Comparison of Modeling Techniques for Flexible Dummy Parts

W. P. Koppens and H. A. Lupker
TNO Crash-Safety Research Centre

C. W. Rademaker
Eindhoven University of Technology

ABSTRACT

An important requisite for an effective use of numerical tools in occupant safety assessment is that suitable models of crash victims are available. Reliable well validated multibody models of several crash dummies have been developed in the last two decades. Flexible parts are modeled as rigid bodies interconnected by joints that account for the flexibility. Recently the finite element method has been used for modeling such parts. In this paper an alternative method is proposed namely as a flexible body with distributed mass and stiffness.

To evaluate these methods, three models of a rib module of a EUROSID-1 dummy are compared with the rib modeled as:
- a chain of nine rigid bodies interconnected by revolute joints, and torsional springs and dampers,
- one flexible body with the deformation approximated by one predefined displacement mode,
- a finite element model using triangular shell elements with a linear elastic material behavior and mass proportional Rayleigh damping.

The numerical results are validated using EUROSID-1 rib module lateral and oblique impactor tests. In these tests the rib module is fixed vertically on one end and the other end is impacted by a falling mass. The models are evaluated with respect to their modeling capabilities and computational efficiency.

It is concluded that the three models can be used to analyze impacts from relevant directions. The displacement modes for the flexible body model can be obtained from finite element analyses or from experiments. The latter approach does not require a detailed description of the flexible structure and its material behavior. The finite element model requires considerably more computation time as compared to the multibody models.

INTRODUCTION

Traffic accidents are a major cause of death and disability. Because of the emanating costs, the authorities tend to make safety regulations of vehicles more and more strict. For this reason and because safety tends to become an important buying reason, the automotive industry takes great efforts to maximize the safety of occupants and other road users within the constraints associated with the purchase price and the usage costs of the vehicle.

To improve the safety of a vehicle, the results of crash tests and numerical analyses are used. Numerical analysis has proven to be a valuable tool that can save much time during the early phases of the design process when used to evaluate the effect of design modifications. The resulting design improvements can
be carried through in this stage without considerable costs and the number of crash tests required for design validation can be reduced.

In crash tests ‘mechanical models’ or ‘crash dummies’ are used as substitutes for human occupants. These dummies are equipped with special instrumentation to assess injuries. Different crash dummies are used for frontal and side crash tests. An example of the latter is the EUROpean Side Impact Dummy, EUROSiD-1 [1], which represents a 50th percentile adult male.

An important requisite for an effective use of numerical tools for occupant safety assessment is that reliable, well validated models of crash dummies are available [2]. In order to avoid duplication of work, standard numerical models of dummies are often developed in cooperation with vehicle manufacturer associations and made available to users of the numerical tool by its software vendor [3, 4]. Such a numerical model of a dummy must represent the behavior of the dummy in all those circumstances that the actual crash dummy is meant for because they are applied by different users.

Dummies are modeled usually as systems of rigid bodies interconnected by joints [5]. Flexible dummy parts are approximated by several rigid bodies interconnected by joints and springs. Recently flexible dummy parts have also been modeled using the finite element method [6]. In this paper a new method is proposed for modeling flexible dummy parts namely as flexible bodies with distributed mass and stiffness. The subject of this paper is to investigate the relative advantages of these techniques for modeling flexible dummy parts. The different techniques are compared with respect to their computational efficiency and capability to develop models of dummy components that can be applied under various conditions. This is done on the basis of a rib module of the EUROSiD-1. This dummy component is ideal for comparing these modeling techniques because it is a relatively simple part for which well defined component tests are available. The multibody module of the program MADYMO-3D was extended with the option to analyze flexible bodies. In addition a shell element was added to the finite element module of this program.

This paper starts with an overview of different techniques for modeling flexible structures. Next a description is given of a EUROSiD-1 rib module and the models based on the different modeling techniques. Finally the results of numerical analyses and experiments of impactor tests on this rib module are presented and discussed.

MODELING FLEXIBLE STRUCTURES

The motion of a flexible structure is fully described by the displacement of every one of its points. Consequently a flexible structure has an infinite number of degrees of freedom. In general it is not possible to obtain a closed-form solution for the motion of a flexible structure. Therefore a flexible structure is approximated by a discrete model with a finite number of degrees of freedom. Three discretisation procedures are compared in this paper, namely the lumped parameter method and the modal synthesis method, both leading to a multibody model, and the finite element method.

LUMPED PARAMETER METHOD - A flexible structure can be approximated by a system of rigid bodies. For this purpose the structure is subdivided into parts. The inertia of such a part is assigned to the corresponding rigid body. The flexibility of a part is accounted for by kinematic joints and springs connecting the body and contiguous bodies. This method has been used by Low and Prasad [7] for modeling the ribcage of the DOT-SID dummy.

MODAL SYNTHESIS METHOD - A flexible structure can be modeled as a flexible body. The motion of a flexible body can be approximated by writing the displacement of the body as a linear combination of displacement fields (modes). However it is not possible
to approximate displacements corresponding to large rotations as a linear combination of modes. Since it is a requisite that also large rotations can be described, the displacements of a flexible body are subdivided into displacements due to a rigid body motion and displacements due to deformation [8]. The displacements due to deformation are then approximated by a linear combination of predefined modes:

$$\mathbf{u}(\mathbf{x},t) = \sum_{i=1}^{N} \alpha_i(t) \Phi_i(\mathbf{x})$$

where \(\mathbf{u}(\mathbf{x},t)\) is the displacement due to deformation of an arbitrary point of the body which has the position vector \(\mathbf{x}\) in the undeformed configuration of the body, \(\Phi_i(\mathbf{x})\) is the \(i\)-th predefined mode, and \(\alpha_i(t)\) is the weighting factor of the \(i\)-th mode. These \(N\) weighting factors are the degrees of freedom of the flexible body. Their values follow from solving the equations of motion.

A mode defines a displacement for every point of the body. Deformations of a body that can be approximated well, are restricted by the specified modes. This is no severe restriction when the possible deformation is known in advance. In order to obtain an accurate solution with only a limited number of modes they should be chosen such that they approximate the actual displacements due to deformation of the body well. The modal synthesis method will lead then to a model with only a small number of degrees of freedom, and consequently requires only a limited amount of computational effort. Modes are determined often either from a finite element analysis or from experiments.

**FINITE ELEMENT METHOD** - A flexible structure can be subdivided into a finite number of parts having a simple shape (elements). Contiguous elements are interconnected at a discrete number of points on the interface, the nodes. The displacements of these nodes are assumed to be unknown. Displacements within an element are determined by interpolation of the displacements of the nodes of that element. The properties of an element, i.e. its stiffness and inertia, can be determined from its geometry, the material properties and the interpolation functions. These can be assembled for all elements leading to a model for the flexible structure. The number of nodes, and consequently the number of degrees of freedom, that is required depends on the complexity of the geometry of the body and the need of having small elements at locations where displacement gradients are large.

**MADYMO**

The different modeling techniques will be compared using a development version of MADYMO [9]. This is an engineering analysis program for the simulation of structures undergoing large motions. The program has been designed especially for the study of the complex dynamical response of humans or human surrogates and their environment due to impacts like they occur in vehicle crashes.

**Fig. 1 Program structure**

MADYMO combines in one simulation program the capabilities offered by multibody and finite element techniques [10]. Interaction between the corresponding modules of the program is through contacts, supports and belts (Fig. 1). For solving the equations of motion, different time integration methods are used by the multibody module (Runge Kutta method) and the finite element module (Central Differences method). These are explicit methods that become unstable when the time step is too large. The maximum time step is proportional to the highest eigenfrequency in the model. Since the highest eigenfrequency of a multibody system is relatively small, the finite
element module is subcycled, i.e. a smaller time step is used, with respect to the multibody module in order to take advantage of the fact that larger time steps may be used for the multibody module.

MADYMO 5.0 has been extended with an option to generate and solve the equations of motion of systems of flexible bodies with a tree structure. In addition a 6 node triangular shell element with constant strain triangle and constant bending moment triangle has been added to the finite element module.

THE RIB MODULE OF THE EUROSID-1

A rib module is built up of three components (see Fig. 2):

- a steel strip bent in the shape of a rib covered with polyurethane foam; the foam is held onto the strip by a cotton sleeve covered with a pvc-based coating,
- a piston-cylinder assembly which connects the ends of the steel strip with a spring in the cylinder (the tuning spring),
- a hydraulic damper fitted with a return spring and a spring cup in series with a spring (the damper spring).

The cylinder is bolted to the spine box of the dummy such that the rib module is impacted from the direction indicated in Fig. 2. Joint stops limit the stroke of the piston-cylinder assembly to the range from 0 till 60 mm. A rubber ring protects the piston-cylinder assembly from being damaged due to contact in a joint stop. The support of the damper spring is such that it can take only a compressive force. The tuning spring and the damper spring are initially unloaded; there is a small initial pretension in the return spring.

RIB MODULE IMPACTOR TESTS

The response of a rib module due to a side impact must satisfy the requirements given in the EUROSID-1 User's Manual [1]. The certification procedure consists of prescribed impactor tests for various impact velocities. Tests are prescribed for the rib module without the damper and the damper spring (rib only), and the full rib module. The maximum rib deflection must be within a specified corridor. The results of the certification tests will be used to validate the different numerical models of the rib module.

In order to assess the power of the numerical models to predict the response of the rib module in other impact conditions, verification tests have been carried out. These consists of perpendicular and oblique impactor tests with a smaller impactor mass and a higher impact velocity.

CERTIFICATION TEST SET-UP - Fig. 3 shows the test rig used for certification. The rib module is bolted to the table at the regular rib-spine interface, impacted side up. The impactor has a mass of 7.78 kg and a flat face with a diameter of 150 mm. The center of mass of the impactor is guided with the center line of the piston by cables. Tests must be performed for four drop heights leading to impact velocities of 1, 2, 3, and 4 m/s. The deflection of the rib is measured with the rib's own displacement transducer.

CERTIFICATION PROCEDURE - The peak deflection of the rib due to impact with the prescribed velocities must be within a given range. In case a rib module does not satisfy this requirement, the tuning spring must be replaced by one of the other four tuning springs (which have a different stiffness) such that the requirements are met. The rib must be replaced in case it is not possible to meet the requirements. In case a full rib module does not satisfy the requirements, the damper must be checked and replaced when necessary.
MODELS OF THE RIB MODULE

The models that have been used for analysing the rib module differ only in the way the rib is modeled; the piston-cylinder assembly, the hydraulic damper and the damper spring, the impactor and the contact between the rib and the impactor are modeled basically the same (Fig. 5).

Fig. 5 Model of rib module without rib

The cylinder, the damper, and the bracket connecting the cylinder and the damper are considered to be rigid. Therefore these parts are modeled as one rigid body, the frame body. This body is rigidly connected to the inertial space. The piston is modeled as a rigid body that is connected to the frame body by a translational joint. The tuning spring, and the rubber ring and joint stops are modeled as a spring and a parallel spring and damper, respectively, between the piston body and the frame body. The damper piston and the spring cup are modeled as one rigid body, the cup body, that is connected by a translational joint to the frame body. The damper and the return spring are modeled as a parallel spring and damper between the frame body and the cup body. The damper spring is modeled as a spring between the spring cup and the rib. The foam covering the end of the rib is modeled by two contact ellipsoids connected to the piston body. The mechanical behavior of the foam on the rib is included in the contact characteristics of the ellipsoids. The data used in this model are taken from the rigid body model of the EUROSID-1 described in [11].

The impactor is modeled as a rigid body
that is connected to the inertial space by a translational joint. The impactor face is modeled as a rigid plane. The contact between the impactor and the foam covering the rib is modeled with the standard model for contact between planes and ellipsoids available in MADYMO.

RIGID BODY MODEL OF THE RIB - The rigid body model is taken from [11]. There the rib is modeled as a chain of nine rigid bodies. The mass of a rib body equals the mass of the segment of the steel strip and foam it represents. The mass of the segment of the rib that is connected to the piston body is added to the piston body mass. The mass of the first body is increased with the mass of the stub and half the mass of the damper spring. The rib bodies are interconnected by revolute joints which allow relative rotation around an axis that is perpendicular to the plane of the rib (Fig. 6). The first body of the chain is connected to the piston body by a revolute joint.

The bending stiffness of the rib is accounted for by torsional springs in the revolute joints. To account for the contribution of the foam to the bending stiffness of the rib, different loading and unloading torques have been specified for these torsional springs and viscous damping is specified. An ellipsoid is connected to each body for defining contact between the rib and the impactor.

FLEXIBLE BODY MODEL OF THE RIB - The deformation of a rib is mainly caused by the impact force of a striking object, and the resulting translation of the piston and the force from the damper spring. The modes that are used for a flexible body model should be such that deformations due to these loads can be approximated well. In general the location where a rib is struck is most likely to be somewhere on the impacted side, though the exact location is not known in advance. However the deformation pattern of the rib does not change drastically when it is struck at another location. This can be verified from Fig. 7 which shows the deformation modes of the rib loaded by a constant load parallel to the piston and a constant load at the location where the impactor hits the rib for the 20° oblique impactor test, respectively. These modes are obtained from two static finite element analyses of the full rib module. Therefore only the displacement of the rib corresponding to the lateral load is used in the flexible body model. Since the deformation does not change drastically with the impact location it is expected that the response of the rib module due to lateral and oblique impacts can be approximated well with this single mode. A modal damping of 90% of the critical damping was needed in order to make the flexible body model satisfy the certification requirements.

Ellipsoids are connected to the flexible body for modeling the contact between the foam and the impactor for the oblique impactor tests.

FINITE ELEMENT MODEL OF THE RIB - The part of the rib between the connection with the piston and the cylinder is modeled with triangular shell elements. Fig. 8 shows the finite element mesh that has been used. The rib end at the impacted side is clamped to the piston body. The other end of the rib is clamped to the frame body. An auxiliary body has been connected to the finite element mesh.
at the location where the damper spring is connected to the rib in order to take the mass of the stub and the damper spring into account. The end of the damper spring is connected to this body. The protruding ends of the rib are included in the piston and frame body.

![Finite element mesh of rib](image)

Fig. 8 Finite element mesh of rib

A linear elastic material behavior is used for the elements \( E = 2.05 \times 10^{11} \text{ N/m}^2, \nu = 0.27 \). The thickness of the elements is set equal to the thickness of the steel strip \( t = 2.7 \text{ mm} \). The contribution of the foam on the bending stiffness of the rib has been neglected. The mass of the foam is accounted for by increasing the specific mass of the rib material \( \rho = 8600 \text{ kg/m}^3 \). Ellipsoids are connected to the finite element structure for modeling the contact between the foam and the impactor for the oblique impactor tests.

The coefficient of the mass proportional Rayleigh damping of the finite element structure has been assigned such a value that the rib deflections obtained with the finite element models are within the certification requirements (the horizontal lines in Fig. 9). The width of the peaks of all numerical models are smaller than that of the experimental results. Similar deviations occurred for the other impact velocities. The range of peak displacements allowed for certification is a measure of the spread in rib deflection which may be expected for different rib modules. Since this spread is larger than the difference that can be observed between the experiments and the numerical models, it can be concluded that all three numerical models predict the response of the rib module with sufficient accuracy.

![Rib displacement vs Time](image)

Fig. 9 Rib deflection for 4 m/s certification test

In order to assess the capability of the resulting models to predict the response of a rib module under other impact conditions, they are used to predict the response of the rib module for the verification experiments. The maximum deflection of the rib appears to be sensitive to the friction between the impactor and the rib. However the friction coefficient was not available. Since it may be expected that the real friction force is small, the friction coefficient was set equal to zero.
Fig. 10 Rib deflection for 4 m/s verification tests

Fig. 11 Rib deflection for 7.5 m/s verification test at impact angle 0°

Fig. 10 shows the deflection of the rib due to impacts at 4 m/s and angles of 0°, 10°, and 20°. The response does not vary much with the impact angle. This is due to the fact that the impacted side of the rib does not deform much as can be seen from Fig. 7. All models are capable of predicting the response well. Note that for the flexible body model only one mode is used corresponding to an impact angle of 0°. The results corresponding to impact angles of 10° and 20° show that this mode is also suited for predicting the response due to oblique impacts.

The required CPU time depends to a great extend on the size of a model (the number of degrees of freedom) and the time step used for the numerical integration. The finite element model has much more degrees of freedom as compared with the multibody models. In addition the largest time step that can be used for the finite element model (4/3 μs) is smaller than that for the rigid body model (10 μs) and the flexible body model (500 μs). As a consequence the finite element model requires considerably more computational effort than the multibody models. The ratio of the average CPU time required for the rigid body model, the flexible body model and the finite element model for the certification tests analyses is 20:1:50.

In actual crash tests the impact velocity is often considerably higher than those used for the certification tests. Fig. 11 shows that also
for higher impact velocities the models give sufficiently accurate results.

DISCUSSION AND CONCLUSIONS

Three numerical models of a EUROSID-1 rib module have been developed using a chain of rigid bodies, one flexible body and finite element structure, respectively, to approximate the behavior of the rib. The models have been applied to analyze the behavior of the rib module in impactor tests. A good agreement between numerical and experimental results is found for each of the three models. However the flexible body model requires significantly less CPU time than the finite element model. The results of the models using one flexible body and that using finite elements are almost identical which indicates that for side and oblique impacts the rib can be represented by one flexible body using a superposition of only one displacement field.

A finite element model of a flexible structure can be created directly from the geometry and material properties of the structure, provided that suited material models are available. A rigid body model and a flexible body model require some processing of these data. However when applied in numerical models of dummies this is done only once during the development of the model and users of the model are not troubled by this.

In creating a flexible body model of a structure, deformation modes of the structure have to be determined in advance by numerical simulations or experiments. When modes are obtained from static finite element analyses all information that is contained in the finite element model, such as deformations, strains and stresses, can also be obtained from a flexible body model by combining the results of the different static finite element analyses using the weighting factors determined in the transient analysis. A rigid body model does not contain detailed information on the deformation of the structure.

Combining the different techniques in models of crash dummies will give an optimal numerical model with respect to accuracy and required CPU time. Dummy parts that are approximately rigid should be modeled with rigid bodies using contact ellipsoids and planes to account for the surface compliance. Dummy parts that experience small deformations should be modeled with flexible bodies. Only those dummy parts that undergo large deformations need to be modeled with finite elements.

Approximations have to be made in developing numerical models of dummies and their environment due to their complexity. Therefore the user of numerical models must be aware of such approximations and must verify whether they are allowable for his application.

REFERENCES


