NON-LINEAR MODAL ROLLING TYRE MODEL FOR DYNAMIC SIMULATION WITH ADAMS

F. Mancosu, D. Da Re Pirelli Pneumatici Spa - D. Minen MDI Italy

1– The problem: road noise simulation

The prediction and the optimization (reduction) of the vehicle vibrations (low and high frequencies 0-400 Hz) are an important goal in the car optimization.

The vibration sources in a running vehicle are different: an important effort is coming from the interaction between the **road/tyre/vehicle** ensemble. The simulation of this interaction (in a running car over a road profile) can help the understanding of the phenomena and can give opportunity of sensitivity analysis and optimizations without any physical constructions.

The full system to be implemented in the **road/tyre/vehicle** ensemble has to be validated (in time domain) taking into account signal in the frequencies range of 0-400 Hz. This is the typical range of the **road-noise** phenomena.

The simulation has to consider the real characteristics of the single components (road, tyre, and vehicle).

2 - The actors: road, tyres, and vehicle

It is important to define the actors (road, tyres, and vehicle) in a way that reproduces the real behaviour of the components.

In particular:

a) The road has to be represented physically with the right profile in 3D: see fig 1;

b) The tyre has to be defined considering:

  b1) the real modal characteristics up to 400 Hz (dumping included)

  b2) how the tyre "reads " the road profile in vertical , longitudinal (brush model).

c) The vehicle has to be represented taking in account all the components influencing the forces generation.

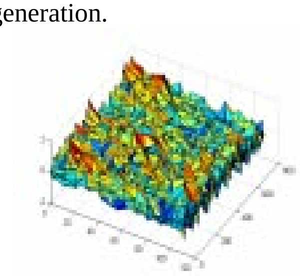

Fig. 1: example of 3D road profile obtained by experimental laser profile-meter

3 - Modeling the actors

The actors have to be modeled considering that the final **road/tyre/vehicle** ensemble has to be used in a common code for the simulations. It is simple to manage some pre-processor for the sub-model, but the common code has to be able to include easily the modeling of the single different components. The more usefully code for the simulation is a multi-body code (ADAMS).

In the past, practical design and analysis of multi-body, large displacement/small strain mechanical systems has been difficult due to the limitations imposed by commercial analysis software. No single software package was capable of "**doing it all**", and the resulting approximations and tedium of translating data among various packages served as a barrier to those seeking to perform the incorporation of flexible body information into ADAMS, or generation of more accurate component dynamic loads for FEM codes.

Such conditions led to the use of necessary, though not always adequate, approximations by practicing engineers: the use of Guyan reduction followed by a forced mass lumping to model flexible components in ADAMS, and the "guesstimation" of critical dynamic loading conditions in FEM code. The recent introduction of **ADAMS/Flex** has overcome many of the problems associated with analysis of flexible multibody dynamics by employing flexible elements whose component modes-based data is provided by finite element codes.

A procedure to realize the interface from FEM code, as HKS/ABAQUS, to ADAMS is now available. It provides an output file (.fil) for quantities such as physical mass, modal mass and stiffness, component modes, etc., for components which had first been built as superelement models. An external translator utility (fil2mnf, first prototype release) completes the process, reading the .fil file, modifying it (it realizes the orthogonalization procedure), and subsequently writing the data into a form which could be read as input to ADAMS (Modal Neutral File format). More, the obtained .mnf file could be completely managed by the new MNF toolkit (especially for Invariant calculation and Graphic reduction), obtaining a size-reduced .mnf file which contains all flexible data for flexible multibody analysis.

Interfacing a Finite Element program with a Large Displacement Multibody Dynamics program achieves two very desirable goals:

- ADAMS mechanical system model fidelity may be dramatically enhanced when component flexibility is accounted for

- Realistic loads for a FEM analyses may be obtained in a natural way by incorporating a FEM model of a component in an ADAMS mechanical system model and simulating in-service events.

3.1 The Road

3.1.1 Current implementation

The road is shown ad a meshed surface which penetrates into the Flexible tyre (see fig.2).
A new extended 3D contact algorithm is currently under development at MDI, and can take care of flexible contacting bodies. Preliminary examples of application together with a short description of the theoretical basis of the algorithm are shown hereby.
The procedure for detecting a compenetration condition between a Flexible Body and a meshed triangular surface and could be summarized into the following steps:

1. An high speed searching algorithm finds potential road contact elements, described as triangular polygons;
2. A vector line along the Z-axis of a Marker belonging to the Flexible Body is drawn;
3. The intersection Point where the Z-axis and the Road Surface intersect is computed (j-point, see fig.1);
4. dz(i,j,i) and vz(i,j,i) are then computed as outputs (j= computed road contact point);
5. A SFOSUB routine generates a force on i-Marker based on dz and vz.

The speed of the searching algorithm is almost independent from the number of road elements. A search radius parameter can be supplied for optimizing the search area.

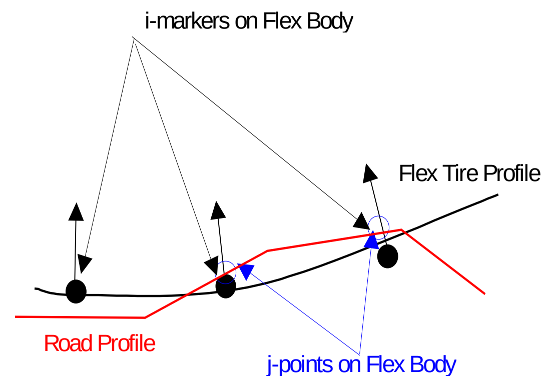

Fig. 2: detail of the deformation of the flexible tyre on a road profile

3.1.2 On going development

To avoid situations like those presented below, a new algorithm to detect compenetration between ground irregularities and Flex Tyre needs to be introduced. The Flex Tyre model is ideally divided into several wedges:

- each wedge has a base constituted by i-markers belonging to the deformable tyre surface, and therefore its volume is subjected to changes at each iteration step
- at every iteration, the intersecting volume between the wedge and the 3D road profile is computed
- by means of a look-up table derived from FE, the resulting volume penetration load is lumped to the nodes that constitute the base of the wedge, proportionally to the distance from the center of gravity of the penetrating volume
- a proportional force is applied to each of the base markers

3.2 Tyre –generality

3.2.1 Fea model: non-rolling tyre

The first step to describe the dynamic behavior of the tyre is a finite element simulation. The finite element model describes any geometrical and physical characteristics of the tyre. The dimensions and the complexity of the model don't allow the simulation in the time domain.
It is fundamental a good experimental characterization of the static and dynamic properties of the materials: all the data of the model was obtained by experimental tests performed both on rubber compound and reinforcing materials. The compound is statically tested with tensile, compression and shear stresses to achieve a meaningful hyperelastic formulation of the strong non-linear behavior.

The dynamic characterization was necessary to describe the frequency-dependent properties, in terms of elastic and hysteretic behavior. The reinforcing materials, such as carcass and belts, have been also experimentally tested to achieve both the positive part of the stress-strain curve and the negative part, using a special methodology and device set up by Pirelli. In figure 3 is shown an example of the dependence of the elastic modulus from the frequency.

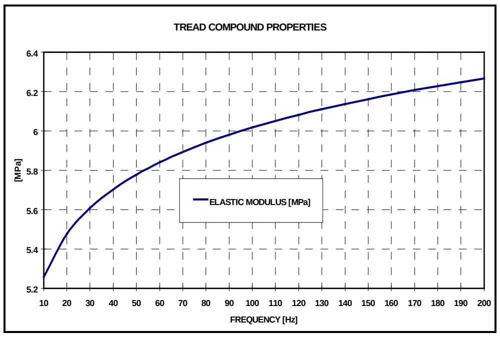

Fig. 3: variations of the elastic modulus of a tread compounds with the frequency.

The reinforcing material was implemented inside the rubber element as additional stiffness as function of the properties of the cords themselves. The rubber compound properties have been implemented using a hyperelastic formulation (Mooney Rivlin).

The finite element model is made up of 20 thousands of brick elements (linear shape functions, 8 nodes each element) that give something like 80 thousands degrees of freedom. The model was generated in an axisymmetric way and it has been demonstrated that requires at least 48 wedges to be able to properly evaluate the eigenfrequency up to about 150 Hz. In this specific case an 80 wedges model was built to investigate higher frequencies, up to 400 Hz. The higher is the maximum frequency of interest the finer should be the mesh of the model and the longer is the computational time. In figure 4 is shown an example of mode shape at 131.4 Hz.

To represent the real static working conditions of the tyre, it was carried out a preliminary set of analysis like the mounting on the rim, the inflation and the deflection with a specified vertical load. These analyses show strong non-linearity due to the contact kinematics and to the materials. After these analysis some steady state analysis was performed in order to calculate the modal damping associated to each eigenvalue, comparing the amplitude of the response at the eigenfrequencies with respect to the static response.

The steady state analysis is a linearized response to harmonic excitation based on the physical degrees of freedom of the model. The response can be achieved through the direct solution of the equations in matrix form, where the stiffness and damping matrices are updated each frequency since the definition of the isotropic linear viscoelasticity allows evaluating stiffness and damping matrices as function of the frequency.

While the response in this analysis is for linear vibrations, the prior static response is strongly nonlinear and the initial stress effects (stress stiffening) are included in the steady state response.

The finite element model can give any kind of data about any nodes of the model itself but it is a huge model, unable to be used for time domain dynamic simulations. As a consequence a strong reduction of the physical degrees of freedom is needed. Since the tyre acts as an interface between the road and the hub it is advisable to retain just the degrees of freedom of the nodes involved in the foot print area and the degrees of freedom of the rim, which has to be linked to the hub.

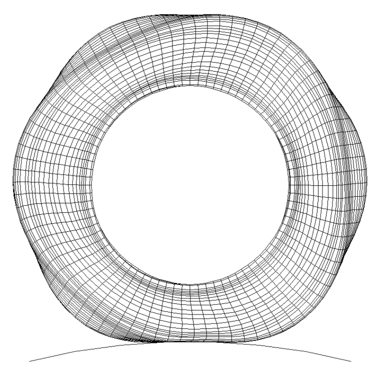

Fig.4: Example of mode shape, which occurs at 131.4 Hz.

The reduction of the degrees of freedom can be made, once known the stiffness matrix and all the applied loads, through a rearrangement of the full stiffness matrix. This procedure uses no approximations. The nonlinear response, due to the materials and the contact between tread and road surface, which occurs during the static pre-analysis, has been defined as pre-load prior to the elimination of the degrees of freedom. As a consequence it has took into account the

stiffening behaviour by pre-loading in a geometrically nonlinear analysis: stress stiffening effects has been included when the rearranged stiffness matrix was created.
While the reduced stiffness matrix uses no approximations, the mass matrix needs more degrees of freedom than just the ones, which refer to the nodes, involved into the contact patch. Therefore some eigenmodes amplitude was restrained as additional degrees of freedom: in this way the choosing of the retained degrees of freedom is no longer critical in order to get an accurate mass matrix representation.

3.2.2 The brush model: rolling-tyre. Discretisation of the tread pattern

The road unevenness excites the tyre dynamics, and slipping phenomena in the contact patch can arise. For the simulation of these phenomena in the foot print area, in lateral and tangential directions, a discrete brush model has been implemented. This model takes into account the main characteristics of the tread: slip stiffness, pitch sequences and size of the contact zone.

The discrete brush model is represented as individual elastic elements linked to the tyre belt. Tangential and lateral slip velocities can result from the tyre dynamics, and consequently the tread elements will have a shear deformation. The maximum deformation of the tread elements is limited by the friction coefficient between the tyre and the road.
Nevertheless the values of longitudinal and lateral slip, due to the roughness of the road, are pretty small; therefore all brush elements in the contact patch adhere to the road surface.

The elastic forces, which arise in the contact zone, depend on the slip stiffness, in tangential and lateral directions, and the shear deformation.
These forces are function of the contact patch size, the tread pattern and the elastic modulus of the tread compound.
In particular the size and shape of the contact depend on the tyre structure; the stiffness of the tread elements is affected from the pitch sequences of the tread pattern. Finally, the slip stiffness of the tread elements also depends on the tread compound.

According to the Lagrange method, we follow the tread elements from the entry to the exit of the contact. In fact, just the elements in contact with the road will generate forces during rolling. The tread elements will enter and leave the contact patch with a law that depends on the pitch sequences.

In this case, the tread has been discretized with *"n"* elements and their deformation and position in the contact patch is considered only at discrete time intervals. For example, in the tangential direction, at a time interval Δt, the position of the tread element increases by Δs, and the deformation by Δu. Denoting with V_r the rolling speed, the following relationships subsist:

$$\Delta s = V_r \Delta t$$

$$\Delta u = V_{sx} \Delta t$$

where V_{sx} is the slip velocity.
For the generic tread element i in the contact zone, the position and the deformation change during the rolling:

$$s_i (t + \Delta t) = s_i (t) + \Delta s$$

$$u_i (t + \Delta t) = u_i (t) + \Delta u$$

The deformation multiplied for the tread element stiffness C_{cpi} gives the local longitudinal force: the sum of these local forces over the contact patch gives the total longitudinal force at a time t :

$$F_c = \sum_{i=1}^{n} C_{cpi} \cdot u_i$$

The tread elements stiffness C_{cpi} has been determined by an internal Code (user subroutine written in Fortran and implemented in ADAMS), that is able to calculate those stiffness directly analyzing the tread pattern in the contact zone. The same procedure applies to the tangential force to evaluate the lateral force. In this case, the tread element deformation is due to the slip velocity V_{sy} in the lateral direction.
Therefore the simple modal model can simulate the rolling conditions when linked to the discrete contact model. In fact, the modal model reproduces only the eigenfrequencies and eigenmodes of the non-rolling tyre. All the dynamic effects affected from the forward velocity are neglected, such as, for example, the increase of the first longitudinal mode relative damping with the velocity of the of the tyre

closed to 35 Hz. As well known, the relative damping variation depends on the slip phenomena in the contact patch.
For these reasons a discrete brush model (fig. 5) has been used as tyre-road interface of the modal model.

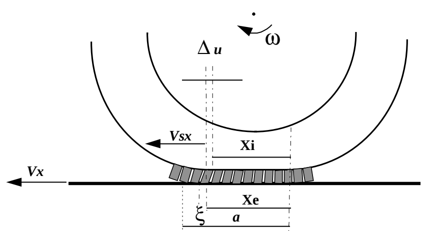

Fig.5: discrete brush model

3.3 Vehicle

The results presented in this paper are obtained using a fixed hub (cleat test) or a simple single-suspension (running on uneven road). However we can join the flexible body in each way to the chassis (through rigid joint or taking into account of hub compliance) of a full vehicle model.

4. Component mode synthesis

The following chapters describe how the flexible body is taken into account in the ADAMS environment

4.1 The CMS approach (Component Modes Synthesis)

A brief review of dynamic reduction techniques in FEA code follows: this section will present Guyan Reduction, Generalized Dynamic Reduction (GDR) and Component Modal Synthesis (CMS).
Let's define the *f*-set, the degrees of freedom which are unconstrained (free) in dynamic analysis. These are what remain of the *g*-set (all of the structural, or grid, degrees of freedom) after removal of the *s*-set (degrees of freedom eliminated by single point constraints) and the *m*-set (degrees of freedom eliminated by multipoint constraints). The *f*-set equations of dynamic equilibrium are

$$M_{ff}\ddot{u}_f + B_{ff}\dot{u}_f + K_{ff}u_f = P_f \qquad (1)$$

In dynamic analysis, we often have more finite element data than is necessary to obtain adequate estimates of dynamic behavior. We may reduce our solution set by partitioning the *f*-set into the *a* (analysis)-set and the *o*-set (omitted degrees of freedom):

$$\begin{bmatrix} M_{aa} & M_{ao} \\ M_{oa} & M_{oo} \end{bmatrix}\begin{Bmatrix} \ddot{u}_a \\ \ddot{u}_o \end{Bmatrix} + \begin{bmatrix} B_{aa} & B_{ao} \\ B_{oa} & B_{oo} \end{bmatrix}\begin{Bmatrix} \dot{u}_a \\ \dot{u}_o \end{Bmatrix} + \begin{bmatrix} K_{aa} & K_{ao} \\ K_{oa} & K_{oo} \end{bmatrix}\begin{Bmatrix} u_a \\ u_o \end{Bmatrix} = \begin{Bmatrix} P_a \\ P_o \end{Bmatrix} \qquad (2)$$

Though equally valid for non-superelement (residual structure only) models, eq. (2) is most familiar in the superelement context, with the *a*-set frequently referred to as the exterior, or boundary, degrees of freedom and the *o*-set, the interior.

For static only, we have two sets of equations in two unknown variable sets (*a* and *o*), allowing a unique (uncoupled) solution for each. Solving for the lower partition of eq. (2) without mass and damping leads to:

$$u_o = -K_{oo}^{-1}K_{oa}u_a + K_{oo}^{-1}P_o \qquad (3)$$

or,

$$u_o = G_{oa}u_a + u_o^o \qquad (4)$$

Note that the solution for the u_o interior degrees of freedom consists of two parts: the $G_{oa}u_a$ response to boundary displacements, and the u_o^o, or fixed-boundary solution to interior loading. The static condensation results in eq. (4) suggest a framework for approximating the coupled dynamic equations in (2).

A consistent way of presenting the various approximate dynamic reduction techniques is through the use of symmetric transformations.
We can introduce the transformation:

$$\{u_f\} = \begin{Bmatrix} u_a \\ u_o \end{Bmatrix} = \begin{bmatrix} I_{aa} & o \\ G_{oa} & I_{oo} \end{bmatrix}\begin{Bmatrix} u_a \\ u_o^o \end{Bmatrix} \qquad (5)$$

which is just the static condensation in matrix form. Eq. (5) and its time derivatives can be used to transform eq. (2), which, ignoring damping, is:

$$\begin{bmatrix} M'_{aa} & M'_{ao} \\ M'_{oa} & M'_{oo} \end{bmatrix} \begin{Bmatrix} \ddot{u}_a \\ \ddot{u}_o \end{Bmatrix} + \begin{bmatrix} K_{aa} & o \\ o & K_{oo} \end{bmatrix} \begin{Bmatrix} u_a \\ u_o^o \end{Bmatrix} = \begin{Bmatrix} P'_a \\ P_o \end{Bmatrix} \quad (6)$$

The advantage of eq. (6) is that the dynamic reduction techniques in FEM code can all be conveniently explained in terms of their corresponding u_o^o approximations.

Guyan Reduction simply assumes:

$$\ddot{u}_o = G_{oa} \ddot{u}_a \quad (7)$$

or, $\ddot{u}_o^o \equiv 0$. The resultant upper partition of eq. (6) can thus be immediately solved for the *a*-set degrees of freedom.

Generalized Dynamic Reduction uses approximate mode shapes to approximate the u_o^o degrees of freedom (Experimentally obtained mode shapes could be used just as well.). Component Modes offer a further logical extension by using the *o*-set eigenvectors to approximate u_o^o. We can simply write:

$$\begin{Bmatrix} u_a \\ u_o \end{Bmatrix} = \begin{bmatrix} I_{aa} & o \\ G_{oa} & \phi_{oq} \end{bmatrix} \begin{Bmatrix} u_a \\ u_q \end{Bmatrix} \quad (8)$$

The *q*-set has been introduced to represent generalized degrees of freedom in dynamic analysis. The *q*-set is included as a partition of the *a*-set, and the *t*-set partition of the *a*-set is used to more clearly identify the "total" physical degrees of freedom on the boundary, hence:

$$\begin{Bmatrix} u'_a \\ u_o \end{Bmatrix} = \begin{bmatrix} I_{tt} & o \\ G_{ot} & \phi_{oq} \end{bmatrix} \begin{Bmatrix} u_t \\ u_q \end{Bmatrix} = [\Gamma]\{u_a\} \quad (9)$$

where the "prime" in u'_a has been introduced simply to distinguish the previous physical boundary degrees of freedom only set from the *a*-set. Each column of the first partition of Γ represents the component's displacements due to boundary motion and are frequently referred to as "constraint modes." The modes of the second partition are, upon proper transformation, referred to as the fixed boundary, or "component" modes.[1] Use of the coordinate transformation (9) yields a set of compact, stiffness-uncoupled, equations of *a*-set dynamic equilibrium. Since the basis vector set of (9) is linearly independent, one possible solution technique is to first orthogonalize the set and then solve the resulting uncoupled equations.

4.2 FEA data conversion to Adams interface

4.2.1 Integration of component flexibility in ADAMS

A high level goal when implementing flexible bodies in ADAMS was that a flexible body could be integrated into a mechanism in a way similar to a rigid body and interact with the mechanism through ADAMS joints and forces.

Early in the development cycle, the need for Component Mode Synthesis became evident. Attempts to model the effect of attachments to the flexible body using only component eigenvectors required an extremely large number of eigenvectors to be considered. While the ADAMS implementation of modal flexibility is general enough to accept any kind of mode shape, the FEM code interface has been set up to export the computationally-determined component modes.

Though *c*-set degrees of freedom are provided in component modes synthesis, they've typically not been used when generating modes for input to ADAMS/Flex. (Though their use holds promise.) Since the resulting simplification yields the familiar Craig-Bampton modes, this section will refer to them as such. There have been certain challenges however:

1.) Embedded in the Craig-Bampton modes are 6 rigid body modes. Since ADAMS provides its own large displacement rigid body motion, these modes need to be removed from modal basis.

2.) The constraint modes partition are static correction modes and provide no information about the resonant frequencies of the degrees of freedom that they provide. An ADAMS user needs to have information about the frequency content contributed by a flexible body mode so that response in these frequencies may be

---

[1] Free-free modes are allowed in component modes via *c*-set degrees of freedom. The transformation:

$\phi_{oq} = \phi'_{oq} - G_{oa} \begin{Bmatrix} o \\ \phi_c \end{Bmatrix}$ eliminates redundant singularities.

controlled to ensure numerical integration robustness.

3.) Craig-Bampton constraint modes can not be safely disabled without imposing an unacceptable constraint effect between the boundary degrees-of-freedom.

Orthogonalizing the Craig-Bampton basis of modes eliminated all of these problems. All modes get an associated frequency and the rigid body modes show up as zero frequency modes and can be easily disabled. A user can choose to enable or disable modes for the dynamic simulation on a mode-by-mode basis.
Let's start to describe of how inertia, stiffness, damping and mode shapes of the flexible body are handled in ADAMS.
Inertia:
The mass matrix of a flexible body in ADAMS is not constant. As the flexible body is deformed, the center of mass shifts, and the inertia tensor changes. These effects are accounted for in ADAMS by formulating the mass matrix in terms of inertia invariants which are computed in a pre-processor. The large translational DOFs, the large rotational DOFs and the modal DOFs are coupled through this mass matrix. This is the mechanism by which spinning gives rise to deformation, etc.

Stiffness:
The generalized stiffness from the finite element analysis is diagonalized and used directly by ADAMS.

Damping:
ADAMS uses modal damping specified separately by the user as a fraction of critical damping. Damping can be specified on a mode-by-mode basis. Users are encouraged to use damping to control modal response. In other words, it is recommended that rather than disabling a mode, because it is assumed to lie outside the frequency range of interest, that the mode should instead be critically damped. This will eliminate the dynamic response of this mode while allowing ADAMS access to it to satisfy boundary conditions.

Mode shapes:
After using the mode shapes to compute the inertia invariants, the modes do not contribute in their entirety to the ADAMS simulations. Only a subset of each mode shape is passed to the ADAMS solver, the subset that corresponds to those nodes where connections have been made or where forces are being applied. This allows the solver to satisfy boundary conditions at connections and to project the applied load on the mode shapes.

4.2.2 Used FEA-ADAMS Interface
The used interface is based on specific ABAQUS instructions and an external translator utility function. After ABAQUS run, the provided output (.fil) contained generated superelement data for the flexible component,; it has been used a good beta version of Abaqus translator (By HKS) that converts this output into a form suitable for ADAMS.
Output quantities for each superelement include:

- Node data
- Element connectivity
- Constraint data
- Physical mass
- Modal mass and stiffness
- Component modes

The above allows a complete characterization of the mass and stiffness properties of a part in terms of its modal components (and, of course, physical mass), as well as graphical display of the part itself (via grid and element data) within ADAMS.

ADAMS uses a special flexible body description file called the Modal Neutral File (MNF) to communicate with a variety of Finite Element Programs. It has been used a beta version of a translator (by HKS). The translator extracts node locations, element connectivity, nodal mass information mode shapes and the corresponding generalized mass and stiffness from the .fil file and deposits this information in the MNF. In addition to formatting the MNF the routine provides the orthogonalization of the component modes
Moreover it has been used a MDI toolkit: a set of library functions suitable for reading, modifying and writing the MNF platform: the User can modify the original .mnf file performing mesh simplification and computing the inertia invariants.

4.3 Next improvements
The FLEX_BODY in ADAMS has deformations that are described via a superposition of mode shapes. In the current implementation, the modes

that make up the modal basis of the FLEX_BODY are assumed to
have been obtained by linearizing the flexible body about an unstressed configuration, as is customary in linear finite element analysis.
This implementation is obviously a limitation for users of nonlinear finite element analysis codes like ABAQUS. An ABAQUS user is likely to wish to linearize the nonlinear finite element model about an operating point which is different from the undeformed position. In this particular case, as the tyre comes into contact with the ground, it reaches a state of nonlinear deformations which ABAQUS can use as an operating point for the linearization.
The source of this limitation is that a linearization about a stressed state contains an associated modal preload which currently the MNF format does not accomodate and ADAMS/Solver does not account for.
The following ingredients are missing to allow ADAMS and ABAQUS to correctly communicate and process flexible components represented by modes obtained in a stressed state:

1. ABAQUS must export node locations to the MNF that correspond to the deformed locations of the nodes, not the input configuration;
2. The MNF format must be enhanced so that it can account for generalized forces associated with the deformed configuration and ABAQUS must compute these forces and store them in the MNF;
3. The ADAMS/Solver must be enhanced to account for this preload.

Note that any future removal of this limitation will probably keep a requirement that the linearization is performed at an operating point that corresponds to a static equilibrium.
To overcome this limitation, the current simulation in FE is done accordingly to what is described in section. This means the Tyre model goes through the following sequence of calculation steps:

1. inflation at the nominal pressure;
2. squeezing against a rigid surface which represents the road;
3. static concentrated loads on contacting nodes based on resulatnt forces coming from 2

At this point the Mode extraction is requested. The model is ouput in the undeformed configuration shape, with a linearized stiffness corresponding to the one at the end of step 3.

5 Adams implementation of the sub-models

5.1 Flexible Body

In order to implement into ADAMS the Pirelli Flexible Tyre model, it has been developed a first release of a customized ADAMS/View interface, including macro and command files.
From the mnf file (after fil2mnf routine), ADAMS reads all modal data to build up a Flexible body, but it doesn’t read the Ids of the interface nodes. Providing an external file which contains the hardpoints (nodes) list of the Flexible Body, the implemented macro automatically creates markers on nodes (hub and foot-print area) and all the objects for Road-Tyre interface: dummy parts, joints, motions, impact forces, state variables, design variables, measures, etc.
A modified Flexible Body dialogs box (see fig. 6) permits the User to select the Node list for the Flexible Tyre and to decide where to attach the Tyre (ground, chassis, etc.)
In order to get kinematics information from ADAMS analysis and to correctly define the brush model of the tyre an equivalent rigid tyre (having equal mass and inertia information, location and orientation) has been created. It is joined to the chassis through a Revolute joint plus a Motion or an Applied Torque, in dependence of type of the performed analysis (deflection, cleat tests, running on uneven road, etc.).
The resulting model after the execution of the macro is represented in figure 7: it is possible to see the rigid part coaxial with the flexible body, the markers in the footprint, the forces and motions applied on the markers.

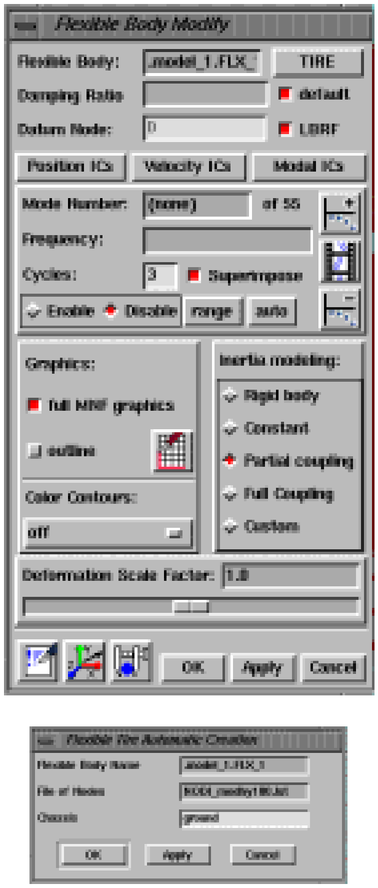

Fig.6: customized dialogs box for the automatic creation of the Flexible Tyre Model

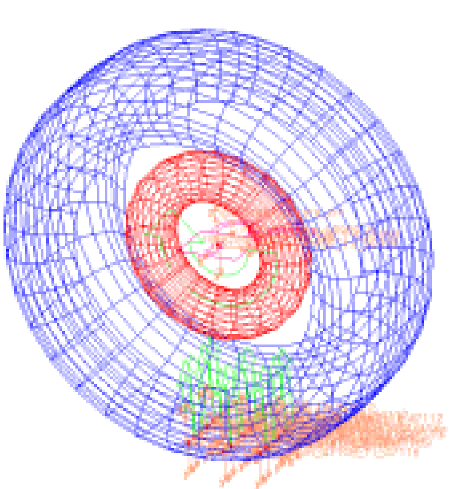

Fig.7: flexible tyre model after the execution of the macro

The impact forces applied on each hard point of the footprint area drive the deflection; the user can decide the deflection time and the deflection amplitude.

5.2 Brush model

To better manage the brush-model behavior, earlier described, we have written an user Adams variable subroutine which receives as inputs the current angular speed of the tyre, the rolling radius and the speed of the hub and returns the global longitudinal slip force. This one is correctly splitted on the nodes of the footprint, moreover the resultant torque (force time rolling radius) is applied on the hub of the equivalent rigid tyre part. In this way it is possible to calculate the 'rigid' contribution of the tyre to the variations of the slip. It is useful to simulate acceleration/deceleration maneuvers of the vehicle.

5.3 Cleat test simulations

The cleat tests simulations are carried out as in the Pirelli comfort approach. The basic curves are read from external files as splines and they provide to excite the flexible tyre.

6 Models Validations

6.1 Generality

It has been modeled a 195/65 R15 tyre size.

The below paragraphs show the different models validations: at first the proposed tyre model static and modal validations, the

The following paragraphs show the validation of the model running over: a) single obstacle, b) single obstacle, c) real road.

6.2 Static and modal validations

In the figure 8 there is the comparison between the static deflection curves obtained by Abaqus and ADAMS.

In figure 9 and 10 there the eigenfrequencies calculated in Abaqus and ADAMS with different boundary conditions. In both the simulations the hub of the flexible body is constrained, then in the case of the figure 9 the nodes of the footprint are free, in the case of the figure 10 the nodes are constrained.

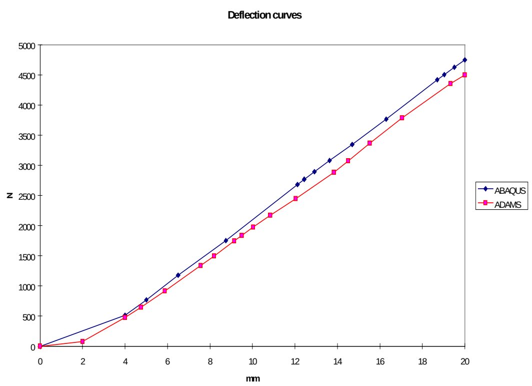

Fig 8: comparison between static deflection curves obtained by Abaqus and ADAMS

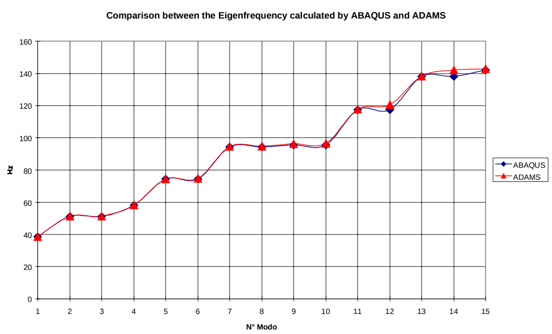

Fig 9: comparison between eigenfrequencies obtained by Abaqus and ADAMS. Footprint nodes ***free****.*

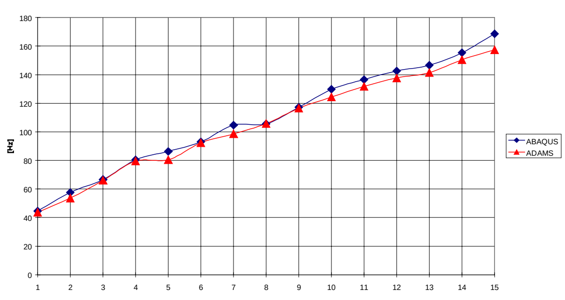

Fig 10: comparison between eigenfrequencies obtained by Abaqus and ADAMS. Footprint nodes ***constrained****.*

It is possible to see the good correlation between the two approaches (Adams, Abaqus).

6.3 Single obstacle

The obstacle is a single triangular obstacle high 15 mm and long 20 mm.

The simulation has followed the "basic curve " approach.

Figures 11 and 12 show the experimental data, i.e. the basic curves: vertical and longitudinal force variations in fig. 5.5, and Rolling radius variation in fig. 5.6. Those data refer to a vertical load of 450 kg.

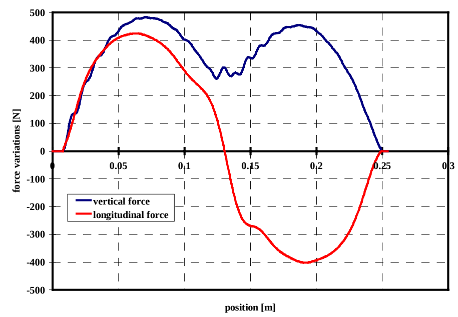

Fig.11: vertical and longitudinal forces at 425 Kg (basic curves measured at 3 km/h)

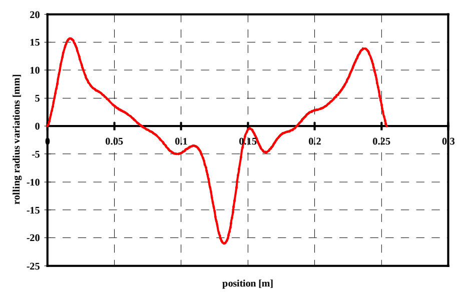

Fig.12: rolling radius variations at 425 Kg (basic curves measured at 3 km/h)

Figures 13 and 14 report the Adams simulation and experimental results, regarding the cleat test at fixed hub and at 30 km/h.

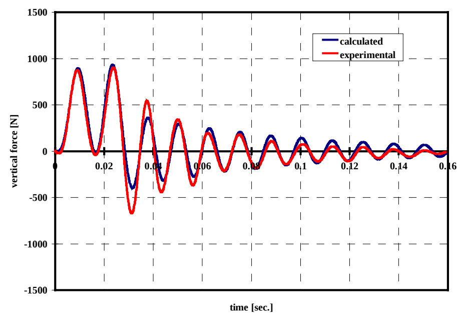

Fig.13: experimental and calculated vertical force at 30 km/h

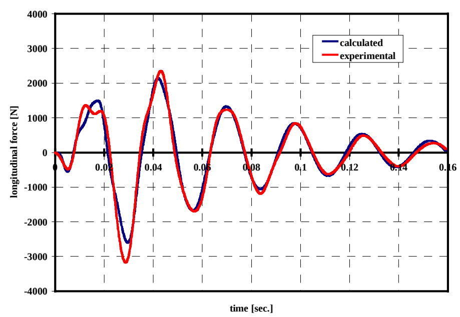

Fig.14: experimental and calculated longitudinal forces at 30 km/h

The experimental and the simulated results are comparable.

6.4 Running on uneven road

The simulation has been performed linking the flexible tyre to a simple mono-suspension with a lumped mass of 400 kg. In this case the experimental data are not available. In figure 15 is shown an example of the results of the simulation: there is the radial acceleration at the hub.

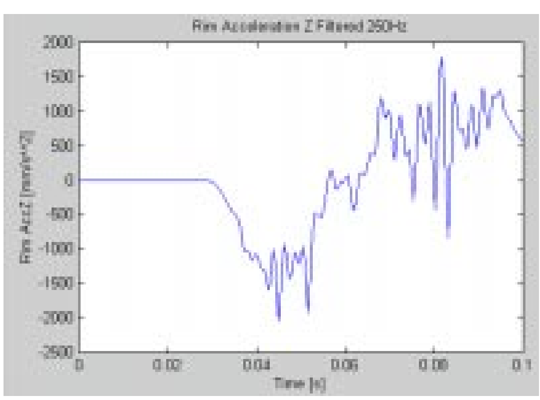

Fig.15: rim acceleration in Z direction in road noise simulations.

7 Conclusions

An automatic procedure to create a modal model tyre is presented. The linking of this model with a multibody vehicle model is very simple

The MNF format must be enhanced so that it can account for generalized forces associated with the deformed configuration and ABAQUS must compute these forces and store them in the MNF.

A new extended 3D contact algorithm is currently under development at MDI; it detects the compenetration between ground irregularities and the Flexible Tire.

The model has to be fully validated for the road noise simulations; at the moment no experimental data are available in order to do a comparison.