

8. ALGOR ®

The Algor software represents a very powerful tool for the engineer who needs to perform a finite element analysis. It allows the engineer to perform mechanical or structural analyses with basis on the finite element method, and to arrive to diverse results, like the stresses caused by change temperatures, the effect of a fluid flowing through any element, etc.

8.1 Introduction

Algor was the first software of FEA with PC interface, and also the first system of analyses which was available for almost every engineer. Nowadays, it is one of the most powerful softwares for finite element analysis. This software was developed by the company Algor Corporation, which specializes in the development of computational tools for the design of mechanical elements with basis in the high technology analysis. The finite element analysis (FEA) was first introduced in 1984, and the interface for CAD software in 1985. In these days, it has around 20,000 users all over the world performing analysis programs.

In 1993, Algor Corporation incorporated the first mesh method from solid blocks in three dimensions, and later the module of mechanical events simulation was incorporated, which combined cinematic phenomena, rigid bodies, flexible bodies, and analysis of non-linear stresses. Besides, to make the software more “friendly”, it was developed in a Windows ambient, and this made easier adding any forces, boundary conditions and any necessary parameters for the analysis.

Algor analyses are based, as mentioned, in the Finite Element Method. This is a very important tool for engineers, because it helps them to predict how an element is going to behave under a wide variety of conditions that can be simulated. This method, as mentioned in the last chapter, works by dividing a real object into millions of small elements which have almost the same shape. Then, the behavior of each element is analyzed by mathematical equations to see how it is going to behave. Then, the computer joins each of the small analyses made to each element, and simulates how they are going to behave as a whole element.

When performing a stress analysis, there are some steps to be followed. In general, there are three big stages: pre-processing, processing and post-processing. Then, these stages can be divided into steps, which are the following:

1. Create a mesh (a grid of nodes and elements) that represents the model.
2. Define a unit system.
3. Define the global properties of the model
4. Define the properties of the element
5. Apply the loads and the boundary conditions
6. Assemble the element stiffness matrices
7. Solve the system of linear algebraic equations
8. Calculate the results
9. Review the results
10. Presentation of results

The first five steps represent the first stage, the next three steps represent the second stage, and the final steps represent the last stage. Once all these steps are completed, a successful analysis is obtained, and the behavior of the element can be predicted.¹

8.2 Applications

With the aid of Algor, engineers can perform stress and displacement analysis in complex parts that are suffering the effect of constant static or dynamic loads, or of loads that vary with time. These analyses can be made with linear or with non linear materials, depending on the need. Another important tool of this software is that it can perform analysis to see how much do thermal loads affect some elements, and as a result, a temperature profile or a heat flow through a particular area can be obtained. Finally, the software provides the necessary elements to make analyses of fluid flow and of electrostatic, and these analyses can be done in a wide range of phenomena.

Summarizing the last paragraph, Algor is employed to predict the behavior of elements under almost all physical phenomena such as:

- Mechanical stress (stress analysis)
- Mechanical vibration
- Heat transfer (conduction, convection and radiation)
- Fluid flow (both liquid and gaseous fluids)
- Various electrical and magnetic phenomena

¹ Data taken from Algor's User Guide, Algor Software version 13 ©

- Acoustics²

Once engineers have performed analyses of any physical phenomena, they can predict how an element is going to behave, and they can decide if those results are what they are looking for, or if the element has to be redesigned to obtain better results. An advantage of this software, at least in the latest versions, is that many types of analyses can be made to one element. For example, first a static stress analysis can be made, and if it fulfills the needs of the engineers, they can then perform a fluid flow analysis to the same element, and then a heat transfer analysis and so forth.

A brief description of each mode of analysis is going to be made next. It is important to understand this in order to realize how powerful Algor is, and to know the tools available for many kinds of analyses. The types of analyses, as mentioned before, are: linear, nonlinear, heat transfer, fluid flow and electrostatic.

For linear analysis, there are subdivisions of it, and these are: linear static stress, linear mode shapes and natural frequencies, linear transient stress (direct integration and modal superposition), linear response spectrum (modal superposition), linear random vibrations (modal superposition), linear frequency response (modal superposition), linear critical buckling load and finally linear mode shapes and natural frequencies with load stiffening.

The most common of these analyses is the Linear Static Stress. This type of analysis means that a load is being applied, and that it does not vary over time. Also, the

² Data taken from Algor's User Guide, Algor Software version 13 ©

change in direction of the loading due to deformation is small, and this load will produce only small deformations, deflections and rotations, and they will be in the linear elastic region of the stress-strain diagram. Other two important characteristics of this type of analysis are that the boundary conditions will not change significantly, and that the element will not suffer inertial effects. The data needed for this type of analysis is: static load (forces, pressures, etc.), elastic material data, boundary conditions, and the direction on which gravity is acting. Then, the static loads will be assumed to be constant, and the dynamic effects will be neglected.

This analysis allows engineers to know the effects of static loads over specific areas of an element. That way, after performing the analysis, the results of stresses, deformations, strains, shear and axial forces are going to be known. Once these results are known, the element can be subjected to redesign and then subjected to analysis again.

The other types of linear analysis are also important, but they may not be as used as the linear static stress analysis. With the other analyses, a lot of things can be done, such as determining the frequencies of bodies that are vibrating, the effects of these vibrations over specific areas of elements (in order to avoid resonance), to determine transient vibrations, the effect of shock (short time and violent conditions), to determine the effects of steady shaking, to determine how much a plate buckles, to determine the effects of forces that are compressing and tensioning an element and that at the same time it is subjected to vibrations.

Now one step more can be taken and the non linear analysis can be described. This analysis can be performed when, as the name indicates, the element is going to enter in a zone of the stress-strain curve that is not linear, which means that the element is going to suffer permanent set or deformation. For this case, the load will vary in a period of time, so it would not be constant. Another important thing to consider is that the model will move during the analysis and that the inertial effects play a very important role. There are two types of nonlinear analyses: Mechanical Event Simulation (MES) with nonlinear material models, and Accupak VE natural mode shapes and frequencies. When performing these analyses, it is very important to consider the consistency of analysis types, material models and physical models. The most important of these analyses is the MES with nonlinear material models, because it simulates how an element moves during any application of variable loads.

The analysis of Mechanical Event Simulation with nonlinear material models allows engineers to see how an element behaves under the action of loads that vary with time and that cause rotation, swinging, buckling or oscillation. So Algor can analyze simultaneously mechanical events that involve long deformations, kinematic motion, nonlinear material properties and forces that arise as the result of motion, and then make a calculation of the resulting stresses. One great advantage of this analysis is that not too many assumptions have to be made, and that leads to smaller chances for mistakes. Almost all of the assumptions are made by the software, for example, "...we assume that the design force is usually indeterminate and results from some type of action or motion. In this scenario, force and stress are functions of displacement or deformation." (Algor

User's Guide, Algor 13®). Maybe the most important advantage of event simulation is that it creates a "frame by frame" record of the event, and not only the final result. In this type of analysis a video can be generated to have a better understanding of the behavior of the element. Summarizing, there are four important things that even simulation allows engineers to do: 1) describe the external influences that control the event, 2) let the software figure out the loading, 3) see the part as it really acts and 4) create realistic presentations that even non-technical people can understand.

In order to perform the even simulation analysis, there are some steps that have to be followed:

1. Determine the type of element to be used
2. Make a finite element model of the geometry of the part
3. Specify linear material properties
4. Select the kind of analysis
5. Specify the length of time the analyzer wants to observe the event
6. Specify boundary conditions
7. Specify the gravity field
8. Specify constant or time-dependent acceleration field
9. Specify any revolutions or angular acceleration
10. Specify the starting position of the model in three-dimensional space
11. Specify any initial displacements along with a displacement vs. time curve
12. Specify the location of all impact surfaces by coordinate position
13. Specify forces with force vs. time curves

14. Specify pressures with pressure vs. time curves
15. Specify initial directions of pressures
16. Specify the density of materials
17. Initiate an interactive process to perform the event
18. Produce graphical plots of stress contours, motion, actual deformations for each picture recorded during the event simulation
19. Produce time based animations of the event from start to finish in real time.

The third type of analysis that Algor software can perform is the Heat Transfer analysis. This kind of analysis is used when the heat distribution through a model wants to be determined. In this case, the loads consist of heat generation, convection, radiation, heat flux and applied temperatures. After a proper heat transfer analysis is done, a thermal stress analysis can be performed to realize how the stress distribution due to temperature distribution is. In general, there are two types of analyses that can be done for heat transfer: Steady State Heat Transfer and Transient Heat Transfer.

The Steady State Heat Transfer analysis is used when the engineer needs the temperature results after the thermal loads have been applied, and when he is sure that the thermal effects are going to be stable (they are not going to change with time). Another important consideration for this analysis is that the thermal loads are constant over time. This analysis can also be divided into two: Linear and Nonlinear analysis. “Linear steady-state heat transfer occurs when the material’s conductivity is not dependent on temperature and when radiation is not applied.” (Algor User’s Guide, Algor 13®). On the

other hand, “when the conduction coefficient is a function of temperature (orthotropic material model) or when radiation is applied to the model, multiple iterations need to be performed to solve the steady state heat transfer analysis problems.” (Idem). The process to follow is that shown in Figure 43. As seen, many iterations have to be done until finding a temperature in which the difference between the prior and the resulting temperatures is acceptable.

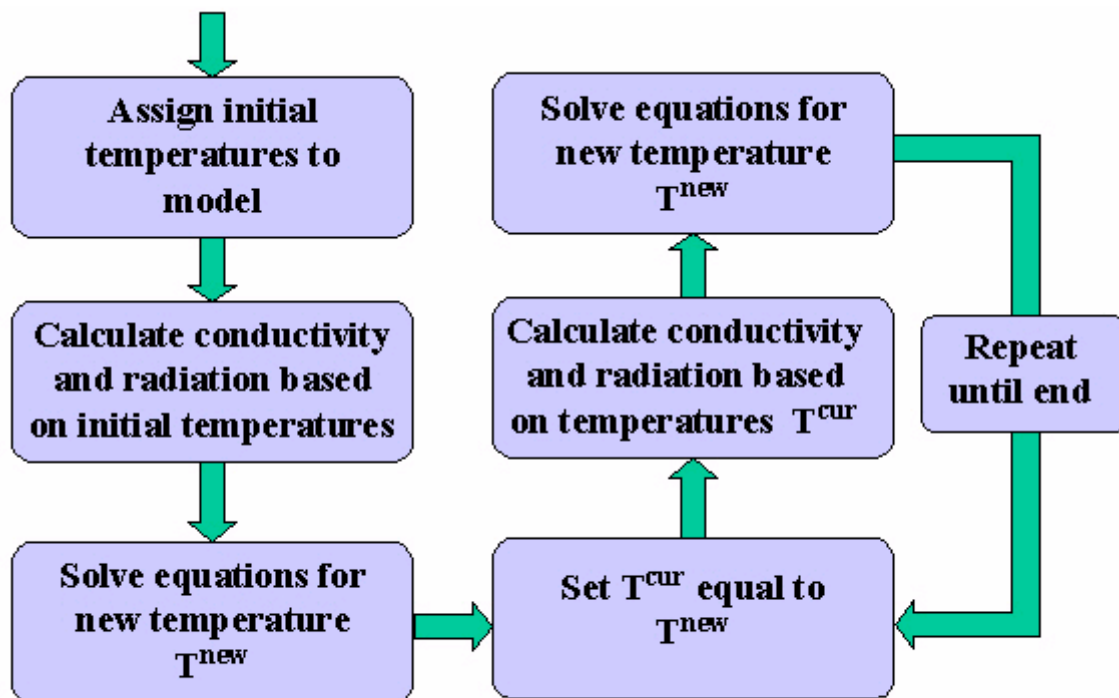


Figure 43 (Algor User’s Guide, Algor 13®)

As mentioned, the other type of heat transfer analysis is the Transient Heat Transfer. This analysis is used when the engineer wants to see an element heat up over a

certain period of time due to thermal loads and when these thermal loads vary with time. This analysis will bring up the temperature distribution and the heat flow of an element that has time dependent temperature conditions. To be able to perform this analysis, the specific heat constant and the density of the material have to be known. This analysis also divides into linear and nonlinear. The analysis and the procedure to do it is the same, but the only difference, as mentioned before, is that temperature is not constant over time.

The next type of analysis is the Fluid Flow analysis. These analyses can be done from two points of view: when the flow is flowing at a constant velocity, and when the fluid suffers the effect of an acceleration during the analysis. This type of analysis "...can simulate the flow of incompressible substances around obstacles such as airfoils or within hollow places, such as lid-driven cavities." (Algor User's Guide, Algor 13®). This software is capable of performing analyses with the following conditions: Newtonian fluids (shear proportional to the viscosity), viscous fluids (nonzero friction), incompressible (constant density), fully filled volume (flow in equals the flow out of the model, no moving air pockets), isothermal (material properties independent of temperature), single fluid (no mixtures), steady state or unsteady (transient) flow, 2-D or 3-D models and laminar flow with turbulence.³ The steps that have to be followed to create a fluid flow analysis are shown in Figure 44.

³ Data taken from Algor User's Guide, Algor Software version 13®

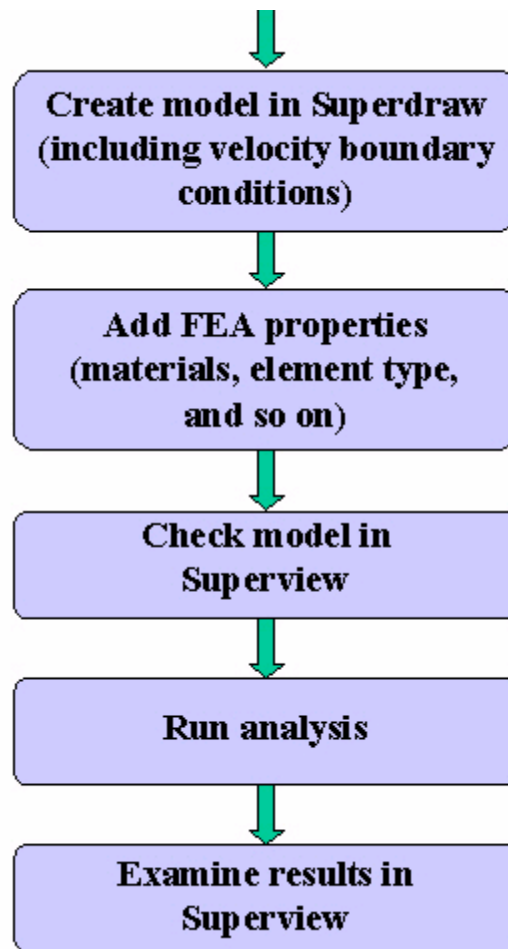


Figure 44 (Algor User's Guide, Algor 13®)

There are two types of analyses that Algor can perform for fluid flow: Steady Fluid Flow and Unsteady Fluid Flow. The first one performs analysis of incompressible flows that have reached steady state, while the unsteady fluid flow performs analysis of incompressible flows that have not yet reached steady state and therefore fluctuate in time.

Finally, the last type of analysis that Algor can perform is the Electrostatic analysis. This is used when studying the "...design of a structure with focus on the structure's electric conduction and insulation properties. It is intended to be used when

electric currents or electric fields are an issue in a product's design..." (Algor User's Guide, Algor 13®). With this software, the model can be created, and then the electric elements can be defined, as well as boundary conditions and properties. Then, the model can be processed with these factors and finally the structure of the current or field results can be viewed. The method to be followed for this analysis is that of Figure 45.

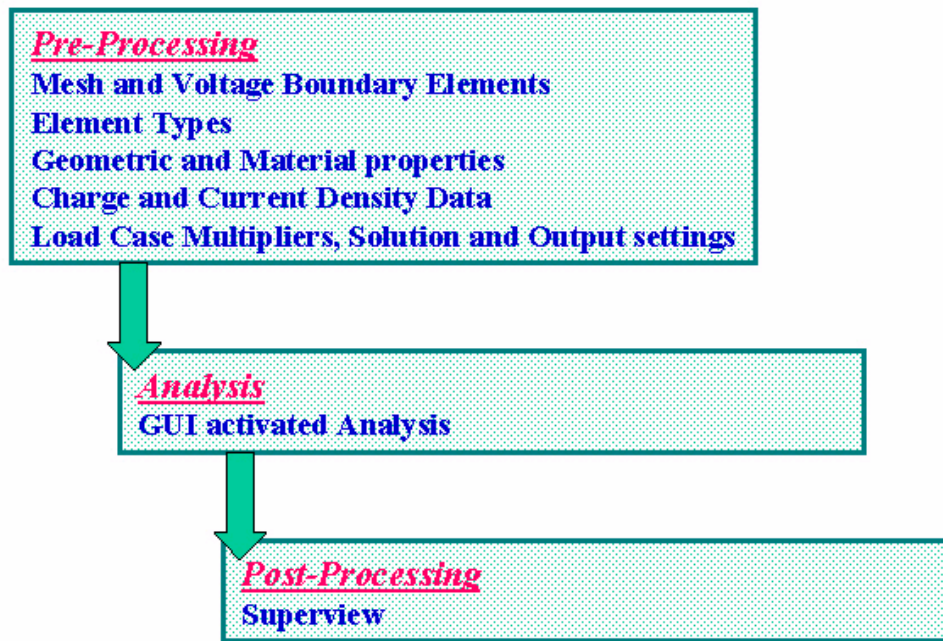


Figure 45 (Algor User's Guide, Algor 13®)

There are two types of electrostatic analysis: Electrostatic Current and Voltage and Electrostatic Field Strength and Voltage. The first analysis is used when engineers are concerned with the current and voltage distribution in a part due to applied voltages and conduction. The second type is used when engineers are concerned with the electric field distribution in a fluid or insulator due to induced voltages. When performing an electrostatic current and voltage analysis, the results of it consists of voltage at nodes, current flux density at centroid and current flow through edges or faces. The data needed

for this analysis is, for input, electrical conductivity, voltage at boundaries, current generation per unit volume and point current sources, and after giving these values, the analysis will give the output data mentioned before. When performing an electrostatic field strength and voltage, the results of the analysis consists of voltage at nodes and electric displacement fields at element centroids. In order to get these results, the data needed for input is permittivity, potential at boundaries, charge density and point charges.

8.3 Modeling

The process of modeling is one of the most important steps for performing an analysis because the success of the analysis depends on how well the element is modeled in the software. This process can be done in many ways. One of the ways is through CAD software, and the only condition that it has to fulfill is that it the file has to be compatible with the versions that Algor can import. For example, an element can be modeled in Pro-E® (which is the case of this thesis) and then imported to Algor to perform the proper analysis, but fulfilling the requirement that the solid has to be saved in one compatible file. This is a very helpful tool when engineers deal with very complex elements, like assemblies. Another option is that the element can be modeled in the Algor software, and then it can be subjected to the desired analysis. The modeling in Algor includes two-dimensional and three-dimensional elements, and they can be solids or simply surfaces, according to what the analysis specifies.

When simulating any phenomena in Algor, some stages have to be followed, and during each one of these stages information is added to the model, the parameters of the

problem and the conditions of the phenomena are defined, and finally the presentation of results. These stages are known as pre-processing, processing and post-processing. Algor presents for each of these stages a different interface. It has five main interfaces, and on each one, different functions can be performed. These five interfaces are, for the case of Algor 13: Superdraw III, CAD Interface, FEA Editor, Superview IV and Superview.

- **Superdraw III**

Superdraw represents a powerful tool of CAD which allows the user to create wireframe or meshed models. If the model needs any modification, this will be done in this interface, because it provides some tools to create basic figures, and also to create any type of mesh, from surface to solid. Another important function of this interface is that the layers can be changed in order to have a better understanding of the model. The same thing can be done with surfaces and with groups. The file format that this interface uses is .esd.

One big problem of this interface is that it is very limited. When modeling complex elements, Superdraw does not represent a very powerful tool to draw. For these cases, it is better to generate the model in CAD software, like Pro-E, AutoCAD or Solid Edge, and then import it to Algor. The use of this interface is recommendable for models that are not too complex, and that can be drawn simply by specifying the coordinates of the element, and using simple drawing tools like circles, rectangles, lines, mirrors, rotation of lines, etc. If any modification wants to be made for a previous model generated in a different CAD software, they can be done in this interface. The

modifications can be performed after meshing the model. Once the model is finished, and it contains all the necessary characteristics for the analysis, it can be exported to the CAD interface.

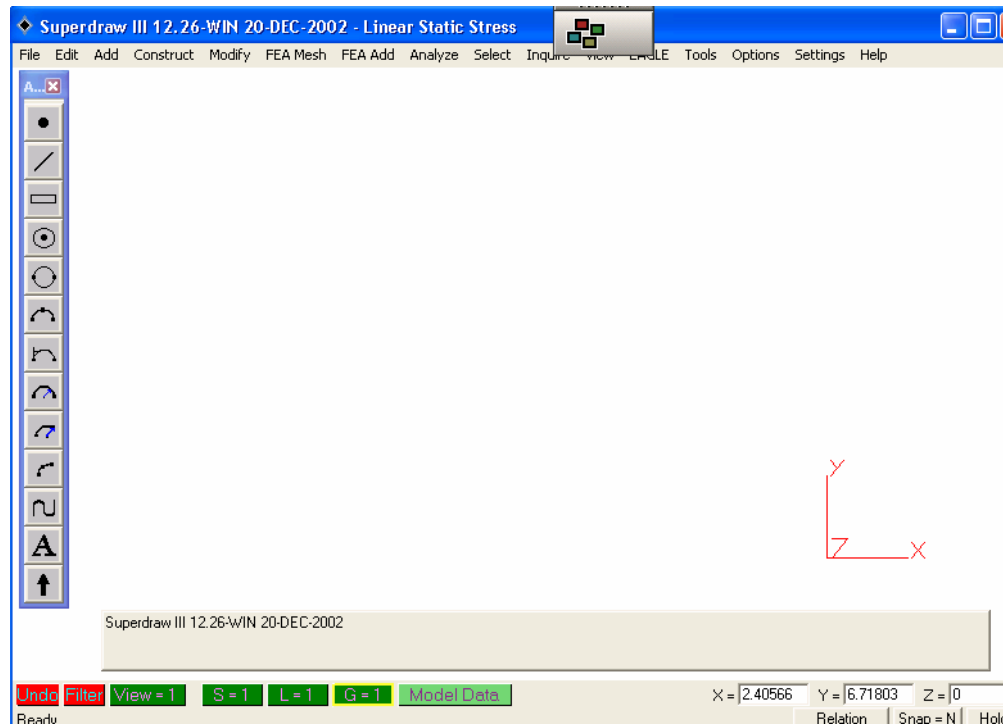


Figure 46.- Superdraw III Interface

- **CAD Interface**

It has been mentioned many times that an element can be created in CAD software and then imported to Algor to perform the proper analysis. When the engineer finds itself in this situation, CAD Interface is very useful. “This interface is used to create meshes on models that were created by a CAD solid model package. These can either be the part or assembly file from the CAD package or can be a universal format.” (Algor User’s Guide, Algor 13®). When using this interface, first a surface mesh can be created,

and after this, the solid mesh can be created. The format file that this interface uses is .dmit.

One important tool of this interface is that it allows generating a mesh interconnected by nodes in the surfaces of the imported solid, and it also allows the user to seek for any errors or mistakes in the geometry and to find critical interconnection points of nodes inside the mesh. An important thing that has to be known in order to use this interface is how to modify the mesh size and density. The mesh has to be designed in a way that it represents the model as close to reality as possible. The mesh has to have a size and density that allows the user to identify all the important characteristics of the model. Maybe the most important thing to remind here is that the mesh has to be as fine as the element allows in the zones where the analysis interests the most. An example of this interface will be seen later in Figure 47.

- **FEA Editor (FEMPRO)**

This interface is, doubtless, the most important of Algor, because here is where the forces and boundary conditions are added to the element, as well as the element specifications. “This interface is used to define the properties of the model and apply all of the loads and constraints. The model can either come from Superdraw III or from the CAD Interface.” (Algor User’s Guide, Algor 13®). The file format used for this interface is .esx.

Managing of the model in this interface plays a very important role, because if a load or a constraint is added wrongfully, then the analysis will be wrong. The user has to be familiarized with all the tools of the interface in order to perform a successful analysis. This is the link between the pre-processing and the post-processing. The most general steps to follow in this interface for one of the simplest analysis is to add the boundary conditions (constraints), then add the loads. Once this is done, the element specifications are loaded (which means if the acceleration is having any effect on the material, in which direction, if it has any pressure or temperature acting, etc). After the specifications of the element are given, the analysis is performed, and it will open the next interface. An example of this interface will be seen later in Figure 47.

- **Superview IV**

Superview is an interface of post-processing. After performing the analysis in FEMPRO, a new window is opened, and the results of the analysis are displayed. “This interface is used to view the results of all linear and thermal analyses. It uses many output files that were created during the analysis to show how the model reacts to the applied loads.” (Algor User’s Guide, Algor 13®). It presents many options to view the results, generally as graphics or as tables. The available options for results are: stresses, deformations, vectors, displacements, etc. And inside each of these options there are other options. For example, for stresses there are maximum stresses, maximum principal, Von mises, etc. Each one of these options makes this interface a really powerful tool for post-processing.

Other important things can be done in this interface. For example, a part of the element can be hidden and then the stresses at unseen points can be seen. The interface presents the model as millions of little squares, and any of these squares can be hidden in order to realize the situation of stresses, deformations, etc. at specific points. An important option of this interface is that the values of any stress or deformation can be known at specific points by simply clicking in the node of interest, and then asking for the desired values.

There is also another interface of Algor named Superview. This is the same as Superview IV, but the results presented here are those from any analysis except from linear and thermal analyses. The way of working is the same as the other interface, with the same options and advantages. An example of the Superview interface can be seen in Figure 47.

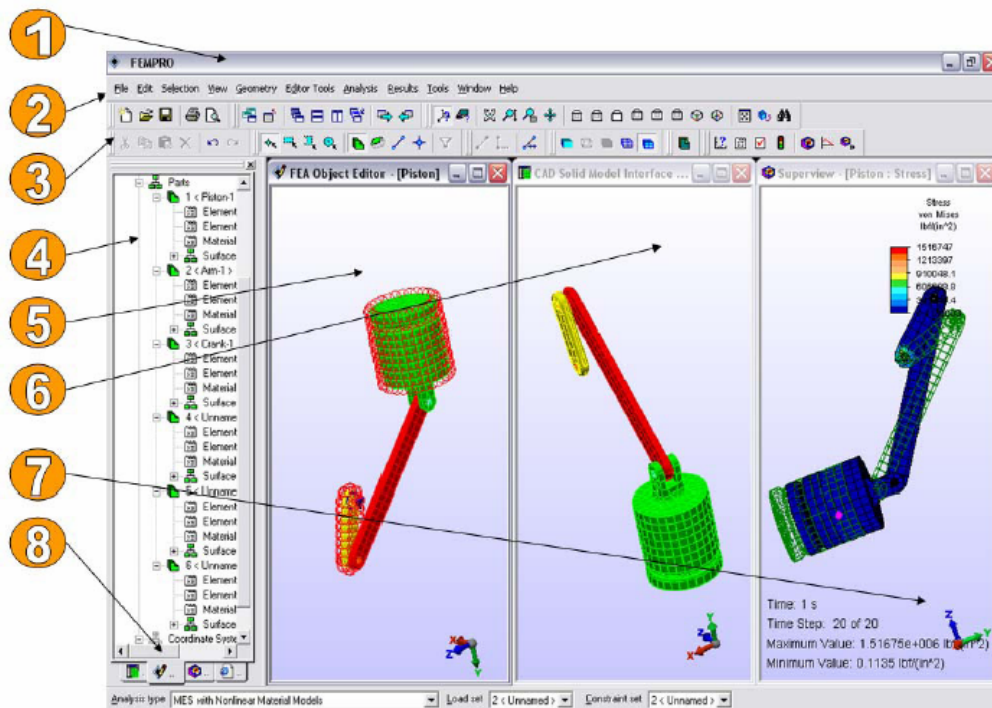


Figure 47 (Algor User's Guide, Algor 13®)

1. Title bar
2. Menu bar
3. Toolbars
4. Model tree
5. Working area (FEMPRO)
6. Working area (CAD Interface)
7. Miniaxis
8. Status bar

For Figure 47, the first window on the left is the window of FEA Object Editor (FEMPRO). Then, the central window is the CAD solid model interface, and finally, the window on the right is the Superview interface.

- **Web Report Interface**

One additional tool of Algor is the web report interface. This is a special interface to create reports of the analyses performed in the software, whichever the analysis may be about. It is useful only for post-processing, and the file type is html, which may be uploaded to the internet, and there consulted by anyone.

The reports can be created with charts, tables and graphics generated by the other interfaces of Algor, and then added to the main page. It includes the integration of images generated during the analysis, charts created after the analysis, and videos of how the element behaves under the effect of any specific load. The main advantage is that the

report is created in a file type that can be read by any internet explorer, so the file is available for everyone who needs it. See Figure 48.

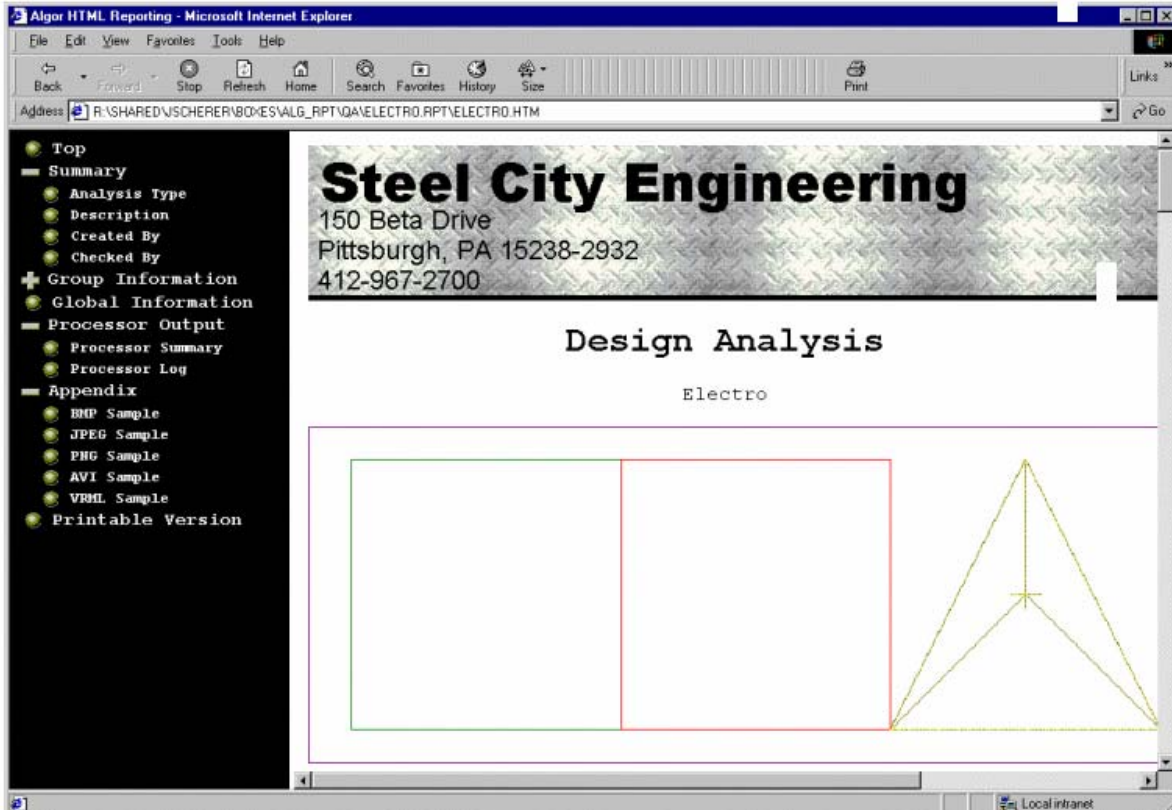


Figure 48 (Algor User's Guide, Algor 13®)

8.4 Determination of the stress concentration factor in shafts with stress concentrators subjected to fatigue, flexion and torsion loads

As mentioned in past chapters, the determination of the stress concentration factor can be done in many ways, for example, photoelastic analysis, tests in laboratory and by the finite element analysis. In this case, the analysis is done by using a FEA software, and this allows the engineer to have more accurate, easier and faster results. Using this tool,

the results can be taken accurately at specific points of interest in the element. For example, for the case of this thesis, the most important zone is the lower point of the central notch; so by simply clicking on any node of interest, the results of Von Mises stress, maximum principal stress and deformation can be known.

The first step to determinate the stress concentration factor is to determine the shape of the element. For this case, a half-circle notch is going to be analyzed at the beginning. The objective is to determine first the stress concentration factor in one three-dimensional element that has only one notch in the center (see Figure 49). The data of the test probe is:

- Length = 100 mm
- Diameter = 10 mm
- Notch radius = 1 mm

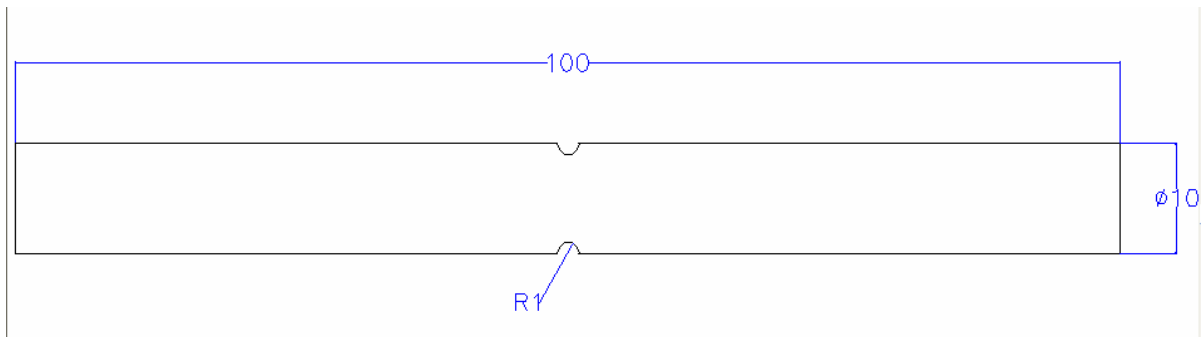


Figure 49.- specimen data

Once this analysis is finished, the next analyses to perform are with three notches. This is done to see the effect that other stress concentrators have in the central notch. The shape of the notches can be any, but the designer has to be aware that the success of the stress concentrator depends on it (if it is a very sharp shape, it will generate bigger stress concentrations, and it will diminish the service life of the element). But an important

point to take into account is that the depth of the complementary notches has to be the same as the depth of the central notch. By doing this, it is being assured that the effect of the complementary stress concentrators is going to be reflected directly on the lowest point of the central notch. Another important point to consider is that, in order to get a stress concentration factor chart, the analyses have to be made at a different distance from the central notch. For example, for the first analyses made in this thesis, the shape of the complementary notches was also half circle of the same radius as central notch ($r = 1$ mm), and in order to get the chart, the distance between them was, at the beginning, of 1 mm, and then it was increased by one millimeter until reaching 12 mm. A chart was built with this data, and it was determined that the important effect zone of the stress concentrators was from 0 to 4 millimeters. So then again, analyses were performed with the same notch shape, but the only difference is that the distance was increased from 0.25 mm in 0.25 mm. That way, after performing sixteen analyses, a more accurate chart was obtained.

Something else that should be considered is that the distance between the central notch and the complementary notches does not have to be too big. This is done because if the distance is too big then the stress concentrators will stop having a direct effect on the central notch, and they will act as separate notches, so instead of having only one big concentration in one notch, the element will have three, and it will be more prone to failure (because it has three high stress concentration zones). In order to figure out which an optimum distance is, a chart can be built, and in the point where the chart overpasses the stress concentration factor of the central notch, the distance can be taken as the

optimum. Once this distance is known, new analyses can be made at a smaller distance than the last one, as was the case of the last analysis explained.

After the shape of the element is known (and taking into account what was mentioned before), the next step is to make the model in a CAD software or in the Superdraw III interface. For the analysis made in this thesis, first it was attempted to make the solids in a CAD software (more specifically Pro-Engineer), but it presented problems when exporting it to Algor. The alternative chosen to this problem was to model the elements in the Superdraw interface. Even if this is a more complex drawing software than Pro-E, it is better to model simple elements in here because it allows the user to make a finer and denser mesh, which will result in more accurate results. What was done to make the model is explained in the next paragraph.

First, after opening the Superdraw interface and giving a name to the model analysis, the units of the element have to be defined. To do this, the user has to enter to the Model Data window, and choose the proper units. For this analysis, the model was drawn in millimeters, so the system used is the International System, but this had to be customized. The International system uses meters as a unit of length, so it only had to be changed to millimeters. After doing this, the window is closed, and the view is changed to YZ Right, and also the miniaxis is set to be displayed permanently at a specific point of the monitor. Once this is done, only one line is drawn in the interface by giving the specific coordinates of the lines and arcs. When that line is finished, it has to be divided into little segments. The number of segments depends on how fine the user wants to mesh

the element (because later this line will be rotated in order to obtain the surface of the element). For the case of this thesis, the central notch was divided into sixteen segments, and that way accurate results can be obtained. To divide lines and arcs, the user has to enter to the Construct menu, and then choose divide. There, the number of segments will have to be specified, as well as if the user wants software to make the division into lines, or just divide the shape in segments. When all the lines and arcs are divided into the required segments, the line has to be rotated to form the surface of the element (in this case a shaft). See Figure 50.

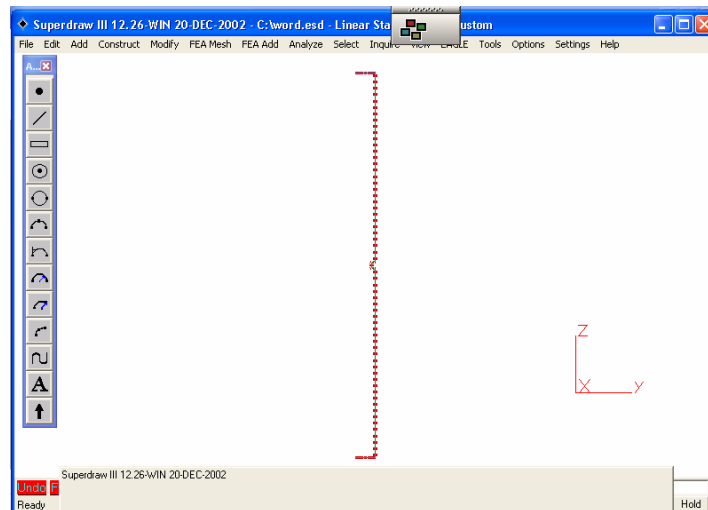


Figure 50.- one line divided in segments

To rotate the line, the user has to specify first the angle and the number of segments in which the line is going to be rotated. To do this, menu Modify has to be accessed and then in Quick RMS Setup. There, the number and angle have to be specified. For this case, the angle was of 10° and the number of 36, that way the line will rotate 360° to form the surface. Once this is done, the user has to accept the information, and then enter again in the Modify menu, but now Copy has to be selected. To perform

the copy, an upper left box has to be checked, which will indicate that the copies will be joined, and then just press in Rotate Last. After clicking there, the model will automatically rotate the 360° specified before. Even if then the model looks ready to create a solid mesh, one last step has to be done. The user has to assure that the model does not contain any duplicated lines. To do this, the user has to access to the Modify menu, then to Clean, and select Duplicate. A new window is opened, and then Perform Cleaning has to be clicked. Now the model is ready to be meshed with a solid mesh (see Figure 51).

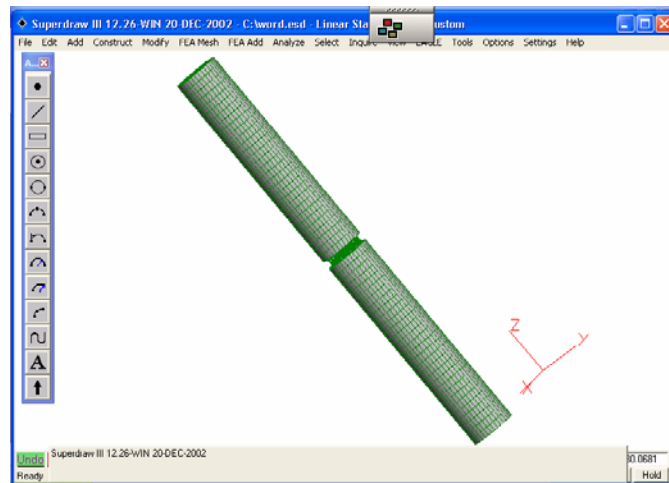


Figure 51.- surface of the shaft generated after rotating a line 360° and joining the copies

Since now the surface of the element is created, a solid mesh has to be applied to it in order to convert it from surfaces to just one solid body. To do this, the user has to enter to FEA Mesh, and there click Solid Mesh Generation. After doing this, a new window will be displayed, where the user has the options of choosing the type of mesh. For this thesis, the mesh used for this type of analysis is Bricks/tetrahedra, and every part of the surface is going to be meshed, so a click has to be given in Mesh All. Once doing this, the user has to click in Generate to start generating the solid mesh. The solid

mesh is generated, and the user only has to accept the next two windows that display on the monitor. The element with a solid mesh would look like Figure 52.

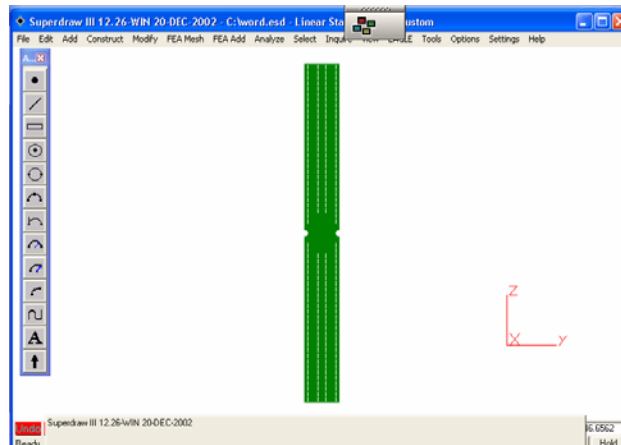


Figure 52.- element with a solid mesh

Once the solid mesh is generated, the element has to be exported to the FEMPRO interface, where the material will be defined, as well as the boundary conditions (restrictions) and the applied loads. To do this, the user has to click in File, and then in Export to FEMPRO. After doing this, the software will automatically transfer the file from Superdraw III to FEMPRO, and it will generate different files for the analysis. Once the element is in the FEMPRO interface, the element can be defined. By clicking on the solid, all of it will be automatically selected. Then, giving a click on the element with the right button of the mouse, a new window will appear. There, the user has to select Modify, and then go to Material. After doing this, the Material Library will be displayed, and the user has the option of choosing between all of the materials existing in there, or there is also the option of defining a new material by specifying its mass density, modulus of elasticity, Poisson's ratio, thermal coefficient of expansion and the shear modulus of elasticity. For this analysis, a predefined material is used, and that is Steel ASTM-A36, also known as Structural Steel. The material window is then closed.

The next step is to add the boundary conditions and the loads. To do this, the user has the options of selecting nodes of the element by clicking on them, or choosing a box where it will select all of the nodes contained inside that box. For this case, Selection menu is entered, then Shape, and then Rectangle. Then, Selection menu is entered again, then Select and then Vertices. By doing this, the user is specifying that by drawing a rectangle on a specific area of the solid, only the vertices will be selected. When all the desired vertices are selected, a right click has to be given on the vertices selected, and a new display will appear. There, the user has to enter to Add and then to Boundary Conditions. A new window will be displayed, where the user has the option of choosing the type of boundary condition: fixed, free, pinned, no rotation, x symmetry, etc. When choosing any of these options, some or all of the degrees of freedom of the system are being restricted. For this analysis, all elements will be fixed. After accepting the information, the user can select the point or points where the loads will be applied. In this case, the load is going to be applied in the center of the lower face of the element, and it will have a value of 100 N. The value and the direction are specified later. To do this, the user has to choose the point where the load is going to be applied. Once this is done, the user gives a right click, and a new window displays, where the option Add has to be selected, and then Forces. There, a new window appears, where the load and the direction of it have to be specified. The load for this analysis is 100 N and the direction is $-Z$. When the user has accomplished all of the steps mentioned before, there is only one more step to perform, and that is to define the Global Analysis Parameters. To do this, the user has to enter to the Analysis menu, and then to Parameters, where a new display will appear. If the element is under the effect of any pressure, temperature or voltage, it can be

specified here. It also can be specified if it is suffering the effect of gravity, of a centrifugal force or of temperatures or voltages at specific nodes. For the analysis performed in this thesis, it is only specified that the part is under the effect of gravity. When all these steps are accomplished, the element is ready for analysis (see Figure 53).

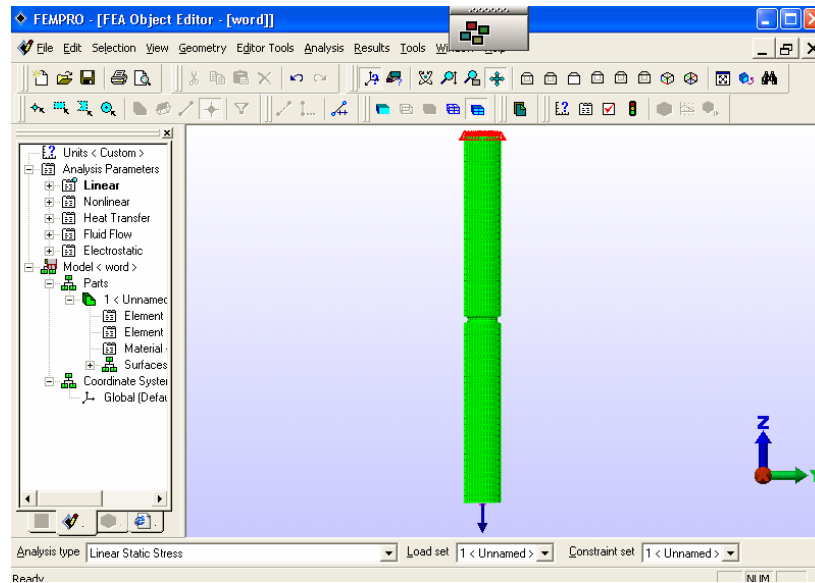


Figure 53.- element ready for analysis

To perform the analysis, the Analysis menu has to be accessed, and then the option Perform Analysis has to be selected. After doing this, the software will start the proper analysis, and will display the results in a different interface, which is the Superview interface. Once the analysis has been performed, a display like the one seen on Figure 54 appears, and there, all data that it is interesting for the engineer can be found, like stress, deformation, strain, reaction vector, etc. And inside every option of stress, deformation, etc, there is also additional data that can be found, like Von Mises, Tresca, Maximum Principal, and so on.

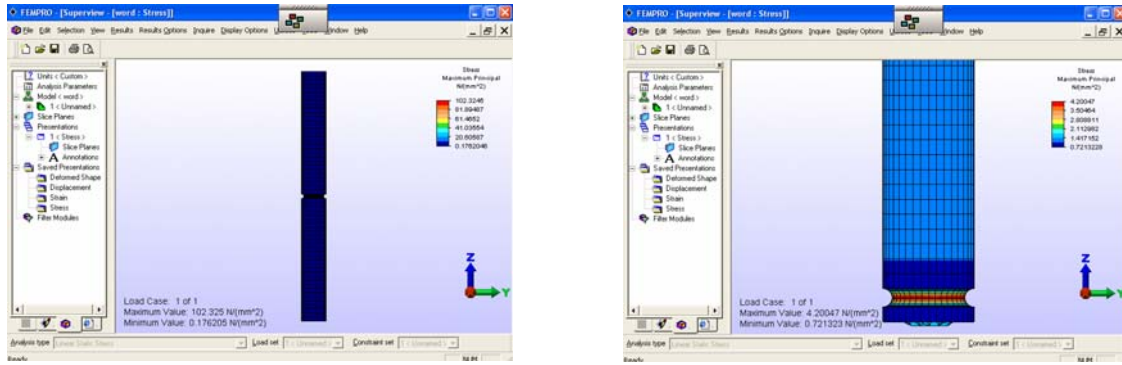


Figure 54.- Results of the analysis displayed in Superview interface

The process described in the past pages is that followed to find the stress concentration factor for shafts. Maybe the only difference to find K_t for different types of loads is the type of load. Instead of adding a tension load, a compression load can be added or maybe a moment. To do this, the same steps have to be followed until reaching the step of adding forces. There, instead of choosing the option of Forces, the user has to choose Add and then Moments, and specify the magnitude and direction of them. When performing the analysis, different stress concentrations will be generated, but K_t will be found equally (by asking the computer for the maximum principal stress value at a specific node of the notch, and then dividing it by the nominal stress). For the element, the same data can be obtained (stress, displacement, deformation, vectors, etc.).

8.5 Multiple concentrators in series

In past chapters it has been talked about stress concentrators and the effect that they might have on an element when subjected to any type of load. It has also been stated that a single notch represents a higher stress concentration than if the element contains a series of spaced notches. But now the arrangements of stress concentrators will be

defined, and the advantages and disadvantages of each one will be stated. There are two types of stress concentrators arrangements: in series and in parallel.

The case of stress concentrators in series is the first used in this thesis. A notch is specified at a point, which is generally the center of the shaft, and two complementary notches are added, one on each side of the central notch (see Figure 55). For the first analysis, and in order to obtain K_t , the radius of the three notches will be the same, and the analyses will be done increasing the distance by 0.25 mm each time, starting from 0 mm and until reaching 4 mm. What it is being looking for is for the maximum principal stress at the center of the central notch (the lowest point of it), but taking into account that the stresses at the same point in the other two notches does not raise too much, because if this happens, the stress concentrators will work against the element.



Figure 55

The shape of the additional notches is going to vary in order to find which one is more effective at reducing stress concentrations. The most important thing to take care of is that the lowest point of the complementary notches has to be at the same depth of the central notch. By doing this, it is being assured that the stress concentrators affect directly to that zone (the critical zone).

8.6 Multiple concentrators in parallel

As mentioned before, there are two types of multiple stress concentrators. The second arrangement is in parallel. This type of stress concentrators is not too common in shafts as it is in plates. For the case of a shaft, which is a three-dimensional element, this type of arrangement is rarely encountered because of its complexity, and because of the shape of the shaft.

Some analysis will be performed in order to understand the behavior of a shaft with stress concentrators in parallel. For this case, a key seat groove can cause high stress concentration at certain point. To see if the concentration can be diminished, some concentrators will be added to the sides of the groove. But it has to be mentioned that this is not very commonly used in engineering.