

APPLICATION OF FINITE ELEMENTS

METHOD TO THE SIZING OF MECHANICAL PARTS

BY

P. SARLIN

SOCIETE NATIONALE INDUSTRIELLE AEROSPATIALE
HELICOPTER DIVISION
MARIGNANE, FRANCE

TENTH EUROPEAN ROTORCRAFT FORUM
AUGUST 28 – 31, 1984 – THE HAGUE, THE NETHERLANDS

APPLICATION OF FINITE ELEMENTS METHOD TO THE SIZING OF MECHANICAL PARTS

by

P. SARLIN

AEROSPATIALE, MARIGNANE - 13725 MARIGNANE Cedex

ABSTRACT

The theoretical bases of the finite elements method are now well established and many large and general programmes are available on the market.

Many drawbacks remain however and this method is not yet quite successful.

Finite elements method implementation i.e. :

Preparation of models

Optimization of calculations with increasingly powerful computers

Interpretation and utilization of results are the main drawbacks.

This method originally developed by design engineers was adopted by mathematicians who laid the theoretical foundations ; it is now time to put it into practical form to have it used by engineers.

The finite elements method may be put to many uses and should become truly operational in the Design Office.

Our convictions and explanations as to how we introduced this method in our Design Office are exposed in the conference.

We present in particular :

- The mesh generator we have developed.
- The liaisons between finite elements and Computer Assisted Design (CAD).

As far as aeronautics are concerned, the finite elements method was first applied to structures (fuselage).

Advanced calculations and a fatigue analysis are most often required on the mechanical components (transmissions, rotor) of a helicopter ; we have devoted many efforts over the last few years to the application of the finite elements method to transmission assemblies.

The various types of calculations used in this field are rapidly presented :

- Stamped shell type structures
- Axisymmetrical structures, structures with cyclic symmetry
- Pure tridimensional structures.

For maximum efficiency, we believe that the finite elements method must be applied as soon as possible in the design stage.

Concrete examples of this application are presented :

- AS 366 G Tail Rotor Hub
- AS 365 Main Rotor Mast
- AS 332 Spheriflex Hub.

It is demonstrated that design, testing and manufacturing costs can be reduced in the example quoted above.

1- INTRODUCTION

The calculation of structures in the various engineering fields is currently posing problems that are becoming increasingly numerous, varied and even more and more complicated because with the development and improvement of the calculation systems, the requirements in the search of a more accurate estimate, a finer interpretation and a more extensive resolution of the problems, also increase, with a view to obtaining a better optimization and an increased safety.

It can even be stated that there are many subjects that no one even dared to approach, in the strong and a priori belief that they would remain unanswered.

In the field of helicopters, we would mention, incidentally the calculations of structures made of composite materials, stress analysis in mechanical parts of complex shape to facilitate fatigue calculations.

The Finite Elements Method (FEM) has progressively become a standard tool in the calculation of structures.

The theoretical bases have long been established, and the arrival of powerful computers on the market has allowed developing many general purpose programs.

It is now an acknowledged fact that this method constitutes the first link that has been developed in the complex chain of computer aided engineering.

The FEM, however, remains a complicated and costly tool. This is why the implementation of such a tool in a truly operational manner, has now become a top priority.

We firmly believe that this task is the responsibility of the manufacturers. Our beliefs regarding the industrial organization of the FEM will be developed in the present paper.

We shall endeavour to illustrate our account by presenting a few practical applications.

2 – SCOPE OF FINITE ELEMENTS TODAY

The answers expected from FEM are varied, however, they can be gathered together in 2 families.

1. Calculations for facilitating the final substantiation

They are used at the final stage of the project, when dimension values have already been chosen, and the results obtained are utilized as an a posteriori verification.

The FEM is used in this manner from the outset, and the limits of such an utilization have been explained.

The results are obtained when it is often too late to modify the dimension values. In certain cases, some test results are available and attempts are then made to reproduce these results by calculation. This is sometimes difficult since the conditions of a test cannot always be simulated on a model.

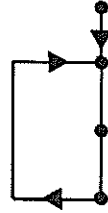
This use of FEM is somewhat static and progressively we tried to meet the second family of requirements.

2. Calculations for aiding the design

These calculations represent a much more dynamic utilization of the FEM and are certainly paving the way of the future for this method. Here the calculations are used for :

- Classifying various technological solutions
- Illustrate the effects of such or such design parameter.

Here, the FEM constitutes a link in the design chain that can be represented as follows :



- Project (pre-dimensioning)
- FE calculation
- Analysis of results, discussion
- Modifications, effects on drawing.

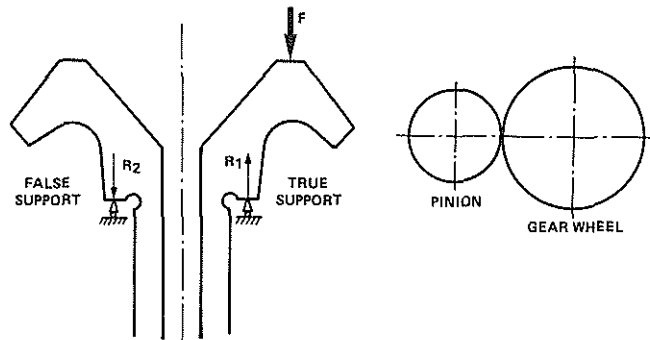
This scheme, applicable to a project or a part of a project, may require the execution of a great number of iterations within a very short time.

To meet these requirements, a new type of organization which will be described in the following part, must be adopted.

For this type of calculations, 2 qualities are essentially required.

a. Confidence in the results

It should be pointed out, here, that confidence does not necessarily mean that very accurate results in absolute value are sought. A simple example showing the impossibility of obtaining a great accuracy is given below. This example reflects the simulation of the rest of a pinion on a support bearing.



Let us examine the problem of the F axial load from the adjoining wheel. The structure is treated as an axi-symmetrical case, with non axi-symmetrical loads (expanded as Fourier series).

At $\theta = 0$, under the F load, we obtain support reactions $R1 > 0$ that are physically correct.

At $\theta = \Pi$, we also obtain R2 support reactions that have no physical significance due to the backlash between the bearing and the pinion.

In consequence, when we refer to the confidence in the results, we do not necessarily mean accuracy. We want to have a tool that gives us, with certainty, the possibility of identifying the critical points to allow separate classification of different solutions.

b. A very short response time

The response time of FEM must be the same time taken for self-interrogation by the designer. If not, the designer will be tempted to use conventional and

therefore more approximate calculations. To obtain a satisfactory response time, a new type of organization, specific of industrial field, must be adopted for FEM. This type of organization is detailed in the following part.

3 – ORGANIZATION OF FINITE ELEMENTS IN THE INDUSTRIAL FIELDS

Only the point of view of the industrial user who wishes to use the FEM as a design aiding tool, is presented here.

The main ingredient in the organization is, of course, the program itself.

A good quality program must be available. This requirement is the easiest to be met, as a large number of good programs are now available on the market.

These programs have been developed either in major universities :

- University of California (SAP program)
- Massachusetts Institute of Technology (ADINA program)
- Brown University (MARC program)
- University of Stuttgart (ASKA program)

or in major Research Centers :

NASA (NASTRAN program)

The technical level reached by these important programs is such that it is useless (because too expensive) for a manufacturer to undertake the drawing up of his own program.

One of these programs must be selected and adhered to, because as it will be seen later in this paper, the organization efficiency depends on a good knowledge of the internal functioning of the program.

Moreover, an attractive program handled and developed by a dynamic team, must be selected whenever possible.

For our part, we have been using for 6 years the SAMCEF program, developed at the University of Liege (Belgium) from 1970 and being continuously expanded ever since.

This program covers all the branches of the Structure Mechanics : Static, free and forced dynamic analysis in the linear and non-linear elastic field.

A contractual maintenance system was jointly developed with the authors that includes :

- Technical assistance service
- Regular maintenance of the program by creating new versions at a yearly rate.

As regards the organization we are convinced of the following. Whatever the calculation program used and whatever its efficiency, its industrial interest will remain almost inexistent if it is not backed by a series of satellite programs ensuring its communication with the users.

This essential requirement is the most difficult to meet, since universal communication programs are not available on the market.

This can be explained by the fact that communication with the calculation program is closely dependent on the specific requirements of each industrial field.

It is, therefore, the responsibility of each group of users to create the communication tools around his program.

We have been using our best endeavour to this end for several years, and this has resulted in an organization as presented in the form of a diagram in Figure 1.

Among all the communication satellites, the Mesh Generator is the most important, since the creation of the mesh remains the most delicate phase of a calculation by FEM.

The quality of the results depends on the mesh.

The model representation technique is not a branch that is currently being taught and, therefore, remains an «art» based on the experience.

3.1 – The G.R.A.T.I.S. mesh generator

(Rapid and Automatic Generator by Iso-parametric Technique). This generator has been entirely developed at Mari-gnane by S.J.M. Achard

3.1.1 – Principle of operation

The nodes of the final mesh are created by interpolation on the nodes of an initial mesh.

The initial mesh is to be prepared by the user. This initial mesh is very coarse, it is made up of Super-Nodes and Super Elements in number that is strictly necessary for description of the geometry.

The fineness of the mesh is then controlled by the user by means of several parameters.

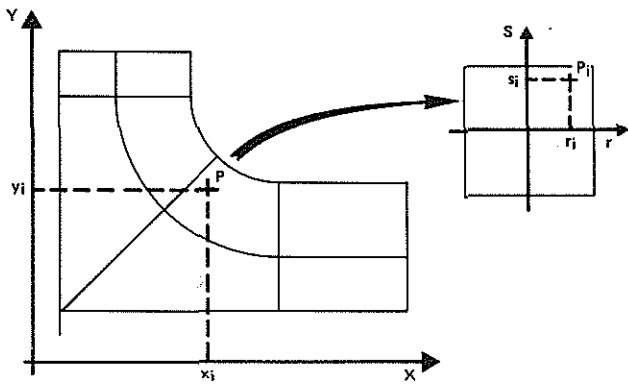
Example of use for thin and thick structures are given in Figures 2, 3, 4 and 5.

For each Super-Element, the technique consists in the use of a geometrical transformation from the actual cartesian space (x, y) into a parent space (r, s) where the local coordinates are represented by $-1 \leq r \leq 1$ and $-1 \leq s \leq 1$.

Thus the cartesian coordinates of a point P (x, y) are obtained as an interpolation on the cartesian coordinates of the Super-Element nodes (xi, yi).

$$\begin{Bmatrix} x \\ y \end{Bmatrix} = \{N\} \begin{Bmatrix} x_i \\ y_i \end{Bmatrix}$$

The form functions (N) of this interpolation are expressed in the parent space (r, s).



The interpolation functions (N) are the same as those used for interpolation of displacements field (Iso-Parametric elements).

The interpolation functions can be achieved for any number of variables, this is why the capacities of the GRATIS mesh generator could be extended to :

- 1-dimension structures (trusses and beams)
- 2-dimension structures in plane and space (plates and shells)
- 3-dimension structures.

3.1.2 – Performance of the mesh generator

1. Rapidity of execution

The programming adopted in GRATIS makes a wide use of interactive graphic techniques.

The mesh is created by the user in a conversational mode with the computer. For this, he has a large number of function keys arranged on a keyboard menu.

The possibilities of instantaneous display allow developing the mesh within a very short time.

Some applications of the GRATIS system are presented in Figures 6, 7, 8 and 9.

2. Multiple choice

Before accepting a mesh, the user can test a large variety of meshes without difficulty.

In particular the assumedly critical areas can be refined by means of the display systems (zoom, change in projection, ...)

In addition to the geometry, the GRATIS system allows creating :

- Boundary conditions
- Loads (distributed or concentrated)
- Material properties.

3.2 – Link with C.A.D.

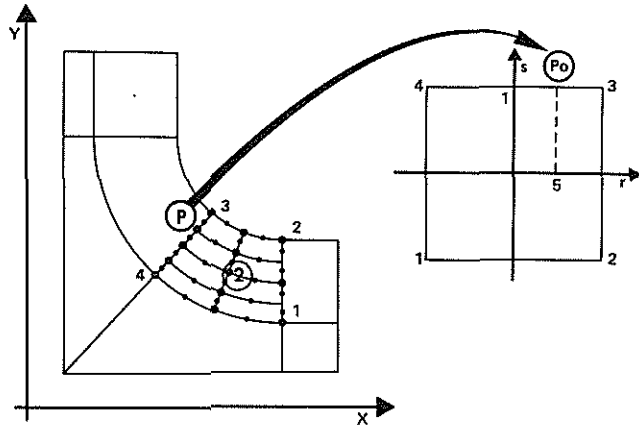
For several years, the Engineering Department has been using C.A.D. systems such as CADAM (IBM) and SYSTRID.

The drawings obtained from these programs include all the necessary data for constituting the Super-Mesh for GRATIS (singular contour points, positions of circle centers, curve intersections, ...).

This is why it is the interest of FEM to take advantage of this data to create the Super-Mesh.

This is possible, with a computer organization represented in Figure 10.

In this case, all manual actions for the preparation of the Super-Mesh are practically suppressed.



Creation of a mesh for Super-Element number 2.

Assuming the following mesh is to be achieved :

- 4 elements on the 1-2 boundary
- 2 elements on the 2-3 boundary

This comes to creating 37 nodes and 8 elements.

Each node can be obtained by using the interpolation described above. Example : coordinate of node P

$$\begin{Bmatrix} x_p \\ y_p \end{Bmatrix} = \{ N_{p0} \} \begin{Bmatrix} x_i \\ y_i \end{Bmatrix}$$

with N_{p0} : interpolation functions at point P_0 where $r = .5$ and $s = 1$.

All the nodes of this Super-Element can thus be automatically created.

The same procedure is used for all the Super-Elements.

The mesh continuity is automatically ensured across the boundaries between Super-Elements.

The double nodes created at boundaries are eliminated.

When the Super-Mesh is built, the user supervises the mesh construction by specifying the type of desired mesh on the Super-Mesh boundaries.

4 – PRACTICAL APPLICATIONS

4.1 – Tail rotor hub – AS 365 N1

This is an example of close relation between design and calculations.

The hub is a stamped shell made of AU4G1, intended for joining the drive shaft of the 11 blades of the tail rotor.

The solution initially retained, resulted in fracture near the lower edge of the hole in the external web.

1. First stage

Determining the origin of the failure.

The calculation for this initial solution, revealed a static stress concentration in the failure area.

A separate analysis of the loads applied, allowed determining that high centrifugal loads generated by the 80 g bearing were involved.

2. Second stage

A series of modifications were then proposed with the designers and immediately calculated.

The final solution retained allowed attenuating the stress concentration and increasing the service life of this part. (see Figures 11 and 12).

The choice of this modification could be determined in 4 days.

During this period, 10 modification solutions have been tried and calculated.

The creation of the first mesh took 4 hours.

In this application, the following results were obtained with FEM :

- Confirming the critical points
- Allowing the drawing of a «dead sure» solution
- Avoiding the execution of many tests.

4.2 – Stresses modulation over 1 revolution for the AS 365 Main Rotor Mast

In this application, the FEM assisted in determining the mast service life.

The purpose of this calculation is to determine the variations of the stresses generated in the mast during one rotor revolution, for 1R flap loading.

1. First stage

Determining the loads exerted by the Starflex hub on the rotor mast plate.

For this, we developed a simplified mesh of «hub-mast» assembly. A half model was sufficient due to the loading symmetry. (see Figure 13).

The Starflex hub material was represented by an equivalent homogeneous material. Its mechanical properties have been adjusted in accordance with the results of rotor arms bending tests, since in this case a correct representation of the assembly rigidity is sufficient (this is not a stress analysis).

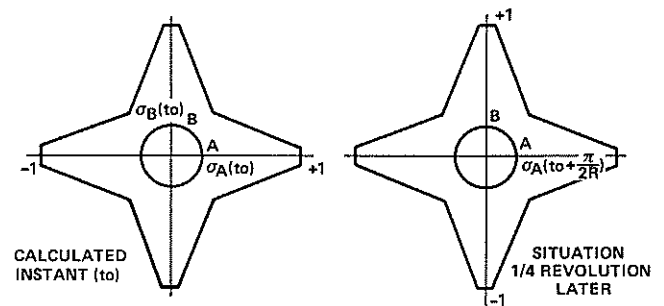
A load distribution representative of the actions of the hub on the mast was thus obtained.

2. Second stage

The rotor mast could be isolated and a finer calculation made on an axi-symmetrical model subjected to a non axi-symmetrical loading.

The expansion of the loads in the Fourier series form, has allowed obtaining the stresses exerted in the mast at a given instant of the rotor revolution.

The nature of the 1R flap loading, leads to the following remark.



The distribution of stresses at an instant located 1/4 revolution later is contained in the following calculation :

$$\sigma A (t^0 + \Pi / 2R) = \sigma B (t^0)$$

The answer to point A over 1 revolution is in the following form :

$$\sigma A = \sigma^0 \text{COS} (Rt + \emptyset)$$

We know 2 points $\sigma A (t^0)$ and $\sigma A (t^0 + \Pi / 2R)$

Thus, σ^0 and \emptyset can be calculated for all the rotor points.

In this way, we obtained the dynamic stresses generated by this loading.

4.3 — Calculation of stresses generated in the Spheriflex Rotor Hub

This purely tri-dimensional calculation was made for the following loadings :

- **Static loads**

- Centrifugal
- Lift
- Drag
- Adapter load

- **Dynamic 1R and 2R loads**

- Centrifugal
- Flap
- Drag
- Adapter load

The static and dynamic 2R loads have been calculated on a 1/4 model of the hub.

The dynamic 1R loads have been calculated on a 1/2 model (see Figure 14).

The main useful purpose of these calculations was the possibility of verifying that no critical area existed in the hub, and clearance for the first flights could be obtained without fatigue test.

At this stage, the calculations confirmed the critical areas detected during the static photo-elastic test of the plate, and did not detect any other areas.

The static substantiation was obtained by means of this calculation.

Another application of this calculation was the assistance offered in positioning the gauges for the calibration and fatigue tests.

- **Calibration test**

FEM has allowed pinpointing the areas where the uncoupling of inter-forces is the best.

The gauges have been located in these areas.

- **Fatigue test**

Stress modulations could be determined by this calculation (refer to the preceding application).

In consequence, gauges for monitoring these modulations have been placed on the hub.

A comparison between tests and calculation is awaited.

5 — CONCLUSIONS

The opinion that one may have on the FEM is closely related to the industrial operating conditions of this tool.

For too long, the disadvantages caused by the bulk of data required for implementation of the method, have been used for argument purpose.

Today, the main action to be conducted is the operational organization of the tool within its industrial environment.

Once this task has been achieved, a list of the disadvantages and advantages of the method can be drawn up.

- **Disadvantages**

- New problems associated with the «model representation».

The FEM is generally thought nowadays, but this is not the case for model representation techniques. So, building the model constitutes the main source of errors, sometimes serious and difficult to detect.

- Illusion in thinking that anything can be calculated by FEM.

The tool is very powerful, it is capable of offering a «calculated solution» to a large number of situations.

The temptation of calculating «anything» at the cost of unrealistic assumptions may be strong, although «certain» results are sometimes obtained in this way.

- **Advantages**

Powerful, general and reliable programs are now available. The FEM has allowed suppressing the usual limitations imposed on the «structures» calculations :

Limitations in the geometry, loading and boundary conditions.

It is now a proven fact that the FEM is capable of having an impact on the design.

The main advantage, however, lies in a reduction of the costs :

- Regarding the design : by the possibility of quickly comparing several solutions at the project stage.
- Regarding the tests : by assisting in an optimum location of the measurement points, and thus, a reduction in the number of tests required.

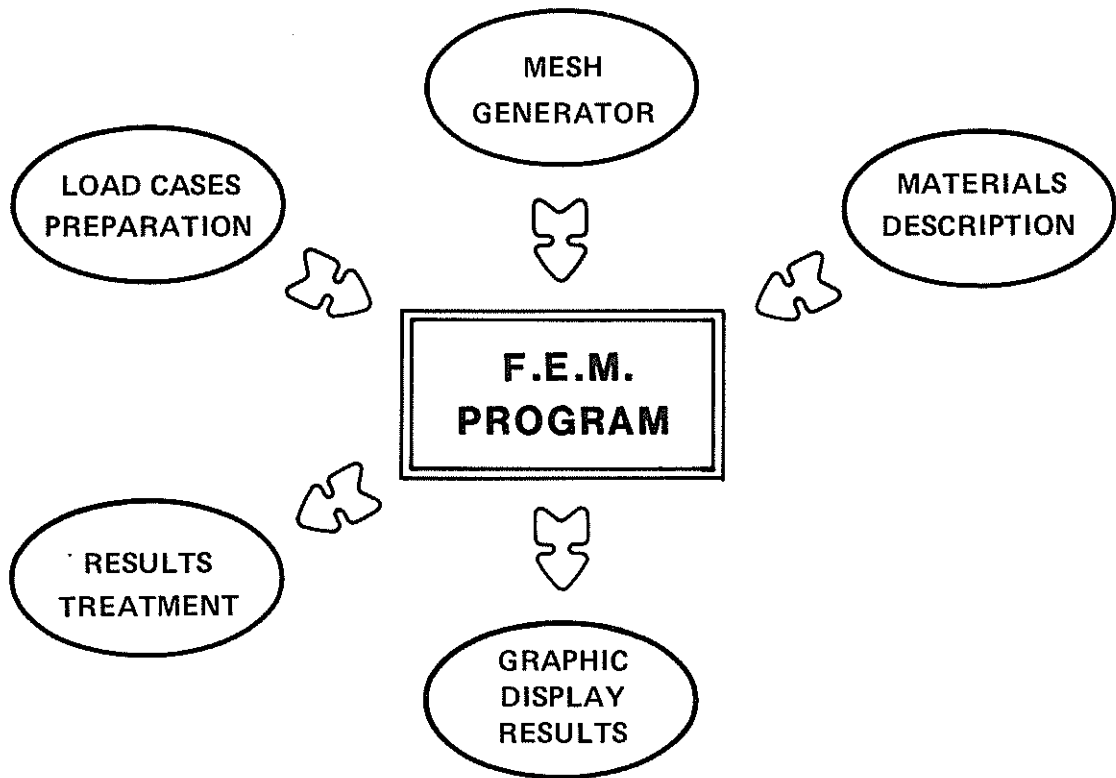

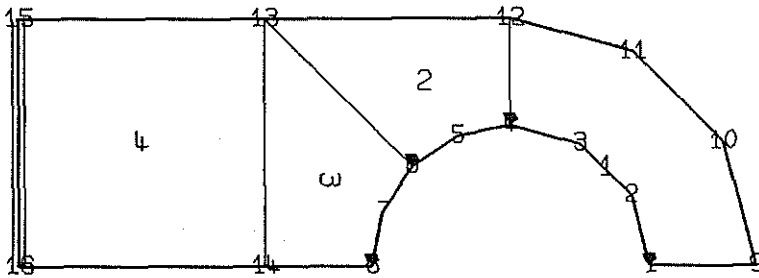


FIGURE 1 SATELLITES OF COMMUNICATION

GRATIS

16 NODES 4 ELEMENTS
6 FIXATIONS 1 PRESSURES

0 0.52860E-05 0
90.000 30.000 0
 $\theta = +90$
 $\beta = +90$
 $\gamma = 0$


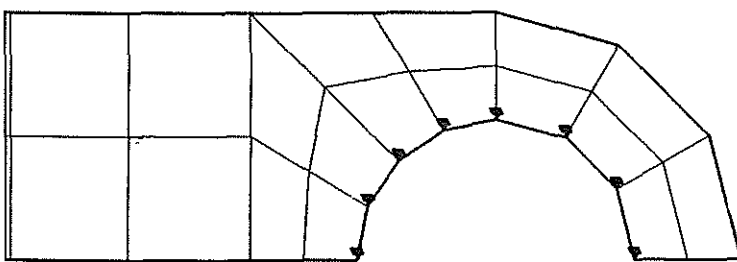
SUPER-MESH

- 1 DATA FILE READING
- 2 DATA FILE STORAGE
- 3 DATA FILE EDITING
- 4 FEM FILE STORAGE
- 5 COMPACTED DATA FILE READING
- 6 COMPACTED DATA FILE STORAGE
- 7 MESH GENERATION
- 8 VOLUME GENERATION
- 9 GEOMETRIC MODIFICATION
- 10 MERGING WITH A COMPACTED DATA FILE
- 11 DATA FILE DISPLAY
- 12 GRAPHIC DISPLAY SELECTION
- 13 CHARACTERISTIC INPUT
- 14 CHANGE IN PROJECTION
- 15 ZOOMING
- 16 PAPER PRINT
- 17 THE END

GRATIS

30 NODES 18 ELEMENTS
10 FIXATIONS 2 PRESSURES

0 0.52860E-05 0
90.000 30.000 0
 $\theta = +90$
 $\beta = +90$
 $\gamma = 0$

MESH FINENESS 1

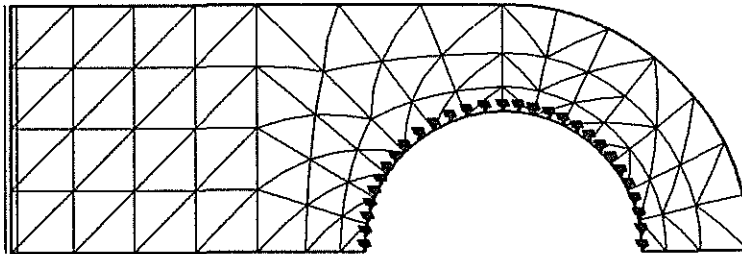
- 1 DATA FILE READING
- 2 DATA FILE STORAGE
- 3 DATA FILE EDITING
- 4 FEM FILE STORAGE
- 5 COMPACTED DATA FILE READING
- 6 COMPACTED DATA FILE STORAGE
- 7 MESH GENERATION
- 8 VOLUME GENERATION
- 9 GEOMETRIC MODIFICATION
- 10 MERGING WITH A COMPACTED DATA FILE
- 11 DATA FILE DISPLAY
- 12 GRAPHIC DISPLAY SELECTION
- 13 CHARACTERISTIC INPUT
- 14 CHANGE IN PROJECTION
- 15 ZOOMING
- 16 PAPER PRINT
- 17 THE END

FIGURE 2

GRATIS

275 NODES 116 ELEMENTS
31 FIXATIONS 4 PRESSURES

0 0.52860E-05 0
90.000 30.000 0
 $\theta = +90$ Y
 $\beta = +90$ |
 $\gamma = 0$ 0 X



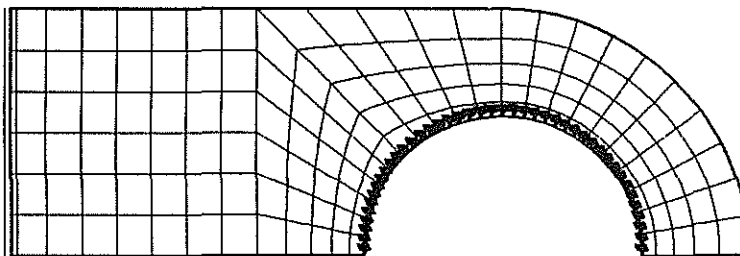
MESH FINENESS 2

- 1 DATA FILE READING
- 2 DATA FILE STORAGE
- 3 DATA FILE EDITING
- 4 FEM FILE STORAGE
- 5 COMPACTED DATA FILE READING
- 6 COMPACTED DATA FILE STORAGE
- 7 MESH GENERATION
- 8 VOLUME GENERATION
- 9 GEOMETRIC MODIFICATION
- 10 MERGING WITH A COMPACTED DATA FILE
- 11 DATA FILE DISPLAY
- 12 GRAPHIC DISPLAY SELECTION
- 13 CHARACTERISTIC INPUT
- 14 CHANGE IN PROJECTION
- 15 ZOOMING
- 16 PAPER PRINT
- 17 THE END

GRATIS

508 NODES 147 ELEMENTS
45 FIXATIONS 8 PRESSURES

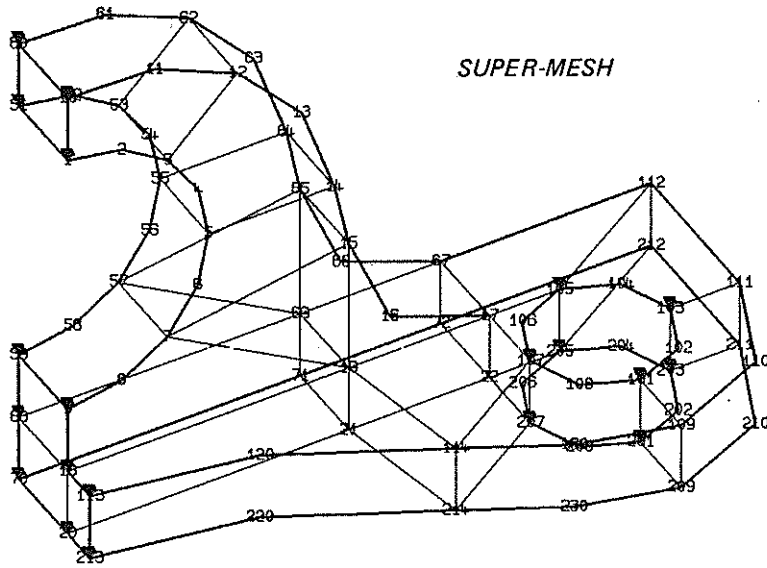
0 0.52860E-05 0
90.000 30.000 0
 $\theta = +90$ Y
 $\beta = +90$ |
 $\gamma = 0$ 0 X



MESH FINENESS 3

- 1 DATA FILE READING
- 2 DATA FILE STORAGE
- 3 DATA FILE EDITING
- 4 FEM FILE STORAGE
- 5 COMPACTED DATA FILE READING
- 6 COMPACTED DATA FILE STORAGE
- 7 MESH GENERATION
- 8 VOLUME GENERATION
- 9 GEOMETRIC MODIFICATION
- 10 MERGING WITH A COMPACTED DATA FILE
- 11 DATA FILE DISPLAY
- 12 GRAPHIC DISPLAY SELECTION
- 13 CHARACTERISTIC INPUT
- 14 CHANGE IN PROJECTION
- 15 ZOOMING
- 16 PAPER PRINT
- 17 THE END

FIGURE 3



SUPER-MESH

GRATIS

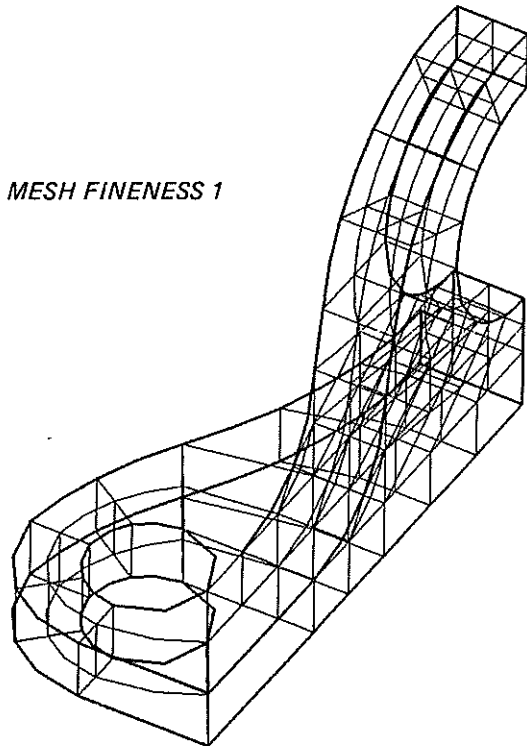
76 NODES 15 ELEMENTS
32 FIXATIONS 2 PRESSURES

4.0000 0.23321E-05 5.0000
25.000 45.000 40.000

$\theta = +30$
 $\beta = +40$
 $\gamma = 0$



- 1 DATA FILE READING
- 2 DATA FILE STORAGE
- 3 DATA FILE EDITING
- 4 FEM FILE STORAGE
- 5 COMPACTED DATA FILE READING
- 6 COMPACTED DATA FILE STORAGE
- 7 MESH GENERATION
- 8 VOLUME GENERATION
- 9 GEOMETRIC MODIFICATION
- 10 MERGING WITH A COMPACTED DATA FILE
- 11 DATA FILE DISPLAY
- 12 GRAPHIC DISPLAY SELECTION
- 13 CHARACTERISTIC INPUT
- 14 CHANGE IN PROJECTION
- 15 ZOOMING
- 16 PAPER PRINT
- 17 THE END



MESH FINENESS 1

GRATIS

539 NODES 76 ELEMENTS
105 FIXATIONS 8 PRESSURES

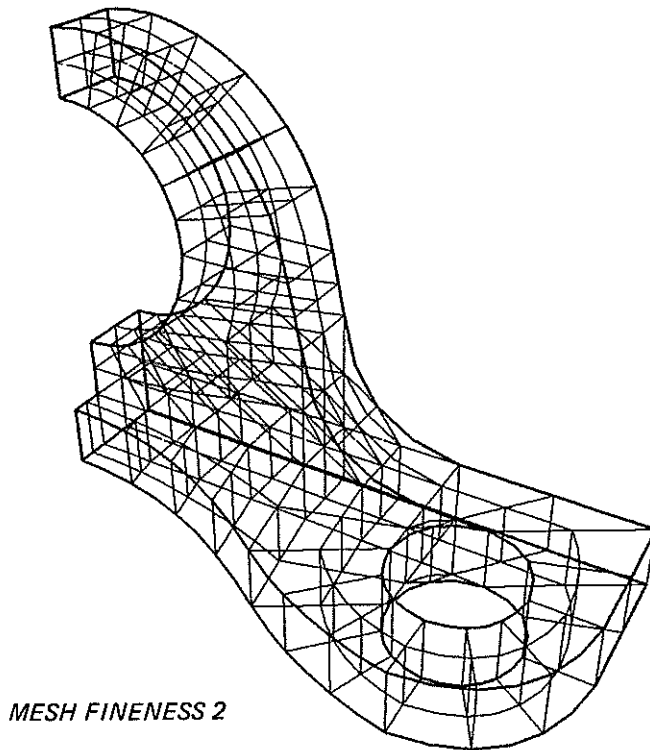
4.0000 0.23321E-05 5.0000
25.000 45.000 40.209

$\theta = -120$
 $\beta = +40$
 $\gamma = 0$



- 1 DATA FILE READING
- 2 DATA FILE STORAGE
- 3 DATA FILE EDITING
- 4 FEM FILE STORAGE
- 5 COMPACTED DATA FILE READING
- 6 COMPACTED DATA FILE STORAGE
- 7 MESH GENERATION
- 8 VOLUME GENERATION
- 9 GEOMETRIC MODIFICATION
- 10 MERGING WITH A COMPACTED DATA FILE
- 11 DATA FILE DISPLAY
- 12 GRAPHIC DISPLAY SELECTION
- 13 CHARACTERISTIC INPUT
- 14 CHANGE IN PROJECTION
- 15 ZOOMING
- 16 PAPER PRINT
- 17 THE END

FIGURE 4

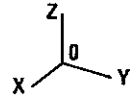


MESH FINENESS 2

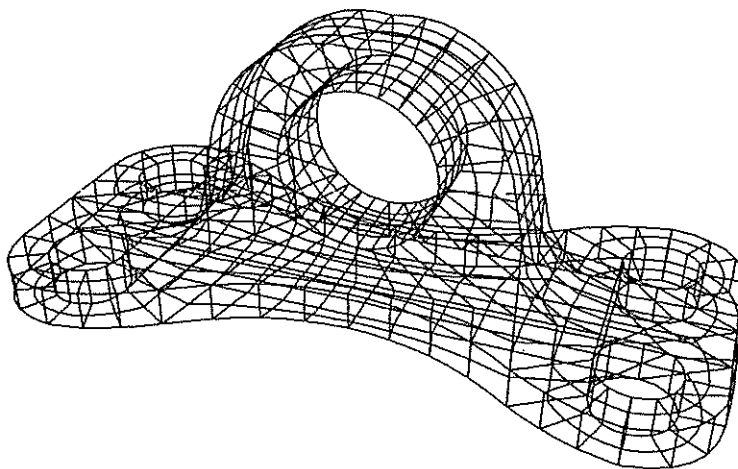
GRATIS

811 NODES 112 ELEMENTS
130 FIXATIONS 12 PRESSURES

4.0000 0.23321E-05 5.0000
25.000 45.000 40.210
 $\theta = -30$
 $\beta = +30$
 $\gamma = 0$
EL = 1.00



- 1 DATA FILE READING
- 2 DATA FILE STORAGE
- 3 DATA FILE EDITING
- 4 FEM FILE STORAGE
- 5 COMPACTED DATA FILE READING
- 6 COMPACTED DATA FILE STORAGE
- 7 MESH GENERATION
- 8 VOLUME GENERATION
- 9 GEOMETRIC MODIFICATION
- 10 MERGING WITH A COMPACTED DATA FILE
- 11 DATA FILE DISPLAY
- 12 GRAPHIC DISPLAY SELECTION
- 13 CHARACTERISTIC INPUT
- 14 CHANGE IN PROJECTION
- 15 ZOOMING
- 16 PAPER PRINT
- 17 THE END

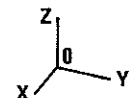


TOTAL STRUCTURE

GRATIS

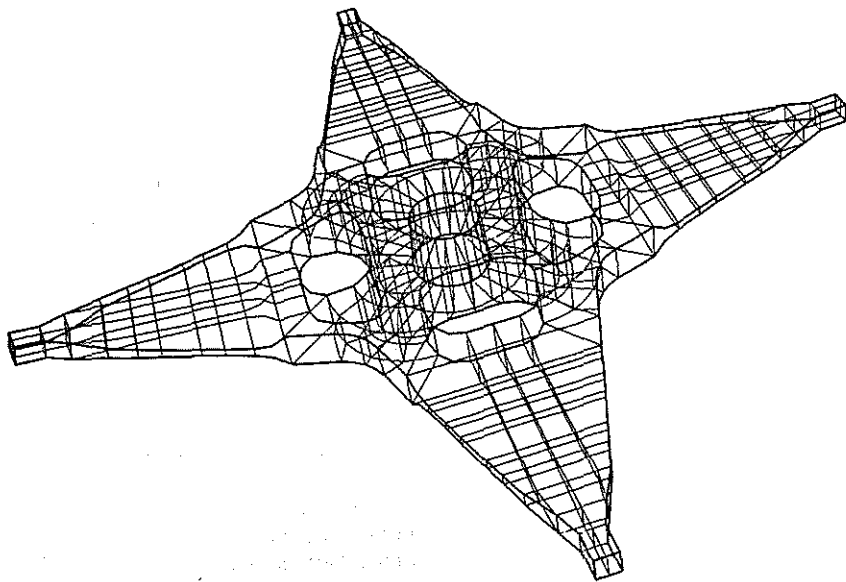
2834 NODES 448 ELEMENTS
520 FIXATIONS 48 PRESSURES

21.000 45.000 5.0000
21.000 45.000 40.210
 $\theta = -20$
 $\beta = +30$
 $\gamma = 0$
EL = 1.00



- 1 DATA FILE READING
- 2 DATA FILE STORAGE
- 3 DATA FILE EDITING
- 4 FEM FILE STORAGE
- 5 COMPACTED DATA FILE READING
- 6 COMPACTED DATA FILE STORAGE
- 7 MESH GENERATION
- 8 VOLUME GENERATION
- 9 GEOMETRIC MODIFICATION
- 10 MERGING WITH A COMPACTED DATA FILE
- 11 DATA FILE DISPLAY
- 12 GRAPHIC DISPLAY SELECTION
- 13 CHARACTERISTIC INPUT
- 14 CHANGE IN PROJECTION
- 15 ZOOMING
- 16 PAPER PRINT
- 17 THE END

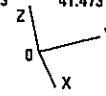
FIGURE 5



GRATIS

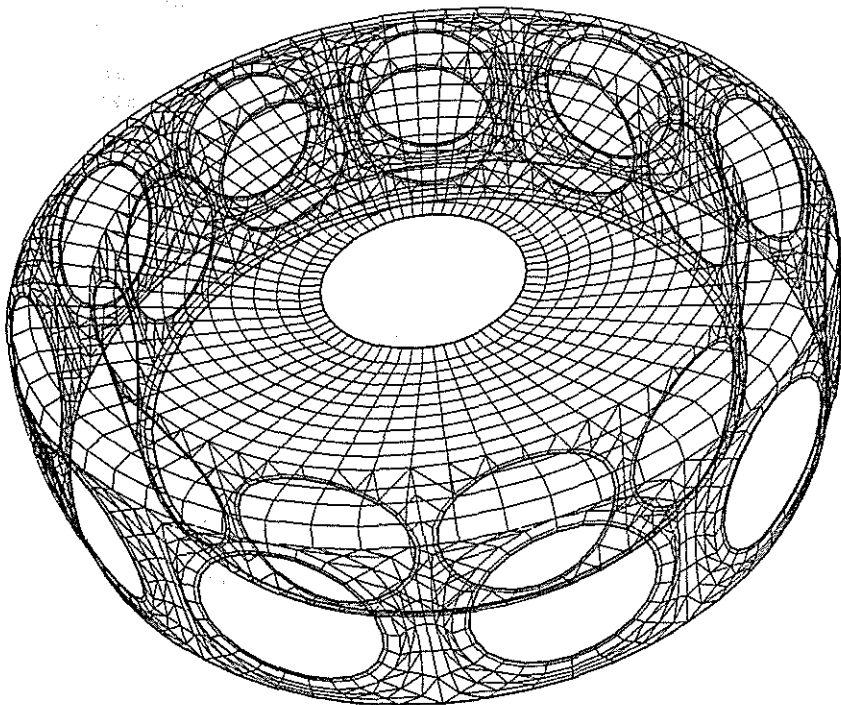
2104 NODES 248 ELEMENTS

601.13 601.13 41.000
 601.13 601.13 41.473
 $\theta = +10$
 $\beta = +40$
 $\gamma = +10$
 EL = 1.00



- 1 DATA FILE READING
- 2 DATA FILE STORAGE
- 3 DATA FILE EDITING
- 4 FEM FILE STORAGE
- 5 COMPACTED DATA FILE READING
- 6 COMPACTED DATA FILE STORAGE
- 7 MESH GENERATION
- 8 VOLUME GENERATION
- 9 GEOMETRIC MODIFICATION
- 10 MERGING WITH A COMPACTED DATA FILE
- 11 DATA FILE DISPLAY
- 12 GRAPHIC DISPLAY SELECTION
- 13 CHARACTERISTIC INPUT
- 14 CHANGE IN PROJECTION
- 15 ZOOMING
- 16 PAPER PRINT
- 17 THE END

FIGURE 6 AS 365 - MAIN ROTOR (STARFLEX)

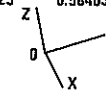


GRATIS

3113 NODES 3410 ELEMENTS

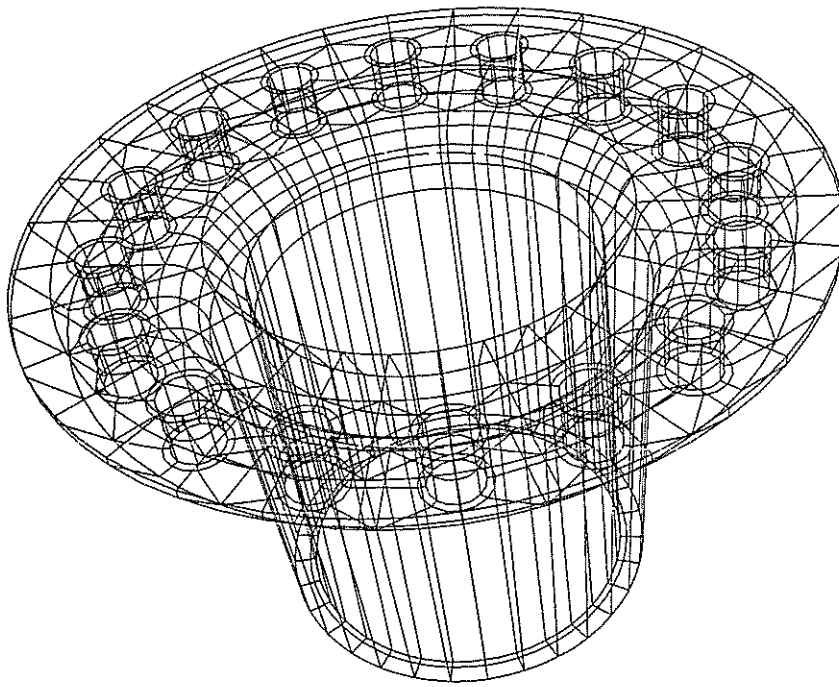
6 CARDS

0.20886 0.20825 0.56370E-01
 0.20340 0.20825 0.58463E-01
 $\theta = +10$
 $\beta = +40$
 $\gamma = +10$
 EL = 1.00



- 1 DATA FILE READING
- 2 DATA FILE STORAGE
- 3 DATA FILE EDITING
- 4 FEM FILE STORAGE
- 5 COMPACTED DATA FILE READING
- 6 COMPACTED DATA FILE STORAGE
- 7 MESH GENERATION
- 8 VOLUME GENERATION
- 9 GEOMETRIC MODIFICATION
- 10 MERGING WITH A COMPACTED DATA FILE
- 11 DATA FILE DISPLAY
- 12 GRAPHIC DISPLAY SELECTION
- 13 CHARACTERISTIC INPUT
- 14 CHANGE IN PROJECTION
- 15 ZOOMING
- 16 PAPER PRINT
- 17 THE END

FIGURE 7 AS 365 N1 - TAIL ROTOR HUB

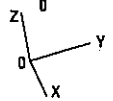


GRATIS

5168 NODES 832 ELEMENTS

3 CARDS

109.00 109.00 130.00
 109.00 109.00 0
 $\theta = +10$
 $\beta = +40$
 $\gamma = +10$
 EL = 1.00



- 1 DATA FILE READING
- 2 DATA FILE STORAGE
- 3 DATA FILE EDITING
- 4 FEM FILE STORAGE
- 5 COMPACTED DATA FILE READING
- 6 COMPACTED DATA FILE STORAGE
- 7 MESH GENERATION
- 8 VOLUME GENERATION
- 9 GEOMETRIC MODIFICATION
- 10 MERGING WITH A COMPACTED DATA FILE
- 11 DATA FILE DISPLAY
- 12 GRAPHIC DISPLAY SELECTION
- 13 CHARACTERISTIC INPUT
- 14 CHANGE IN PROJECTION
- 15 ZOOMING
- 16 PAPER PRINT
- 17 THE END

FIGURE 8 AS 365 - MAIN ROTOR MAST

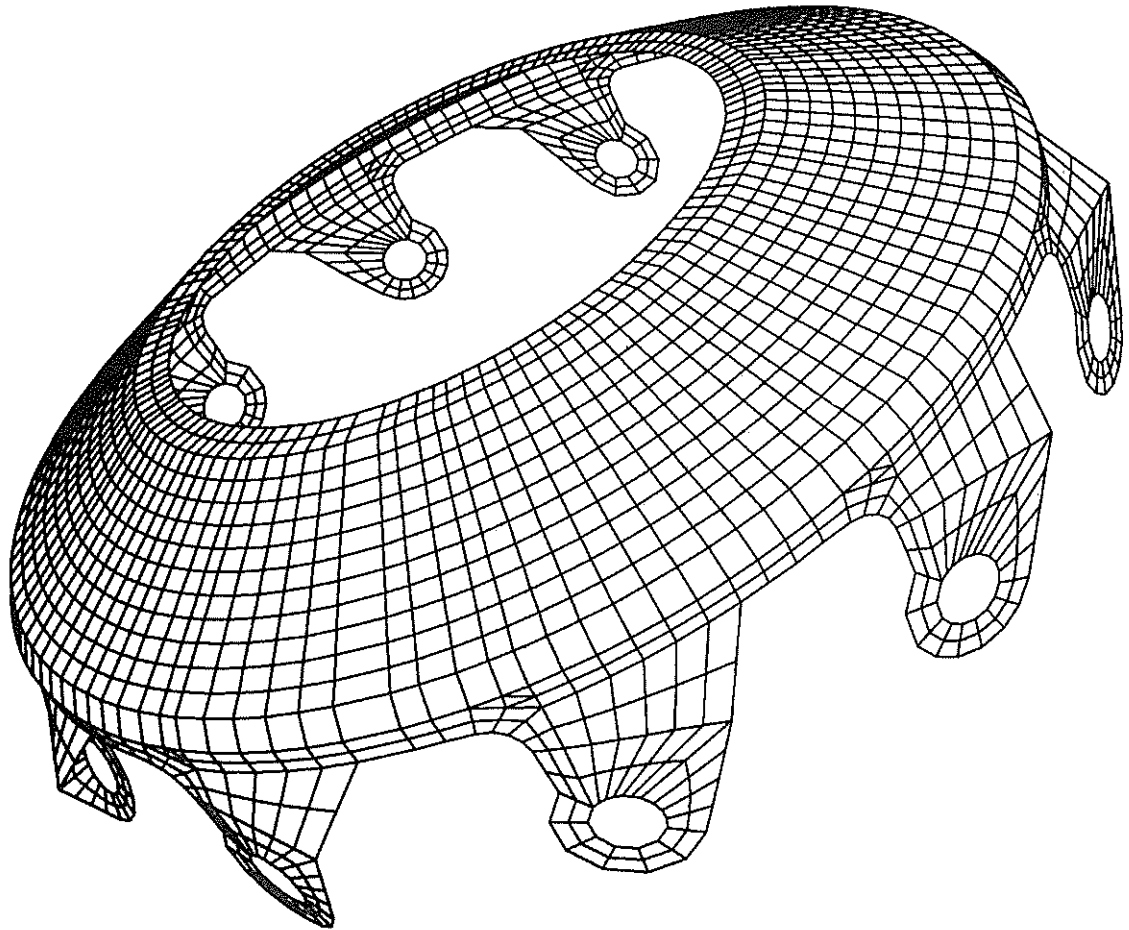


FIGURE 9 AS 365 N1 - PITCH CHANGE SPIDER

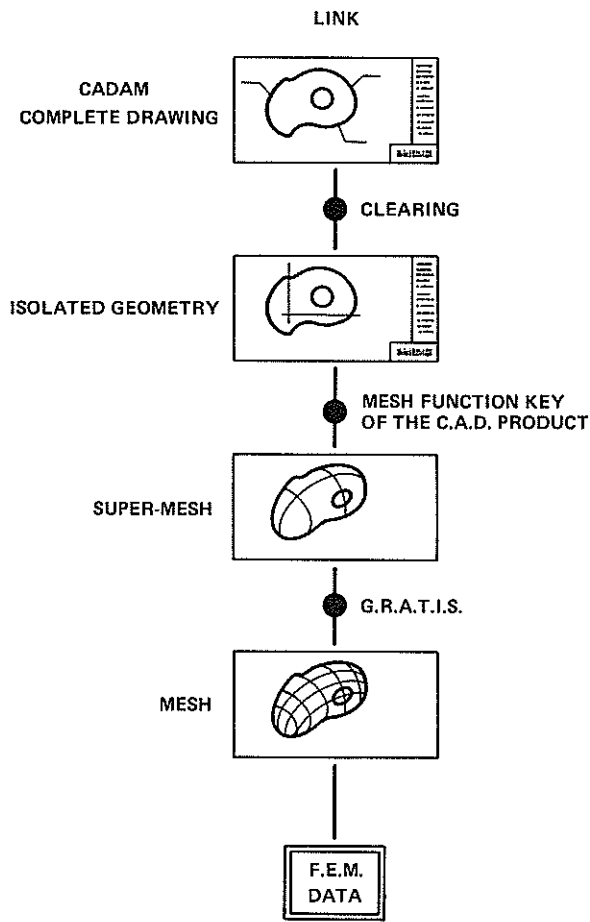


FIGURE 10 C.A.D. → F.E.M.

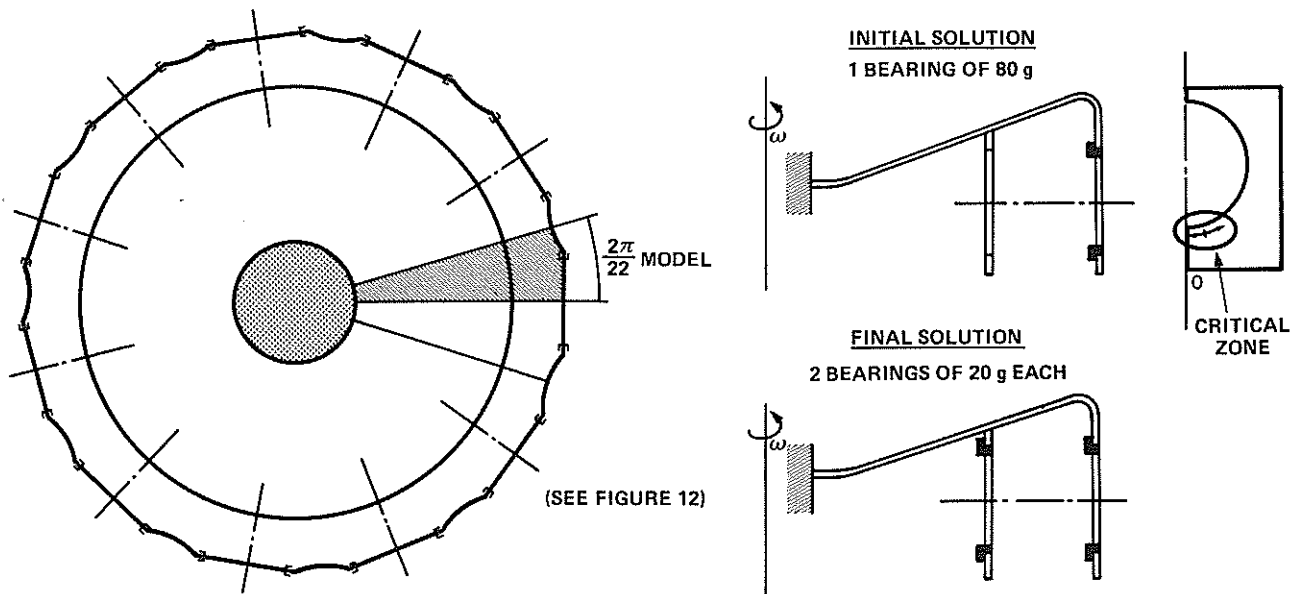


FIGURE 11 AS 365 N1 - TAIL ROTOR HUB

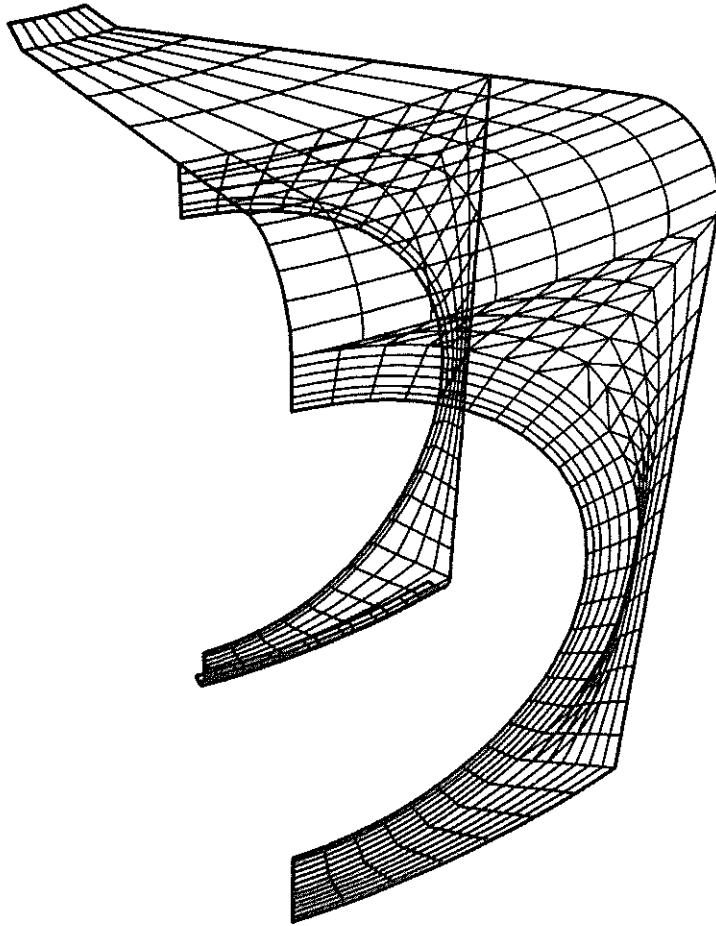
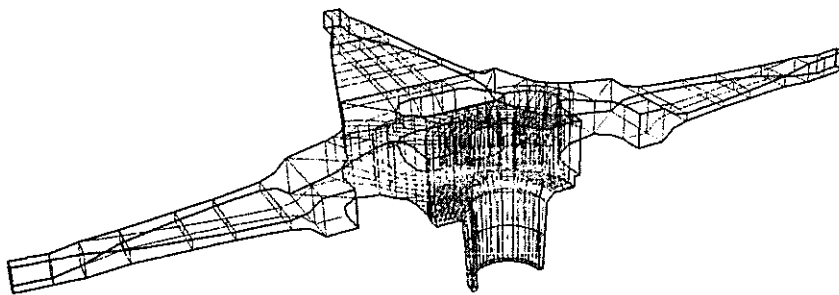


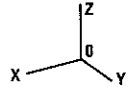
FIGURE 12 AS 365 N1 - TAIL ROTOR HUB - $\frac{27\pi}{22}$ MODEL



GRATIS

3340 NODES 480 ELEMENTS
 474 FIXATIONS
 16 LOADS
 3 CARDS

601.13 601.13 171.00
 601.13 0.66878E-04 41.473
 $\theta = -60$
 $\beta = +25$
 $\gamma = 0$
 EL = 1.00



- 1 DATA FILE READING
- 2 DATA FILE STORAGE
- 3 DATA FILE EDITING
- 4 FEM FILE STORAGE
- 5 COMPACTED DATA FILE READING
- 6 COMPACTED DATA FILE STORAGE
- 7 MESH GENERATION
- 8 VOLUME GENERATION
- 9 GEOMETRIC MODIFICATION
- 10 MERGING WITH A COMPACTED DATA FILE
- 11 DATA FILE DISPLAY
- 12 GRAPHIC DISPLAY SELECTION
- 13 CHARACTERISTIC INPUT
- 14 CHANGE IN PROJECTION
- 15 ZOOMING
- 16 PAPER PRINT
- 17 THE END

FIGURE 13 AS 365 N - SIMPLIFIED HALF MODEL OF "HUB-MAST" ASSEMBLY

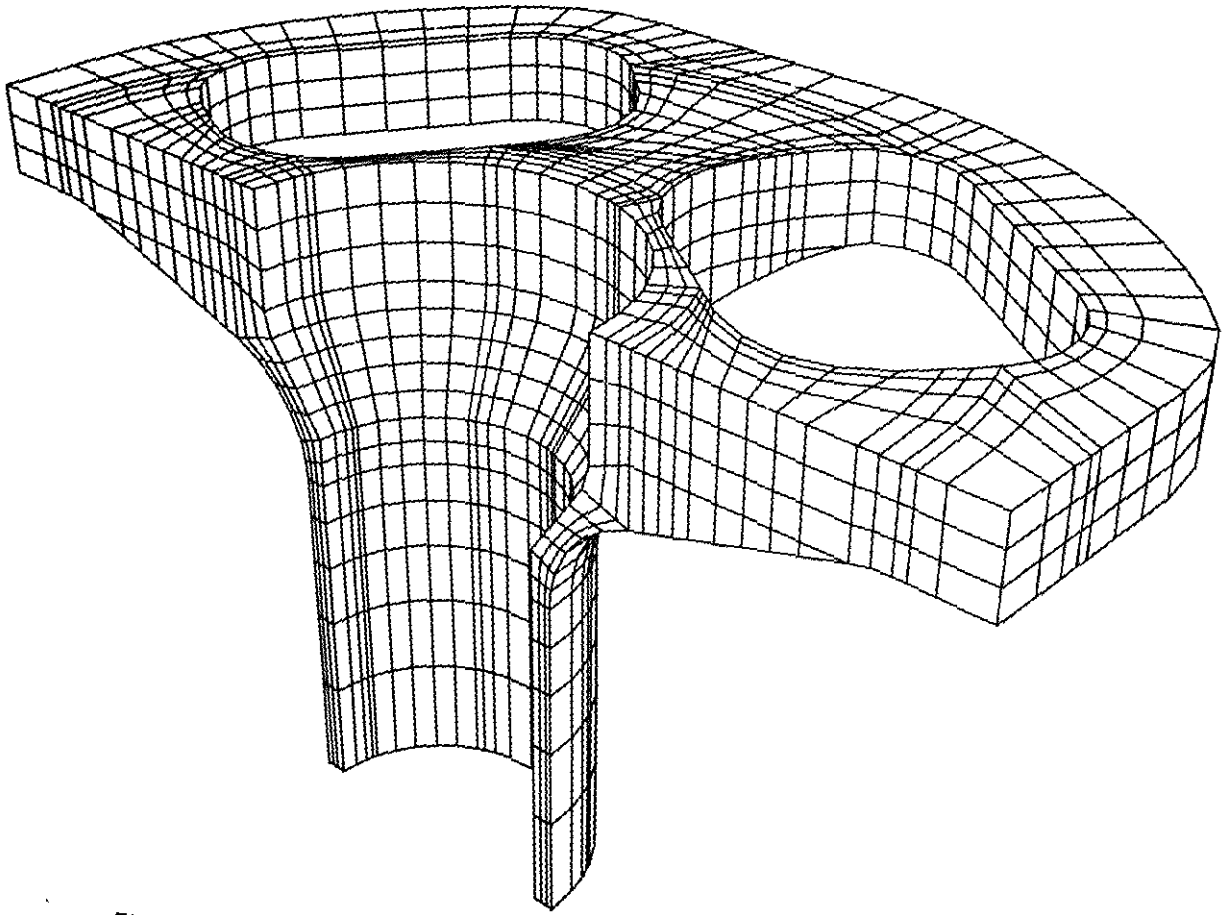


FIGURE 14 AS 332 - SPHERIFLEX ROTOR HUB - HALF MODEL