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

Dimitrios Giagopoulos, Sotirios Natsiavas
1DYnamical Systems TechnologY, Thessaloniki, Greece
2Department 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 - DYYNAMIS (DYNAmics of MechanIcal Systems) 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, DYYNAMIS 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 DYYNAMIS 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, DYYNAMIS 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.
DYNA\textit{MIS}: 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

ANSYS  →  DYNAVIS  →  μETA

Nastran Input File  Nastran Output (op2) File

Applications:

⇒ NVH
⇒ Biodynamics
⇒ Optimization
⇒ Identification and Control

Class of Mechanical Systems  Finite Element - Car Model

Finite Element Analysis

Diagram showing displacement and force graphs.
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
2nd ANSA & μETA International Congress
June 14-15, 2007 Olympic Convention Center, Porto Carras Grand Resort Hotel, Halkidiki Greece

Code Verification

Exact Solution

Eigenvalue & Transient Analysis on a Beam Model

Code Verification

NAFEMS

Static Analysis on a Thick Plate Model

<table>
<thead>
<tr>
<th>Normal Stress $\sigma_{yy}$ at point D [MPa]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
</tr>
<tr>
<td>-5.380</td>
</tr>
</tbody>
</table>
Code Verification

Vehicle Model

MSC.NASTRAN

<table>
<thead>
<tr>
<th></th>
<th>[100Hz]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1,532,964 dof</td>
<td></td>
</tr>
<tr>
<td>Memory Required [MB]</td>
<td>661</td>
</tr>
<tr>
<td>CPU Time [HH:MM:SS]</td>
<td>00:37:31</td>
</tr>
<tr>
<td>Elapsed Time [HH:MM:SS]</td>
<td>00:56:13</td>
</tr>
</tbody>
</table>

Structural Analyses Performed

Mechanical Model

Coulomb friction

Viscous Damping
Structural Analyses Performed

Static Equilibrium Analysis

Vehicle Speed

radius=50m

Structural Analyses Performed

Eigenvalue Analysis [Real / Complex]

Real Mode

Complex Mode
Structural Analyses Performed

Substructuring

![Graph of substructuring](image)

Structural Analyses Performed

Transient Response Analysis

![Graph of transient response analysis](image)
Structural Analyses Performed

Frequency Response Analysis

![Graph showing frequency response analysis with two lines representing Direct and Modal solutions.](image)

Random Analysis

![Graph showing random analysis with two lines representing Good Quality Road and Poor Quality Road.](image)
Additional Features

Non-Linear Frequency Response Analysis

![Graph showing Non-Linear Frequency Response Analysis]

- Linear (sol 108)
- Non-Linear (sol 128)

Additional Features

MLDS Substructuring on Non-Linear Transient Response Analysis

![Graph showing MLDS Substructuring]

- Non-Linear (sol 129)
Additional Features

MLDS Restart Combination

<table>
<thead>
<tr>
<th>δYNAMIS</th>
<th>Elapsed Time [HH:MM:SS]</th>
<th>Time Save (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Analysis 1</td>
<td>00:37:06</td>
<td></td>
</tr>
<tr>
<td>Analysis 2</td>
<td>00:45:58</td>
<td>0.0</td>
</tr>
<tr>
<td>Analysis 2</td>
<td>00:18:06</td>
<td>60.0</td>
</tr>
</tbody>
</table>

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)