δYNAMIS: A NEW SOLVER FOR LINEAR AND NONLINEAR FINITE ELEMENT MODELS

[1]**Dimitrios Giagopoulos**, [2]**Sotirios Natsiavas***
[1]DYNAmical Systems TechnologY, Thessaloniki, Greece
[2]Department of Mechanical Engineering, Aristotle University, Thessaloniki, Greece

KEYWORDS - automated finite element modeling, substructuring, static and transient analysis, linear and non-linear frequency response, deterministic and random analysis.

ABSTRACT - **δ**YNAMIS (**DYNA**mics of **M**echan**I**cal **S**ystems) is a numerical code leading to an automated determination of the response of mechanical systems, in an efficient way. These systems are geometrically discretized by finite elements. Frequently, this leads to models with a quite large number of degrees of freedom and strongly nonlinear characteristics. Therefore, one of the basic ideas employed in designing the code was to provide the option of reducing the order of the systems examined by applying appropriate methodologies, when necessary. Apart from increasing the computational efficiency and speed, this reduction makes amenable the application of numerical techniques, which are efficient for low order systems.

In particular, δYNAMIS starts by reading all the necessary data in standard NASTRAN format from pre-processor ANSA, in order to set up the corresponding equations of motion. Then, a number of static and dynamic analyses can be performed. Currently, the code can be used to carry successfully out the following types of analysis:

- Static Analysis (for linear and nonlinear systems),
- Eigenvalue Analysis (real and complex natural frequencies and mode shapes),
- Substructuring (static condensation, component mode synthesis, multi-level dynamic substructuring),
- Linear Transient Response Analysis (direct integration and modal analysis),
- Linear Frequency Response Analysis (direct determination of frequency response functions and modal analysis),
- Nonlinear Transient Response Analysis (direct integration),
- Nonlinear Frequency Response Analysis (direct determination of periodic steady-state response diagrams and stability properties under periodic excitation) and
- Random Analyses.

Once the analysis chosen is completed, the results are directed to post-processor μETA. In fact, in order to achieve compatibility with most state of the art CAE post-processors, these results are written in NASTRAN (op2 or punch file) format.

The validity and effectiveness of δYNAMIS was first illustrated by passing standard tests on the finite element level. Moreover, direct comparison was performed (in terms of accuracy, memory required, data transferred and numerical speed) with MSC.Nastran for several quite complex structures. Besides the computational advantages, the code developed brings a lot of new features and improvements, which can be enhanced further by a proper coupling with the pre- and post-processor. For instance, the structure of the code facilitates greatly the application of parallel processing procedures. In addition, a distinct capability of the code is related to the special contact elements developed for both static and dynamic analyses. Another remarkable and unique strength is the ability to perform frequency response of periodically excited nonlinear dynamical systems in a direct way. Finally, δYNAMIS provides valuable tools towards employing a single original model for all the possible analyses (e.g., vibration, acoustics, fatigue) that need to be run for systems of any geometric complexity.

δYNAMIS: A NEW SOLVER FOR LINEAR AND NONLINEAR FINITE ELEMENT MODELS

Dr. Dimitrios Giagopoulos
Dynamical Systems Technology
Thessaloniki, Greece

Prof. Sotirios Natsiavas
Department of Mechanical Engineering
Aristotle University
Thessaloniki, Greece

OUTLINE OF PRESENTATION

- **Design Process**
- **Class of Mechanical Systems**
- **Solver Capabilities**
- **Code Verification** (Exact Solution, NAFEMS, NASTRAN)
- **Types of Analyses Performed**
- **Additional Features**
- **Synopsis-Extensions**

Design Process

Pre-Processor	Solver	Post-Processor
ANSA	δYNAMIS	μETA

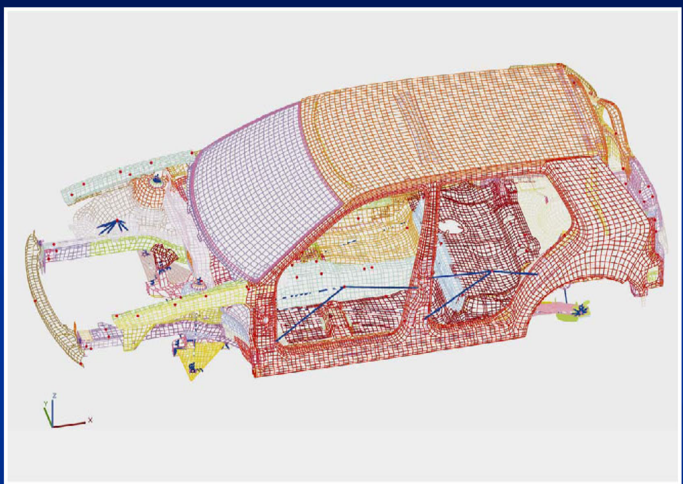
ANSA → (Nastran Input File) → δYNAMIS → (Nastran Output2 (op2) File) → μETA

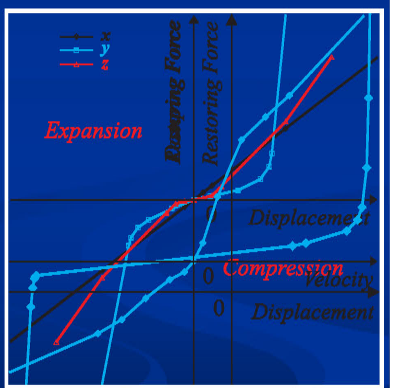
Applications:

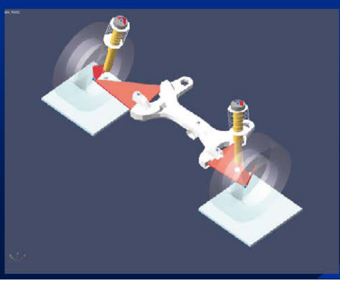
- ⇒ NVH
- ⇒ Biodynamics
- ⇒ Optimization
- ⇒ Identification and Control

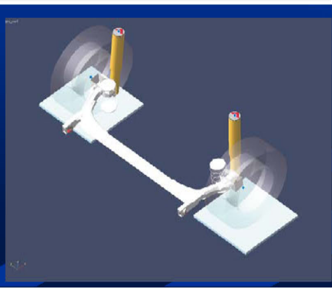
Class of Mechanical Systems

Finite Element - Car Model









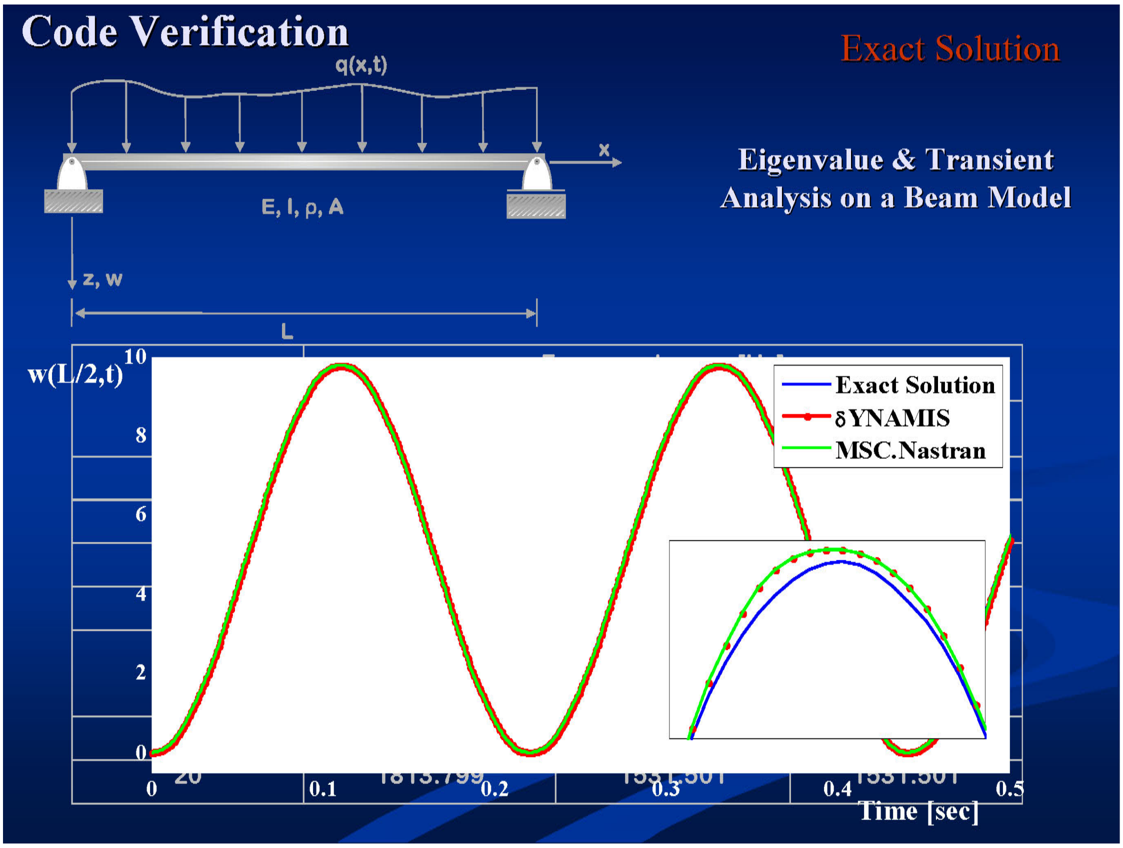
Solver Capabilities

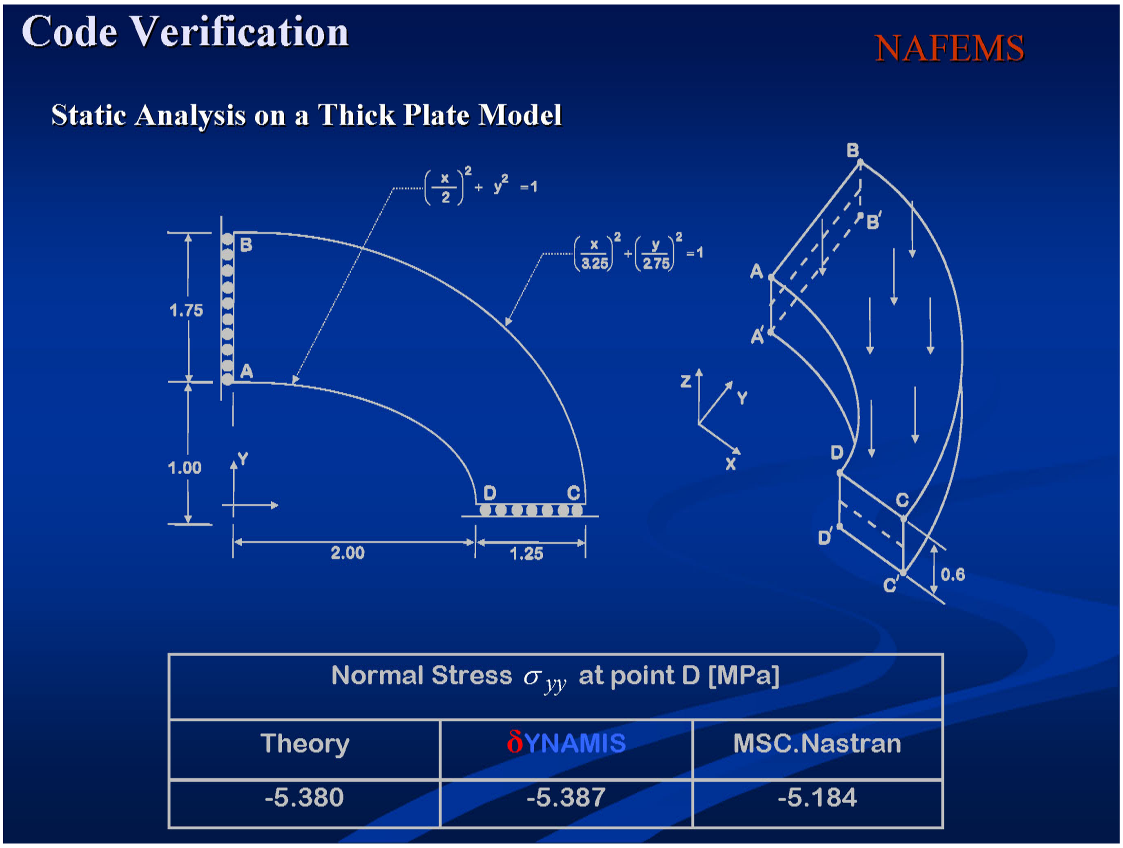
Types of Analyses Performed

- Static Equilibrium Analysis [Linear (101) / Nonlinear (106)]
- Eigenvalue Analysis [Real (103) / Complex (107)]
- Substructuring [Guyan, CMS, MLDS]
- Transient Response Analysis [Linear (109, 112) / Nonlinear (129)]
- Frequency Response Analysis [Linear (108, 111) / Nonlinear (128)]
- Random Analysis [Linear (108, 111)]

Solver Capabilities

- Element Types
  - Surface Elements : CQUAD4, CQUADR, CTRIA3, CTRIAR
  - Solid Elements : CHEXA, CPENTA, CTETRA
  - Other Elements : CBEAM, CBAR, CROD, CMASSi, CONMi, CELASi, CDAMPi, RBE2, RBE3, CBUSH1D, CBUSH, CWELD, CGAP
- Material Types
  - MAT1, MAT2, MAT8, MAT9, MATT1, MATT2, MATT8, MATT9
- Boundary Conditions Types
  - SPC, SPC1, SPCD, SPCADD, MPC, MPCADD
- Loads
  - FORCE, FORCEi, RFORCE, MOMENT, MOMENTi, SLOAD, PLOAD, PLOADi, GRAV, RLOADi, TLOADi, NONLINi, DAREA, DLOAD, DELAY, DPHASE, LSEQ
- Solution Control
  - EIGR, EIGRL, EIGC, FREQ, FREQi, TIC, TSTEP, TSTEPNL, RANDPS, RANDT1, RCROSS, NLPARM

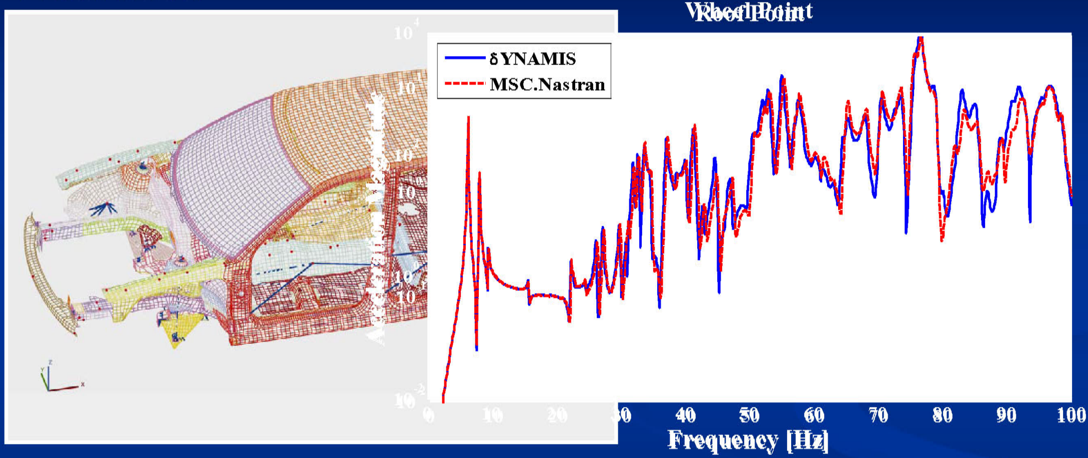

Code Verification
Exact Solution
Eigenvalue & Transient Analysis on a Beam Model
q(x,t)
x
E, I, ρ, A
z, w
L
w(L/2,t)
Exact Solution
δYNAMIS
MSC.Nastran
Time [sec]


Code Verification
NAFEMS
Static Analysis on a Thick Plate Model
1.75
1.00
2.00
1.25
0.6
Normal Stress σ_{yy} at point D [MPa]
Theory
δYNAMIS
MSC.Nastran
-5.380
-5.387
-5.184

Code Verification

MSC.NASTRAN

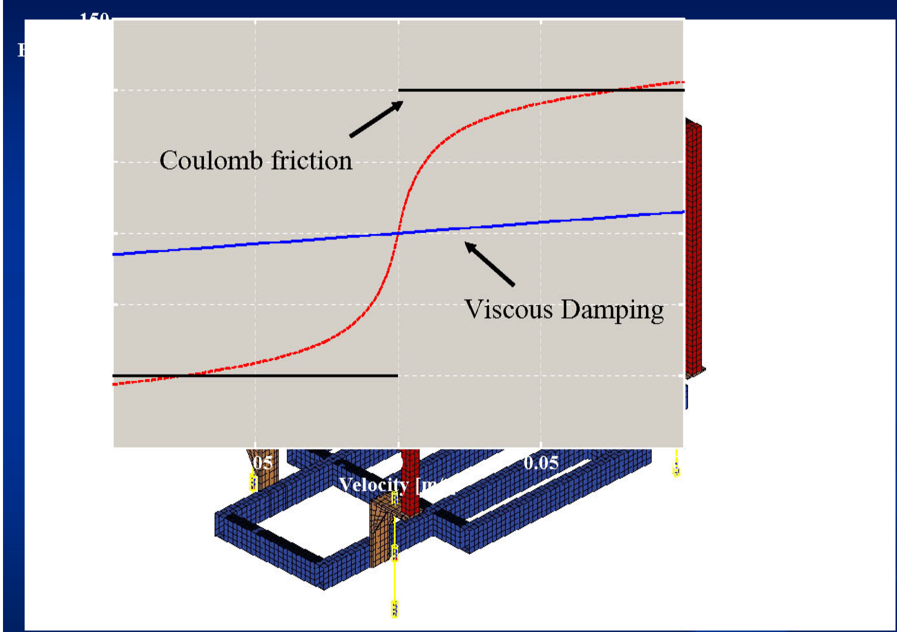
Vehicle Model

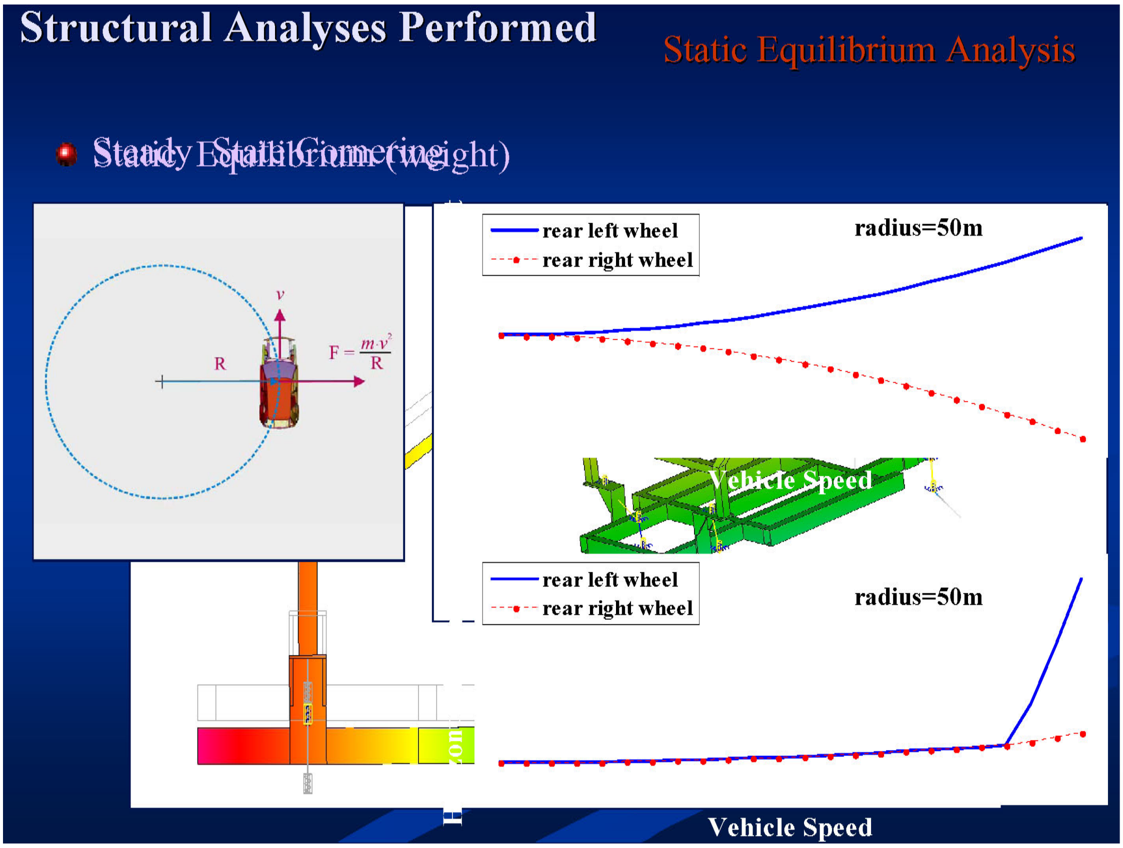


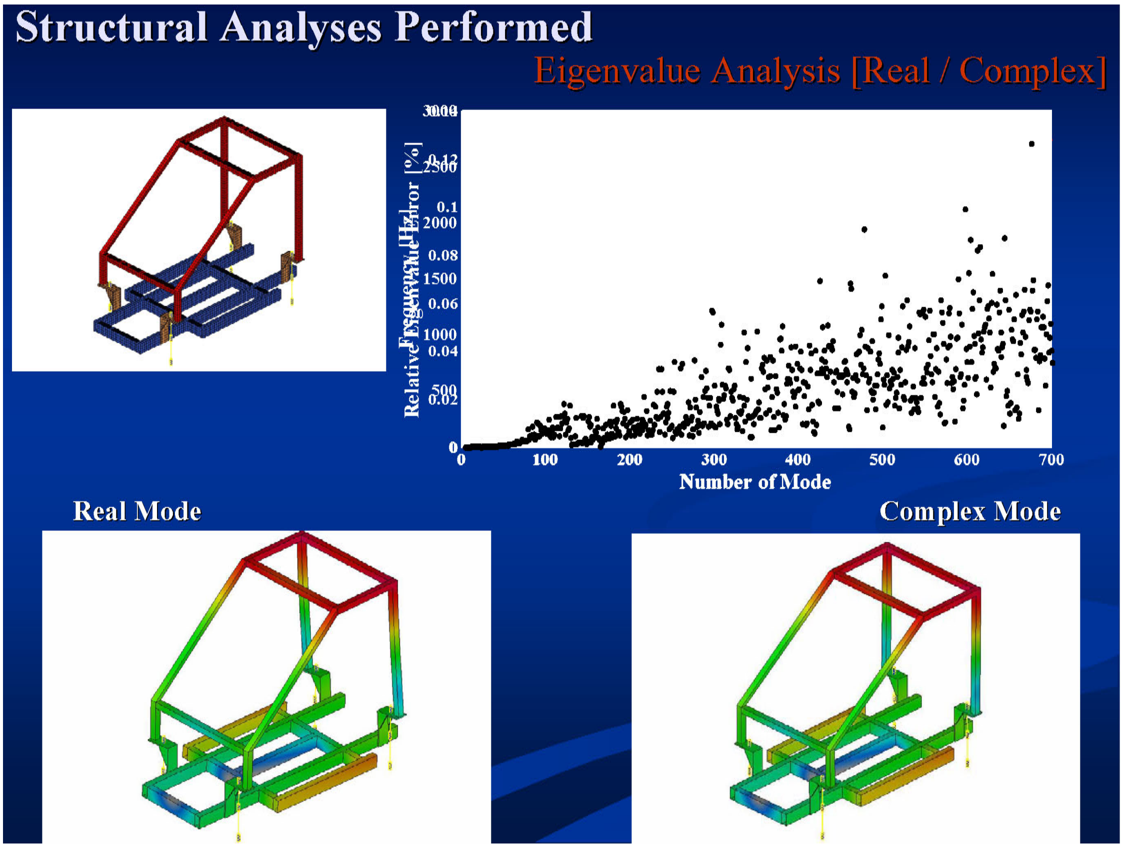
1,532,964 dof	[100Hz]	
	δYNAMIS	MSC. Nastran
Memory Required [MB]	661	632
CPU Time [HH:MM:SS]	00:37:31	00:56:13
Elapsed Time [HH:MM:SS]	00:56:13	01:21:36

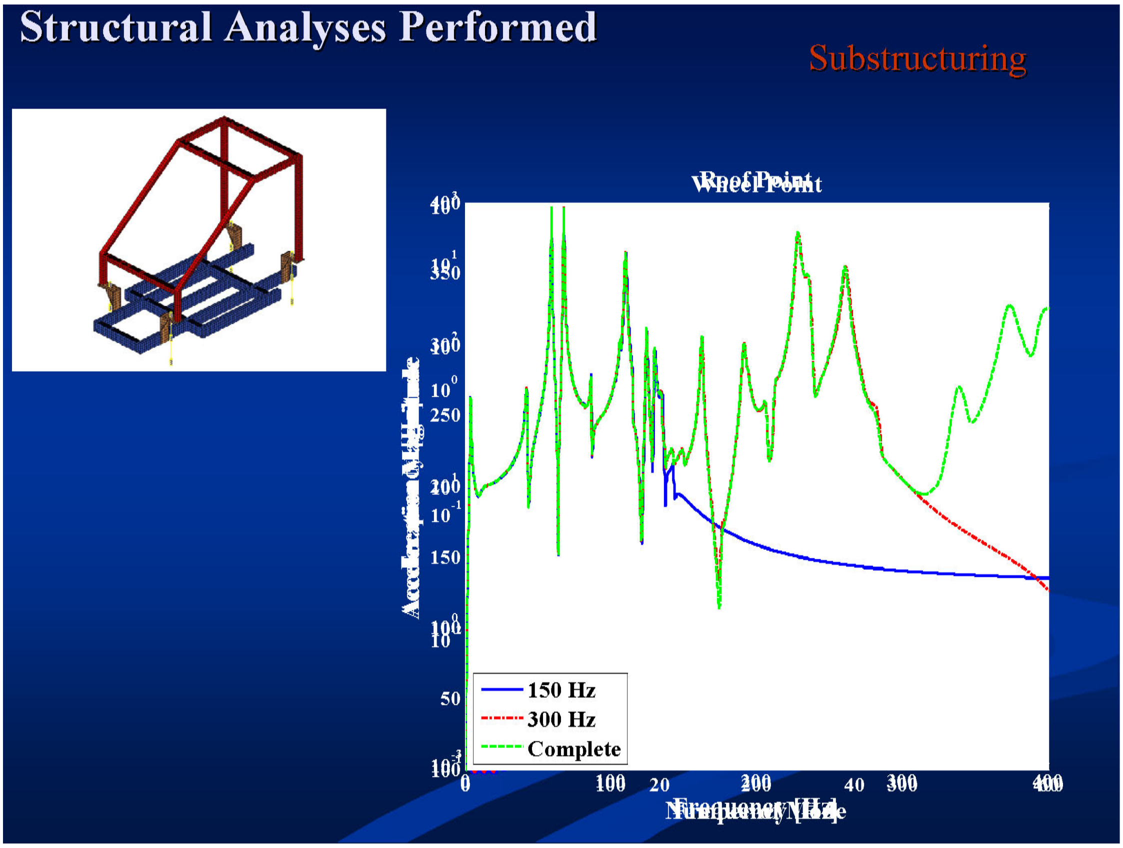
Structural Analyses Performed

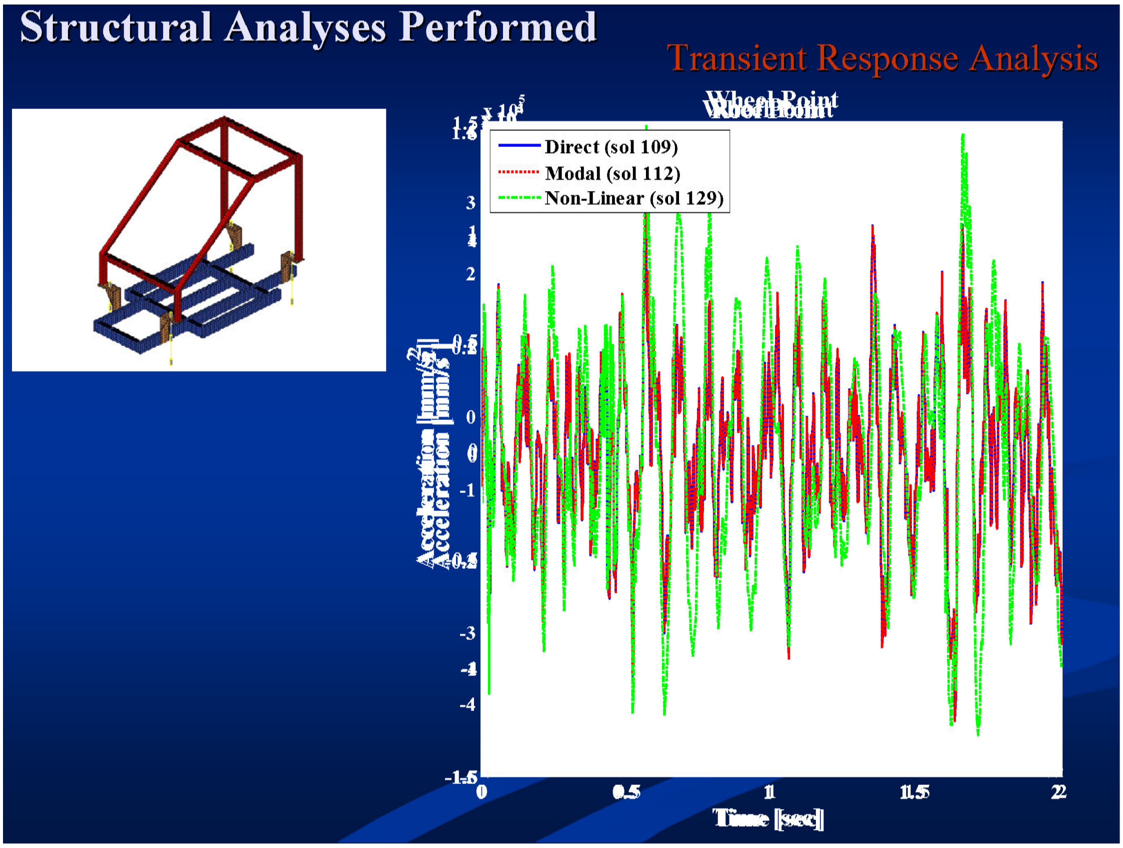
Mechanical Model

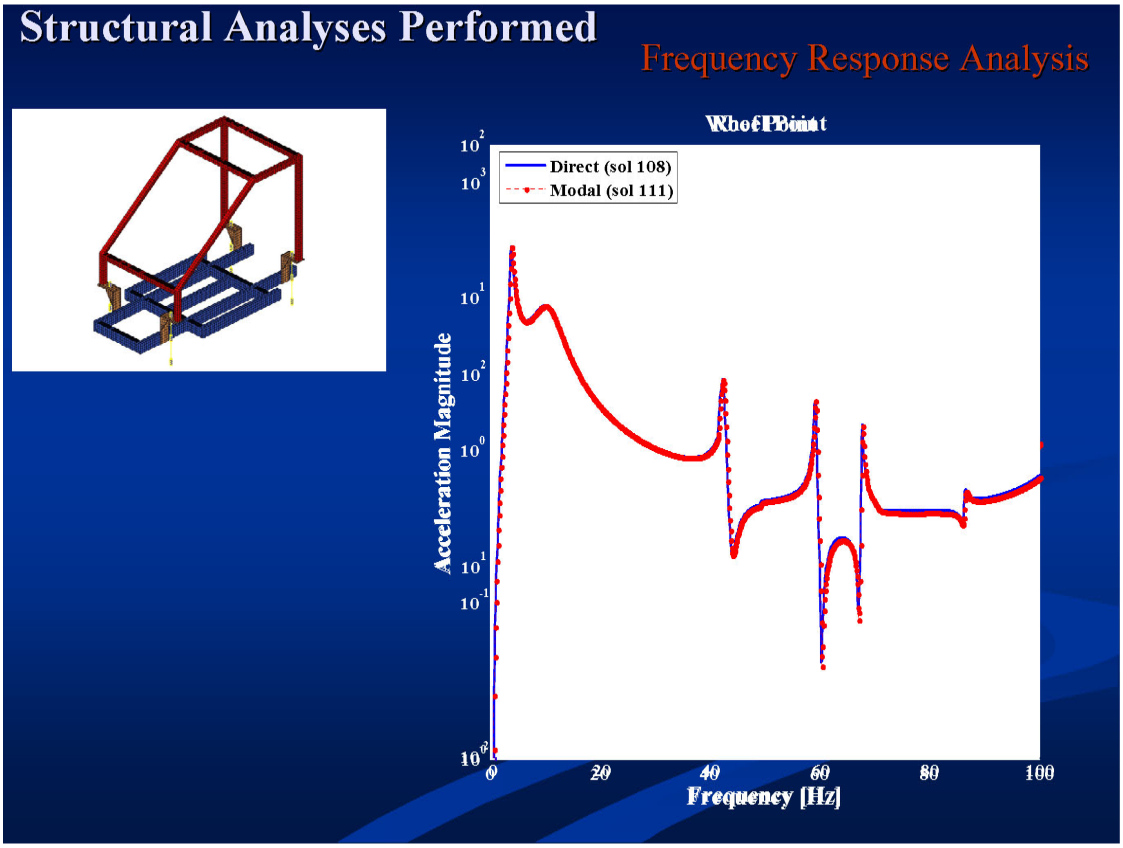


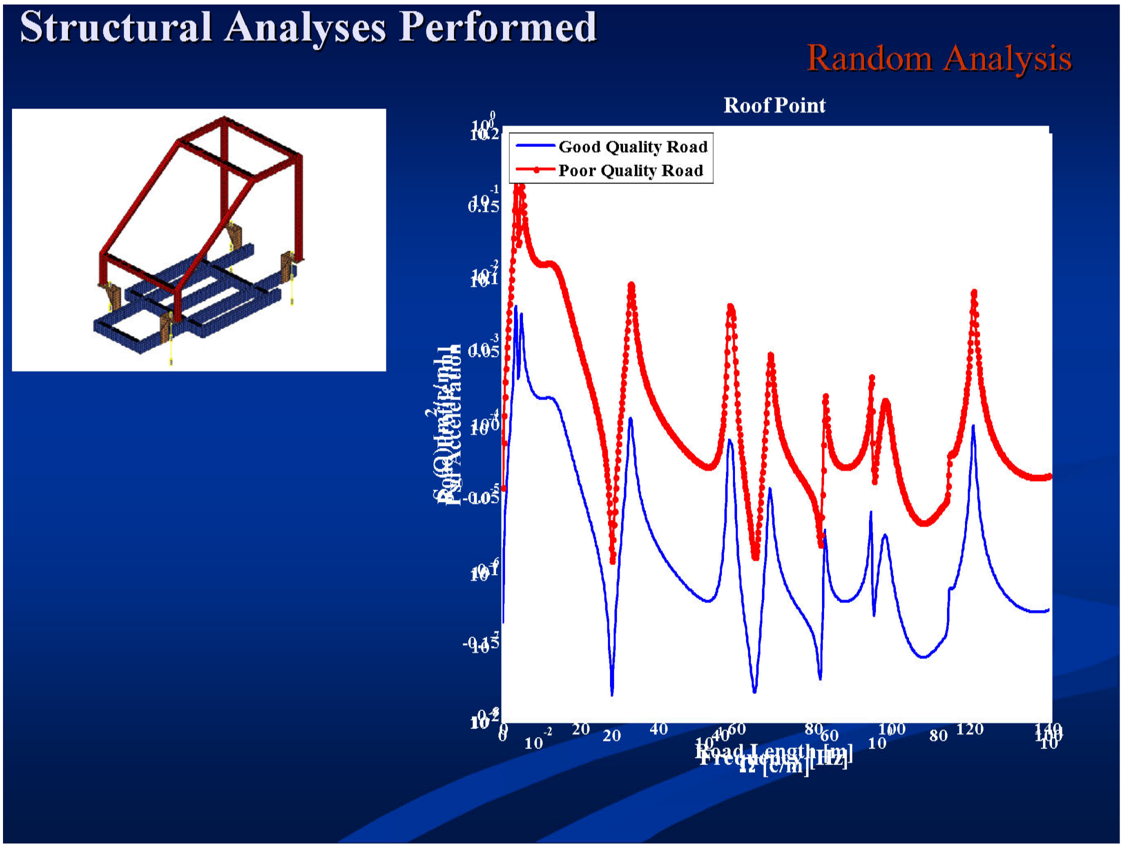

Structural Analyses Performed
Static Equilibrium Analysis
v
R
F = m·v²/R
rear left wheel
rear right wheel
radius=50m
Vehicle Speed
rear left wheel
rear right wheel
radius=50m
Vehicle Speed

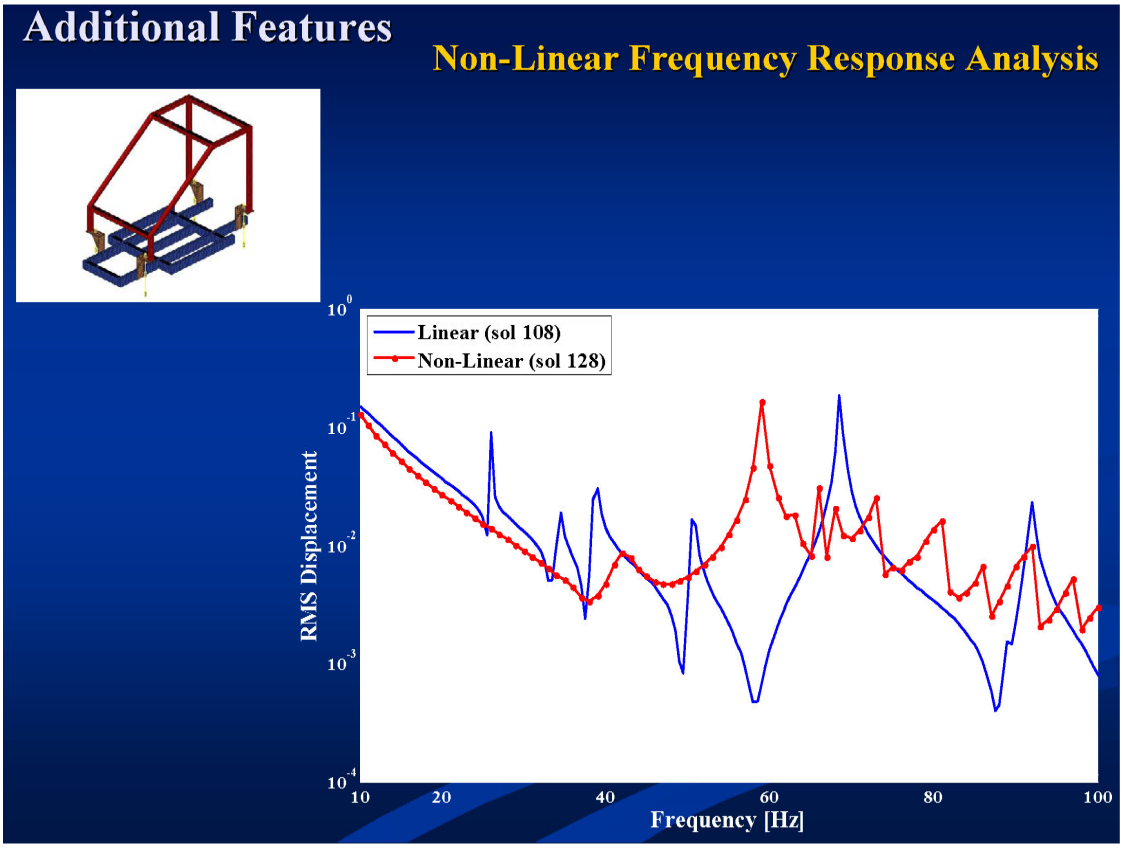

Structural Analyses Performed
Eigenvalue Analysis [Real / Complex]
0
100
200
300
400
500
600
700
Number of Mode
Real Mode
Complex Mode

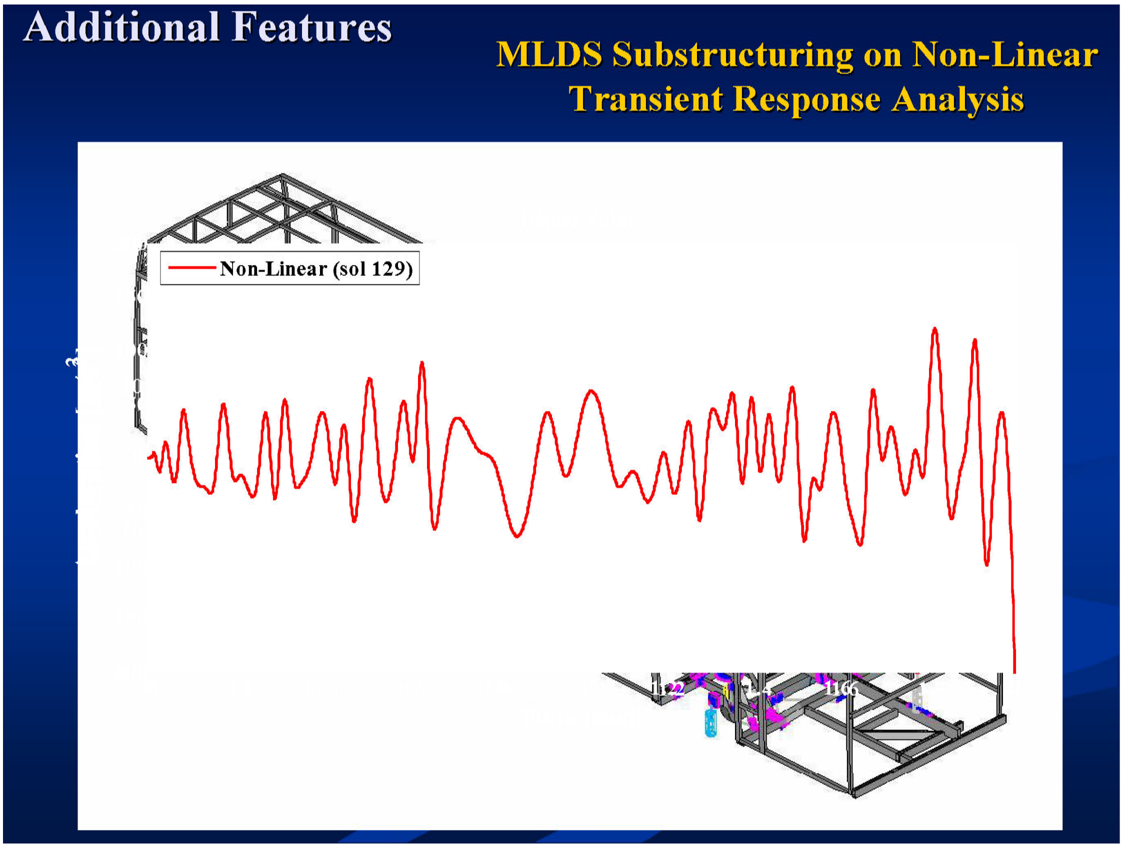

Structural Analyses Performed
Substructuring
150 Hz
300 Hz
Complete


Structural Analyses Performed
Transient Response Analysis
Direct (sol 109)
Modal (sol 112)
Non-Linear (sol 129)


Structural Analyses Performed
Frequency Response Analysis
Direct (sol 108)
Modal (sol 111)
Acceleration Magnitude
Frequency [Hz]
0
20
40
60
80
100

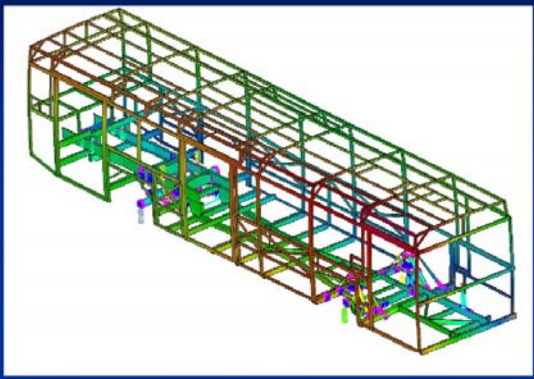

Structural Analyses Performed
Random Analysis
Roof Point
Good Quality Road
Poor Quality Road


Additional Features
Non-Linear Frequency Response Analysis
Linear (sol 108)
Non-Linear (sol 128)
RMS Displacement
10^0
10^-1
10^-2
10^-3
10^-4
10
20
40
60
80
100
Frequency [Hz]


Additional Features
MLDS Substructuring on Non-Linear Transient Response Analysis
Non-Linear (sol 129)

Additional Features

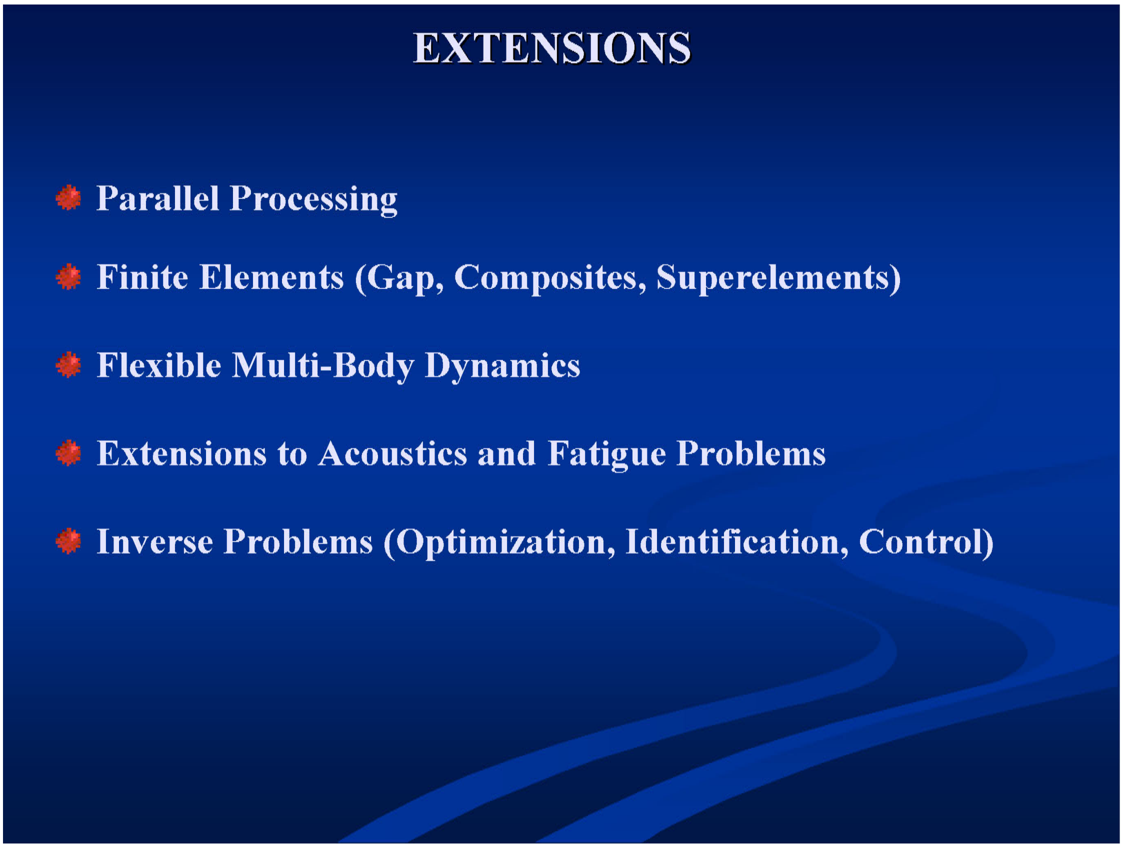
MLDS Restart Combination



δYNAMIS	Elapsed Time [HH:MM:SS]	Time Save (%)
Analysis 1	00:37:06	
Analysis 2	00:45:58	0.0
Analysis 2 Restarted from Analysis 1	00:18:06	60.0

SYNOPSIS

- **Usability:** Full compatibility with Nastran data input files. Output in standard Output2 (op2) files
- **Completeness:** Full suite of analysis capabilities. Complete library of geometric cards, elements, constrains, loads, properties and materials for all kinds of structural analysis
- **Accuracy:** Extremely accurate line, shell and solid finite elements
- **Performance:** Highly sophisticated solvers exploit the very unique characteristics of the equations delivering solution in a fraction of time
- **Innovation:** New Non-Linear steady state analysis takes the classical linear frequency response into the non-linear domain, including continuation methods and stability analysis


EXTENSIONS
Parallel Processing
Finite Elements (Gap, Composites, Superelements)
Flexible Multi-Body Dynamics
Extensions to Acoustics and Fatigue Problems
Inverse Problems (Optimization, Identification, Control)


δYNAMIS