FEATURE-BASED GEOMETRIC MODELLING AND ANALYSIS OF MULTIBODY MECHANICAL SYSTEM BEHAVIOUR

D. PLAY*, J. F. RIGAL* and Tamás ENDRŐDY**[1]

* Laboratory (CASM) Conception et Analyse de Systèmes Mécaniques,
INSA de Lyon
** Laboratory CASM and Institute of Machine Design
Technical University of Budapest
H–1521 Budapest, Hungary

Received: Jan. 31, 1995

Abstract

The aim of this paper is to present a modelling system and an application to analyse the mechanical behaviour of a gear-box. Designing a new type of gear-box, there is a real need for the concurrent engineering activities of the designers of geometric modelling, mesh generation and mechanical analysis. In this system own developed contact types of finite elements (FEM) have been used.

The new system contains some programs of the earlier system ASSYM (CASM Lab.) for modelling the casing as the superstructure of a gear-box and its bearings as FEM contact-macroelements. In the proposed system the designers can use the modeller CAEDS (IBM Corp.) for feature-based geometric modelling and for generating the global mesh model of casing, the FEM analysis processor Mac/NASTRAN (MacNeal-Schwendler Corp.) and the new version of the N_SIM program (CASM Lab.) for modelling the pairs of gears and the bearings, furthermore the New Grid (N.G.) preprocessor (CASM Lab.). The N.G. can introduce the local mesh models of macroelements to the global mesh of gear-box casing.

Keywords: CAD, gear-box, geometric, static model, mesh, FEM.

1. Introduction

In this paper a geometric and mechanical modelling system and one of its applications is presented, in which one has concentrated on the relation between the feature-based modelling and the analysis of a gear-box. The mechanical system behaviour is analysed on the basis of the discretized components of a gear-box. It is focused on the different processes of the machine design. So this work is situated in the frame of the research on

[1]This work has been done during Tamás Endrődy's 4 months' stay (1st October 1994 – 31st January 1995) as a visiting professor in Laboratory CASM, INSA de Lyon. The publication is based on the cooperation between Laboratory CASM and Institute

concurrent engineering activities by an effective and fruitful cooperative planning.

The feature-based, parametrized, solid geometric modelling is examined through CAEDS functionalities. The mechanical modelling is eyed through the FEM processor Mac/NASTRAN and the former simulating program ASSYM, moreover the new processor N_SIM. The proposed system permits to calculate the real positions under charging and the load distribution in a gear-box, taking into consideration the technological constraints such as axial and radial clearances, the correction of toothed gearings or rolling elements.

After the presentation of the flowchart proposed for this modelling and the architecture of the software system, one can find the main ideas about the conception of geometric and mechanical modelling.

2. The Aim of Geometric and Mechanical Modelling

2.1 The Precedents

The earlier system ASSYM [1, 2] allows the statical computation of the positions of the components under charging and the load distribution in a gear-box. In the ASSYM one could use simplified beam 3D and shell FEM elements to model the gear-boxes of automobiles and helicopters. The main components of the gear-box model were the casing ribbed on its sides and around the housing of bearings, the bearings **Bi** (where $i = 1, \dots 6$), the pairs of gears **Gi** (where $j = 1, \dots, 7$), its elastic fixing pins to the chassis and with its links to the bridge (*Fig. 1*).

In the model the casing was used as a superelement of the gear-box, **Bi** and **Gi** were used as its macroelements. In this system own developed FEM element types: elastic, straight beam, circular beam and shell elements were used.

Using these numerical simulation tools some gains could be obtained in the machine design process. The rolling element loadings on one hand, gear and shaft positions on the other hand, are quantified. But the real difficulties remain in choosing the finite element types and in generating consistent geometric, mechanical and mesh models.

2.2 Modelling of a Gear-box and its Components

The design process of a new type of gear-box needs really concurrent design activities. In an integrated CAD/CAM/CAE system like e.g. in CAEDS one can find all the services which are necessary for the complex design process.

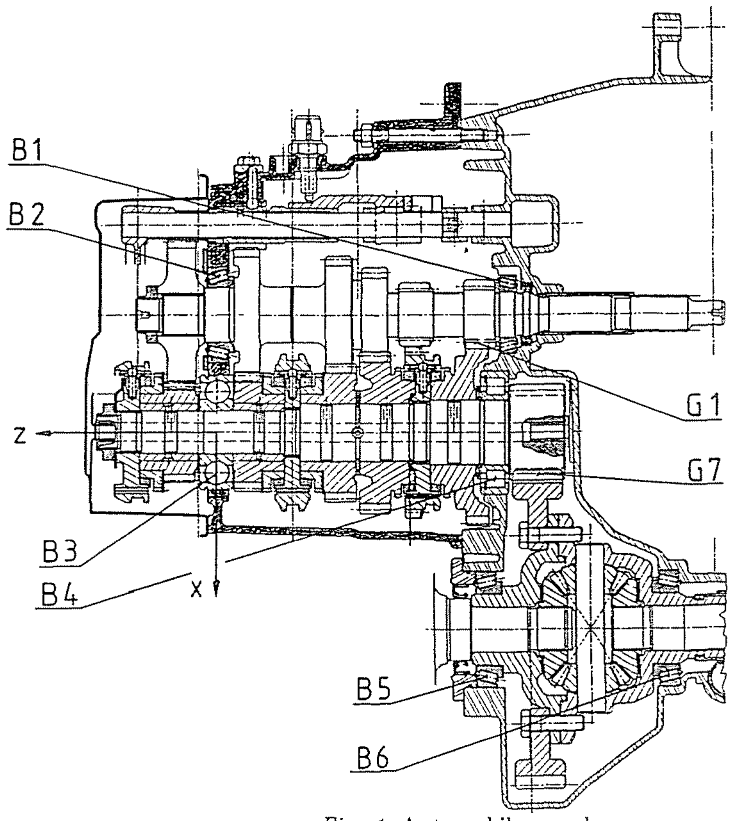


Fig. 1. Automobile gear-box

In such a system all the components: CAD, CAGD (Geometric design), CAE (Engineering, FEM) have the same or very similar model representation and the same data base, here Pearl Data Base, for solid modelling, FE mesh modelling, machining, rapid prototyping, etc. [6]. Besides, it has very effective and direct services: pre- and postprocessor to an advanced analysis FEM processor, like MAC/NASTRAN. Its main services can help to generate on the same model basis solid, surface and wireframe meshes in 3D and the Feature-Based modelling. The designer can use interactively a Feature Library which has parametric design possibilities, too.

Feature-Based and parametrical design can give new type of effectiveness in the *design–analysis–prototyping* process. After having a parametrized prototype for an object or a mechanism created together with all the characteristic details, called features, one can modify very effectively the existing prototype by modifying or changing its features or the connections between them. Pearl Data Base has good services for the components and features and for the linking tables between groups of different elements in the relational model and finally for the FE-types.

2.3 Linking Zones between the Components

A multibody mechanism has joining faces between its components by which the functions of mechanism can be realised, but for the achievement of these functions some constraints are also essential.

In a gear-box these constraints also work through these common faces, between the pairs of components: e.g.: between the casing and bearings, shaft of gears and bearings and between the elements of kinetic links. In this type of mechanical analysis the constraints and the functions of mechanism act through the common edges and mutual surfaces of the disjunct volumes of its components. That is why the joining faces have very important role in the linking process of the geometrical modelling and the static analysis.

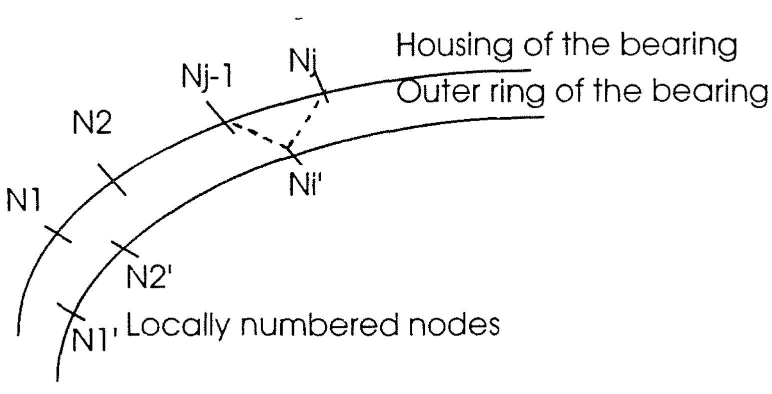


Fig. 2. Joining of the housing and the outer ring of the bearing

There are two possibilities for joining the nodes, e.g. $N(j)$ of the housing of casing and the nodes $N(i)'$ of the outer rings:

a) *in an optimal case*: the user can have the same nodes for $N(j)$ and $N(i)'$, in consequence that at first he has coded the mutual surfaces, and than one could freeze and label some of the common Mesh Curves nodes and Mesh Areas between this housing and the outer ring.

b) *in a few cases*: when the nodes of the casing-box housing for a bearing and the joining outer ring of the bearing have different nodes, one can apply also the **MPC** (Multi Point Constraints) instruction of the NASTRAN for joining two or more nodes $N(j)$ of housing to each node $N(i)'$ of the outer ring of bearing (*Fig. 2*). But the right method is to thaw and modify the earlier nodes of the common curves or the earlier Mesh Areas of the mutual surfaces between the above mentioned components.

In every case if one would like to join a contact or a macroelement to another (superelement), one must pay attention to make first the Mesh Areas and Curves for the joining surfaces then to freeze and label their nodes, finally he can generate the rest of the Mesh Areas and Mesh Volumes for the whole mechanism. So less difficulties can be found in joining their contact and macroelements.

2.4 Geometric Modelling

2.4.1 Solid Modelling In the CAD systems the most important design modules are: geometric modelling, construction, analysis, manufacturing, measuring, testing, etc. Solid modelling is a very effective type of the geometric modelling in which the form of the objects is represented in 3D by volumetric solid elements. A complex object can be built up of few primitives (3D solid elements) by a set of regularised operations: as union, subtract and intersection. This process is stored by the historical (C.S.G.) tree, so one can control and modify a really complex object very efficaciously. The surface topology of the object is handled by the BREP (Boundary Representations) which is generally added to the solid modeller.

The typical documentation can be generated by a lot of 3D/2D graphical and geometrical manipulations, so on the other hand the solid modeller knows all the services as well as other types of geometrical modelling systems. Now this sophisticated solid modeller is extended by a 3D parametrical design module by which one can build up a parametrized prototype for the designing object as well.

2.4.2 Feature-Based Modelling

Features have a lot of applications in various fields: design, geometric modelling, analysis and manufacturing. In our project several types of features are used such as blends, fillets, chamfers, ribs, bosses, slots, grooves, etc. However, there are great differences among the notions of the feature concepts used in different contexts. Usually we can find the common definition in the literature [7]: 'features represent the engineering meaning of the geometry of a part or of an assembly'. After SHAH [8, 9] one can define the form features as an entity which satisfies the following criteria:

- it is a physical constituent of a part or of an assembly,

- it is mappable to a generic shape,
- it has engineering meanings and predictable properties.

The feature concept can be considered from different points of view, the most interesting ones for us are geometric modelling, design and analysis and manufacturing:

- *Geometric modelling*: the features are groupings of the geometric and topological entities together with their reference to one another.
- *Design and analysis*: the features are elements used for creating an object or analysing the mechanical or other properties of this object.
- *Manufacturing*: the features represent shapes and technological attributes associated with manufacturing operations, materials and tools.

The features can have several attributes and we can use different feature taxonomies:

- explicit and implicit form features,
- surfatic- and solid features,
- in connection with the analysis and design: statical, dynamic and kinematic features.

On the base of the above mentioned notions, we can find (also in the CAEDS) the utilization of solid features in a parametrical way as it can give a very effective tool for modelling and for the analysis of a gear-box.

2.5 Mechanical Modelling

2.5.1 The Choice of a Finite Element Type The Finite Element Method (FEM) is now commonly used during the mechanical design process. This method, developed in the sixties, is the theoretical base of various softwares. So, one can perform static or dynamic analysis of the mechanical structures, such as fluid or thermical studies.

In the past few years, many aeronautic and automobile gear-boxes were analysed with the earlier version of the proposed system and many opportunities occurred comparing their experimental and numerical results. On the base of the conclusions of more than 25 gear-box meshes made up till now with ASSYM software, now it is possible to propose a good panel of elements (*Figs. 5, 7*). In this system one used own developed FEM element types: elastic linear and circular beam elements and 2D shell elements. Besides, one can verify that the mechanical and mesh models must be simultaneously constituted with the solid geometric model in all cases: in the case of solid, surface and wire-frame type elements.

A complex multibody system is generally composed of several basic components. Nevertheless, the mechanical and mesh model of a gear-box is complex enough: it has other than elastic and linear elements, e.g. in modelling a gear-box, the contact and the joining elements are also used. This way, we would like to make easier the meshing process, so we propose the separation of the structural and kinematic or joint elements. The concept of modelling by FEM macroelements makes it possible to solve the elastic and the contact problem separately as we can see in *Fig 3*. In the first step a reduced stiffness matrix can be obtained, thus in the second step less nodes of mesh must be computed.

In the dynamic analysis case, the problem is more difficult to solve, because of the eigenmodes of the structural parts. Much research has been made in this field [2, 4, 5], but finally we think that the same geometrical and structural approaches can be performed.

2.5.2 Generation of the Mesh Model of Components

The mesh generation of the structural parts must be based on the well-known procedures:

- *the free method*: automatic generation and refinement meshes by *h-method* and *p-method* and
- *the mapped method*: in which the user can create regulated meshes in all cases. In both cases the main criteria such as 'Mesh Volumes', 'Mesh Areas' and 'Mesh Curves' of CAEDS are used [4]. On the other hand Feature-Based modelling and design method is needed in this project [7]. The object is to join the geometric modelling concepts with technological or mechanical ones.

First let us see geometric modelling. In the case of a complex object we can find that solid modelling is a very effective geometric modelling method. One can build up and modify the model very quickly by the regularised set operation, furthermore by manipulating the structure of the historical CSG tree. Moreover one can use the parametrical design possibility, too, together with the feature-based design services of a C.A.X. system like CAEDS. One can collect all the features, little details of the complex object and some of them can be found in the Feature Library parametrized. On the other hand, the user can define parametrically all the features, details for the Feature Library determining the outgoing parameters joining them to the basic object (feature). The final result will be a modularly built up geometrical (solid) model of the complex part (e.g.: the casing of the gear-box) which can be modified very easily, parametrically, as well.

The second step is the generation of the FE mesh – in this case a mesh of solid finite elements which can be made:

a) – freely, automatically or *b*) – by a mapped method.

In the case of *a*), the *adaptive* generating FE solid mesh one can generally use a 3D tetrahedrical element type. In most of the sophisticated C.A.X. systems one can find the so-called *h-method* and the *p-method*, too. That is, the system can automatically regenerate the mesh in some environments if the error in the elements of this environment is bigger than the globally estimated error for all the elements. In the case of the *h-method* the same elements appear in the process of the mesh refinement, but their measure becomes less and less. In the case of the *p-method* in the adaptive mesh generation process the algorithm uses 2nd, 3rd and higher ordered elements instead of the linearly interpolated elements in the places where the regeneration of the mesh is necessary. In both cases of *a*) the user can control the mesh generation process, of course, to give the first density of nodes on the curves and area bound by the actual object.

In the case *b*) of the mapped method the user can make the mesh generation by hand one after the other, so one can use generally 3D Brick elements.

2.5.3 The Role of the Mutual Faces in the Modelling

In our case the main problem comes from the multibody character of the gear-box. The joining faces (linking zones) between the parts of the gear-box have very important role in the modelling as seen in *Paragraph 2.3.*

The process of analysing and defining the structure of the joining parts requires the following steps:

- defining the principal joining element pairs, finally the serial of the joining pieces of surfaces,
- starting from the casing one can find the first joining element pair (common surfaces) for the driving shaft, between the housing and the outer ring of the bearing,
- the next joining element pair is between the inner ring and the shaft of the bearing element pairs,
- throughout the transmission system one can find all the pairs of the joining elements up to the bridge.

Notes: In our macroelements we could model the bearings together with their outer ring, rolling elements and inner ring, and also the pairs of the gears as contact macroelements. Now we omitted the common surfaces between the shaft and the inner ring of the bearing, because we supposed that this is one solid element, without contact element between them.

One can define some other joining element pairs (common surfaces) among the supporting elements of bearings and between the parts of the kinematic links.

After all these the mesh generation process will have the following steps for every mutual surface:

- generating the meshes of *nodes* for the *mutual faces* (e.g. between the casing-box housing and the outer ring of bearings), these will be the *Mesh Areas* which are essential in modelling. These Mesh Areas have to be *labelled* and *fixed* in the CAEDS system as well;
- then one can generate the *Mesh Volumes* – by an arbitrary method – for all the disjunct objects, taking into consideration the Mesh Areas between them, which had been labelled and frozen before.

This information will be very important when the user of the CAEDS system would like to communicate with a user of another system like MAC/NASTRAN or the N_SIM analysis system. This means that one can have all the information which is necessary to introduce the local meshes of the contact elements as macroelements to the global mesh of nodes (to the superelement) in a very simple way, even if the mesh of the superelement was generated by another system.

All the contacts or joined zones must be simulated by several numerical elements, but from the technological point of view they belong to the same mechanical components as gears, bearings, etc. The preprocessor 'NEW GRID' (N.G.) is developed also by the Feature-Based concept. Three main technological components are defined by the N.G. preprocessor as input macroelements for the global mesh model of the gear-box:

- pairs of gears,
- rolling bearings and
- contact elements or kinematic links.

The functions of this program are the following:

- to generate numerical values from the catalogue data,
- to generate numerical models for the macroelements,
- to generate local mesh models for each macroelement and
- to link them to the global mesh of the casing, that is to the superelement of the gear-box.

Finally one has to define the environment of the gear-box: the nodes for suspending the motor (engine) and their fixing places, but these may as well be simplified to points (nodes), forces and moments according to the valid degrees of freedom (DOF).

3. The Proposed Method

3.1 The Flowchart of the Proposed Modelling Process

Returning to the statical analysis process, according to *Fig. 3* a flowchart is proposed. In connection with the geometric solid modelling and the mesh generation we studied [3, 6] and used the CAEDS (from IBM Corporation). In connection with the static analysing, in addition one used the FEM processor MAC/NASTRAN (from MacNeal-Schwendler Corporation) and the new processor N_SIM (of CASM), developed especially for analysing the contacts of the bearings and the pairs of gears in the gear-boxes. We are also developing a new preprocessor, namely New Grid (N.G.) for the NASTRAN and the N_SIM processors as well.

From the mesh generation point of view two separated stages are defined.

- The first one is concerned with the meshes of the structural parts such as the casing-box, the bearings, the shafts and some other supporting elements or the kinetic links, but one must take into consideration the common faces (Mesh Areas) between the pairs of the joining components of the gear-box. In this case we will have the global mesh of the gear-box.
- The second stage is concerned only with contact and joint elements. In this case we must create the local mesh models for the contact elements and then we must introduce them to the global mesh of the gear-box.

The global mesh of the superelement and the local meshes of the other components of the gear-box are built sometimes independently of their solid modelling and from each other, sometimes on other types of machines and on other types of CAX systems. Some parts of them can be used several times and in different situations. That is why one has to use very good interfaced format in each case.

In a designing office the engineers have to work in a close cooperation, thus one must concentrate on the effectiveness of the whole process: creating the geometric, mechanical, mesh modelling and the environment of the modelling, that is the loading, the fixing points and the constraints as well.

According to the proposed flowchart (*Fig. 3*) the user first creates the mesh model of the casing and other gear-box components together with the main data for loads and fixing of the nodes of the structure by the system CAEDS. Parallel with this process the user can give the local mesh models for the macroelements (gears, bearings and other contact elements) of the gear-box by the preprocessor New Grid (N.G.). So one can run the first job of the NASTRAN FEM process for having the reduced stiffness matrix.

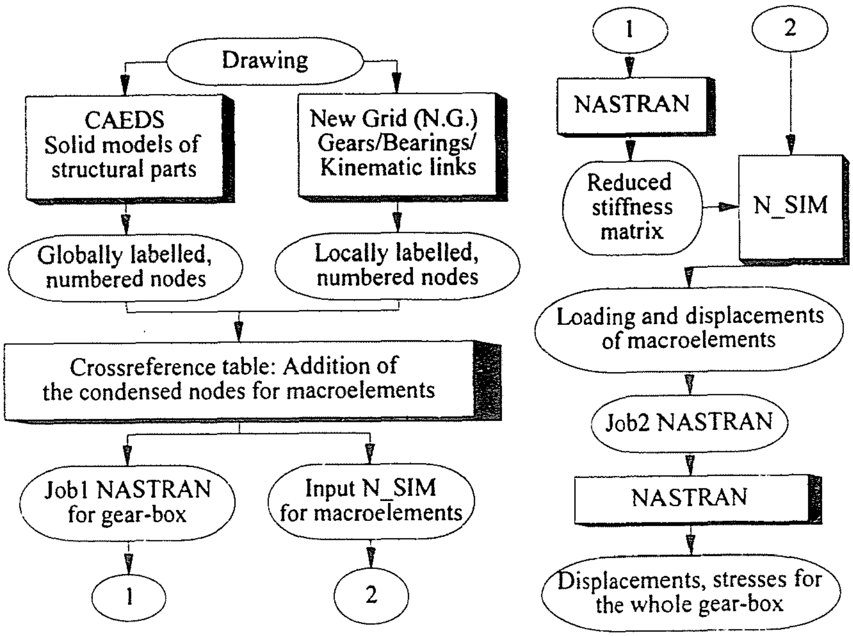


Fig. 3. The flowchart of the proposed process

The second step is the preparation of the input data for the simulator N_SIM: the models of the macroelements of the gear-box and the reduced stiffness matrix by which one can compute the loading and the displacements of macroelements.

Finally one can modify the first NASTRAN job by the new data: real loads and displacements of macroelements, and he can run the second NASTRAN job for the stresses and displacements of the whole gear-box.

Several types of gear-box loadings and positions can be tested fast with this system. Finally one can compute with the right loadings and displacements together with all the static characteristics of the whole structure.

3.2 The Architecture of the Proposed Software System

After analysing the CAEDS functionalities (*Fig. 3*) we can study the main components of the software system proposed for geometric and mechanical modelling of the gear-box by *Fig. 4*.

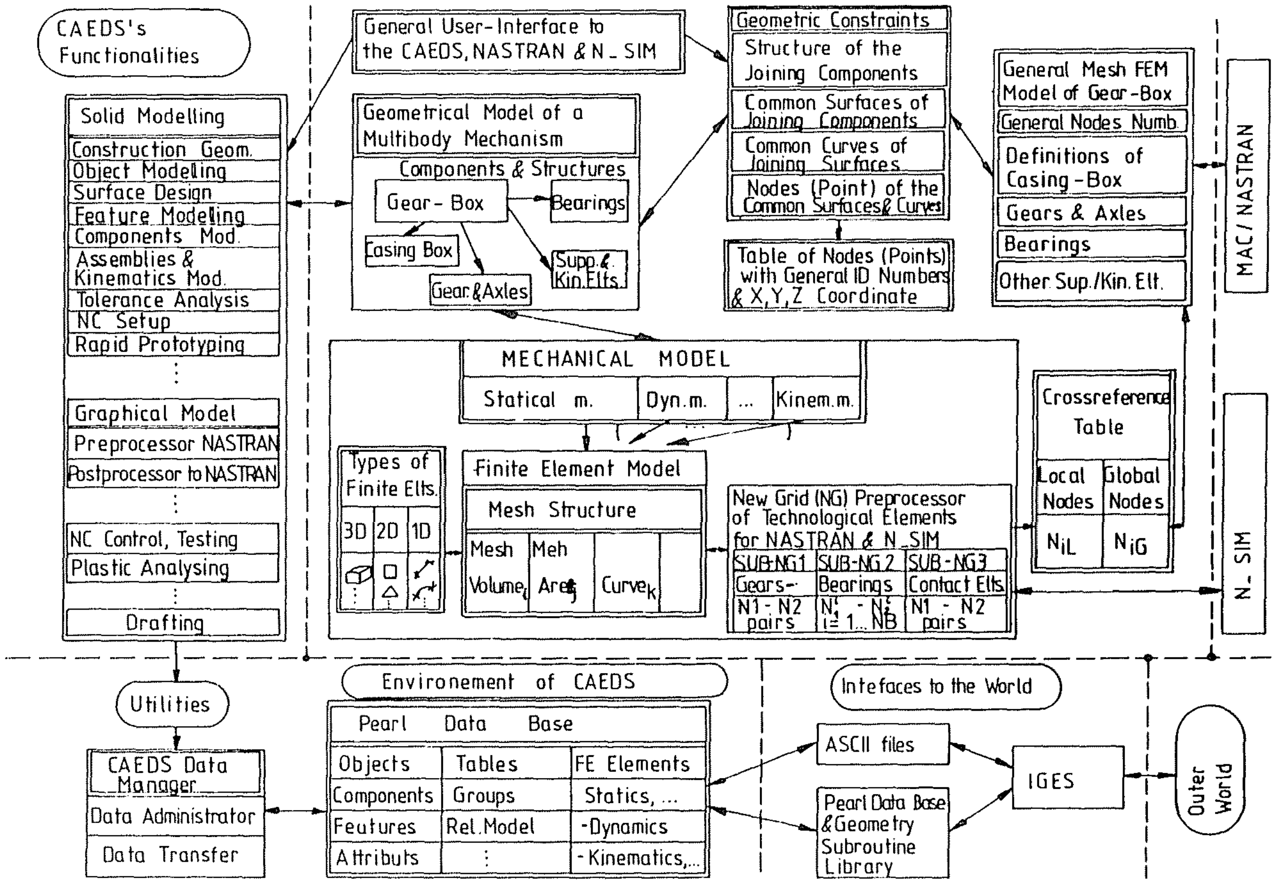


Fig. 4. The architecture of the proposed software system

General User Interface

The user of the geometric modelling (by CAEDS) can be the same person as the user of the mesh generation (by NASTRAN, New Grid) and the user of the mechanical analysis (by NASTRAN, N_SIM) or the users of these modules must have very intensive and direct communication among themselves. The General User Interface is the tool of this communication to help the use of all the modules of the software system proposed: it gives some proposals, formats and tables for using them more easily.

Geometric Modelling of the Multibody Mechanism

This module has solid models for all the components the gear-box:
- casing-box (namely superelement),
- contact macroelements:
  - pairs of gears and shafts,
  - bearings,
  - other contact elements and the kinematic links.

All these models can have features, parametrized as well, and geometric constraints between them.

Geometric Constraints between the Joining Parts

This is a structure of the joining components: namely a structure of the joining surfaces and curves throughout the whole multibody mechanism. These pieces of faces and curves shared with the components of the gear-box are very important tools of the functioning (working) of the gear-box. This way one can control all the geometric and mechanical modelling processes by these mutual faces and curves. On the other hand these geometric constraints will be very important in General Mesh Generation process, too.

General Mesh Model

First one must generate the mesh of nodes for mutual curves and pieces of faces between the components of the gear-box. After having the Mesh Curves and nodes, one can define the Mesh Areas for the mutual surfaces between the components of the gear-box and finally one can generate all the Mesh Volumes, that is the mesh models for all the components of the gear-box: for the casing box (Superelement) and for macroelements (pairs of gears and shafts, bearings and other auxiliary elements).

Mechanical Model

The statical modelling is being developed now, the dynamic and kinematic modelling will be added in the near future.

There are two software tools in this system:

–MAC/NASTRAN – through an interface to CAEDS,

–N_SIM (developed by Laboratory C.A.S.M.) – through an interface to CAEDS and NASTRAN.

Finite Element Model

This module contains the following ones:

- the mesh structure, based on the geometric constraints of the joining components of the gear-box,
- its disjunct components are the Mesh Volumes,
- the common surfaces between the neighbouring components are the Mesh Areas,
- the curves between two or more joining surfaces are the Mesh Curves containing mutual nodes.

Types of Finite Elements

There are many types of the F.E., namely:

- 3D, 2D and 1D (Dimensional) elements,
- simple (first ordered -linearly interpolated) or 2nd and 3rd ordered and interpolated elements,
- in 2D: there are 4 sided, 3 sided (special) elements,
- in 3D: there are solid brick, tetrahedral elements, etc.

Statical Model for FEM Analysis by NASTRAN and N_SIM:

This module has the name New Grid (N.G.) which has the following main characteristics:

- New Grid (N.G.) is a special preprocessor for NASTRAN and for N_SIM, the own developed static analysis program for a gear-box;
- by the N.G. preprocessor one can generate macroelements for the statical analysis with MAC/NASTRAN and N_SIM.

Pairs of gears and axles: this macroelement type has a pair of Nodes (N1–N2) - locally coded, which is a special contact element, *Bearings*: in this macroelement type the most important types of bearings are modelled with balls, cylindrical rollings and taper rollings, which contain multiplied times 2, 3, 4 pairs of contact elements, multiplied by (NB) the number of the rolling elements of the bearing. Finally this can have NB pairs of nodes: $(N1 - N2)$ i-times where $i = 1, \dots, NB$, – by this macroelement type all types of bearings can be modelled.

Contact elements: this macroelement contains a pair of contact elements, that is a pair of nodes: $(N1 - N2)$, for modelling any type of contact links of a kinematic or static element of a mechanism.

The N.G. preprocessor gives locally numbered nodes (pairs of nodes) which can be introduced to a globally coded mesh of nodes: e.g. the mesh generated by CAEDS or NASTRAN can be extended by locally identified pairs of nodes and finite elements. Naturally these macroelements have all the characteristics which are necessary.

Crossreference Table:

This is the tool of the identification and introduction of the locally coded nodes of the macroelements: the gears, bearings and other contact elements. This contains:

- all the important globally coded nodes of the superstructure,
- the same locally coded nodes of the pairs of gears and axles, bearings, and contact macroelement types joining to the casing.

Returning to *Fig. 4*, after one has introduced the models of the macroelements (technological elements) generated by the N.G. preprocessor, to the global FEM mesh model of the casing generated by the CAEDS, the whole FEM model can be processed by the MAC/NASTRAN FEM processor. The results of this job give the *reduced stiffness matrix* of the whole gear-box FEM mesh model. The Processor N_SIM can compute the displacements of the pairs of gears and shafts by the reduced stiffness matrix of the whole gear-box model. Together with these displacements as limit conditions and together with all the other data of the FEM mesh model, one makes the MAC/NASTRAN processor to run once again. The MAC/NASTRAN can compute all the statical characteristics important for the construction process of the gear-box as we could see in *Fig. 3*.

4. Application

4.1 The Objective of the Mechanical Analysis

The previous process has been applied to analyse the behaviour of an automobile gear-box with five ratios. The complete mechanical system is presented in *Fig. 1.* The object of the static analysis was to compute load distribution on bearings **Bi** and the position of the gears **Gj** with real external loadings.

4.2 The Discretization of the Mechanical System

4.2.1 Meshing the Casings

The mesh of the casing is shown in *Fig. 5.* Two graphical representations of the FEM mesh of the casing box are shown

1st: by solid and shell elements,

2nd: by bar and shell elements.

Beam and shell FEM elements with 3 nodes have been used for this application. Because of the complexity of the shapes only some parts have been automatically meshed. Some manual inputs have been necessary to obtain the final mesh. The reduced stiffness matrix has been computed with the NASTRAN FEM program.

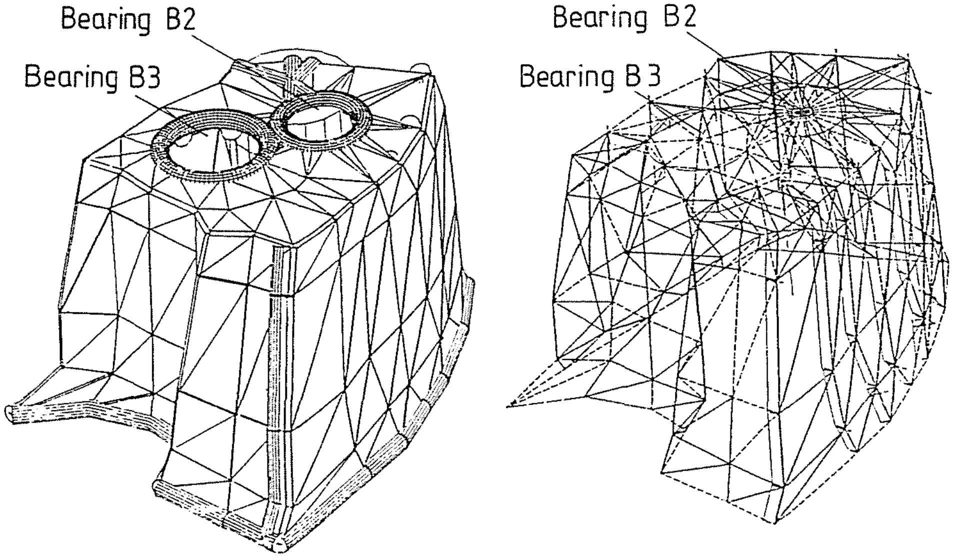
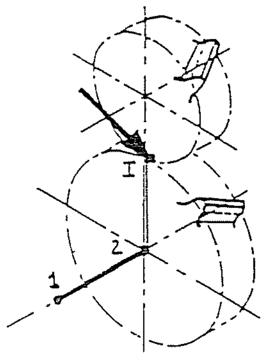


Fig. 5. Two graphical representations of the casing-box model

4.2.2 Discretization of Shafts and Gears

All the internal parts have been manually discretized. Shafts are modelled with beam elements and the deformations of teeth are neglected (*Fig. 6*). A contact point element is placed at the pitch point along the action line



1-2 : Beam element,
2-I : Non-deformable bar,
on I: Contact point.

Fig. 6. Shafts and gears model

of the gear. A non-deformable bar element joins this point to the beam section centre.

4.2.3 Rolling Bearing Modelling

Rolling bearings are discretized according to the rolling element number. *Fig. 7* shows the outer ring reduced to a medium line. On one hand, it is connected with the external node of the rolling element model. On the other hand, it is linked to the nodes of the bore surface with the help of contact elements. The rolling element model internal nodes are connected all together with the same point which is the centre of the beam section, modelling the shaft.

During the computation, real contact positions and orientations are calculated for each rolling element. Some models are presented in *Fig. 8*. Some input geometrical parameters are shown, such as the connection nodes ($N1$ and $N2$) of the element with the structure. Naturally material characteristics are also introduced in the model.

4.3 Results

Several numerical results have been obtained with this process. Only some examples are presented in this paper. Different loadings of the rolling elements are shown in *Fig. 9a*, rollings B2 and B3 of the *Fig. 1* are concerned, and 3 different ratios of the gear-box are noticed as case 1, 2 and 3.

For the same bearings, *Fig. 9b* represents the ring positions in the same cases. Three views are designed with a scale. The positions of the

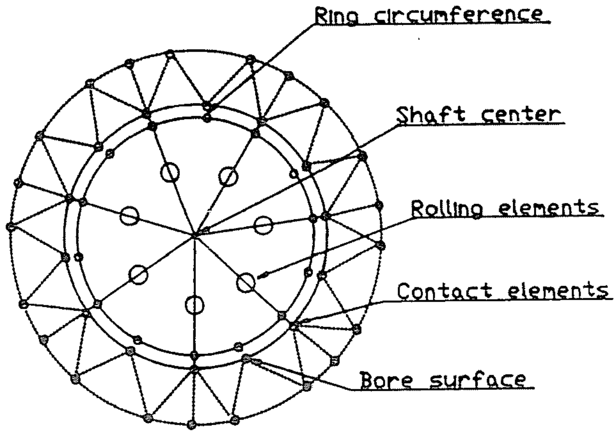
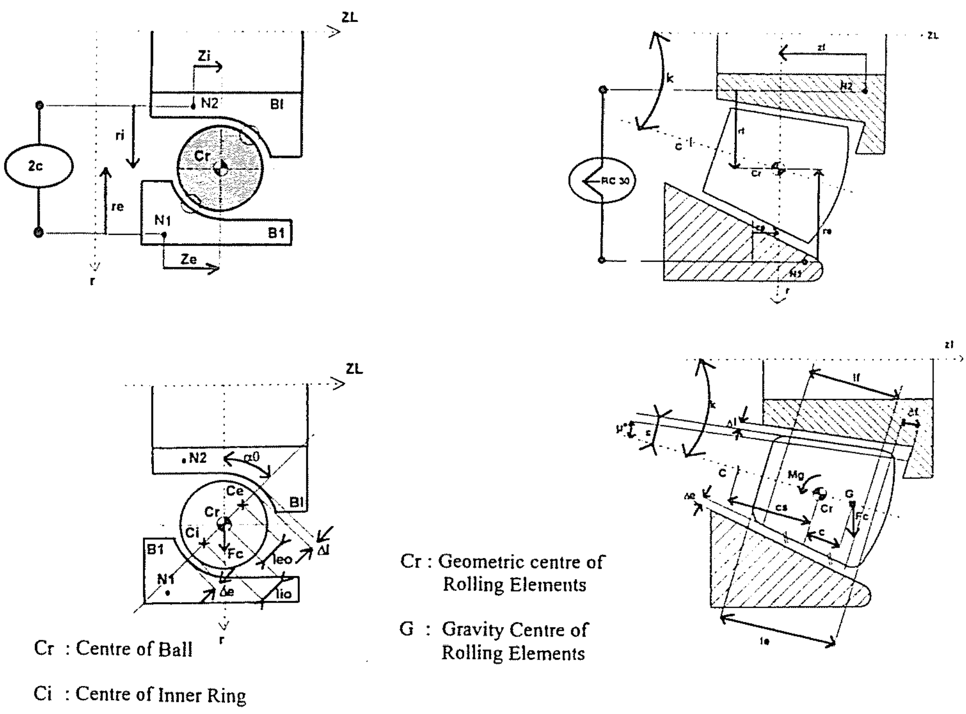


Fig. 7. Bearings' discretization



Ce : Centre of Extern Ring

Fig. 8. Rolling elements definition

shafts and relative positions of the gear pitch point are also calculated for the different ratios and different loadings of the gear-box. These results

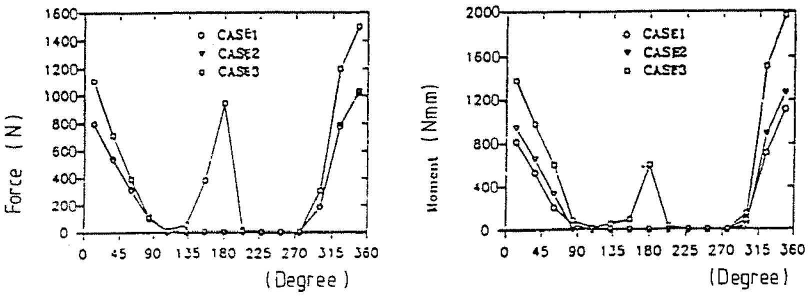


Fig. 9a. Load distribution on rolling elements of bearings: $B2$, $B3$

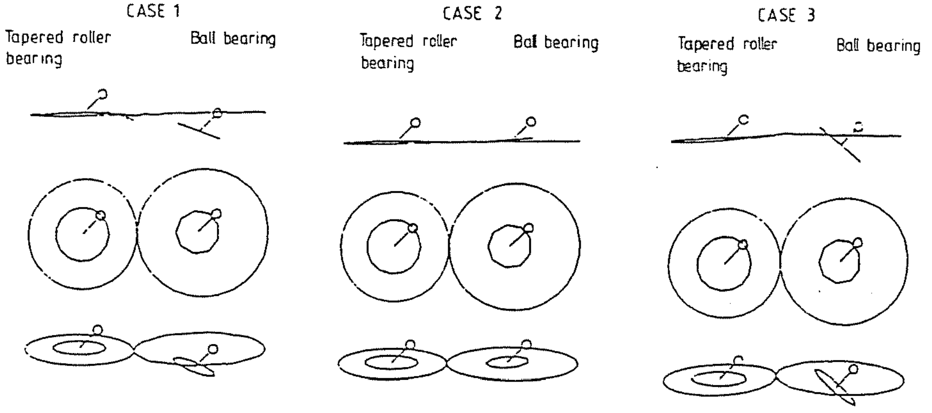


Fig. 9b. Some results: Ring positions of bearings $B2$ and $B3$

might be important to optimize the tooth profile without the help of expensive prototypes.

5. Conclusion and Future Developments

This simulating and analysis process is very useful for the designers developing a new mechanism in the Design Offices in the vehicle industry. These tools can reduce the design time, the number of iterations and prototypes in the design process. The process is very complex: it has a FEM programme system and a specific one, such as N_SIM, the later one must be used to perform the complete analysis.

Geometrical modelling appears as an important part of the engineers' job. So the proposed approach is based on concurrent activities and it is supported by the geometric model. The feature based geometric model is a good starting point. It is directly related with a discretized mechanical one. Machine parts and elements can be associated with basic features of a solid modeller. Later mesh can be automatically generated directly from the geometric data base.

The actual implemented software allows to join curves and 2D finite elements and also surfaces and shell elements. The new numerical developments are in relation between volumes and bricks or 3D finite elements. On the other hand from a mechanical point of view new processors are developed to perform a vibration analysis of the global mechanical system.

References

1. CHOI, I. S.: Notice du Logiciel ASSYM, Resp.: J. F. Rigal, Prof. D. Play, Labo. CASM, p. 70, 1992.
2. RIGAL, J. F.: L'Habilitation à Diriger des Recherches en Sciences Spécialité Mécanique, Jury: Prof. D. Play et autres, INSA de Lyon,Univ. Claud Bernard – Lyon I, p. 143, Mai 1994.
3. BLAJ, F.: Notice d'Utilisation du Module CAEDS- 'Graphics Finite Element Modeler', Resp.: J. F. Rigal, Prof. D. Play, INSA de Lyon, p. 74, 1994.
4. LIM, C. I. – SINGH, H.: A Review of Gear Housing Dynamics and Acoustics Literature, NASA Report N185148., T. M. 98_C_009, (N90_21387), 1989.
5. REMOND, D. – VELLEX, P. – SABOT, J. : Comportement Dynamique et Acoustique des transmissions pour engrenages, Synthèse bibliographique, publication de CETIM, 1993.
6. *CAEDS*, Graphics Finite Element Modeler (GFEM), User's Guide, IBM, 1990.
7. LAAKKO, T.: Incremental Feature Modelling: Methodology for Integrating Features and Solid Models, *Acta Polytechnica Scandinavica, Math. & Computer Scie.* Series N 63, Helsinki UT, p. 84, 1993.
8. SHAH, J. J.: Conceptual Development of Form Features and Feature Modelers, *Research in Engineering Design*, Vol. 2, pp. 93–108, 1991.
9. SHAH, J. J.: Assessment of Features Technology, Computer Aided Design, Vol. 23, 5, pp. 58-66, June, 1991.