isostatic conditions, considering homogeneous material, the stresses appearing in the structure, when heating it uniformly, are negligible

FEM Quality Check/Modal Analysis – "Summary" Sheet (2)

- **"Summary" sheet – Example struc01 – General**
  - **struc01_SUMRY.xlsx** file, is given with this presentation
  - It represents the Check Results of **struc01.bdf** submitted Input File
  - It is a **Complete Check**: Model + Basic + Thermo-Elastic + Modal (searching until 70 Hz)
  - Information has been generated in the 4 blocks
  - Information related to these 4 blocks has been generated and dumped into 26 **Sheets** (until a total of 31 Sheets can be created)
  - Many links activated in "Summary" sheet, but also in other sheets, lead to complementary Information, located inside the EXCEL, and outside, in some **external files**
  - In the given Excel, the external files cannot be reached, because, intentionally, directory containing it has not been delivered

- **"Summary" Sheet and Other Sheets Information**
  - In the next 4 slides, corresponding Outputs, for each of the 4 Blocks of the "Summary" sheet, are shown
  - All information, for all sheets, can be seen in detail, surfing the given Excel file
  - Besides, for each sheet, a summarized description of its contain, is given in slides 15 to 18

FEM Quality Check/Modal Analysis – "Summary" Sheet (3)
Block 1

FEM Information Summary				
Coordinates Bounds	2.500 2.500 5.000	Units - N, m, Kg, s		Coord. Systems 4 #
Number of Grids			3 780	
Number of d.o.f.s (F-set)			22 146	
Number of Elements			3 790	
Dominant Elements - 95 %			3 599	CQUAD4
TRIA3 / (QUAD4+TRIA3) Ratio		0.0%		
Grid Point Singularity Table #		*1 Grid is Probably Unconnected #*		
MASS (Kg)		2263.26	2263.26	2263.26
CoG (m)		.136	.038	2.285
First Elastic Mode Frequency Free-free (Hz) - 7th mode			3.0E+00	
First Constrained Mode Frequency (Hz) [Total : 46] #			2.88	
PARAM Values	AUTOSPC = YES			
	K6ROT = 100.			
	WTMASS = 1.			

Elements Summary	
Type	Number
CBAR	79
CBUSH	13
CELAS1	84
CHEXA	12
CONM2	3
CQUAD4	3 599
TOTAL	3 790

Properties	
	Number
Material Properties #	12
Physical Properties #	20
All Material Properties are Used	
All Physical Properties are Used	
Suspicious Values #	2

Rigid Elements / MPC Summary	
Type	Number
RBE2	3
TOTAL	3

Average Nb. of Elem. by Property	190

Shell Material Orientation (THETA / MCID Coherences) #	Normal Coherence #
Shell Off-Set Coherences #	PCOMP - Shell Off-Set

Then, Normal Check is Mandatory

FEM Quality Check/Modal Analysis – "Summary" Sheet (4)
Block 2

Shell Mesh Quality / Duplicate Elements / RBE2, CELAS, CBUSH Length				
SHELL ELEMENTS DISTORTION #	Min Value (Average) *Number*	Max Value (Average) *Number*	Minimun Limit	Maximum limit
Aspect Ratio - QUAD4	------		------	5.0
Taper - QUAD4	------		------	0.5
Warping Factor - QUAD4 [Therm-Elas. Check] #	*500 Worst Warped QUAD4 #*	2.24E-02 (2.60E-04) *500*	------	1.0E-05
Skew Angle - QUAD4		------	30°	------
Skew Angle - TRIA3		------	10°	------
Internal Angle - QUAD4			30°	150°
Internal Angle - TRIA3	------		------	160°
Element Connectivity #	Existence of Elements with Duplicated Topology #		0-Length RBE2 #	0-Length CBUSH
0-Length CELAS #	12 (0.80E-03)			

Solid Mesh Quality (TETRA, PENTA, HEXA)			
	Extreme Value (Average) *Number*	Minimun Limit	Maximum limit
Edge Length Aspect Ratio		------	100.0
Edge Point Length Ratio		0.50	------
Edge Point Included Angle		150°	------
Face Warp Coefficient		0.71	------
Jacobian Det. at Vertex - TETRA		0	------

BAR / BEAM Off-Set Length Ratio	
	BAR
	BEAM

FEM Quality Check/Modal Analysis – "Summary" Sheet (5)
Block 3

FEM Numerical Quality Check - Free-free conditions / Constrained conditions	Value	Minimum Limit	Maximum Limit	Units
Free-free conditions				
Max - 3 Levels Strain Energy (PARAM,GRDPNT at CoG) #	6.4E+03	------	1.0E-03	N * m
Rigid Body Modes - Highest Frequency Value - 6th mode #	2.5E-02	------	5.0E-03	Hz
Lowest Elastic / Highest Rigid Freq. Ratio . F7 / F6 #	120	10000	------	Dimensionless
Relevant Strain Energy Concentration Area can be Visualised in Patran posting the Group 'fail_strain'				
Constrained conditions - SPC / X, Y, Z Unit Gravity				
Stiffness Matrix Conditioning (Max. Ratio)	2.7E+05	*More Info #*	1.0E+07	Dimensionless
Epsilon	5.0E-11	------	1.0E-09	Dimensionless
Balance between Applied Loads & Resulting Reactions (Ratio Residual/Applied) #	3.9E-04	------	1.0E-04	Dimensionless
SPC Set Considered	1			

Model exhibits 6 Rigid Body Modes #

Load Path Check - Constrained conditions - SPC / X, Y, Z Unit Gravity	
Number of Elements with Zero Loads on All Directions #	4
These Elements can be Visualised in Patran posting the Group 'null_forces'	

FEM Quality Check/Modal Analysis – "Summary" Sheet (6)
Block 4

Thermo-Elastic Check - Homogeneous Heating (ΔT=100°C) - α Constant (1.E-5 °C^{-1}) and Iso-Static conditions				
Iso-Static Conditions Automatically Computed	Maximum Value	Localisation	Maximum Limit	Units
Rotations #	4.4E-06	GRID 21198	1.0E-07	Radian
Stresses #	30611	CQUAD4 21122	100	Pa
Alpha: Some value Not Specified or Suspicious #	Tref: Some value Not Specified #		*High Stresses are NOT Due to Warping*	
When splitting on whole FEM, each CQUAD into 2 TRIAs, stresses don't decrease	*29773*		*Maximum Rotation*	*3.09E-06*

FEM Quality Check/Modal Analysis – Sheets Content (1)

Sheets Content

For each sheet, the information listed below is present in the sheet, or can be acceded from this sheet

- *Characteristics*
  - FEM Main Characteristics (Type, Bounds, Mass); Global Mass Breakdown; Structural Composition (Volume & Mass Breakdowns by Material)
- *Model_Misc*
  - Documentation tables for "FEM Description" ; CELAS & RBE2 Length; Shell Material Orientation & Off-Set Checks: detailed information
- *Model_Prop*
  - Material & Physical Properties: Input value for each field; Material Identification; PBARL/PBEAML Section; PCOMP Type & Characteristics Information
- *Properties*
  - Material/Physical Properties – Summary & Bounds Check (Material: Only Linear Elastic and No Temperature Dependent Materials)
- *Free-free*
  - Results of modal analysis (the first 20 modes) in free-free conditions: Check Rigid body modes (number and freq. values) and Strain Energy values
- *Constrained*
  - Results of linear static analysis considering unitary gravity loads in constrained conditions: equilibrium, max ratio, load path checks
- *Thermo-Elas*
  - Input Checks: α, Tref – Guidelines – Warning if some potential problem is detected related to further Thermo-Elastic Analysis
  - Results: Thermo-Elastic Check considering homogeneous heating ($\Delta T = 100°C$), with α constant (1.E-5 $°C^{-1}$), under isostatic conditions
  - In case of non-compliance, higher stresses are localized, possible causes are listed, and more probable ones are highlighted
- *Rel-Warp*
  - This sheet is activated, if warped CQUAD4 elements are found. These CQUAD4 are listed as well as related entities, in order to localize the problem
  - Also, a derivate model, obtained splitting each warped CQUAD into 2 TRIAs, is generated and run, to confirm if warping can be the cause of high stresses

FEM Quality Check/Modal Analysis – Sheets Content (2)

- ***Modal 46*** *(The name of the sheet indicates the number of requested modes)*
  - Modal Analysis Results : Frequency range; Elastic Mass Percentage reached in the computed range; Main modes identification.
    Complete Table for the computed modes: frequency, effective masses (value and %); Modal density.
- ***Eff_Mass***
  - Table for the computed modes: frequency, effective masses (value and %) – 2 different formats
- ***Eff_Mass_Gr***
  - Mx, My, Mz Effective Masses Graphs (value and cumulative)
- ***Freq_Margin***
  - Assessment of modes frequency values evolution, considering several mass margins (with the hypothesis of a homogeneous variation of the mass for the whole FEM)
- ***Zero_Load***
  - In Gravity Load Check, load path is verified identifying Elements with Zero Loads on all directions (processing Forces/Moments or Stresses). These elements are listed and can also be easily plotted in MSC.Patran, by means of a group previously created.
  - A special treatment is done to CBUSH Elements, where Zero Load is verified for each Translation direction. Besides, concerning the Grids connecting these detected CBUSH, a searching of all entities related to the GRID is performed.
- ***Grav_Stress***
  - Maximum Stresses Assessment in Gravity Check X, Y, Z dir.
  - In this case (struc01), the considered load is 5 g
- ***Mesh Size*** *(Shell Elements)*
  - Length of smallest and largest edge. Percentage of Elements inside a given range size, below & above
  - Dynamic Shell Mesh validity: frequency threshold
  - For Explicit FEM Analysis: maximum time step Δt to avoid un-stability in the integration scheme
- ***Warning***
  - Suspicious values for Material & Physical Properties; Unused Physical Properties; Unused Material Properties

FEM Quality Check/Modal Analysis – Sheets Content (3)

- *Grid-Singularity*
  - Found dofs in Grid Point Singularity Table are organized in 2 lists: dofs with Zero Stiffness & dofs with Non-zero Stiffness
- *Rel-Strain*
  - This sheet is activated, when Strain energy is above the acceptable limit in at least one of the 3 Nastran-Set levels (G, N, F)
  - It helps to find the problem, tracking entities related with the Grids with more contribution and suggesting possible causes
- *For-Info*
  - Remind of some other Checks that have to be done, not performed automatically in OrbiNas, requiring visualisation with a Pre/Post Graphical tool
  - Miscellaneous information regarding specific modelisation and possible Nastran and/or OrbiNas limitations
- *Bulk-Sum*
  - FEM Bulk Data Entries organized by Type of entries.
  - Input Files Summary: Include Input Structure or Single File
- *Coord_Sys*
  - Coordinate Systems present in the Model. Indication of Not used ones. Entities Referring to each Coordinate System.
  - For delivery purposes (for a further integration of the FEM at higher system level), an anticipate diagnostic about the success of 'Displaced & Rotated Modal Test' can be done. Then, the corresponding CS, that has to be considered as Main, is indicated.
- *Prop_Mass*
  - Mass Breakdown Table: Physical Properties Assignment indicating corresponding Item, Material, Structural Mass, NSM, Nb of Elements, Numbering Range
- *Prop_No_Mass*
  - Physical Properties Assignment (PBUSH – PELAS – PGAP) – Stiffnesses, Related CBUSH / Coord. Systems , Nb of Elements, Numbering Range
- *Used_Materials*
  - Table integrating only Materials used in the FEM. Composite Material is considered as a Single Material – Prepared for MSC.Patran Material Visualisation

FEM Quality Check/Modal Analysis – Sheets Content (4)

- **Other Sheets Not Appearing in the presented example**
  - *FATAL*
    - When one of the Nastran Run, related with the requested check, has finished with a Fatal Error. Message error is located, isolated and highlighted
  - *MATS1*
    - Stress-Dependent Materials Inputs - Information and Check: MATS1 & TABLES1(for Stress-Strain Curve option) entries
    - Warning if:
      - Unused Material
      - Suspicious values are found for Yield Stress Limit or Hardening Slope
      - No existence of referenced TABLES1 entry
      - Both Stress-Strain Curve and Hardening Slope are specified
    - Stress-Strain Curve TABLES1 Plot
  - *MATT1*
    - Temperature-Dependent Materials Inputs - Information and Check: MATT1 & TABLEM1 entries
    - Young Modulus & Thermal Expansion Coefficient are supported
    - Warning if:
      - Unused Material
      - No existence of referenced TABLEM1 entry
    - E (Temp) - Alpha(Temp) Plots
  - *Freq-Rep*
    - Inputs information and check Entries related with Frequency Response Analysis: RLOAD1, RLOAD2, Damping, Random
    - Excitation Type Identification: Applied Load or Enforced Motion; For Random verifies if PSD is an Auto Spectral one
    - Plot or Warning if no-existence of: TABLED1 for RLOAD1, RLOAD2; TABRND1 for RANDPS (PSD); TABDMP1 for Modal Damping
  - *Rel-Sing*
    - This sheet helps to find problems related with Grid Point Singularity Table
    - In general, it is activated with "Rel-Strain", when Strain Energy is above limit, and when this can be due to existence of Near-Singular Dofs Not Constrained

Condensed Matricial Models Generation – Inputs

- **Inputs** (for DynCond & StatCond)
  - **Nastran Input file** (Bulk Data Only)
  - **Output Files Format**
    - ***Option 1: Standard*** → matrices are generated in Standard Nastran format (files 'op4', '.asm', '.pch')
    - ***Option 2: Via DMAP*** → matrices are generated in NASTRAN OUTPUT4 extended precision, ASCII format, Format 3E23.16, as specified in Astrium Document "Generic Mechanical Specification"-ADS.E.0787 (files 'matcb'/'matas', '.part', '.spoint')
  - **OTM Generation** (available only for Output files Standard Format)
    - Input: requested displacements, stresses, forces,… for which Output Transformation Matrices will be generated

- **Other Inputs for DynCond** (Craig-Bampton Modal Synthesis)
  - **Retained Modes.** This Input drives the size/accuracy of the generated condensed model;
    Modal Synthesis is a "base change truncature" method → exact if all modes are retained
  - **External Grids Range (ASET).** Set of External or Retained Physical Dofs
    - In general, due to a logic of imposed numbering within components, input is easy; for example: 1,thru,159
    - In case of a "more random" numbering, as mentioned in cell commentary, **entering 0**, the **".aset" file**, containing ASET entries, will be used
  - **SPOINT Initial Range**. SPOINT are necessary to define modal degrees of freedom. Total number=number of retained modes

- **Other Inputs for StatCond** (Guyan)
  - In Static Condensation, only physical dofs are considered. No more Inputs are needed
  - Nevertheless, as reminded in the launcher: **A-Set entries must be included in ".bdf" file**

Condensed Matricial Models Generation – Outputs

- **Outputs – 'Standard' Option (for DynCond & StatCond)**
  - **".op4"** → Stiffness & Mass Matrices corresponding to dofs: Physical + Modal (for DynCond) + requested OTM (Output Matrices Transformation), optionally
  - **".asm"** → Assembly Punch File
  - **".pch"** → Standard Punch File

- **Outputs – 'Via DMAP' Option for DynCond → Craig-Bampton Condensed Model**
  - **".matcb"** → Stiffness & Mass Matrices corresponding to [Physical+Modal] dofs defined in the partitioning vector
  - **".part"** → Partitioning Vector : Physical + Modal dofs retained
  - **".spoint"** → Scalar Points representing retained Modal dofs

- **Outputs – 'Via DMAP' Option for StatCond → Guyan Condensed Model**
  - **".matas"** → Stiffness & Mass Matrices corresponding to External dofs defined in the partitioning vector
  - **".part"** → Partitioning Vector : External Physical dofs retained

Frequency Response Analysis & Model Comparison

- **FreqRep**
  - FreqRep performs a Quick Frequency Response Analysis, considering an Enforced Unit Acceleration applied in Constrained Area.
  - It is strongly recommended to use it, as a complementary Check, before any dynamic analysis.
  - Analysed results:
    - Check that responses at Zero Hz, match those providing from Static Excitation
    - For each Grid, Maximum Amplification Factor and at what frequency it appears
    - Amplifications for each Grid at all frequencies can be consulted (TF is plotted for the GRID with the maximum amplification)

- **ModelComp**
  - In the next slide, can be seen an example of Model Comparison ("Summary" sheet & extract of "ComparFem" sheet).
  - **"struc"** is the original model (the Reference). Whole the model is Physical.
  - **"stru_c100"** is the Target model. One of the components (the cone) of this model is a matricial model (this matricial model has been previously generated, condensing the original physical cone of "stru", via Craig-Bampton Modal Synthesis method, considering I/F nodes as Physical dofs + 100 Modal dofs). The rest of the components are the original physical ones.
  - The result is that *"Compared Models Cannot be Considered Equivalent"*, due to non-compliance regarding Individual Effective Masses (OK for Frequencies and OK for Cumulative Effective masses)
  - Probably, if we had retained 200 modes of the cone (instead of 100), it would be fine.
  - In any case, the acceptable deviation values considered here are very severe (intentionally to show non-compliance)

OrbiNas - V-2023.5

stru_c100

FEM Models Comparisons

General Information / Assumptions

Target Model Name	stru_c100
Reference Model Name	struc
Number of Computed/Compared Modes	30
Frequency Range	2.920 - 46.904
All effective masses with significative weight (above .8%) are compared	

	Mx	My	Mz	Ix	Iy	Iz
Cumulative Effective Masses	17%	22%	8%	39%	27%	33%

Maximum Deviations in the Model respect to the Referenced one

	Component	Deviation Value	Maximum Allowed
Frequencies	-	0.11%	0.20%
Cumulative Effective Masses	Mx	0.01%	0.50%
Effective Masses	Iy	0.91%	0.70%

Compared Models Cannot be Considered Equivalent from Dynamic Behaviour point of view

Summary | ComparFem | stru_c100 | struc

```
=== FEM Models Comparison ===   stru_c100

Reference Model:  struc

* All the   30 computed frequencies are compared
* All effective masses with significative weight (above   .8%) are compared
* Frequency Range :      2.920    -    46.904

=== Maximum deviations in the model respect to the referenced one

    Frequencies         :   .11%              in mode    4
    Cumul Eff. Masses   :   .01%  [Mx 17%]
    Effective Masses    :   .91%  [Iy  2%]   in mode   24

    Mx Effective Masses :   .10%  [Mx  1%]   in mode   26
    My Effective Masses :   .55%  [My  2%]   in mode   22
    Mz Effective Masses :   .38%  [Mz  1%]   in mode   15
    Ix Effective Masses :   .91%  [Ix  3%]   in mode   22
    Iy Effective Masses :   .91%  [Iy  2%]   in mode   24
    Iz Effective Masses :   .74%  [Iz  2%]   in mode   16

Maximum deviation for Frequencies.......  <  .20%
Maximum deviation for Cumul Eff. Masses..  <  .50%
Maximum deviation for Effective Masses...  >  .70%

* Compared Models Cannot be Considered Equivalent from Dynamic Behaviour point of view
```

FEM Models Comparison — stru_c100
Frequencies and Effective Masses — struc

Mode	Freq	Dev	Mx %T	Mx Dev	My %T	My Dev	Mz %T	Mz Dev	Ix %T	Ix Dev	Iy %T	Iy Dev	Iz %T	Iz Dev
1	2.920	-			1%	-			2%	-			2%	-
2	4.928	-												
3	6.749	-.03%			5%	-			4%	.01%			10%	.01%
4	7.482	-.11%	4%	.02%							6%	-.03%	7%	-.01%
5	9.476	-												
6	12.388	-					1%	.01%						
7	15.332	-							1%	-.04%			2%	-.04%
8	17.244	-												
9	17.750	-.01%												
10	18.665	-.07%												
11	19.191	-.06%												
12	19.591	-.02%											1%	.30%
13	19.848	-												