ADAMS

Principles and Examples of
Stress Recovery

D. Minen

F. Ambrogi
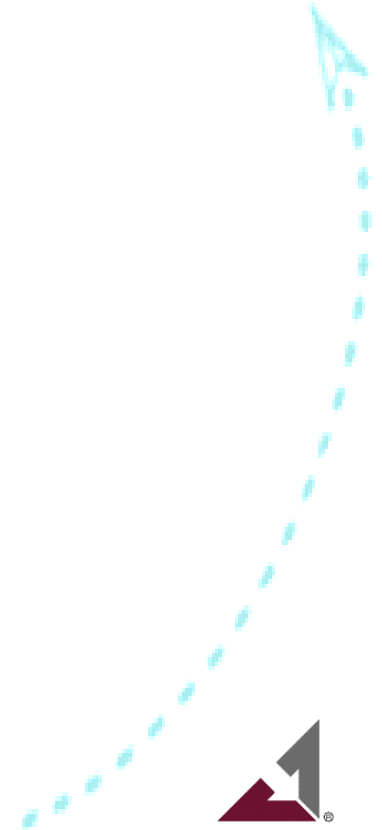
Mechanical Dynamics

May-2000

Mechanical Dynamics

AGENDA

- *Motivations*

- *Modal Stress Recovery Principles*

- *Applications at MDI*
  - *ADAMS to Nastran*
  - *ADAMS to ANSYS*

- *Application Examples*

- *Conclusions*

Mechanical Dynamics

Motivations

Why combined MB-FE Stress Recovery?

- *Stress computation is fundamental for mechanical components design*

- *MB & FE together are <u>the</u> winning solution because dynamic effects are correctly taken into account*

- *Stress(and/or Strain) computation is the base for fatigue analysis*

- *Virtual system prototyping is a superior technique for approaching durability*

Load Export and Flex Components

■ **Why bother with Load Export on Flex Bodies?**

  ◆ <u>Task:</u> Compute Strains on Flexible Components

  ◆ Nodal Deformations for FBs are available in form of Modal Expandable Displacements

  - Need something that computes local Strains from Nodal Deformations

  - => Finite Element Equation Formulation

  - Material Model(s)

  - => Stress Components at Nodes

  ◆ Why to export joint REACTIONS and FORCES from ADAMS and compute Deformations again?

  ◆ Why not to take advantage of the already computed Dynamics?

  ◆ Why to run a bunch of FE Static Load Cases with enormous waste of time?

  ◆ Forces are never accurate in ADAMS as displacements are

Mechanical Dynamics

Modal Stress Recovery

- Accurate Dynamic Loading and Boundary Conditions automatically computed on the embedded Flex Component

- Immediate and very accurate Dynamic Stress distribution computation compared to a full DOFs FE solution

- No need to export Loads and/or Nodal Deformation

Mechanical Dynamics

ADAMS

Principles of Modal Stress Recovery - 1

Assuming that the reduction of the full set of Mode Shapes of the Flexible Body to a sub-set is correct and comprehensive of all the required effects, the Stress Distribution related to the body deformation can be calculated from:

Stress Mode Shapes Method

$$\{\sigma\} = [\Phi_\sigma] \cdot \{p\}$$

$$[\phi_\sigma] = [\{\phi_\sigma\}_1, \ldots, \{\phi_\sigma\}_{P+S}]$$

ortho-normalized **Modal Stress Matrix**

Deformation Mode Shapes Method

$$\{u\} = [\Phi_u] \cdot \{p\}$$

$$[\phi_u] = [\{\phi_u\}_1, \ldots, \{\phi_u\}_{P+S}]$$

ortho-normalized **Modal Deformation Matrix**

$\{\phi_\sigma\}_{1 \ldots P+S}$ (P=number of Normal Constrained Modes S = number of Static Correction Modes)

$\{\phi_u\}_{1 \ldots P+S}$ (P=number of Normal Constrained Modes S = number of Static Correction Modes)

$\{p\}$ Vector of Modal Coordinates

$\{\sigma\}$ Stress Component Matrix

$\{u\}$ Physical Displacement Matrix – Input for a imposed displacement FE analysis

Mechanical Dynamics

Principles of Modal Stress Recovery - 2

- **Stress Computation from ADAMS results and FB Modes**

  - ◆ from Modal Precomputed Stress Vectors

    - • Directly combined with Modal Coords to get Physical Stress Components

  - ◆ from Modal Displacement Vectors

    - • Directly combined with Modal Coords to get Physical Displ Components to FE

  - ◆ from Modal Coords only

    - • Exporting them to FE

Mechanical Dynamics

Principles of Modal Stress Recovery - 3

- **Modal Stress Vectors**
  - Storable into the .mnf file or elsewhere
  - Require a-priori knowledge of the mesh characteristics
  - Enable Stress computation in ADAMS directly
  - Could expand the size of the .mnf a lot
    - which stress component to store?
    - each node has its associated top, middle and bottom shell surface

- **Modal Displacement Vectors**
  - Stored in the .mnf file (part of the FB definition)
  - Allow any type of Stress calculation w.o. knowing the mesh characteristics
  - Require to be expanded to Physical Displ (in ADAMS) to generate full Nodal Displacement in FE

- **Modal Coordinates**
  - Stored in the .res, $q_i(t)$
  - Ideally, could be directly brought to FE and used to expand the Modal Stress Vectors or the Modal Displacement Vectors to Physical Stresses

VONMISES
Requests von Mises stress.
MAXS or SHEAR
Requests maximum shear for shell elements and octahedral stress for solid elements.
CENTER
Requests CQUAD4 element stresses at the center only.
CUBIC
Requests CQUAD4 element stresses at the center and grid points using strain gage approach with cubic bending correction.
SGAGE
Requests CQUAD4 element stresses at center and grid points using strain gage approach.
CORNER or BILIN
Requests CQUAD4 element stresses at center and grid points using bilinear extrapolation.

ADAMS

Accuracy using Modal Stress Recovery
## *Example on a simple filleted bar*

- **Filleted Bar Model**
    - ◆ ANSYS Model vs. ADAMS Flexible Body
    - ◆ 7194 Shell (shell43) Elements
    - ◆ 6 Static Correction Modes for the cantilever point
    - ◆ 20 Normal Constraint Modes in addition
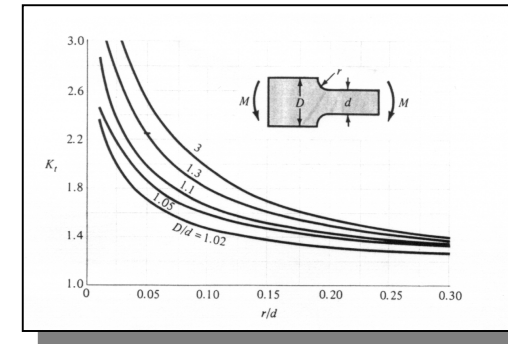
Material	Young Modulus (N/m²)	D (m)	d (m)	r (m)	s (m)	M (N m)
Steel	2.1E11	5.0E-2	3.85E-2	2E-3	1.0E-2	1

Stress (N/m²)	$\sigma_{von\ mises}$	K_t
ANSYS Node 408	8.9E+05	2.1922
ANSYS Node 4040	4.0E+05	
ADAMS Node 408	8.7E+05	2.1640
ADAMS Node 4040	4.0E+05	

Mechanical Dynamics
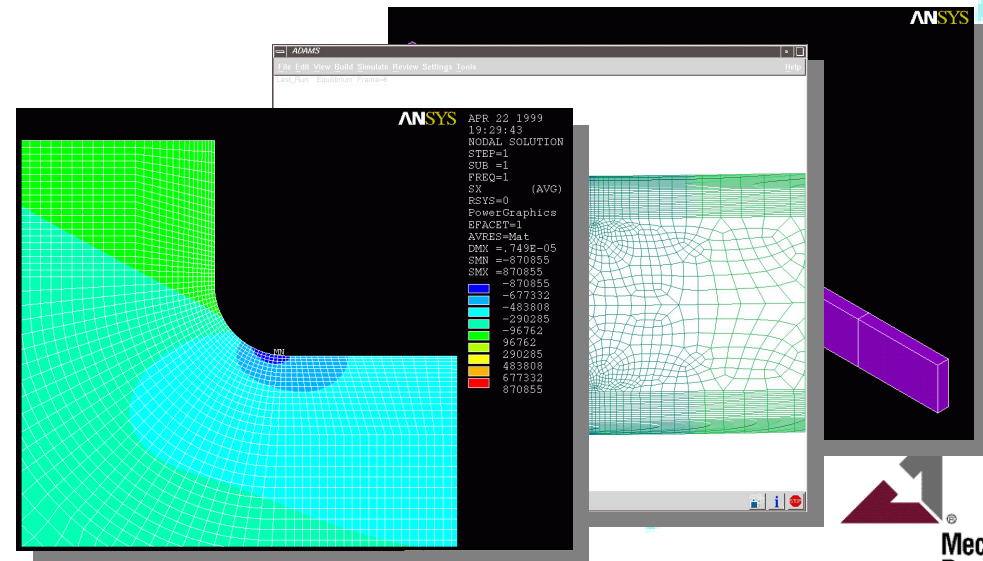
Modal Stress Recovery
Currently available technology at MDI

- MSC/Nastran

- ANSYS

Mechanical Dynamics
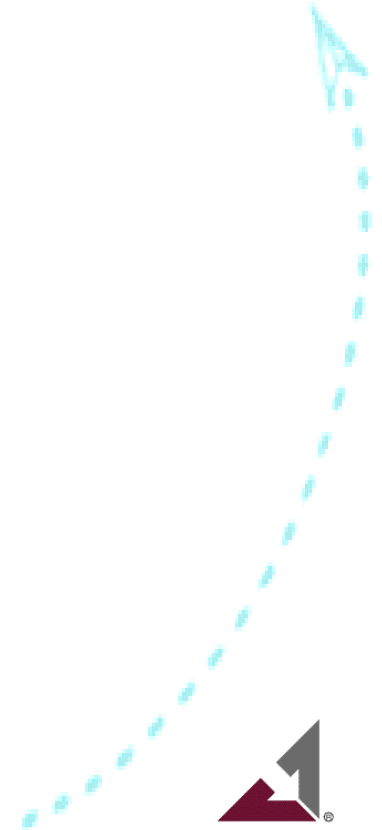
ADAMS-ANSYS Stress Recovery - 1

- **Modal Stress Vectors**

  - ◆ Stored in the .mnf for homogeneous type of FE

  - ◆ Expandable and plottable in A/PPT

- **Modal Displacement Vectors**

  - ◆ Eigenvectors in the .mnf file are read in and expanded to Physical Coordinates

  - ◆ A file ("D") containing all the Physical Nodal Displacement in ANSYS format is created via a special function in A/View

  - ◆ The "D" file is automatically read into ANSYS and a specialized macro controls multiple runs

Mechanical Dynamics

ADAMS-ANSYS Stress Recovery - 2

■ **Modal Superposition**

◆ A file containing the **Orthonormalized Eigenvectors** and the **Modal Coordinates** from ADAMS simulation is created via a special function in A/View

◆ The file is automatically read into ANSYS via a macro

- A set of static analyses is run, imposing to nodes the nodal displacements from orthonormalized mode shapes, to get all modal stress components

- A load step number is assigned to each static analysis

- The results from the previous loading steps are combined, using ADAMS modal coordinates as "weigthing factors"

ADAMS-ANSYS Stress Recovery - 3
Modal Dispalcements Vectors vs. Modal Superposition

■ Modal Displacements Vectors

◆ Time Domain Stress Recovery is advantageous when the number of time steps is small (less then the number of Flexible Body active modes)

■ Modal Superposition

◆ It is currently limited to max 99 Flexible Body active modes, due to ANSYS limitation. It should be very easy to expand it to a much higher number

Mechanical Dynamics

Stresses from Modal Stress Vectors

.mnf File

Topology

Mode Shapes

Gen.Stiffness

Gen.Mass

Stress Vectors

...

A/View

Graphical Representation

Connectability

Mode Selection

A/PPT

Plot nodal stress components

Modal Expansion

FE
Run CM

Compute Nodal Stress Components

Flexible body

Analysis

Node to add to select list

Selected Nodes list

☑ sigma Von Mises

☐ sigma x   ☐ sigma y   ☐ sigma z

☐ tau xy   ☐ tau yz   ☐ tau zx

OK   Apply   Cancel

Options

Inertia Invariants Pre-computation Flag

Inertia Invariants

Gen.Stiffness

Mode Shapes

Mechanical Dynamics

ADAMS

ANSYS®

Stresses from Modal Displacement Vectors

.mnf File

Topology

Mode Shapes

Gen.Stiffness

Gen.Mass

...

ANSYS

Run CMS

A/View

Export nodal results for ANSYS Analysis   ✕

Flexible body   [   ]

Analysis   [   ]

Ansys input file   [   ]

MNF to ANSYS scale option   [none   ▾]

☐ Binary format   ☐ Partial Recovery

[OK]   [Apply]   [Cancel]

...

ANSYS®

POST1 or POST26

countour plots,

animations, …

Accessories

Units

Portability

Simplified Graphics Options

Inertia Invariants Pre-computation Flag

A/Solver

Retained Modes

Connectivity

Inertia Invariants

Gen.Stiffness

Mode Shapes

Mechanical Dynamics®

ADAMS-ANSYS Stress Recovery

Proof of concept

- Flexible body:
  - 522 nodes, 450 shell43 elements
  - 15 NM, 12 SCM

- 50 Hz Harmonic force input

- Bending mode at 66 Hz

- Partial elem plotting could be applied

ADAMS-ANSYS Stress Recovery

Proof of concept

- Resonant frequency corresponding to plate bending mode
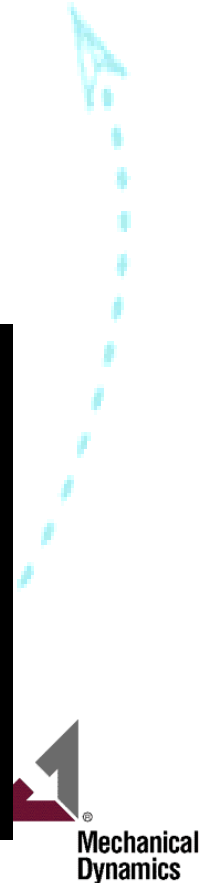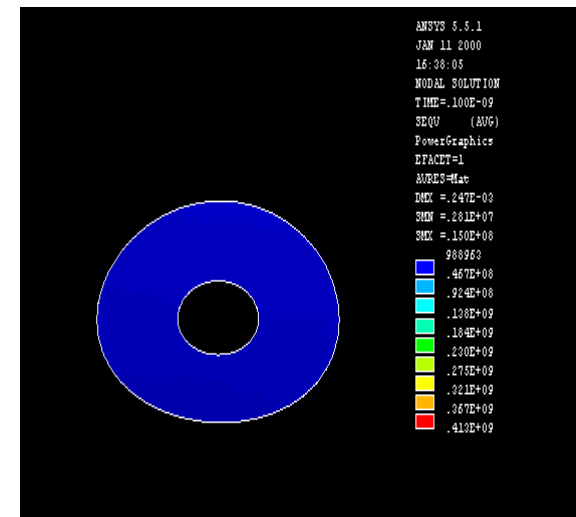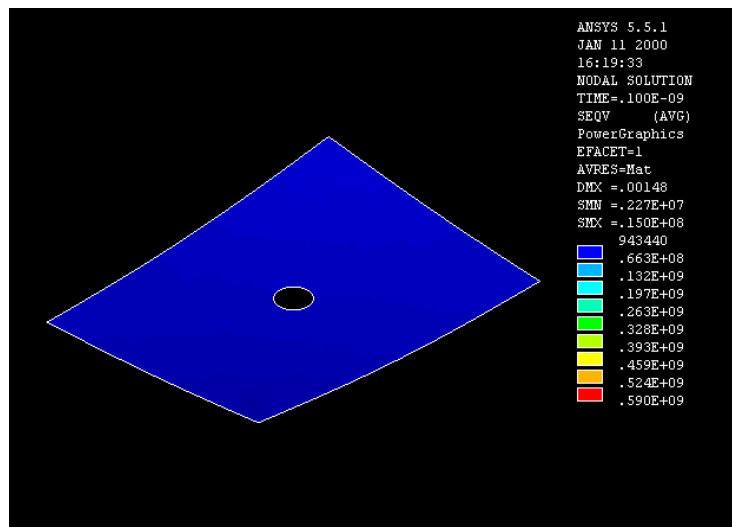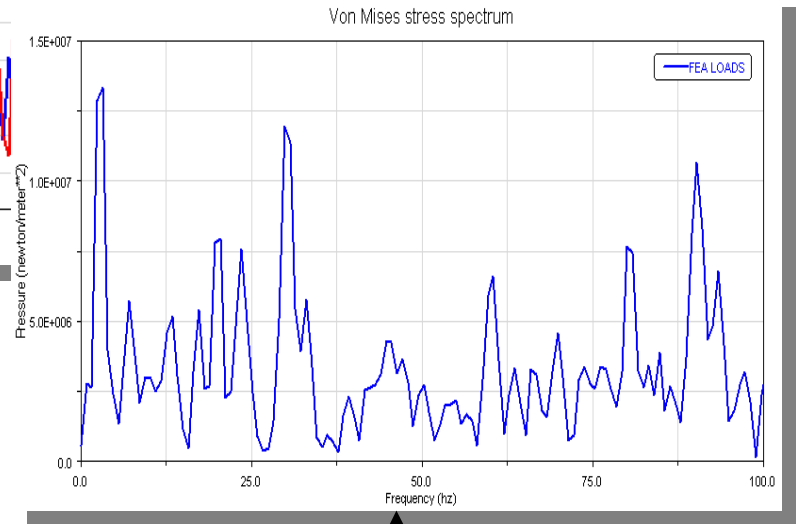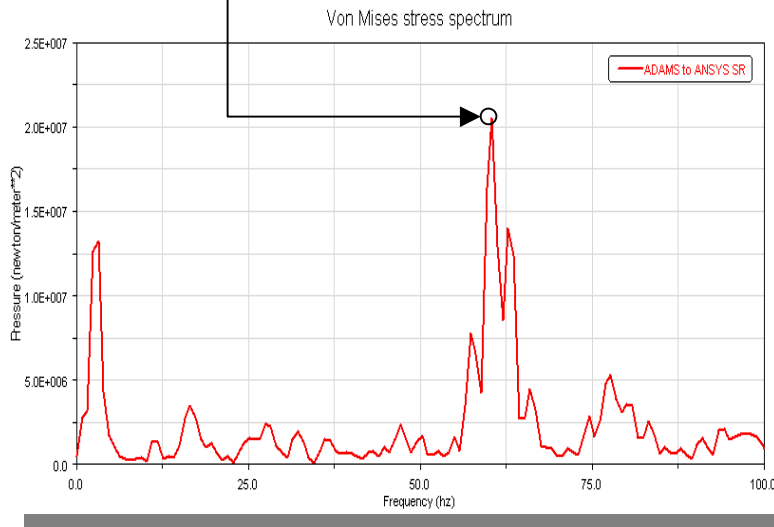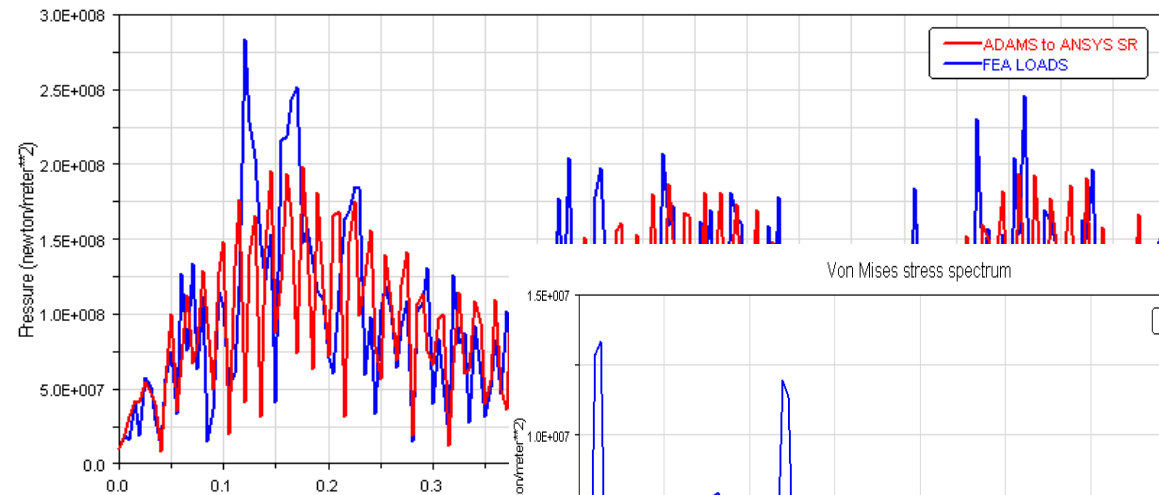
- Totally unrelated frequency content

Comparison between ADAMS to ANSYS SR and FEA LOADS application

Von Mises Stress

Von Mises stress spectrum

Von Mises stress spectrum

ADAMS-ANSYS Stress Recovery
Examples

- [Flexible rail axles on flexible track](#)

- [Flexible bus frame and chassis](#)

- [Flexible crankshaft](#)

Mechanical Dynamics

ADAMS-ANSYS Stress Recovery

Stress recovery analysis on a bus flexible frame
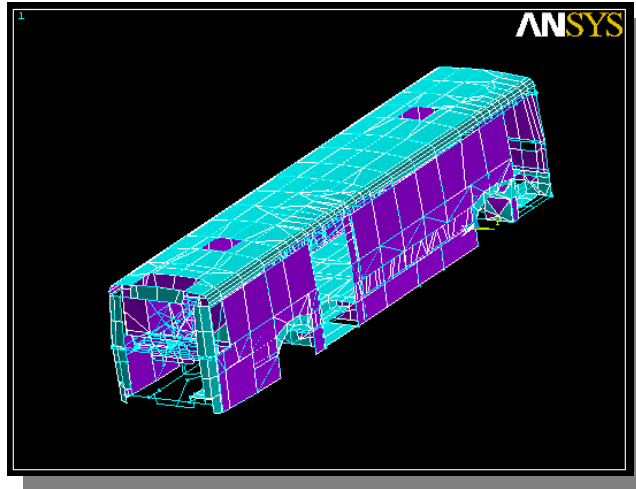
- ■ ANSYS 5.5 model:
    - ◆ 3980 elements:
        - • beam44, shell63,mass21,combin14, link8
    - ◆ 4474 nodes
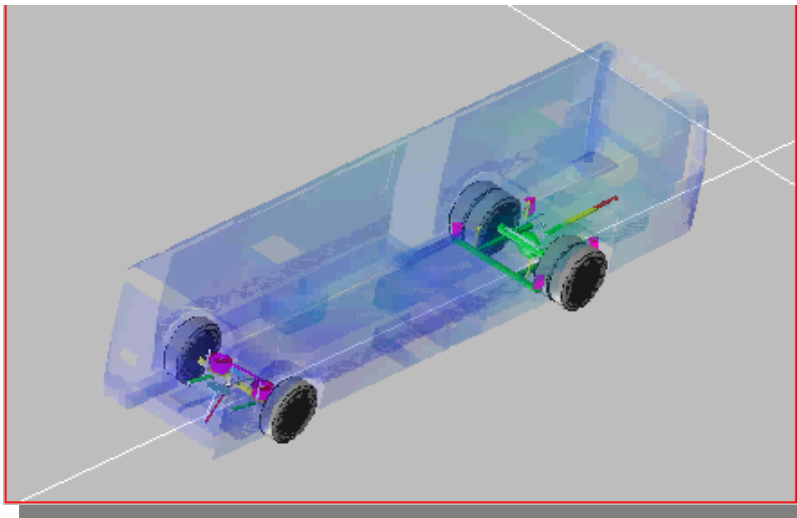
- ■ ADAMS/Pre model:
    - ◆ 241 dofs
    - ◆ Flexible frame:
        - • 25 IFC points
        - • 150 SC modes
        - • 16 further N modes
    - ◆ Fiala tire model
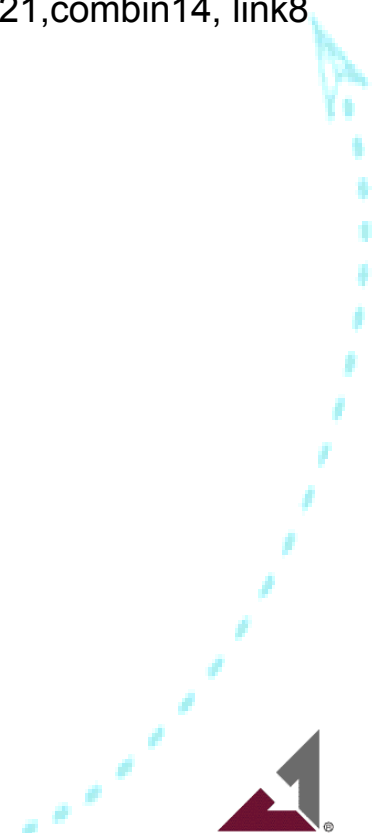
Mechanical Dynamics

ADAMS-ANSYS Stress Recovery
Stress recovery analysis on a bus flexible frame
ADAMS simulations
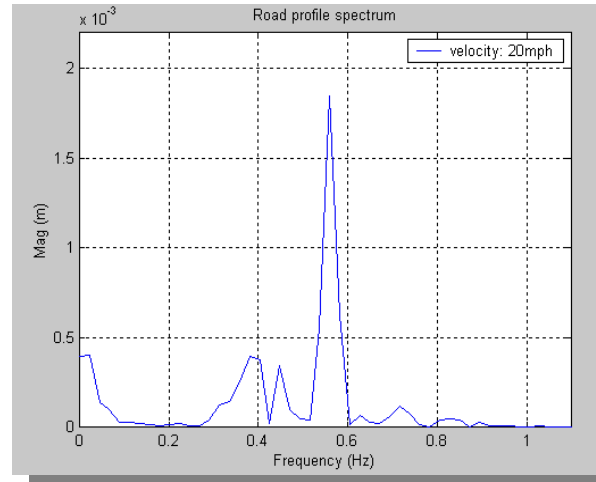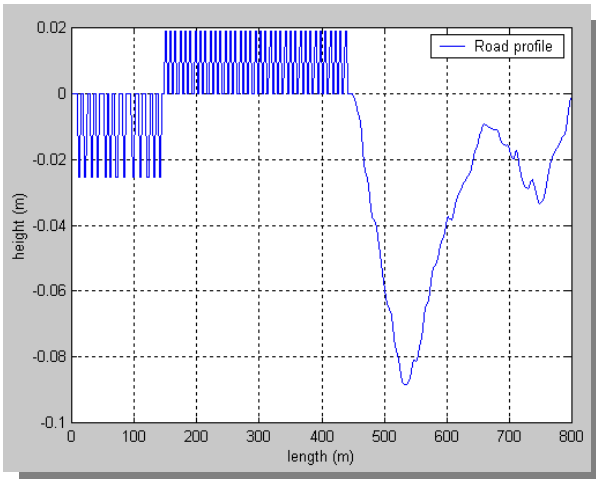
- Road profile

- Dynamic simulation
  - ◆ 40 secs 400 steps
  - ◆ CPU time 7540 secs on Pentium 300
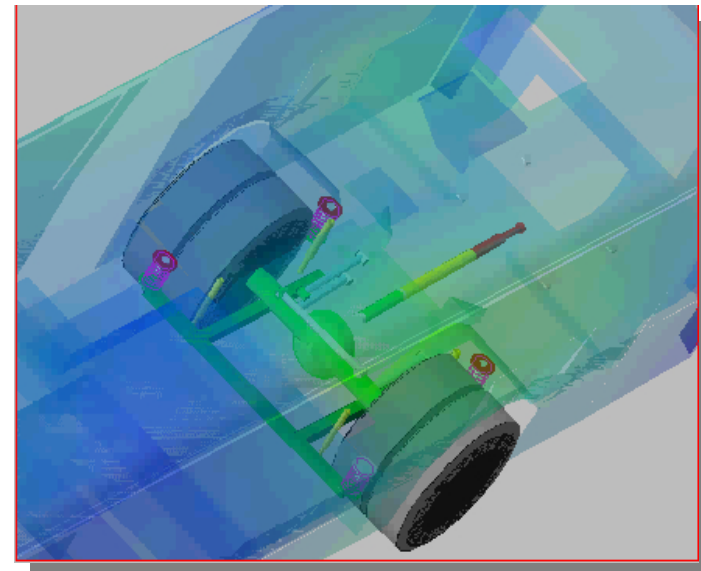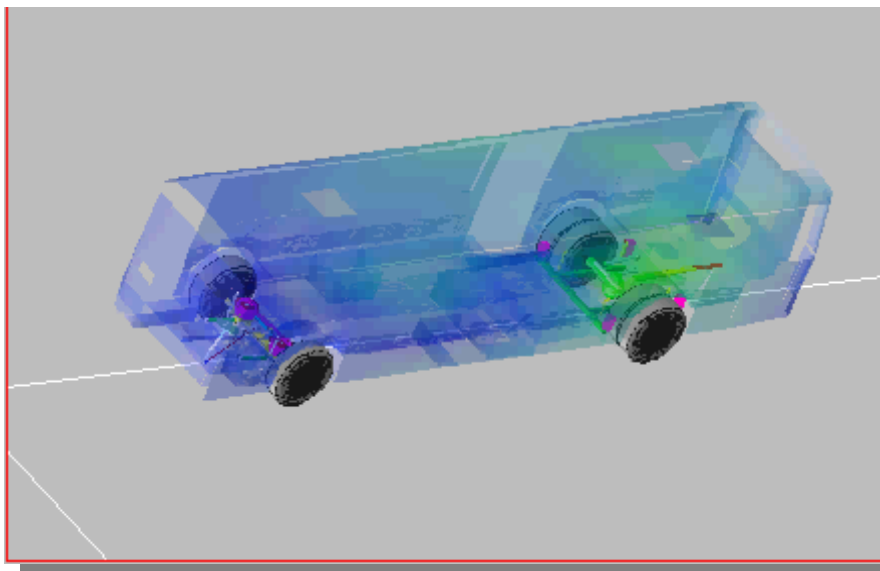
ADAMS-ANSYS Stress Recovery
Stress recovery analysis on a bus flexible frame
ANSYS simulations
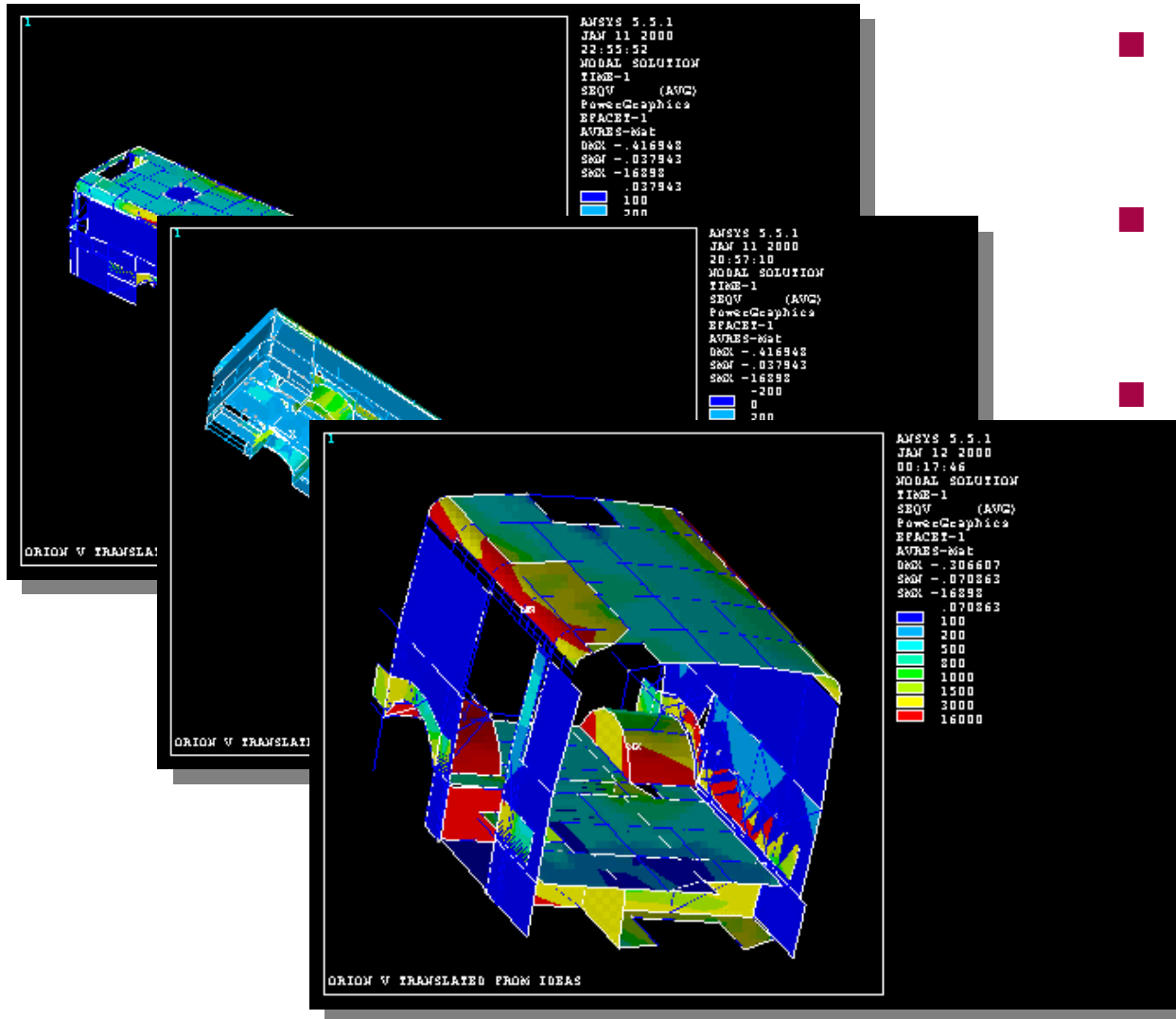
- **Input file**
  - ◆ 80Mb in bynary format,230 ascii

- **CPU time**
  - ◆ 8000 secs on SGI Octane (256 Mb RAM Irix 6.4)

- **Output file**
  - ◆ 1Gb .rst file (using a reduced set of resuts)
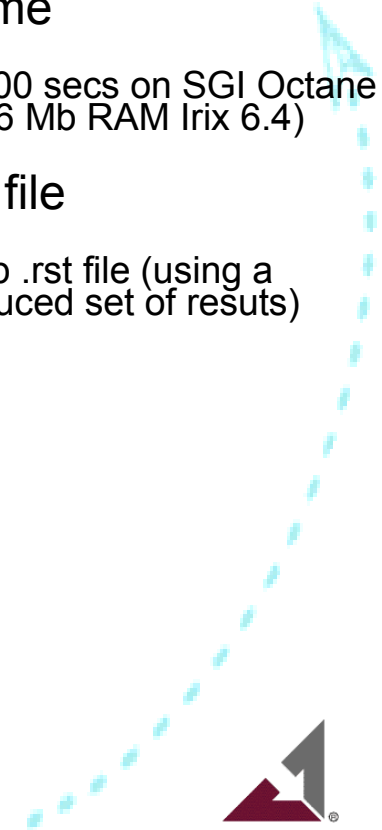
ADAMS-ANSYS Stress Recovery

Stress recovery analysis on a bus flexible frame
some RESULTS
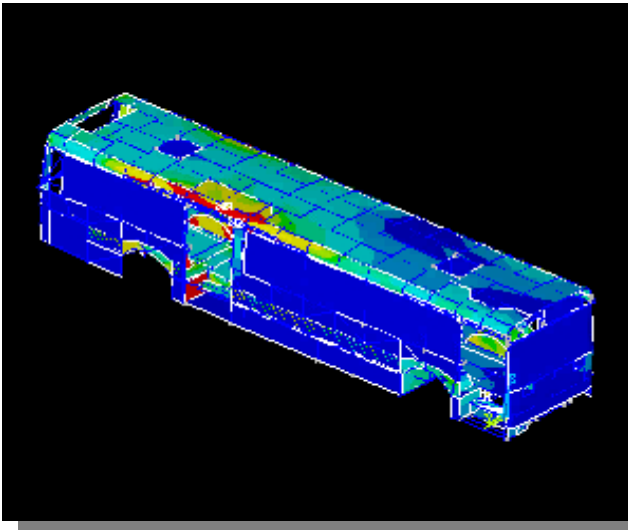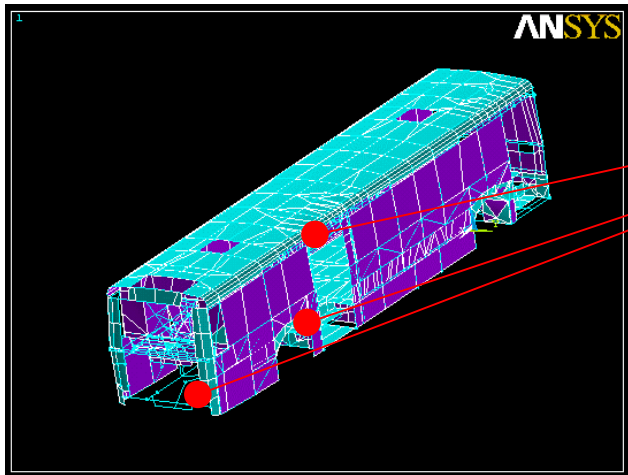
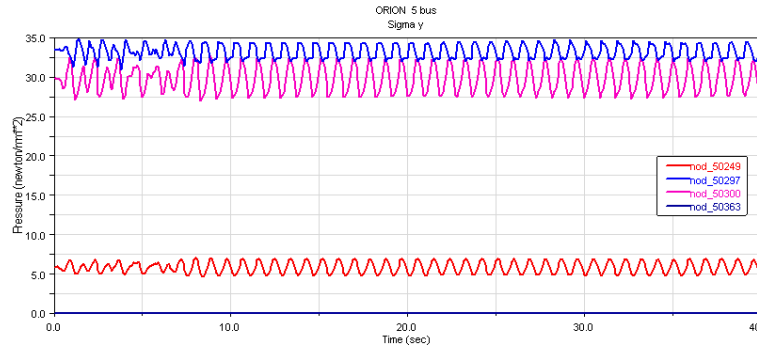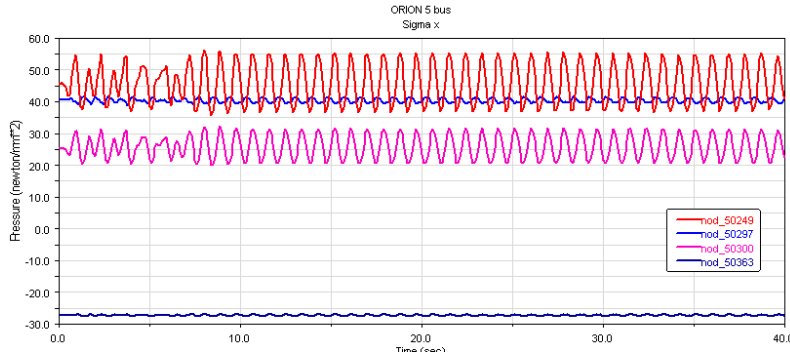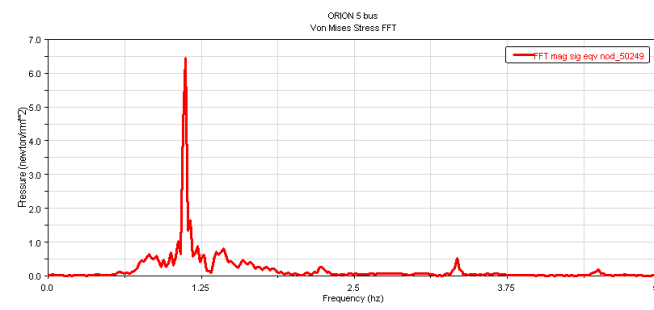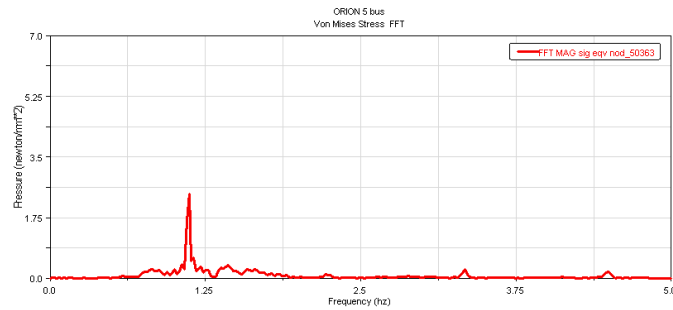- Stress maximum values

  - on nodes
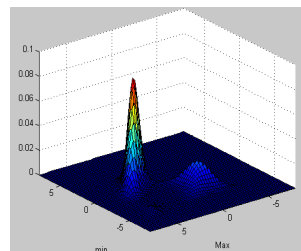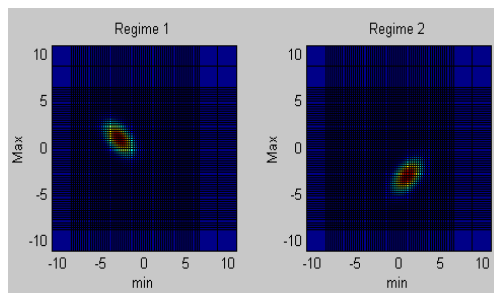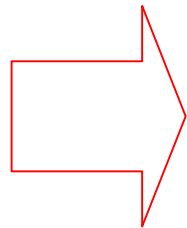   50363,50349,50297,50300
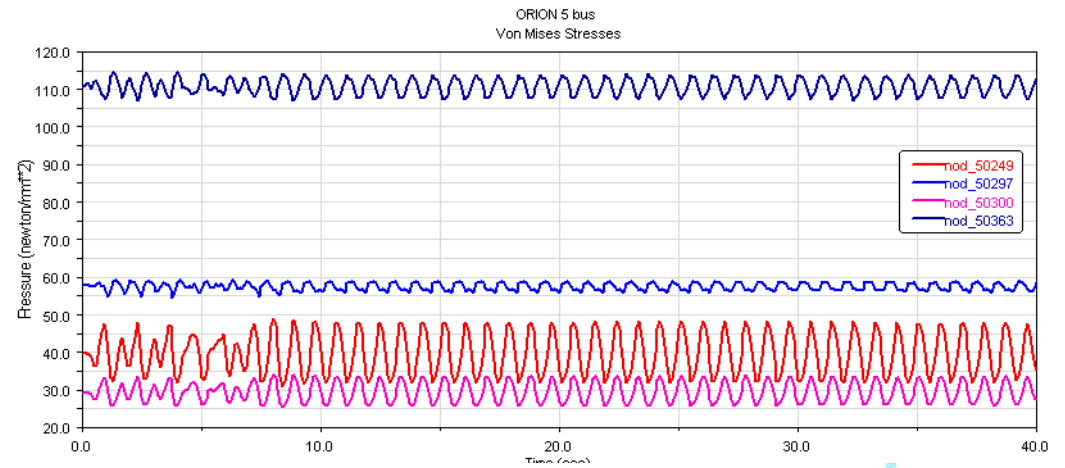
ADAMS-ANSYS Stress Recovery

Stress recovery analysis on a bus flexible frame
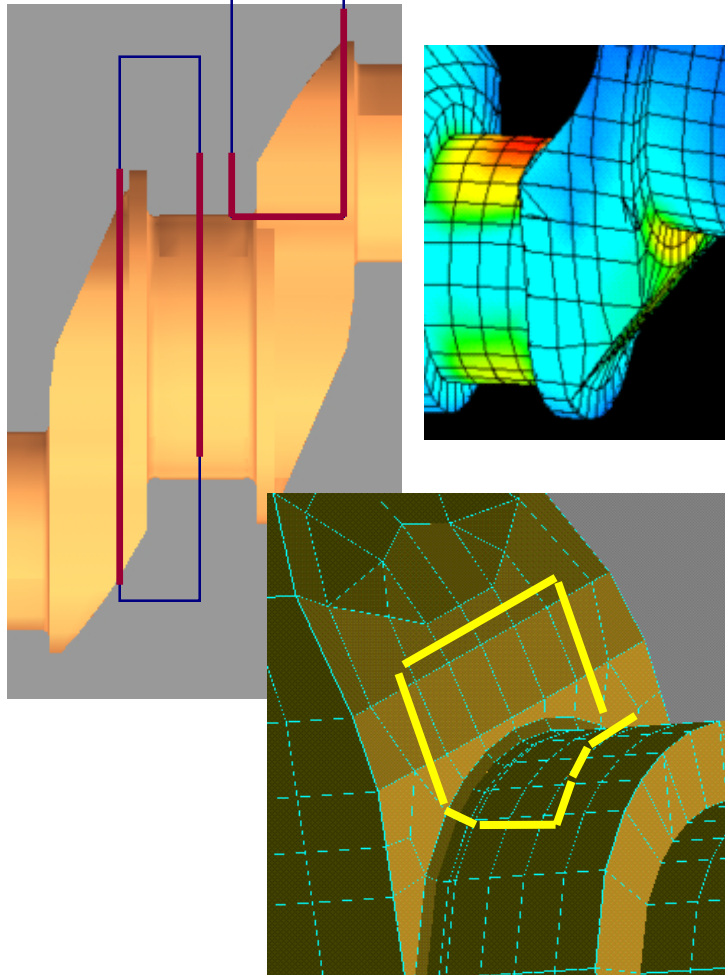## some RESULTS

Stress

time histories

Stress

components

spectrum

Example of Fatigue life
cycle evaluation

(rainflow counting)

Stress Recovery
Considerations for an efficient environment

**Engineering problem with stress recovery: solid models (e.g. crankshaft)**

• **model size : even 1.000.000 nodes are too coarse for accurate stresses in stress concentration areas (e.g.fillets)**

• **accuracy for deformation converges quickly with increasing number of nodes**

• **map results of overall model on detail model(s)**

ADAMS-ANSYS Stress Recovery
Limitations of Stress Vector Method

- Element Types

  - ◆ Stresses on beam elements are calculated only at Post-Processing level so they are not available to be written as Stress Vectors

  - ◆ Some FE nodes may belong to different type of element (i.e. shell and beam), so:

    - • *How to expand Stresses there?*

- Some FE elements feature different stress components

  - ◆ On a shell: top, bottom or middle

- Stress contour plots are not currently available in ADAMS

Mechanical Dynamics
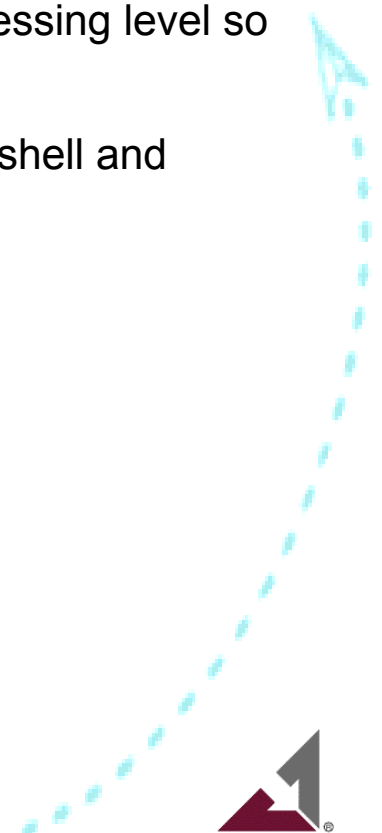
ADAMS-ANSYS Stress Recovery
Limitations of Displacement Vector Method

- ■ **Known issues**

  - ◆ Most of the ANSYS SR analysis cpu time is used to set "D,..."

  - ◆ analysis may become very time consuming on big FE models

  - ◆ Huge ANSYS input file if using ascii format

ANSYS Features a modal dynamic analysis (works using modal superposition) so we could:

  - ◆ Use mnf modal base and ADAMS modal coords to run ANSYS solution (few data to input and fast solutions)

  - ◆ Expand Stresses in ANSYS only
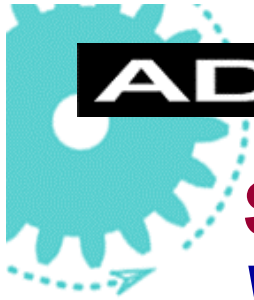
Mechanical Dynamics

Stress Recovery
Vision for better technology integration

- Continue to develop and improve our third part interfaces
  - ◆ MSC/NASTRAN
  - ◆ ANSYS
  - ◆ ABAQUS
  - ◆ I-DEAS

- Export <u>JUST</u> Modal Coordinates from .res file

- Create our own internal MSR using VKI Toolkits
  - ◆ Minimum amount of work to re-use FB data
  - ◆ Modal Coordinates read in a Post-Processing tool
  - ◆ Re-use Displacement Vectors and expand them to compute Nodal Displacements
    - • use FE equation formulation to derive
      - • Element Strains
      - • Element Stresses
  - ◆ Selection of local or global references
  - ◆ Selection of element regions