Modelling of flexible members for simulation of vehicle dynamics
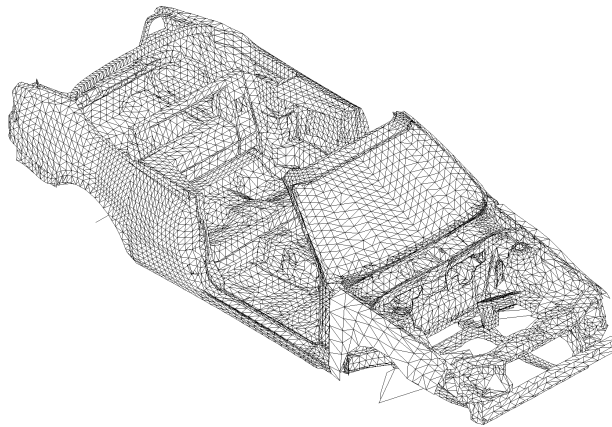
Report VALB-424

Tero Kiviniemi & Timo Holopainen

VTT
MANUFACTURING TECHNOLOGY

A	Work report	
B	Public report	X
C	Confidential report	

Title of report	
Modelling of flexible members for simulation of vehicle dynamics	

Client/sponsor of project and order	Report No.
TEKES research project LIIKKUVÄRE	VALB-424

Project	Project No.
LIIKKUMAL	V9SU00155

Author(s)	No. of pages/appendices
Kiviniemi, Tero & Holopainen, Timo	37

Keywords

vehicle dynamics, deformable structures, simulation models, component mode synthesis method

Summary

Advanced simulation models of vehicle dynamics consists of interconnected rigid and deformable structural members. The modelling of deformable structures as a part of simulation model can be carried out by different approaches. A relatively new approach is to employ component modes. In this approach, each deformable structural member is first modelled by finite element method with great accuracy. Next, the flexibility and inertia data of this structural member is translated to the current simulation software in the form of component modes and respective data. By this approach, the number of original degrees-of-freedoms of current structural component can be reduced effectively, and thus, in many applications, the numerical efficiency enhanced without any remarkable loss of accuracy.

In this report the general background for modelling deformable components is reviewed first. Especially, the component mode synthesis method and its modifications are considered. Secondly, the modelling alternatives offered by some of the simulation software products are reviewed. The software products included in this study are ADAMS, DADS, Simpack and Working Model Motion. The alternative possibilities to model deformable bodies by these software's are compared. Thirdly, future trends and some application examples are presented. Finally, some preliminary recommendations are given and several open questions are listed.

Date	Espoo 29.12.1999

Matti K. Hakala	Timo Holopainen	
Research Manager	Research Scientist	Checked

Distribution:

Project support group, 7 copies
HUT Laboratory of Automotive Engineering, 1 copy
VTT Manufacturing Technology, 6 copies, Archives 1 copy, Research Director 1 copy
www-pages of LiikkuVÄRE-project

VTT Manufacturing Technology	Tel. +358 9 4561
	Fax +358 9 456 5888
	E-mail: Timo.Holopainen@vtt.fi
	WWW:http://www.vtt.fi/manu/

Foreword

This report is made as a literature survey of a project named Design methods for vehicle vibration control (LIIKKUMAL). This project is a part of a TEKES-technology programme named Control of Vibration and Sound - VÄRE.

The fact that all authors' practical experience about simulation software is from using ADAMS –products leads automatically to few unintentional things. First, modelling elements discussed here are classified somehow. As it can be seen, this classification is derived from ADAMS. So we are not sure, if this is a general comparison of software's properties, or more or less comparing properties of other software to ADAMS. Second, when having long experience of one software and getting familiar with other software by reading few days of their manuals, there is very big difference in level of knowledge. For this reason other software may have some elements or element combinations, which can be use for certain purpose described here, but which are not mentioned.

The authors have written this report in close co-operation. However, Kiviniemi has been the main writer of Chapters 3 - 6 (excluding Section 3.4), and Holopainen in Chapter 2. Remaining chapters and sections have been written together. In addition to the authors, Ismo Vessonen, VTT Manufacturing Technology, has been closely involved with this project.

Espoo 22.12.1999,

Authors

Table of contents

1 Introduction

Advanced simulation models of vehicle dynamics consist of interconnected rigid and deformable structural members. Up to the present, the modelling of deformable structural members has been based mainly on discrete spring and mass elements. During the last few years a new approach has been introduced, and it has arrived already in some of the commercial simulation software's. This relatively new approach in the flexible multibody dynamics is based partly on the component mode synthesis method, which has a long history in structural dynamics starting on 1960's.

In this report we try to go through the different aspects of the modelling process. The purpose is to bring out the different choices, which will be encountered by modelling engineers. The authors feel that they can not give the right answers, but they hope that they can help to set up the right questions. In the text some references to the current and earlier literature are given in order to offer some starting points to go further.

Short introduction of five simulation software is presented. More detailed is discussed, what kind of possibilities and elements are available for modelling massless connecting components and flexible structures. Some major differences are tried to point out between software in this area.

The viewpoint of this report is of an engineer, who is modelling flexible members as structural component in order to construct an adequate simulation model. The goal of this report is to consider all aspects, which should be taken into account during the modelling process. However, the authors recognise the incapability to review as large a topic as the modelling of flexible members is. The level of the considerations will be very general and the experiences of the authors will direct the treatment of the matters.

Some of the questions we try to shed light on:
- How to model adequately one flexible structural component for simulation purposes?
- What is the most effective way to model a structural component?
- What are the inherent requirements to model a structural component?
- What are the requirements sets by the surrounding structural system?
- What are the requirements sets by the simulation process?
- How it is possible to balance between accuracy, stability and numerical efficiency?
- What kind of modelling elements are available for modelling massless connecting components?

The authors have very limited experiences of modelling flexible members as a part of simulation models. The experiences are gained by using ADAMS/Flex together with FE-software I-Deas. ADAMS/Flex™ is an add-on module to ADAMS/View that enables the use of flexible bodies in simulation models. I-Deas is one the FE-software's, that has a built-in interface to transfer the flexibility characteristics to ADAMS/Flex. However, the aim of the treatment will be on a more general level and the viewpoint will be above stand-alone software.

2   Flexible bodies in multibody dynamics

We start this chapter by making a classification of approaches for the dynamic analysis of mechanical systems. Then we introduce and compare some basic modelling alternatives for structural components. In this report the focus will be on the modelling of flexibility by employing, broadly speaking, the assumed modes method. This method is closely related to the component mode synthesis method, and in Section 2.3 a classification of variations of this method will be made. In subsequent sections some modelling aspects are reviewed including the damping, the gyroscopic effects and the joints.

2.1   Classification of dynamic analyses

Table 1 shows a set of different approaches for the dynamic analysis of mechanical systems. It can be mentioned that the table is not exhaustive, and that there are several other descriptive terms in use. Additional information concerning the classification of non-linear analyses can be found e.g. in Ref. [Bathe 1982, p. 302]. The idea of this table is to compare the assumptions related to the kinematics and stress-strain relations of different approaches. In this report the focus will be on the flexible multibody dynamics. This approach merges the linear structural dynamics of individual structural members and traditional rigid body dynamics of whole mechanical system.

Table 1. A classification of dynamical analyses in relation to flexible multibody dynamics.

	Type of analysis	Displacements and rotations	Strains	Stress-strain relation
1	Linear structural dynamics	infinitesimal	infinitesimal	linear
2	Materially-nonlinear structural dynamics	infinitesimal	infinitesimal	non-linear
3	Rigid multibody dynamics	large	neglected	-
4	*Flexible multibody dynamics*	*large*	*infinitesimal*	*linear*
5	Flexible multibody dynamics with large deformations	large	large	linear
6	Flexible multibody dynamics with large deformations and non-linear materials	large	large	non-linear

2.2   Modelling alternatives of structural components

During the modelling process of a flexible multibody system an analyst has a myriad of alternatives to be chosen. The same is true already in the modelling of individual structural members. A division of basic modelling approaches is shown in Table 2. The division is based on the modelling (or not) the flexibility and inertia properties. All the approaches shown in Table 2 are frequently used for structural members in structural and multibody dynamics. The simplest approach can be found in the upper left corner of the table and the most advanced in the lower right corner. The repertoire of approaches is motivated by a common statement: a gen-

eral objective in modelling is to use as simple model as possible, but as complex as is necessary [Eberhard et al. 1998].

Table 2. Basic modelling approaches for structural members.

		Flexibility of the member	
		no	yes
Inertia of	**no**	rigid link	massless flexible body
the member	**yes**	rigid body	flexible body

Next, a brief survey is given on the modelling approaches presented in Table 2.

2.2.1　Rigid link

A structural member can be idealised as a massless rigid link, if the stiffness of the structure is high compared both to the transmitted forces (and bending and torsion moments), and if the inertia force due the accelerations of the component has only minor effects on the system behaviour. In practice, relatively light and stiff structural members (compared to surrounding structural components) are most often modelled by rigid links. In post processing phase of the analysis the transmitted forces and moments are available but the stress distributions can not be evaluated without additional models.

Formally, a rigid link is a collection of points, lines and surfaces, which are kinematically fixed. In addition, as a result of dimensional reduction each point can have fixed rotational degrees-of-freedom. A rigid link reduces the number of degrees-of-freedom of the system by setting kinematic constraints between two or more bodies.

2.2.2　Rigid body

A structural member can be idealised as a rigid body. In this case the inertia properties of the component are taken into account but the flexibility of the member is ignored. The employment of rigid body approach is traditional and very common approach in the dynamics of multibody systems. In rigid body mechanics the relative motions between the components can be large, but the deformations of the components are ignored. In this approach the rigid bodies are connected by discrete, potentially flexible, coupling elements.

The rigid body approach is justified, if the elasticity of the component has only insignificant role in the dynamical behaviour of the system. This can be estimated (also in advance) by considering the magnitude and frequency content of transmitted forces and moments.

Formally, a rigid body is a collection of surface and inertia information. The inertia information of the body can be presented by the total mass, the centre of gravity, and the inertia tensor. In general case, a rigid body has six independent degrees-of-freedom.

2.2.3　Massless flexible body

A structural member can be idealised as an elastic body without inertia properties. In this case the structural behaviour of the component can be described as a generalised spring. This kind

of an approximation can be justified, if the inertia forces of the component have only minor effects on the system dynamics. In practice, this means that the rigid body accelerations of the component are insignificant (acceleration induced forces can be compared to other transmitted forces), and the natural frequencies of the component are distinctly higher than the frequency content of the transmitted forces.

The elastic behaviour of the component can be determined by the boundary value problem. This boundary value problem can be analysed by the FE-method. The total number-of-degrees of freedom depends on the discretisation, i.e., on the element type and mesh refinement. Because the system is linear the information of structural behaviour can be condensed into the interface degrees-of-freedom. Thus, the final number of degrees-of-freedom, which can be only a fraction of the original number, is determined by the discretisation of the interface surfaces. The same number of degrees-of-freedom is required later on to describe the behaviour of this structural component in a simulation model.

2.2.4　Flexible body

In this context, the most general description of a structural component includes the elastic and inertia properties. The refinement of the model and the number of degrees-of-freedom is dependent on the discretisation. The number of degrees-of-freedom can not be reduced by the same way as can be done with static substructures. The reason for this is the inner dynamic behaviour of the component. However, other reduction methods can be employed. One of these is the component mode synthesis method.

2.3　Classification of component mode synthesis methods

The component mode synthesis (CMS) method has been employed in numerous variations for structural dynamics from the early 1960s onwards. The abundant literature has been reviewed e.g. by Hintz [1975], Craig [1987], and Seshu [1997]. Originally CMS-methods were developed for the linear dynamic analysis of complex structures. Using this approach large numerical problems could be solved in smaller parts. During the last years the CMS-method has been employed to model flexible substructures as a part of simulations models for multibody dynamics.

In this section, the main purpose is to outline and classify different CMS-methods. The idea is to sort out different methods on grounds of the component mode sets they include. The main characteristics of this classification originate from Craig [1987].

2.3.1　Normal modes

The component modes of each substructure are divided into two main categories: the normal modes and the interface modes. The normal modes are used to represent the dynamic behaviour of the substructure. There are several possibilities to generate a normal mode set to be used in the analysis. The generated sets differ from each others by the applied boundary conditions. Table 3 shows a classification of normal mode sets. The most frequently applied

mode sets are the fixed-interface modes, e.g. in Craig-Bampton [1968] method, and free-interface modes, e.g. Craig-Chang [1977] or Rubin method [1975].

Table 3. A classification of normal mode sets [Craig 1987] and their translations into Finnish [Klinge 1999].

	Symb	Normal mode set	Translation into Finnish
1	Φ_{fixed}	Fixed-interface modes	kiinnitetyn liitoksen muodot
2	Φ_{free}	Free-interface modes	vapaan liitoksen muodot
3	Φ_{hybr}	Hybrid-interface modes	sekaliitoksen muodot
4	Φ_{load}	Loaded-interface modes	kuormitetun liitoksen muodot

Generally, all the normal modes could be used in the analysis. On the other hand, the total number of component modes determines the required computational resources. Thus, typically, only a chosen number of the lowest normal modes are used in the analysis. An often encountered practical problem will be, how to determine the adequate number of normal modes.

2.3.2   Interface modes

The interface modes add the contribution of the boundaries of the substructure. Table 4 shows a classification of various interface modes. The number of generated interface modes is dependent on the number of interface degrees-of-freedom. If a refined element mesh is used to model the interface of a flexible component, the number of generated interface modes can be very large. Therefore, advanced methods have been developed in order to reduce the number of interface modes employing eigensolution approach for interfaces [Bourquin & d'Hennezel 1992, Klinge 1998].

Table 4. A classification of interface mode sets [Craig 1987]. The constraint modes can be divided into the rigid-body modes and redundant constraint modes.

	Sym	Interface mode set	Translation into Finnish [Klinge 1999]
1	ψ_a	Attachment modes	joustomuodot
2		Constraint modes:	pakkomuodot
	ψ_r	- rigid-body modes	jäykän kappaleen muodot
	ψ_c	- redundant constraint modes	redundantit pakkomuodot
3	ψ_m	Inertia-relief modes	hitausvoimamuodot
4	ψ_b	Inertia-relief attachment modes	vapaan kappaleen joustomuodot
5	ψ_d	Residual attachment modes	jäännösjoustomuodot

Craig [1987] has defined three statically-complete mode supersets which are sufficient to determine exactly the quasi-static response of the component subjected to external forces. This quasi-static response includes the effect of d'Alembert forces due to the "rigid-body" response of a free component. These statically-complete mode supersets are equivalent in the meaning

that they span exactly the same kinematic subspace. The names of these supersets and the related subsets are shown in Table 5.

Table 5. The statically-complete mode supersets and their modal content [Craig 1987].

Name of the superset	Sym.	Consisting of the modes (see Table 4)					
		ψ	ψ	ψ	ψ_d	ψ	ψ_r
Attachment mode superset	ψ^a	X				X	X
Inertia-relief attachment mode superset	ψ^b		X				X
Constraint mode superset	ψ^c			X		X	X
Residual attachment mode superset	ψ^d				X		X

2.3.3   Dynamic component mode supersets

The dynamic component mode supersets are acquired by adding a set of normal modes to a statically-complete mode superset. Table 6 shows a list of combinations, which were tested during the early years of CMS-method development [Craig 1987].

Table 6. A dynamic component mode superset is obtained by combining a normal mode superset and an interface mode superset. The references given in the cells of the table stand for an application of each combination.

		Dynamic mode set	
		Φ_{fixed}	Φ_{free}
Interface mode set	ψ^a	Hintz 1975	Hintz 1975
	ψ^c	Hurty 1965 *, Hintz 1975, Craig & Bampton 1968 *	Herting 1979
	ψ^d		MacNeal 1971, Rubin 1975, Craig & Chang 1977

* the inertia-relief modes are not included

2.4   Inertia coupling of deformable bodies

The flexibility properties, i.e. the stiffness matrix, of a deformable body are independent of the body motion. Thus, the translation of stiffness properties from a structural FE-analysis to a flexible multibody simulation is relatively straightforward.

The modelling and translation of inertia properties is more complicated. This is a result of the inertia coupling between the reference motion and the elastic deformation of the body [Shabana 1987, p. 203 - 216]. However, it is possible to derive the inertia terms for deformable bodies that undergo large reference rotations. The general mass matrix of a deformable body can be written in the form [Shabana 1998, p. 204]

$$\mathbf{M} = \begin{bmatrix} \mathbf{m}_{RR} & \mathbf{m}_{R\theta} & \mathbf{m}_{Rf} \\ & \mathbf{m}_{\theta\theta} & \mathbf{m}_{\theta f} \\ sym. & & \mathbf{m}_{ff} \end{bmatrix}$$

where the submatrices are defined as [Shabana 1998, p. 204]

$$\mathbf{m}_{RR} = \int_V \rho \mathbf{I} dV$$

$$\mathbf{m}_{R\theta} = \int_V \rho \mathbf{B} dV$$

$$\mathbf{m}_{Rf} = \mathbf{A} \int_V \rho \mathbf{S} dV$$

$$\mathbf{m}_{\theta\theta} = \int_V \rho \mathbf{B}^{\mathrm{T}} \mathbf{B} dV$$

$$\mathbf{m}_{\theta f} = \int_V \rho \mathbf{B}^{\mathrm{T}} \mathbf{A} \mathbf{S} dV$$

$$\mathbf{m}_{ff} = \int_V \rho \mathbf{S}^{\mathrm{T}} \mathbf{S} dV$$

where ρ is the density, $\mathbf{I}$ is the identity matrix, $\mathbf{A}$ is the transformation matrix between the body reference and the global co-ordinate system, $\mathbf{B}$ is the transformation matrix associated with the differentiation of the transformation matrix $\mathbf{A}$ with respect to time, and $\mathbf{S}$ is the matrix consisting of assumed shape functions.

Some of the characteristic properties of the inertia coupling can be mentioned. Firstly, if the body is rigid, elastic deformations vanish and the submatrices $\mathbf{m}_{Rf}$, $\mathbf{m}_{\theta f}$ and $\mathbf{m}_{ff}$ are null matrices. In this case the inertia of the rigid body is represented by submatrices $\mathbf{m}_{RR}$, $\mathbf{m}_{R\theta}$ and $\mathbf{m}_{\theta\theta}$. Secondly, if the reference motion is not allowed, the only remaining submatrix is $\mathbf{m}_{ff}$. This matrix is equivalent to the generalised mass matrix used in the linear structural analysis. Thirdly, when a deformable body undergoes large rotational motions, the non-linear coupling terms between the reference motion and elastic deformation, represented by matrices $\mathbf{m}_{f}$ and $\mathbf{m}_{\theta f}$, are required.

From the computational point of view the updating of the mass matrix has a significant effect on the simulation time. It can be noticed that the submatrices $\mathbf{m}_{RR}$ and $\mathbf{m}_{ff}$ are constant matrices and they can be calculated in advance. The other submatrices depend on the orientation and motion of the body reference, which is represented in the equations by matrices $\mathbf{A}$ and $\mathbf{B}$. It is advantageous to calculate the terms of mass matrix as far as possible in advance. This can be accomplished by using the inertia shape integrals. The inertia shape integrals of a deformable body are [Shabana 1998, p. 231]

$$\mathbf{I}_1 = \int_V \rho \bar{\mathbf{u}}_0 dV$$
$$I_{kl} = \int_V \rho x_k x_l dV, \quad k,l = 1,2,3$$

$$\overline{\mathbf{S}} = \int_V \rho \mathbf{S} dV$$

$$\overline{\mathbf{S}}_{kl} = \int_V \rho \mathbf{S}_k^T \mathbf{S}_l dV, \quad k,l = 1,2,3$$

$$\overline{\mathbf{I}}_{kl} = \int_V \rho x_k \mathbf{S}_l dV, \quad k,l = 1,2,3$$

where $\overline{\mathbf{u}}_0$ is the position vector of an arbitrary point of the undeformed body, and $\mathbf{S}_k$ is k th row in the matrix of assumed shape functions $\mathbf{S}$.

In a general computational algorithm these inertia shape integrals can be calculated in advance in pre-processing phase. Particularly, these calculations can be done for complicated geometries using the FE-method.

The inertia shape functions are as integrals of continuous functions. This results in consistent mass matrix. Another formulation is based on discrete masses in selected grid points. This results in lumped mass matrix. In the lumped mass formulation the integral equations are substituted by the finite series. Shabana [1998] presents both consistent and lumped mass formulation.

2.5   Gyroscopic effects

The equations of motion for a deformable body undergoing large motions can be written using the generalised co-ordinates [Shabana 1998, p. 227]. These equations include the gyroscopic and Coriolis force components. A numerical algorithm for the analysis of flexible multibody systems can be derived including these terms [Shabana 1998, p. 265]. Thus, the gyroscopic and other effects due to rigid body accelerations can be included into the analysis together with modal approach. A short review of component mode synthesis methods applied for gyroscopic systems is included in Ref. [Seshu 1997].

Generally, the gyroscopic effect of the wheels and other rotating parts generates gyroscopic moments, which depend on the inertia properties and velocity terms. In steady-state motion gyroscopic moments are generally small. However, gyroscopic moments of the wheels can be significant in non-stationary conditions [Genta 1997, p. 426].

2.6   Damping

Damping, i.e. the dissipation of energy, is present in all mechanical systems. Usually, it is assumed, that the damping of mechanical systems is connected to velocity dependent terms. The different damping mechanisms can be modelled either by discrete dampers or by distributed damping properties. A discrete damper with non-linear velocity dependent parameters is a natural part of a simulation model, which is inherently based on the non-linear equations of motion. All the same, a discrete damper is a structural component as such, and not a property of a flexible structural member.

Distributed damping of a flexible member is more difficult to take into account in system models. Originally the component mode synthesis method was developed for lightly damped systems. In that approach the damping is initially neglected and the normal modes are deter-

mined for undamped substructures. This is advantageous and leads to symmetric coefficient matrices. Later on, the damping is taken into account as assumed modal damping.

During the subsequent years, more general approaches for modelling of damping were developed. These approaches were based on the employment of complex component modes. A list of current literature can be found in Ref. [Seshu 1997]. Wang and Kirkhope [1995] compared free-interface real (= traditional) and complex CMS method using as an example damped rotors. The conclusion was that sufficiently accurate solutions can be obtained by real CMS method when damping ratios of about 70 % or less are present. Otherwise, they recommended the use of complex CMS method.

In addition to model actual dissipation of energy, the damping characteristics of a flexible body can be used to enhance the efficiency of numerical solution. Certain numerical difficulties arise if the simulation model includes high frequency components. The flexible bodies can contribute a large amount of high frequency components. A solution is to use larger damping characteristics for higher natural modes. As an example, in ADAMS/Flex, the default values for damping ratio are [Anon. 1998]:

- 0.01 for all modes with frequency lower than 100 Hz
- 0.10 for modes with frequency in the 100 - 1000 Hz range
- 1.00 for modes with frequency above 1000 Hz

This approach eliminates the oscillating components with high frequency content during the time integration. However, this approach is a numerical trick and there is no relation to physical dissipation of energy.

2.7   Joints and contacts

Structural members of a mechanical system are connected to each others by different connections or joints. Some of the connections are rigid and others offer possibility to relative translational and rotational displacements between the system components. The mechanical joints such as revolute, spherical and cylindrical joints are most important in flexible multibody dynamics.

We consider an arbitrary joint, which enable two connected system components to undergo a large relative motion. Actually, the contact surface between these two bodies has a finite area. The modelling of two contact surfaces, which interact and have a relative motion is a very hard task. Thus, simplified approaches are required in dynamics of multibody systems. Usually, in practical analyses the contact is modelled by an idealised joint represented by a contact point and a set of joint variables. These joint variables represent relative translational and rotational displacements between the components.

In addition, there is another reason for modelling the joints as simply as possible. Using the CMS methods the connecting interfaces of structural components has a special effect on the problem size. The number of static modes is usually dependent on the number of nodes on the interface boundary. If the area of the interface is relatively large and the element mesh is refined, the number of static modes is large. This problem of model size is emphasised when solid FE-models are employed.

It can be concluded, that the pointwise models of mechanical joints is a traditional and straightforward approach leading to a relatively simple numerical model. The modelling of joints more accurately taking into account the contact area is a hard task and the solution requires a lot of numerical resources.

3   Software products

3.1   Software included in this report

3.1.1   ADAMS

ADAMS (**A**utomatic **D**ynamic **A**nalysis of **M**echanical **S**ystems) is developed and distributed by Mechanical Dynamics Inc.

The software consists of several different modules. Two main modules are ADAMS/View and ADAMS/Solver. View is an interactive and graphical pre- and post-processor and Solver is a calculation engine. Basic modelling and analysis can be done with these modules. Additional modules are listed in Table 7.

Table 7. Additional modules of ADAMS.

Module	Description of properties
ADAMS/Linear	The equations of motion formulated by ADAMS can be simplified to a linear form by this module to more conveniently assess the stability and vibration behaviour of mechanical system model.
ADAMS/Controls	Module allows users to combine their block diagram models from MATLAB/Simulink, MATRIXx/SystemBuild or EASY5 with their ADAMS models.
ADAMS/Flex	Module allows flexible bodies to be created within ADAMS models from modal frequency data.
ADAMS/Hydraulics	Module helps user smoothly integrate system-level motion simulation and hydraulic system design. Hydraulic circuits are presented with block-diagrams.
ADAMS/Tire	Engineers can use module to model forces and torques acting on a tire moving down a roadway or over irregular terrain. This includes vertical, longitudinal, and lateral forces as well as aligning, overturning, and rolling resistance torques at the tire hub.
ADAMS/Exchange	Module automates the exchange of geometry data between ADAMS and CAD systems supporting any of variety of standards including IGES, DXF, STEP, VDAFS and STL.
ADAMS/Car	With this module automotive engineers build computer models of entire vehicles, complete with suspensions, powertrains, engines, steering mechanisms, anti-lock braking systems and other complex assemblies.
ADAMS/Driver	Control actions of a vehicle driver such as steering, braking, accelerating, gear shifting and clutching for a variety of manoeuvres can be simulated with this module.
ADAMS/Engine	Module offers specialised utilities for valve train design and testing timing mechanisms, belts, cranks and pistons.

ADAMS/Pre	Module provides automakers and their suppliers with a suite of specialised pre-processing capabilities to quickly create a complete computer model of a full-vehicle or vehicle-subsystem design.
ADAMS/Rail	With this module user can quickly and easily build complete, parameterised rail-way vehicle models, defining suspensions, wheelsets, rail-wheel contact and other vital characteristics.
ADAMS/Animation	This module allows rapid generation of high-quality moving images to visualise the behaviour of mechanical systems. Design presentations can be enhanced with features such as multiple light sources and camera angles, texture mapping and translucency.
CAT/ADAMS MECHANISM/Pro	Interfaces between ADAMS and CAE-software.

All information of ADAMS-software presented in this document is based on its online documentation mentioned in the reference list. More information about this software can be found from the internet-address http://www.adams.com.

3.1.2   DADS

DADS (**D**ynamic **A**nalysis and **D**esign **S**ystems) is developed and distributed by LMS CADSI, which is the Mechanical Simulation Technologies Division of LMS International.

DADS-package has three starting levels and some additional modules. The lowest level DADS/Basic consists of DADS Model, graphical pre-processor, DADS Analysis, calculation engine, and DADS Graph, post-processor. Next level, DADS/Standard, includes also module M1 (user defined forces and subroutines and contact elements) and module M2 (control and hydraulic elements). Highest level, DADS/Advanced, includes also module M3, DADS/Flex. Additional modules are listed in Table 8.

Table 8. Additional modules of DADS.

Module	**Description of properties**
DADS/Plant	Integration of DADS with Control System Simulation Software (MA-TRIXx/Systembuild, MATLAB/Simulink, EASY5)
DADS/Engine	Module enables the dynamic simulation of engines and powertrain performance.
DADS/DOX (Design of Experiment)	Simultaneous variation of multiple design parameters enabling trade-off studies and design optimization.
Advanced Tire Element	Includes the Pacejka "Magic Formula" to integrate tire test data into DADS using a complex curve fitting formulation.
DADS/Linear	Reports system eigenvalues and animates system rigid body modes. Coefficient matrixes are available for export to external control system packages.
DADS/Track	Simulates road wheels on a tracked vehicle. The track, wheel, soil interaction is modelled in detail to predict how the vehicle will respond.
CAT/DADS DADS/IMS DADS/Pro	Embedded in CATIA Embedded in I-DEAS Embedded in Pro/Engineer

All information of DADS-software presented in this document is based on its online documentation mentioned in the reference list. More information about software can be found from the internet-address http://www.cadsi.com.

3.1.3   Simpack

Simpack (**Sim**ulation **Pack**age) is developed by the department of Vehicle System Dynamics at the Deutsches Zentrum für Luft-und Raumfahrt, DLR. Software is distributed by Intec GmbH.

Simpack - Kinematics & Dynamics contains the basic module for kinematics and dynamics including the complete scope of modelling and the graphic user interface with pre- and post-processors. Additional modules are listed in Table 9.

Table 9. Additional modules of Simpack.

Module	Description of properties
Simpack – Wheel/Rail	Module to set up complete railway vehicle models which includes non-linear description of the contact of wheel and rail, the definition of the lining and the irregularities of the track.
Simpack – Automotive	Module to set up complete road vehicles which includes the non-linear description of the contact of the tyres and the road.
Simpack – Control	Module for setting up the control loops and the controllers, which may be analogue and/or discrete.
Simpack – User	A Fortran library of user modules, which enables user to introduce their own programming code of specific user items, like force elements, sensors, controllers and actuators.
Simpack – Symbolic code	Module provides the generation and the export of the real time programming code of the dynamic model to another simulation environment e.g. with hardware in the loop.
FEMBS	Interface between FEA program packages and Simpack for modal description of the elastic bodies.
SIMAT SIMAX	Interface between MATLAB/Simulink and Simpack. Interface between MATRIXx/Systembuild and Simpack.
ProSIM CatSIM	Interface between Pro/Engineer and Simpack. Interface between CATIA and Simpack.

All information of Simpack-software presented in this document is based on its online documentation mentioned in the reference list. More information about software can be found from the internet-address http://www.simpack.de.

3.1.4   Working Model Motion

Working Model Motion is developed and distributed by MSC.Working Knowledge, which is a division of MSC.Software Corporation.

Working Model Motion seems to have less elements than the other software included in this report. User can import geometry for his model from leading CAD-packages. Kinematic capabilities are quite similar to the other software. Kinematic analysis and collision detection can be performed. Dynamic capabilities and force elements needed for dynamic modelling are limited compared to the other software. These limitations have a positive effect on price of the software. This software is much cheaper than the other ones. That makes it a potential choice for some cases.

All information of this software presented in this document is based on its online documentation mentioned in reference list. More information about this software can be found from the internet-address http://www.workingmodel.com.

3.1.5   eta/VPG - Virtual Proving Ground

eta/VPG is developed and distributed by Engineering Technology Associates, Inc. (ETA)

eta/VPG is a CAE software package for simulation of vehicle dynamics and structural response to proving ground testing events.

This software would have been very interesting for this report, because it has different background when compared to the other software products reviewed here. VPG is built on the finite element program LS Dyna. Because of that, it can be used to model the flexible components of a system using beam, shell and solid elements. It offers the choice to define material properties for these elements, allowing them to represent the actual component flexibilty (or they may be defined as rigid material). The material characteristics may be defined by linear or nonlinear properties, and it can capture the material and geometric nonlinearity of the problems. In this case, there has been added a capability to model mechanical systems with special elements.

Unfortunately, the requested detailed information of this software was never received. So it was impossible to find out the properties of this software.

More information about this software can be found from the internet-address http://www.eta.com.

3.2   Modelling of flexible connecting components (massless)

3.2.1   Spring-Damper

Spring-Damper–element is typically used to model springs and dampers. It can describe at the same time both the spring and damping characteristics. Spring-Damper-element can be used for modelling also other kind of components in appropriate cases.

Spring-damper is defined by selecting a point from two bodies. The element is generated between these two points using the stiffness and damping characteristics. The force, which is acting between these two points, depends on the relative displacement and velocity of these two points.

The simplified mathematical formulation of a non-linear spring-damper can be

$$F = -k \cdot x^a - c \cdot v^b$$

In the linear case the exponents a and b have the value of 1. The other method, for the nonlinear case, is to use tabular data in the form of xy-points and interpolate between those

points. Experimental values measured from the actual structural components can be easily imported into the model by this method. It can take into account the typical behaviour of real dampers; force generated at a certain velocity is different in compression and tension modes.

In ADAMS both the stiffness and damping characteristics can be linear or non-linear. In non-linear case, the dependencies are defined by the tabular data. Non-linear mathematical equation can not be used in this case. Initial conditions are defined with preload and length of spring at that preload.

In DADS spring-damper element is called TSDA, Translational Spring Damper Actuator. Spring-damper term of the element can be linear or non-linear. In non-linear case the dependencies are defined by a curve (tabular data). This element can have an actuator term. That can be either linear or non-linear. The linear actuator term means constant force value, so it is similar to the preload term of ADAMS. The non-linear actuator means, that the preload term can be time dependent.

In Simpack there are separate elements for a linear spring (FE-1), linear damper (FE-2), linear or non-linear spring-damper in parallel (FE-4), and linear or non-linear spring-damper in series (FE-6). The non-linear characteristics are defined with the input table.

In Working Model Motion the stiffness characteristics can be linear or one of the following non-linear cases: $-kx^3$, $-kx^2$, k/x or $-k/x^2$. The damping characteristics have similar options.

3.2.2   Force acting between two points

The Spring-damper-element, described above, can be said to be a special case of Force acting between two points -element. It is pre-formatted so that the user needs only to give some coefficients or curves to describe the dependencies.

In ADAMS there is a general force element, which is acting between two points. With this element the user can freely form up a mathematical equation to describe a force and from what characteristics it is dependent on. The magnitude of this force component can depend on the value of any system variable, like time, displacement, velocity, acceleration and force of other force element or joint. There is also a possibility to link a user written subroutine for defining the value of a force.

In DADS, Simpack and Working Model Motion there are not this kind of Force acting between two points -element.

3.2.3   Force acting at point

If there is not any suitable element for the modelling of flexible mounting components, which would fulfil all the requirements, usually, there is a possibility to use some kind of a general force. There can be several variations of general force; it can have three force components (force vector), three torque components (torque vector) or three force and three torque components (general force vector). The behaviour of this kind of a force element is defined with the mathematical equations of each component separately. There can also be a possibility to use some user-written subroutines.

In ADAMS there are elements for a force vector, a torque vector, and a general force vector . Both the mathematical equation and the user-defined subroutines can be used when this kind of elements are employed. The magnitude of a force component can depend on the value of any system elements, like time, displacement, velocity, acceleration or force of other force element or joint.

In DADS there is an element, which is called Three-point force. This force has three components. This element can act as a force or as a torque. The magnitude of this force can be defined with a constant value, or a time-varying curve. During the writing of this report a new version, numbered 9.5, of DADS was introduced. In the brochure of Key Enhancements it is mentioned two new elements, Expression Load and Expression Element. With use of them, it is possible to model a custom force or torque so that the magnitude is defined with mathematical formula that incorporates system parameters and results (position, velocity, acceleration) to compute a scalar value. In DADS there is also an element called User-Defined Force Element. It can be used if the other elements do not fulfil the requirements. Unfortunately, in the manuals available during the writing of this report there were not any detailed description of this element, how it is defined and how it can be used.

In Simpack there is Time excitation by u(t)-vector for forces and torques with given direction of action (FE-93) and Time excitation by u(t)-vector for forces and torques acting along the connection line between the coupling points (FE-94). The magnitude of force and torque in these elements can be defined with a function of time.

In Working Model Motion there is not any of these general force elements.

3.2.4 Bushing

Bushing element is typically used to model elastic mounting components like rubber bushings. A bushing-element is a six degree of freedom force element. With a bushing element two bodies can be connected together. A bushing is defined by selecting a point from both bodies and the force-torque components are calculated in tree axis co-ordinate system. Generally, the bushing characteristics are defined by six stiffness coefficients and six damping coefficients and with the optional preload terms. The definition of initial conditions can be done also with the initial displacements instead of the preload terms.

The general mathematical formulation of the bushing element is

$$
\begin{bmatrix} F_x \\ F_y \\ F_z \\ T_x \\ T_y \\ T_z \end{bmatrix} = - \begin{bmatrix} k_{11} & 0 & 0 & 0 & 0 & 0 \\ 0 & k_{22} & 0 & 0 & 0 & 0 \\ 0 & 0 & k_{33} & 0 & 0 & 0 \\ 0 & 0 & 0 & k_{44} & 0 & 0 \\ 0 & 0 & 0 & 0 & k_{55} & 0 \\ 0 & 0 & 0 & 0 & 0 & k_{66} \end{bmatrix} \cdot \begin{bmatrix} x \\ y \\ z \\ \alpha \\ \beta \\ \chi \end{bmatrix} - \begin{bmatrix} c_{11} & 0 & 0 & 0 & 0 & 0 \\ 0 & c_{22} & 0 & 0 & 0 & 0 \\ 0 & 0 & c_{33} & 0 & 0 & 0 \\ 0 & 0 & 0 & c_{44} & 0 & 0 \\ 0 & 0 & 0 & 0 & c_{55} & 0 \\ 0 & 0 & 0 & 0 & 0 & c_{66} \end{bmatrix} \cdot \begin{bmatrix} \dot{x} \\ \dot{y} \\ \dot{z} \\ \dot{\alpha} \\ \dot{\beta} \\ \dot{\chi} \end{bmatrix} + \begin{bmatrix} F_{x0} \\ F_{y0} \\ F_{z0} \\ T_{x0} \\ T_{y0} \\ T_{z0} \end{bmatrix}
$$

In ADAMS the bushing element is linear. The stiffness and damping characteristics are defined by a set of constant coefficients. The equilibrium situation is defined with the preload terms. There are some limitations in rotational displacement. Accurate result is achieved only

if α and β are under 10 degrees. If α or β is more than 90 degrees, the other one of those comes erratic. Only χ can be more than 90 degrees without causing convergence problems.

In DADS both stiffness and damping characteristics of bushing can be linear or non-linear. In non-linear case the dependencies are defined with force-displacement or force-velocity curves instead of constant coefficients in the mathematical equation. The bushing element is assumed to be axisymmetric. The stiffness and damping are defined by the axial and radial terms. Mathematically this means that $k_{11}=k_{22}$, $k_{44}=k_{55}$, $c_{11}=c_{22}$ and $c_{44}=c_{55}$. DADS have two sub-types of bushing elements. The standard bushing element assumes that in equilibrium situation the two points of the respective two bodies are coincident. The so-called bushing link assumes that there is a free length between attachment points. This is defined with free length along x-direction instead of preload term. Limitations for rotations are more restrictive in DADS than in ADAMS. It is mentioned that α and β must be small. What is small in this case, it is not mentioned as an angular value. Only χ can be more, but it has to still be less than 90 degrees.

In Simpack there are two elements such as bushing; Compact spring element with damping, non- linear characteristics and clearance (spatial rubber) (FE-5) and Spherical Linear Spring and Damper Parallel (FE-41). With element FE-5 both the stiffness and damping characteristics can be linear or non-linear. The non-linear characteristics are defined with tabular data. The stiffness and damping are defined separately in three directions. This element is only a force element and it does not include torque terms. So, mathematically the stiffness and damping are defined with 3x3 matrixes. Element (FE-41) uses 6x6 matrixes. So, with this element also torque can be applied. This element has only linear characteristics.

In Working Model Motion both stiffness and damping characteristics of bushing can be linear or non-linear with some limitations. In non-linear case for stiffness dependencies can be formulated as $-kx^2$ or $-kx^3$. The damping characteristics have similar options. The initial conditions can not be defined separately. There are not any remarks of limitations of the bushing element in the users manual.

3.2.5 Beam

Beam-element can be used to describe the flexibility of a beam structure. It can be used when the structure has a constant area of cross-section like in Figure 1.
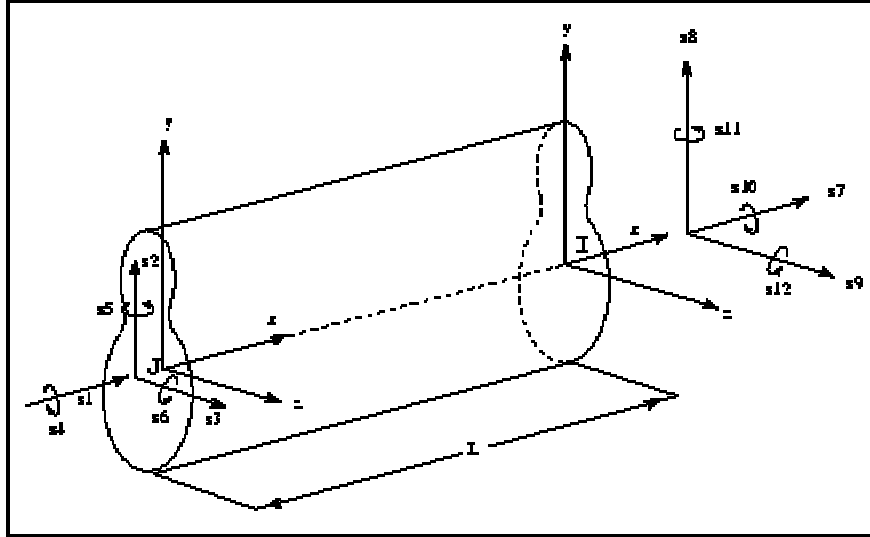
Figure 1. Definition of beam. [Anon. 1998, ADAMS/View documentation]

A beam element creates the linear translational and rotational forces between two locations that define the endpoints of the beam. The forces produced by the beam are dependent on the relative displacements and velocities of the endpoints of the beams.

The general mathematical formulation of beam element is analogous to those in the finite element method.

$$
\begin{bmatrix} F_x \\ F_y \\ F_z \\ T_x \\ T_y \\ T_z \end{bmatrix} = - \begin{bmatrix} \dfrac{EA}{L} & 0 & 0 & 0 & 0 & 0 \\ 0 & \dfrac{12EI_{zz}}{L^3} & 0 & 0 & 0 & \dfrac{-6EI_{zz}}{L^2} \\ 0 & 0 & \dfrac{12EI_{yy}}{L^3} & 0 & \dfrac{6EI_{yy}}{L^2} & 0 \\ 0 & 0 & 0 & \dfrac{GI_{xx}}{L} & 0 & 0 \\ 0 & 0 & \dfrac{6EI_{yy}}{L^2} & 0 & \dfrac{4EI_{yy}}{L} & 0 \\ 0 & \dfrac{-6EI_{zz}}{L^2} & 0 & 0 & 0 & \dfrac{4EI_{zz}}{L} \end{bmatrix} \cdot \begin{bmatrix} x-L \\ y \\ z \\ \alpha \\ \beta \\ \chi \end{bmatrix} - \begin{bmatrix} c_{11} & c_{12} & c_{13} & c_{14} & c_{15} & c_{16} \\ c_{21} & c_{22} & c_{23} & c_{24} & c_{25} & c_{26} \\ c_{31} & c_{32} & c_{33} & c_{34} & c_{35} & c_{36} \\ c_{41} & c_{42} & c_{43} & c_{44} & c_{45} & c_{46} \\ c_{51} & c_{52} & c_{53} & c_{54} & c_{55} & c_{56} \\ c_{61} & c_{62} & c_{63} & c_{64} & c_{65} & c_{66} \end{bmatrix} \cdot \begin{bmatrix} \dot{x} \\ \dot{y} \\ \dot{z} \\ \dot{\alpha} \\ \dot{\beta} \\ \dot{\chi} \end{bmatrix}
$$

In ADAMS the beam is defined by giving some material and geometric parameters of structure. The material parameters needed are the Young's modulus and the shear modulus. The geometric parameters are the length of the beam element, the area of cross-section, the area moments of inertia I_{xx}, I_{yy} and I_{zz}, Y shear area ratio and Z shear area ratio. The Y and Z shear area ratios are needed, because ADAMS uses the theory of Timoshenko beams. So, in the stiffness matrix there is some correction terms included, like below

$$\begin{bmatrix} F_x \\ F_y \\ F_z \\ T_x \\ T_y \\ T_z \end{bmatrix} = - \begin{bmatrix} \dfrac{EA}{L} & 0 & 0 & 0 & 0 & 0 \\[2mm] 0 & \dfrac{12EI_{zz}}{L^3(1+P_y)} & 0 & 0 & 0 & \dfrac{-6EI_{zz}}{L^2(1+P_y)} \\[2mm] 0 & 0 & \dfrac{12EI_{yy}}{L^3(1+P_z)} & 0 & \dfrac{6EI_{yy}}{L^2(1+P_z)} & 0 \\[2mm] 0 & 0 & 0 & \dfrac{GI_{xx}}{L} & 0 & 0 \\[2mm] 0 & 0 & \dfrac{6EI_{yy}}{L^2(1+P_z)} & 0 & \dfrac{(4+P_z)EI_{yy}}{L(1+P_z)} & 0 \\[2mm] 0 & \dfrac{-6EI_{zz}}{L^2(1+P_y)} & 0 & 0 & 0 & \dfrac{(4+P_y)EI_{zz}}{L(1+P_y)} \end{bmatrix} \cdot \begin{bmatrix} x-L \\ y \\ z \\ \alpha \\ \beta \\ \chi \end{bmatrix}$$

`P`$_y$ `= 12 E I`$_{zz}$ `ASY/(G A L`^{2}`)`
`P`$_z$ `= 12 E I`$_{yy}$ `ASZ/(G A L`^{2}`)`
`ASY` = Correction coefficient (shear area ratio) for shear deflection in the y direction for Timoshenko beams.
`ASZ` = Shear area ratio for shear deflection in the z direction for Timoshenko beams.

In ADAMS the damping of the beam-element can be defined with a damping ratio or a damping matrix. When using the damping ratio the stiffness matrix is multiplied by the value entered for the damping ratio to obtain the damping matrix. When using the damping matrix user gives damping matrix coefficients. The damping matrix is symmetrical, so only one-half of the matrix needs to be specified. The range of linearity is not defined specifically. It is said that when the beam element angular deflections are small, the stiffness matrix provides a meaningful description of the beam behaviour. If translational displacements exceed 10 percent of the original length, warning is issued during the calculation.

In DADS the modelling of beam element with geometrical and material properties is based on the simplified beam theory and the shear deformation is ignored. So the mathematical presentation of the beam element is like in the general formulation. If the shear deformation is wanted to be taken into account, then the stiffness characteristics must be defined with a 6x6 matrix. In DADS the damping can be defined with damping ratio or damping matrix. If damping ratio is specified, applied damping is calculated with equation

$$D = 2(zeta)\sqrt{M_{diag} \cdot K_{diag}}$$

where D =   the applied proportional damping used during the simulation.
             zeta   =   the damping ratio
             M =   the diagonal mass matrix (from body with greater mass)
             K =   the diagonal stiffness matrix

When user gives the damping matrix coefficients, matrix has to be given in full form with 6x6 terms. About the range of linearity it is said that the angular deflections should be less than 10°.

In standard Simpack there is not a separate beam element. Because the modelling of a beam is based on the use of general 6x6 stiffness and damping matrixes, a Simpack element (FE-41) can be used for this purpose too. Instead of defining the geometric and material properties, user has to calculate all terms of the matrix by himself. There is more information of this element in Chapter 3.2.4.

There is available a separate module called BEAM for Simpack. With that module an arbitrary beam structure can be modelled. The module will calculate the mode shapes for the structure. These mode shapes are transferred to the simulation model. So at that stage the element is not massless any more.

In Working Model Motion there is not any beam element available.

3.2.6 Force element defined by 6x6 matrix

This force element is similar to the beam element discussed above. It is a general method to define the force components between two points with a 6x6 matrix. The force element can be used instead of a beam element in order to define a beam with a non-uniform cross section or a beam with non-linear material characteristics.

The general mathematical formulation of a force element is analogous to those used in the finite element method.

$$
\begin{bmatrix} F_x \\ F_y \\ F_z \\ T_x \\ T_y \\ T_z \end{bmatrix} = -\begin{bmatrix} k_{11} & k_{12} & k_{13} & k_{14} & k_{15} & k_{16} \\ k_{21} & k_{22} & k_{23} & k_{24} & k_{25} & k_{26} \\ k_{31} & k_{32} & k_{33} & k_{34} & k_{35} & k_{36} \\ k_{41} & k_{42} & k_{43} & k_{44} & k_{45} & k_{46} \\ k_{51} & k_{52} & k_{53} & k_{54} & k_{55} & k_{56} \\ k_{61} & k_{62} & k_{63} & k_{64} & k_{65} & k_{66} \end{bmatrix} \cdot \begin{bmatrix} x - x_0 \\ y - y_0 \\ z - z_0 \\ \alpha - \alpha_0 \\ \beta - \beta_0 \\ \chi - \chi_0 \end{bmatrix} - \begin{bmatrix} c_{11} & c_{12} & c_{13} & c_{14} & c_{15} & c_{16} \\ c_{21} & c_{22} & c_{23} & c_{24} & c_{25} & c_{26} \\ c_{31} & c_{32} & c_{33} & c_{34} & c_{35} & c_{36} \\ c_{41} & c_{42} & c_{43} & c_{44} & c_{45} & c_{46} \\ c_{51} & c_{52} & c_{53} & c_{54} & c_{55} & c_{56} \\ c_{61} & c_{62} & c_{63} & c_{64} & c_{65} & c_{66} \end{bmatrix} \cdot \begin{bmatrix} \dot{x} \\ \dot{y} \\ \dot{z} \\ \dot{\alpha} \\ \dot{\beta} \\ \dot{\chi} \end{bmatrix} + \begin{bmatrix} F_{x0} \\ F_{y0} \\ F_{z0} \\ T_{x0} \\ T_{y0} \\ T_{z0} \end{bmatrix}
$$

In ADAMS this element has the name field. It can be either linear or non-linear. In the linear case the characteristics are defined with a 6x6 stiffness matrix, translational and rotational preload values, and a 6x6 damping matrix. The stiffness and damping matrices must be positive semidefinite, but need not be symmetric. When defining the damping properties also a damping ratio can be used instead of a damping matrix. The non-linear field element is defined with a user-written subroutine.

In DADS, with beam element, there is a possibility to define the 6x6 stiffness and 6x6 damping matrixes by the user.

In Simpack element FE-41 can be used for this purpose. With it user can define the 6x6 stiffness and damping matrixes. There is more information of this element in Chapter 3.2.4.

In Working Model Motion there is not any element like field available.

3.2.7   Other elements

When we are interested about modelling flexible connecting components, there are two very interesting elements in Simpack. They are Dynamic force element (spring, damper) dependent on frequency response (FE-102) and Non-linear spring damper parallel with hysteresis (FE-104).

Dynamic force element dependent on frequency response (FE-102) can be either a spring or a damper. The stiffness coefficient, or respectively the damping coefficient, is represented by a dynamic weighting function, a filter of maximal order 6. Output of these filters is a force value, which is amplified by a constant gain coefficient. The input value to this filter is the relative displacement of the spring, or respectively the relative velocity for the damper. Additionally the input value can be filtered by a low pass filter, a high pass filter or a band pass filter. In a pre-processing step the coefficients of the numerator and denominator polynomial of the transfer function can be calculated from a table of measured points of the frequency response or of the power spectral density.

Non-linear spring damper parallel with hysteresis (FE-104) force element includes both non-linear spring and damper characteristics. Additionally a hysteresis function is included. Basic principle of hysteresis can be seen from Figure 2.
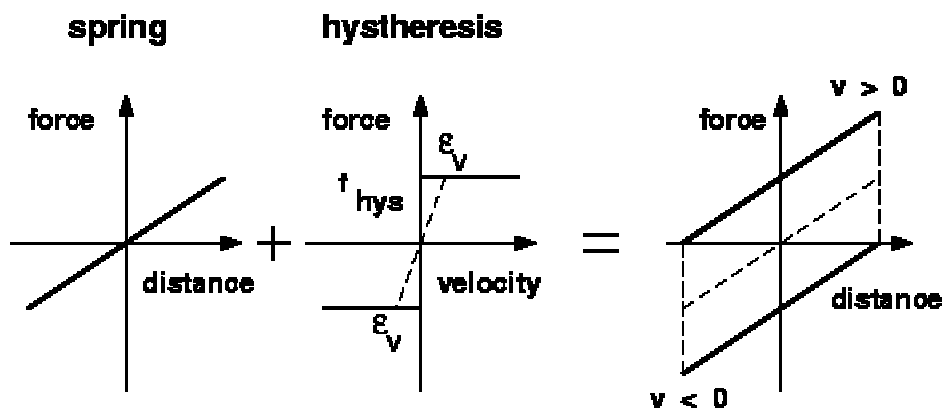

Figure 2. Principle of hysteresis modelling. [Anon. 1999, Simpack documentation]

The size of hysteresis force may depend on the size of displacement. Then the case can be like in Figure 3.
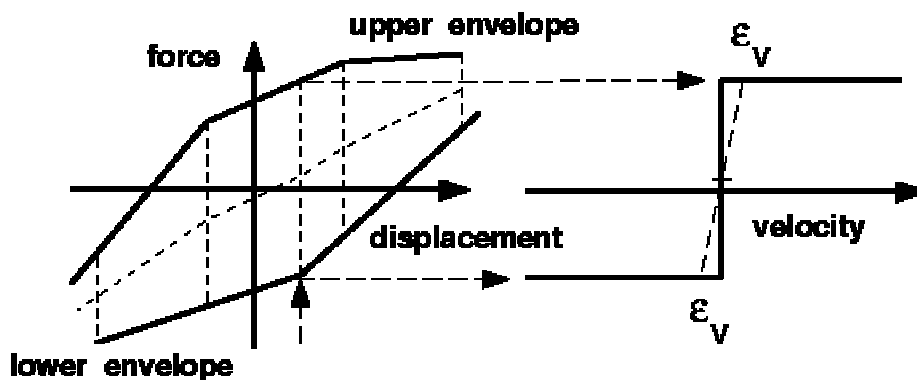

Figure 3. Hysteresis depending on displacement. [Anon. 1999, Simpack documentation]

3.2.8   Summary

The elements available in the selected software products can be summarised like in Table 10. The categories are based somehow on the elements available in ADAMS.

Table 10. The element types available in the selected software's.

Element	ADAMS	DADS	Simpack	Working Model Motion
Spring-Damper	X	X	X	X
Force between two points	X			
Force element (3-comp.)	X	X		
Torque element (3-comp.)	X	X		
General force element (6-comp.)	X		X	
Bushing	X	X	X	X
Beam	X	X		
Force element defined by 6x6 matrix	X	X	X	

Table 10 shows more or less how easy things are to model. If there is not an element with a similar name in some software like in ADAMS, it does not necessarily mean that the element with that kind of the properties can not be modelled in that software.

If there is a general force element defined by a 6x6 stiffness matrix, also a beam and a bushing element can be modelled with it. The definition of a bushing is a 6x6 matrix, where only the diagonal terms are used. The beam-element uses few terms more. It is just easier to use some of the beam elements for modelling an arbitrary beam structure than to use a general 6x6 matrix. With a beam element the user have to give only the geometrical dimensions and material properties to define the element instead of calculating all necessary terms of a 6x6 matrix himself.

With the similar logic a 6-component force can be constructed with a 3-component force element and a 3-component torque element modelled at the same location.

Major difference is if some element is linear or non-linear element. Also some practical differences come from how action force and reaction force are focused and directed to the bodies.

3.3   Discrete flexibility

First approach to model flexible structures is usually use the approach of discrete flexibility. In this approach an arbitrary structure of real world is divided to several bodies (masses) in the simulation model and these bodies are connected together with massless beam elements (or force elements defined by 6x6 stiffness matrices), Figure 4. The number of suitable divisions depends on the loading case, the expected deformations and required accuracy.
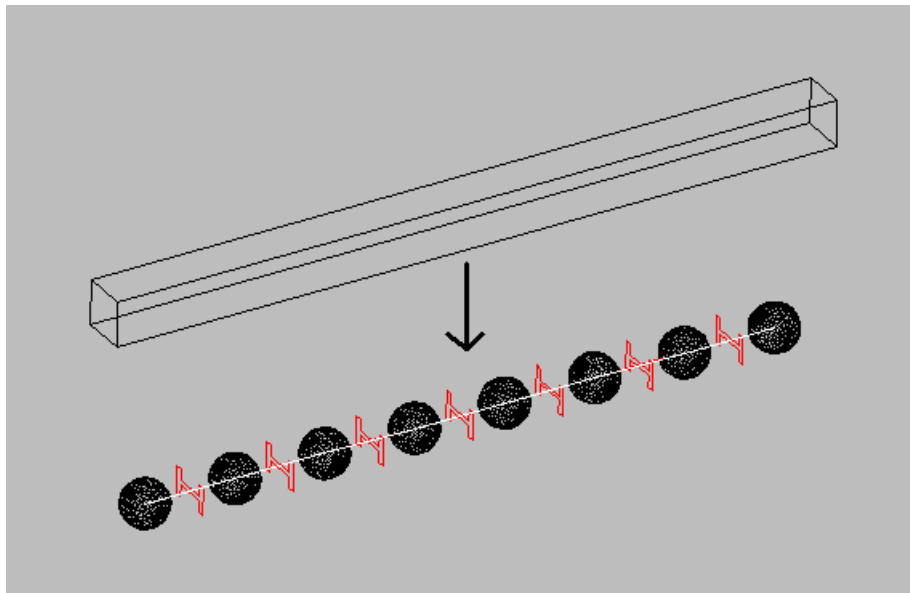
Figure 4. Defining flexibility of structure with discrete method.

One advantage of this method is that by connecting several linear elements, non-linear phenomena can be described.

3.4 Modal flexibility in ADAMS

Because the authors do not have experience of using other flexibility tools than ADAMS/Flex, this chapter will be based on it. In Chapter 3.5 some items are presented where other software's seem to employ different approaches according to their manuals.

3.4.1 Set of component modes

The modal flexibility method in ADAMS/Flex is based on the Craig-Bampton approach [Craig & Bamton 1968]. However, some minor modifications are made. As was presented earlier in this report (Section 2.3), the set of component modes, employed in Craig-Bampton approach, is not quasi-statically complete. The set of inertia-relief modes is not included. Thus, the kinematic space spanned by these modes can describe the static, but not the quasi-static, behaviour exactly. The meaning of this deficiency in practical analyses is not known for the authors.

3.4.2 Linearity

First, when using flexible bodies it has to be remembered that modal method of describing flexibility in ADAMS/Flex is a linear method. Another thing to remember is that stress stiffening effects are not included in ADAMS/Flex. These limitations can be override by dividing the flexible structure to several flexible bodies and connecting them together similar like using lumped masses in discrete flexibility method.

3.4.3 Connecting interfaces

Before starting of calculation of modes in FEA software it is not necessary to know attachment points for joints and forces in simulation model. Joints and forces can be applied to any

node of the body during the simulation phase. When the attachment points are known beforehand, defining them in FEA software helps to achieve solution with a minimal number of modes. That is a basic rule for many cases. But having a large number of attachment points can results unwieldy datafiles and can significantly impact CPU time if the associated modes are enabled during the dynamic simulation. So it seems that there is optimum point somewhere in the middle.

For example in I-DEAS there is two possibilities to define a boundary condition set. First is to constrain a node to have a zero displacement. Using this can lead easily to unrealistic situation. In ADAMS nodes constrained this way are fixed to Local Body Reference Frame. So they are not able to move relative to each other. This boundary condition should be used with great care. Another and recommended way is to constrain a node to have a displacement of 1,0.

3.4.4   Data transfer from FEA

The interface between a FEA-software and ADAMS/Flex is realised by using a modal neutral file (MNF). At the moment MNF for ADAMS/Flex can be generated with Abaqus, Ansys, Ideas and MSC/Nastran FEA-software. In order to get an overview of the interface we study as an example the data translation from I-Deas. The translated data entities related to flexible body properties are as follows:

1. Nodal co-ordinates
2. Element topology table
3. Global body properties (total mass, centre of mass, inertia tensor)
4. Eigenvalues
5. Mode shapes
6. Generalised mass of each mode
7. Generalised stiffness of each mode
8. Lumped nodal masses
9. Nodal inertia's (inertia tensor of each node)

The nodal co-ordinates and the element topology table describe the spatial properties of the translated flexible body. The rigid body inertia characteristics are described by the total mass, the centre of mass and the inertia tensor (3 x 3 matrix). The translated eigenvalues and mode shapes together with generalised mass and stiffness of each mode describe the flexibility and inertia properties based on assumed modes. The accuracy of the description can be enhanced by using more assumed modes. The inertia data is translated in the form of lumped masses and inertia tensors of each node. It means that the translated "mass matrix" is diagonal added by the non-diagonal term of nodal inertia tensors.

The translation of inertia data is central, because the coupling between the large rigid body motions and the small amplitude deformations are coupled by the inertia terms. In addition, the translated inertia information is the starting point for the calculation of inertia shape functions. It can be mentioned that it could be possible to calculate the inertia shape functions outside of the simulation solver, e.g. in some of the possible FEA software.

When importing a flexible body into ADAMS with MNF, ADAMS creates necessary geometry displaying flexible body and mesh representing the flexible body nodes. Mesh geometry

can be simplified later if the animation speed is too slow. Flexible body is located so, that its local body reference frame is in the origin of the global co-ordinate system. Body can be moved with normal moving methods in ADAMS/View.

3.4.5   Orthogonalisation of component modes

In ADAMS/Flex the component modes obtained by a FE-analysis are orthogonalised. By this technique the rigid body modes can be identified. In addition, all orthonormalised component modes will get an associated frequency. The ADAMS/Flex manual presents some other favourable consequences [Anon. 1998]. However, the orthonormalisation interweave the constraint modes and normal modes. Thus, the distinction between the quasistatic and dynamic behaviour of the member is lost.

The orthonormalisation of component modes brings out a question associated to the truncation or disabling of component modes. The truncation or disabling can be done in two different phases: before or after the orthonormalisation. The first choice is made when the number of normal modes ecported from the FE-analysis is fixed. The second choice is made in ADAMS/Flex, when the set of enabled modes for the simulation model is fixed. The essential question can be stated as: in which phase it is best to eliminate unnecessary component modes and which kind of criteria could be used for that ?

3.4.6   Damping

In importing stage the type of damping used is selected. User can enter a scalar damping ratio that is applied to all modes. Also the default damping method can be selected. Then ADAMS/Flex applies non-zero damping as follows:

- 1% damping for all modes with frequency lower than 100.
- 10% damping for modes with frequency in the 100 - 1000 range.
- 100% critical damping for modes with frequency above 1000.

Third possibility is to use user-written subroutine to define damping. That can be done later after body is created. With this method damping can have different levels for different modes and it can vary over time.

3.4.7   Connecting flexible bodies

There are some limitations, when flexible body is attached to other bodies in the model. Force elements beam, bushing and field can not be connected to flexible bodies. General force, force vector and torque vector can be connected only when the flexible body is the action body. Joints having moving contact point, like translational or inplane joints, can not be connected to flexible body. Also motion generators can not be used, when other body of joint is flexible body. A marker can be created to any node of flexible body but not between nodes.

In suitable case these limitations can be override with use of massless link. This massless link is connected to flexible body with allowed joint and limited element or marker is then connected to that massless link. This method can have some unwanted effects, so it has to be careful when using it. It is also recommended to use massless link when connecting force to

flexible body even if it is a action body. This is because evaluating the force on massless link can be computationally more efficient than evaluating force on flexible body.

3.4.8   Choice of component modes for simulation

Modifying flexible body opens up a flexible body modify dialog box. It can be used for changing some properties of flexible body. One is to enable and disable modes active in dynamic simulation. Another is to choose the method of inertia modelling.

First step is to check that six rigid body modes are disabled. That should be done automatically, but sometime procedure fails to recognise rigid body modes. Increasing computational efficiency of simulations can be done by disabling any modes that do not contribute to the motion which is wanted to model. Modes can be disabled based on their mode number, as a group based on their mode number or frequency and through a table. Selecting these modes to be disabled is demanding task to do.

There are some guidelines and warnings given in ADAMS/Flex manual [Anon. 1998].
"The participation of a mode is not based solely on their resonant frequency and the frequency of excitation. For example static load can cause the deformation of the component in such way that can only be captured by the shape that corresponds to a particular high frequency mode. You should only disable modes because of unacceptable computational overhead or when you are certain that a particular mode does not contribute to overall response. You should never disable modes based on an arbitrary notion of a frequency range of interest."

ADAMS offers few tools that help on choosing right modes to be disabled (Or should it be said that the user doesn't disable necessary modes). Animate tool can be used for seeing deformation of particular mode. This can be used for estimating if that deformation is interesting one. When disabling modes with higher order frequencies, always keep track of the number of actual DOFs that ADAMS computes for the assembled model. If redundant constraints start to appear, it means that the flexible body has too few modes retained to be compatible with the external constraints. Third way is to use help of ADAMS/Linear. With test model having only flexible body constrained to ground similar to how it will be constrained in the final model. After running ADAMS/Linear the natural frequencies achieved should match on those predicted by FEA-software.

3.4.9   Inertia terms

There are four choices for the method of inertia modelling: rigid body, constant, partial coupling (default) and full coupling. Choosing rigid body means that all modes are disabled and body behaves like rigid body. Constant-option is said to have an academic value, because it has only small effect on computational savings but dramatic effect on results. Default, partial coupling-option disables ninth invariant of mass matrix of flexible body. This invariant imposes the greatest computational overhead when having minor impact on results. Full accuracy is achieved with full coupling -option. This is recommended at least when working with very flexible bodies.

3.5   Modal flexibility in other software products

Other flexibility packages studied were DADS/Flex and Simpack FEMBS. Their manuals did not cover very detailed information of the mathematical background. It seems that basic working principle and methods are similar to ADAMS/Flex.

Data transfer format from FEA software to simulation software have own format in each software. DADS/Flex uses Flexible Database File and Simpack FEMBS uses Standard Input Data (SID). At the moment, input file for DADS/Flex can be generated with Abaqus, Ansys, Ideas and Nastran FEA-software's. Input data for Simpack FEMBS can be generated with Abaqus, Ansys, and Nastran FEA-software's.

DADS/Flex manual does cover the available types of damping. In Simpack FEMBS there is two types to choose from: modal damping proportional to the stiffness values or the natural damping (via Lehr).

At the moment, in Working Model Motion it is not possible to use modal flexibility to model flexible structures.

4   Hierarchical design approaches

Lot of input data is needed for detailed vehicle model. When starting a new vehicle model project, every detail is not necessarily known at the starting moment. Modelling existing vehicle may have also problems with input data. Sometimes it is very hard to find out all necessary properties of important component, especially if you are not a vehicle manufacturer or if you don't have very good testing facilities around.

Making detailed model of vehicle is demanding modelling process. It consists of many very detailed items. If adding all the modelling details at the same time, it is very easy to encounter serious debugging problems when starting to test the vehicle model.

These two potential problems are good reasons for doing the modelling step by step. Starting from simplified model or sketch and adding more details after testing simplified stages of model.

A suspension with elasto-kinematic properties can be good example of this modelling approach. At the first stage modelled bodies can be connected with idealised joints. Idealised joints are quick to add and they don't need any special parameters, only location and type of freedom is needed to know. After modelling suspension by this way, it can be tested, so that it works in principle. Kinematic analysis can also be done, getting ideas about how different angles changes during suspension movements.

When this is all working and need for elastic properties come essential, these joints can be replaced with flexible mounting components. Joints used in earlier stage don't have to be deleted completely. They can just be deactivated. Then it is possible to return to kinematic model, if encountering problems later in some stage or another detail area. Logical way to go is to model real rubber bushing with bushing-elements. Bushing-element needs only stiffness

and damping coefficients in different directions so that they can be modelled. Limitation with bushings is that they are linear components (in ADAMS at least). That is not a problem simulation smooth and not so critical driving situations. If there is a feeling, that linear range is exceeded in some situation, these bushing elements can be replaced with general force elements. Then more detailed information is needed, but of cause better result is achieved.

This same strategy is suitable for modelling flexible structures. Start always with rigid bodies. model everything else and be sure, that those other details are working correctly. Then start to test with flexible bodies.

5 Future trends

Representatives of different software were asked what are the future trends on this area of modelling and simulating flexible bodies. They told, that there will be improvements in all areas of flexibility modelling; pre-processing, simulation and post-processing. When this all will happen, that is another thing.

In area of pre-processing there will be possibilities to generate flexible bodies directly from rigid body in simulation software without external FEA-software. First this will be possible for bodies with simple geometry and later for more complex ones.

Simulating (many) complex flexible structures can be very time consuming task. So naturally one important task is to improve calculation efficiency.

On post-processing area one new feature will be that it will be possible to solve and see (coloured animations) stresses directly in simulation software. This feature together with ideas in pre-processing side means implementing of FEA-code in some level to simulation software.

One area for improvements is to make loading case data transfer to other software easier.

Also other possibilities, than find out stresses with FEA-software, are coming for post-processing of simulation data. Loading cases of dynamic simulation with flexible bodies can be transferred to fatigue and noise –prediction software. That is quite interesting trend when thinking this VÄRE technology programme.

6 Application examples

Vehicles have a lot of components, which are flexible during the operation. In many components flexibility is designed property and functionality of component is based on flexibility. Of cause there are components, where flexibility is unwanted property and it has to be accepted because of other design criteria.

There are three typical areas, where flexible bodies are widely used in vehicle simulations:

- Engine
  - crankshaft
  - connection rod
  - cylinder line
  - cylinder block
  - valve mechanism
- Suspension
  - suspension arm
  - leaf spring
  - antiroll bar
- Frame or bodywork

A report "*Commercial Vehicle Modelling and the Influence of Frame Flexibility on Vehicle Response*", [Koppenaal, 1998] can be presented as an example of using flexibility in vehicle simulations. There the frame of a truck has been modelled as a flexible body and it has been used in analysis of handling and vibration studies. Frame, Figure 5, has been modelled with NASTRAN FEA-software. Original FEA-model has 90 000 degrees of freedom.
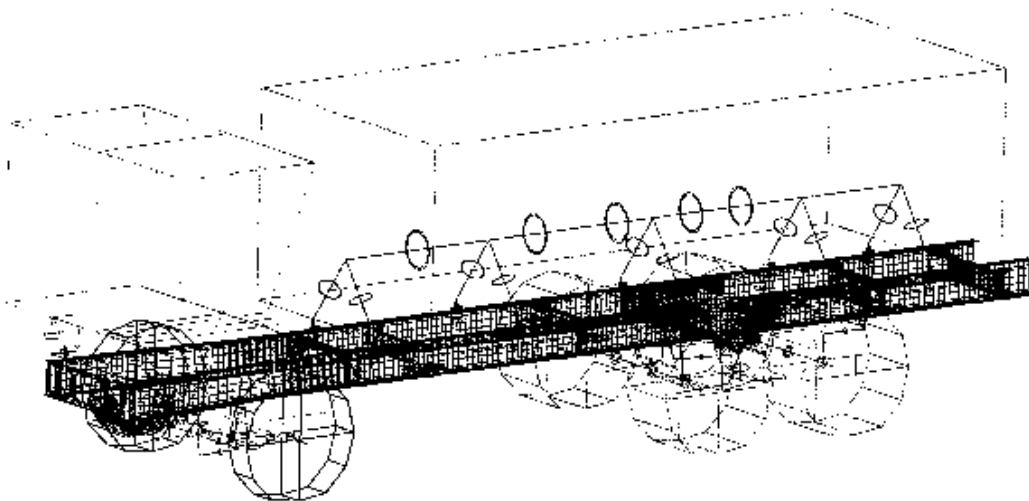

Figure 5. The flexible frame of a truck [Koppenaal, 1997].

Some comparison of rigid body model, simple torsional spring model and flexible body model of frame has been done.

An other example can be given from engine simulations; "*3D Piston Secondary Motion*", [Bamford, 1998]. Objective of that study has been to modell full piston motion inside the cylinder bore using flexible piston and connecting rod parts. Piston and connecting rod has been automeshed in I-Deas and then the modes were processed using NASTRAN. Model of piston and connecting rod can be seen in Figure 6.
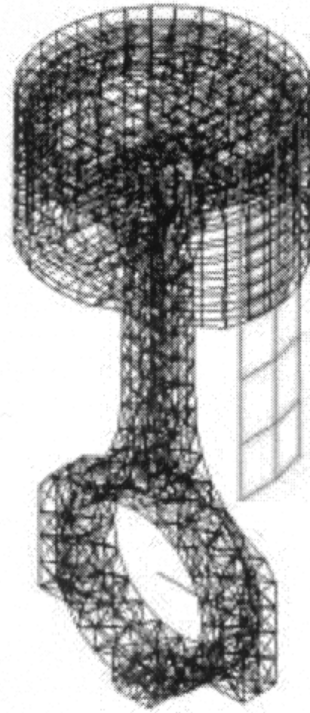
Figure 6.The flexible piston and connection rod model [Bamford, 1998].

Here is a references-list of articles about use of flexibility in MBS-simulations. These articles have been presented on ADAMS Users' Conference and they downloadable from internet location mentioned.

1997 International ADAMS User Conference
May 13-14, 1997
Eastern Michigan University's Corporate Education Center
Ypsilanti, Michigan, U. S. A.

12th European ADAMS Users' Conference
November 18-19, 1997
SORAT Hotel in Marburg, Germany

http://support.adams.com/support/conf_pap97.htm
- "*Experimental Vibration Analysis to ADAMS Integration: Flexible Chassis Full Vehicle Simulation,*" E. Pisino, L. Guglielmetto, Fiat CRF; D. Catelani, Mechanical Dynamics srl Italy
- "*Commercial Vehicle Modeling and the Influence of Frame Flexibility on Vehicle Response,*"J. Koppenaal, S. Edlund, Volvo Truck Corporation
- "*Vehicle Ride Study with Flexible Bodies in ADAMS,*" H.T. Chang, Delphi Chassis System; M. Rushbrook, NAO Chassis Center; S. Joy, NAO Small Car Group
- "*Structural Crash Analysis with ADAMS: A Comparison Between Multibody and FEM Approaches,*" N. Bertorelli, A. Gugliotta, R. Montanini, R. Vadori, Dipartimento di Meccanica, Politecnico di Torino

- "*Optimum Design of Engine Mounts by Integrating Triaxial Test Data and Analytical ADAMS Model*," S. Ochi, Cooper Engineered Products; B. Bigot, MTS; K. Bowland, Mechanical Dynamics, Inc.
- "*A Piston Secondary Motion Model Including Structural Flexibility*," M. Tussing, Cummins Engine Company; D. Minen, Mechanical Dynamics srl Italy

13th European ADAMS Users' Conference
November 18-19, 1998
Hotel New York in Disneyland Paris, France

http://support.adams.com/support/euc98day1.htm
- "*Study of Component Flexibility Effects on Full-System Performance via Mixed Body Dynamics Approach*," F. Palcak & M. Hulla, SOVA a.s. - Slovak Republic

http://support.adams.com/support/euc98day2.htm
- "*Use of ADAMS/Flex for Handling, Steering, Primary and Secondary Ride Dynamics*," M. Pilling, Ricardo FFD - UK
- "*Modelling of the Leyland 55 Series Truck using a Combination of Multibody Dynamics and Finite Element Techniques*," M. Pendlebury & M. Green, Leyland Trucks – UK
- "*3-D Piston Secondary Motion*," M. Bamford, Ford Powertrain Division - USA

1998 International ADAMS User Conference
June 9-10, 1998
Morris J. Lawrence Building, Washtenaw Community College
Ann Arbor, Michigan, U. S. A.

http://support.adams.com/support/iuc/pap_pres.html
- "*A Study of Vibrational Behaviour of a Medium Truck Considering Frame Flexibility*," Dr. Alvaro Costa Neto, Univ. de Sao Paulo, L. Ferraro, V. Veissid, and C. Freitas, Mercedes-Benz M. Argentino, R. Ripoli, and Odilon Perseguim, Humaita
- "*3-D Piston Secondary Motion*," Mike Bamford and Yeongchin Lin, Ford Powertrain
- "*Engine Structural Vibrations*," Jeff Jones and John Raub, Cummins Engine

1999 International ADAMS User Conference
November 17-19, 1999
The Kongresshalle
Berlin, Germany

http://support.adams.com/userconf/euc99/euc99_thu.htm
- "*Nonlinear Vibration Analysis of FIAT Brava Chassis with 3-Cylinder DI Diesel Engine by Elastic Multibody Simulation*," Dr. Fischer, D. Witteveen & D. Premstaller, Steyr-Daimler-Puch, Steyr
- "*Load Determination in Small Urban Bus using a Combined Finite Element and Multibody System Approach,*" J. Pasquini, Debis Humaita
- "*Modeling a Nonlinear Flexible Mechanism Using ADAMS/Flex and Nonlinear Co-rotational FE-Techniques,*" Mr. Clerici, ETH-Zürich & M. Brandenberger, CADFEM

7 Discussion and conclusions

In this report the modelling of flexible structural components was considered to be used as a part of a simulation model for vehicle dynamics. Firstly, the modelling of deformable components was considered on a general level. Secondly, different modifications of the component mode synthesis method were discussed. Thirdly, various modelling alternatives offered by different software products were reviewed. Finally, future trends and some application examples were presented.

The use of component mode method in flexible multibody dynamics offers many advantages. Perhaps the most important of these is the enhanced numerical efficiency, which is mainly a consequence of the reduced number of degrees-of-freedom of the system. In addition, the use of component modes and the respective generalised co-ordinates can give better insight into the dynamics of a flexible multibody system compared to the physical co-ordinates.

The component mode method has a set of restrictions, which delimit the applicational scope. In this method all flexible bodies are assumed to encounter only infinitesimal deformations, and the stress-strain relation is assumed to be linear. The research and development work proceeds in order to get rid of these limitations, but it will probably take several years before the advanced methods are available in commercial software products.

In this report some sort of a preliminary understanding has been acquired; at least the authors have that kind of an experience. However, many open questions remain to be inquired in the future. Some open and important questions are for example:

- How can be chosen a sufficient but not too large number of normal modes during the FE-analysis?
- What is the procedure to chose or compose adequate interface degrees-of-freedoms for a flexible body?
- What is the actual implementation of inertia terms of deformable bodies in a certain simulation software (e.g. ADMAS/Flex)?
- What is the implication of high frequency components (e.g. 10 kHz) of flexible bodies to solution efficiency?
- What are the criteria, which can be applied to disable component modes after orthogonalisation in order to enhance numerical efficiency?

Some recommendations, as working hypotheses, can be given in order to test the obtained ideas in practice:

1. Build up the interfaces of the FE-model carefully: a) make sure that the stiffness of all interface degrees-of-freedoms are sufficiently high, b) try to reduce all unnecessary interface degrees-of-freedoms.
2. Keep all the component modes, which describe static behaviour of the component. The respective supersets are given in Table 5.
3. Keep all the lowest normal modes up to a frequency f_{kl}, which can be chosen by employing the frequency content of the loading and the required accuracy of the analysis.
4. Calculate the natural frequencies and modes of each structural member in current simulation software. Compare these eigenvalue results with those obtained by FEM with the

same boundary conditions. Check the correspondence of natural modes and frequencies up to the limit frequency f_{kl}.

5. Be very careful, if you remove component modes after orthogonalisation.

The studied software products offer quite wide range of possibilities to model massless connecting components. In that sense only Working Model motion has major limitations because lack of suitable modelling elements. With other software included it can be said that basically limitation of what can be done what not, is not in software products. If user can describe the properties of element he wants to model in mathematical format, it can be modelled in simulation software. More or less it is question how easy it is to do. Do you have to calculate and input all the terms of 6x6 stiffness matrix by yourself ? Or is it enough to define some stiffness coefficienst, geometrical dimensions and material parameters and after that the software takes care of the rest? This is also main difference between ADAMS, DADS and Simpack software.

This can be classified by defining two user levels; engineer user and expert user. An engineer user needs a tool, which is easy and quick to use. In that kind of tool there is wide range of elements, where user needs only to input some parameters and software takes care of the mathematical formulation of that element. Typically these input parameters has strongly relation to reality. An expert user needs open environment without limitations. Elements available can be low level basic elements, where user has to describe mathematical formulation of the component he want to model.

Ranking of software is not unambiguous task and so it is not done here. It can be said that by combining all elements of software included it could be created a user friendly software where is no limitations.

References

Anon. 1998. Using ADAMS/View. Online documentation of ADAMS software. (Version 9.1, June 24, 1998)

Anon. 1998. Using ADAMS/Flex. Online documentation of ADAMS software. (Version 9.1, July 27, 1998)

Anon. 1998. DADS User's guide. Online documentation of DADS software. (Revision 9.0, 1998)

Anon. 1998. DADS Reference Manual, chapter 12. Online documentation of DADS software. (Revision 9.0, 1998)

Anon. 1998. DADS/Flex Manual. Online documentation of DADS software. (Revision 9.0, 1998, excluding chapter 5)

Anon. 1999. MSC/Working Model, User's Manual for Windows 95, Windows 98 and Windows NT. Online documentation of Working Model software. (Version 5.0, 1999)

Anon. 1999. SIMDOC 7.1. Online documentation of Simpack software. (Version 7.1, February 21, 1999)

Anon. 1999. SIMDOC. FEMBS User Manual. Online documentation of Simpack software. (Version 3.0, August 13, 1997)

Bamford, M., Lin, Y., Jiang, Y., and Harrish, B. 1998. 3D Piston Secondary Motion. Presentation in International ADAMS User Conference, June 9-10, 1998, Ann Arbor, U. S. A. Source of presentation: http://support.adams.com/support/iuc/pap_pres.html

Bathe, K.-J. 1982. Finite element procedures in engineering analysis. New Jersey: Prentice-Hall. 735 p. ISBN 0-13-317305-4

Bourquin, F. & d'Hennezel, F. 1992. Intrinsic component mode synthesis and plate vibrations. Computers & Structures, Vol. 44, pp. 315 - 324.

Craig, R. R. Jr. & Bampton, M. C. C. 1968. Coupling of substructures for dynamic analyses. AIAA Journal, Vol. 6, No. 7, pp. 1313 - 1319.

Craig, R. R. Jr. & Chang, C.-J. 1977. On the use of attachment modes in substructure coupling for dynamic analysis. Proceedings of AIAA/ASME 18th Structures, Structural Dynamics & Materials Conference. Vol. B., pp. 89 - 99.

Craig, R. R., Jr. 1981. Structural dynamics. New York: John Wiley & Sons. 527 p. ISBN 0-471-04499-7

Craig, R. R., Jr. 1987. A review of time-domain and frequency-domain component mode synthesis method. International Journal of Analytical and Experimental Modal Analysis, Vol. 2, No. 2, pp. 59 - 72.

Eberhard, P. & Schiehlen, W. 1998. Hierarchical modeling in multibody dynamics. Archive of Applied Mechanics, Vol. 68, pp. 237 - 246.

Genta, G. 1997. Motor vehicle dynamics: modeling and simulation. Singapore: World Scientific Publishing. 539 p. (Series on advances in mathematics for applied sciences: vol. 43.) ISBN 9810229119

Herting, D.N. 1979. A general purpose, multi-stage, component modal synthesis method. AIAA/ASME/ASCE/AHS 20th Structures, Structural Dynamics and Materials Conference. St. Louis, Mo. 1979.

Hintz, R. M. 1975. Analytical methods in component modal synthesis. AIAA Journal, Vol. 13, No. 8, pp. 1007 - 1016.

Hurty, W. C. 1965. Dynamic analysis of structural systems using component modes. AIAA Journal, Vol. 3, No. 4, pp. 678 - 685.

Klinge, P. 1998. Värähtelymuotojen synteesi matriisin osituksella (Component mode synthesis by matrix decomposition). Licentiate's thesis. Espoo, Helsinki University of Technology, Laboratory for Strength of Materials. 69 p.

Klinge, P. 1999. Private discussion conserning the finnish terms. 16.12.1999.

Koppenaal Jacco, Edlund Stefan 1997. Commercial Vehicle Modelling and the Influence of Frame Flexibility on Vehicle Response. Presentation in 12th European ADAMS Users' Conference, November 18-19, 1997 in Marburg, Germany. Source of presentation: http://support.adams.com/support/conf_pap97.htm.

MacNeal, R. H. 1971. A hybrid method of component mode synthesis. Computers & Structures, Vol. 1, No. 4, pp. 581 - 601.

Óttarson, G. 1998. Modal flexibility method in ADAMS/Flex. ADAMS Technical Support, Knowledge Base, Article 7247, 31.3.1998. 18 p. Electronic version: http://support.adams.com/kb-files/id_7247/modalflexmethod.pdf (This article applies to ADAMS/Flex (9.0.4) on all platforms.)

Rubin, S. 1975. Improved component-mode representation for structural dynamic analysis. AIAA Journal, Vol. 13, No. 8, pp. 995 - 1006.

Seshu, P. 1997. Substructuring and component mode synthesis. Shock and Vibration, Vol. 4, No. 3, pp. 199 - 210.

Shabana, A. A. 1998. Dynamics of multibody systems. 2nd ed. Cambridge: Cambridge University Press. 372 p. ISBN 0 521 59446 4

Wang, W. & Kirkhope, J. 1995. Complex component mode synthesis for damped systems. Journal of Sound and Vibration, Vol. 181, No. 5, pp. 781 - 800.