

DYNAMIC DESIGNER

DYNAMIC

ADAMS



Flexible Bodies Automatic Creation and Dynamic Stress Visualization Using ADAMS

*Diego Minen
Mechanical Dynamics*



Agenda



- Product Vision
- Graphical User Interface



- ◆ Automeshing
- ◆ Modal Stress Generation
- ◆ Extrusion Meshing
- ◆ Modal Truckk Frame
- ◆ Modal Leaf Spring



- Modelling considerations
- FE partnership



- ◆ Tools
- ◆ Validation

- AUTOFLEX and beyond

Vision



- Need to generate flex components at an early stage in virtual prototyping



- FE-MBS-FE too long loop
- system level integrated approach



- ◆ import or define geometry in ADAMS
- ◆ FE mesh in ADAMS
- ◆ anticipate structural component detailed development



- stress pre-calculation in ADAMS
- fatigue pre-calculation in ADAMS
- export components mesh to ASCII files



- ◆ ↻ structural design from dynamic design req ↻
- ◆ ↻ dynamic design from structural design req ↻



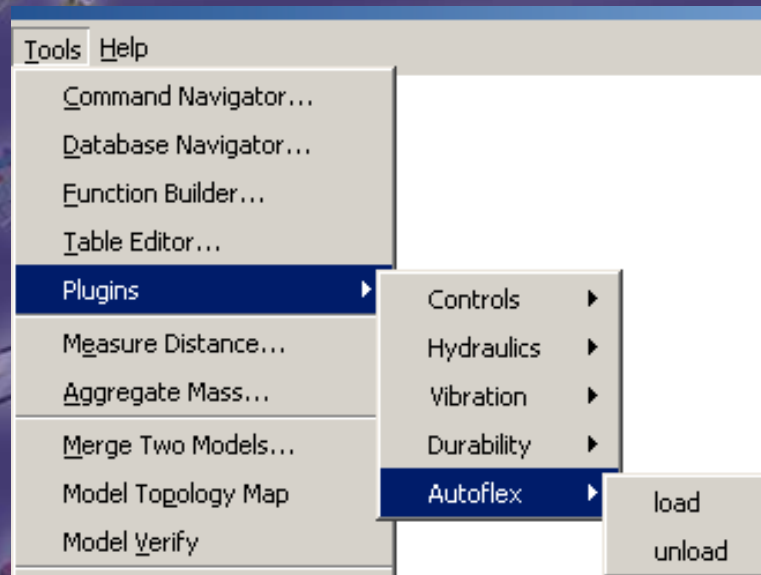
- More efficient concurrent system design

AutoFlex Implementation

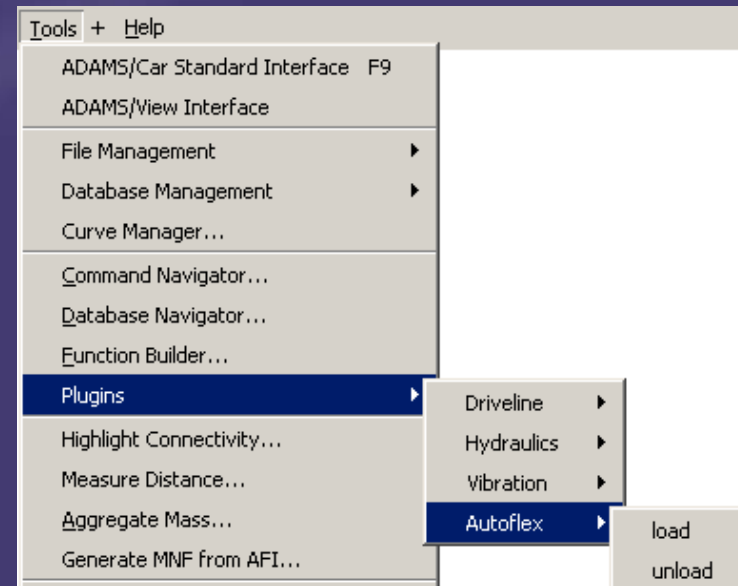
Plug-in feature



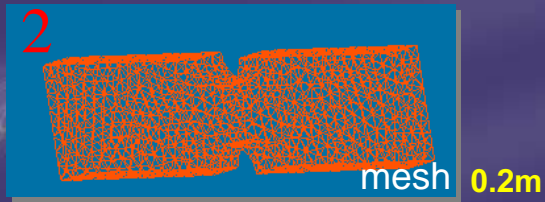
■ View Environment



■ TPL Environment

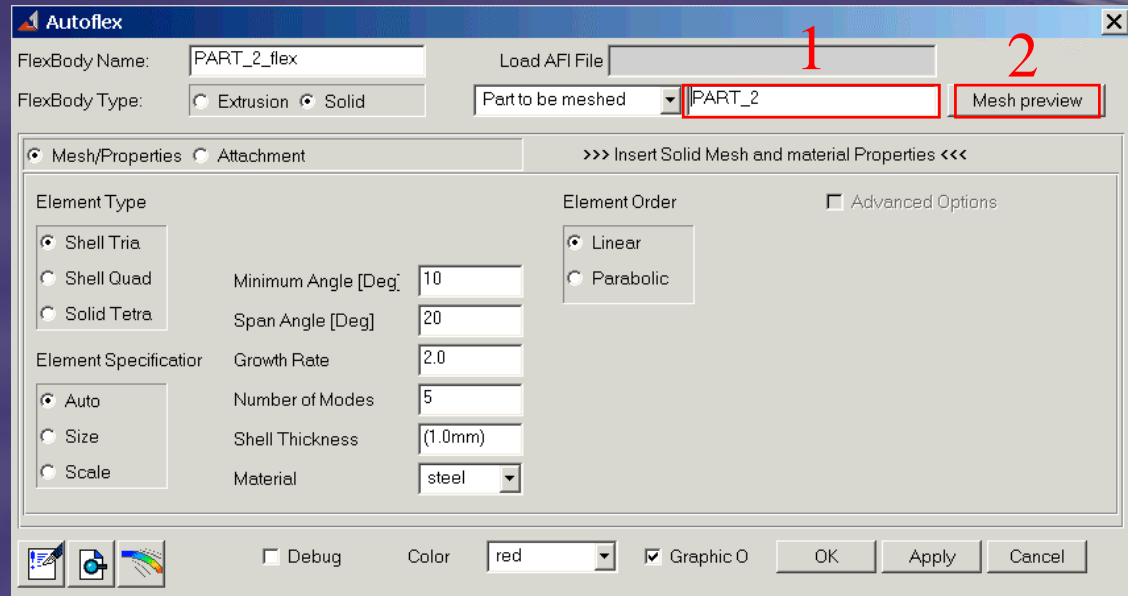


The interface – Object meshing



■ Meshing Control

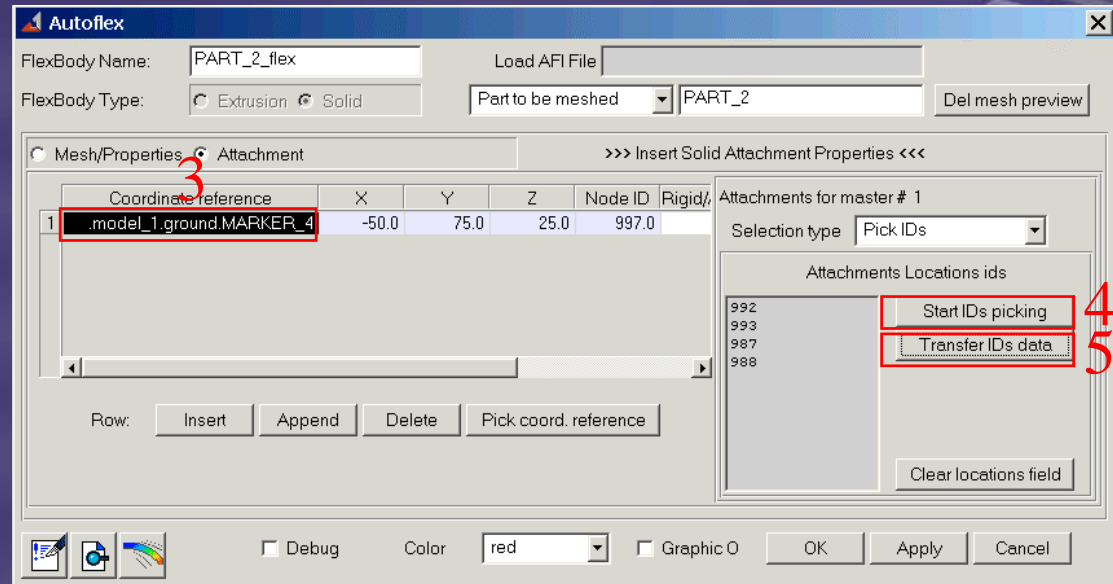
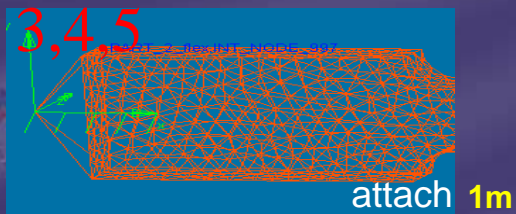
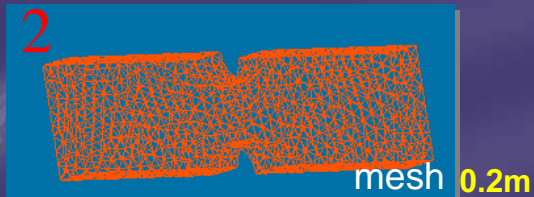
- ◆ Size
- ◆ Min Internal Angle
- ◆ Span Angle
- ◆ Elems Grow Rate
- ◆ Material
- ◆ Shell Thickness



■ A/VIEW and Template Products

- ◆ Solid Objects Meshing
 - Parasolid File
 - A/View objects with booleans
 - ADAMS shell file (surf mesh only)
- ◆ Linear and Parabolic Element Types
 - Surf Mesh Tria
 - Volume Mesh Tetra

The interface – Object meshing



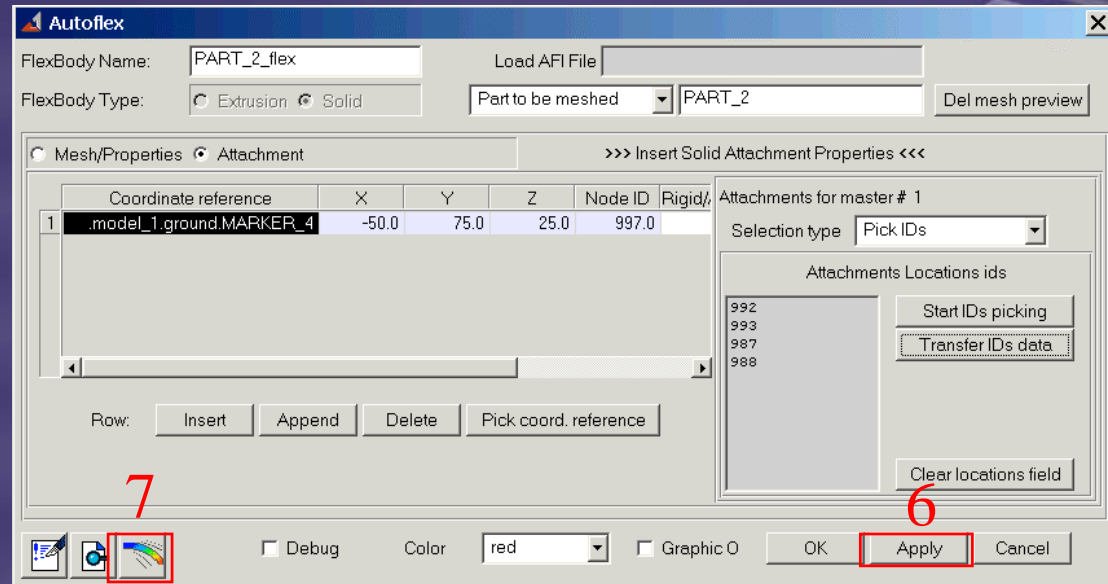
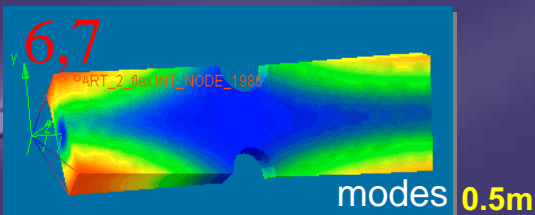
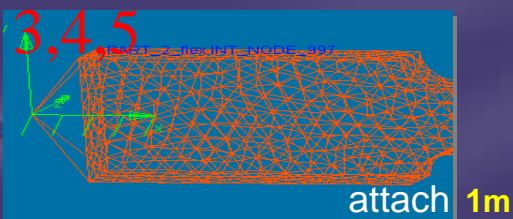
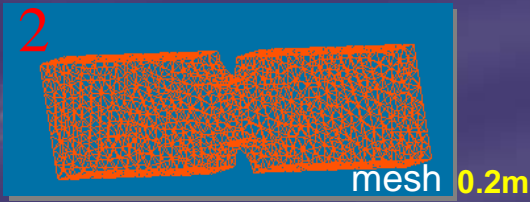
Attachment Control

- ◆ Pick FB node(s)
- ◆ Automatic nodes capturing with primitive intersection
 - ◆ Cylinder
 - ◆ Sphere

A/VIEW and Template Products

- ◆ Attachments parametrized to ground Markers in AVIEW, on HPs or CFs in TPL based products
- ◆ Save parametrization to .afi file
- ◆ While loading file, attachment points modification is recognized
- ◆ Rigid or averaged attachment stiffness (RBE2, RBE3)
- ◆ Release DOFs on each attachment point
- ◆ Control node ID of attachment (for saving connections when FB is modified)

The interface – Object meshing



■ A/VIEW and Template Products

- ◆ Solution monitored while running
- ◆ .log file generation
- ◆ ASCII file .afi describing component features
- ◆ Edit .afi file to compute "MODAL_STRESS" (*)
- ◆ Re-run from command line



```

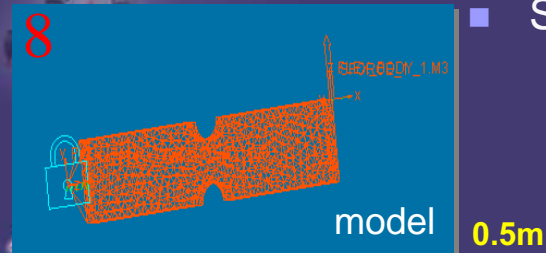
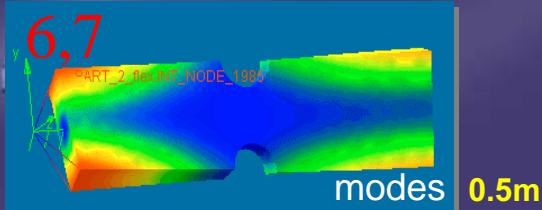
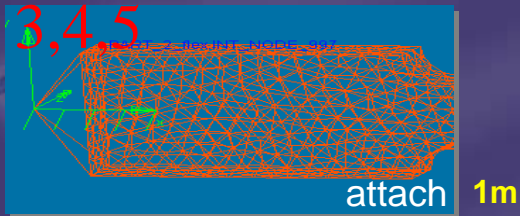
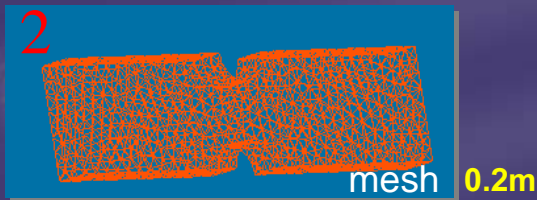
$-----
[ ANALYSIS ]
TYPE = 'MODAL_STRESS'
MODES = 5
    
```

(*) experimental in V12

■ FE Solution

- ◆ A Flex Body is automatically generated and loaded in
- ◆ Mode Shapes are visually checked

Solve in ADAMS and Display Results



Mechanical Dynamics

Compute Nodal Plot Components A/Durability

Analysis: Last_Run
 Flexible Body: FLEX_BODY_1
 Select Node List: 663
 Node to Add to List:

Stress Strain

Normal Y Normal Z
 Shear YZ Shear ZX

Apply Cancel

Animation View Camera Record Overlay **Contours** PPT/Durability

Display Legend
 Plot Type: Von Mises Stress
 Legend Placement: Left
 Legend Title: Von Mises Stress (MPa)
 Colors: 255
 Legend Gradients: 10

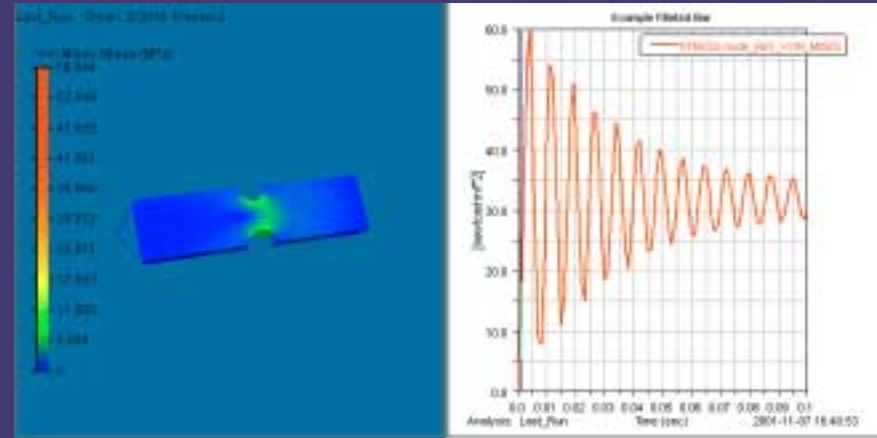
Minimum Value: 0.000000
 Maximum Value: 59.943676
 Reset Limits

■ ADAMS/Durability V12

- ◆ Stress Contours on AutoFlex Flex Bodies (*)
- ◆ Plot Stress Component and Invariants
- ◆ DOE and Optim on Stress Results available

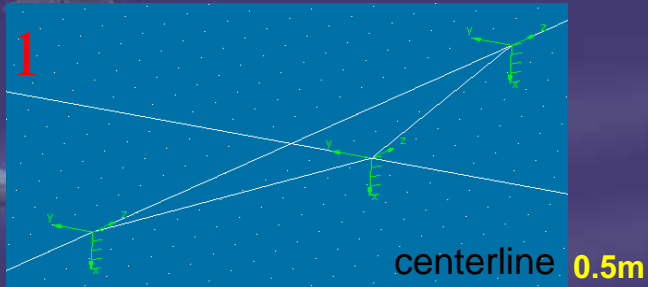
■ Solution

- ◆ Define the mechanism environment
 - ◆ End Step Load
 - ◆ Fix Joint at Attachment
- ◆ Run Analysis



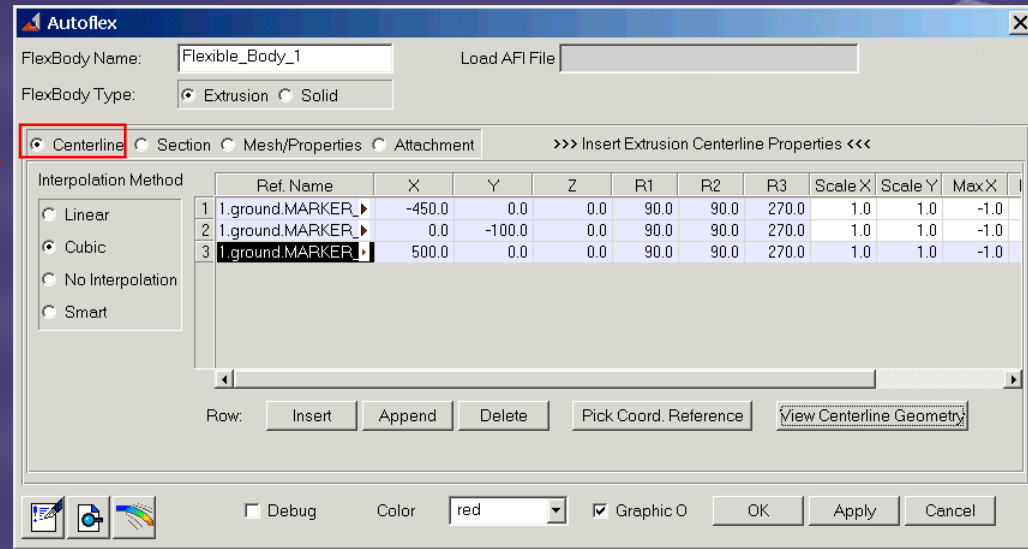
0.5m

The interface – Extrusion meshing



Centerline Points Interpolation

- ◆ Linear
- ◆ Cubic
- ◆ No interpolation
- ◆ Smart
 - ◆ 3 colinear points are connected with straight line, cubic spline elsewhere



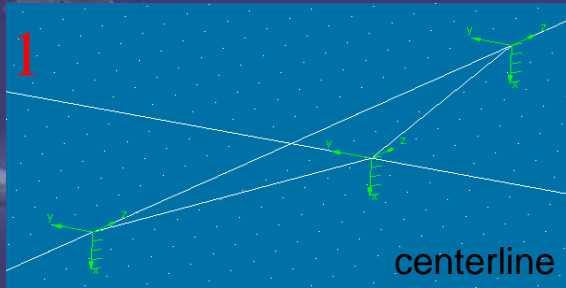
A/VIEW and Template Products

- ◆ Shell mesh on open and closed sections
- ◆ Brick mesh on closed convex sections
- ◆ Linear Element Types
 - 4 node shells
 - 8 nodes bricks

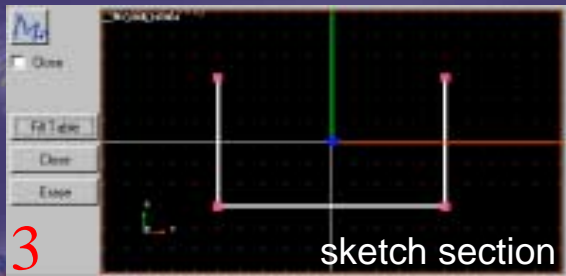
Centerline Points Control

- ◆ Sections scalable X,Y
- ◆ Sections trimmable to max X,Y
- ◆ Variable thickness on sections (shell)

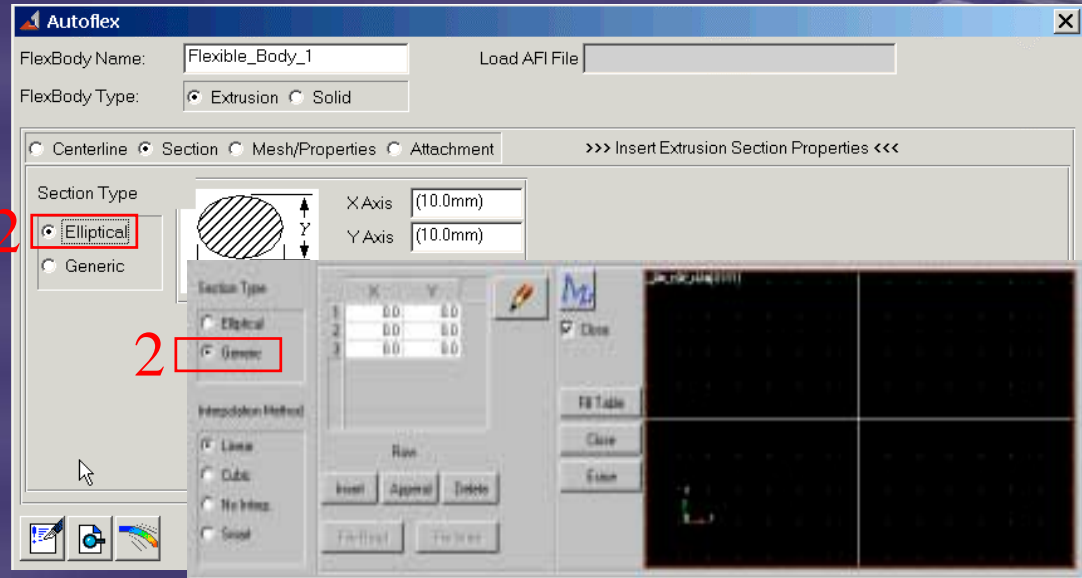
The interface – Extrusion meshing



centerline 0.5m



sketch section 0.5m



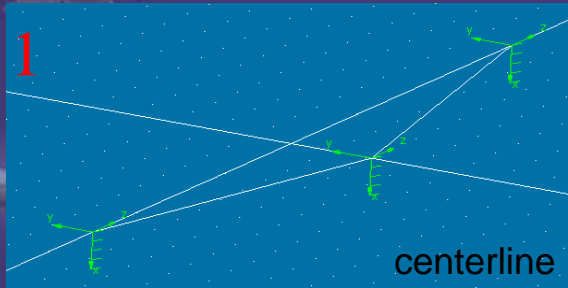
■ Section Points Interpolation

- ◆ Linear
- ◆ Cubic
- ◆ No interpolation
- ◆ Smart
 - ◆ 3 colinear points are connected with straight line, cubic spline elsewhere

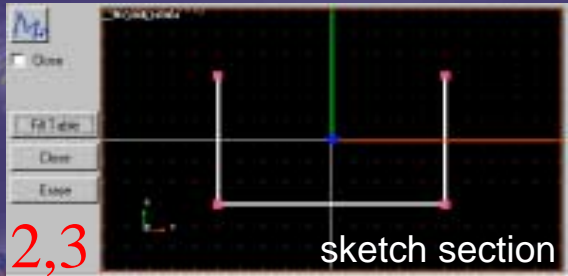
■ A/VIEW and Template Products

- ◆ Generic open or closed sections
- ◆ Points are editable in a table

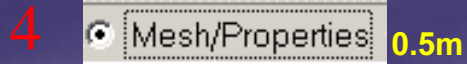
The interface – Extrusion meshing



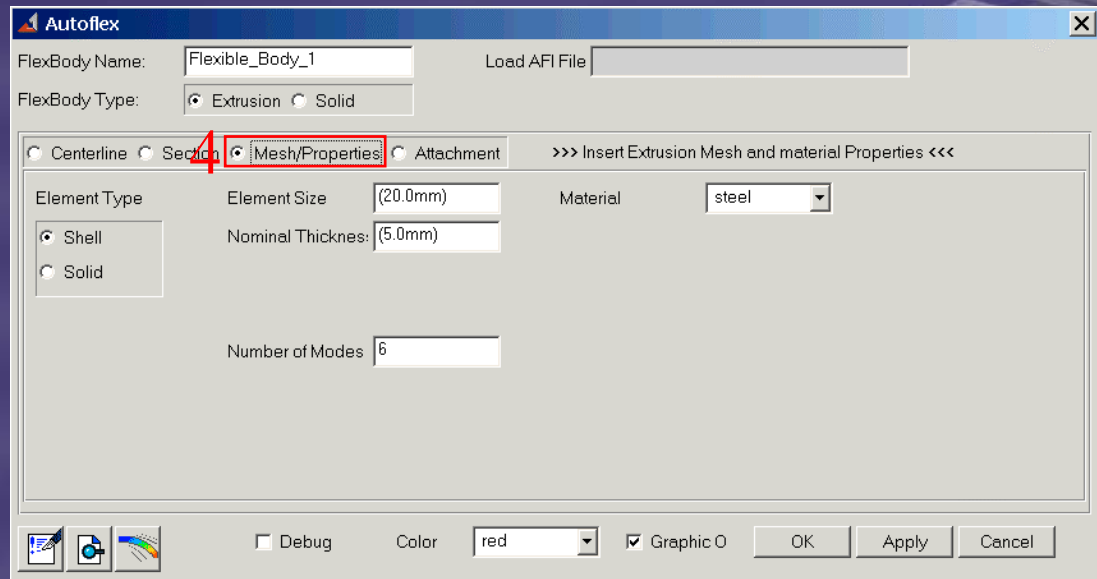
centerline 0.5m



sketch section 0.5m



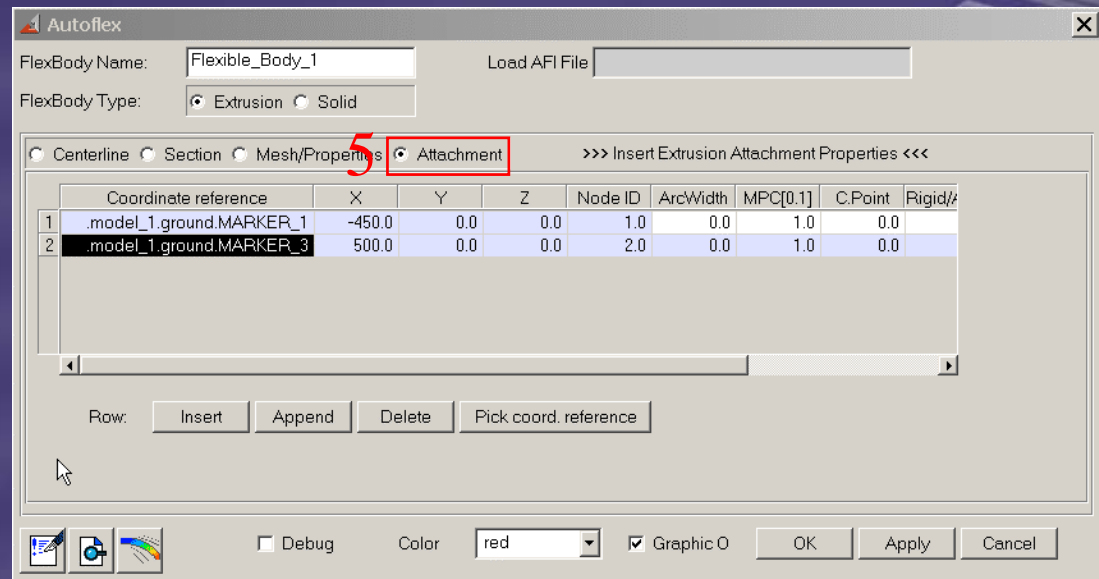
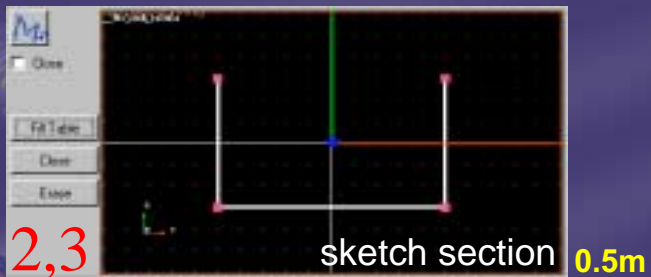
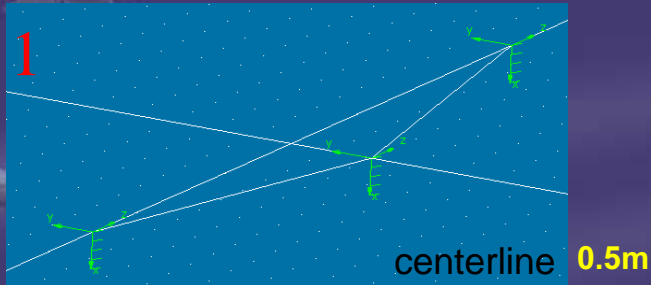
Mesh/Properties 0.5m



■ Mesh Properties

- ◆ Element size
- ◆ Element thickness (shell)
- ◆ Material
- ◆ Number of Normal Modes

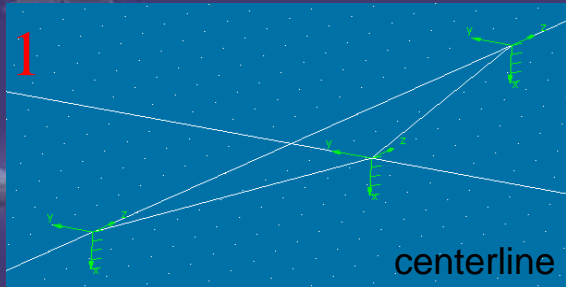
The interface – Extrusion meshing



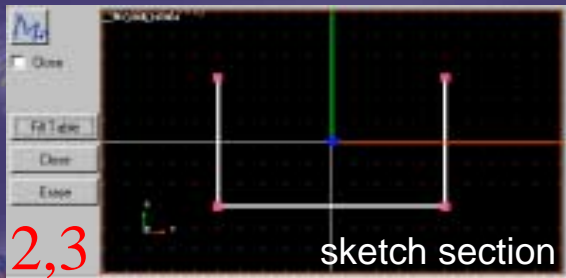
■ Attachment points control

- ◆ Could be same centerline Markers (HPs, CFs) or other ground point
- ◆ Attachment width (along centerline length)
- ◆ Attachment section nodes %age
- ◆ Attachment to a centerline point section
- ◆ Attachment to closest section
- ◆ Rigid or averaged attachment stiffness (RBE2, RBE3)
- ◆ Release DOFs on each attachment point
- ◆ Control node ID of attachment (for saving connections when FB is modified)

The interface – Extrusion meshing



centerline 0.5m



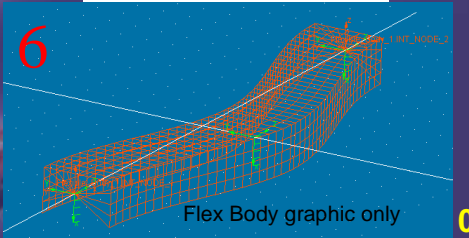
sketch section 0.5m



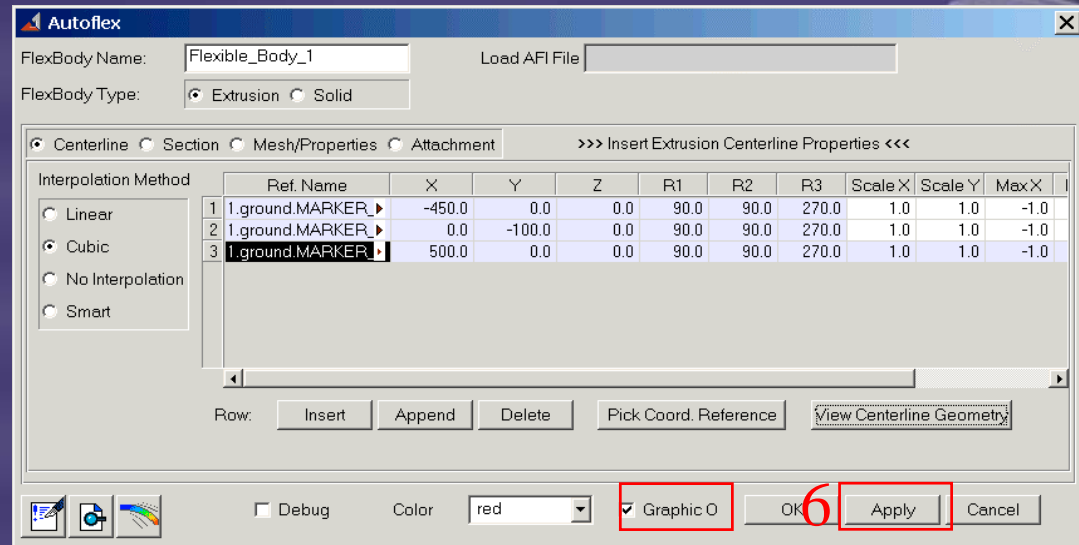
0.5m



0.5m



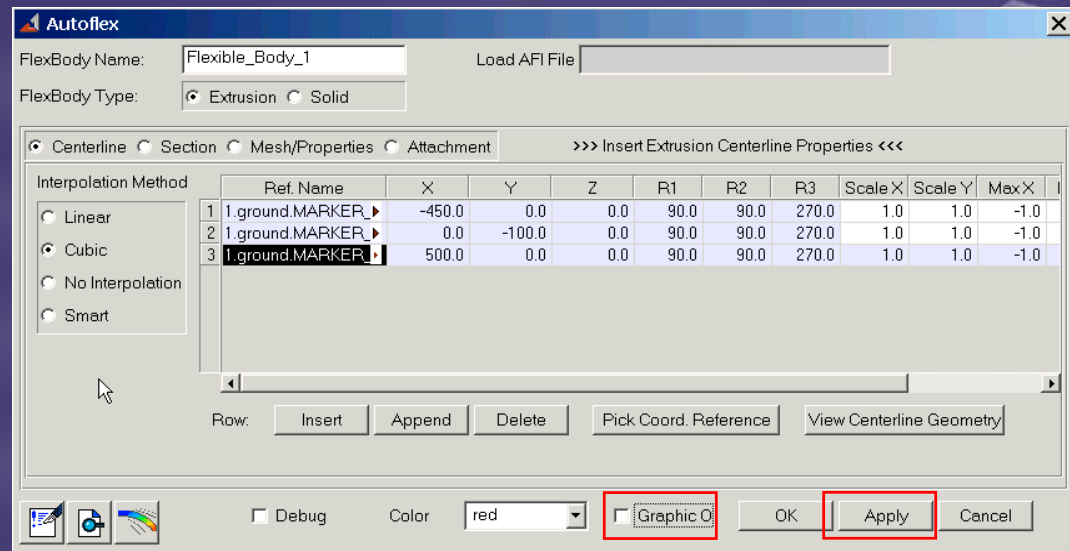
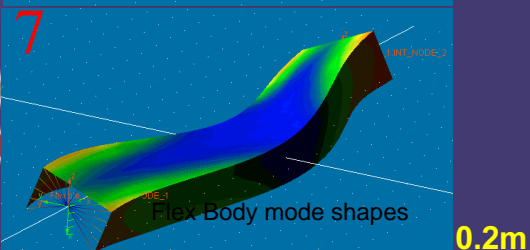
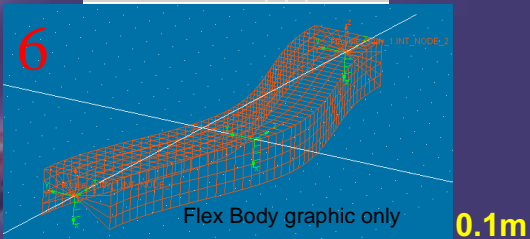
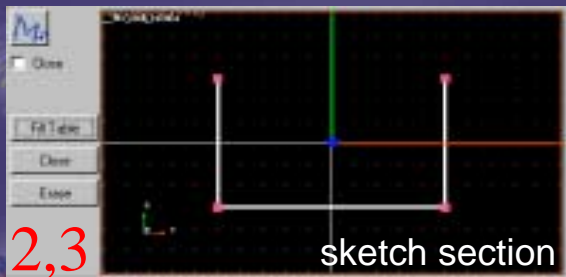
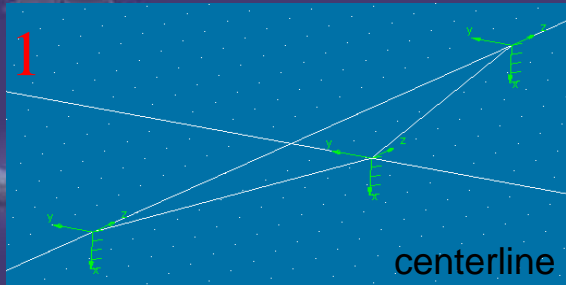
Flex Body graphic only 0.1m



■ Solve Flexible Body

- ◆ Graphic only feedback
- ◆ Modal Neutral File generation w.o. mode shapes
- ◆ .afi file generation

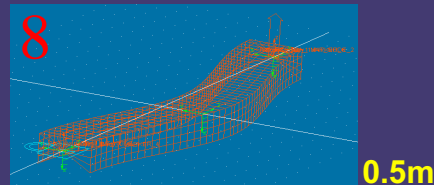
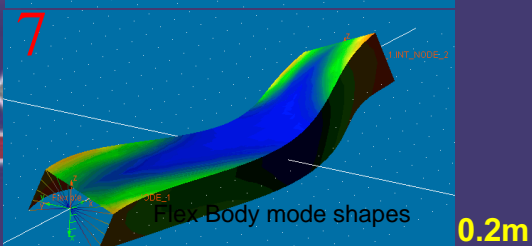
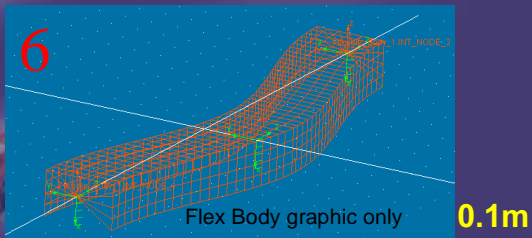
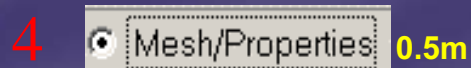
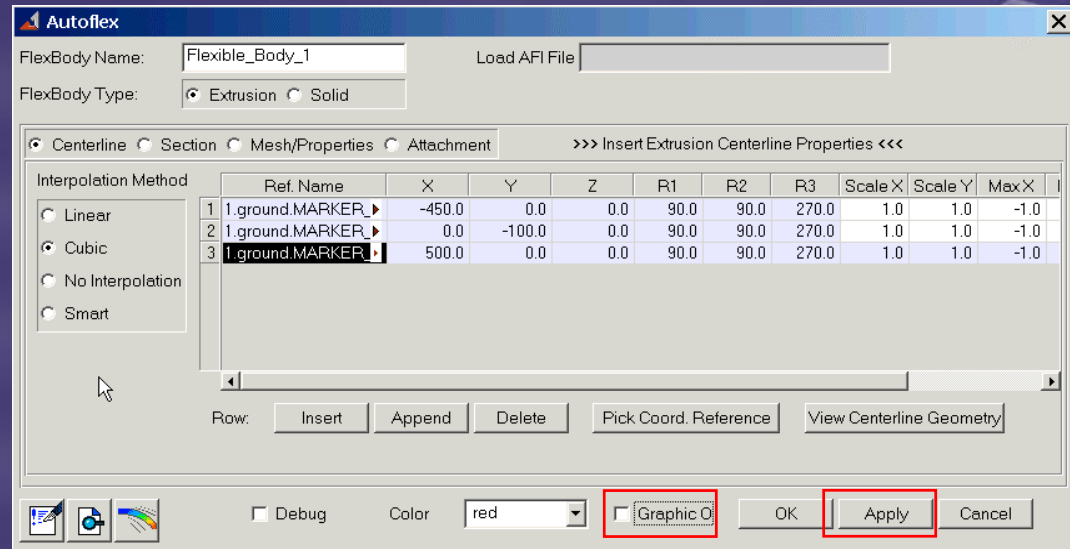
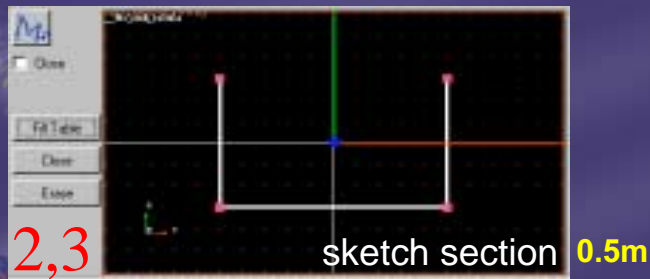
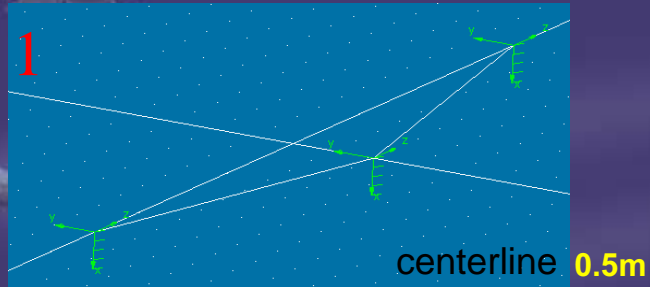
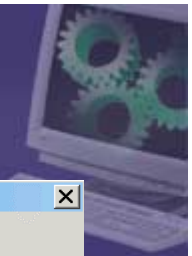
The interface – Extrusion meshing



■ Solve Flexible Body

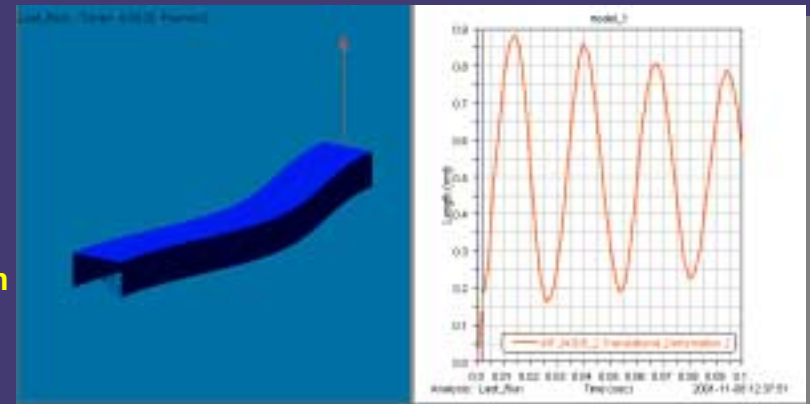
- ◆ Graphic only feedback off
- ◆ Modal Neutral File generation with mode shapes
- ◆ .afi file generation

Solve in ADAMS and Display Results



Solution

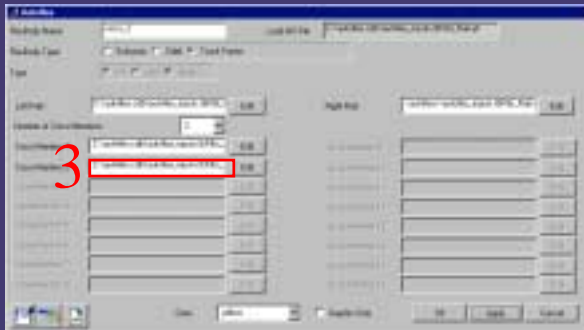
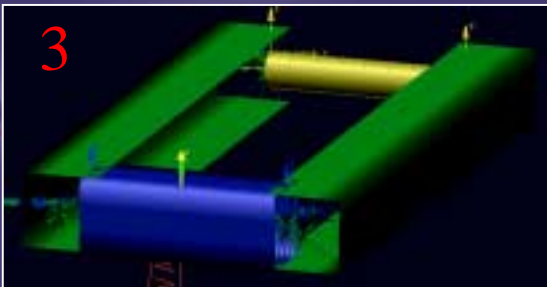
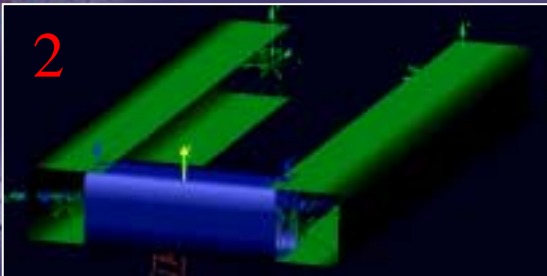
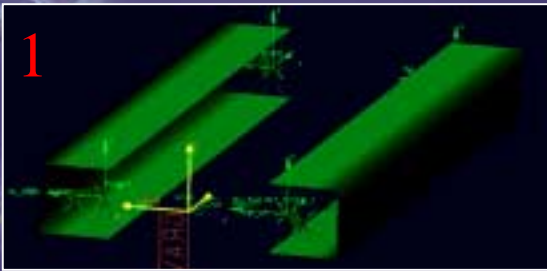
- ◆ Define the mechanism environment
- ◆ End Step Load
- ◆ Fix Joint at Attachment
- ◆ Run Analysis



Modal Truck Frame Builder



- Available in TPL based products



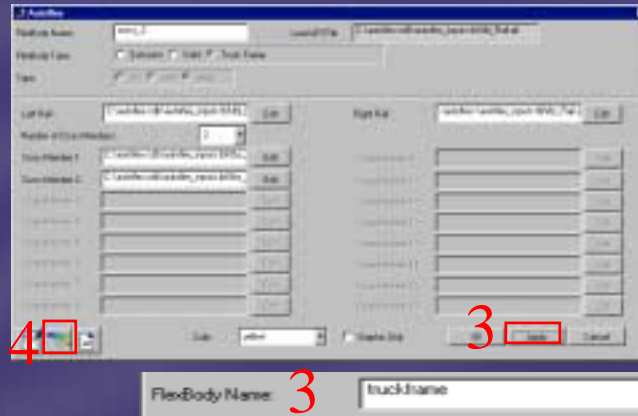
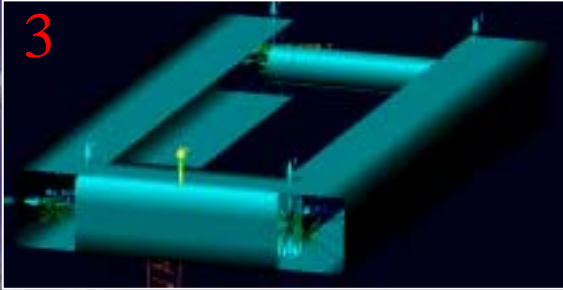
- Define Rails

- ◆ Use AutoFlex Extrusion for editing or importing
- ◆ Left/Right objects
- ◆ Max 2 rails definable

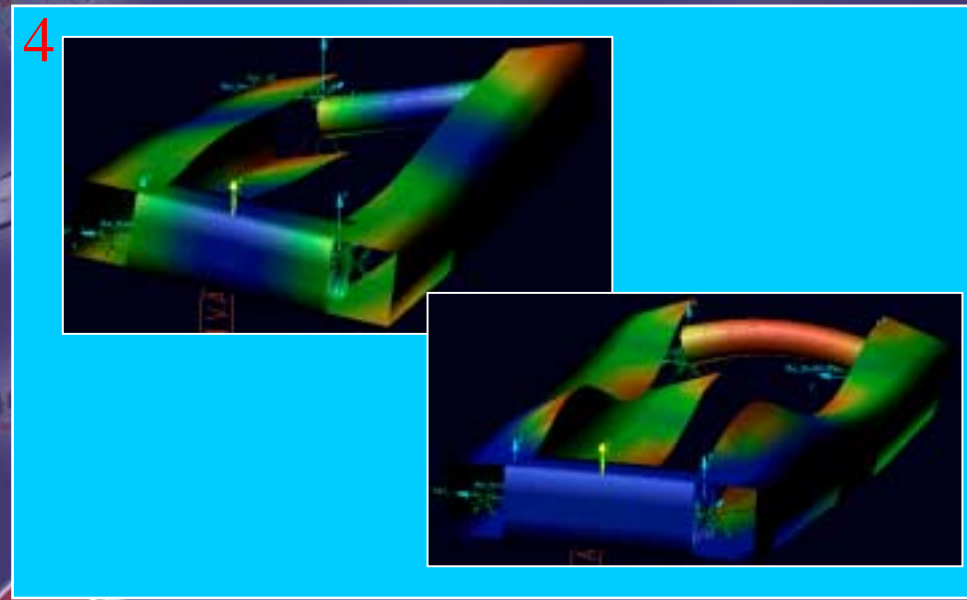
- Define Cross Members

- ◆ Use AutoFlex Extrusion for editing or importing
- ◆ Single type objects
- ◆ Max 16 cross members definable

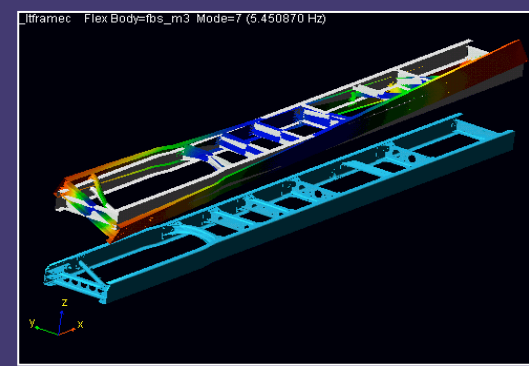
Modal Truck Frame Builder



- Define Merged Frame
 - ◆ Name of the assembled FB
 - ◆ Investigate Mode Shapes
 - ◆ Static Modes are not computed on merged points
 - ◆ Attachment points parametrization is preserved



- Only Extrusion Objects are mergeable with GUI
- Object meshes are mergeable from command line
- To connect extrusions-objects ADAMS joints are needed



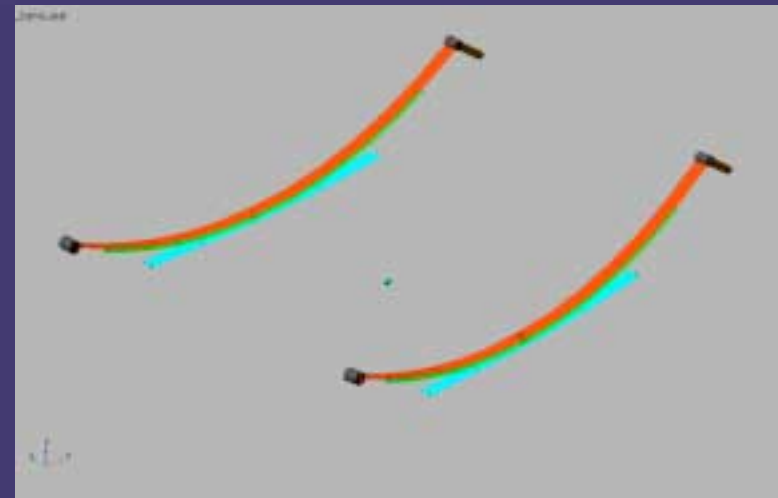
AutoFlex Modal Leaf Spring



- Automatic generation of leaf-packages with flexible body
- Available in TPL based products
- Multileaves levels definable
- MultiFlexBodies for each leaf



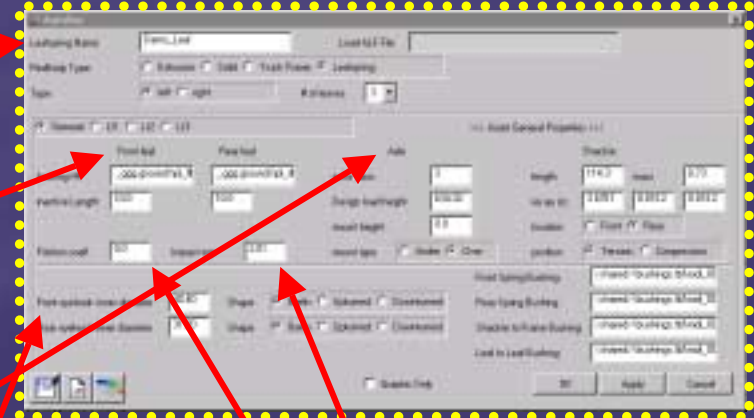
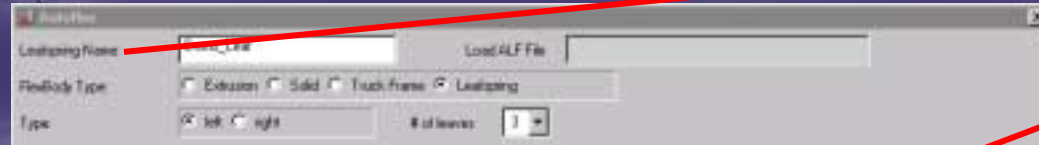
- ◆ Centerline properties
- ◆ Section properties
- ◆ Material
- ◆ Element types
- ◆ Attachment properties
- ◆ Number of modes
- ◆ Color
- ◆ Contact stiffness between leaves
- ◆ Friction between leaves



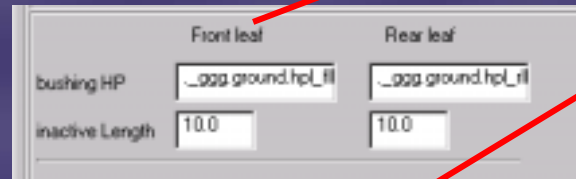
AutoFlex Modal Leaf Spring

General Data Panel

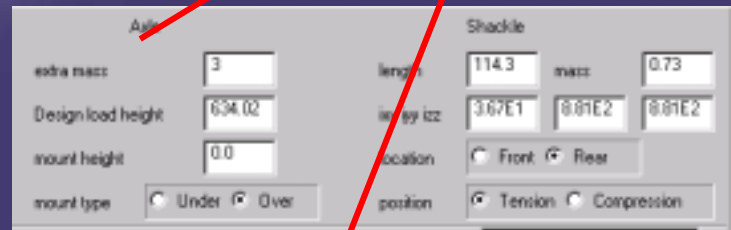
Object name and type



Location



Axle & Shackle properties



Contact exponent

Contact friction

Contact stiffness is specified in the UDE

Eyehooks & Attachments



AutoFlex Modal Leaf Spring

Leaves Data Panel

Leaf number

General
 L1
 L2
 L3

Geometry definition

Front length: 382
 # front FBs: 2
 Rear length: 457
 # rear FBs: 2
 Section thickness: 5.0
 Section width: 50.0
 Z offset: 22.0
 Gap U1/U2/U3: 2.54

Profile coordinates

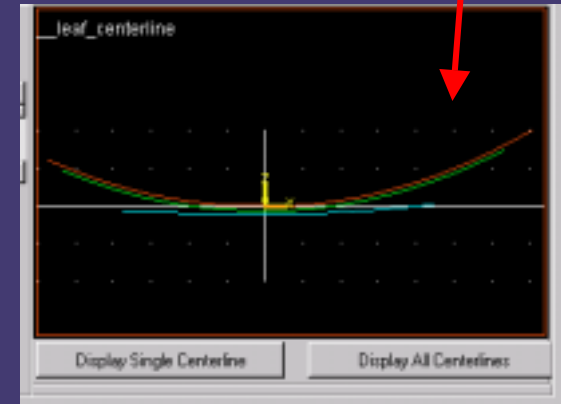
Centerline Properties

	X	Z	Spine W	Thick.
1	-373.08	4.35	1.0	-1.0
2	-344.39	4.26	1.0	-1.0
3	-311.07	4.09	1.0	-1.0
4	-275.89	3.52	1.0	-1.0
5	285.72	3.08	1.0	-1.0

Row:
 cautoflex\vacar_leaf_cr

FEM properties

Material: steel Damping: 0.0
 Shell Size: 20.0 # Nodes: 6
 Graphic Only



Centerline preview

AutoFlex Modal Leaf Spring Profile Generation



Centerline Properties

	X	Z	Scale W	Thick.
1	-373.08	4.35	1.0	-1.0
2	-344.39	4.26	1.0	-1.0
3	-311.07	4.08	1.0	-1.0
4	-275.89	3.52	1.0	-1.0
W	145.72	1.98	1.0	1.0

Row: [Insert] [Append] [Delete]
Get from file: C:\autoflex\locat_leaf_cr

- Profile definition
 - ◆ Profile coordinates
 - ◆ Width coefficients
 - ◆ Thickness
 - ◆ Inactive length
 - ◆ Eyehook shape
 - ◆ Shackle properties

- Final Leaf points calculation
 - ◆ Profile fitting
 - ◆ Extension
 - ◆ Orientation
 - ◆ Eyehook location
 - ◆

MLSPTS
Standalone
Executable



- Leaf construction

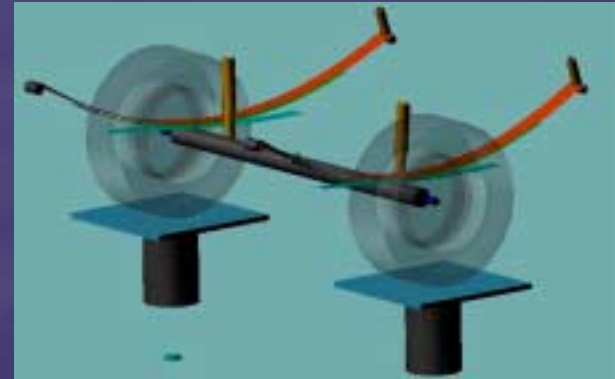
AutoFlex Modal Leaf Spring Assembling



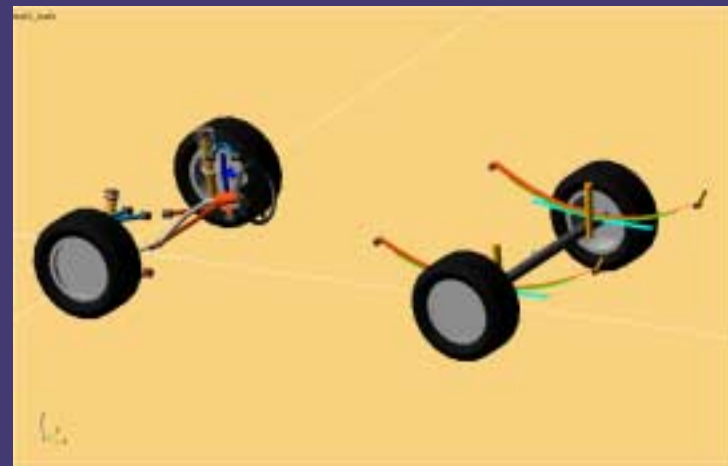
Automatic leaf pack
template generation



Shared axle template



Suspension & Vehicle assembly



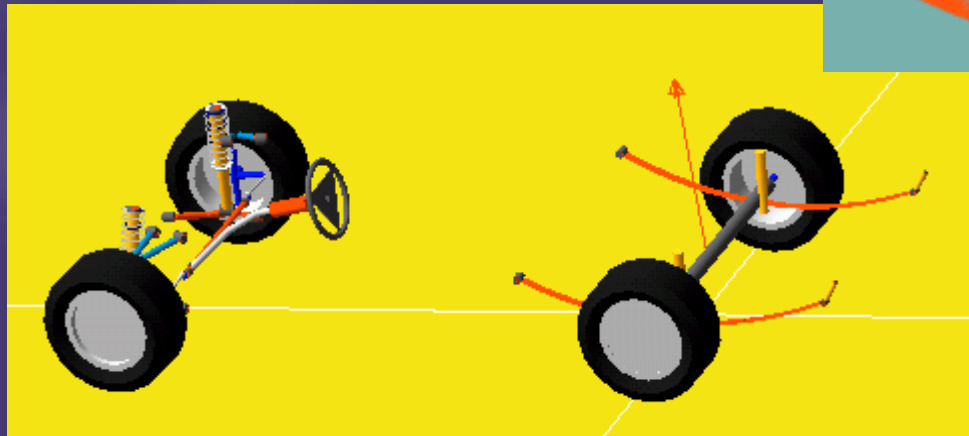
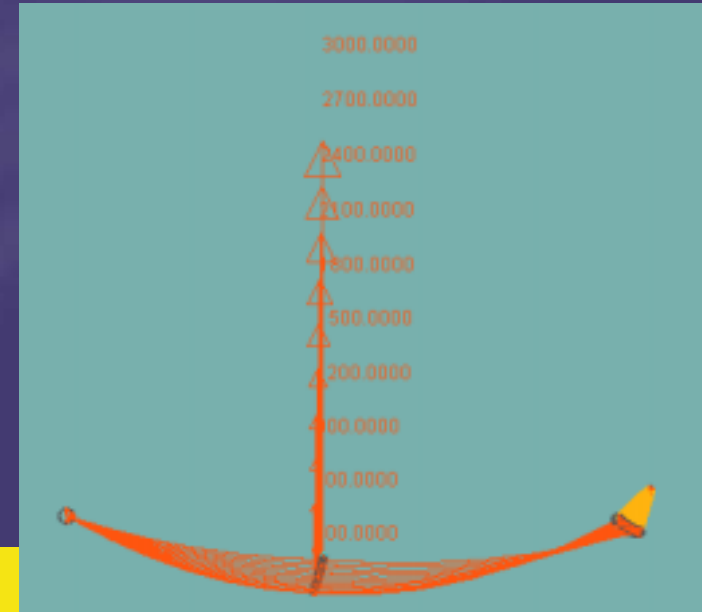
AutoFlex Modal Leaf Spring

Preload analysis



■ Features

- ◆ Preload the leaf pack up to a desired height
- ◆ Compatible with all standard Car analysis
- ◆ Preloaded configuration reached during initial static analysis



AutoFlex Modelling Considerations “-ves & +ves”



- AVIEW and TPL based product are not CAD systems
- Objective is not detail meshing
 - ◆ Local mesh editing is not allowed
- AutoFlex nodes and elements file is not editable
- AutoFlex is not an alternative to popular FE codes, rather complementary
- Beams are not available in AutoFlex
 - ◆ In Component Mode Synthesis number of DOFs is controlled by number of modes and not number of nodes
 - ◆ Beam Elements are not of any help to reduce model size and DOFs in ADAMS
- AVIEW and TPL based products communicate with CAD and FE programs
 - ◆ Import external files for meshing
 - ◆ Export mesh ASCII files in popular FE code format
- Mesh on objects (ParaSolid)
- Mesh on concept (Extrusion)
- Mesh on assemblies
 - ◆ Modal Truck Frame – Multiple FB merge in one
 - ◆ Modal Leaf Spring
- AutoFlex produces editable object file (.afi)
 - ◆ Contains object (PS) or concept (Ext) references



MDI FE Partner for AutoFlex Visual Kinematics Inc.



■ Company Background

- ◆ Founded in 1989
- ◆ Located in Saratoga, CA (Silicon Valley)
- ◆ Component software libraries
- ◆ OEM and joint development agreements
- ◆ Major clients in CAD, CAE, CFD, E&M



■ Development Libraries for Engineering Applications

- ◆ Applied mechanics, fluid dynamics, kinematics and electromagnetics
- ◆ CAE end-to-end application development tools
- ◆ CAE integration and interoperability tools
- ◆ Specialize in finite element analysis
- ◆ Object oriented programming



VKI - Component Software Libraries



■ Products

- ◆ **DevTools** -- low level foundation components
 - **VisTools**, data structures, visualization algorithms, FEA data manipulation
 - **VisTools/Mesh**, automatic mesh generation
 - **VglTools**, graphics device interface, rendering
 - **VdmTools**, finite element database access
 - **VfeTools**, finite element formulations, material models
 - **VfsTools**, equation solver, eigen solver, solution procedures
- ◆ **FOCUS/SDK**-- embeddable finite element post processor
- ◆ Embedded in over 20,000 seats of FEA software

■ Consulting

- ◆ Accelerate application development
- ◆ Visualization and graphics techniques
- ◆ Element formulations and material models
- ◆ Solution procedures



VKI - Flexible Body Generation

- VisTools, VisTools/Mesh

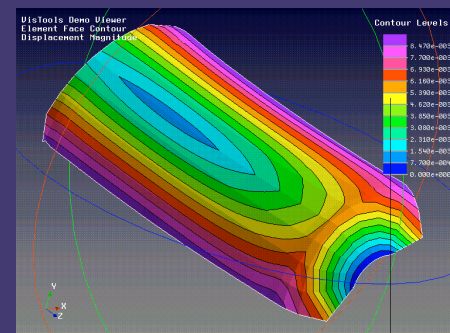
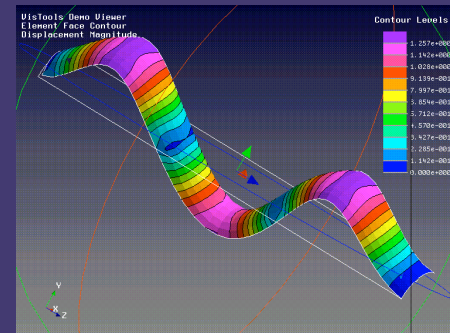
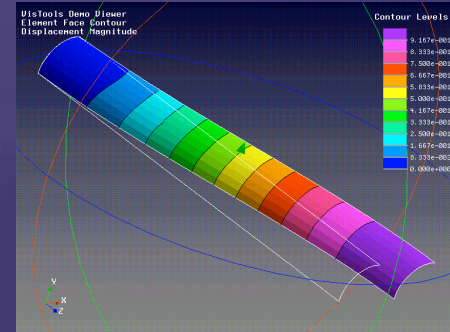
- ◆ Finite element mesh generation
- ◆ Data structures

- VfeTools

- ◆ Linear solid, shell stiffness, mass, loads
- ◆ Kinematic and distributed couplings (rigid elements)
- ◆ Linear elastic material model
- ◆ Element distortion checking
- ◆ Displacement coordinate system transformation
- ◆ Stress mode generation

- VfsTools

- ◆ Attachment point modes, direct sparse solver
- ◆ Vibration modes, large scale eigensolver
- ◆ Residual load modes
- ◆ Full system reduction, flex body stiffness and mass



VKI - Mesh Generation



■ Design Considerations

- ◆ Comprehensive meshing toolkit
- ◆ Combination of meshing tools generally required
- ◆ Linear, Parabolic, Serendipity and Lagrange element types
- ◆ 2D Plane, 3D Volume
- ◆ Curve and Surface meshing
- ◆ Input geometry – discrete geometry
 - Linear or parabolic lines or triangles
 - Optional line or triangle vertex normals

■ Unstructured Meshing

- ◆ Delaunay based - triangulation, tetrahedralization
- ◆ Constrained smoothing
- ◆ Indirect methods
- ◆ Quadrilateralization of triangulation (on going work)



VKI - Curve and Surface Meshing



■ General Features

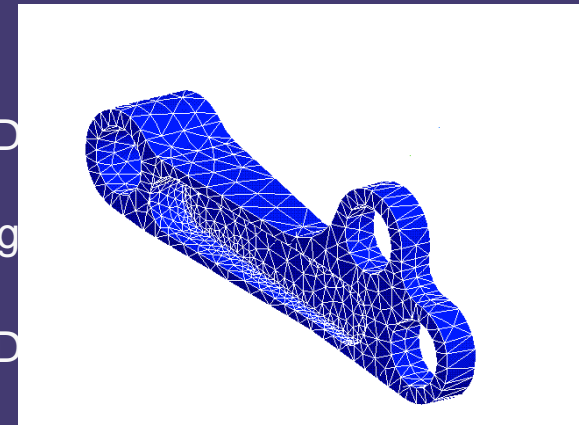
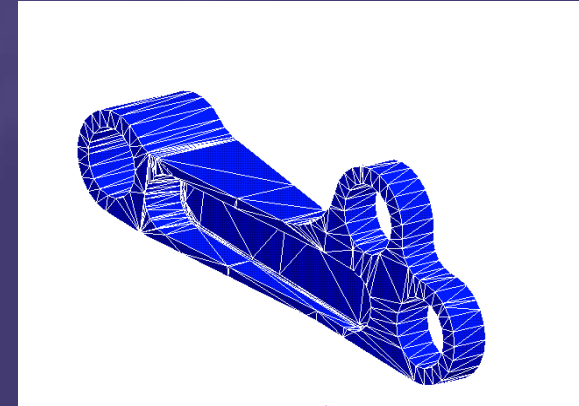
- ◆ Input geometry
 - Linear or parabolic lines or triangles
 - Preserved line endpoints, triangle edges
 - Option triangle node normals
 - Optional geometry functions
- ◆ Regrid existing FE mesh, ParaSolid tessellation
- ◆ Incorporate arbitrary user defined points into mesh
- ◆ Allows meshing across geometry topology (edges)
- ◆ Target edge length, curvature tolerance controls
- ◆ Optional node based edge length control

■ Curve Meshing - CurvMesh

- ◆ Generates linear and parabolic line elements in 2D/3D space
- ◆ Used as precursor to 2D plane or 3D surface meshing

■ Surface Meshing - SurfMesh

- ◆ Generates linear and parabolic triangles (quads) in 3D space
- ◆ Used as precursor to 3D volume meshing



VKI - Finite Element Analysis



■ Finite Element Formulations

◆ Structures

- Stiffness, geometric stiffness, mass, applied forces, reactions, stress, strain
- Geometric nonlinearities, large strain

◆ Heat Transfer

- Conductance, capacitance, heat generation, temperature gradient, heat flux
- Convection, contact resistance

◆ Solids, shells, beams, trusses

- Linear, parabolic
- 3D and 2D plane stress, plane strain and axisymmetric

◆ Discrete

- Spring, damper, concentrated mass, rigid body, MPC

◆ Contact

- Point-to-point, small sliding interface, large sliding point-to-surface

◆ Host application owns and controls input and output

- Calculate element matrices and vectors
- Calculate element stress and strain



VKI - Finite Element Analysis



■ Material Models

- ◆ Isotropic, orthotropic and anisotropic linear material properties
- ◆ Large strain plasticity, viscoelasticity, hyperelasticity
- ◆ Surface interactions
 - Contact/tied, slip/no-slip/friction
- ◆ All materials are temperature dependent
- ◆ Abstract interface to material subsystem (~12 functions)

■ Element Properties

- ◆ Geometry of degenerate finite elements
- ◆ Shell properties - monocoque, general integration, matrix input
- ◆ All element properties may vary along element nodes

■ Element Independent Utilities

- ◆ Degree of freedom rotation, offset, condensation
- ◆ Co-rotational formulation



VfeTools - Accuracy



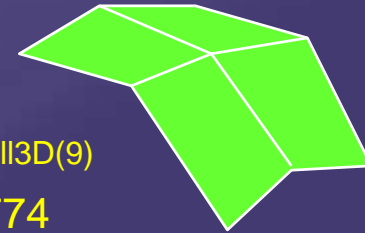
■ MacNeal - Harder Spherical Shell

◆ Doubly curved shell, displacement under load = .0940

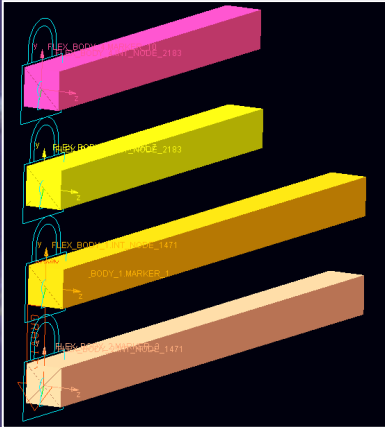
◆ VKI	Shell3D(4)	Shell3D(6)	Shell3D(8)	Shell3D(9)
2	0.909	0.906	1.333	1.774
4	0.998	0.900	0.467	0.956
8	0.990	1.021	0.892	0.985
12	0.990	1.020	0.986	0.988

◆ NASTRAN	QUAD4	QUAD8
2	0.972	0.025
4	1.024	0.121
8	1.005	0.823
12	0.998	0.992

◆ ABAQUS S4	STR165	S8R5	S9R5
2	0.444	0.229	0.796
4	0.925	0.916	0.951
8	0.974	0.967	0.982



VfeTools - Accuracy



- Steel cantilevered brick subjected to gravity

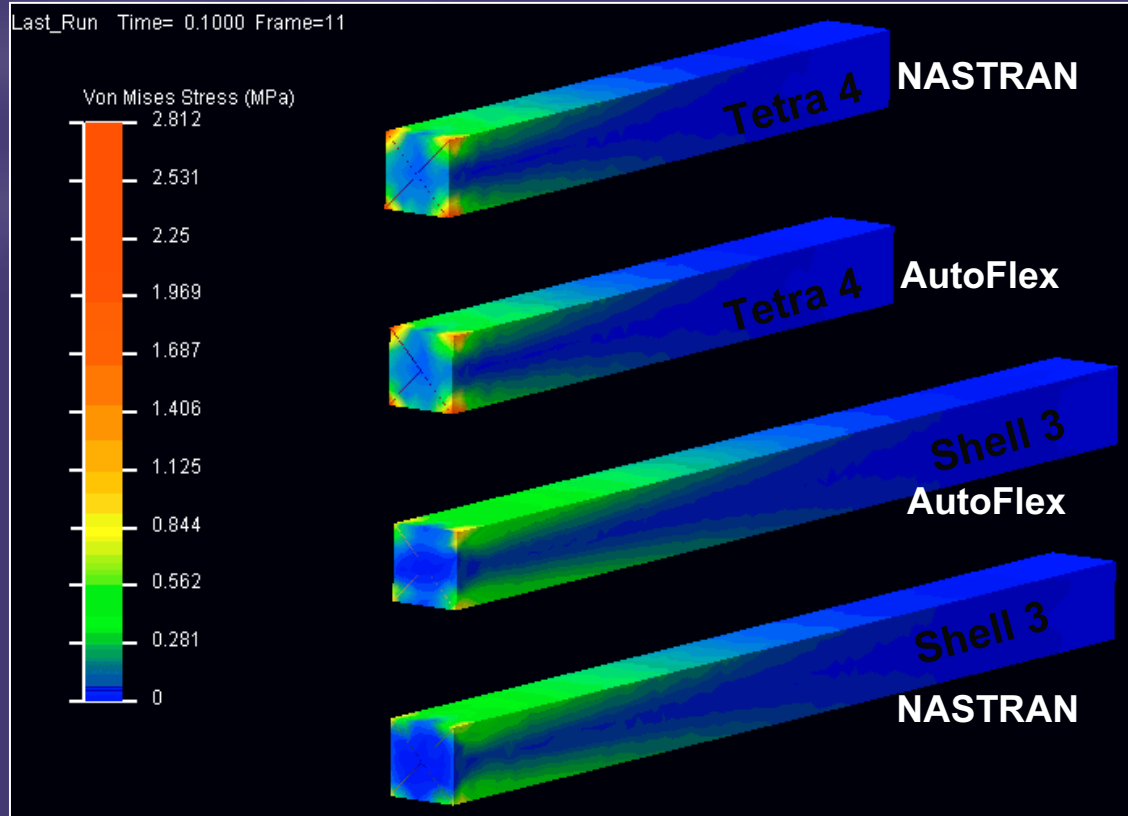
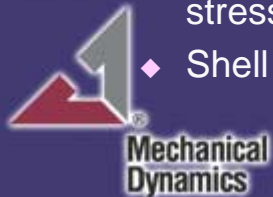
- ◆ Tet Mesh
 - NASTRAN
 - VKI
- ◆ Shell TRIA mesh
 - NASTRAN
 - VKI

- Dimension

- ◆ Tet Mesh
 - 20*20*200[mm]
- ◆ Shell TRIA mesh
 - 20*20*300[mm]
 - Thickness=1mm

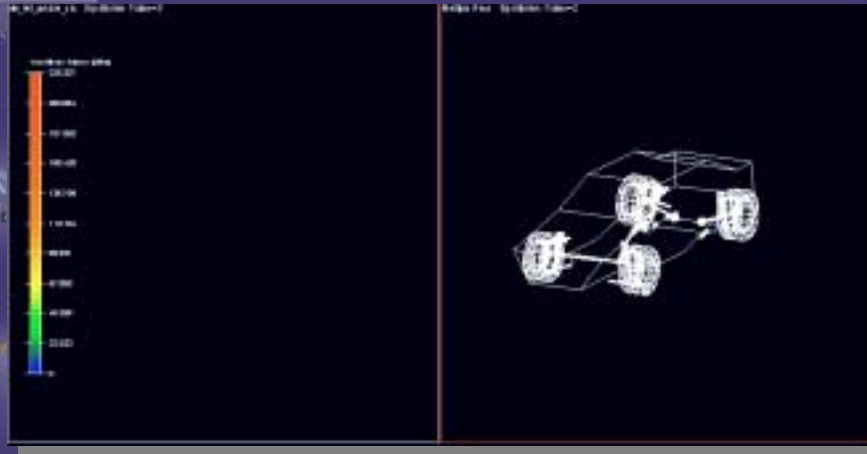
- Results (on-going validation)

- ◆ Tetra 4: within 1% VM stress
- ◆ Shells 3
 - NASTRAN does not project centroidal stress to grids
 - VKI does grid projection
 - Within 1% VM stress in ABAQUS
- ◆ Tetra 10: within 1% VM stress
- ◆ Shell 6: within 1% VM stress

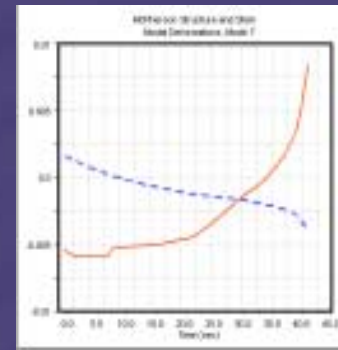


Applications

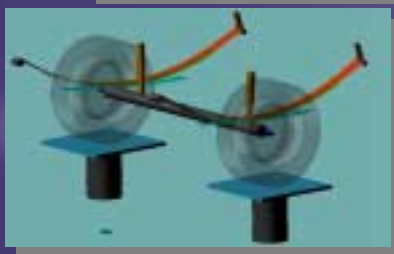
DESIGNER
DYNAMIC
ADAMS



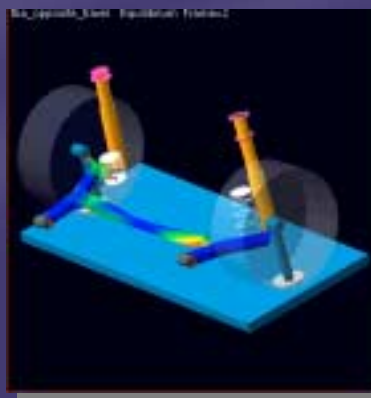
LCA



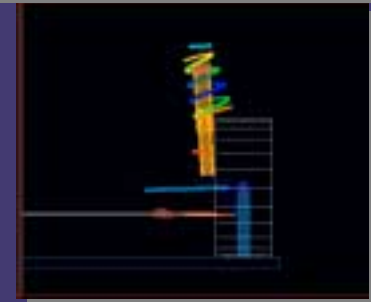
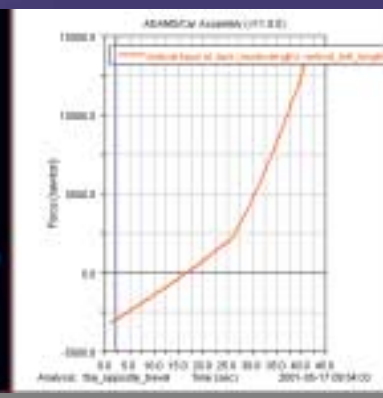
McPherson Struct



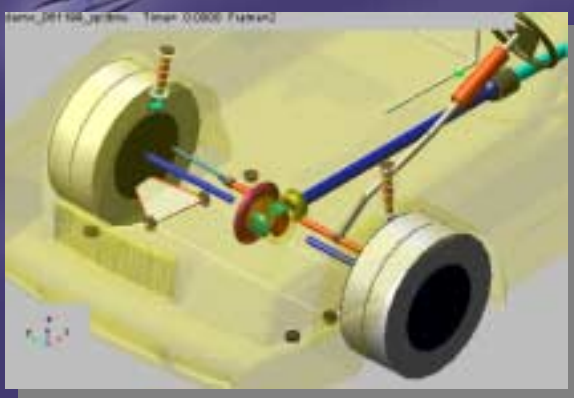
Leaf Spring



Twist Beam Susp



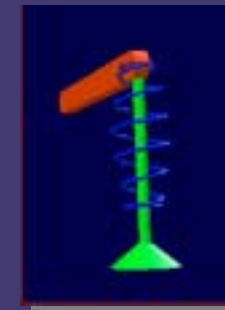
Suspension Coil



Flexible Driveshafts
Mechanical Dynamics



EngineCoil Springs



Flexible Camshaft

AutoFlex Development V13 and beyond



■ Meshing

◆ Using ParaSolid features

- For refining
- For attachments
- Re-Mesh for Stresses
- Spot welds between different FBs

■ Solve

- ◆ Extend Solution for Residual Vectors (distributed loads) and Pre-Loads
- ◆ Modal Stresses and Strains
- ◆ MAC
- ◆ Explore light non-linearities (load-stiffening)

■ I/O

- ◆ Import and output third part meshes

