OptiStruct NVH 해석에 도움이 되는 Special Features 소개

한국 알테어 김한구

OptiStruct basic run option

: 메모리관리 / 고유치 솔버

Altair

OptiStruct basic run option

- 메모리관리

Memory assignment

- OptiStruct will automatically estimate the required memory – see .out and .stat file

OptiStruct has different memory options:

- In core – fast
  - Performs all mathematical operations in memory
- Out of core – slow
  - Writes information to disk
  - Use when solution requires more memory than you have available
- Minimum core – very slow
  - Writes even more information to disk
  - Use when you don't have enough memory to solve out of core
- **Wherever possible, run OptiStruct with enough memory for In-Core solution**
  - Assign more memory than default with option –len memory_mb, –maxlen upper hard limit ex) –len 50000

Scratch area

- Whilst calculating the solution, OptiStruct will write data to the disk
- Default is the location where the fem file is stored
  - Can be modified with configuration file OptiStruct.cfg or run option -scr
- If read/write speed to scratch area is slow, this can significantly impact solution time
  - **Avoid having scratch area on network drives as this can slow down the solution**

```
 ***************************************************************************
 **                                                                       **
 **                                                                       **
 **                        OptiStruct 2017.2.3                            **
 **                                                                       **
 **             Advanced Engineering Analysis, Design and                 **
 **         Optimization Software from Altair Engineering, Inc.           **
 **                                                                       **
 **                                                                       **
 **                 Windows 8.1  (Build 9200)  ALKR0158                   **
 **          8 CPU:  Intel(R) Core(TM) i7-4712HQ CPU @ 2.30GHz            **
 **                    9351 MB RAM,  18719 MB swap                        **
 **                                                                       **
 ** Build tag: 0913157_9433xxx_Ce64RBW8UH14M:142311-2  4000020004000      **
 ***************************************************************************
 **  COPYRIGHT (C) 1996-2018                 Altair Engineering, Inc.     **
 ** All Rights Reserved.  Copyright notice does not imply publication.    **
 **         Contains trade secrets of Altair Engineering, Inc.            **
 ** Decompilation or disassembly of this software strictly prohibited.    **
 ***************************************************************************

MEMORY ESTIMATION INFORMATION :
-------------------------------

 Solver Type is:  Sparse-Matrix Solver
                  Direct Method

 Current Memory (RAM)                                        :     800 MB
   In addition a ramdisk area was allocated in memory        :      10 MB
 Estimated Minimum Memory (RAM) for Out of Core Solution     :      32 MB
 Recommended Memory (RAM) for Out of Core Solution           :      32 MB
 Recommended Memory (RAM) for In-Core Solution               :      32 MB
 Recommended Number of Nodes for SPMD Load Decomposition Run :   >   3
 (Note: The Minimum Memory Requirement is limited by Output Calculation.)

DISK SPACE ESTIMATION INFORMATION :
-----------------------------------

 Estimated Disk Space for Output Data Files                  :       6 MB
 Estimated Scratch Disk Space for In-Core Solution           :       1 MB
 Estimated Scratch Disk Space for Out of Core Solution       :       1 MB

***************************************************************************
```

OptiStruct basic run option

- 고유치 솔버소개(Lanczos, AMSES)

Solver run time can be affected by the use of solver (e.g. Lanczos vs AMSES)

- Lanczos in not efficient for solving large eigenvalue problems
  - Lanczos calculates the entire eigenvector which leads to huge amounts of disk I/O (I/O time and disk space)
  - Will not work for thousands of modes for millions of DOF

- The OptiStruct large scale eigensolver (AMSES) is very efficient
  - AMSES is short for Automatic Multilevel Substructuring Eigen Solver
  - Only the portion of the eigenvector needed for output degrees of freedom are calculated
  - For modal FRF or transient analysis:
    - The load vectors are reduced into modal space
    - Structural Damping (GE) matrix reduction to modal space
    - Residual Vector (RESVEC) generation is done automatically
  - Solutions for frequency response, transient response, and CMS Super Element creation
  - Is SMP parallel for efficiency

OptiStruct basic run option

- 고유치 솔버소개(Lanczos, AMSES)

Solver run time can be affected by the use of solver (e.g. Lanczos vs AMSES)

- EIGRL (Lanczos)
  - No approximation of the eigenvectors
  - Full eigenvector calculation
  - Leads to large run times for large problems
  - Limited to about 2000 modes

- EIGRA (AMSES)
  - Approximate solution
  - Accurate enough for NVH
  - Partial eigenvector calculation
  - Efficient reduction to modal space
  - No additional cost over EIGRL (both are 25 HyperWorks Units)
  - Can be used on both Windows and LINUX computers

EIGRA

Bulk Data Entry

EIGRA – Real Eigenvalue Extraction Data using Automated Multi-Level Sub-structuring

Description:
Defines the data need to perform eigenvalue analysis with the Automated Multi-Level Sub-structuring technique

Format:

(1)	(2)	(3)	(4)	(5)	(6)	(7)	(8)	(9)	(10)
EIGRA	SID	V1	V2	ND		AMPFACT		NORM	

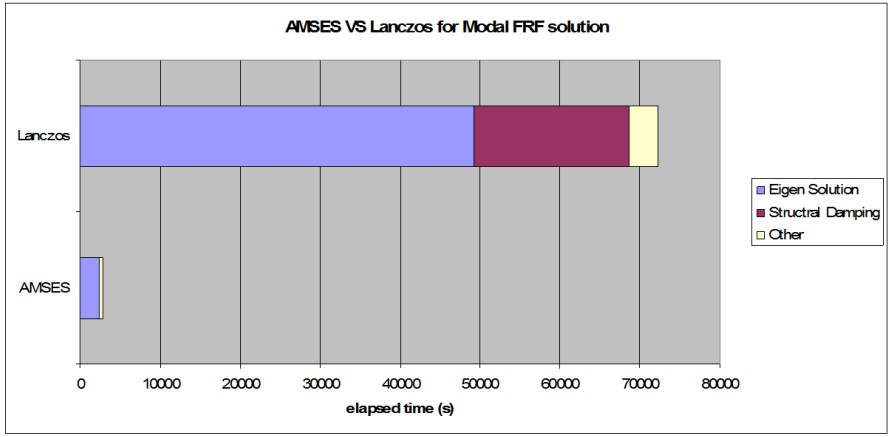
OptiStruct basic run option

- 고유치 솔버소개(Lanczos, AMSES)

Solver run time can be affected by the use of solver (e.g. Lanczos vs AMSES)

➢ AMSES runs with Windows or LINUX

➢ AMSES runs on 32 bit or 64 bit machines



• 2.2 Million Degree of freedom, **1369** modes, w/ structural damping, 200 excitation frequencies, 1CPU
• Dell T7500, Xeon X5550 2.67GHz, 12GB RAM, regular disk, 64bit Windows Xp

OptiStruct NVH special features

- 고유치 솔버소개(Lanczos, AMSES), Faster FRF 솔버

OptiStruct also offers a **FASTFR** solver for **Large Scale Modal Frequency Response**

- Fast Frequency Response Solution based on Thousands of Modes with Viscous and/or Structural Damping
- Works with Frequency Dependent Materials
- Multi-threaded with nearly linear speedup
- No Additional Cost
- Runs on Windows and Linux
- **Controlled by PARAM, FASTFR**
- FASTFR has some special technique to diagonalize the modal FRF equation.

PARAM, FASTFR

Parameter	Values	Description
FASTFR	<YES, NO> Default = AUTO	If, AUTO (Default), the faster method is automatically chosen by the program for Modal Frequency Response Analysis. If the FASTFR method is not activated, the standard method is used (see comments). If YES, and if FASTFR is allowed (see comments), it activates an alternative method to run Modal Frequency Response Analysis that enhances the performance. If NO, the FASTFR method is deselected and the program is run using the standard solution method. **Comments**: The FASTFR method **will be ignored** for Modal Frequency Response Analysis, if: 1. Multiple modal spaces exist in the model. 2. Single Program, Multiple Data (SPMD) parallelization has been requested. 3. Viscous degrees of freedom are significantly large. The FASTFR method will be allowed, if Shared Memory Parallelism (SMP) parallelization has been requested by specifying the number of processors using the `-nproc` run option.

Altair

OptiStruct NVH special features

: Automated Peak Response(PEAKOUT) 활용

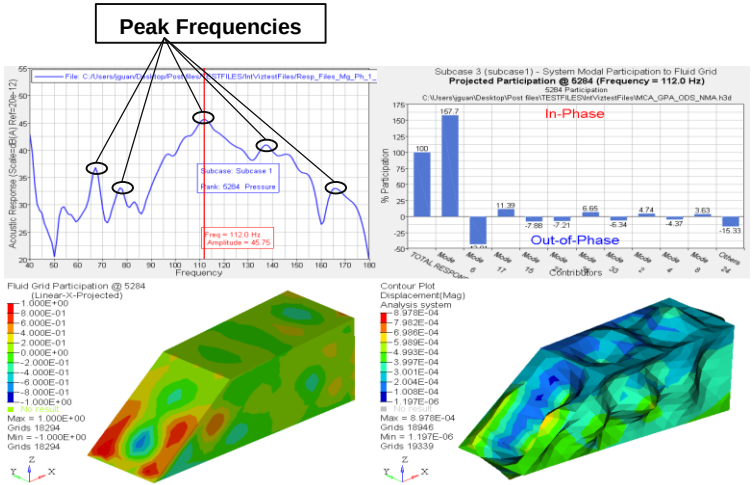
Altair

OptiStruct NVH special features

- Automated Peak Response(PEAKOUT) 활용

The PEAKOUT feature can be use to automatically identify peaks in frequency response results and extract further results at these frequencies

  - No need to do this in 2 separate simulations
  - Find response node peak and then write out all node displacements at this frequency



OptiStruct NVH special features

- Automated Peak Response(PEAKOUT) 활용
  - To effectively diagnose the NVH problems require multiple output requests
    - Operating Deflection Shape
    - Modal participation factor
    - Grid Participation Factor
    - Energy Contours
    - Power flow, Stress,.. many NVH results
  - Huge Result Files (~10-100 GB) when results are requested for all frequencies
    - Post processing performance suffers
    - Data management becomes challenging
  - Using PEAKOUT card
    - Reduction of file size on the order of 10-40 times
    - Huge performance advantages without any loss of accuracy

OptiStruct NVH special features

- Automated Peak Response(PEAKOUT) 활용
  - PEAKOUT operates through identifying peaks in frequency response
  - This can be used to output detailed results for peak response frequencies in the same run
  - Peak control parameters
    - Number of PEAKS
    - Cutoff value
    - Min, Max Spacing

Bulk Data Entry

PEAKOUT – Peak Identification Criteria

Description

Defines criteria used for the automatic identification of loading frequencies at which result peaks occur. Other result output may then be requested at these "peak" loading frequencies. This feature is only supported for frequency response solution sequences.

Format

(1)	(2)	(3)	(4)	(5)	(6)	(7)	(8)	(9)	(10)
PEAKOUT	SID	NPEAK	NEAR	FAR	LFREQ	HFREQ	RTYPE	PSCALE	
	GRIDC	GID1	CID1	CUTOFF1	GID2	CID2	CUTOFF2		
		GID3	CID3	CUTOFF3	...	...	...		

OptiStruct NVH special features

- NPEAK : Desired number of peaks. Default = 5 (Integer > 0)

- NEAR : Minimum allowed distance between two peaks

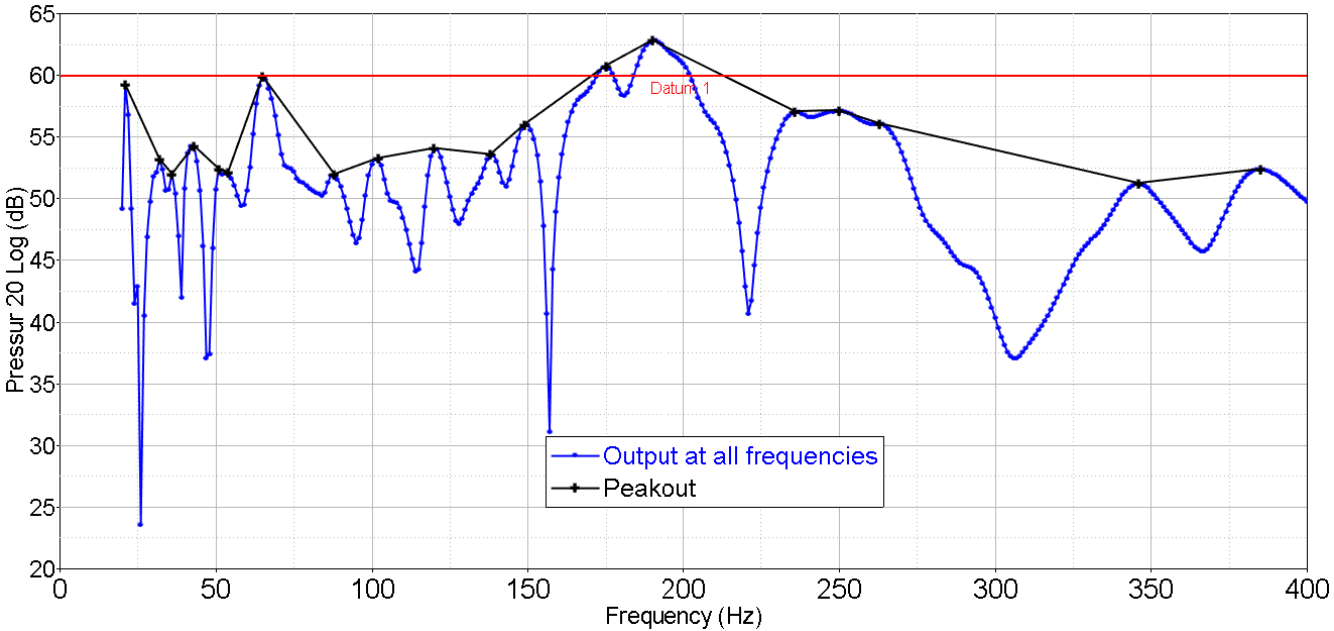
If two peaks are closer than this value, the loading frequency of the lower peak will be ignored

- FAR : Maximum allowed distance between two peaks. Additional peaks will be selected (in addition to NPEAK) if the distance between the peaks is greater than this value

- CUTOFF# : The cutoff value can be a real value or an integer value

OptiStruct NVH special features

- Automated Peak Response(PEAKOUT) 활용
  - Sample PEAKOUT Results



OptiStruct NVH special features

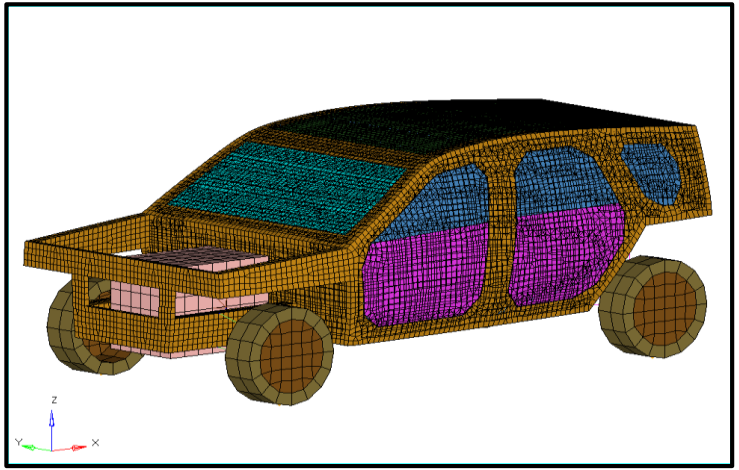
- Automated Peak Response(PEAKOUT) 활용

File Name and Location:

`…\PEAKOUT_AUTO\BaseModel.fem`

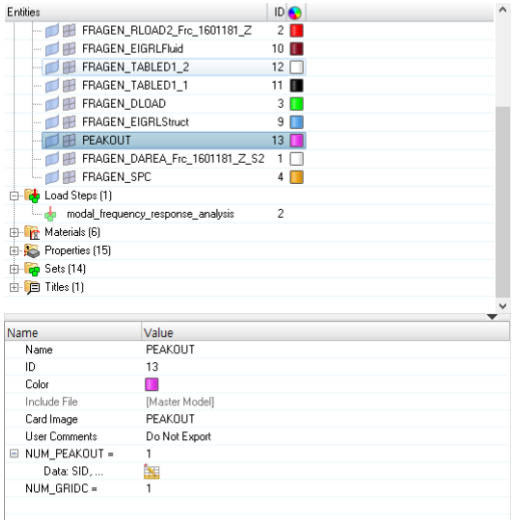
Objectives:

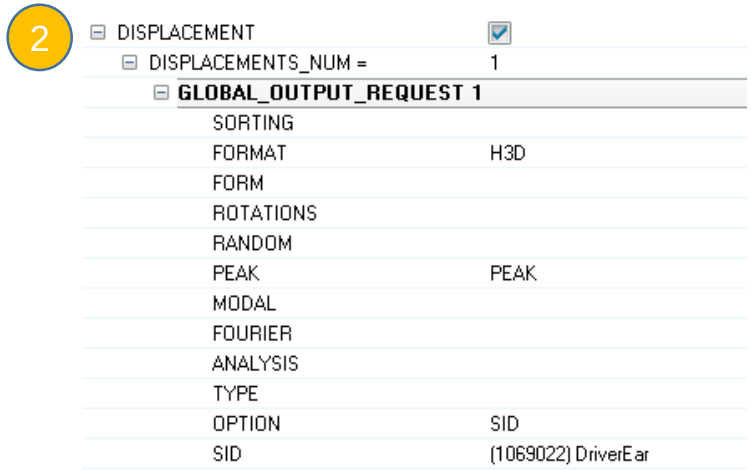
- Import **`BaseModel.hm`** in HyperMesh Desktop
- Create PEAKOUT load collector named **`PEAKOUT`**
  - NUM_PEAKOUT: **`1`**
  - NPEAK: **`10`**
  - **LFREQ**:10.0Hz **HFREQ**:100.0Hz
  - RTYPE: **`DISP`**, RPSCALE: **`DBA`**
  - GID(0): **`1081121`**, CID(0): **`1`**, RPC_OPT: **`VALUE`**
- Update the subcase to include **`PEAKOUT`**
  - PEAKOUT: **`<checked>`**, ID: **`PEAKOUT`**
- Edit a DISPLACEMENT control card
  - FORMAT(1): **`H3D`**, PEAK(1): **`PEAK`** ,OPTION(1): **`SID`**, SID(1): **`DriverEar`**
- Run the Analysis as **`BaseModel_peakout.fem`**
- Post-process the Results

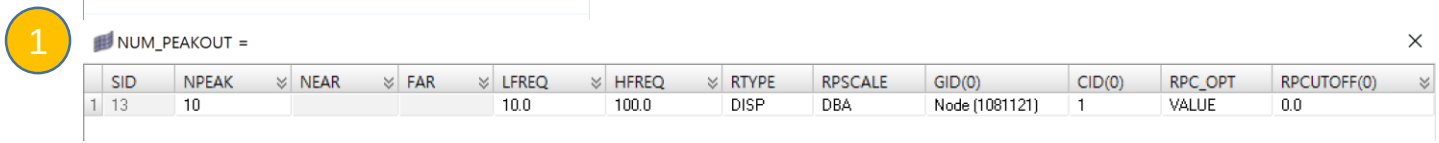


OptiStruct NVH special features

- Automated Peak Response(PEAKOUT) 활용







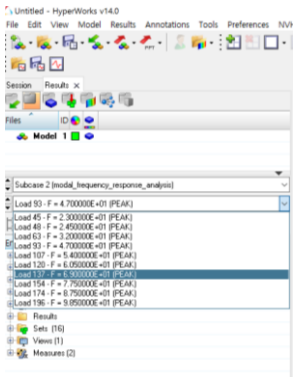
OptiStruct NVH Faster Solutions for Optimization

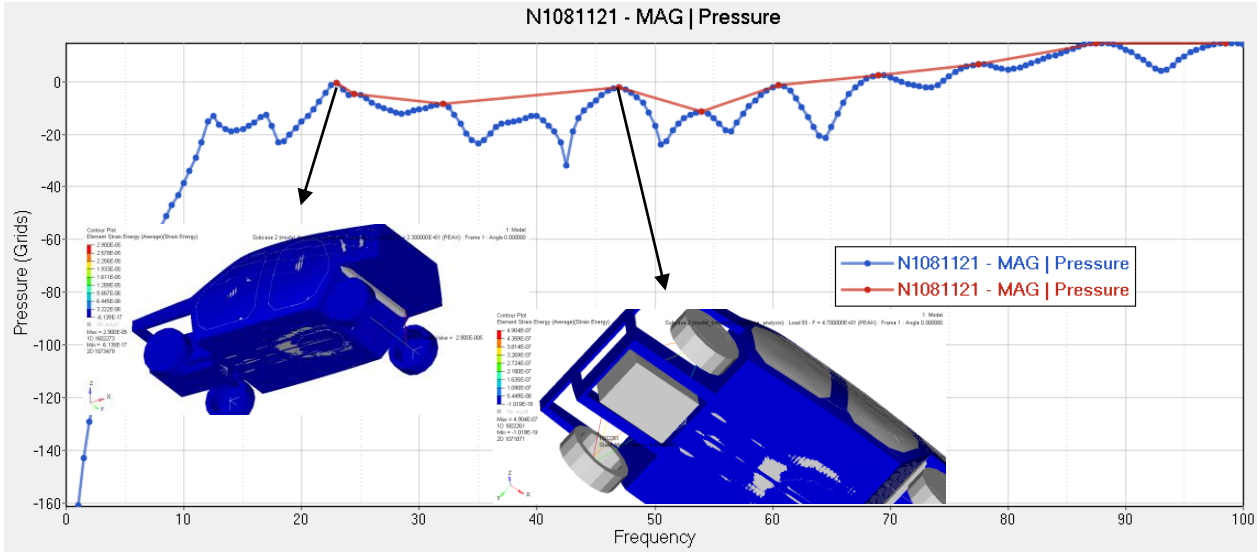
- Automated Peak Response(PEAKOUT) 활용

Exercise Results: input >> BaseModel_peakout.fem

- File: `BaseModel.h3d vs BaseModel_peakout.h3d`
- Plot: `Frequency` vs. `Pressure (Grids) of node 1081121(DBA Reference value 20e-12)`
- Contour: `element strain energy`

```
PEAKOUT = 13
DISPLACEMENT(H3D,PEAKOUT) = 1069022
ESE(H3D,PEAKOUT) = ALL
PEAKOUT      13     10                    0.0DISP    DBA
+       GRIDC    1081121      1     0.0
```





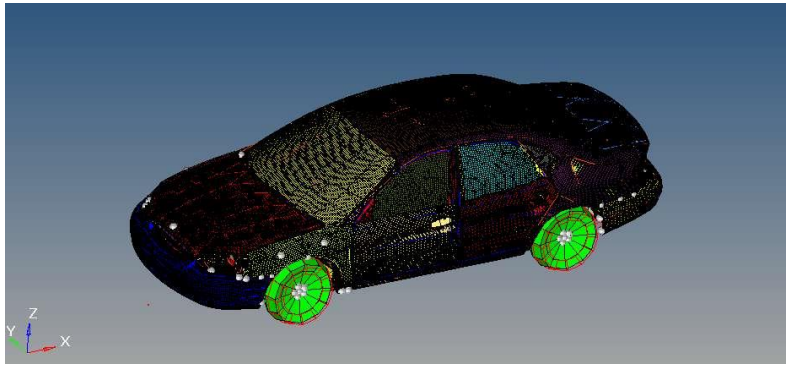
Altair

OptiStruct NVH special features

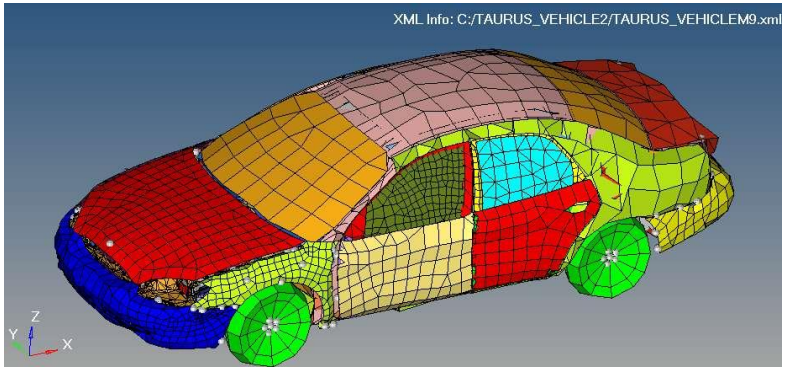
: Coarse Display 모델링과 해석응용

OptiStruct NVH special features

- Coarse Display 모델링과 해석응용
  - Coarse Display Models can be utilized to simplify post-processing and review



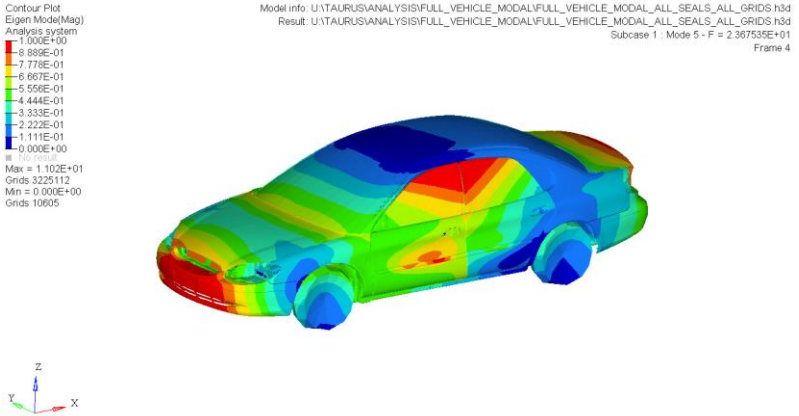
Detailed Mesh



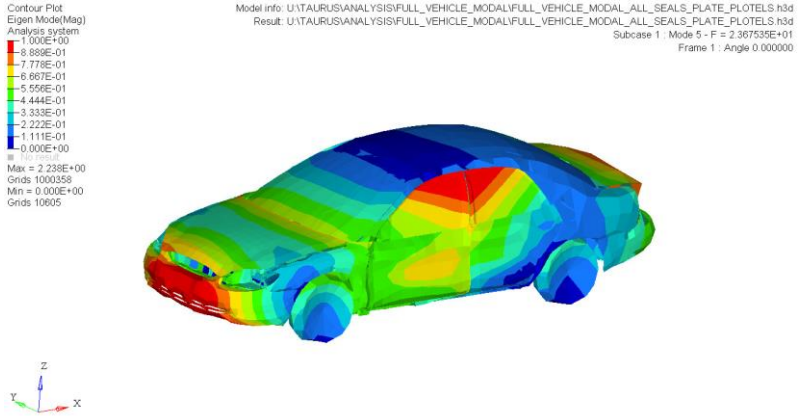
Coarse Display Mesh

OptiStruct NVH special features

- Coarse Display 모델링과 해석응용
  - Coarse Display Models do not result in a loss of result fidelity



Results on Detailed Mesh

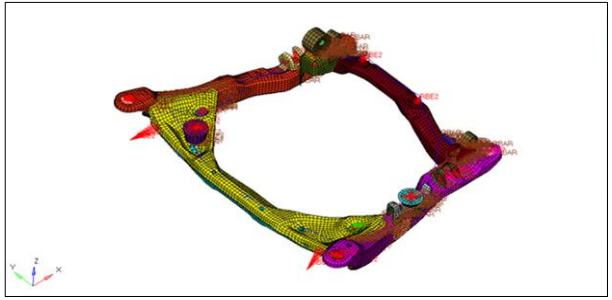


Results on Coarse Display Mesh

Altair

OptiStruct NVH special features

- Coarse Display 모델링과 해석응용
  - Normal Mode – Front Subframe in Automotive
    - Use PLOTEL3/4 Plate element to reduce results size for mode shape



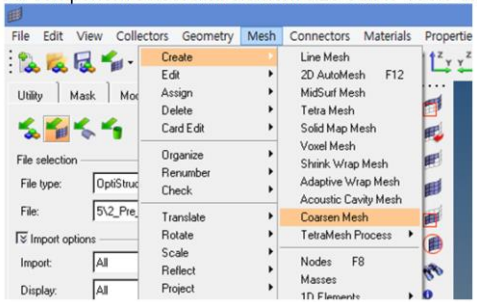
File Name and Location:

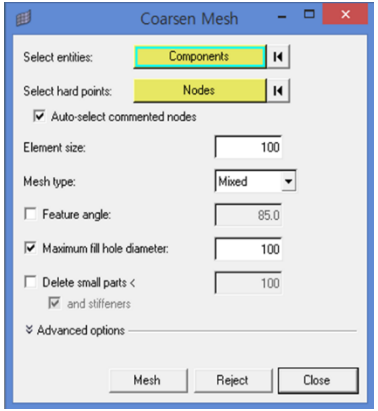
`…\Coarse_Display\subframe.fem`

Objectives:

- Free Free Normal Mode
- Import **subframe.fem** in HyperMesh
- Use Coarse Mesh Utility
  - Element Size **100**

Mesh full down menu → Create → Coarse Mesh
All component select and element size edit to 100.0 then hit Mesh button





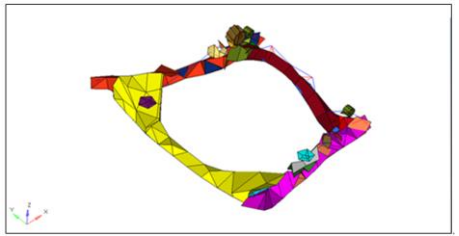
Altair

OptiStruct NVH special features

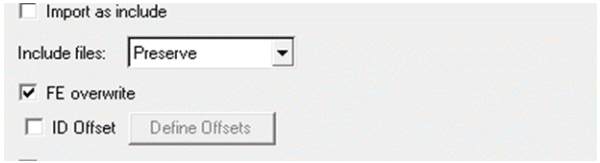
- Coarse Display 모델링과 해석응용
  - Normal Mode – Front Subframe in Automotive
    - Use PLOTEL3/4 Plate element to reduce results size for mode shape

Objectives:

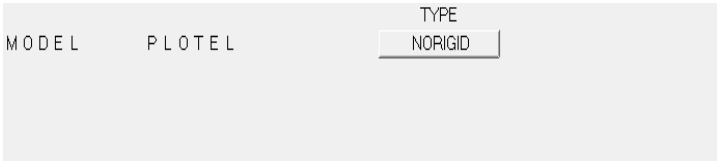
- New coarse model is PLOTEL3/4 and original model is completely deleted.
- You can export **subframe**.fem model
- Do Not Save this Model with the original file name. You will lose your detail model if you don't give it a new name.
- Then you can import original model (normal mode) file to HM. You have to use "**FE overwrite**" for import option.
- **MODEL** card creation (Analysis Page → Control Card → MODEL)



>> 생성된 PLOTEL3/4



>> Import 옵션 "FE Overwrite"



>> PLOTEL 모델 저장

OptiStruct NVH special features

- Coarse Display 모델링과 해석응용
  - Normal Mode – Front Subframe in Automotive
    - Use PLOTEL3/4 Plate element to reduce results size for mode shape

Detailed model vs coarse display model

Same mode shape and much smaller result file size