MODELING AND FINITE ELEMENT ANALYSIS OF A COMPLEX HELICOPTER TRANSMISSION INCLUDING HOUSING, SHAFTS AND GEARS.

A. Bianchi - Stress Engineer - Transmission Dept.
S. Rossi - Stress Engineer - Transmission Dept.

AGUSTA
C. Costa - Italy

Abstract

The availability of very fast computers and the extensive use of 3 dimensional geometric modeling allow the solution of very complex shapes using the Finite Element Modeling technique.
As the calculation of Helicopter Transmission housing and gear deflection and stress is influenced by several interacting effects, Finite Element Analysis should include housings, shafts, bearings and gears.
The actual main case design of the EH101 Transmission has been studied using integrated Computer Aided Design and Finite Element modeling and analysis.
The EH101 Helicopter has a three engine, four reduction stage 5200 HP Transmission with a reduction rate of about 100.
The main case design including the II/III stage reduction gears has been analysed.
The activity covered the generation of the complex geometric CAD shape using CATIA exact solid modelling and the generation of the Finite Element model of the housing.
The mathematical model has been completed with the introduction of shafts, gears and a simplified representation of bearing effects.
Using the NASTRAN super-element approach, it has been possible to analyse separately each element and at the same time taking account of the interacting effects.
Using this technique, the deflection and stress behaviour of housing, shafts, bearings and gears can be predicted in the same integrated analysis.

Nomenclature

3D	Three dimensional
CAD	Computer Aided Design
CPU	Central Processing Unit
dof	degree of freedom
FEA	Finite Element Analysis
FEM	Finite Element Method or Finite Element Model
MPC	Multi Point Constraint
RAM	Random Access Memory
SE	Super-Element

Introduction

The Finite Element Analysis has been intensively used to calculate displacement, stress and strain behaviour of aerospace structure since the '70s.

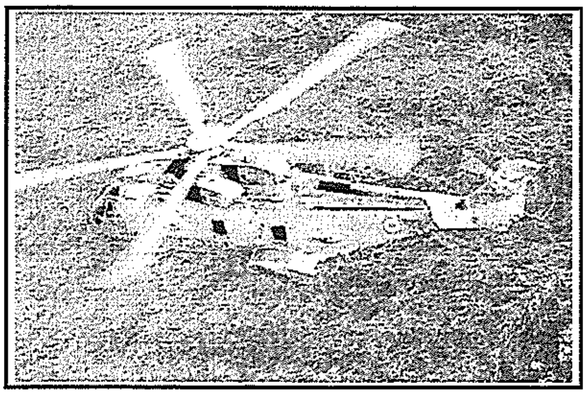

Fig. 1- Agusta/Westland EH101

Late in the '80s, the introduction and use of three dimensional element has enlarged the applicability of Finite Element Analysis from structures composed of a combination of elements with a one (linear) or two (plane) predominant direction to general three dimensional geometry.

The introduction in the '90s of very fast computers and the intense use of 3D modeling of complex geometric shapes for design purposes, allowed the start of finite element simulations of thick castings, and in particular transmission cases using three dimensional elements.

This combination of 3D geometric modeling and finite element analysis have been used in Agusta to study the structural response to external and internal loads of the main transmission case of the Agusta - Westland EH101 helicopter.

Geometric Modeling

The geometry of the EH101 main case is very complex due to its three engine transmission design.

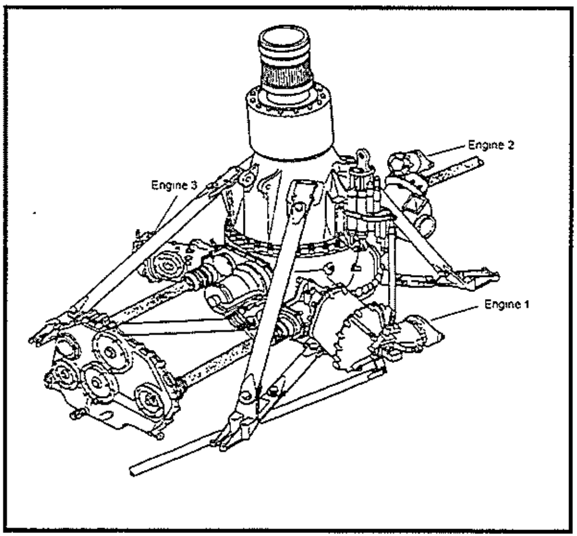

Fig. 2 - EH101 Transmission General Sketch

The reduction ratio and input and output R.P.M.s for each reduction stage are reported in table 1.

Table 1

Reduction Stage	Reduction Ratio	Input R.P.M.	Output R.P.M.
I	2.1923	20872	9521
II	2.3548	9521	4043
III	4.8709	4043	830
IV	3.8750	830	214

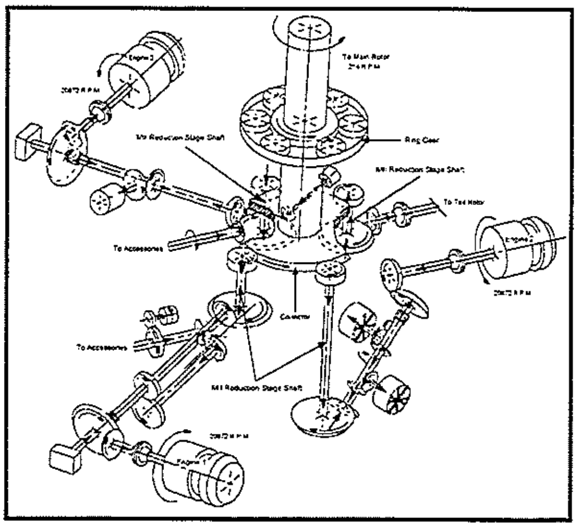

Fig. 3 - EH101 Transmission Gear Train Sketch

The CATIA exact solid 3D modeling of the main case has been a critical point and a very time consuming activity due to its complexity.

The geometry has been created starting from 2D drawings and the actual part.

A combination of solid CATIA modeling in connection with a large use of the surface modeler had been necessary to resolve the very complex shape geometry of the EH101 main transmission case.

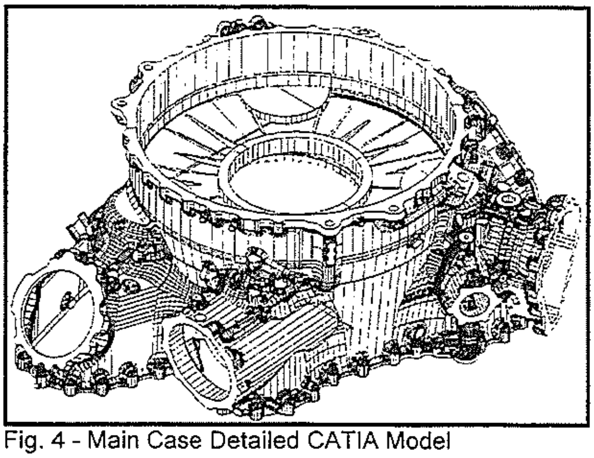

Fig. 4 - Main Case Detailed CATIA Model

The main case CATIA 3D model has been simplified (as shown on Fig. 5 on next page) to eliminate all the elements not relevant for the structural analysis and to reduce the overall geometric complexity.

Post processing or local analysis could be utilised to calculate the structural behaviour in these simplified areas.

In particular the features included in the following table have been discarded.

Table 2

Feature name	Reason
Bosses	Introduced directly in the FEM model
Pipeline (with the exception of the upper one)	Not structural
Liners	Introduced directly in the FEM model
Very small fillet	Too complex and not important for structural analysis

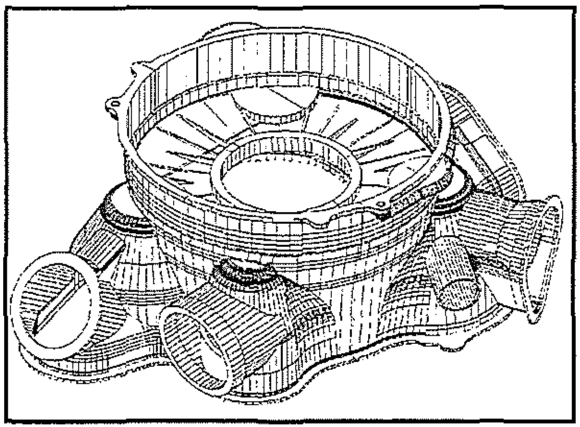

Fig. 5 - Main Case Simplified CATIA Model

Minimum dimensions have been used to take into account geometric tolerances.
More than 1000 surfaces have been used to represent the geometry of the EH101 transmission main case.

Geometry Translation

A mathematical model of the main case has been generated starting from a translation of the geometric CATIA model.
NASTRAN version 69 has been chosen as the solver and I-DEAS Master Series as the finite element model generator and post-processor.
The CATIA geometric model and in particular the surfaces that enclosed the main case volume, have been transferred into an I-DEAS geometric model. This model is identical to the CATIA one with the only exception of the partition into eight smaller sub-volumes (using appropriate split planes) to make easier the finite element generation.
This translation of the model between the two softwares is another very complex and time consuming task as very small surfaces, curves or curvature radii, totally acceptable for the CATIA model, have to be eliminated into I-DEAS, avoiding significant modification to the geometry, to allow rational mesh generation. Very small geometric element, if compared to the overall dimension of the object, means in fact a concentration of low dimension finite elements that could lead to model generation failure or excessive number of elements, and consequently nodes, in structurally not relevant zone.
Moreover, due to the different method of evaluating geometric coincident tolerance between CATIA and I-DEAS, a 10-15 % of the surfaces had to be corrected or regenerated into I-DEAS software as the interface was not able to translate them.
Furthermore a “non native” (translated from CATIA or other software) geometry is very hard to modify or recreate into the I-DEAS modeler.

Mesh Generation

A very controlled semi-automatic mesh generation, a well chosen overall finite element dimension, an as smooth as possible transition from coarse to more detailed mesh zone and the partition of the model into 8 sub-volumes have been used to make possible the creation of the finite element model of the EH101 main case.
This control is necessary as letting the mesh automatic generation software free, led to the impossibility of creating the mesh or to unreliable results.
Several iterative mesh generations had been necessary to minimise the node and element number and to reduce stretch and distortion from the basic shape of each element.
In this way it had been possible to represent the very complex geometry of the main case using “only” about 50000 tetraedric 10 nodes elements (the tetraedric element is a must due to the complexity of the geometry) and 120000 grid points (350000 dof ca.).
The overall dimension of finite elements is 17.5 mm with an imposed lower value where the geometry, that means surfaces, edges, thickness or curvature radius, is not compatible with this dimension.

The finite element model is really very close to the EH101 main case real geometry as the only significant simplifications have been introduced into CATIA model as shown on Fig. 6.

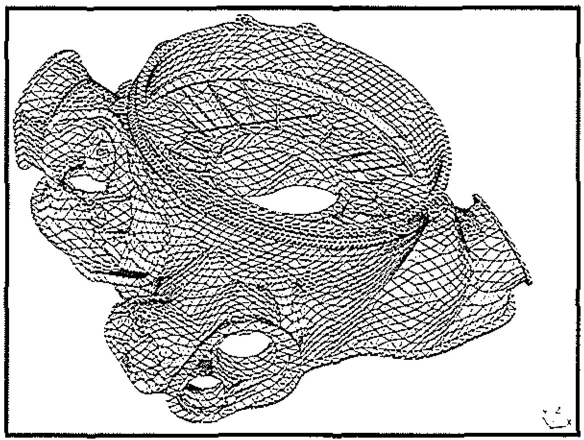
Fig. 6 - Main Case Finite Element Model

Mesh Assembly

As the main case response to external (from the connection to other component) and internal (from gears and shafts) loads is heavily influenced by the other transmission cases (lower and upper case in particular) and by the joint to the helicopter main structure, these elements had to be included into the analysis.

In this analysis the component to be better investigated is the main case, so structurally representative but simplified lower and upper cases and main helicopter structure connection struts have been used to reduce element and grid number.

In any case a more detailed Finite Element Model could be introduced later into the analysis.

The FEM of the dummy lower and upper case sare shown on Fig. 7 and 8.

For the upper case 650 elements and 6599 nodes have been used.

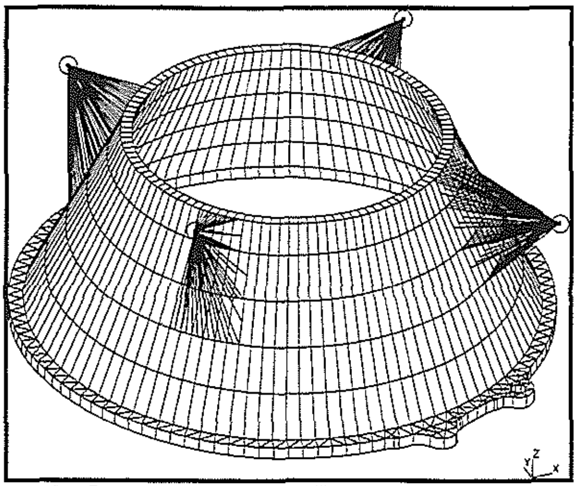
Fig. 7 - Dummy Upper Case Finite Element Model

For the lower upper case 4100 elements and 25378 nodes have been used.

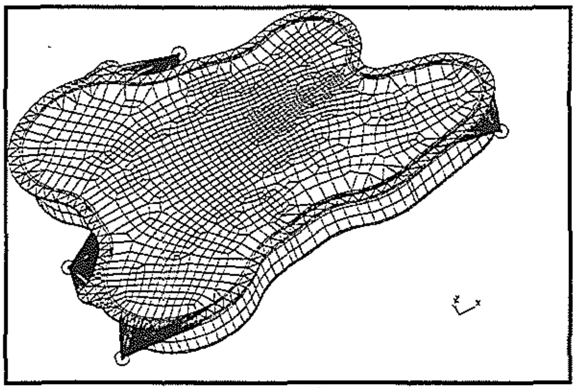
Fig. 8 -Dummy Lower Case Finite Element Model

The connection between main-upper case and main-lower case has been simulated using RBE3 rigid elements for the bosses and CELAS elements for stud bolts (in plane and axial stiffness) and pins (in plane stiffness). In-plane CELAS stiffnesses have been matched to assure the correct load sharing of torque load between the pins and the bolt friction.

This CELAS elements were also used to easily verify the transfer of load along the case connection flange.

The link between upper and lower case and the main helicopter structure connection struts has been modeled using RBE3 rigid elements.

Shafts and Gears

To analyse the interaction between case deformations, shaft deformations and gear behaviours, a detailed mesh using solid parabolic bricks and wedges of the shafts reported in table 3 has been generated and introduced into the mathematical model.

Table 3

Shaft	Elements	Grids
II/III stage reduction eng. 1	5454	7560
II/III stage reduction eng. 2	2470	4453
II/III stage reduction eng. 3	5772	7994
tail rotor	6840	10164
collector	6104	9488

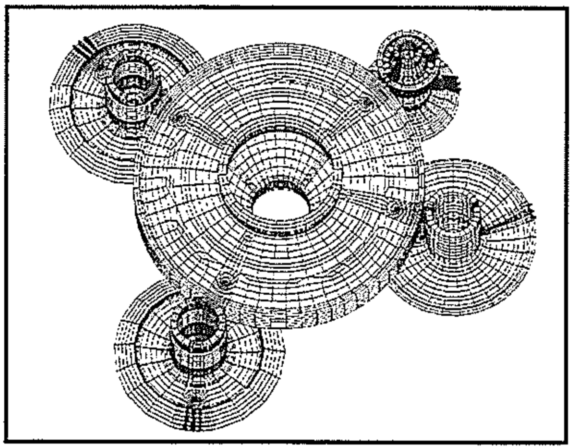

Fig. 9 - Shafts and Gears Finite Element Model

On every gear only three teeth (the mating one plus the adjacent two) have been modeled. The interaction between mating teeth has been simulated using MPC connection directed along the pressure line on grids lying on the contact line.
The connection of the mating gear teeth assure that torque introduced at pinion level of II/III reduction stage is transferred correctly to the collector.
Reaction forces of support bearings are then a consequence of the equilibrium under the two mating loads (gleason on II/III reduction stage pinion and helical on collector) and have not to be calculated.

Roller and Ball Bearings

To take account of bearing effects, shafts and cases have been connected to the main case using CELAS elements between the shaft and the case bearing reference point. Spring stiffness values have been chosen to represent the bearing stiffness (axial and radial for single line ball bearing, radial and flexural for roller bearing or duplex and triplex ball bearing) calculated from conventional analysis or taken from supplier information.

Complete FEM Model

The mathematical final dimension of the full finite element model is demonstrated by the following data summary:

- 26 COORDINATE SYSTEM
- 172724 GRID POINTS
- 1042 ELAS
- 10418 HEXA 20
- 1246 PENTA 15
- 51277 TETRA 10
- 39 RBE2
- 194 RBE3

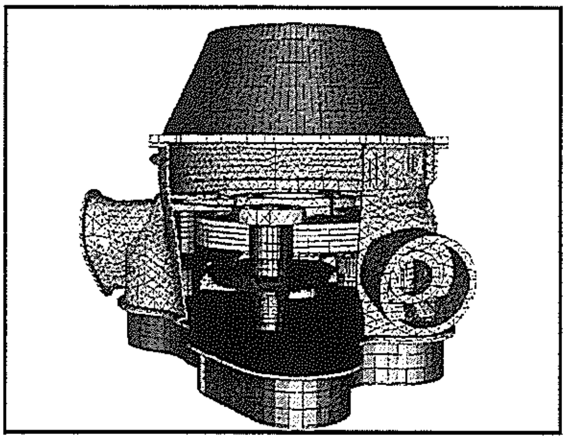

Fig. 10 - Complete Finite Element Model (broken)

Model Arrangement

The full FEM model has a very high level of complexity but each sub-element (main case, lower case, upper case, shafts, bearings, connection struts) has well defined and limited interfaces.
Cases are in fact connected by bolts and pins on the upper flange (main and upper case) and lower flange (main and lower case), shafts by ball and roller bearings (represented by

CELAS elements) and by contact lines on teeth (represented by MPC).
Main structure attachment struts are connected to upper and lower case using RBE2 elements on one side and restraint on ground on the other side.
For these reasons a NASTRAN super-element approach could be useful and has been used for the analysis.
A multi-level super-element tree has been chosen with a collector super-element interposed between sub-element and the residual structure.
The super-element tree has been organised as follows:

Table 4

	super element id
main case	11
upper case	12
lower case	13
gears	21
SE collector	101
residual structure	0

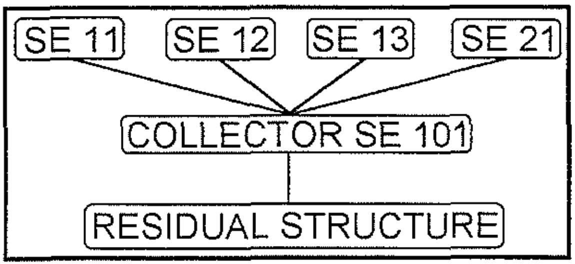

Fig. 11 - Super Element Tree Sketch

The presence of the SE collector the inclusion of nodes that represent the interfaces between each component that are not significant for the analysis (otherwise obligatory included into the residual structure) and let free the residual structure to include only reference and control points such as bearing reference points, gear contact points, restraint points, load points, etc..
In this way the data recovery of only a very simplified residual structure (composed by a very low number of grid points - no more than 200) has to be performed to check the overall effects of new loads, new components, slightly different interfaces, etc.

Restraints and Loads

The FEM model has been restrained at the connection of the support struts to the helicopter main structure that has been considered rigid.
The collector gear has been torsionally restrained at the spline (interface with the sun gear) to simulate torque introduction into the mast.
Loads have been applied on the upper flange (from ring gear) and on the pitch line of mating gear teeth of the pinion (II/III stage reduction).
The load distribution on the pins and bolts of the upper flange due to ring gear torque has been calculated using a dedicated local Finite Element Analysis.

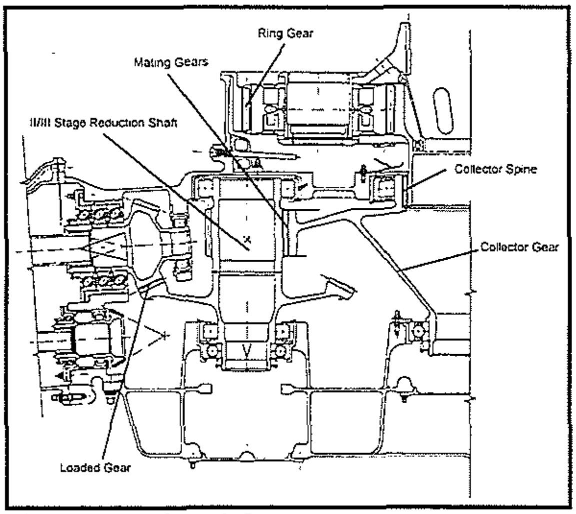

Fig. 12 - Shaft and Gear Arrangement

Finite Element Model Quality Assurance

The EH101 transmission main case model had been quality assured using a set of standard check (geometric, mathematical, effectivity assurance, etc.) to reach a higher confidence level of the final results and achive a good compliance between reality and mathematical representation.
Only the qualified model has been released for analysis and every modification introduced into the model means a new activity of quality assurance.

Cross over checks have been performed using available test results to assure conformity between mathematical simulation and test.

Mathematical Solution

An IBM Risc 6000, 58H model, with 256 Mbytes of RAM and 6 GBytes of free disk space has been used to solve the finite element problem and evaluate displacement and stress results.

A time varying (according to the data recovery and output request) between 8 and 12 hours (elapsed time), about 10000 second of CPU and 4.5 GBytes of disk space have been necessary for the solution using NASTRAN version 69.

Post Processing

Due to the complexity of the finite element model and the very high number of grids and elements, the post processing of result data had been a very delicate task.

The FEM partition chosen for the mesh generation has been used to limit the post processing activity to only one sub-part at a time.

Furthermore, as a super-element analysis technique has been used, a separate data recovery for each superelement has been performed limiting in this way the mass of data to be generated and checked for each step solution.

The post processing of NASTRAN output has been very limited due to the use of 3D elements and geometry very close to reality. For this reason the use of graphic interfaces (deformed shape animation, contour plot of stresses, arrow plot of restraint forces, etc.) has been shown to be particularly useful to understand and check the results better.

The results obtained and verified are the following:

1. General displacement behaviour of the transmission.
2. General stress state of the transmission.
3. Displacement of shafts in the bearing zones due to the deformation interactions of all cases.
4. Displacement of shafts in the gear contact zone due to interactive case and shaft deformations.
5. Stress in the shafts induced by case deformations.

Conclusion

The activity has demonstrated the contemporary feasibility of the following tasks:

1. CATIA 3D modeling of a very complex transmission case without any geometric simplification using solid and surface modelers.
2. Transfer of a highly detailed model to I-DEAS pre-processor without any loss of accuracy.
3. Generation of a 3D finite element model fully representative of the complex geometry.
4. Integration into the same finite element model and analysis of almost the whole main transmission (main case, lower and upper case, main helicopter structure connection struts, II/III reduction stage shafts and gears, collector gear and bearings).
5. Finite element solution using NASTRAN super-element technique of the fully integrated model with limited hardware and elapsed time resources.
6. Post processing and result interpretation of the displacement and stress behaviour of the almost complete main transmission taking into account interaction effects.

Further Developments

This activity involved the applicability of CAD and FEA methodologies to actual casting cases geometrically and structurally very complex, taking into account several interactive effects.

Next step will be the application of these approaches to preliminary and detailed component design.

The methodologies used during this activity have shown the following advantages from this point of view:

1. Feasibility of an iterative approach to finite element modeling and analysis till the best obtainable structural and/or functional solution has been reached.
2. Possibility to substitute and re-design only one element of the transmission (for

example one shaft or one case) without the necessity of re-performing all the calculation by following these steps:

a) Re-design of the new component using 3 dimensional CAD technique.
b) Generation of the Finite Element Model of the new component.
c) Analysis of the new component alone using iterative CAD-FEA approach till a structural and functional good compromise has been reached.
d) Substitution of the old component with the new one in the integrated analysis.
e) Finite element solution of the new component using a stiffness matrix technique to reduce complexity, cost and time (stiffens matrix technique means the substitution of the elements not modify with the only stiffness matrices reduced at the interface point) and to take account of the interaction effect.
f) Finite element solution of the unchanged components with the same stiffness matrix technique to take account of the effects of changing one component on all the others.

3. Availability of finite element analysis results in the early steps of design for a rapid and cost-effective evaluation of modifications.

4. Possibility of verifying the mutual effects of several different component solutions on all the other components under analysis.

5. Better confidence of reaching final design goals honouring time and effectiveness requests.

At the moment the most critical points of the activity are the following:

1. A hard and time consuming activity of CATIA 3D modeling if the geometry of the component to be designed and/or analysed is very complex.
2. An improvable integration between CAD and FEA more effective software actually limits and slows the double sense connection between design and analysis tasks and makes the necessary translation of geometry more difficult than expected.
3. The complexity of modeling and analysis and the necessity to include several components obliged the use of a high class hardware.
4. The very remarkable mass of output data makes indispensable a very accurate post processing to avoid misinterpretation or loss of result informations.
5. The several interaction effects and the complex structural behaviour makes it necessary to have cross-checking with reliable test data.