TRABAJO FIN DE GRADO

E.T.S.I.D

UNIVERSIDAD POLITÉCNICA DE VALENCIA

EVALUATION OF THE STRESS CONCENTRATION ON TUBES WITH TRANSVERSAL HOLE BY MEANS OF THE FINITE ELEMENT METHOD

Author:

Ben Jordan Baxter

Tutor:

Juan José Ródenas García

Mechanical Engineering Degree

Valencia, June 2016

Escuela Técnica Superior de Ingeniería del Diseño

Documents

1. Report.
2. Specification Document.
3. Budget.

Escuela Técnica Superior de Ingeniería del Diseño

Document 1

REPORT

Escuela Técnica Superior de Ingeniería del Diseño

INDEX

ILLUSTRATION INDEX

1. SUMMARY

In the following project there is described a method for calculating the theoretical stress concentrator factor for a specific solid, further on mentioned, by means of reproducing its behaviour curve assisted by finite element method, no matter the geometry involved.
The stress concentrator analysed has been chosen from Peterson´s Stress Concentration Factors 3rd edition book and is shown in Illustration 1.

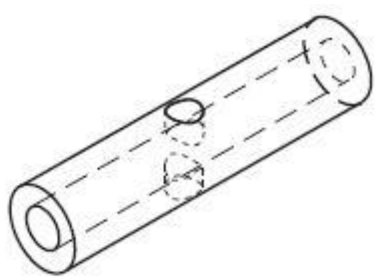


Illustration 1. Circular section tube with transverse hole

In order to obtain the curve corresponding to the stress raiser, the piece will be modelled in ANSYS and loaded with three different load cases, which are a uniaxial stress along the cross section, a bending stress perpendicular to cross section and finally a torsion stress. From the analysis of these three stress states, the maximum stress is calculated. When achieving maximum stress from ANSYS and knowing the theoretical stress, stress factor Kt can be obtained.

After many analyses the results are plotted and then fitted to a curve using generalized least squares method (GLS). As a result of applying this method, the equation for Kt, function of all its variables, appears. Finally, the results are compared to the before mentioned bibliography.

2. HISTORY

Designing problems are restricted in certain ways by the quality and resolution offered by the actual technique. A stress concentrator or also known as stress raiser is a geometrical singularity that alters stress distribution in its surroundings. Now, a stress concentrator can be defined as any section change throughout the whole piece, notch, groove, hole (which is the case of this project) or even a change of direction that the piece holds.

If one uses the traditional, elemental equations and formulas to design structural members, one notices that these equations are applied for constant sections or with slight and gradual changes. So, as expected, stresses obtained via this method are smaller than if we apply a stress concentrating factor that takes account for this effect. In consequence, not applying this Kt factor can result into mechanical failure and subsequent repercussions.

In real life and through every industry it is hard not to mention a solid that doesn´t have any type of shoulder, notch or groove that leads to a stress concentrator but as it seems, not all stress raisers have the same impact and influence on the piece. Consulting further bibliography related to this subject it is found that theoretical stress concentrating factors depend on geometry and involved loads.

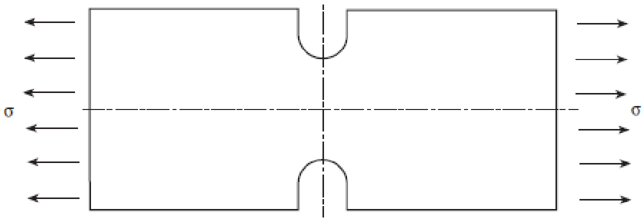


Illustration 2. Theoretical loaded component

In many cases, authors that refer to stress concentration factors differ or simply aren´t capable of advancing in research because the available information is so limited. The need of practical information and research forces erroneous solutions that are not always liable, and less accurate having mechanical designing and calculus on standby.

Never the less, there are some ways for calculating stress concentration factors. There are experimental methods for measuring stress concentration factors including photoelastic stress analysis, brittle coatings or strain gauges. At the same time that these approaches are successful, all methods mentioned have experimental accuracy and/or measurement disadvantages.

Apart from experimental methods during design phase, there are multiple approaches to estimating stress concentration factors. Some catalogues include stress concentration and maybe the most popular is Stress Concentration Design Factors by Peterson, first published in 1953. Also, nowadays and in this project, finite element methods are used in designing processes.

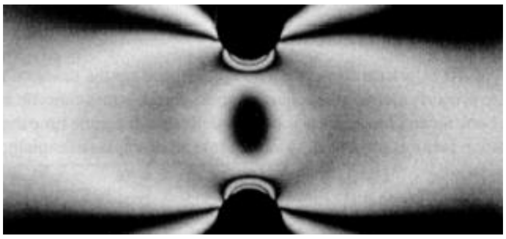


Illustration 3. Stress distribution

As it can seem logic, there will always be small differences between the catalogue, FEM and theoretical values calculated and this is because each method has its advantages and disadvantages. Actually, many catalogue curves were derived from experimental data. The result and final conclusion is that engineering judgement must be used when selecting which data applies to making a design decision correct. Many theoretical stress concentration factors have been derived which may not be analysable and are not testable in a stress lab, but approaching the problem using two or more of these methods will allow an engineer to achieve an accurate conclusion.

As a visual aid to the problem, in illustrations 2 and 3 there is a piece (not the bar analysed in this project) under uniaxial stress. Illustration 3 shows the stress flow through the whole piece and it is seen that stress discontinuities are near notch.

3. INTRODUCTION

Machine components often have areas where the state of stress is reasonably higher than any theoretical prediction. This can be due to:

1. Geometrical discontinuities or commonly known in engineering as stress raisers, being these holes, notches, etc.
2. Internal, microscopic irregularities known under the name of non-homogeneities of material. This defects can de fruit of non-precise manufacturing processes like casting and / or moulding.
3. Surface imperfections. This states as the following level after manufacturing processes. Usually cracks and scratches are surface imperfections created by machining operations.

These stress concentrations are local effects which, as said previously, are function of geometry and loads. In order to quantify the effect of this phenomenon, a stress concentrating factor was defined which englobes all elements related to stress raising. The study of this stress concentrating factor is the main object of study of this project and following up is some relevant information concerning it.

The stress concentration factor Kt can be formulated as the ratio between the maximum or peak stress in the body and another stress taken as a reference, commonly called nominal stress.

$$K_t = \frac{\sigma_{max}}{\sigma_{nom}} \quad (1)$$

$$K_{ts} = \frac{\tau_{max}}{\tau_{nom}} \quad (2)$$

Where Kt is used for normal stress (bending and uniaxial stress) and Kts is used for shear stress, like the bending load case.
The subscript t stands for theoretical stress factor.
The nominal stress of equations 1 and 2 is typically obtained through elementary strength of materials equation, using either net or gross cross section. Further on, both of these concepts will be detailed and explained.
The transverse hole through a pipe or bar with a circular cross section occurs more often than it could be assumed, appearing in lubricant and coolant ducts in shafts, connectors or transmission rods and in many tubular frameworks.

After studying these effects on different components, with alternative materials and other parameters, engineers developing these experiments came to some conclusions, which could be understood as characteristics of the Kt factor.
The first characteristic discovered was that Kt is function of the geometry or shape, but not its size neither material. Additionally, it`s also function of the type of loading but not the actual load. This is, Kt relies on if the load is an axial load, bending or torsional, but not the Newton`s or Newton/m that are applied. The next characteristic may seem obvious but is also mentioned, being that the result obtained isn`t the same having a notch than a hole, for example. When studying the effect of a stress raiser, one must select one stress raiser, if not, results will not be coincident.

As said before, Kt is a ratio between two stress (maximum and a nominal stress) making Kt a dimensionless value. Well, Kt is defined respect to a particular nominal stress, having to choose between a net or gross cross section.
Finally, the last main characteristic is set as an assumption. The hypothesis adopted is to consider the material elastic, homogeneous and isotropic. This affects vastly the material definition limiting the parameters down to two (which the isotropic material possesses), being these Young`s Modulus (EX) and Poisson`s ratio (vxy). This last assumption is narrowly related to characteristic number one mentioned above. Material isn`t an issue when studying stress raisers so why not define an easy material, with only two parameters.

3.1. USE OF STRESS CONCENTRATION IN FATIGUE ANALYSIS

Stress concentration factors play a critical role in any detailed fatigue analysis. Within a structure or component, fatigue cracks are most likely to nucleate in the region where the stress is at its peak. The highest stresses in a body will occur at geometric features including holes, notches, etc. as mentioned before and speaking in theoretical terms, at these points, stress tends to infinite. These peak stresses can be calculated from stress concentration factors.

Therefore, it is primordial to accurately calculate and analyse in order to ensure a reasonable and liable fatigue life. In some special areas of engineering, such as aeronautics, stress concentrators gain strong in importance due to the amount of micro cracks in a plane, but this problem is tackled by Fracture Mechanics theories which are aware of these cracks and tend to control instead of eliminating. For fatigue analysis, the net stress concentration factor, denoted with the symbol Ktn, is used.

One must separate both elastic and plastic ranges when analysing stress concentration factors. All said above is related to an elastic, almost perfect range, where calculus is near enough easily done. Although, in the plastic range one must consider by separate, stress and strain concentration factors.

In conclusion, the existence of notches or a stress raiser has a critical impact towards fatigue in some materials.

3.2. DETERMINATION OF Kt VALUE

The stress concentration factor, associated with a specific geometry and loading condition can be studied by means of experimentation, analysis or computational methods.

Experimental methods: Optical methods, such as photoelasticity, are widely used for experimentally determining stress concentration factor. However, several alternative methods have been used historically: the grid method, brittle-coating, brittle-model and strain gauge.

Analytical methods: The theory of elasticity can be used to analyse certain geometrical shapes to calculate stress concentration factors but it´s not recommended for complex shapes and calculus due to its simplicity.

Computational methods: Finite element techniques provide a powerful and cheap computational method of tackling stress concentration factors. The universal availability of powerful, effective computational capabilities, usually based on the finite element method, has altered the use of and the need for stress concentration factors. As said before, this is the method used for this project.

3.3. NOMINAL STRESS SELECTION

The definition of a reference stress, being either τ_{nom} or σ_{nom}, depends on the each type of problem tackling. It`s important to properly identify this, so called, reference stress in order to develop a correct approach to establishing and finally obtaining a stress concentration factor.

For the definition of the nominal stress, basically, there are two ways. One is setting nominal stress as the stress calculated at any point of the component where the stress raiser isn`t involved. This is, analysing a component without a stress raiser and calculating the stress at this section. Normally, one can go the embedment section and calculate the stress there, being this area a common and known geometry but any other cross section without a stress raiser is valid. This stress is called **gross nominal stress**.

The second way into calculating a nominal stress is approaching the section where the stress raiser relies. This task is a bit more difficult depending on the geometry of the component and specially, the cross section that is generated at that section. From all the possible sections inside a stress raiser such as a notch, groove, hole, the section where the stress will be calculated will be at a plane coinciding with a symmetry plane, usually being this section the middle plane of the imperfection. This stress is referred to as **net nominal stress**.

In this present project, to make calculus simple and avoiding difficult equations related to inertia moments for awkward cross section, results will be calculated with gross nominal stress.

Now the question is how to calculate a nominal stress (either gross or net). It is stated as a stress, so by definition, it`ll be a force divided by an area, resulting in any force unit, for instance Pascals. In this entire project the unit used are those from the International System.

In this project, according to Peterson`s Stress Concentration Factors 3$^{rd}$ edition, there`ll be three load cases or types of loads. The component studied is a circular section pipe with a transverse hole and on end of the pipe with be the embedment and the other will engage all force or moments creating the different load cases. The three load cases are an axial, traction force, a bending moment and a torsional moment. These are the cases described in the bibliography previously mentioned, where all curves are recreated. Let`s describe the formulation behind the nominal stress calculus for each case.

Axial Force: This load case is the easiest to treat with due to the simplicity in its formulation.

$$\sigma_{gross} = \frac{P}{A_{tube}} = \frac{P}{\frac{\pi}{4} * (D^2 - d_i^2)} \tag{3}$$

Where P is the axial load applied and D is exterior diameter.

Bending Moment: The formulation for this case is a bit more difficult but can be deducted.

$$\sigma_{gross} = \frac{M}{Z_{tube}} = \frac{M * D}{2 * I_{tube}} = \frac{32 * M * D}{\pi * (D^4 - d_i^4)} \tag{4}$$

Where M is the bending moment applied, D is exterior diameter and I_{tube} is the inertia moment of the gross cross section.

Torsional Moment: The last load case has a very similar formulation, being such that it`s half of bending moment gross stress, if the loads applied are the same.

$$\tau_{gross} = \frac{T * D}{2 * J_{tube}} = \frac{16 * T * D}{\pi * (D^4 - d_i^4)} \tag{5}$$

Where T is the torsional moment applied, D is exterior diameter and J is polar moment of inertia for the gross section.

4. FAILURE CRITERIONS

If a mechanical component is under uniaxial stresses, is easy to predict either break or yield failure but, generally, stress states are biaxial or triaxial. To predict failure, some criterions are needed. There are some theories that define static failure under multiaxial stresses, some more precise for yield, and others for breakage but all validated by experiments according to material.

In order to obtain a stress raiser factor, two stresses need to be compared, being them the maximum stress obtained by means of the analysis run in ANSYS and the nominal gross stress mentioned previously. In mechanical designing there are three major stress criterions to follow, although there are some others such as Coulomb – Mohr, which isn`t mentioned. When the analyses are done, it`s the designer/engineer that must decide which of the following three criterions is the most adequate. When reached the Result Processing step, the results of all three criterions will be compared.

4.1. RANKINE CRITERION

It is also called as the maximum normal stress criterion and it`s usually used for fragile materials. It`s only used as a break criterion due to it links well to fragile materials. Rankine established that the studied material will resist any load whenever the maximum normal stress at that point never overpasses the admissible stress obtained by a traction essay and vice versa for compression loads. The equation that represents this criterion is:

$$\sigma_{Rankine} = MAX\ (|\sigma_1|, |\sigma_2|, |\sigma_3|) \tag{8}$$

Being $\sigma_1, \sigma_2, \sigma_3$ the principle stress in those directions. According to his criterion, failure occurs when $\sigma_{rankine} \geq S_u$, being Su the ultimate stress. Here is considered always plane stress so that σ_3 is null. This supposition can be made because components are going to break on the surface and not in the bulk. There isn`t failure for any σ_1, σ_2 combination inside the square from illustration 6. For three dimensional cases, failure surface transforms into a cube but follows the same philosophy.

This is a failure criterion but in this project it`s not of interest the study of failure or non-failure in the component. The purpose of using this criterion is to use the equivalent stress obtained by it.

According to ANSYS, Rankine`s criterion is called as 1st Principle Stress.

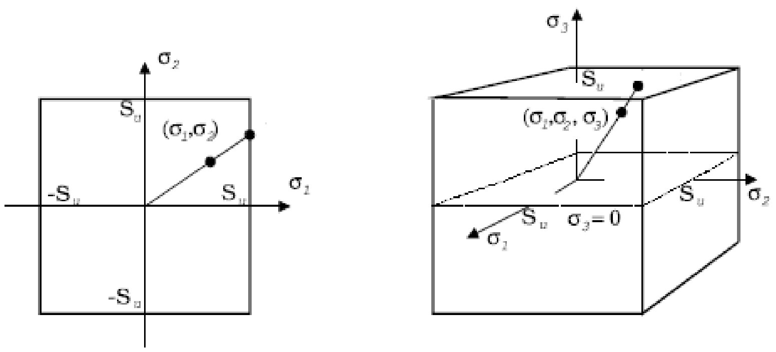


Illustration 4. Rankine`s criterion

4.2. TRESCA CRITERION

Also named as maximum shear stress. It foresees yield deformations and isn`t adequate for pure ductile materials. This theory predicts failure when the maximum shear stress in a multiaxial state reaches the value corresponding to failure in an experimental essay. The maximum shear stresses can be obtained from the normal stresses.

$$\tau_1 = \frac{|\sigma_2 - \sigma_3|}{2} \qquad \tau_2 = \frac{|\sigma_1 - \sigma_3|}{2} \qquad \tau_3 = \frac{|\sigma_1 - \sigma_2|}{2} \tag{9}$$

After operating with some equations obtained by traction essays, the final Tresca criterion is set as:

$$\sigma_{Tresca} = MAX\ (|\sigma_1 - \sigma_2|, |\sigma_2 - \sigma_3|, |\sigma_3 - \sigma_1|) \tag{10}$$

According to his criterion, failure occurs when $\sigma_{Tresca} \geq S_y$, being Sy the yield stress.

For plane stress ($\sigma_3 = 0$), maximum shear stress criterion can plotted in a diagram like in illustration 7. In the interior points of the hexagon, there isn`t yield, according to this criterion. For triaxial stress states, failure is predicted by a hexagonal based cylinder, which axis is σ_1 = σ_2 = σ_3.

This is also a failure criterion but in this project it`s not of interest the study of failure or non-failure in the component. The purpose of using this criterion is to use the equivalent stress obtained by it.

This stress, in ANSYS, is identified as Stress Intensity.

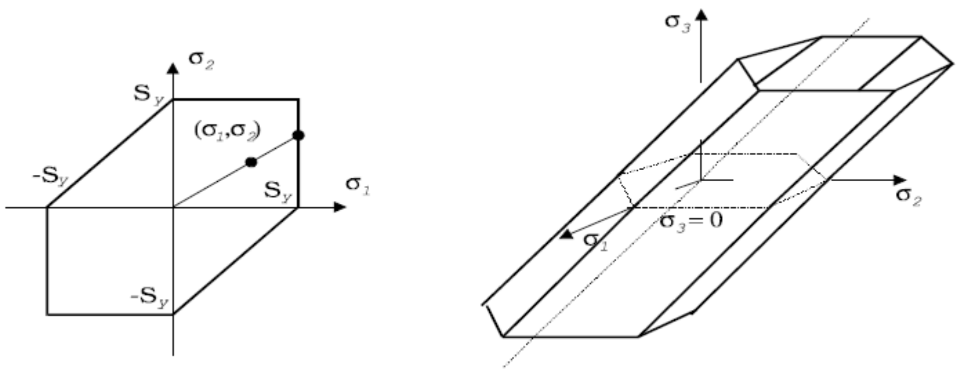


Illustration 5. Tresca`s criterion

4.3. VON MISSES CRITERION

This criterion was born fruit of studying the previous one, Tresca. Von Misses worked on Tresca`s theory and saw that it didn`t predict exactly failure by yield. In any point of an elastic body there`s strain energy and it can be divided into two different components:

- Volumetric strain energy: energy associated to volume changes without hanging form or shape of point.
- Distortion energy: with a constant volume, the energy capable of changing the shape. This is the inverse of volumetric strain energy. Equation 8.

Von Misses theory is based on studying distortion energy, receiving a parallel name for the theory called, distortion energy. Equation 9.

There`s yield when distortion energy reaches a value of distortion energy form a traction essay. The equation for Von Misses stress theory is based on the distortion energy principle and can be obtained from it with some equation operations.

$$\text{Strain energy} = \frac{1}{2} * (\sigma_1 * \varepsilon_1 + \sigma_2 * \varepsilon_2 + \sigma_3 * \varepsilon_3) \quad (11)$$

$$\sigma_{Von\ Misses} = \frac{1}{\sqrt{2}} * \sqrt{(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2} \quad (12)$$

According to his criterion, failure occurs when $\sigma_{VM} \geq S_y$, being Sy the yield stress.

The difference between both Tresca and Von Misses around the 10% mark, being Von Misses more conservative. When plotted, this criterion adopts an ellipse shape, coinciding some points with Tresca`s theory.

As said before, this is a failure criterion. It`s not of interest the study of failure or non-failure in the component. The purpose of using this criterion is to use the equivalent stress obtained by it.

For a three dimensional stress state, failure surface is a cylinder, with a circular base where its axis is $\sigma_1 = \sigma_2 = \sigma_3$.

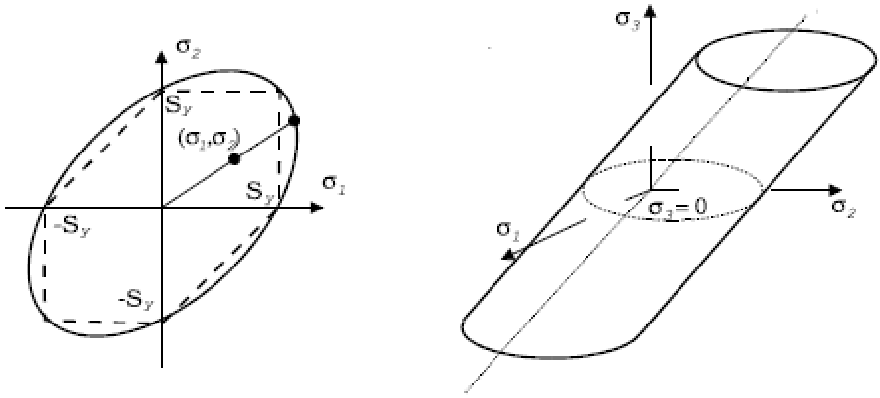


Illustration 6. Von Misses criterion

5. COMPONENT MODELLING PROCEDURE

From this point on, is the modelling part of the project where all technical features are taken for granted and the component for this project must be modelled, created and analysed in the most efficient way as possible.

Now, in order to model this component in ANSYS, here is displayed a method that has in mind every aspect of the pipe. The modelling process has been done according to the following stages:

1- Previous considerations.
2- Element type and material definition.
3- Geometrical model insertion.
4- Mesh generation.
5- Boundary conditions and loads.
6- Solving problem

5.1. PREVIOUS CONSIDERATIONS

As in any project, before getting started with calculus, it`s highly recommended to take some time and analyse the objective of the project. It`s better to spend some time thinking on how to do a task instead of directly diving in and committing many mistakes that in long term will translate into time loss and therefore efficiency. Before starting ANSYS and introducing parameters into the program, let us take a look at the component studied in the present project.

At one end of the pipe, the displacement restrictions will be set and on the other extreme of the pipe, the loads must be applied (axial, bending and torsional stresses). Looking at the aspect of the component, some variables are needed to define diameters and lengths belonging to the pipe. In this project, D has been established as the exterior diameter of the pipe, di is set as the interior diameter of the pipe (which creates the hollow cavity), L is the length of the pipe and finally, in order to incorporate the stress raiser, d is established as the diameter of the cylindrical tube running transversally down through the pipe.

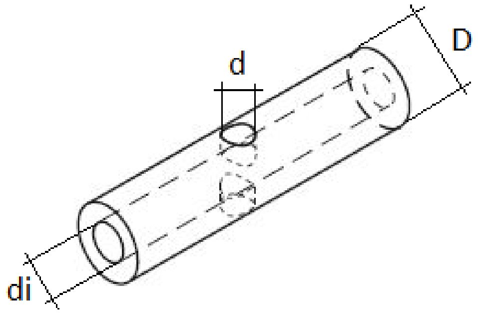


Illustration 7. Component`s parameters

For further processing inside ANSYS and macro setting, some variables are needed to relate all these parameters. This information can be found in Peterson`s Stress Concentration Factors 3rd edition book or elsewise it can be mathematically deduced. In order to replicate the curves from Peterson`s Stress Concentrating book and verify them, the same variables are required to recreate the curves. The three parameters used in Peterson`s Stress Concentrator book and also used in this project are: the ratio between stress raiser`s diameter and exterior pipe diameter (d/D), the ratio between interior and exterior diameters from pipe (di/D) and finally the exterior diameter of pipe (D). When resolving the component in ANSYS these variables will have a set value, corresponding to one point of the Kt curve. In order to achieve more points from that curve it´s necessary to change the values of those variables (always in a logical ranges of possible values).

A key factor that has to be taken seriously is that when modelling the component, the stress concentrator must be far enough away from the fixed end due to that when analysing in ANSYS, the program has difficulties to detect differences between stress near stress raiser and the embedment, so in order to reach reasonable results the transverse hole needs to be far enough from displacement restrictions at constrains. The question now is; how much distance is "far enough" away from fixed end? The question is answered in point 5 where the geometrical model is created.

For this project, a finite element method program is needed for all numerical analyses. Both student and tutor agreed on using the academic version of ANSYS, which is freely available for students and personnel of the university. The main inconvenient of using this version of software is that the amount of knots is limited to a total of 32000. It may not seem a major problem but it has probably been the biggest obstacle in the realization of the project, needing for the designer to rearrange ideas, changing strategy until a satisfactory result was reached. Further on into this report is mentioned more about this important aspect.

Last but not least, is the issue about any possible limitations inside the geometry. The only limitation concerning geometry, taken account for in this present project, is that stress raiser`s diameter must always be smaller than interior pipe diameter, due to structural matters. If one looks perpendicularly to cross section of pipe, then this issue can be seen better.

Geometrical limitation:

$$d < d_i \tag{13}$$

This condition is set in one way to restrict the amount of possible cases and therefore analysis possible. If the interest was more specific in this solid, then a different plan could be thought about but that would take more effort, time and resources and this is out of range for this project.

5.2. ELEMENT TYPE AND MATERIAL DEFINITION

Once clarified the previous considerations, one can start launching ANSYS, through the application of ***ANSYS Mechanical APDL Product Launcher***. It is to mention that any student or personnel from the Polytechnic University of Valencia (UPV) can get access to a free version of ANSYS during a limited time period, if they belong to the university, of course. Also, the amount of knots that can be introduced into this ANSYS version are limited, but for doing this project and any similar analysis this version is more than enough.

Once clicked twice on the product launcher, a black window will appear. First of all, there are many modules inside ANSYS but as students using a free version of this software then the ***Academic Teaching Introductory*** version is sufficient for these calculus. Then, a work directory is required, being this the path to where the files are going to be saved. It`s highly recommended not to save files at the desktop because once solved the component there are several files as output, so having them isolated and located is an advantage. Also, if user is working in a different language than English where there are accent marks or letters such as ñ then ANSYS doesn`t work properly. It may save incorrectly or either shut down automatically when solving, so for the directory path, keep it simple and easy.

Finally, a name for the file is requested, in this case ***Solve Tube***. Once all this information is completed, the user is in conditions of pressing ***Run*** to initialize the program. The main menu window should be similar to illustration 8.

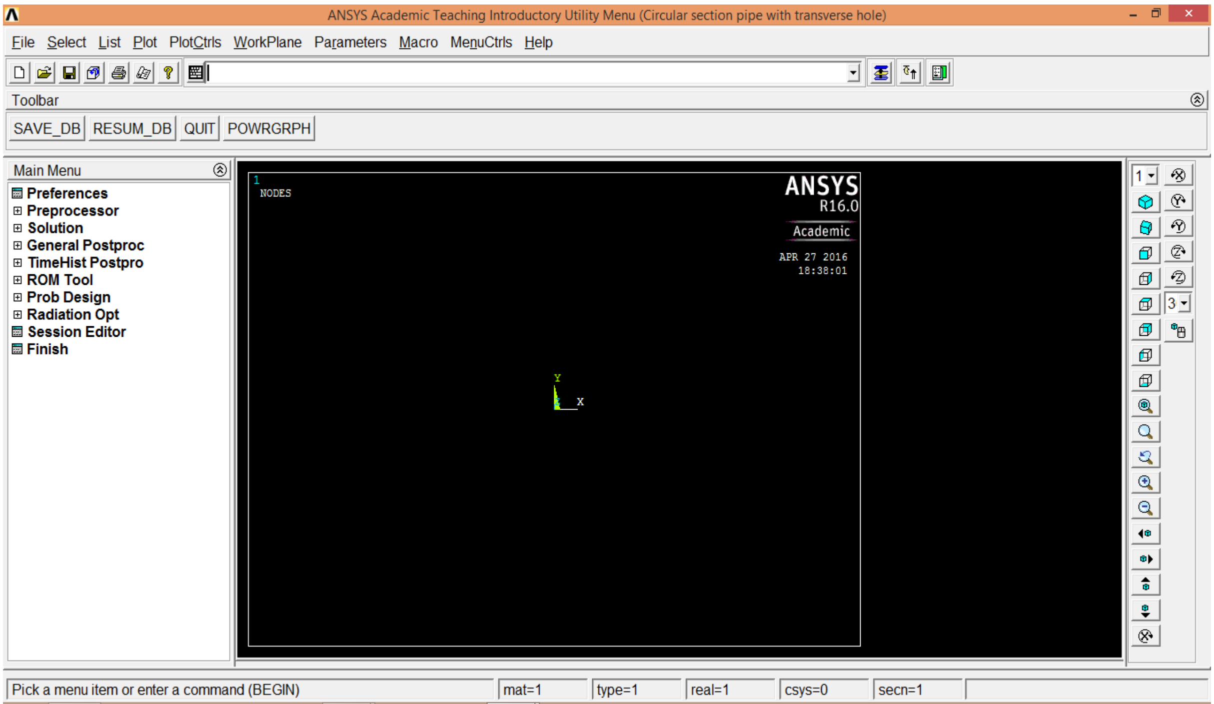


Illustration 8. Main menu screen

This is the main window when staring ANSYS and there are many windows in the top part of this utility window but here they are not explained because it isn´t the objective of this project.

At the left hand side of the window there is a main menu. This is going to be the sequence followed while operating with ANSYS. Even if the user doesn´t know how to manipulate the program the main menu sets a path to follow. Not all stages will require information so those can be left in blank or ignored when one knows what is asked. Following the steps established by the main menu, the first thing required is ***Preferences***.

The user needs to tick a box corresponding to the type of analysis desired, being this case ***Structural***. Then by default the box of h method is marked and that will be left untouchable. H-method is referenced to the type of technique for resolution after meshing the geometry. According to this method, the degree of the polynomials will maintain fixed while the number of element (nodes) will increase.

Once pressed ***OK***, the program redirects back to utility menu, where the next stage is waiting to be fulfilled. This is the ***Preprocessor***. The pre-processor in any type of finite element programs is a vital stage. Basically, success or failure is based on what and how is completed in this section, because afterwards the following step is Solution and these calculus programs can solve anything possible even if the results are extremely wrong. There is no capacity of ANSYS to correct the ***Preprocessor*** so whatever is set in this stage is what is calculated. The majority of time spent in these type of project is entering the desired values in the way the program understands it because once modelled the component correctly, hundreds of solutions can be carried out per day, but they necessarily need to be correctly modelled.

When opened the tab of ***Preprocessor*** the first information needed to be filled is ***Element Type***. Inside here a new element type needs to be created. Press ***Add/Edit/Delete***. Due to it is the first time ANSYS is started, there are no configurations saved by default so press add a new element type.

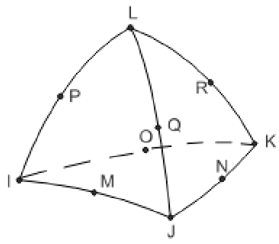


Illustration 9. Tet 10 nodes element

Here is where the user establishes which type of element is wanted to mesh geometry. Depending on the component analysed, the element type chosen will vary in order to recreate in a better way the meshed geometry. After searching all different types of elements for meshing, the best one for the pipe in this project is ***Solid Tet 10 node***. This is a solid tetrahedral element with 10 nodes, which means more calculus points to solve, resulting in a more accurate result. For this element, 4 integration points are needed.

If chosen the same element type as described, the resulting window should show the following shape. Press ***OK*** to exit this window.

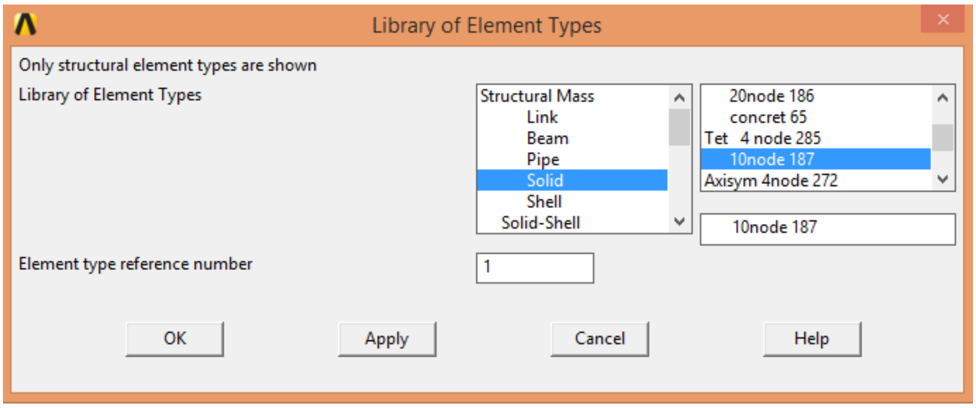


Illustration 10. Selecting element type

Also we are going to need to incorporate another type of element, so we must reopen the previous window and select ***Structural Mass*** and ***3D Mass 21***. This element will be used when meshing the component because this mass element includes rotations and inertias in its freedom degrees. The reason of this step will be explained in detail when meshing and applying loads.

Following this sequence, the real constants for the ***Structural Mass*** must be established. This is done by clicking into ***Real Constants*** and ***Add/Edit/Delete*** and then ***Add.*** All the real constants will be left in blank excepting the set number which will stay as 1.

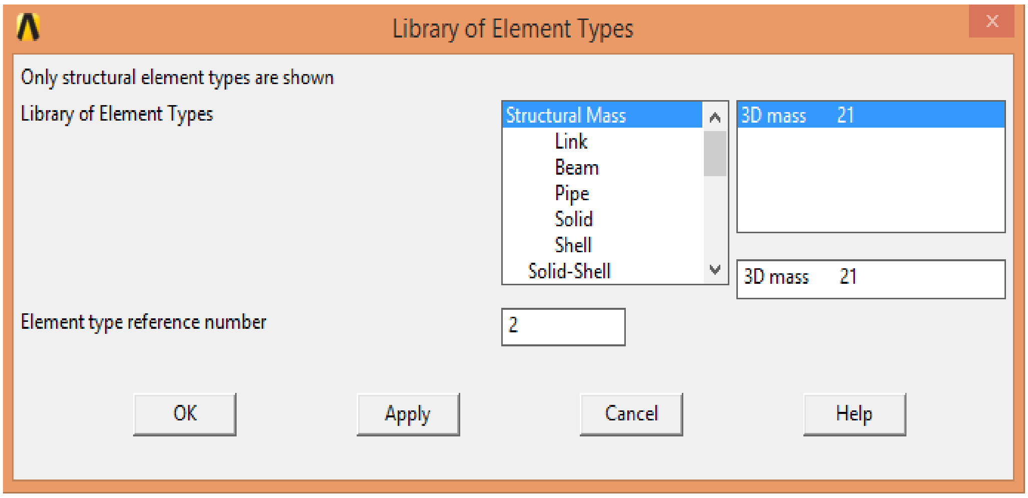


Illustration 11. Selecting mass element

This concludes the part of defining the element used for meshing and now the material of the component must be set. No real constants are needed in this project so the next stop is ***Material Props***. When opened this tab, ***Material Models*** is where to go. As mentioned in the introduction of this project, material doesn`t have an influence on the results so any could be valid. In this case, a steel material has been selected due to that the constants to define a steel are commonly known. To define a steel, Inside ***Material Model Available*** press, ***Structural, Linear, Elastic*** and ***Isotropic***. When a new window flashes out it asks for the variables that define a steel, being these only two, Young`s modulus and Poisson`s ratio. For steels, Young's modulus (EX) is $2.1 \cdot 10^{11}$ Pascal and Poisson's ratio (PRXY) is 0.3. Introduced these values, the window should have the following appearance.

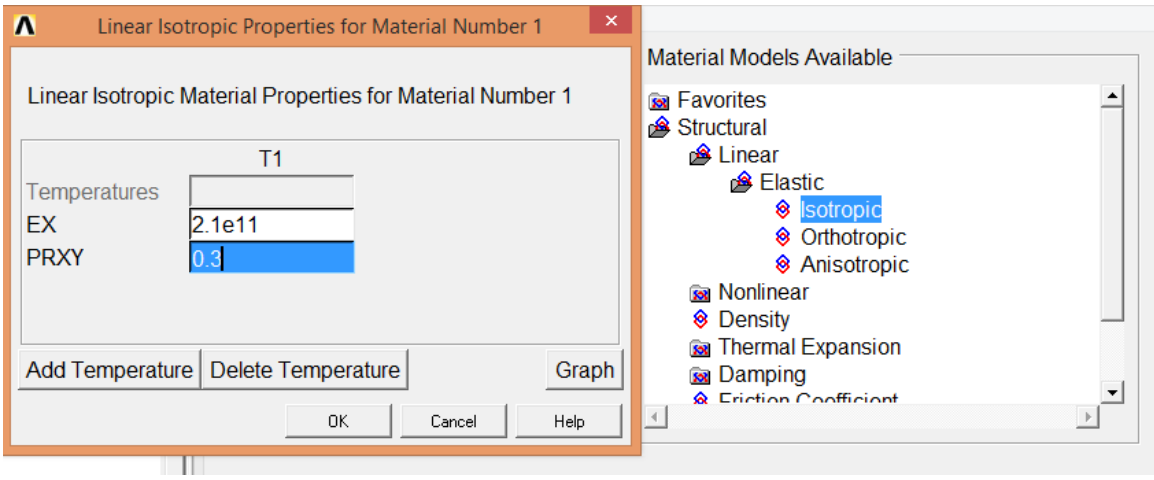


Illustration 12. Material properties

5.3. GEOMETRICAL MODEL INSERTION

Now the question is how to recreate the geometry in ANSYS. First of all, it must be said that the recreation of the geometry has been done following Peterson`s Stress Concentration Factors 3rd edition and the image can be seen in illustration 1. It also need to be said that there are many ways of modelling this solid, being so that any other designer would create the geometry in a different way and technically, they all would be valid.

When one creates a model in any finite element method program, it is done so that it helps the designer in post chores such as meshing and applying loads. There is no point in creating a sophisticated model when meshing the solid is going to be extremely hard. The model needs to contain the basic and essential information detailed in planes while being as simple as possible. In this case, the essential information given by Peterson is basic, opening up a handful of possibilities when creating the geometry.

Here is described one way of modelling the geometry but as said, it isn`t the only. While doing this process in ANSYS, three different models were necessary in order to finally select a correct geometry where no problems were involved. But for one to prevent problems ahead, sometimes these problems aren`t held in mind, so it could be called as a trial and error process until eventually hitting with the correct solution.

There are many ways of modelling the geometry and here is exposed one, simple method for this task. The three main variables must be taken account for, being these d/D, di/D and D, meaning that the length of the component must be function of these variables. In this project, L is directly proportional to D.

The strategy adopted for the creation of the geometry is fractioned into three. This is because in this project, there are three different cases identified, which need to be modelled in different ways in order to obtain easy but efficient results.

Only having 32000 nodes, which sounds a lot but really isn`t, the main task is to economize the available nodes. They must be used in the most efficient way possible so that precision is where higher stresses are foreseen. The precision is directly related to the amount of nodes at one certain location so as it seems logical the majority of the nodes need to be set at the stress raiser. In order to confine these nodes around the stress raiser, the main plan followed in this project (which gathers 70% of all possible combinations) is called for case 1.

Now in case 1, the basic hollow pipe is created easily and, following that, comes the creation of the stress raiser, which is also fairly easy to do. These two steps would comprehend the basic information of the solid but the geometry is taken a few steps further. If only a transverse hole is created, the mesh will be finer at the inner walls of the hole but the user doesn`t know if the maximum stress is going to occur in the exterior side or a slight distance inside the component. For capturing the most information near the stress raiser (not only on the outside wall) a method needs to be set so that the mesh is finer a small distance inside the pipe. This is solved by with a geometrical solution. An extra cylinder is created which has a bigger radius than the stress raiser so it surrounds the hole. This way, the stress raiser can have a mesh size and the exterior cylinder can have another, both independent of the rough size for the rest the tube.

Also, it is wanted for the majority of nodes to be confined at the stress raiser so a solution adopted for this statement is dividing he pipe in several sections. If one looks at the gross cross section, this is the volume wanted to be sectioned. ANSYS can revolute an area into many parts as wanted but all parts are equal in size. What is wanted here is to section the pipe into a certain amount of parts but not equal in size because the sections must divide pipe just around the stress raiser and all the rest of the pipe can be one big section where the elements will be bigger. On top of that, when the macro creation step arrives, the user will want to vary the inclination of the dividing line in order to optimize the process. To clarify this idea, look at illustrations 13 and 14. These could possibly be two cases where in illustration 13 the hole is small so the line will section the pipe into volumes with different sizes and in illustration 14, the hole is bigger so the line that divides all the pipe must rotate and divide into volumes with another size.

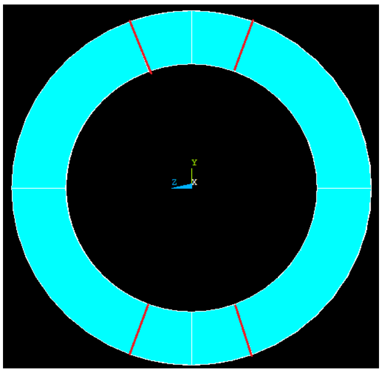


Illustration 13. Possible configuration

This type of geometry creation is called case 1 in this present project, with both the exterior cylinder and the pipe division. Further on is explained how to do this process.

This is the most complete model contained in this project because it takes account for many features and solve many future problems but if one was to create only this type of geometry and start calculating for results, some would legitimately work perfectly but in some D, d/D and di/D combinations, ANSYS isn`t capable of creating the geometry (the reason is explained in further points). ANSYS cannot model all cases with only case 1, so the chore now is to identify these special cases and set another case for them.

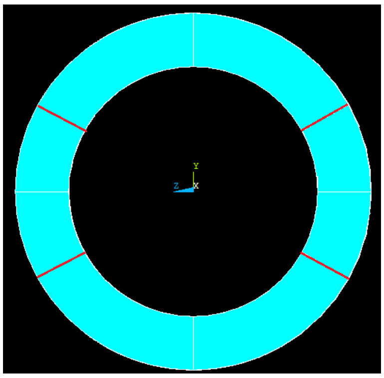


Illustration 14. Another possible configuration

The cases where case 1 is invalid is at the extremes, being these when stress raiser is very big or very small. In order to know where the line is, an arc sine is calculated and as known, the arc sine of a number over 1 is impossible, so here is the source of one of the problems. The user needs to detect when this is going to happen and change the case for creating the geometry. But also what is observed is that the exterior cylinder is always compatible with the division of volume, which give the geometry an extra plus.

So now, there are two differentiated cases where we can and cannot divide into smaller section the pipe. Inside the case where the division can`t be done, it`s observed that there will be big holes and small holes as stress raisers. It has been decided to eliminate the exterior cylinder when the hole is too big because it becomes unnecessary. When the hole is large enough it`s not worth it creating the exterior cylinder and it is more efficient to mesh with small elements all the middle section of the component.

So now all possible cases have been mentioned and set, what is left is to establish a limit between cases, responding to the question when to follow case 1, 2 or 3. For this chore some parameters further on used need to be mentioned. As for the volume dividing process, the line that does the division needs to be located at all time and this is done via angles.

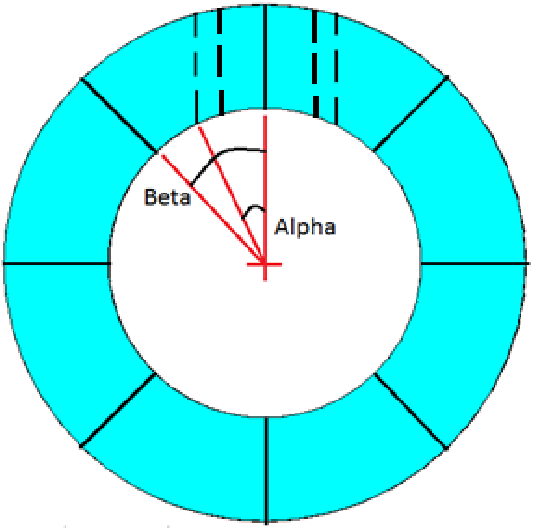


Illustration 15. Alpha and beta definition

When the user introduces the parameters that define the geometry of the component (D, d/D, di/D), two new auxiliary variables are created. These are alpha and beta which can be seen in figure 13. Alpha is the angle from the centre of the pipe to the external cylinder`s radius. Additionally, beta will be alpha plus an extra angle that sections the component where desired.

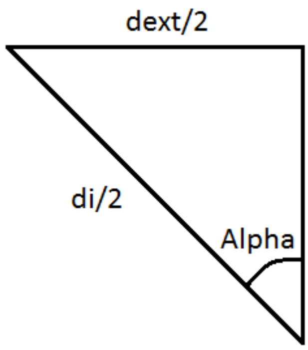


Illustration 16. Alpha calculus

All values are known except for alpha, which can be obtained using the definition of the sine of an angle. This value will change when the stress raiser hole changes, which was one of the main objectives. Now alpha is known, beta can be stipulated based on it, adding a certain amount of degrees which will divide the pipe into portions with the wanted size. After trying some analysis, it was determined, for this specific case that the perfect definition for beta was to be alpha plus 5 degrees. Less than that would compact the volumes too much and there would be problems when meshing and more than 5 degrees would expand the dividing areas. It`s not easy to find a balance between these variables because the stress raiser grows in one plane and the beta angle grows in another making the chore, in some cases, difficult.

The next question to be answered is: where are the limits of using this type of geometry creation? Well, after the experimental method of trial and error it was determined that for this project, when alpha was bigger than 60 degrees, that meant that beta was 65 degrees, some of the resulting volumes were too small in comparison with the others, making the dividing process no use. The limit is established at alpha being less than 60 degrees and being able to calculate the arc sine. Beyond that, something different must be done in geometry creation terms.

When running some geometry tests it was seen that, when dividing wasn`t possible, there were two clearly differentiated combinations, those where the hole was very small and those where the hole was very big. As mentioned previously, when the hole is too big, then it`s not worth creating the surrounding external cylinder but when the stress raiser is small then it is interesting to have the external cylinder to capture what is happening inside the hole`s walls. The limitation set for saying that there is a small hole which needs a surrounding cylinder is fixed at d/D being less or equal to 0.3. So after this deliberation two other cases have been created. After trying all possible geometry combinations, it was seen that every single one could be created via case 1, 2 or 3, having no exceptions. In illustration 15 is a logical process image of how the future macro should work, having to know which case to jump to.

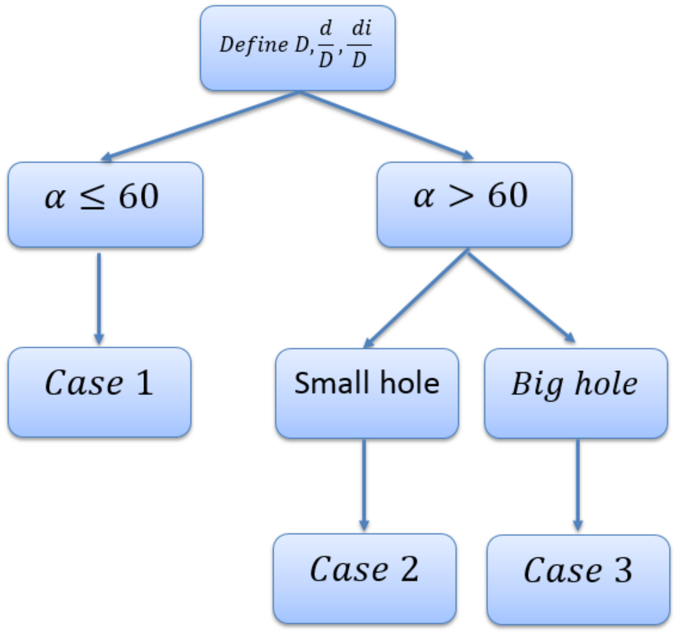


Illustration 17. Logical process for case loading

Where case 1 involves dividing pipe into 8 sections plus adding the surrounding cylinder. Case 2 doesn`t contemplate the dividing process but does create the surrounding cylinder and finally, case 3 doesn`t create neither the external cylinder nor the division of volumes. This is the case distinction, the reason of being of the three cases, the need of them but following up in points 5.3.1, 5.3.2 and 5.3.3 is explained how to do this task.

5.3.1. CASE 1

It must be held in mind that when creating the geometry, one is foreseeing it to an adequate meshing, so geometry is dependant and limited by meshing process. For setting the manner of geometry creation for case 1, the parameters used are D = 0.75, d/D = 0.2, di/D = 0.6. A sketch of the spatial colocation of keypoints can be seen in illustration 18.

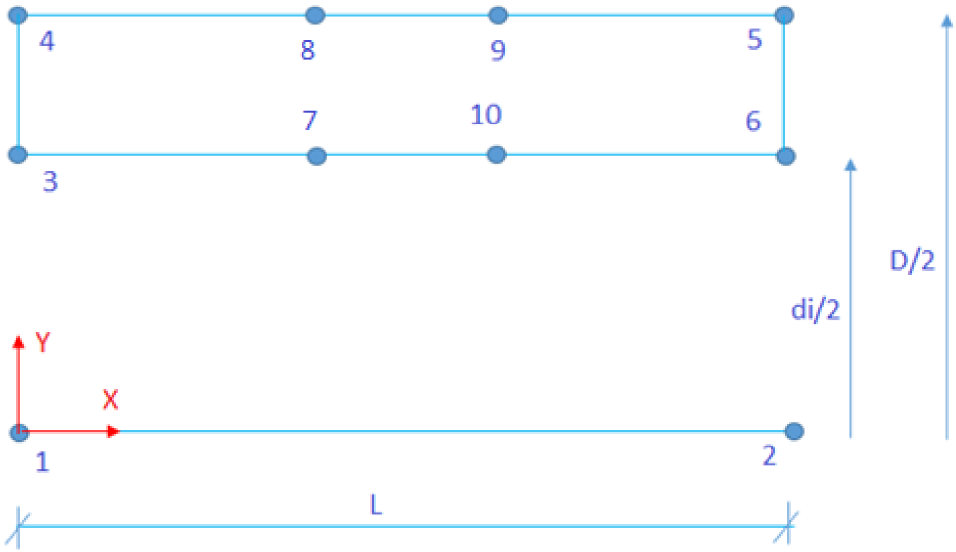


Illustration 18. Keypoint layout

Keypoint	Coordinate X	Coordinate Y
1	0	0
2	3.67*D	0
3	0	di/2
4	0	D/2
5	3.67*D	D/2
6	3.67*D	di/2
7	(3.67*D)/2 - (d/2) - 0.15	D/2
8	(3.67*D)/2 - (d/2) - 0.15	di/2
9	(3.67*D)/2 + (d/2) + 0.15	D/2
10	(3.67*D)/2 + (d/2) + 0.15	di/2

Illustration 19. Keypoint coordinates

As shown, only ten keypoints are needed and the coordinates, relative to world reference system (coloured in red) for each keypoint, are collected in illustration 19.

Keypoint 2 sets the length of the pipe and as shown, it`s value is 3.67 times outer diameter. As said before, the length must be function of D and it`s directly proportional.

For the tube`s length it has been adopted L = 3.67 * D so that stress concentrator is far enough away from embedment. This is done to avoid possible errors with the results. Multiplying D by 4 seemed a good, reasonable length for the pipe. But inside the macro, if the user wanted to identify that parameter and modify it, this might be a task to find a 4 because inside that macro there could be many 4's. So in order to make the parameter`s search an easier task a random number near to 4 has been assigned, in this case 3.67. For keypoints 7, 8, 9 and 10, their X coordinate is justified in the following way. Further on, a cylinder is going to be created to pass straight through the pipe to create the stress raiser. In order to view, later on, the component inside, some lines are put near the stress raiser, helping meshing task also. But where to put these lines? The solution adopted is: for 7 and 8, start at L/2 (3.67*D/2), then take away radius of stress raiser (d/2). Now comes the safety margin, which is set at 0.15. So this safety margin needs to be subtracted from the other values mentioned before, having the X coordinate. For keypoints 9 and 10, which are on the right hand side of stress raiser, we need to add safety margin along with d/2 in order to have their X coordinate.

Keypoint	Coordinate X	Coordinate Y
1	0	0
2	2.75250	0
3	0	0.2250
4	0	0.3750
5	2.75250	0.3750
6	2.75250	0.2250
7	1.15125	0.3750
8	1.15125	0.2250
9	1.60125	0.3750
10	1.60125	0.2250

Parameters	
D	0.75
d/D	0.2
di/D	0.6
L	2.7525

Illustration 20. Numerical values for keypoint coordinates

With the aid of a Microsoft Excel Worksheet these variables can be translated to numbers, assigning values to D, d/D and di/D. These numbers are the ones introduced into ANSYS manually. Coordinates must be at positions set in illustration 20.

Now, in order to introduce these number into ANSYS and to draw them on the workplane, one must open up ***Modelling***, ***Create, Keypoints.*** Choose option ***In Active CS***. Immediately a window appears asking for the number of keypoint and its location, coming in X, Y and Z coordinates. Mention that in this case Z direction isn`t used to define any point so must be left in blank. Introduce the values from illustration 21 and press ***Apply*** to carry on creating more keypoints.

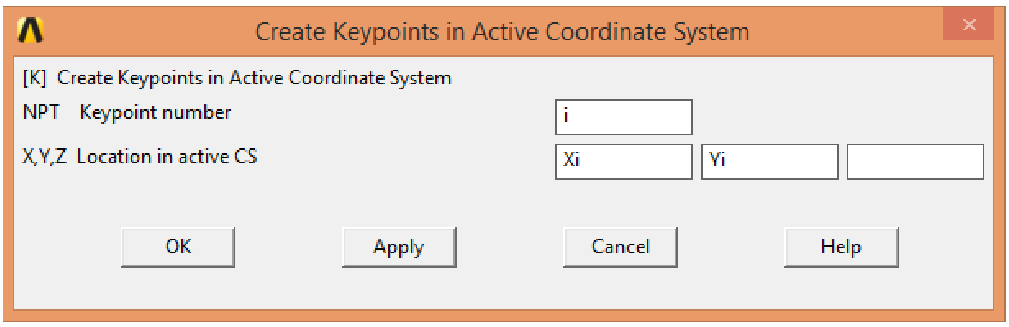


Illustration 21. How to create keypoints

Where i goes from 1 to 10, passing through all 10 keypoints needed.

Once introduced all keypoints, the user should have an image exactly like in illustration 22.

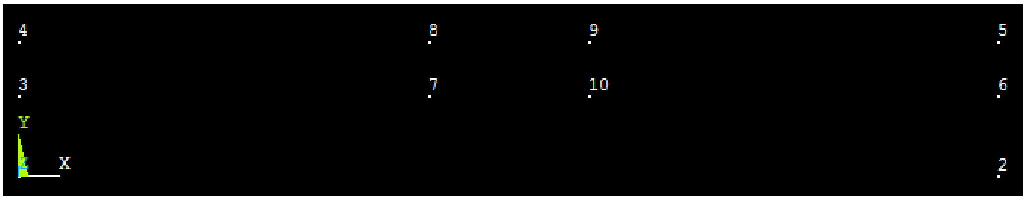


Illustration 22. Keypoints

These are the keypoints for the hollow pipe creation but now some other keypoints are going to be created for dividing the hollow pipe into 8 sections. In order to achieve the correct inclination (set by alpha) another coordinate system is created. This is done by going to the ***Workplane*** tab, ***Local Coordinate System***, ***Create Local CS, At Specified Loc.*** First of all, appears a window asking where the new coordinate system will be set. For commodity, it`s created at the origin (0,0,0) because only the rotation is wanted.

Press ***OK*** to continue and suddenly a new window flashes up asking for more details about the new CS. First of all is the reference number which will stay as 11. Type of coordinate system is Cartesian and the location is once again shown. Beneath is the rotation, which is the interesting part of this stage. How much to rotate? For that, alpha and beta need to be calculated. It must be said that for this first time these variables need to be calculated manually but for the rest of cases, this calculus will be automatized in the macro.

As it can be seen in the next equation, the diameter of the external cylinder is needed in order to calculate alpha, even before creating the stress raiser and its surrounding cylinder. An appropriate value for this external cylinder`s radius is the stress raiser`s radius plus an extra, which is set at 0.03.

In the next equations is shown how to calculate alpha and posteriorly beta. For the parameters that are used (D = 0.75, d/D = 0.2, di/D = 0.6), alpha and beta result to be:

$$Sine\ \alpha = \frac{\frac{d_{ext\ cyl}}{2}}{\frac{d_i}{2}} = \frac{\frac{d_{hole}}{2} + 0.03}{\frac{d_i}{2}} \rightarrow \alpha = 27.818º \tag{14}$$

$$\beta = \alpha + 5 = 32.818º \tag{15}$$

Now that alpha and beta are known the rotation can be established. Z and Y rotations are going to be null. For this first coordinate system, enter the following value: -(90 - beta) which corresponds in this case to -57.182º.

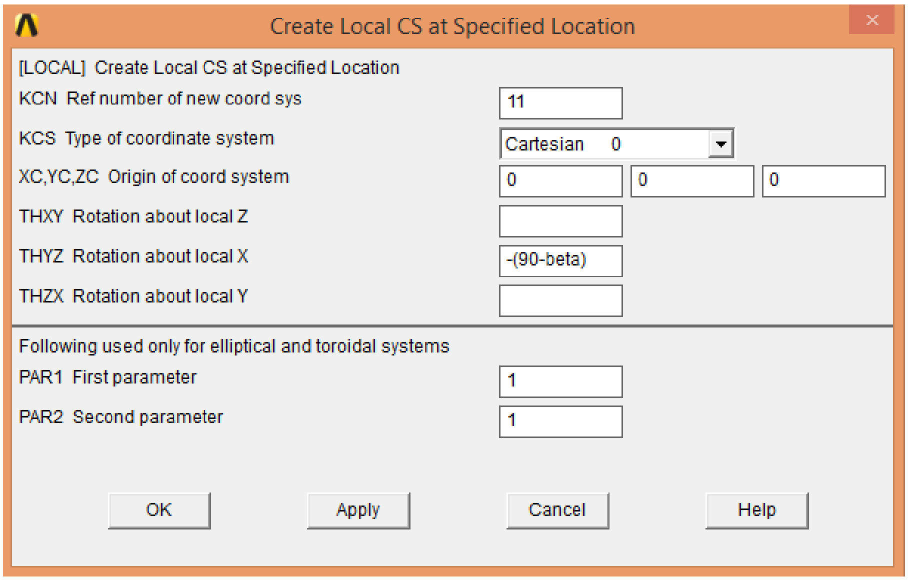


Illustration 23. Creating secondary coordinate system

Once this is coordinate system is created, the user needs to tell ANSYS to change to coordinate system 11 and use it for further operations. This is done by going back to ***WorkPlane, Align WP with, Specified Coordinate System.*** Another window flashes up asking which C.S to align with, in this, asking for the name. Previously, the new C.S was named 11, so enter 11 in the box and press ***OK***.

At this point, four new keypoints are created, using the exact same method as before but changing the coordinate values (don`t forget that coordinate system has now changed).

Keypoint	X coordinate	Y coordinate	Z coordinate
11	3.67*D	0	di/2
12	3.67*D	0	D/2
13	3.67*D	0	-di/2
14	3.67*D	0	- D/2

Repeat this sequence once again to recreate 4 other keypoints. Another coordinate system needs to be created, aligned and then create another four keypoints. The process is the same as mentioned but changing values. Create another coordinate system with a reference number set at 12 and with the same location as CS 11 (0,0,0). For the rotation, punch in the X rotation box 90 – beta (which in this case is 57.182). Finally create keypoints number 15, 16, 17, 18 with the same X, Y and Z coordinates as the table above. Notice that all vital keypoints are created before lines and areas and this is due to a numbering problem. When areas and volumes are created, more keypoints associated are too, so in order to keep track of some keypoints, they are created first, so there won`t be any problems.

This is the procedure to insert as many keypoints as wanted but now they need connecting. They are going to be connected by lines. Each line starts and ends at a different keypoint, so the lines are created following indications in table below.

Line	Keypoints
1	11 and 12
2	17 and 18
3	13 and 14
4	15 and 16
5	1 and 2
6	3 and 4
7	4 and 8
8	8 and 7
9	3 and 7
10	7 and 10
11	9 and 10
12	8 and 9
13	9 and 5
14	10 and 6
15	5 and 6

The manner to create lines in ANSYS is by entering the ***Preprocessor***, ***Modelling, Create Lines, Lines*** another time and from all the possible ways here is explained the ***In Active Coord.*** When

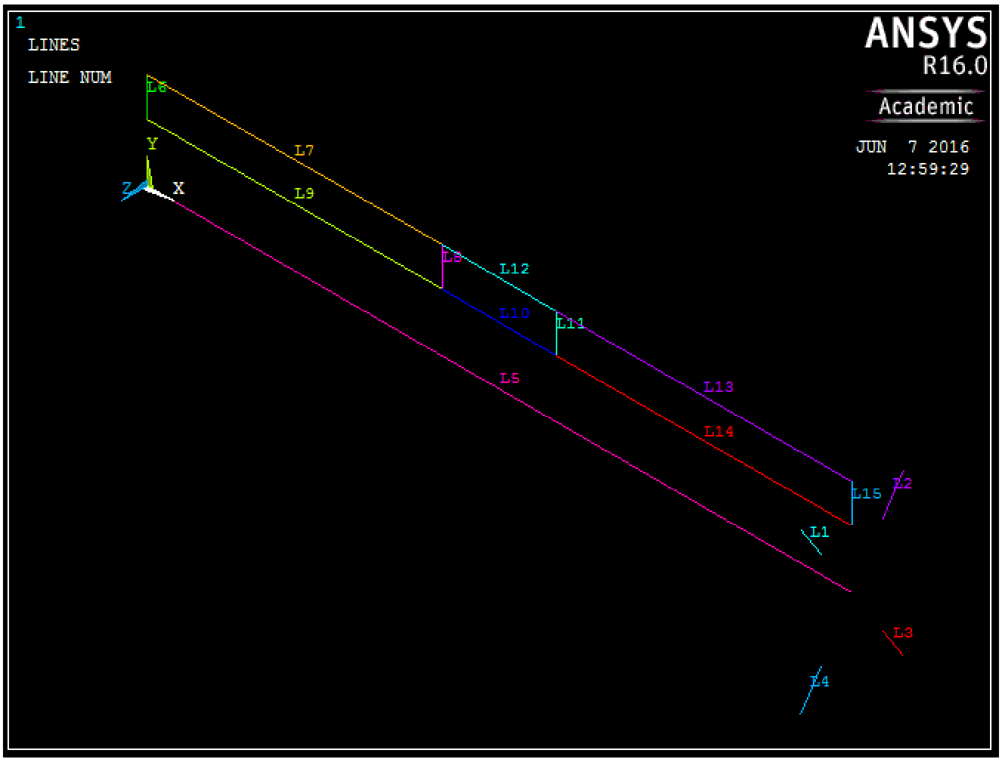


Illustration 24. Line creation

choosing this option an auxiliary window comes up. By default, the option ***Pick*** is set. Now, to create a line, enter the number of one keypoint, press intro and then enter the other keypoint belonging to that line and the line will be created automatically. Having above the table with all lines and keypoints, this chore is easy. Once finished creating all lines, press ***OK*** and the result necessarily needs to be just as shown in illustration 24.

Now that the lines are created, the next step is to fill in the appropriate spaces to have areas. Three areas are needed in order to recreate the hollow pipe and they are numbered as:

Area 1: compromises lines 6, 7, 8 and 9.
Area 2: boundaries are lines 8, 10, 11 and 12.
Area 3: surrounding lines are 11, 13, 14 and 15.

The only task left is to introduce these areas in ANSYS. This can be done in the following way, inside ***Preprocessor, Modelling, Create, Areas.*** In the many possibilities offered here is mentioned ***Arbitrary.*** To make area creation easier, instead of creating by keypoints, choose ***By Lines*** and then tick ***Loop*** box so that ANSYS recognises enclosed areas by lines. After creating all areas and pressing ***OK,*** user should have the following scenario.

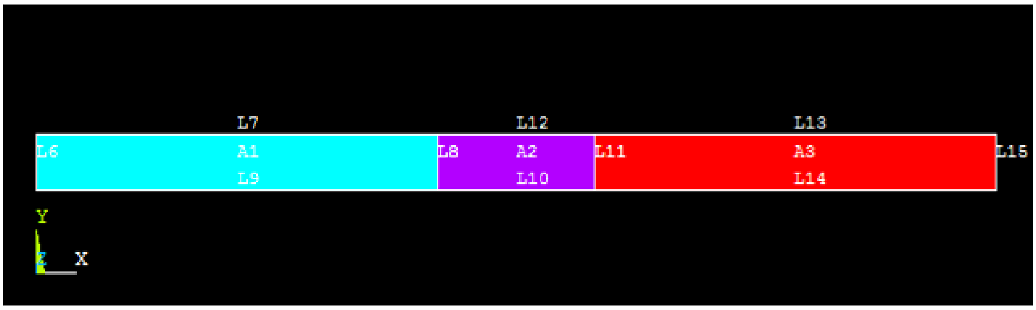


Illustration 25. Hollow pipe area creation

Now the areas that are going to divide the hollow pipe into 8 sections are created. Unlike before, the manner to do this is to ***Operate, Extrude, Lines, Along Line.*** A familiar picking window will appear. Choose line 1, press ***OK*** and then click on line 5. This will drag line 1 all along the length of line 5, which is the component`s length. Repeat this same process for lines 2, 3 and 4, always dragging along line 5. When this is done, the result should take the form of illustration 26 (the first three areas have been obviated to show the result of this last process).

The next step in the process is to create volumes (3-D) with the areas recently created. The strategy taken in this project to create the components` geometry is to pick an area and revolute around an axis to create a cylinder.

The hollow pipe is created by selecting areas 1, 2, and 3 and the axis for revolution is line 1. This process in words is translated into ANSYS language following these indications: first enter ***Preprocessor, Modelling, Operate*** and ***Extrude, Areas, About Axis.*** The user must select first the areas, press ***OK*** and then select, through two keypoints, the axis wanted. As a guidance, ANSYS shows on screen what parameters are needed when doing any operation. This help is shown at the left, bottom corner. Select areas 1, 2 and 3, press ***OK*** and then select keypoints 1 and 2 and press ***OK*** again.

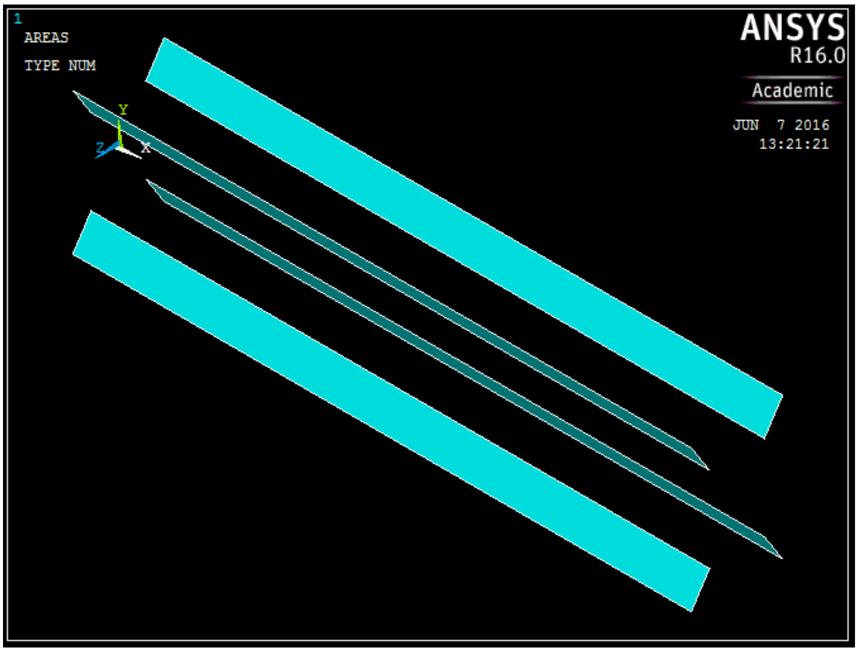


Illustration 26. Dividing area creation

Another important aspect to mention is that once chosen the areas and the axis, ANSYS askes user in how many parts it should revolute the piece, this is, ANSYS cannot revolute areas around an axis 360 degrees. The maximum angle allowed is 180 degrees and this is translated in ANSYS in stages. If user establishes 2 stages, then the piece will be sectioned in half, 3 stages sections in 3 parts of 120 degrees... Well, in this project, for post visualization, the hollow pipe is sectioned in 4 parts of 90 degrees each, making the meshing task much easier.

The next step is dividing volume that has just been created by the areas which will section it into 8 pieces. This is done going to ***Preprocessor, Modelling, Operate, Booleans, Divide.*** Now here choose the ***Volume by Area*** choice. First of all, ANSYS asks for the volume which is to be divided. Press ***Pick All*** and ***OK***. Then the areas dividing the volume are required. Enter numbers 4, 5, 6 and 7 and press ***OK***. Remember that these areas are the ones created before. The component should adopt the following shape:

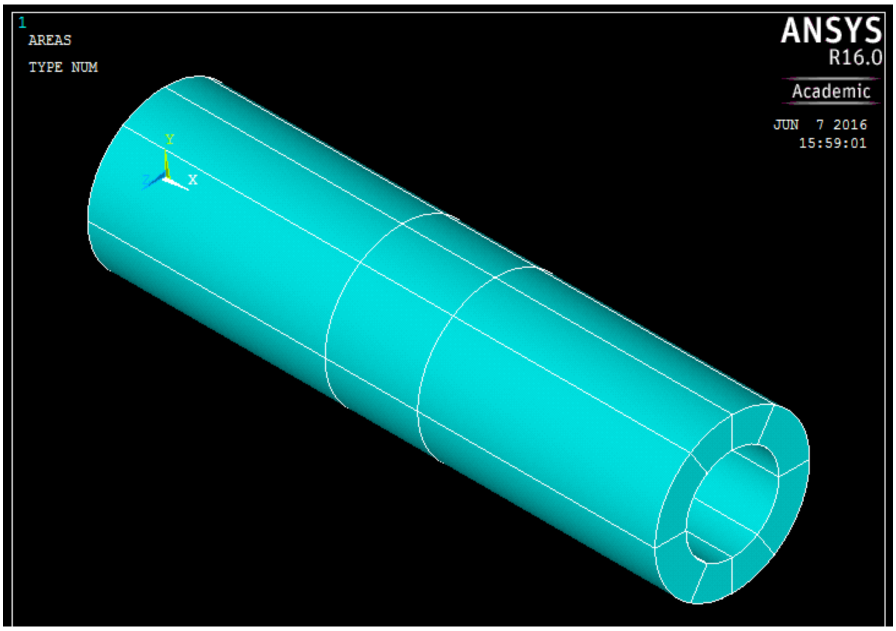


Illustration 27. Hollow pipe and dividing processes

Once this operation is done, the user will have a hollow pipe needed to be intersected, perpendicularly, by a solid cylinder down through the middle. The most sensible thing to do next is to create a cylinder an intersect the volumes, coming out with the final geometry wanted. The cylinder mentioned necessarily has to be created on the XZ plane with Y direction giving it depth or, in this vision, height.

From all possible ways of creating the cylinder, here is presented one easy way. First of all, one must orientate the cylinder in the proper way and this is done by creating another coordinate system, with direction and location wanted. This coordinate system is created at: X coordinate will adopt a value of L/2 and Y coordinate will adopt a value of D/2 + (D/2 –di/2). The reason of Y coordinate having that value is simple, it must be at least as much as D/2 to overpass the hollow pipe created, and then, as done before, there must be a safety margin, that tips over the pipe. In this case, the extra height isn`t essentially important because at the end of the process, both volumes will be subtracted. But to make it visually easier, the safety margin added here is D/2 + di/2, but it could have been any other number above that. Then, the radius of cylinder will be d/2 and the depth or height can be any value over D. The cylinder must come out from both top and bottom section to creating the stress raiser, so anything over D would be valid.

The way in ANSYS to achieve this goal is following the next steps. First, to create the new coordinate system in place, just like before. Go to top part of the ANSYS window and press ***WorkPlane, Local Coordinate Systems, Create C.S, At specified Location.*** The location coordinates for the new coordinate system will be:

$$X = L/2 \qquad Y = D/2 + (D/2 - di/2) \qquad Z = 0$$

In this case, these parameters will be 1.37625 for X and Y as 0.525.

Press ***OK*** to confirm this location. Now the next step is to set the orientation of this new coordinate system, which is set by filling in the next window that shows up. First thing that is asked for is for the name, which will be 13, following the existing numbering. Then, which type of coordinate system it is, remaining as Cartesian. The coordinates are then reviewed and next comes the orientation. This cylinder`s length must go in Y direction but it can go +Y or –Y direction, it doesn`t matter. For this project, the cylinder`s length is according to –Y direction and this means rotating the X direction of the old reference system 90 degrees. The window should be filled in just as in illustration 28.

Press ***OK*** to finalize this creation and check that the new coordinate system has been created and is seen. Now, there are two coordinate systems on screen, but ANSYS can only use one at a time for creating areas and volumes. So, before creating the cylinder, one must tell ANSYS to work with the new C.S. This is done by going back to ***WorkPlane, Align WP with, Specified Coordinate System.*** Another window flashes up asking which C.S to align with, in this, asking for the name. Previously, the new C.S was named 13, so enter 13 in the box and press ***OK***. Now is when the cylinder can be created. If one wished to check if the new coordinate system is engaged, go to ***Workplane, Display Workplane***. This action shows on screen the current coordinate system working.

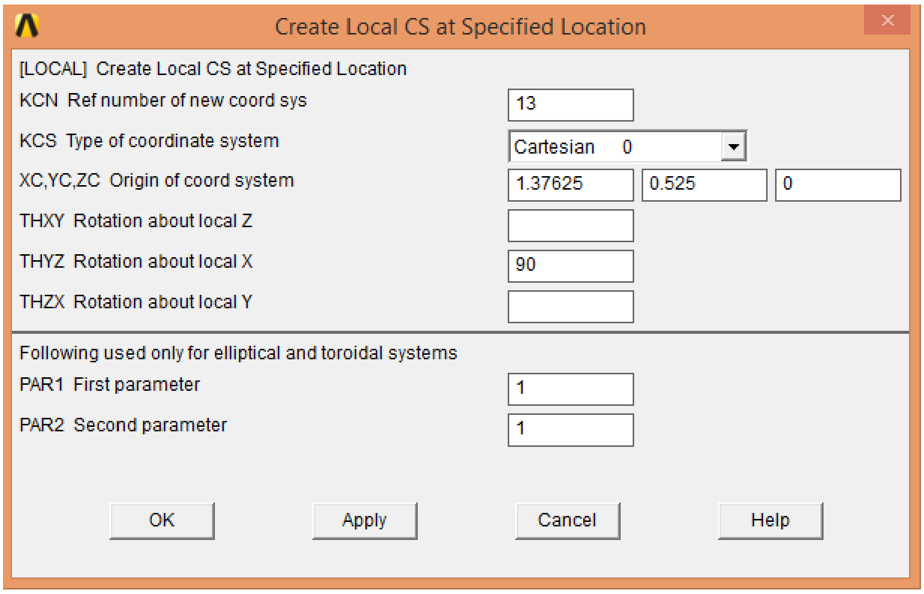


Illustration 28. Creating coordinate system 13

The procedure to create the cylinder is very simple. Click on ***Preprocessor***, ***Modelling, Create, Volumes, Cylinder, Solid Cylinder.*** To create the cylinder ANSYS asks the user via a window, the location (X and Y) of the centre of the cross section, radius of the cross section and the depth or height. Because coordinate system number 13 is in use, the location of the centre must be at this exact point, so enter two zeros in X and Y location. The next value asked for is radius, which will be d/2 (in this case 0.075). Finally, the depth is required being, as said before, any number over D (which in this case is 0.75). In this present project, the extra distance is stated at a value of D + D/2 (being in this case 1.125). Enter these number as shown in figure 29.

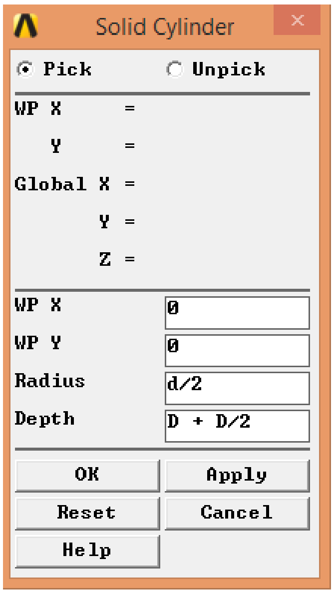


Illustration 29. Solid cylinder coordinates

Once this volume is created, the next stage in geometrical creation is the subtraction of volumes. The process needed is take away solid cylinder`s volume from hollow pipe. This, in ANSYS, is done inside ***Preprocessor, Modelling, Operate,*** and ***Booleans.*** From all the possible Boolean operations allowed, press ***Subtract, Volumes.*** Following the left, bottom corner instructions, ANSYS asks for the "volumes from which to subtract", being these volumes concerning the hollow pipe. This chore can be done with the picking application incorporated in ANSYS. When chosen all volumes, press ***OK*** and then ANSYS will ask for subtracting volumes, which are volumes related to solid cylinder. Press ***OK***, wait a few seconds and the final result should be as like in illustration 30.

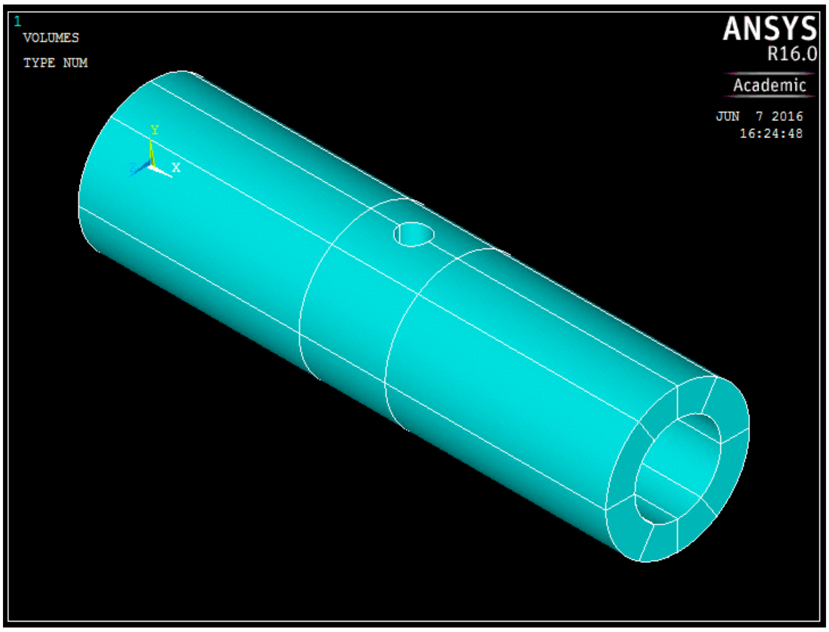


Illustration 30. Creating stress raiser

Reaching the end of geometry creation comes the external surrounding cylinder. Creating this cylinder is exactly the same as for the stress raiser: ***Preprocessor***, ***Modelling, Create, Volumes, Cylinder, Solid Cylinder***. The centre of the base will still be located at 0,0 (in CS 13) but the radius will change. It will be the value of the stress raiser`s radius plus 0.03, as mentioned at the beginning of commenting geometrical creation. The depth will be set at the same value as for the hole while it doesn`t have a major effect. In this case, the external cylinder has a radius of 0.105 and the depth remains as 1.125.

At this point, there is a volume which is unwanted. Enter ***Modelling***, ***Delete, Volumes Only***. Select the volume of the recently created cylinder (external) and press ***OK***. Now, go to ***Modelling, Operate,*** and ***Booleans.*** Choose ***Divide Volume by Area***. This same command has been used previously so follow the instructions mentioned before. The volume to be divided can be set with ***Pick All***. Press ***OK*** and now user needs to select the areas belonging to the cylinder. Enter in the command bar ***aplot*** to only plot areas, which makes picking areas much easier. Select the top and bottom lids of the cylinder as well as the two sections that conform the cylinder.

To end with the geometry creation of case 1, it`s recommended to split or divide the pipe in half along the X axis. This operation has no functional meaning alone it's a great help for seeing inside the pipe what stresses are where etcetera. This is done using the current coordinate system number 13. Go to the top tab where it says ***Workplane, WP Settings.*** Here all workplane options can be personalized but in this case, the interesting part is to show the grid that ANSYS has incorporated. Change ***Triad Only*** for ***Grid Only***. The other parameter needed to be changed here is the maximum and minimum. This refers to how big the user wants the

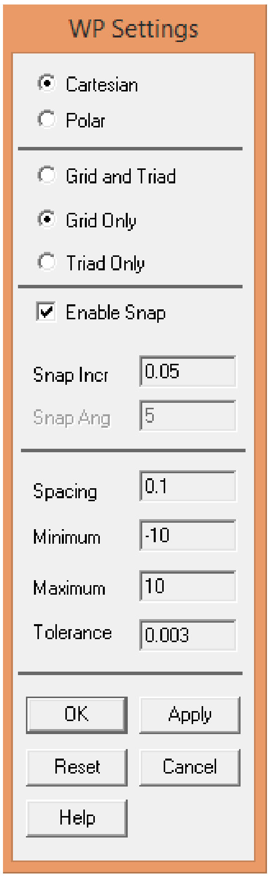


Illustration 31. Workplane settings

grid. Change -1 for 10 and 1 for 10, in minimum and maximum grid sizes. With these values, surely the grid will be bigger than the pipe`s diameter. Press ***Ok*** to confirm changes and the workplane`s grid will emerge, but due to the rotation established for CS 13, the grid is in the XZ plane and doesn`t contact the pipe. Therefore, it must be turned. This is done by going to ***Workplane, Offset WP by Increments***. Go to the middle part of the window that has just appeared (degrees) and change the slider value from 30 to 90 (a 90-degree turn is wanted). Then click on the button stating +Y and a spinning arrow on it. Immediately, the grid will turn into the right position. Press ***OK*** to confirm this movement and exit to the main screen. See illustrations 31 and 32.

Go back to ***Modelling, Operate, Booleans, Divide*** and this time select ***Divide Volume by Workplane.*** A well-known window appears asking for the volumes and for this case, it picked by ***Pick All***. Automatically, the operation takes place and the result should be as expected. At this point, grid isn`t useful so proceed to not show it in favour of the triad. To do this is to undo what was said in order to show the grid. Go back to ***WP Settings*** and tick the ***Triad Only*** box and accept. Also the 90-degree turn can be undone if wanted but it is not necessary because coordinate system number 13 is not going to be used again. In ***Offset WP by Increments*** make a 90-degree turn pressing button –Y with the arrow.

Now, with these operations done, set the original coordinate system number 0 as the workplane. Remember that it can be done undoing process done before. ***WorkPlane, Align WP with, Specified Coordinate System*** and enter number 0 in the box and finally press ***OK***. The final result of geometry creation for case 1, after creating the external cylinder, operating with it and dividing the pipe in half by the workplane, is shown in illustration 33.

In the next point is specified how to do the geometry creation procedure for case 2, which has similar stages to case 1 but is slightly different.

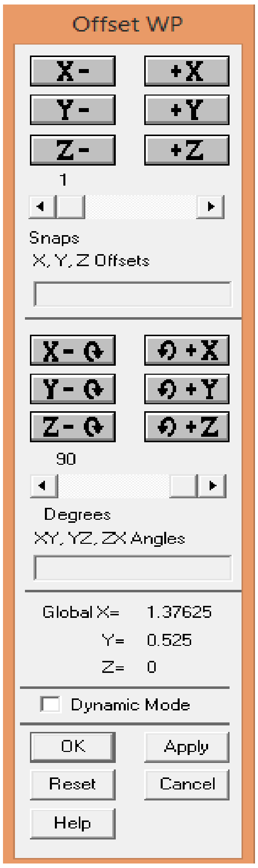


Illustration 32. Offsets by increments

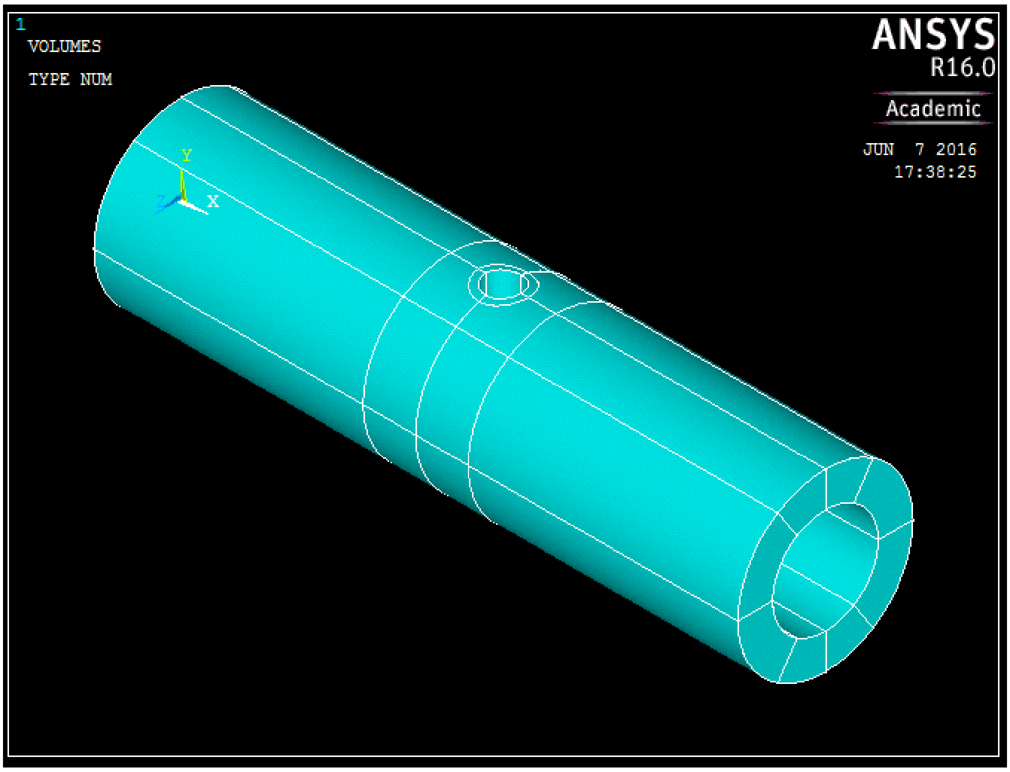


Illustration 33. Final geometry creation for case 1

5.3.2. CASE 2

As said in point 5.3. where all geometry cases were defined and justified, the difference between case 1 and 2 is dividing the pipe into 8 sections or not. Cases 2 and 3 (see 5.3.3) have the peculiarity that alpha is over 60 degrees or simply cannot be calculated using the arc sine function. In these cases, dividing the pipe into 8 isn`t possible so that operation is avoided, but then there could be cases where the hole was very small or very big. For the first options, the external cylinder is useful and this was called as geometry case 2. The condition defining a small hole was set at d/D to be less or equal to 0.3. In order to show how to create the geometry of these cases, the essential parameters are going to be D = 0.75, d/D = 0.1 and di/D = 0.2. Due to that the process is very similar, extracting some operations, this type of geometry creation will be lightly commented and only the innovating features will be mentioned.

First of all, material type and element definition is the same as for case 1 and will be the same for case 3 also. Then keypoints need to be created but in case 2, only the keypoints belonging to the hollow pipe section need to be created and these are only keypoints 1 to 10. The coordinates for this case can be obtained using the same excel worksheet and changing the parameters. See illustration 34.

Keypoint	Coordinate X	Coordinate Y
1	0	0
2	2.75250	0
3	0	0.0750
4	0	0.3750
5	2.75250	0.3750
6	2.75250	0.0750
7	1.18875	0.3750
8	1.18875	0.0750
9	1.56375	0.3750
10	1.56375	0.0750

Parameters	
D	0.75
d/D	0.1
di/D	0.2
L	2.7525

Illustration 34. Keypoint coordinates for case 2

The way to create keypoints was explained previously and the result is very similar to case 1 geometry. The next step is to create the lines to connect these created keypoints and this also, is the same method as explained in case 1. For the lines, follow illustration 24, because the result is very similar taking away the dividing lines.

Now, the areas need to enclose these lines. The same method is used in case 1 and 2 so follow instructions mentioned above in 5.3.1.

Area 1: compromises lines 2, 3, 4 and 5.
Area 2: boundaries are lines 4, 6, 7 and 8.
Area 3: surrounding lines are 7, 9, 10 and 11.

To obtain a volume, in case 1 these areas were revolved around an axis, which was the line going from keypoint 1 to 2 and this will extrapolate exactly the same for case 2, spinning that area in 4 steps. Nothing different has been done yet, except for changing the location of each keypoint.

Without being able to divide this component into 8 sections, the next task is to create the stress raiser and its surrounding cylinder. Recall that in case 1 a new coordinate system was needed in order to create the cylinder in the correct plane. The operation is identical as before but changing values for the size of the hole and outside cylinder, because now they are much smaller. These values were stipulated previously and can be found in case 1.

Parameters	
D	0.75
d/D	0.1
di/D	0.2
L	2.7525

Stress raiser	
X	0
Y	0
R	0.0375
Depth	1.125

Coordinate system	
X	1.37625
Y	0.675
Z	0
Rotation X	90
Rotation Y	0
Rotation Z	0

Exterior cylinder	
X	0
Y	0
R	0.0675
Depth	1.125

Illustration 35. Parameters for stress raiser and exterior cylinder in case 2

After doing this, user is in condition of creating both the stress raiser and the surrounding cylinder and this was done by ***Modelling, Create, Volume, Cylinder, Solid Cylinder***. This was realized by introducing the coordinates of where the base would be and then radius and depth of the cylinder. For this particular case, the values of the coordinate system, stress raiser info and external cylinder info are gathered in illustration 35.

When the stress raiser cylinder is created, it needs to be subtracted from the hollow pipe volume. Remember that this was done by entering ***Preprocessor, Modelling, Operate,*** and

Booleans, Subtract, Volumes. Press ***Pick All,*** click ***OK*** and select volume of the stress raiser cylinder. Immediately, the operation will take place.

Next comes the external surrounding cylinder. Creating this cylinder is exactly the same as for the stress raiser: ***Preprocessor***, ***Modelling, Create, Volumes, Cylinder, Solid Cylinder***. In this case, the external cylinder has a radius of 0.0675 and the depth remains as 1.125. Enter ***Modelling***, ***Delete, Volumes Only*** and delete the volume belonging to the external surrounding cylinder. Now, go to ***Modelling, Operate,*** and ***Booleans.*** Choose ***Divide Volume by Area***. The volume to be divided can be set with ***Pick All***. Press ***OK*** and now user needs to select the areas belonging to the cylinder. Enter in the command bar ***aplot*** to only plot areas, which makes picking areas much easier. Select the top and bottom lids of the cylinder as well as the two sections that conform the cylinder.

And as operated in case 1, for post visualization matters, it`s recommended to divide the pipe in half along the X axis. This will enable user to choose certain volumes so that the stress raiser will be seen better and in more detail. The procedure is exactly the same as before in case 1. Go to the top tab in ***Workplane, WP Settings***. Change ***Triad Only*** for ***Grid Only***. Also change -1 for 10 and 1 for 10, in minimum and maximum grid sizes. Do a 90-degree by going to ***Workplane, Offset WP by Increments***. This process is the same for all three cases so recall to illustrations 31 and 32.

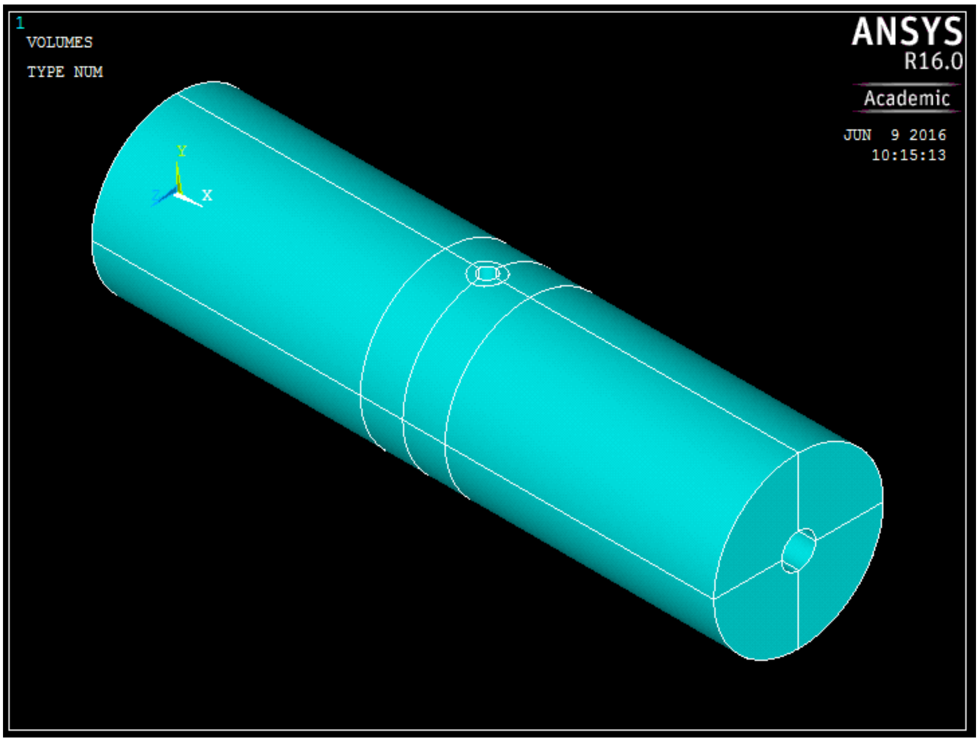


Illustration 36. Final geometry for case 2

Following up is the procedure in order to create geometry for case 3 and if case 2 wasn`t explained deeply, then case 3 will be merely mentioned because the operations are simple and all have been explained at least once in this project previously.

5.3.3. CASE 3

An example for reconstructing geometry belonging to a case 3 could be defined by the following parameters: D = 0.75, d/D = 0.45, di/D = 0.6. Recall past information, case 3 was a situation where dividing the pipe into 8 sections was impossible and also d/D was larger than 0.3. In these types of cases, the solution was not to create the exterior cylinder and mesh the entire central area of pipe finer. The surrounding cylinder had no function in these cases so it was erased. So, creating this type of geometry was basically creating the hollow pipe and then dividing that volume by the workplane to see better the inside of the component.

Keypoint	Coordinate X	Coordinate Y
1	0	0
2	2.75250	0
3	0	0.2250
4	0	0.3750
5	2.75250	0.3750
6	2.75250	0.2250
7	1.05750	0.3750
8	1.05750	0.2250
9	1.69500	0.3750
10	1.69500	0.2250

Parameters	
D	0.75
d/D	0.45
di/D	0.6
L	2.7525

Illustration 37. Keypoint coodinates for case 3

First of all, material type and element definition is the same as for case 1 and will be the same for case 3 also. Then keypoints need to be created. The coordinates for this case can be obtained using the same excel worksheet and changing the parameters. See illustration 37.

The next step was to create lines between keypoints. For case 2 and 3, the keypoints that link to create lines are in the exact same order as well creating areas. The numbering is identical for cases 2 and 3 so recall to 5.3.1. and 5.3.2 and follow the instructions there given. At this point of creation, the result should be the same as in case 2 but differing the values of keypoints and therefore the position.

The hollow pipe is created by selecting areas 1, 2, and 3 and the axis for revolution is line 1. Enter ***Preprocessor, Modelling, Operate*** and ***Extrude, Areas, About Axis.*** Select areas 1, 2 and 3, press ***OK*** and then select keypoints 1 and 2 and press ***OK*** again.

Next was creating the stress raiser cylinder and subtracting both volumes at that time, but before that, in order to orientate the cylinder in the correct plane, it was necessary to create an auxiliary coordinate system, by default called as number 11. Go to top part of the ANSYS window and press ***WorkPlane, Local Coordinate Systems, Create C.S, At specified Location.***

To create the cylinder remember that user had to click on ***Preprocessor***, ***Modelling, Create, Volumes, Cylinder, Solid Cylinder*** and then for subtracting volumes the path set in the main menu was ***Preprocessor, Modelling, Operate,*** and ***Booleans, Subtract, Volumes.*** In this particular case, the values needed to introduce in the corresponding boxes are represented in illustration 38.

Parameters	
D	0.75
d/D	0.45
di/D	0.6
L	2.7525

Stress raiser	
X	0
Y	0
R	0.16875
Depth	1.125

Coordinate system	
X	1.37625
Y	0.525
Z	0
Rotation X	90
Rotation Y	0
Rotation Z	0

Illustration 38. Values for stress raiser creation

The last task to do in this type of case is to divide the pipe in half by workplane, so that stress is seen better in the inner areas the component. This operation has already been done in cases 1 and 2, and therefore, the process is exactly the same for case 3, so it is not repeated again. To recall this step, go to point 5.3.1. where it is mentioned in detail. No other operation is done for these types of cases, in geometry terms (meshing is different) so the final result for this combination should be identical to illustration 39.

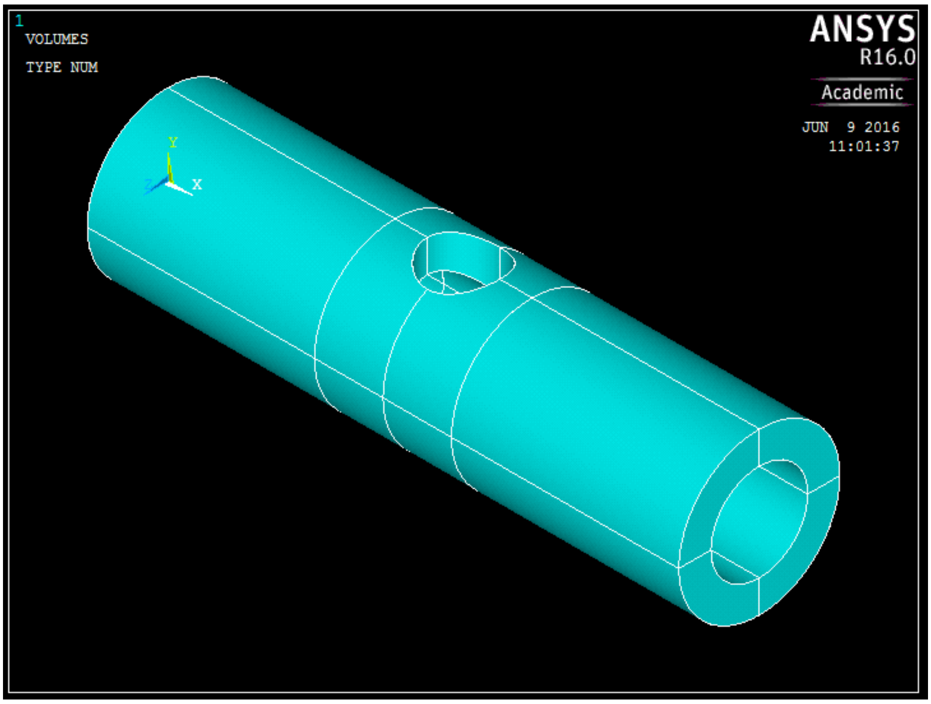


Illustration 39. Final geometry for case 3

5.4. MESH GENERATION

This next stage is probably the most important step in the analysis of any finite element program. The solution`s precision is going to be based, generally, on the degree of refining of the mesh. A finer mesh will lead to more accurate results when a coarse mesh will extrapolate into erroneous results due to the lack information in the resolution. But, as one can expect, as the mesh gets finer and the elements size reduces there has to be a price to pay and it comes in form of calculus time and a bigger data base o store information.

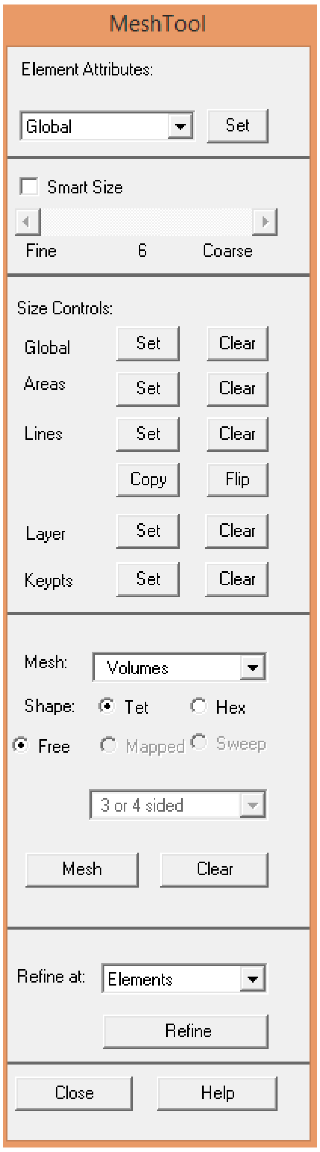


Illustration 40. Mesh Tool

The second issue mentioned isn`t a major problem because modern computers have big hard drives (HDD) that allow a massive information storage. The biggest problem is calculus time, which is the time that it takes the computer (ANSYS) to solve the problem. Finer the mesh, more elements in that mesh and more differential equations need to be solved.

For engineers who work in this area, time is precious and can`t be wasted waiting for a program to solve some problem, that initially might be wrongly set out. So in real life problem, what occurs is a compromise between engineer and software. It is not recommended to mesh, uniformly, the model, but take in consideration the most critical areas of it. When analysing a component, it is rare to want the total information of the model, instead, the critical sections are looked at in detail, so all computational efforts are directed to critical and most solicited sections. This could sound as a big problem when the engineer does not know or cannot foresee where critical sections are but in the case of this project, this area is known, being the stress raiser. This means that the mesh is going to be finer in the middle section where the stress raiser is and here is where the most nodes are going to be. At both ends of the pipe, no massive stress is going to appear and also, this is not the area desired as study. Here, the elements will be bigger and the mesh will be coarse. How big or how small the elements are, depends on the engineer`s experience so, because the user doesn`t necessarily need to be an expert in ANSYS, one must try a size of element, mesh and see the result and refine until a final, satisfactory result is reached.

Inside the software used for this project there is a mesh generation procedure to make the refining easier. It can be done automatically or manually, telling ANSYS the size of element wanted in each place. This, translated into ANSYS language is done inside ***Preprocessor***, ***Meshing*** and select ***Mesh Tool.*** There are many different ways to generate a mesh in ANSYS which all can lead to the same result but choosing ***Mesh Tool*** is one of the most complete ways to do it because it summarises, into one, all the other options. The ***Mesh Tool*** window should show as illustration 40.

ANSYS can create an automatic mesh enabling ***Smart Size*** and pressing ***Mesh***, it can then refine the created mesh with function ***Refine*** but this is not this case. For this project, mesh parameters have been introduced manually within ***Size Controls.*** Here, one can pick areas, lines or keypoints and establish a certain size for these elements. Automatically ANSYS will create a mesh according to these orders.

Due to the fact that in this project there are three different cases, each one with a different geometry, the meshing task necessarily is individual for each case because the numeration of keypoints, lines, areas and volumes changes between each case. First of all, the procedure to mesh case 1 will be mentioned.

5.4.1. CASE 1

Here comes a very important aspect to have in mind. Looking towards the load applying stage, in order to apply momentums and torsional forces on this component, an element that includes rotations in its freedom degrees is needed. One cannot apply these types of forces on the TET 10 node, all forced onto one keypoint or node. In order to avoid singular configurations, these forces need to be applied on a surface instead of a point.

This means telling ANSYS that the entire front section moves and rotates in the same way. Later on in point 5.5, this will be explained in detail, creating a rigid section, making al nodes on that surface move and rotate as a stiff solid element. But for now, while meshing, it is clear that not all keypoints can be meshed with the same element. This involves meshing the central keypoint of the right handed section of the component (keypoint number 2) with ***Mass Element*** while the rest of the component will be meshed with ***TET 10 node*** as said before. Now, it`s very important to mesh this keypoint before doing anything else, because when ANSYS is told to mesh, it selects an element to mesh the desired feature with x amount of nodes. Due to a keypoint is meshed (unidimensional character) ANSYS will mesh keypoint number 2 with a unidimensional element containing only one node. Because the entire component is going to be meshed, thousands of nodes are created so in order to supervise and know which is the one belonging to keypoint number 2, node which will be 'master node', it`s important to mesh keypoint 2 first so that the first node created (node number 1) will always be at that same place. This is a great help when looking into the macro.

Go to top part of window and click ***Plot, Keypoints*** and ***Keypoints*** again or enter the ***kplot*** command. Now, only the keypoints should be visible. To include this in ANSYS go to ***Meshing, Mesh Tool.*** Now first, the element attributes for each keypoint must be stipulated. At the top part of ***Mesh Tool*** select ***Keypoints*** and then by the side press ***Set.*** Select every keypoint with ***Pick All***. Illustration 41.

Once selected it, press ***OK*** and another window will jump out. Here is where the attributes are selected. The attributes of all these keypoints are going to be ***TET SOLID 187***. Press ***Apply*** and then select keypoint number 2 and set that the element type for meshing is ***Mass 21.*** This means repeating the process descripted previously. See illustrations 42 and 43.

Illustration 41. Keypoint attribute selection

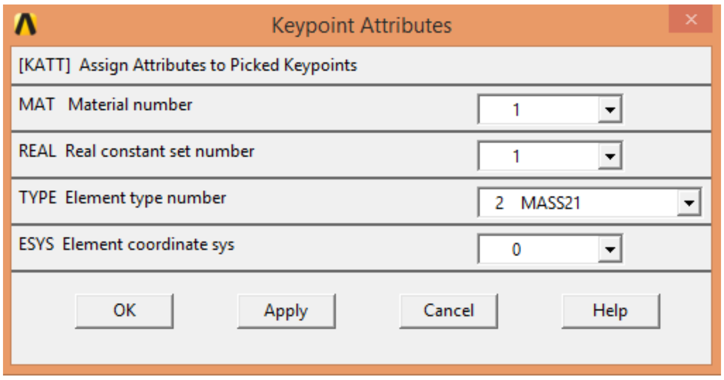


Illustration 42. Setting keypoint number 2 attributes

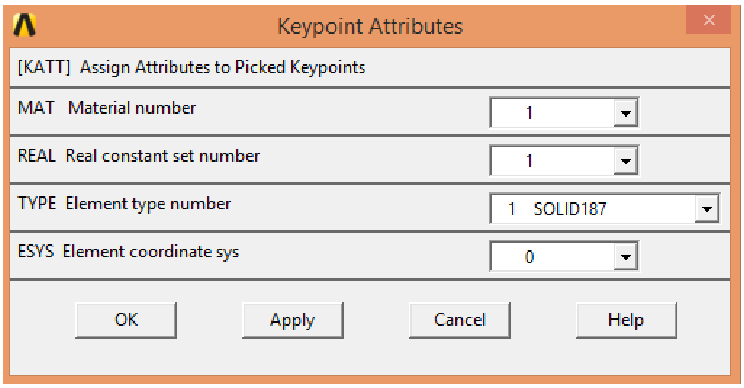


Illustration 43. Setting total component`s attributes

Once set all attributes to keypoints, one can mesh keypoint number two in order to create that one dimensional element with one node mentioned before. After that, all the component can be meshed, but first it`s needed to mesh keypoint 2. This is done by going to ***Mesh Tool***, at mid table where it says ***Mesh***. Here pick, from all available, ***Keypoints*** and beneath that, next to the ***Clear*** button, press ***Mesh.*** A selection window appears waiting for user to select keypoint number 2. Select it and press ***OK***. This keypoint has been meshed and to double check that there is only one element with one node there, enter ***Elist*** and/or ***Nlist*** commands to obtain a list of all elements and/or nodes, respectively.

When this is finalized, the next step is to set the size of each keypoint line and area. First, the size of some keypoints will be set, followed by line sizes and area sizes. Go back to ***Mesh Tool*** and scroll down to ***Size Controls***. Find Keypts and click ***Set***. In order to select the wanted keypoints, the safest way is to select keypoints by location because, independently of the parameters, these keypoints will always lay at this location. To do this, go to the top tab and press ***Select***, ***Entities***. ***Keypoints by Location.*** By default, a black dot is found inside the X coordinate label and it must remain like that. The keypoints wanted are at a distance of 0 in the X axis. Press ***Apply*** to confirm and then, also in the X coordinate label, enter the number 2.7525, which belongs to the end section of the pipe. But this time one must change the ***From Full*** option to ***Also Select***, in order to maintain the previous selected keypoints. Finally, press ***OK*** to confirm. Now, user can set the size for these keypoints. Go back to ***Mesh Tool*** and press ***Keypoint*** and ***Set***. Now that only the desired keypoints are selected, press ***Pick All*** (remember that the ***Select Entities*** only allows user to manipulate what is selected). A window will flash up, just as in illustration 44 asking for the size of these keypoints. In this case it will be 0.45. It is to mention that these first values defining sizes are merely orientating because later on, in the macro, these values will be in function of the three main parameters (D, d/D and di/D).

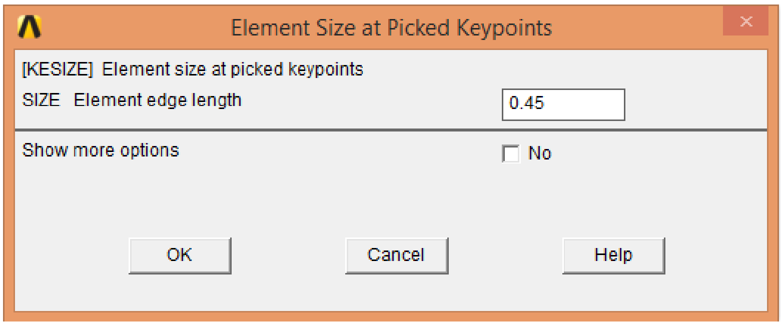


Illustration 44. End section element size

The next entities to be meshed are the lines. The lines that are interesting to set their size are those belonging to the stress raiser and also those corresponding to the external cylinder. The idea is to set a very fine size in the stress raiser area and then just outside there, mesh a little bit gross, but still fine in order to capture information near the hole. First of all, as before, the entities wanted are the lines so enter in the command bar ***Lplot*** which tells ANSYS to plot only the lines. Zoom in if necessary so to pick all lines corresponding to the inner stress raiser. In total, there will be 24 lines selected and the size assigned to these lines will be 0.01 (notably smaller than the end sections). See illustration 45.

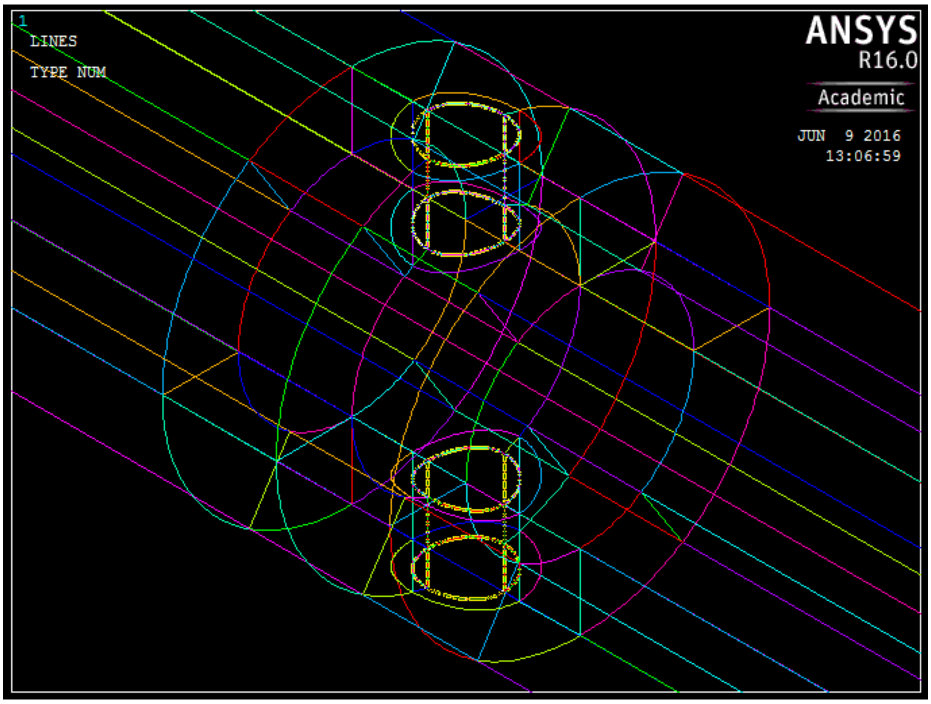


Illustration 45. Stress raiser line size

Do the same process for the lines belonging to the external cylinder that surrounds the hole. The selection is the same as for in illustration 46 and then the size for these lines will be 0.03. This size is a little bit bigger but still fine enough to capture stress with accuracy.

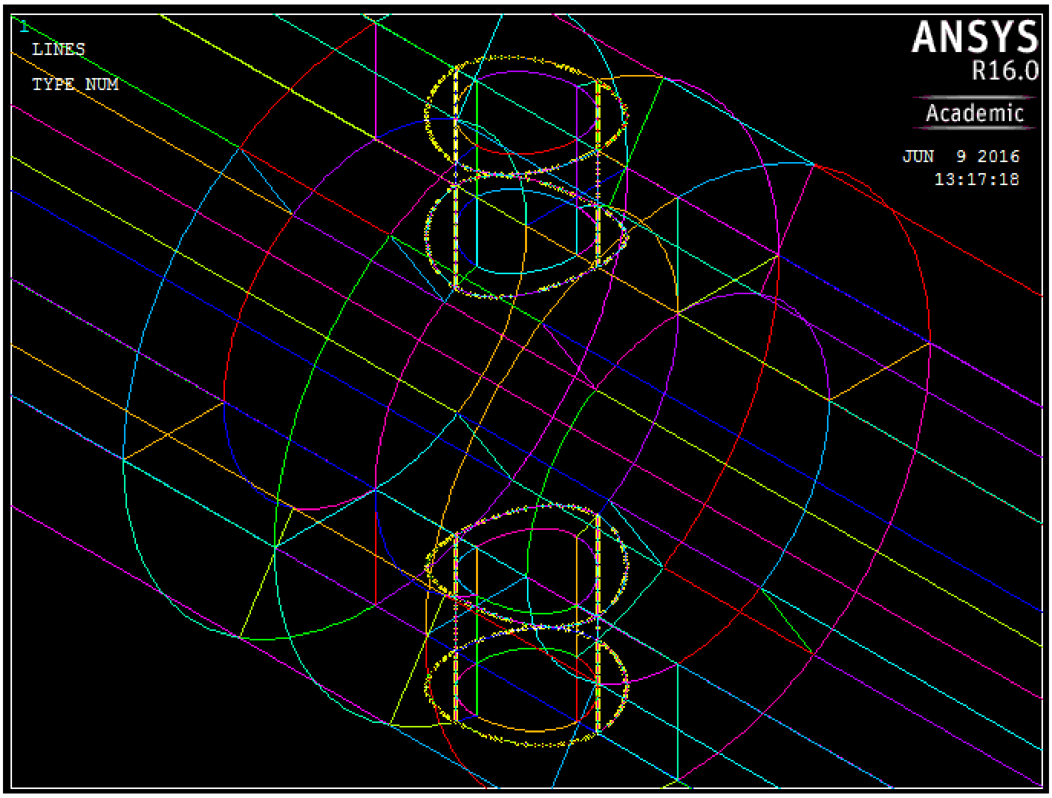


Illustration 46. External cylinder line size

Finally, the last entity that is going to be mesh in this case 1 is area and this explains dividing in 8 sections the pipe. The user needs to economize node because there are only 32000 available, so they need to be put in certain areas instead of others. Dividing in 8 the pipe helps a lot when setting area sizes. The strategy adopted is to tell ANSYS that all areas near the stress raiser are going to have a certain (small) size and those areas away from hole, which don`t have any importance in studying them, will have a bigger, gross size. This way most of the nodes will concentrate at the stress raiser.

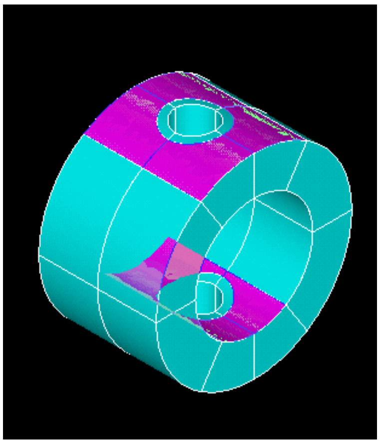


Illustration 47. Fine area sizing

Now, the procedure to do this is similar to before when setting line sizes but in this case, with areas. Go back to the ***Mesh Tool*** and press the button ***Set*** next to ***Areas.*** The areas which will have a smaller size, near the hole, are those shown in illustration 47. The number of areas selected should be equal to 16, 8 from the top and another 8 at the bottom. When these areas are all selected, a window just like in figure 50 will come up asking for the size desired at these areas and user must introduce 0.04. The ***Select Entities*** option is a great help in these chores.

Now, the exactly same thing needs to be done with the areas far from the stress raiser, represented in illustration 48. These areas will have a size of 0.1. Notice that this last size is notably larger than those near the stress raiser. As usual, when selected an entity and pressed the ***OK*** button, ANSYS asks for size. Complete the blank space with the number 0.1.

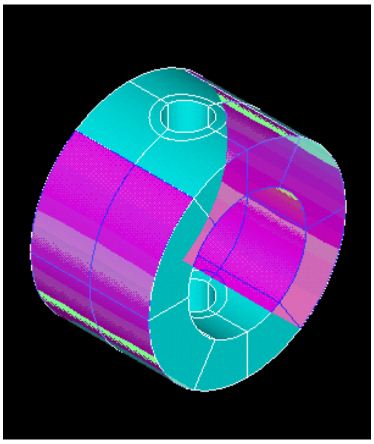


Illustration 48. Area selection

Press ***OK***, as always to exit this phase. With all previous conditions inserted into the program, the only thing left is to mesh. Beneath ***Size Controls*** go to ***Mesh*** where there is a scroll, which askes what to mesh. Select ***Volumes***. Shape keep it as ***TET*** and by default stay with ***Free*** meshing, allowing elements to adopt certain configurations differently to mapped where it`s all squared out. Under these commands press the button ***Mesh***. A window will flash up asking for which volumes to mesh, so obviously, all volumes are desired to be meshed so pick all volumes of component or, even quicker, press ***Pick All*** and then press ***OK*** and wait for program to finalize meshing process. The final result should be as shown in illustration 49 with the total component meshed.

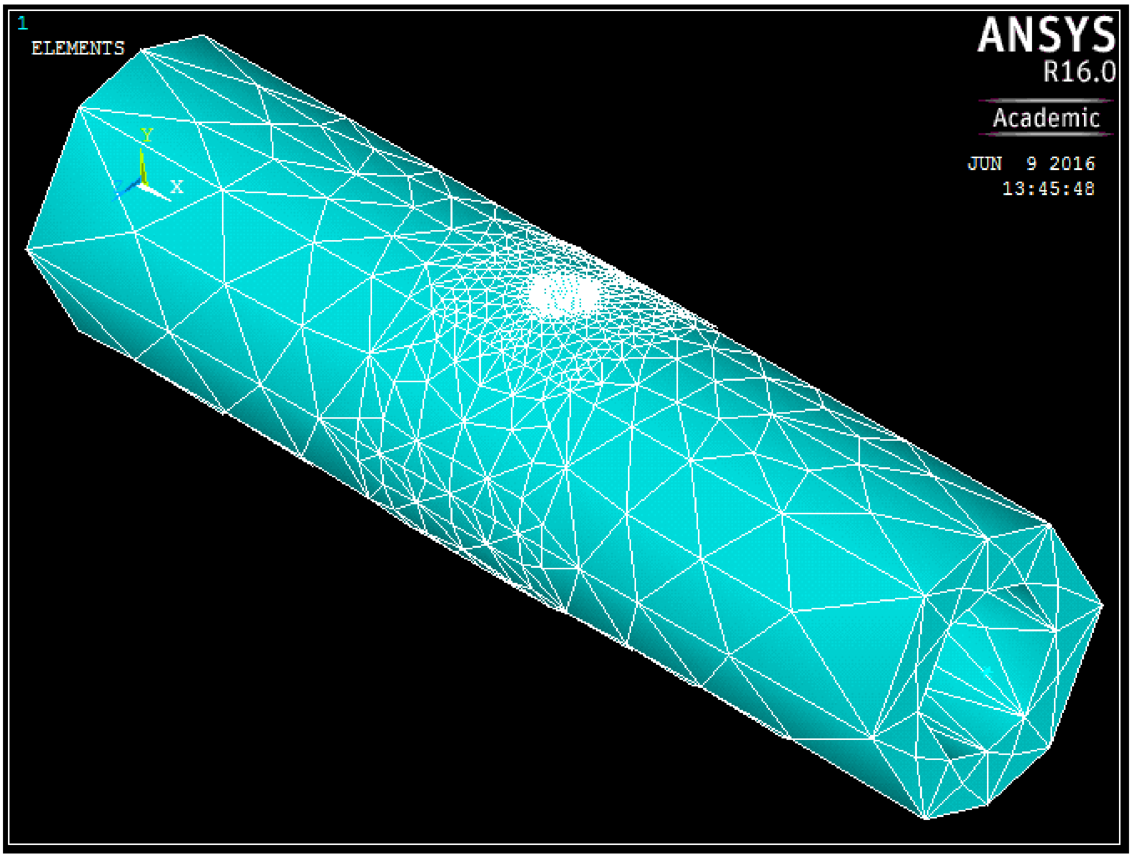


Illustration 49. Mesh for case 1

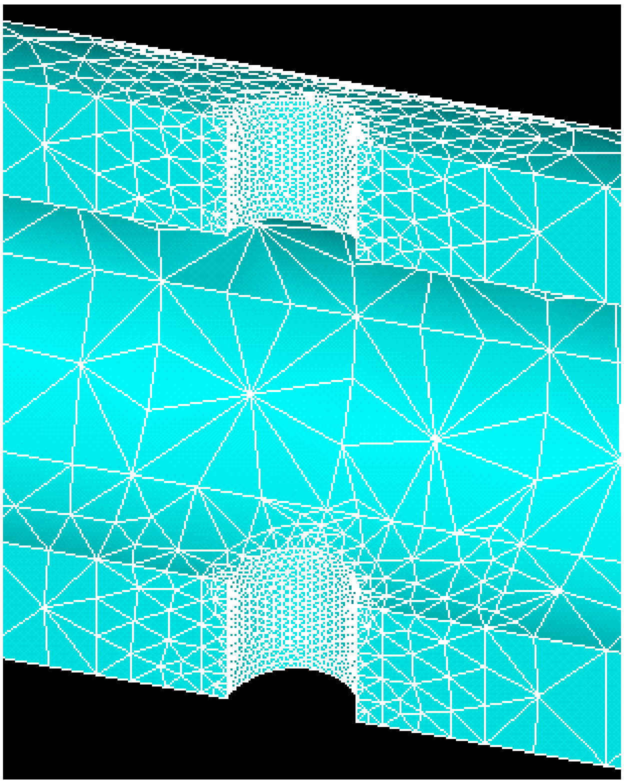


Illustration 50. Inside mesh view

As it can be appreciated, elements are bigger nearer to the extremes and finer as approaching stress raiser. One can see inside the pipe by using the buttons at the right hand side of the screen or go up to the top part of the screen to ***Select***, ***Entities, Volumes.*** After selected the wanted volumes, return to ***Select***, ***Entities*** but this time press on ***Elements Attached to Volumes*** and press ***Apply***. This will now show the elements attached to the selected volumes and is quite handy when plotting results or seeing other features of the component. Also, to verify that all nodes are near the stress raiser is, there is a chance into plotting nodes. Enter the following commands: ***Allsel,all*** which reselects everything again and then ***nplot***. This last command plots all nodes and allows user to see where nodes are concentrated and by the looks of illustration 51, the objective has been accomplished.

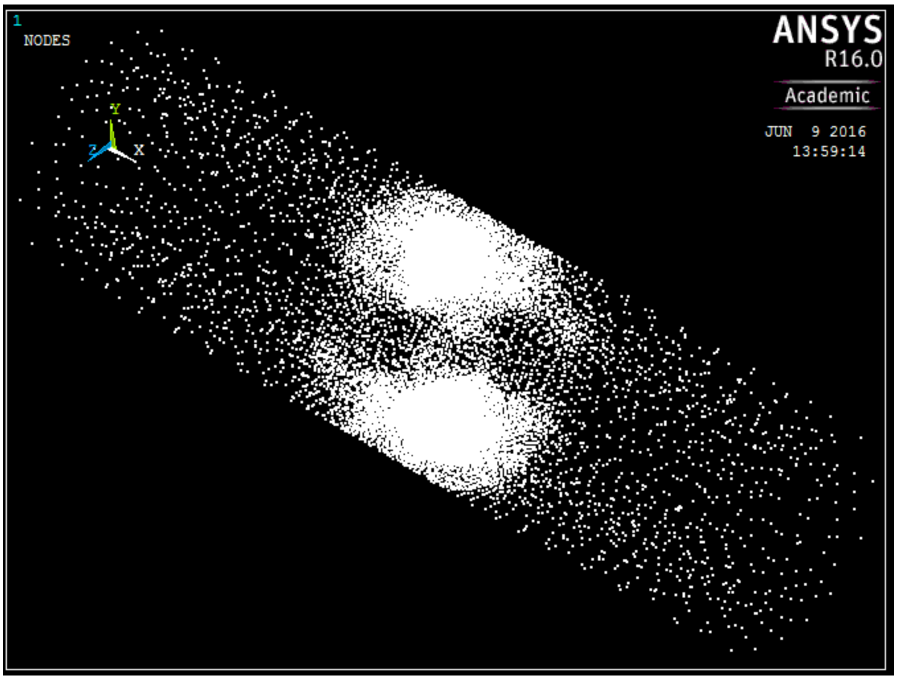


Illustration 51. Node plot

The elements at the fixed end can be big, as shown in figure 49 because this area is not of calculus interest. This area is only for applying boundary conditions. The representation shown in this figure only represents vertex nodes, so apparently, the geometry isn`t perfectly described. But, the geometrical approximation used in numerical calculus uses all nodes of the elements, showing that the boundary is perfectly represented and the result is shown in figure 51.

Finally, the last issue to answer is about the size of each element at each place. Well this isn`t a number that came in a manual or catalogue, it`s based on the engineer`s or designer`s experience. For this project, due to nobody needs to be an expert in meshing elements, the sizes are fruit of hours of trying different configurations and adjusting sizes, meshing lines, or areas until a final, satisfactory result is achieved. This process is time consuming and it finds an obstacle in the way. This project has been done using an academic introduction ANSYS version and as one can suppose, this isn`t the same as for companies who acquire licenses. These academic versions are limited in the amount of nodes resulting after meshing any component and due to the fact that here is chosen a 10 node element to mesh, well, after meshing everything, thousands of nodes are created, specially where the stress raiser lays. Here is where most of the nodes are, concentrated in that area. This academic version is limited to 32000 nodes so when user is in the phase of finding the correct size for each area, sometimes it might not be possible because the 32000 node limit will be overpassed. One must mesh considering the maximum amount of nodes possible supported by ANSYS while achieving a satisfactory mesh which wraps up most of the important information at the output.

As said, the meshing process took some time to configure and to optimize. Different combinations, with different sizes were tried until a satisfactory result was reached. In this project many different configurations were tried, reflected in illustration 52.

	NODES	PRERR 1	PRERR 2	PRERR 3
1	26866	2.88	3.87	3.55
2	26000	3.94	5.28	4.74
3	+32000	-	-	-
4	22133	4.89	6.70	5.90
5	22000	5.55	7.62	7.15
6	25663	3.05	4.06	3.91
7	23200	4.93	6.80	5.80
8	26873	2.81	3.70	3.55
9	29500	2.77	3.70	3.52
10	28110	2.67	3.59	3.41
NEW GEO	24711	3.86	5.08	4.86
A	27573	3.10	4.10	3.86
B	24637	3.96	5.13	4.83
C	29864	3.09	3.95	3.86
D	+32000	-	-	-
E	28511	3.16	4.13	3.90
F	28840	3.23	4.20	4.12

Illustration 52. Meshing process

In order to understand this chart, some few details must be explained. Each row represents a different configuration which was tried out (17 in total). When a configuration was set, the only way the know if it was better than the previous was comparing the number of nodes and specially, the percent error in structural energy norm (PRERR). This name comes directly from the command that allows user to acknowledge the value of this error. As defined in the Command Reference inside ANSYS own Help Module, PRERR is defined as: *The structural approximation is based on the energy error and represents the error associated with the discrepancy between the calculated stress field and the globally continuous stress field. This discrepancy is due to the assumption in the elements that only the displacements are continuous at the nodes. The stress field is calculated from the displacements and should also be continuous, but generally is not.*

In simple words, this PRERR is a precision indicator saying how close user is to the real solution. It is said that a good analysis (in these conditions) is that which has a PRERR under 5%. Based on the two restrictions set by the 32000 node limit and the 5% PRERR, then after many attempts, the optimum meshing configuration was established as C, which has been shown in this project. All cases have been meshed with this configuration or with similar patterns (cases 2 and 3).

An important aspect to say about the PRERR command is that it executes the error evaluation at every element selected. If user has the entire pipe selected and executes the PRERR command, it will evaluate the whole component and the PRERR would be around 10 or 20 percent. Why is this? Well, it`s due to the fact that at the end section of the pipe, keypoint sizes were 0.45, which is gross, therefore the error in this region, in which we aren`t interested, is very rich. So, in order to obtain these results, one must select central volumes with the corresponding elements and evaluate PRERR in these conditions.

5.4.2. CASE 2

The following point describes how to mesh cases like case 2. Following the previous, implemented combination explained to create geometry, here is detailed how to mesh this case. The fundamental parameters were: D = 0.75, d/D = 0.1, di/D =0.2. It`s to mention that in this project is explained how to create and mesh a component with a certain combination of essential parameters but it`s applicable to all cases within the limits established pages before. Also, as done when describing how to create geometries for cases 2 and 3, some details were neglected because they were already explained. The same principle is applied in the meshing process. Some operations are exactly the same for cases 1, 2 and 3, so being explained once in detail in case 1 is enough. Repeating the same descriptions isn`t adequate and unnecessary.

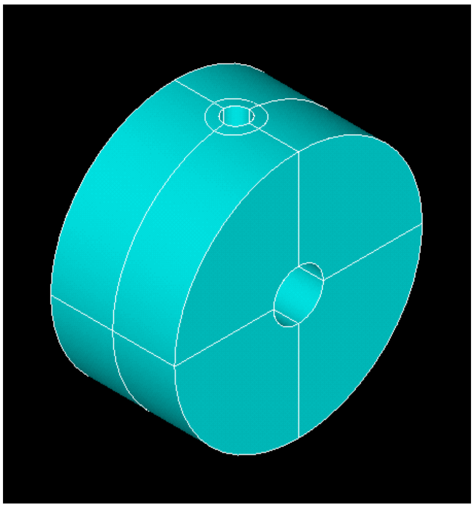


Illustration 53. Area selection for case 2

The first thing to be done is entering the ***Mesh Tool*** and setting the attributes for keypoints. This is the same sequence as for case 1, where user selected every single keypoint and set their attributes as ***TET SOLID 187***. Then, only keypoint number 2 was selected and its attribute was defined as ***Mass 21***. This was due to the boundary condition established ahead. To recreate this process recall to the beginning of point 5.4.1.

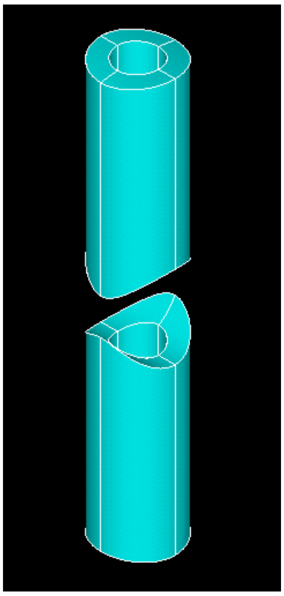
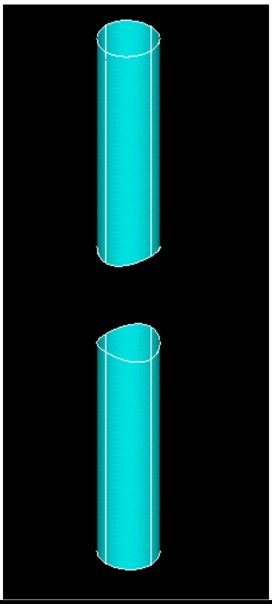


Illustration 54. External cylinder area selection

The next step was to set sizes at different entities, being these keypoints, lines and areas. The difference between case 1 and 2 is slight so the process will be similar. First thing to be sized are the end section keypoints, where the elements are bigger. Follow instructions given in case 1 when keypoints were selected by location, assuring that only the desired keypoints were sized. The size of these keypoints is set at 0.45, the same as for case 1. Now here comes the difference between case 1. Remember that case 1 had the pipe divided into 8 sections but in cases 2 and 3 this isn`t the case. Instead of setting sizes for lines, area sizes will be set. First, select areas belonging to the central part of the component which will have a reasonably fine size (in one hand to mesh finer and on the other to decrease PRERR).

Select areas shown in illustration 53 and set as size for these areas of 0.06. Use ***Select Entities*** for picking. Now repeat the same picking process but now select only areas belonging to the external surrounding cylinder and define the area size as 0.03 (areas reflected in illustration 54). Finally, pick just, in the same manner as before, the areas belonging to the stress raiser and establish a size of 0.01. See Illustration 55.



All sizes for this case have been set and the only task left is to mesh. Remember to mesh in first place keypoint number 2, in order to create node number 1 for post tracking it. This was done inside ***Mesh Tool, Mesh Keypoints***. Select keypoint number two and press ***OK***. The same process was done in case 1 so recall previous information. After this, user can mesh all volumes with ***Mesh Tool,*** obtaining the result shown in illustration 56.

Illustration 55. Stress raiser area selection

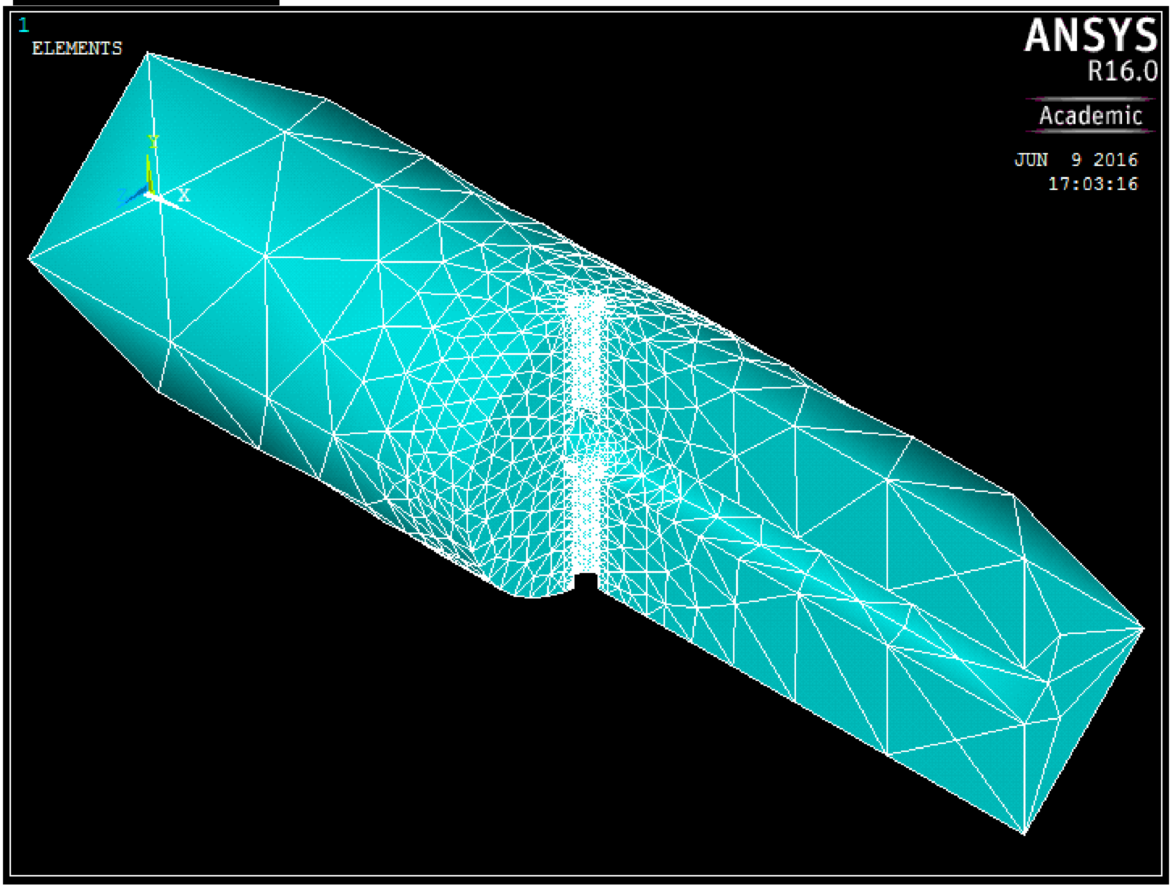


Illustration 56. Final mesh for case 2

Remember that this is a representation of ANSYS. It might seem that the volume doesn`t describe the circle shape because of an element plot but in the numerical approximation calculus, all nodes are taken account for, being these mid side nodes which aren`t represented in illustration 56. Recall figure 51 to see a node plot where the circular section is perfectly described.

5.4.3. CASE 3

At last, the final type of geometry case found in this project is case 3. It is the simplest case existing due to its shape. Case 3, as case 2, could not bear the dividing into 8 operation and as an addition, the surrounding cylinder was also eliminated because it didn`t offer any advantages. For this particular case, the entire mid-section area would be meshed with the same size, excepting the stress raiser, which would be much finer. The parameters will be the same for the geometry creation demonstration being D = 0.75, d/D = 0.45, di/D = 0.6.

The initial step was to set the attributes for keypoints, being all of them ***TET SOLID 187*** except for keypoint number 2, which would be ***Mass 21.*** This has been explained and done already in cases 1 and 2 and it`s the same for case 3 so it`s not going to be repeated. The next step is to set the sizes for keypoints, lines and areas.

First, the end section keypoints will be sized using exactly the same method as in cases 1 and 2 and the size for these keypoints is 0.45. Then, as in case 1, the lines belonging to the inner stress raiser cylinder are sized at 0.015. The result of picking these lines should be as in illustration 45. Posteriorly, the all mid-section areas are selected just the same as in illustration 53 and the size is 0.06. Last but not least, the areas that conform the inner stress raiser cylinder are selected (just the same shown in illustration 55) and the size set at 0.015, the same size for the lines. After meshing keypoint number 2 first and then meshing all volumes, like in previous cases, the result is shown in illustration 57.

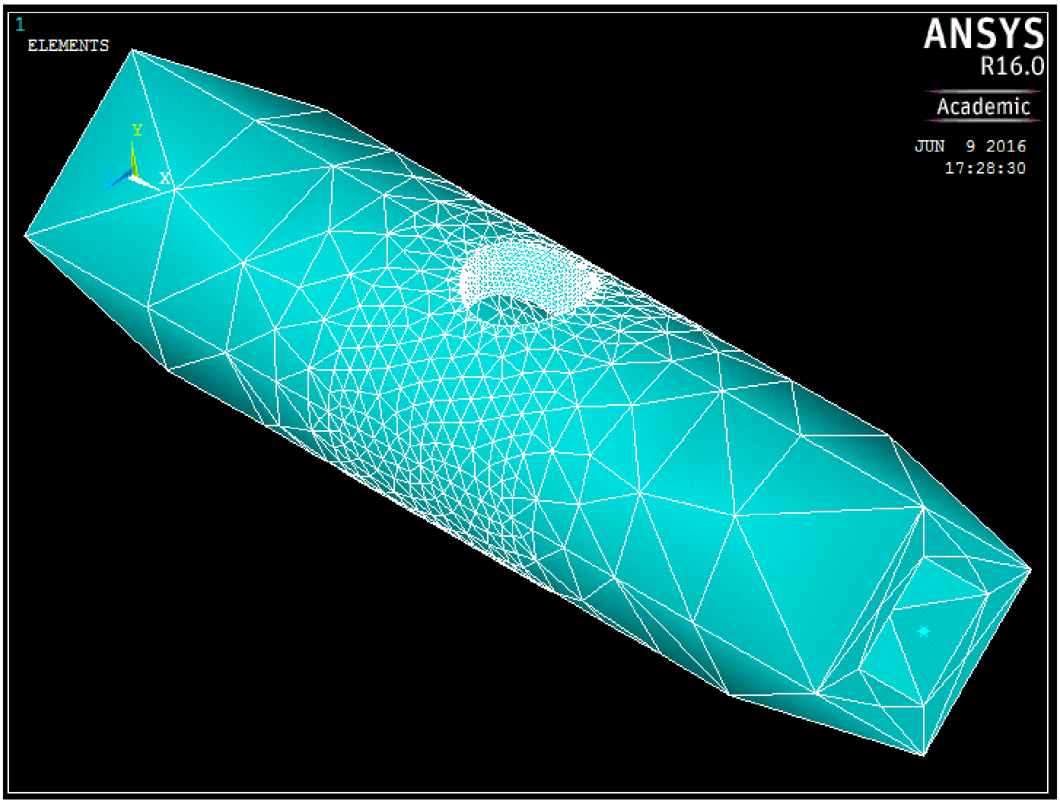


Illustration 57. Final mesh for case 3

Looking at the end section of the pipe, it can be appreciated that the resulting mesh has a triangular shape (this occurs in cases two and 3). This shape appears fruit of meshing sizes and one could think that it is an inappropriate mesh because it doesn`t capture the circular boundary of the pipe. Well, this can be answered by plotting volumes and nodes and it will be proven that nodes follow the circular limit. The elements don`t follow this pattern but it doesn`t matter because this element has 10 nodes so the boundary is perfectly covered.

This is merely an element representation but in numerical calculus, nodes come into play. It can be seen in illustration 51 that when nodes are plotted, the circular section is shown.

5.5. BOUNDARY CONDITIONS AND LOADS

The last step in the modelling procedure is establishing boundary conditions and the loads applied on the component. It is said before that the left end of the component will be affixed, not allowing any type of movement or rotation and at the other extreme of the component is where all the loads are going to be applied. Boundary conditions are important in analyses because, primarily, without them ANSYS cannot calculate any results. As far as ANSYS is concerned, there is a component floating in a workspace without any restrictions. If any loads were applied, it is impossible to calculate stress, reactions, forces etc. Luckily, if one forgets to apply them while modelling, it will show up in a flashing window reminding you that there are no loads and/or restrictions set on the component, but it`s still a vital and necessary step to do in any Finite Element Method program.

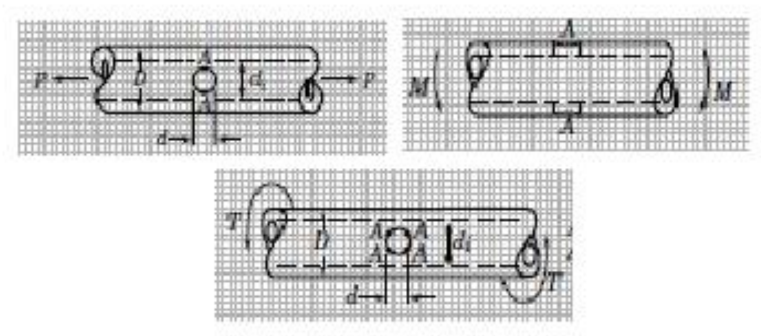


Illustration 58. Load cases

Now, according to Peterson`s Stress Concentration Factors 3rd edition, there are three different load cases for this component, being axial stress called tension according to bibliography (axial tension in +X direction), bending and torsion. An example of all three cases can be seen in figure 66, courtesy of the mentioned bibliography.

It`s to mention that 5.5, applying load conditions is common to all three cases. Geometry or meshing isn`t involved so it will be explained using parameters from case 1. Well, first is shown how to affix the left side of the pipe so it cannot move or rotate due to it is easier and quicker to do. Afterwards all loads will be set.

With all the nodes recently created, it might seem a task to only select the left side section. This is the only area wanted to pick so that it`s rotation and displacement is null. First of all, go to ***Select, Entities***. The first scroll asks what to select so say ***Areas.*** Beneath that there is another scroll displaying all the different techniques implemented in ANSYS to pick an element. In this case, choose ***By Location***. The areas wanted are set in X, Y and Z coordinates relative to world coordinate system. As it can be noticed, these areas, all, are at X equals to zero. Tick the box ***X coordinates*** and in min, max enter a 0. Finally, press ***OK*** and all those areas will be selected. To make sure, go to ***Plot, Areas.*** See illustration 59.

Now that the areas are selected they need to be affixed. This is done going inside ***Preprocessor*** and one must jump to the ***Loads*** tab. Here, go into ***Define Loads***. ***Apply, Structural.*** For now, select ***Displacement, On Areas.*** Suddenly a window, which is quite familiar at this point of project, shows up. ANSYS needs to know which areas want to be affixed. Because previously the wanted have been selected (using ***Select*** tool) even if user wanted to select any other areas, they wouldn`t appear. Choose all 8 areas and press ***OK***.

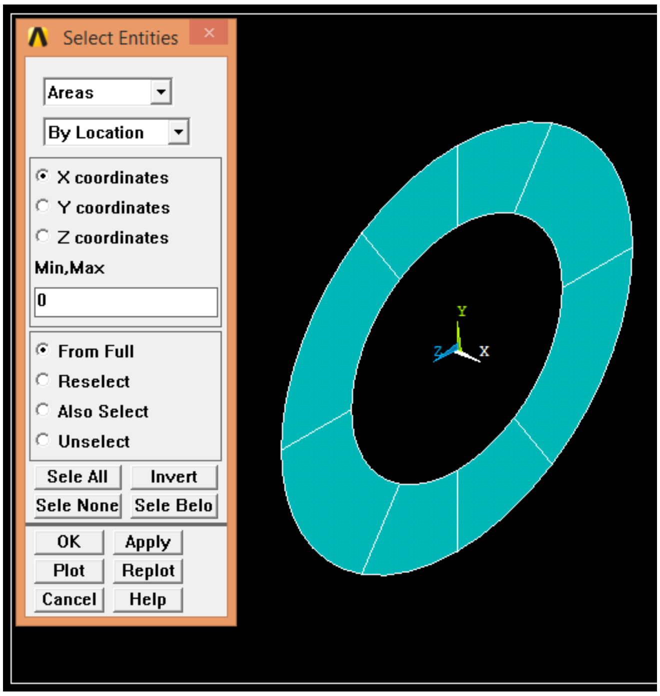


Illustration 59. Embedment area selection

After pressing ***OK***, another window flashes up saying in what way does the user want to apply a structural displacement, this is, how many freedom degrees can the area have. For this case, no displacements or rotations are wanted so choose ***All DOF*** (all freedom degrees).

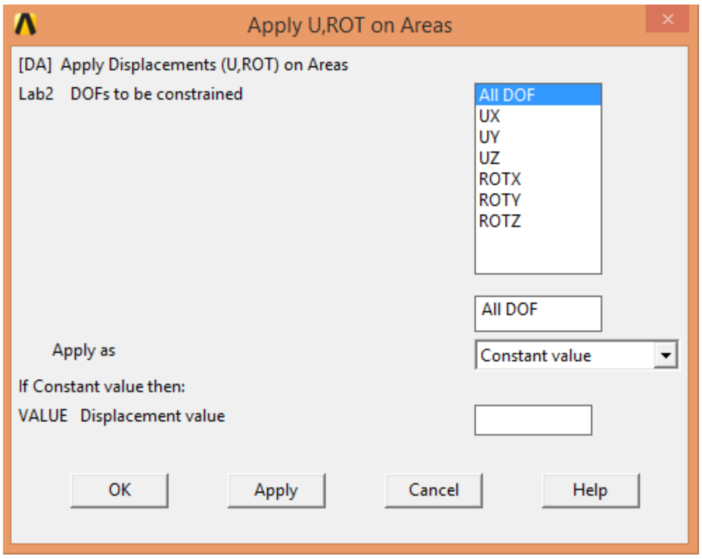


Illustration 60. Boundary conditions on embedment

Now that the embedding is set, the last task in order to complete the component modelling is to apply the desired loads. This could seem easy but it holds some unwanted surprises. As mentioned before when meshing, due to a three-dimensional problem and the vary of loads, one cannot apply these loads on a single point. In order to obtain correct results and avoid singularities, loads are applied per surface. This necessarily needs a rigid cross section at the right extreme of the pipe. A rigid section is so, that all elements contained in it will move and rotate in the same way, just as a stiff solid. The condition into creating a rigid region is to create what is called in ANSYS a 'master node' which will dictate the way its 'slave nodes' will move. In this project, the 'master node' is node number 1, created in 5.3. on purpose, just for this task (centre of cross section) and the 'slave nodes' will be all nodes on that face excepting node number 1.

Before even creating this rigid section, to make the chore easier, the user must know that creating a rigid area concerns nodes. The nodes that are of interest are the one from the right end face and node 1 as master node. So it`s recommended to go to ***Select, Entities*** and choose ***Nodes, By Location***. Enter in ***X coordinates*** 2.7525 which is the value of the length of the tube, so that all nodes at that distance are selected. This avoids problems when not selecting correctly or doubting if the selection is the appropriate. For the rigid area creation enter ***Preprocessor*** and click on ***Coupling / Ceqn.*** After pressing ***Rigid Region,*** a selecting window will appear and reading what is said below, ANSYS asks for the master node. Either pick node number 1 or enter its number manually so it is selected. After that, press ***OK*** and ANSYS will ask for the slave nodes. These are all the nodes on that face excepting node 1, so pick them and once again press ***OK***. The next window that appears leave in blank and press ***OK*** to create the region, resulting in a model just as in illustration 61.

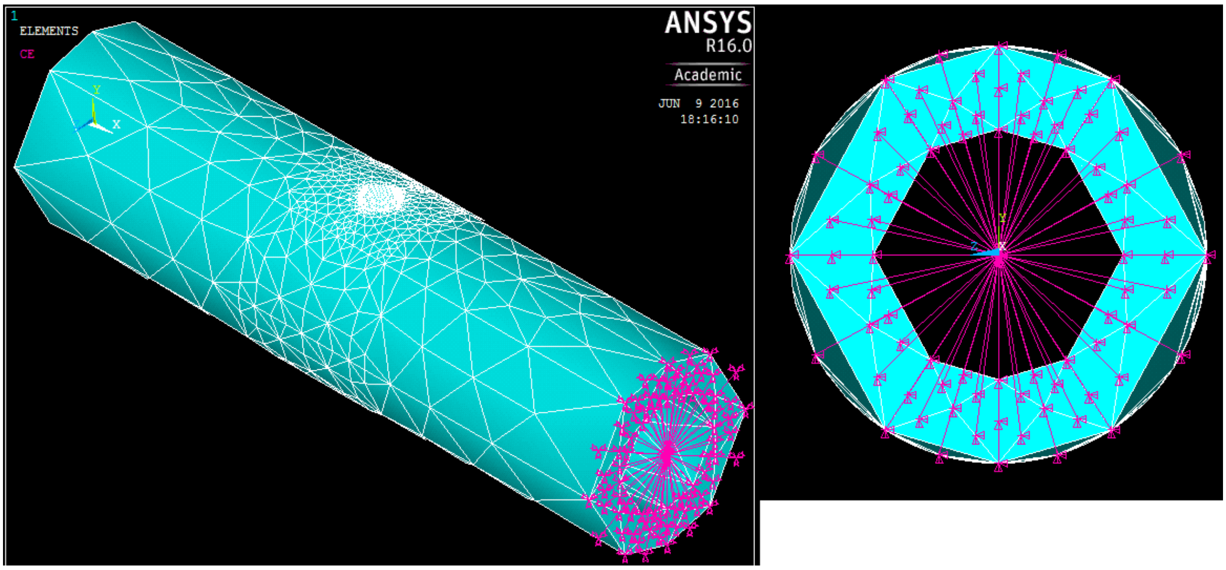


Illustration 61. Rigid region

All that is left to do is apply loads on the component. As mentioned in this project, Peterson`s Stress Concentration Factor 3rd edition sets three different load cases, being an axial load, bending momentum and a torsional force.

Referenced to what has been mentioned previously, here one can see the interaction between master node (node number 1) and the rest of nodes on that circular section plane. Although the geometry of the elements don`t adopt the circular section, when nodes are plotted, one can observe the mid side nodes of these elements and prove that the circular shape is indeed accomplished. The element plot provided by ANSYS only shows vertex nodes and can create confusion over the geometrical model.

Instead of introducing an axial force and solving, then introducing the bending force and solving and finally setting a torsional force and solving, this process is going to be optimized. There is a more efficient way of doing this and it by means of load cases (LS Files). These forces are applied on a keypoint (number 2). Solving via load cases means defining a force and saving it inside ANSYS`s memory, needing to define a number for each load case. The axial force will be load case number 1, bending force is load case 2 and torsional force will be load case 3. This procedure is much quicker and efficient, solving wise.

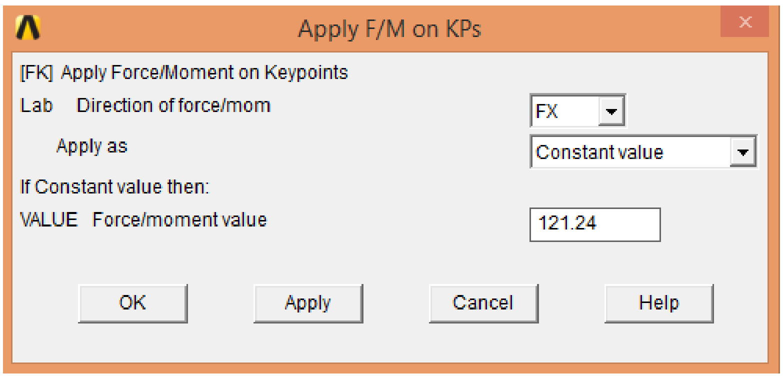


Illustration 62. Applying axial force

To do this in ANSYS, enter ***Preprocessor***, ***Loads, Define Loads, Apply, Structural***. Now this time select ***Force / Moment.*** All forces and moments are desired to be applied on keypoint number two because previously it has been meshed with a mass element, allowing rotations and inertias. So press ***On Keypoints***. When this is done, another selecting tool merges asking on which keypoint to apply the force. Pick or type in keypoint number 2 and press ***OK*** to continue. Another window shall flash up asking for the value of the force. Now, any value of force could be valid so a reasonable number would be 100 but, as when setting the length of the pipe, if one introduces a value of 100 N and later on wants to change that value inside the macro then it could be a nightmare trying to seek for that number, when there could be many 100`s inside that code. So, in order to make this number shout out a bit more if ever wanted to pin point, the value of the force is set at a random number (near 100) but easier to identify. See illustration 64.

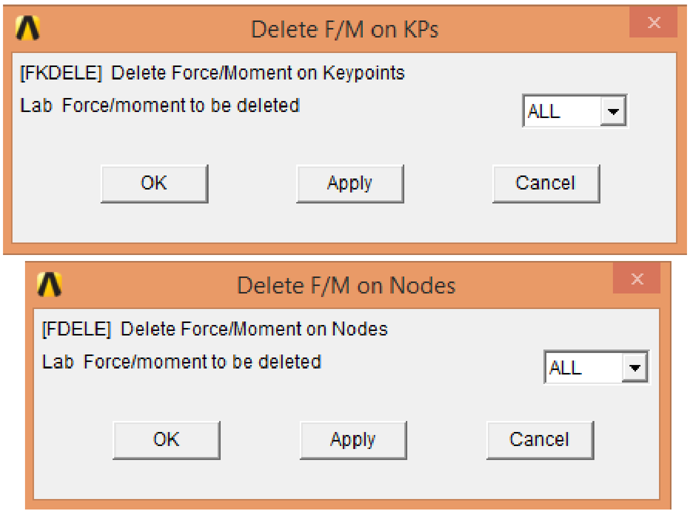


Illustration 63. Deleting forces on keypoints and nodes

Once this load is established, the user must tell ANSYS to save this configuration. This is done by going into ***Preprocessor, Loads, Load Step Opts***. Click the ***Write LS file*** in order to save a new file into ANSYS`s memory. A number is needed so as said before, enter number 1 for axial force. After this, a new file is necessary but before applying any loads, user must delete all previous ones. In this project it`s said to delete all forces on keypoints and on nodes, just to make sure. This is done by entering ***Preprocessor***, ***Loads, Define Loads, Delete, Structural, Force/Moment, On Keypoints*** and then ***On Nodes.*** When deleting, on keypoints, select keypoint number 2 and when deleting on nodes, select node number 1.

	LS file	Load	Direction
Axial Force	1	121.24	FX
Bending Moment	2	-121.24	MZ
Torsional Moment	3	121.24	MX

Illustration 64. Values for each load case

Once these operations are completed, user is in conditions of creating a new LS file, with a new load. The process is exactly the same as before when applying the axial force but to complete the three cases in hand, LS file number 2 will represent the bending moment and the moment applied on keypoint 2 will be -121.24 selecting MZ. The minus sign is due to the way the axis is set and in which direction this moment is applied. When applied this moment, save LS file in ***Preprocessor, Loads, Load Step Opts***. Click the ***Write LS file*** with the number 2. Delete forces and moments on keypoint 2 and node number 1 and apply the last moment on keypoint 2, which will be a MX with the same value as before, 121.24. Finally, create another LS file numbered 3 and delete all forces and moments on keypoints and nodes. Illustration 64 summarises all this information. This is the final step for boundary and load conditions. The next step is to solve the conditions imposed but before it`s recommended to ***Select Everything***.

5.6. SOLUTION

This is the easiest step to do. It consists in telling ANSYS to solve the problem that user has gave him and wait for ANSYS to do it. It`s a simple step but if any operation in pre-processor is incorrect, then the results will be invalid. Instead of solving a current LS, in this problem,

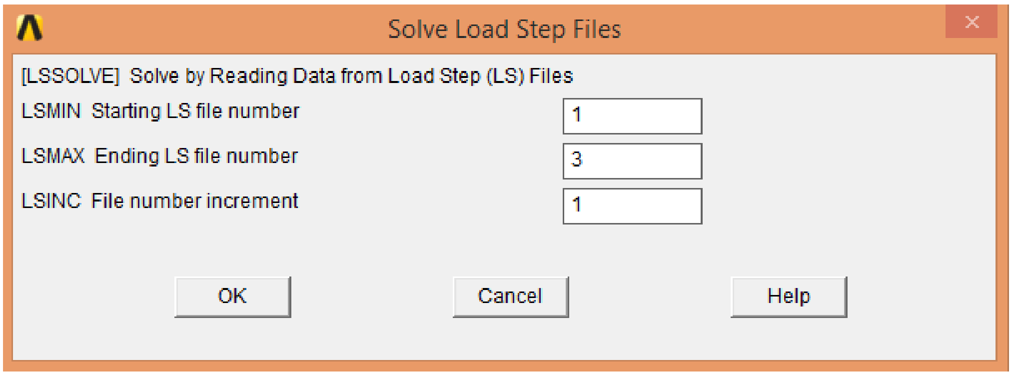


Illustration 65. Solving LS files

there`s three different cases to solve so the way to do it is to exit ***Preprocessor*** and enter ***Solution.*** Pick ***From LS Files*** and now user needs to tell the software which cases to solve. Indicate to start at file number 1, end at file 3 (go through all files) and solve them one by one, which is the file number increment. Press ***OK*** and wait for ANSYS to calculate. Depending on the number of nodes and other factors, it will take more or less time.

6. RESULT PROCESSING

The first thing to be done is to obtain the results and this is considered as a post processing task, so user must enter the ***General Post Processor.*** Once here, ANSYS needs to know which case is which and number them, so user must enter the following commands: ***LCDEF,1,1*** which tells ANSYS that case number 1 is for the axial force, ***LCDEF,2,2*** is for bending moment and ***LCDEF,3,3*** is for torsional moment.

Now the user can load any case wanted, for example, if the axial force case is of interest, type in the command ***LCASE,1*** and all information for this case will be available. The same is applied for ***LCASE,2*** for the bending moment and ***LCASE,3*** for torsional moment. Mention that when typed these commands, user must be inside the general post processor. Now is the time for the user to check the percentual error that was mentioned previously when establishing the geometry and mesh. The volumes that should be chosen for the PRERR command evaluation are the ones shown in figure 74. For PRERR command to be executed, one must be in the post processor and turn off the POWERGRAPHICS option. The results of the PRERR command for D = 0.75, d/D = 0.2 and di/D = 0.6 are shown in illustration 66.

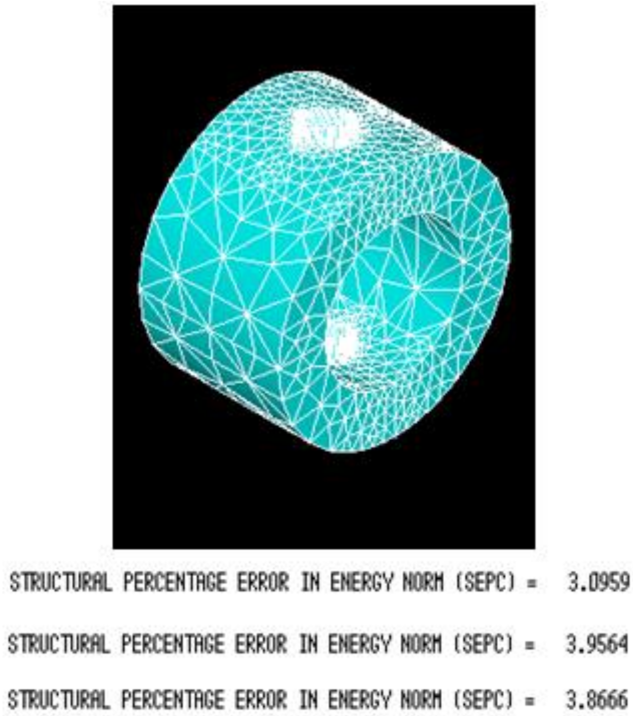


Illustration 66. PRERR results

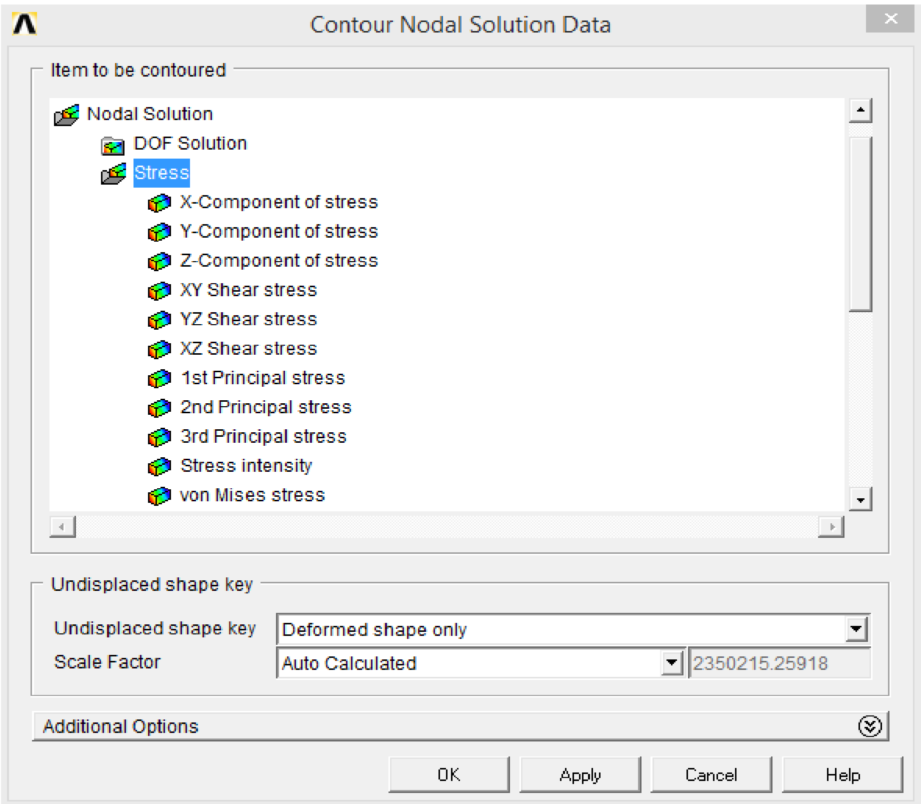


Illustration 67. Nodal solution window

After solving this case, one must recall why for this process and what is the objective. Using ANSYS as a calculus assistant was only so that user could obtain the maximum stress, necessary for calculating Kt. So, the way to plot the maximum stress in this case is to enter ***General Post Processor, Plot Results, Contour Plot, Nodal Solution, Stress.*** The window that should appear is shown in illustration 67.

There are many stresses represented as an option but at the beginning of this report, three criterions were explained and those are the one wanted, but ANSYS has changed the name of them from how they were defined.

Rankine`s criterion is ***1st Principle Stress***, Tresca`s criterion is called as ***Stress Intensity*** and obviously, Von Misses`s criterion is called ***von Mises Stress.*** Choose one of them, for example Rankine`s criterion and press ***OK***. The component will embrace some colour code indicating the maximum and minimum stress and where it`s located. It is important to obtain a visual image of the results because not only the maximum stress is desired but also, where it is found. Imagine that the maximum stress was held at the embedment (for example), that would mean an incorrect result for this analysis and designer would have to discriminate this result and choose stress found at the stress raiser.

Spot out that for each analysis, being this each combination of D, d/D and di/D, there are three different load cases, each load case containing three different results (one for each criteria). This means that for each analysis, 9 are the results (photos) created. After x amount of analysis, a big data base will be stored, which will require post processing (manual) work.

The results obtained for case 1, followed throughout all this project, are shown in the following figures.

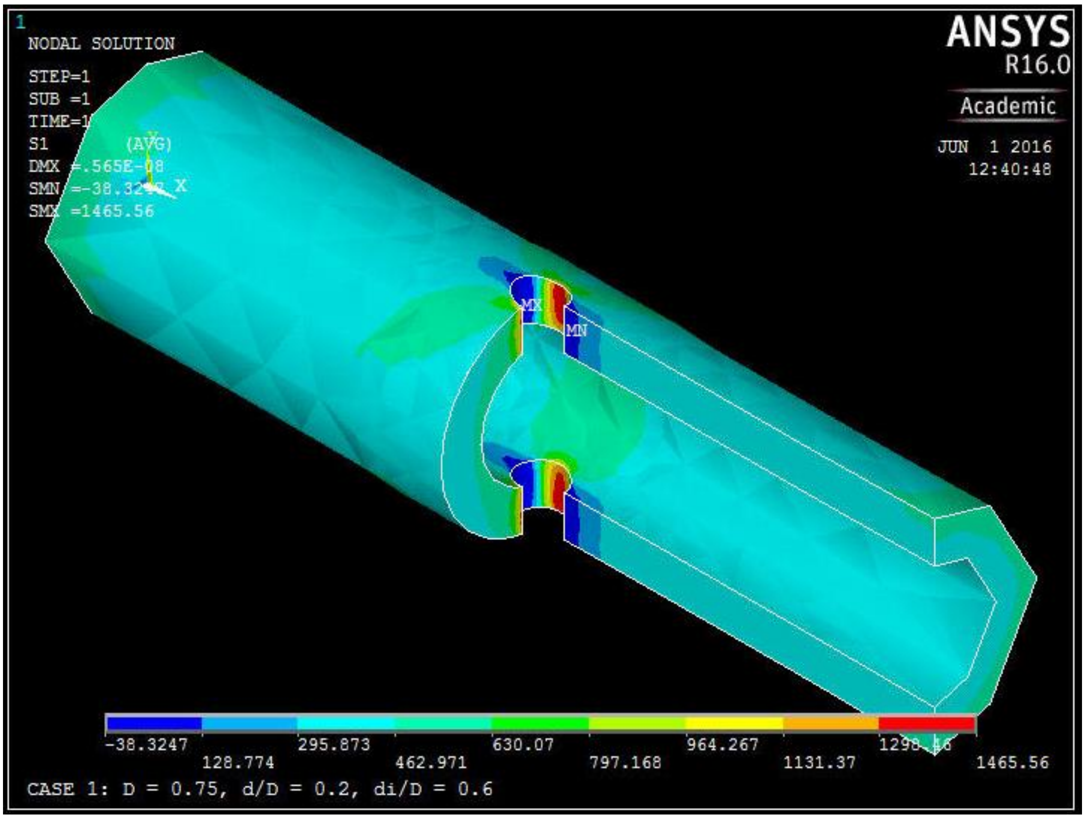


Illustration 68. Axial Force. 1st Principle Stress

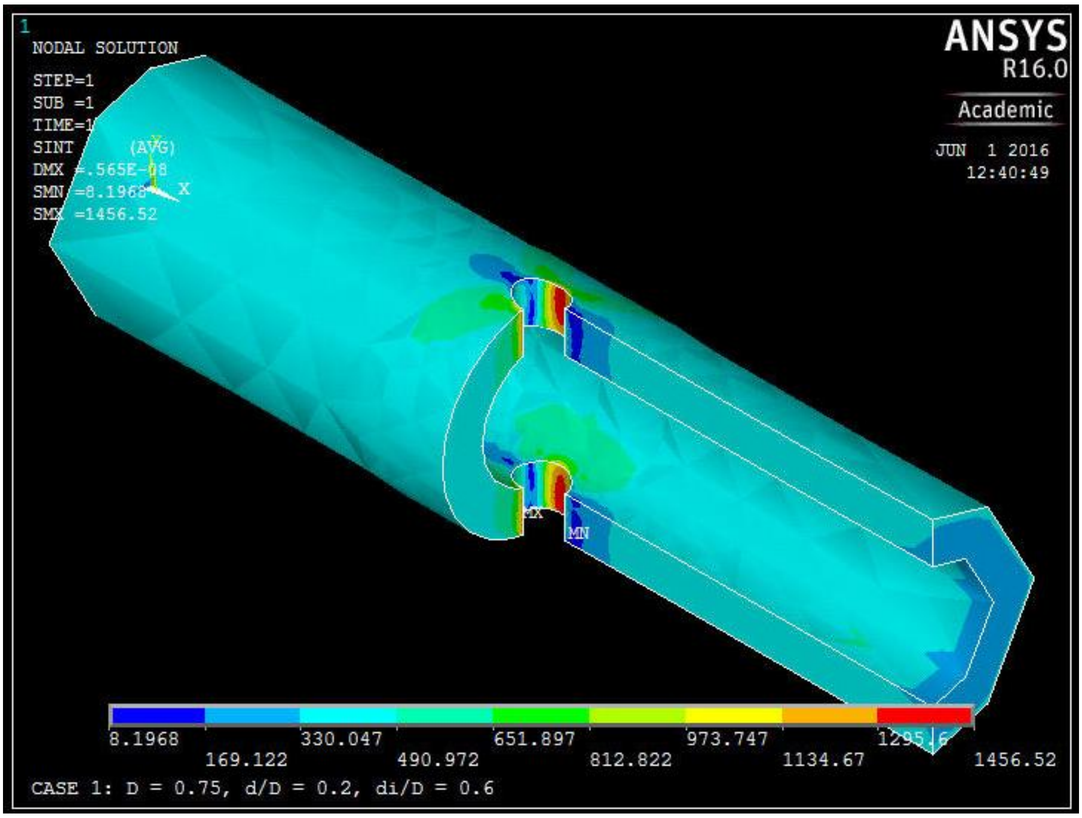


Illustration 69. Axial Force. Stress Intensity

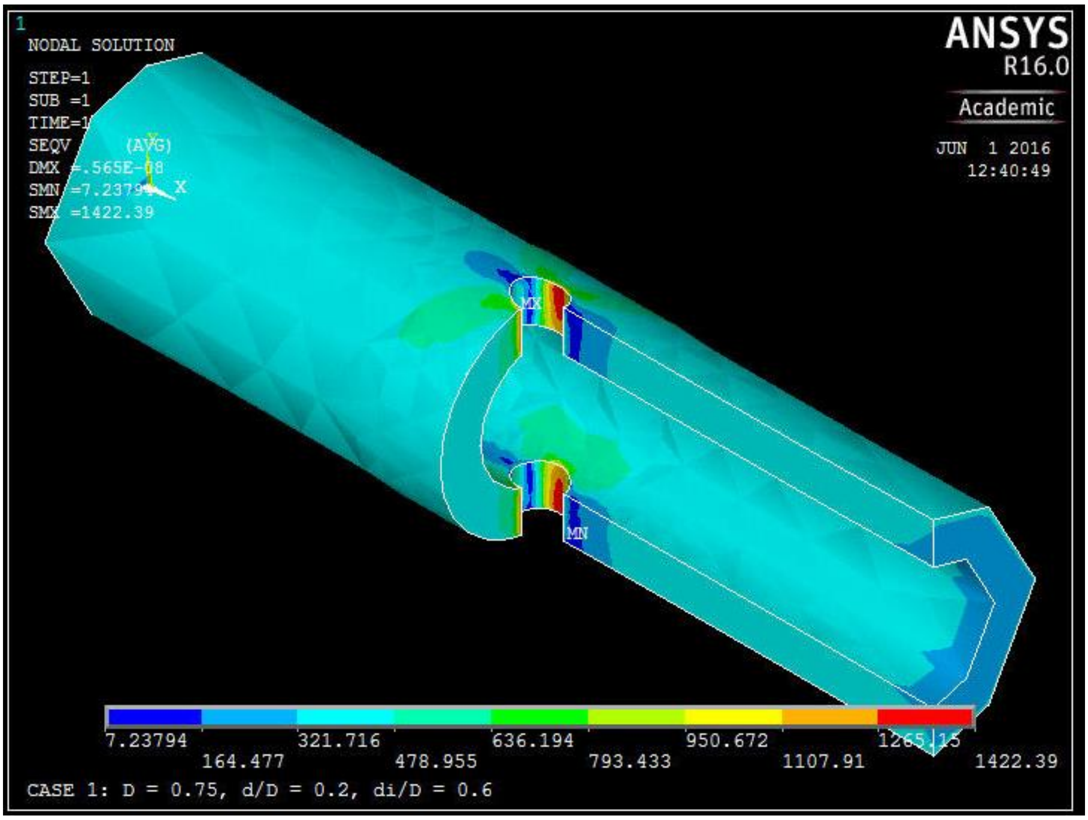


Illustration 70. Axial Force. Von Misses Stress

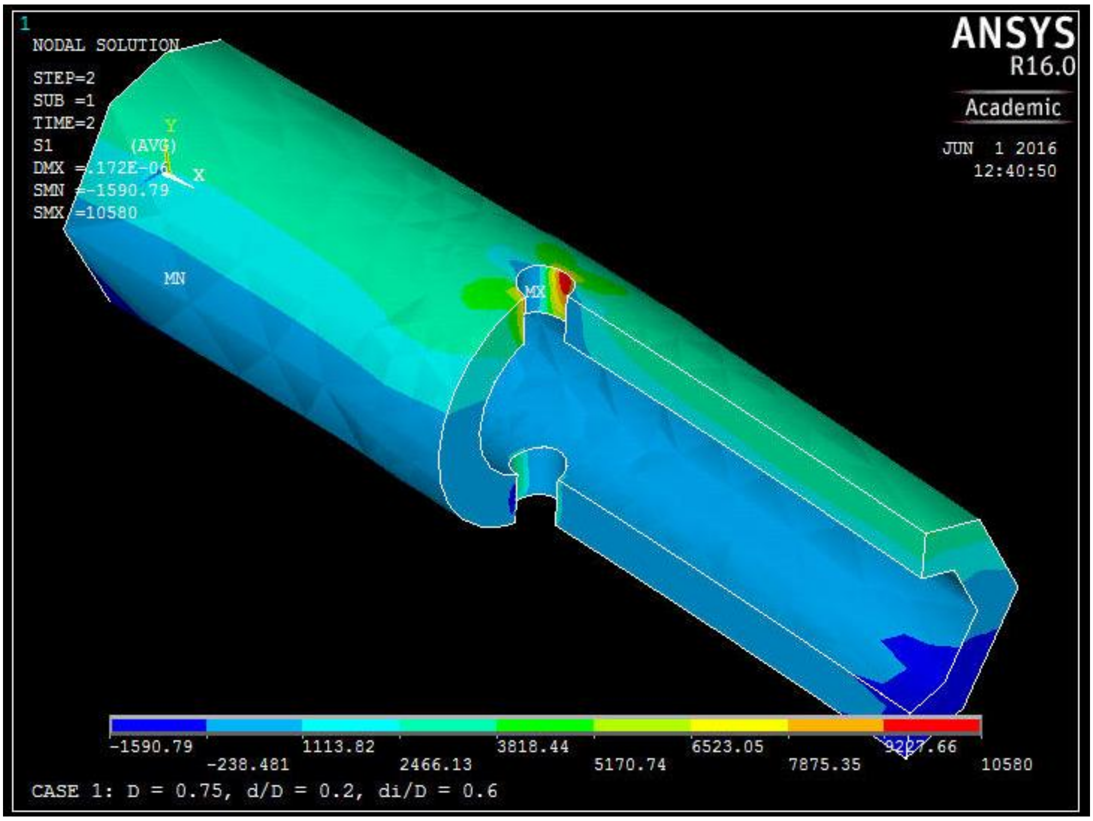


Illustration 71. Bending Moment. 1st Principle Stress

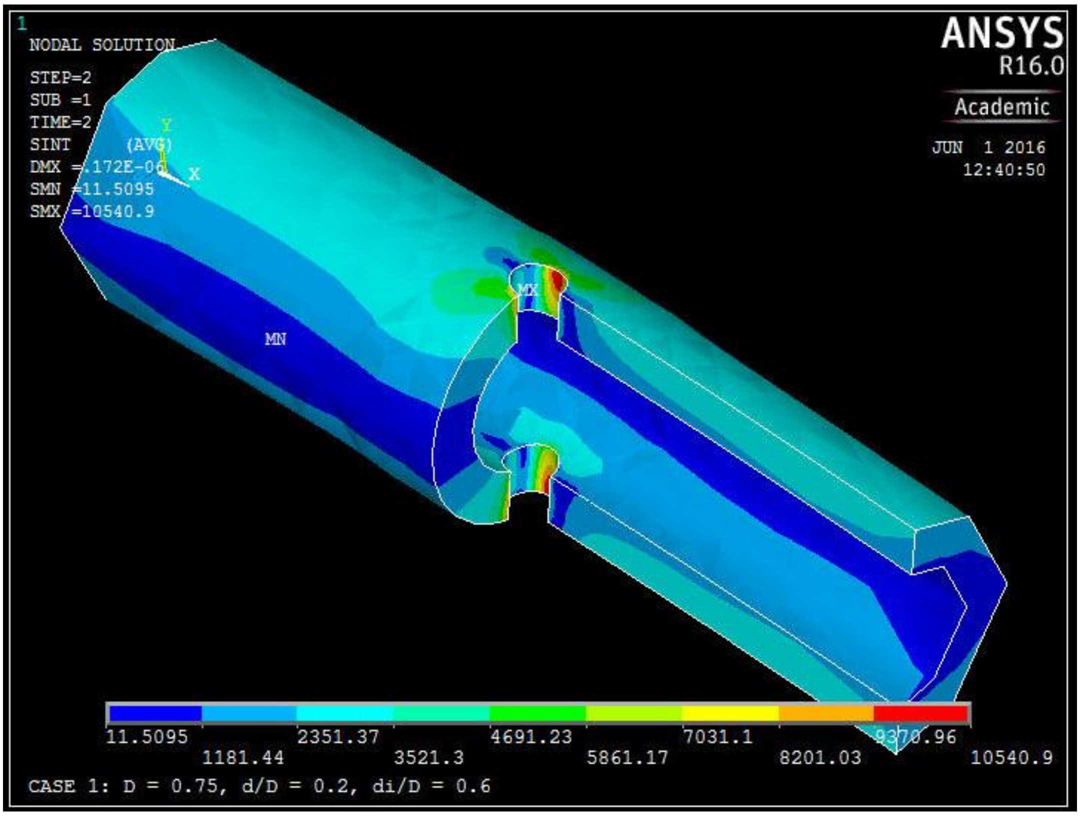


Illustration 72. Bending Moment. Stress Intensity

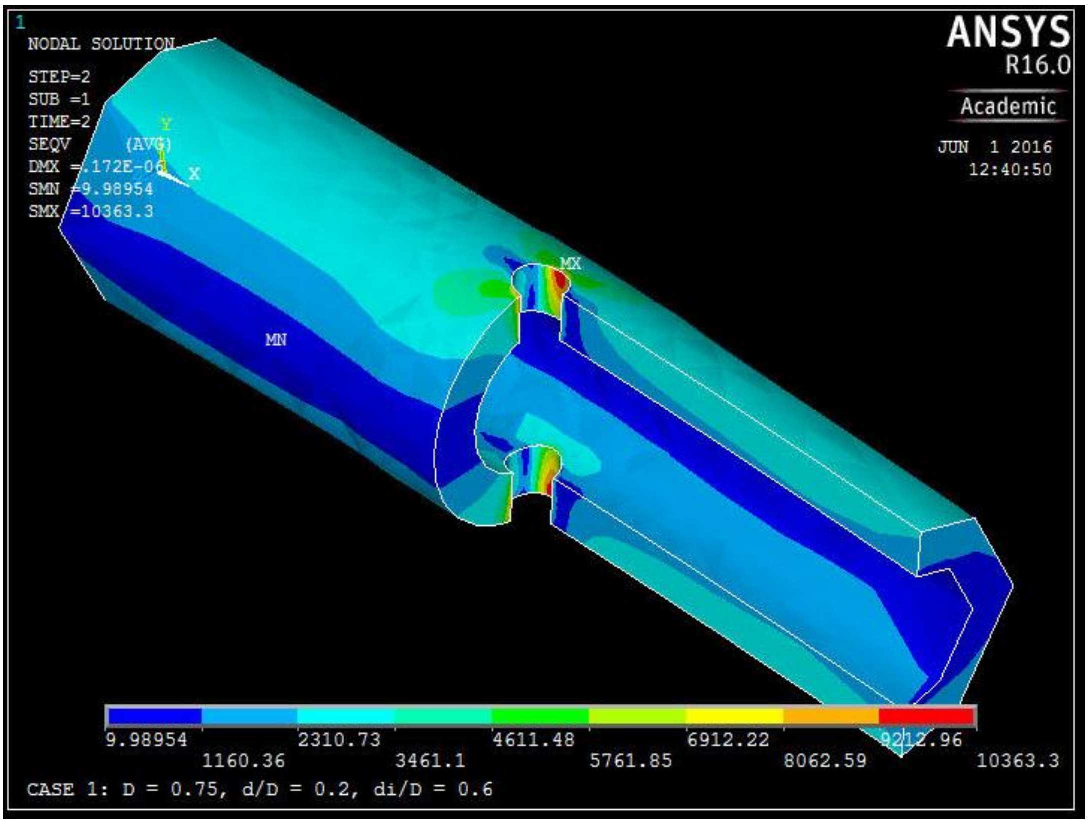


Illustration 73. Bending Moment. Von Misses Stress

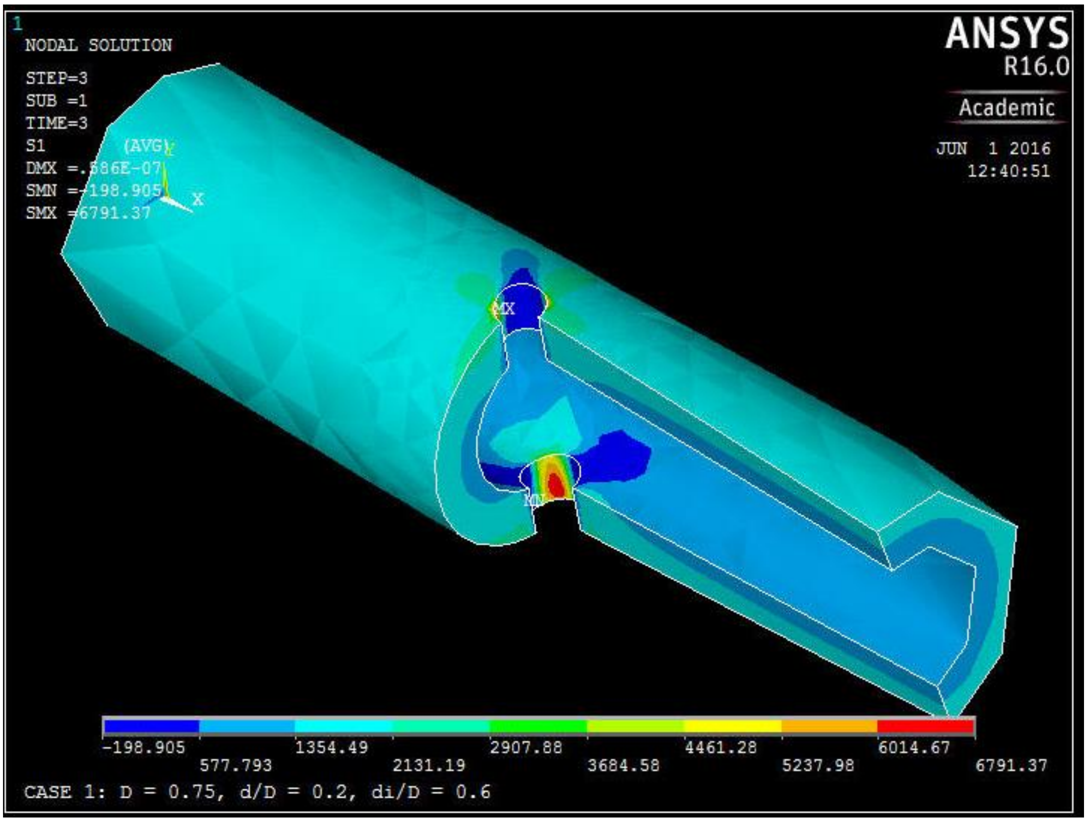


Illustration 74. Torsional Moment. 1st Principle Stress

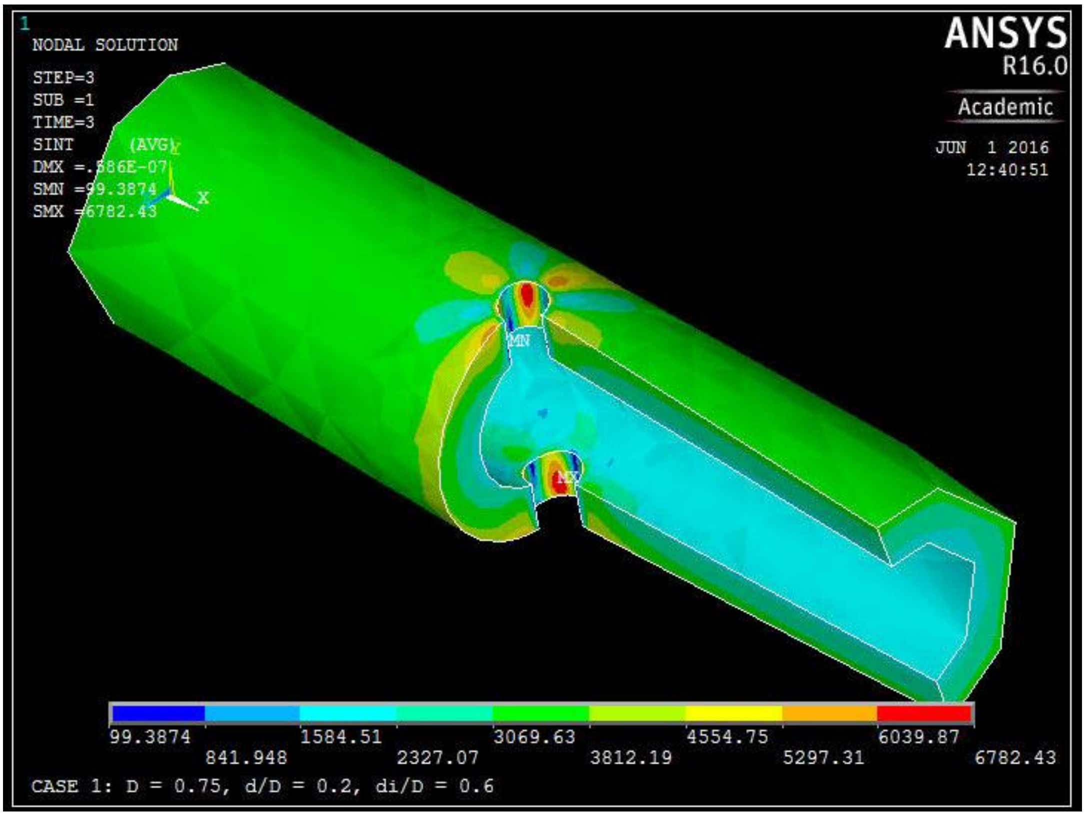


Illustration 75. Torsional Moment. Stress Intensity

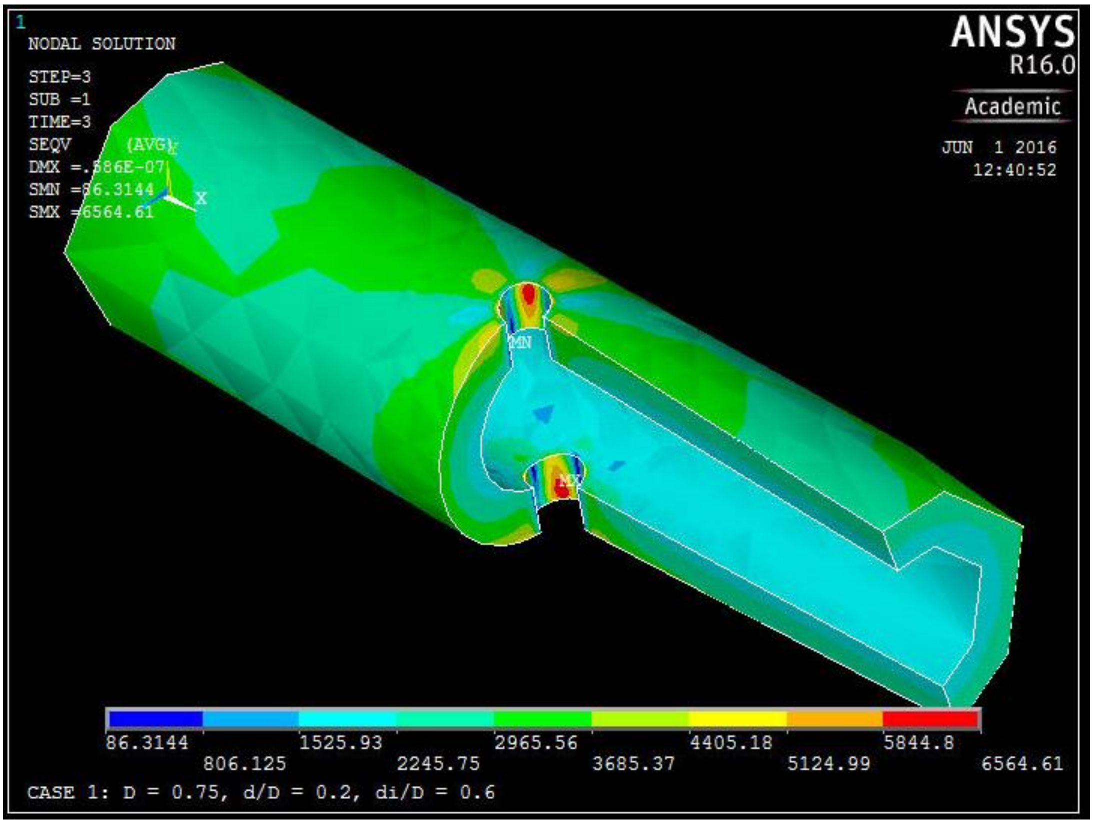


Illustration 76. Torsional Moment. Von Misses Stress

7. MACRO CREATION

The objective, don`t forget, is to obtain a graphic and an equation function of D, d/D and di/D so that for any combination, Kt can be easily obtained.

This is the process to be done for each analysis. Now, the more analyses launched, more precise will be the final result but it`s not viable doing these steps for every combination of D, d/D and di/D. It would take too much time, increasing the probability of committing an error, so this project proposes an alternative, automatized solution in order to obtain as many results and analysis as wanted.

When user does any action in ANSYS, the software stores the information in a file, so that there`s control and a tracking program of the process been done. Everything that user enters manually or via commands is copied and saved in a .log file, in the working directory chosen at start up. ANSYS`s command bar has the property of reading and executing the commands written manually and this is one of the key factor for using macros. The next step is to retrieve that .log file, save it with the extension ***.mac***, open it with a text editor (like Notepad) and create a macro that satisfies all three types of cases mentioned.

Now, the macro designed for this present project contains many lines, being so that it would take up a large amount of space of this report, so it has been copied and attached to the CD that comes with this project and it`s also found as an addendum, part of this project.

Obviously, the entire macro cannot be dissected in this report, so only a few important aspects will be mentioned. The first being how the macro is going to be executed and how to introduce the parameters. This is done by *arguments* which allows user to enter the values that characterize the analysis. The three *arguments* used in this project are D, d/D and di/D and these are introduced in the macro as D = ARG1, DD = ARG2 and DID = ARG3. The way to execute the macro is to type in its name (*SolveTube*) followed by a coma and the three parameters mentioned before, all separated by comas, so for instance, to run the analysis for case 1, user must type in the command bar ***SolveTube,0.75,0.2,0.6.***

The next aspect to be mentioned is how the macro distinguishes the parameters so that it enters case 1, 2 or 3. This is done by logical functions built inside ANSYS but before, recall to illustration 17 to see how to differentiate cases 1, 2 and 3. Logical condition for entering case 1 is *IF,ALPHA,LE,60,THEN. This translates into: if alpha is less or equal than 60 degrees then carry on reading the next line, which enters case 1. If this condition is not satisfied, then ANSYS will jump to the next logical condition, which happens to be *ELSEIF,DD,LE,0.3,THEN. If d/D is less or equal than 0.3, then enter case 2. The rest of cases are solved by case 3, which can be entered simply by typing *ELSE.

The last aspect which is of most interest is obtaining results. If user runs 50 analyses in one go, it is impossible to go back and look at the stresses, so in order to extract information from tests, here is shown how to save an image. The command for this chore is: /UI,COPY,SAVE,JPEG,FULL,COLOR,NORM,PORTRAIT,NO,100. This command saves an image of the plotted stress in a .jpg file in the working directory. The problem now is that if 50 analyses are done, there will be 50*9 = 450 images with three different criterions. This can be an issue post processing so in this project the images have been renamed and saved in new, separate folders. Previously, the folders where images are saved need to be created in working directory.

/RENAME,SolveTube000,jpg,,C:\Users\BenJordan\Documents\University\TFG\ANSYS\Results\AxialForce\1stPrincipleStress/%D% - %DD% - %DID%,jpg.

This is the command that realizes the renaming and ordering process. First tell ANSYS the name of the file wanted to be renamed followed by its format (jpg). Then the new directory where the image is sent is needed, followed by its new name, which in this report has been set as the three parameters of the analysis. As it can be seen in the macro, just changing the directory is enough to save the image in a different folder. This command is for the axial load and saves the image in a folder called *1st Principle Stress,* inside another one called *Axial Load* (all inside the working directory).

In this project, in order to achieve a reasonably good data base about this component, 64 analyses were executed, leading to 576 images. The parameters for all analyses will be a combination of the fundamental parameters. Parameter d/D will adopt the following values: 0.1, 0.15, 0.2, 0.25, 0.3, 0.35, 0.4, 0.45, 0.5, 0.55, 0.6, 0.65, 0.7, 0.75, 0.8. Parameter di/D will take the next values: 0.2, 0.3, 0.4, 0.5, 0.6, 0.7, 0.8, 0.9 and finally D will be constant adopting a value of 0.75.

8. RESULT ANALYSIS

After running all 64 analyses comes the post processing step. At this point user will have all the information needed from ANSYS, obtaining, ultimately, σ_{max}. In order to obtain Kt, the other stress needed is the nominal stress, which mentioned at the beginning of the report, is chosen as the gross nominal stress. The formulation to calculate this stress has been mentioned in point 3.3.

For the example followed in this report, D = 0.75, d/D = 0.2, di/D = 0.6, nominal stress is calculated as shown in the following figure.

D	d/D	di/D
0.75	0.2	0.6

	Formulation	Load (N)	Gross Stress (Pa)
Axial Force	$\frac{P}{\frac{\pi}{4}(D^2 - di^2)}$	121.24	428.799

	Formulation	Load (Nm)	Gross Stress (Pa)
Bending Moment	$\frac{32\,M\,D}{\pi\,(D^4 - di^4)}$	121.24	3363.128

	Formulation	Load (Nm)	Gross Stress (Pa)
Torsional Moment	$\frac{16\,T\,D}{\pi\,(D^4 - di^4)}$	121.24	1681.564

Illustration 77. Calculating nominal gross stress

Apply the same method and formulation for all possible combinations and the nominal gross stress is easily obtained via an Excel sheet. So, the final step is to obtain Kt for each load and for each criterion, dividing the maximum stress form the finite element method by the nominal gross stress, calculated recently. Due to the vast amount of information (574 Kt values), all of them are not going to be shown in this report, but only some few examples to show the methodology and the rest are the same. All results will be incorporated in the CD coming with this and at the end of this project as an addendum.

To enquire information about any load case and criterion, the addendum is available at the end of this project.

When all Kt values have been calculated, a bunch of numbers isn`t practical to manage and even less to compare with Peterson`s graphs, so the next step is to plot, in independent graphics, Kt values and see the evolution and verify the results. This is done with the help of an Excel Worksheet.

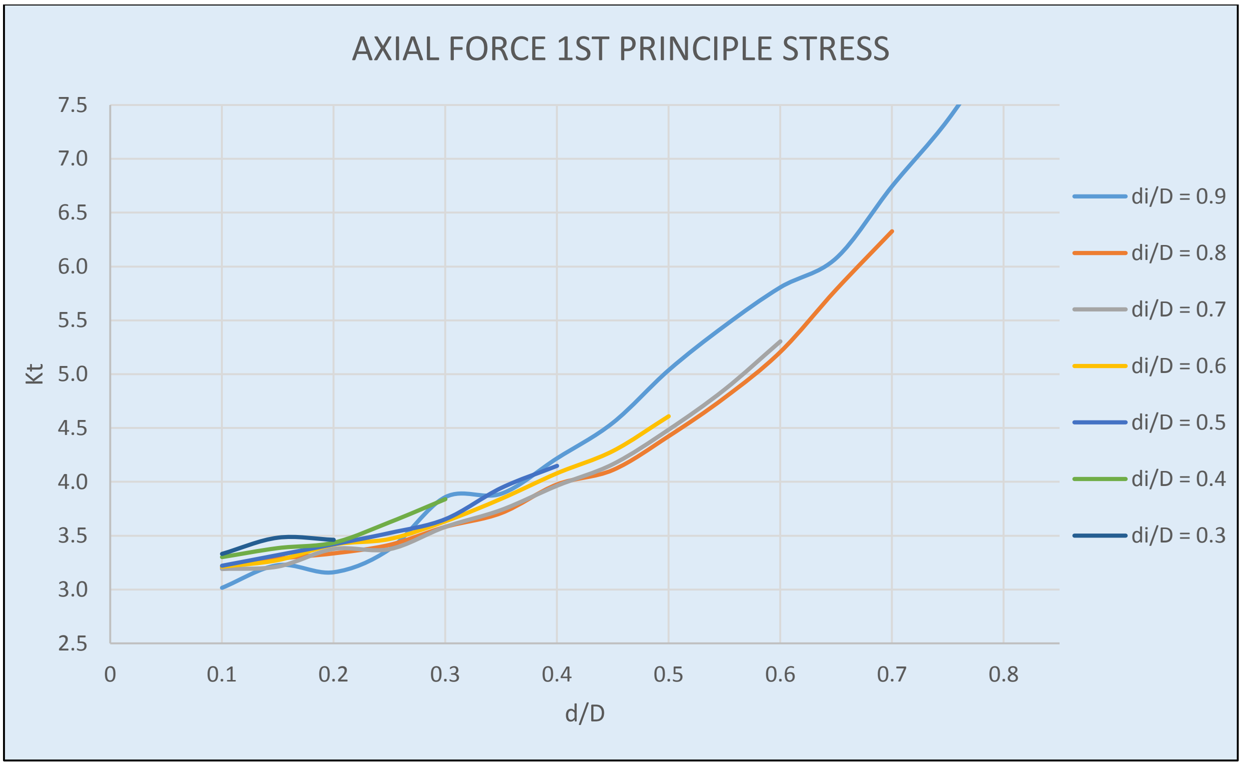


Illustration 78. Axial Force 1st Principle stress chart

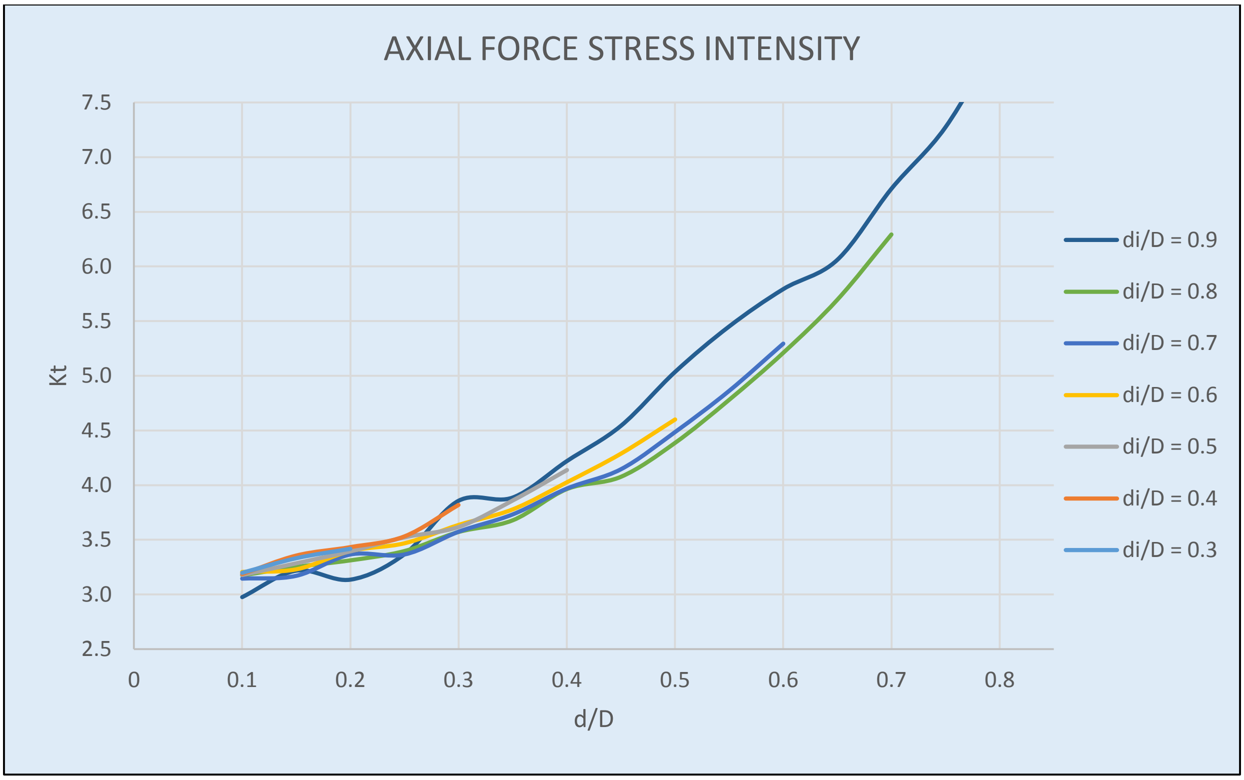


Illustration 79. Axial Force Stress Intensity chart

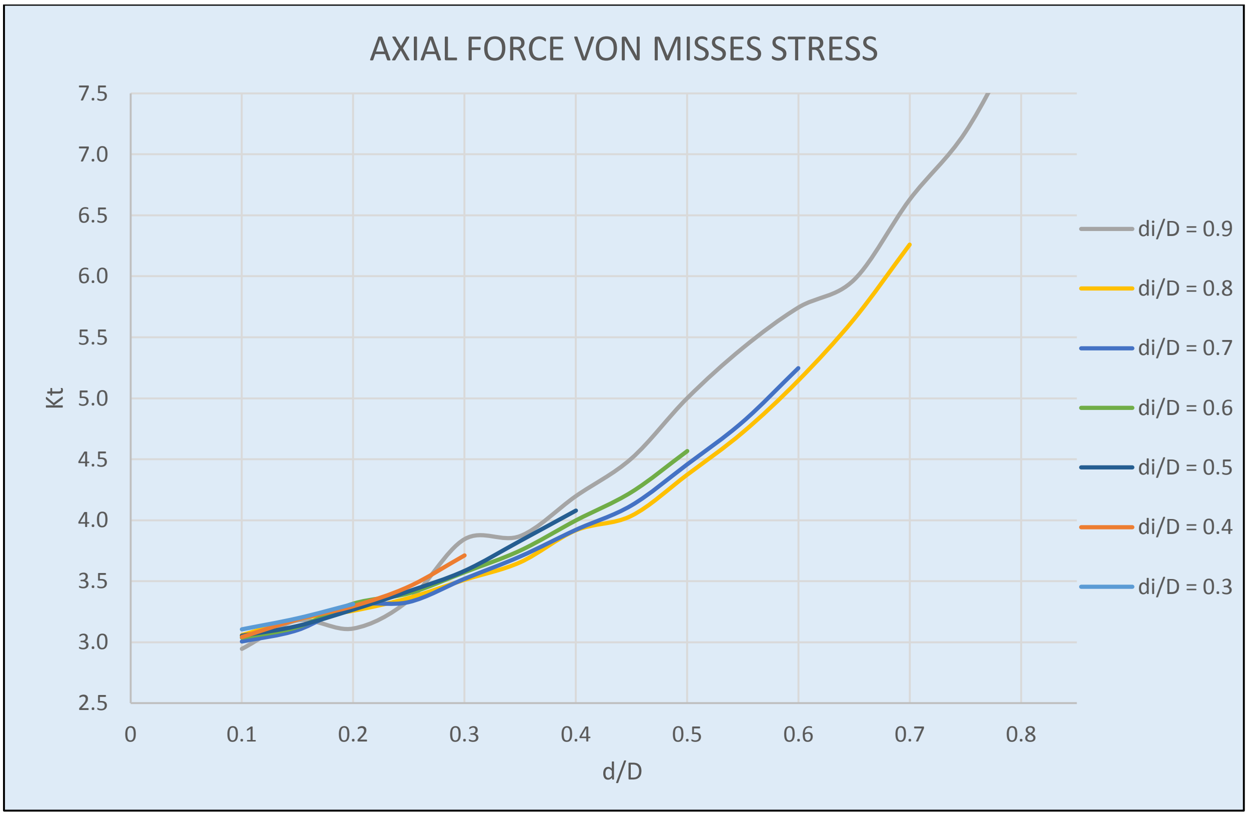


Illustration 80. Axial Force Von Misses chart

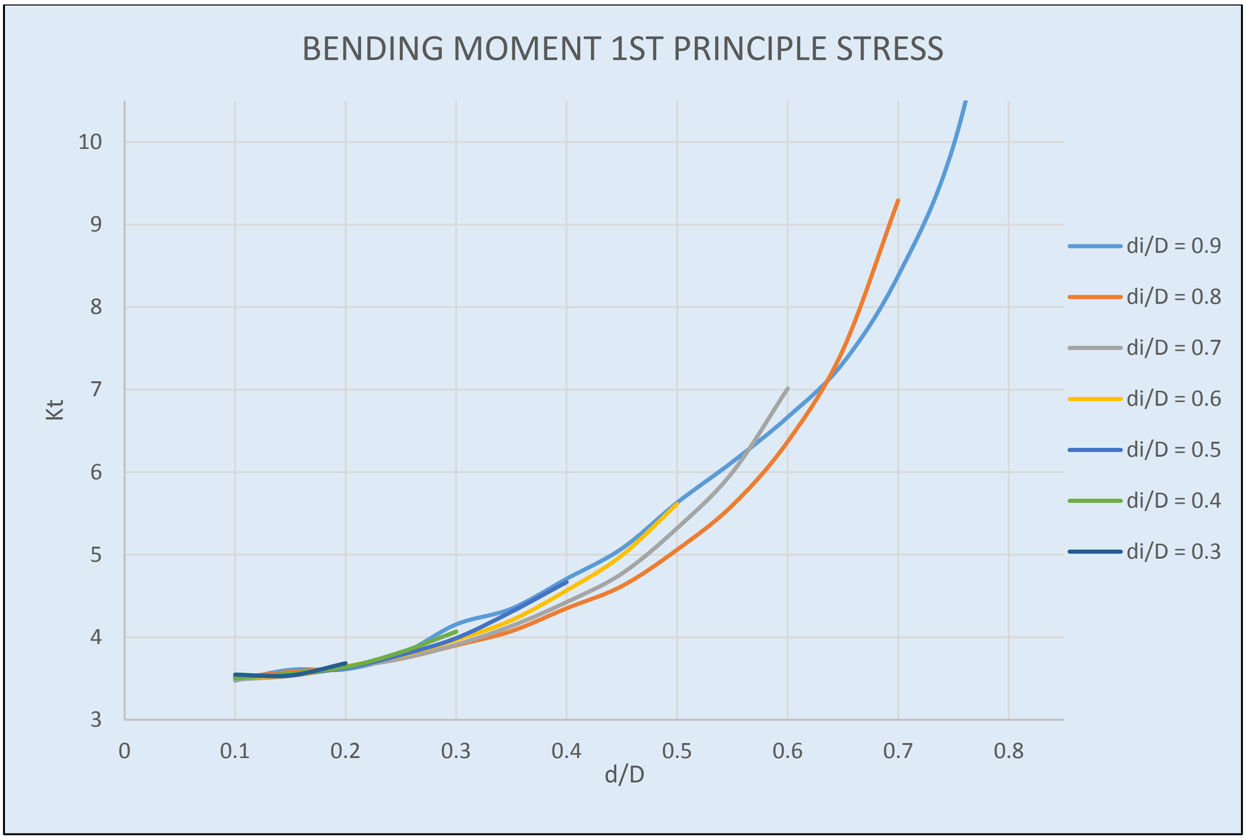


Illustration 81. Bending Moment 1st Principle stress chart

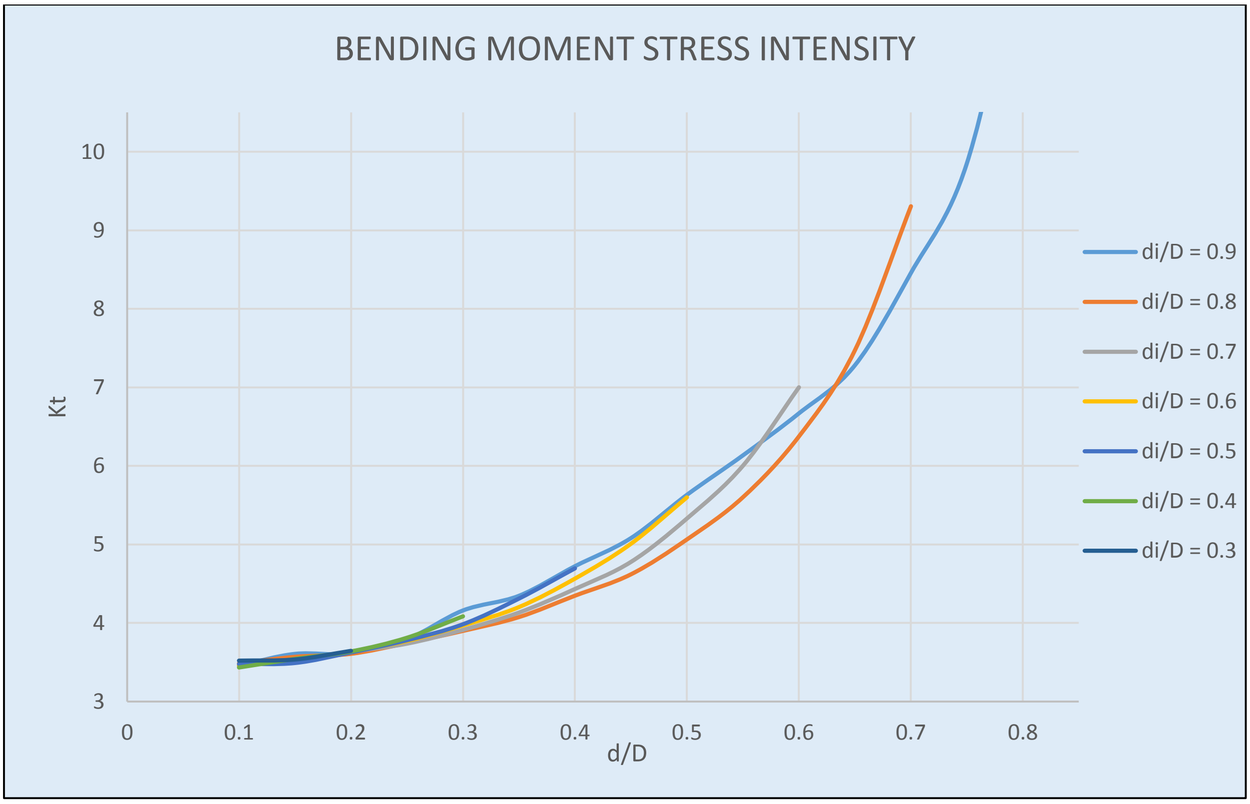


Illustration 82. Bending Moment Stress Intensity chart

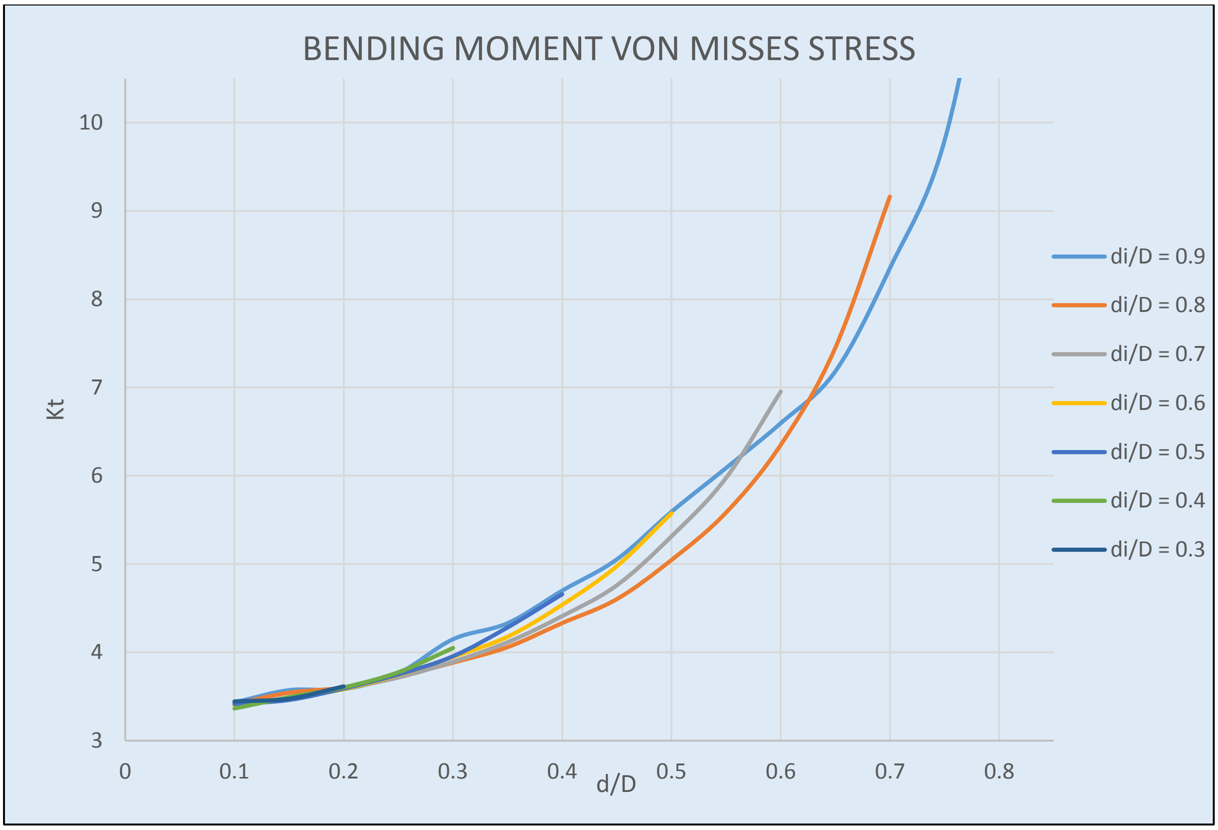


Illustration 83. Bending Moment Von Misses chart

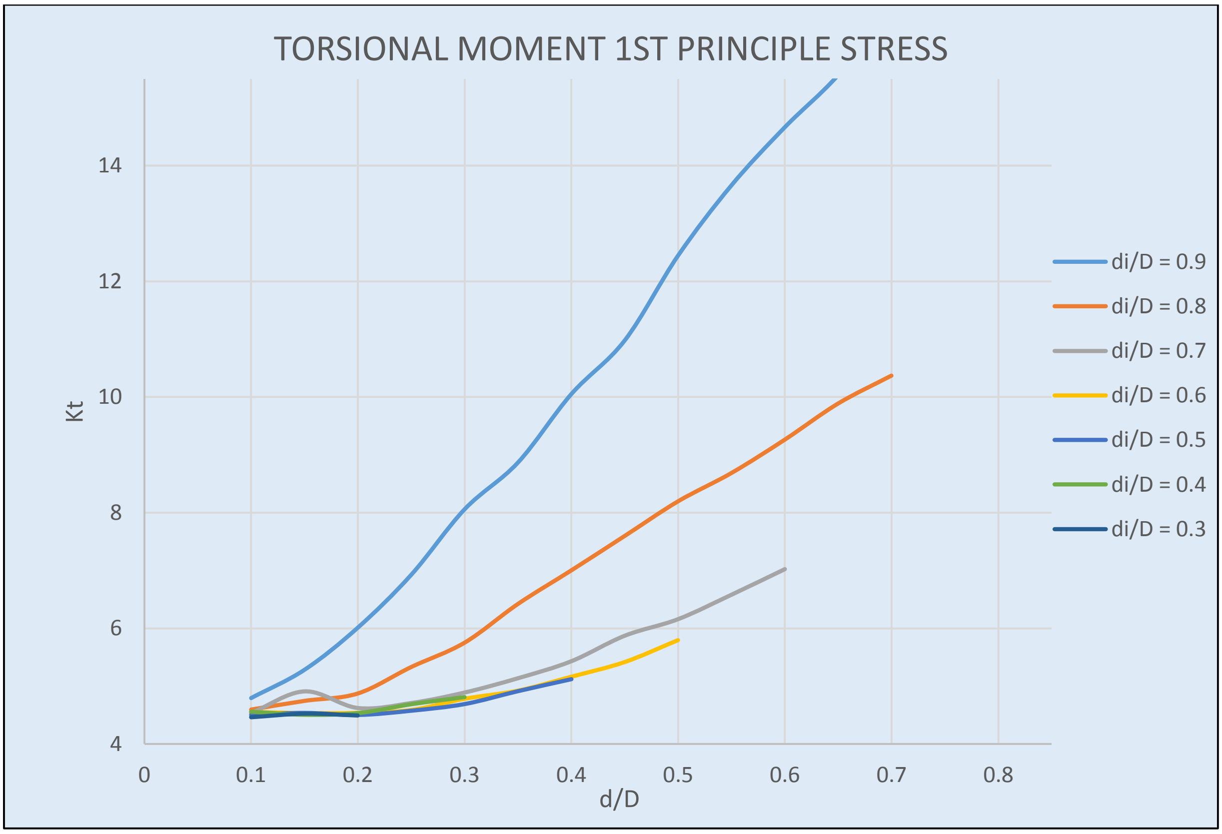


Illustration 84. Torsional Moment 1st Principle stress chart

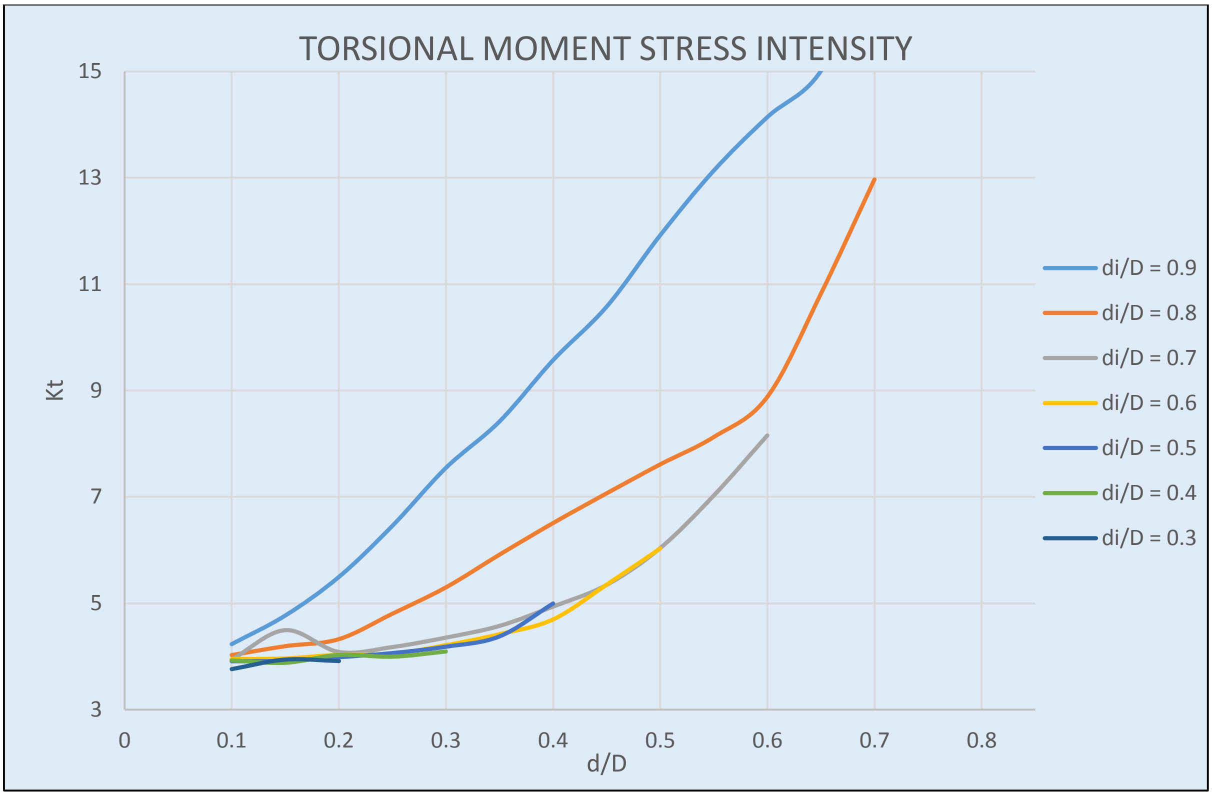


Illustration 85. Torsional Moment Stress Intensity chart

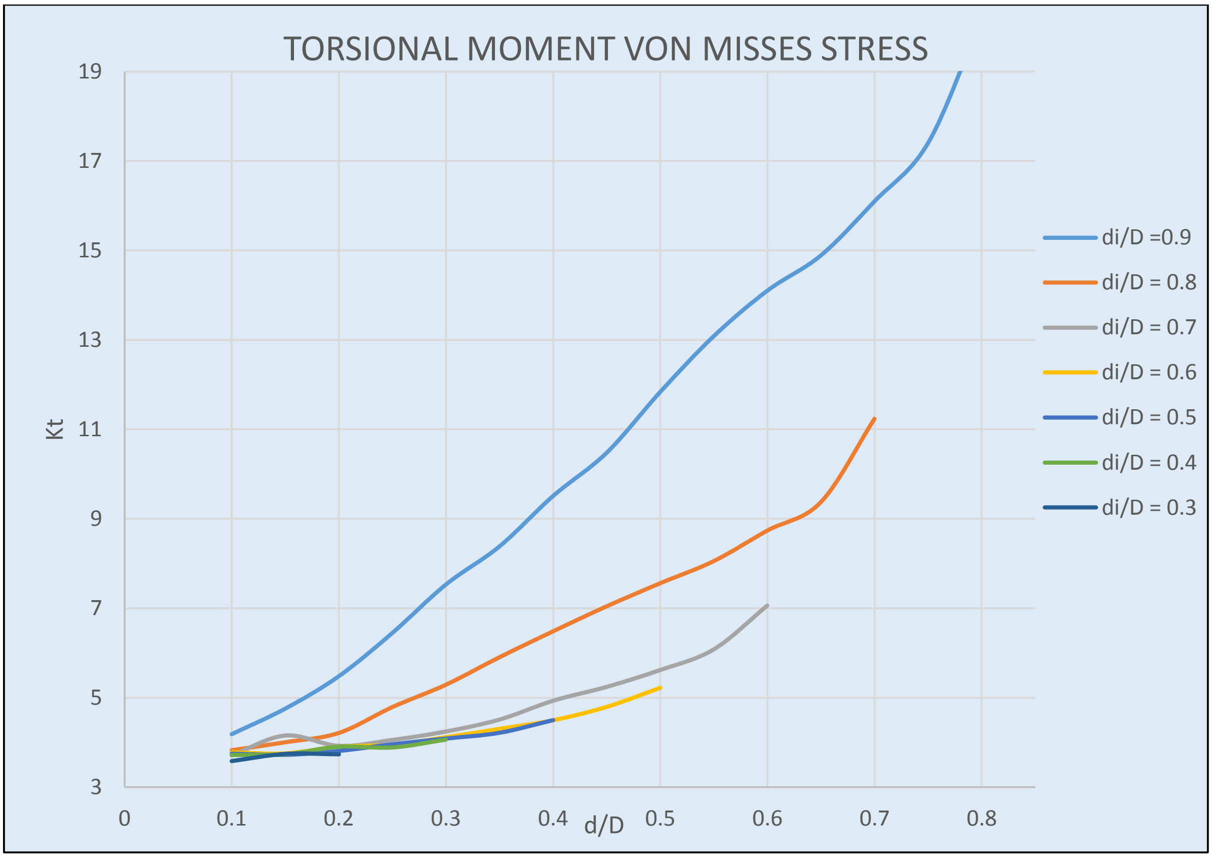


Illustration 86. Torsional Moment Von Misses chart

These are the graphics for all three load cases distinguishing three different criterions. In order to verify finite element method against Peterson`s Stress Concentration Factors or vice versa, both plots need to be overlapped and in addition, the most appropriate criterion can be selected from the three mentioned.

If one compares both graphics, it is seen that most of the curves for each di/D coincide quite well, having a good match, except for all curves where di/D = 0.9. All curves follow a pattern of growth in a certain direction, tending to an asymptote, but when di/D reaches a value of 0.9, the curve doesn`t grow in the same manner, also presenting some ups and downs, when for the rest of cases, this growth is gradual but constant. The next point of this report will focus on the study and understanding of this phenomenon.

8.1. FINITE ELEMENT METHOD AND PETERSON DISCREPANCY

The first aspect to pin point out is that the major differences occur when di/D = 0.9 and this means that discrepancies come when the hole is large (in fact, as large as it can get). In order to understand and interpret the results, the focus was turned to one single load case and choosing one criterion, because, if a reason/explanation or an error was found it could be applied for the rest of cases. In this report, the focus highlights the axial force according to 1st Principle Stress.

The differences are increased when d/D and di/D are high, so the attention was focused on the following cases:

D = 0.75, d/D = 0.65, di/D = 0.8
D = 0.75, d/D = 0.65, di/D = 0.85
D = 0.75, d/D = 0.65, di/D = 0.9
D = 0.75, d/D = 0.65, di/D = 0.95

The first analysis was launched again and the view was enhanced where the maximum stresses were hold, being at the stress raiser. By entering ***Post Processor, Query Results, Sub grid Solution*** and choosing ***Stress, 1st Principle Stress***, user can see stress values from any point of the component. It was spotted that the maximum stress was at the exterior layers of the pipe and that the stress was uniformly distributed throughout a plane perpendicular section. As a sprout of curiosity, the stress at the inside layers of the tube was measured, resulting, obviously, in a slight decrease, resulting in 3682.62. The results of this study are shown in the following figure.

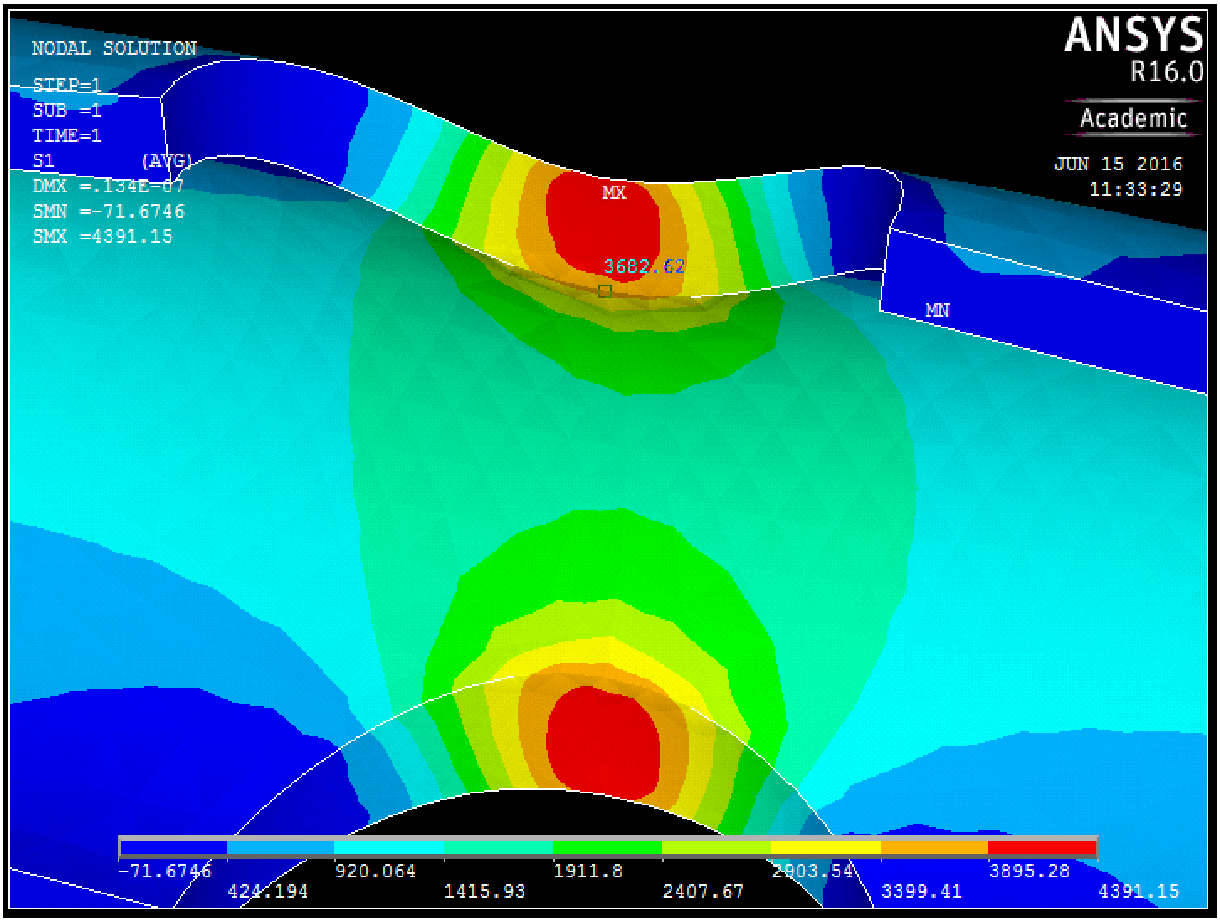


Illustration 87. Enhanced view of maximum stress location for D = 0.75, d/D = 0.65, di/D = 0.8

It was an interesting analysis to see that at the exterior layers lays the maximum stress but at the interior layers if the tube, at a slight distance inside, stress decreases a little, not being the maximum stress in a perpendicular cross section. To see this phenomenon's evolution, the next case was executed (D = 0.75, d/D = 0.65, di/D = 0.85). it was proven that the maximum stress is located at the exterior layers of the pipe, just the same as for the case where di/D was equal to 0.8. Again, the stress at the inside sections of the pipe were taken to see the difference, noticing the same results as the previous case, where the inside layer stress is slightly inferior to the maximum. The results aren`t shown because the image is very similar to the previous one, but the numerical values are: maximum stress at top layer is 5783 and at the inside layer, stress is 5335.7.

To keep track of this tendency, another analysis was done, corresponding to the parameters D = 0.75, d/D = 0.65, di/D = 0.9. In this case, the hole is the same but the tube gets thinner. Here is where an important discovery was made, an unexpected appreciation was noticed, which could seem useless but ultimately was a major discovery for the result analysis of this project. A zoomed image of this analysis is shown in the next figure.

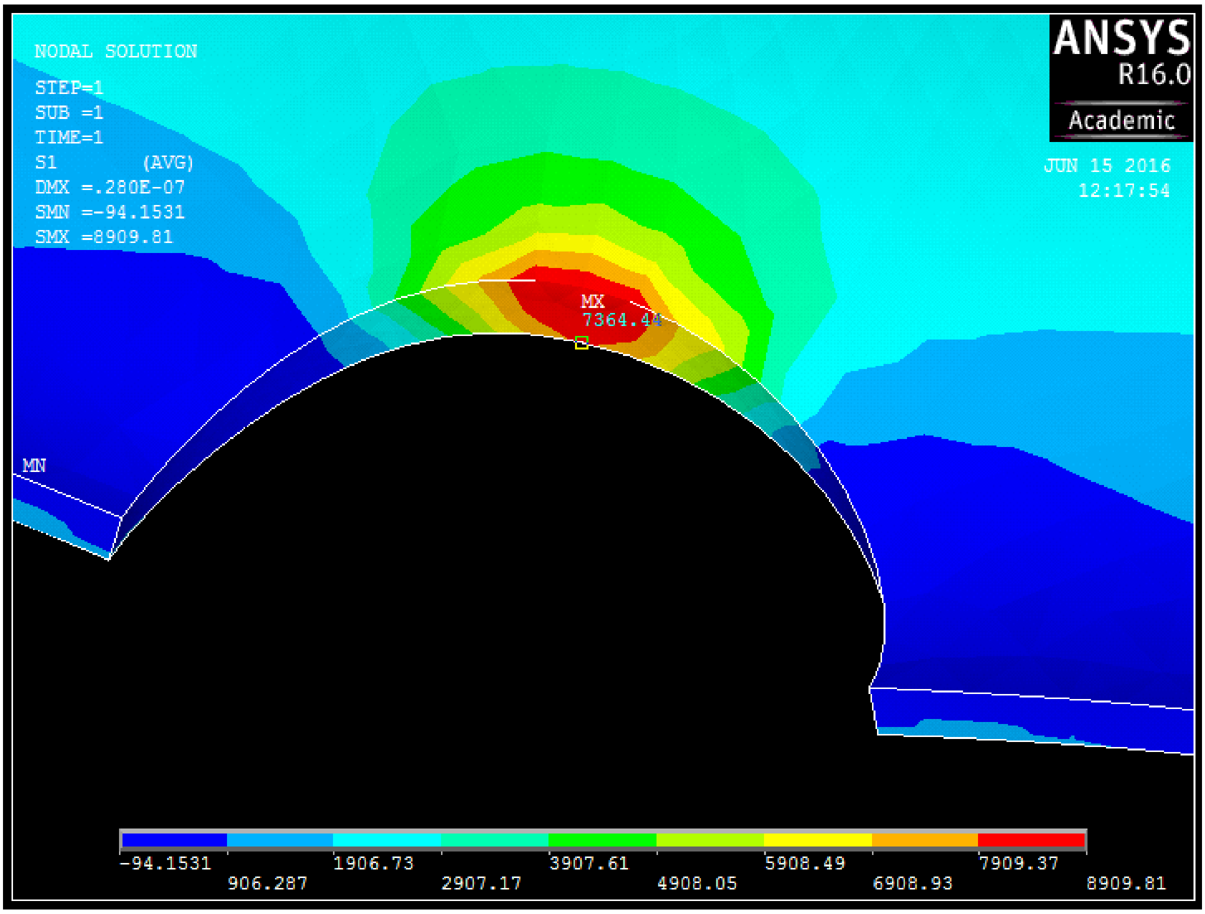


Illustration 88. Enhanced view of maximum stress location for D = 0.75, d/D = 0.65, di/D = 0.9

As it can be seen, now, the maximum stress area (coloured in red) is not a plane perpendicular section. Instead, this area enters inner layers of the inside of the pipe. But that is not the most shocking appreciation. The major difference between the previous cases and this one is that, now, the maximum stress is located in the inner section of the tube and not at the exterior layers. The maximum stress has changed in location. At the inner layers, the maximum stress is 8909.81 and amazingly, at the outside section, stress is 7364.44. To corroborate these results, another trial was executed to see if this phenomenon was a fluke or not. This time, the pipe would be even thinner, having all parameters the same value as before except for di/D which would now be 0.95 (extra thin pipe). Surprisingly or not, it was shown that the maximum stress also had changed its location from the exterior layers to the inside of pipe.

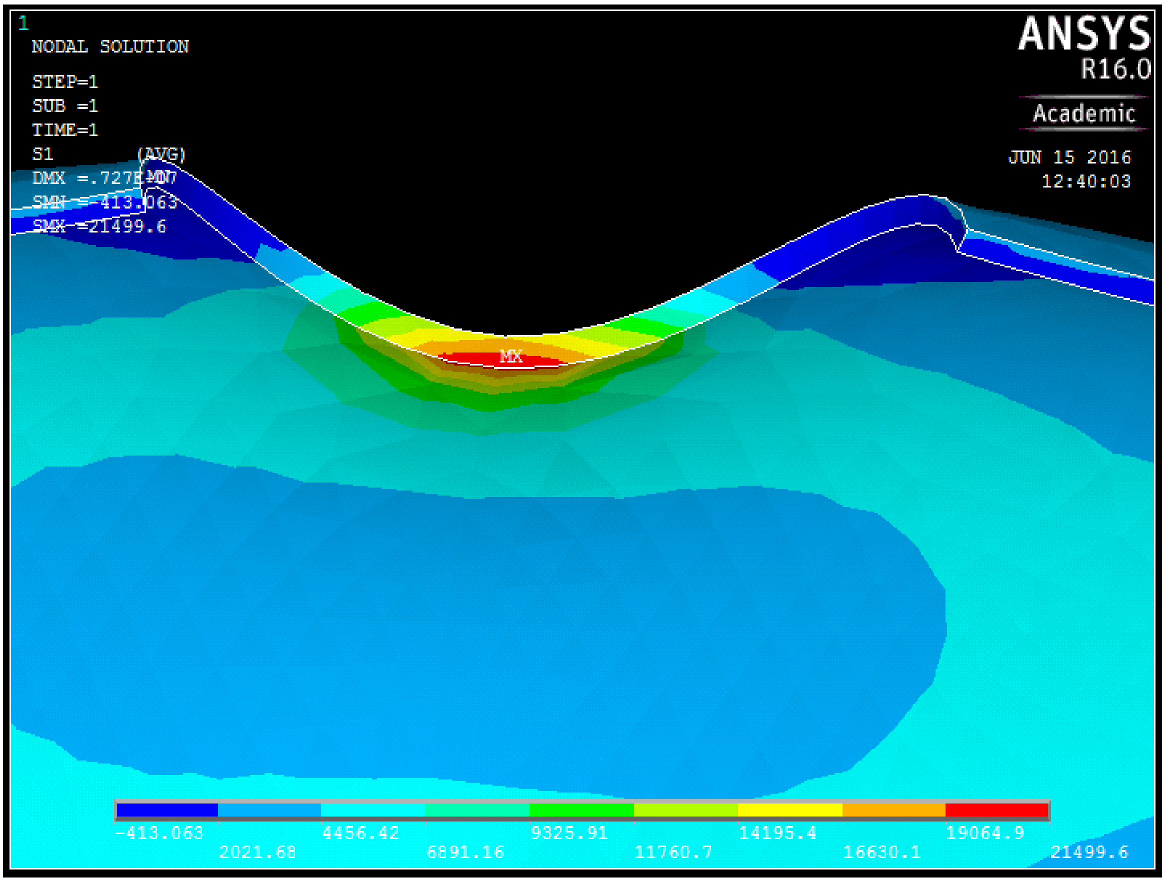


Illustration 89. Enhanced view of maximum stress location for D = 0.75, d/D = 0.65, di/D = 0.95

Maximum stress for this case is 21499.6 and in the exterior zone, stress is 14988.1 The difference from the exterior and interior layers is increased and this can be proven by the image above. As it can be seen, the red coloured area is no longer a perpendicular, plane section but it`s covering one of the edges. With the information extracted from the 4 analyses ran, the following table was created, aside with an interesting graph that sums up this study.

Outside					
D	d/D	di/D	Gross Stress (Pa)	Maximum Stress (Pa)	Kt
0.75	0.65	0.8	762.309	4391	5.760
0.75	0.65	0.85	988.941	5783	5.848
0.75	0.65	0.9	1444.375	7364.44	5.099
0.75	0.65	0.95	2814.679	14988.1	5.325

Inside					
D	d/D	di/D	Gross Stress (Pa)	Maximum Stress (Pa)	Kt
0.75	0.65	0.8	762.309	3682.62	4.831
0.75	0.65	0.85	988.941	5335.7	5.395
0.75	0.65	0.9	1444.375	8909.81	6.169
0.75	0.65	0.95	2814.679	21499.6	7.638

Illustration 90. Inside and outside layer Kt comparison

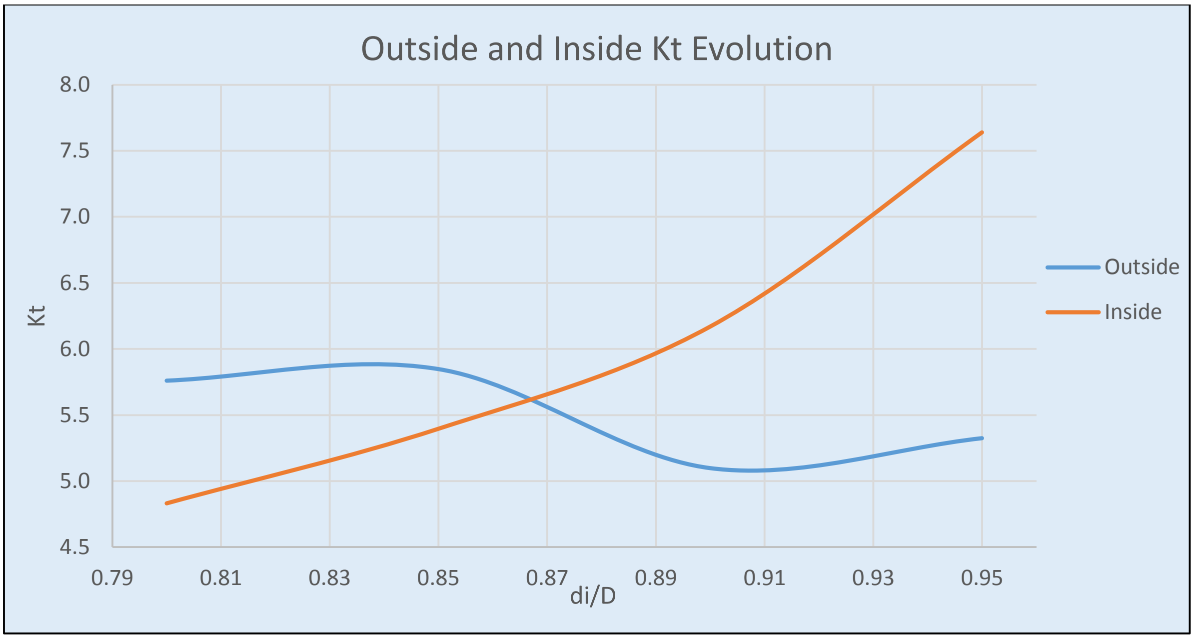


Illustration 91. Outside vs inside layer Kt chart

As shown in this plot, when di/D gets larger, which indirectly means that di is getting bigger and therefore the tube thinner, on the inside layers, Kt is always growing but on the outside of the tube, Kt has some ups and downs, meaning that there isn`t a clear tendency when stress grows. These fluctuations are due to the fact that the maximum stress can be on the interior or on the outside of the component. At approximately di/D = 0.87 is when maximum stress changes from outside layers to the interior of the tube.

This statement has been discovered for the axial load case but if one looks at the other load case charts, some other unexpected fluctuations are spotted, for example for the torsional moment. When di/D = 0.7, there is a bump in the graph so this also was studied, obtaining the same results as for the axial load. The maximum stress varies its position changing from the outside layers to the inside ones. Notice that for torsional moment cases, maximum stress is at a 45-degree angle, confirming that the analysis is correct. So it can be said that all the fluctuations in the graphs are due to the location of maximum stress and this derives into affirming that maximum stress is not always found at the same location.

So now, an important question to answer is why Finite Element Method results have the same structure, tendency and sequence as for Peterson`s method, unless for some cases where the Kt`s evolution isn`t so smooth. Well, first it has to be known how Peterson obtained his results and which method he followed. As it was mentioned in the first points of this report, Kt can be obtained experimentally by photo elastic essays or with strain gages and as it has to be, after a brief research on how Peterson (and colleagues) obtained their charts, it was found that most of them were obtained through the two previous methods. Some of these charts are from the nineteen forties and fifties and the finite element method wasn`t even rooted into engineering at that time. The way Peterson elaborated his charts was via strain gages.

Once that it`s known how Peterson plotted his graphs, one can compare them to the results found in this report and ask why some curves aren`t the same. It happens to be that Peterson fixed his strain gages where he thought that the maximum stress would appear, located near the stress raiser. He assumed where the maximum stress would occur, and was right fix the strain gages on the outside of the pipe but he didn`t take account for two important aspects, that can be discussed now. The first is that maximum stress doesn`t occur exactly on the exterior layer of the pipe but a few millimetres inside material. This problem could be saved assuming that the strain gages collected that information but the main aspect he didn`t count on was that Peterson stuck his strain gages on the outside of the pipe because he knew the approximate location of maximum stress and also because the outside of the pipe is the most accessible place to fix them. Imagine Peterson testing a tube with a small interior diameter and a small hole as stress raiser. It is practically impossible to stick strain gages on the inside (interior) of the pipe, so as an alternative, they were put on the outside. Now, for the majority of cases, maximum stress is located on the exterior layers of the tube and the results would be accurate but on the other hand, as it is shown in this present project, sometimes, maximum stress changes its location being at the interior layers of the pipe. Peterson would have his strain gages on the outside and would obtain a fake result, being the stress obtained slightly inferior to the real maximum stress, which would be at the inner layers of the tube.

This is the reason why some graphs are very similar to Peterson`s, when maximum stress was located on the outside, but when it was on the inside, the results were different, having an underrated stress value and therefore an underrated Kt value. Peterson calculated Kt values for a fixed location (where strain gages were), which was on the outside of the tube but following the research done in this report, maximum stress was calculated at a global level, wherever it was, being more accurate. This discovery reinforces the use of finite element method, being able to detect maximum stress no matter the location, affirming that it`s a powerful tool in post analysis terms.

Finally, before passing on to the final step in this project, out of the three criterions for each load case, one must be chosen, being the most accurate one. After observing the graphs presented previously, it is concluded that for the axial force and for bending moment, the most adequate criterion is 1st Principle Stress because these loads produce, mainly, normal stress and stresses in the X direction. And then for the torsional moment, the stresses created here are basically shear stresses so it`s logical to go for either Tresca or Von Misses. After viewing both criterions, the most accurate one is Von Misses.

9. POLYNOMIAL FITTING

9.1. PREVIOUS CONCEPTS

Reached this point, this project could be concluded and present as results the graphs shown before but in order to present results in a more adequate way, instead of presenting some graphs here is given an equation. It`s more comfortable for user to have an equation where he introduces the parameters and obtains the Kt final value. It`s much more practical than having to go into some graphs, allowing a certain error.

So the task now is to achieve an equation that represents the Kt information only by introducing d/D and di/D values. There are many ways into doing this but here is shown a least square method with aid of Matlab. The idea is to create a function in Matlab that receives known information, such as d/D, di/D and Kt and outputs the equation belonging to that least square fitting. Introduced the known information, this Matlab program must adjust to these values via a polynomial, calculating the equation values and as an addition a residue will be outputted as well. This residue is a measurement of how good enough the fitting is, being a good fitting holding a low residue.

First of all, one must determine which polynomial to converge, defining precision. As an initial trial, Kt field was adjusted to a quadratic polynomial just like:

$$P = [1 \quad \mathrm{X} \quad \mathrm{Y} \quad X^2 \quad XY \quad Y^2\,] \tag{16}$$

Now, say that a Q function is created as the multiplication of P and a vector containing some random values, say A.

$$A = \begin{vmatrix} a0 \\ a1 \\ a2 \\ a3 \\ a4 \\ a5 \end{vmatrix} \tag{17}$$

$$Q\,(x, y) = a_0 + a_1\,X + a_2\,Y + a_3\,X^2 + a_4\,XY + a_5\,Y^2 \tag{18}$$

X and Y would be d/D and di/D values and from these two parameters will arise a Z field which is the value of Kt for each combination of X and Y, being Z (X, Y). Now, if one subtracts Q (x, y) from Z (x, y), for each possible combination and search for the minimum of that operation, this comprises the least square method. The only unknown of this operation is a vector, which will be the values of the polynomial fitting. The mathematical operation is:

$$\min \textstyle\sum_i (Q_i\,(x, y) - Z_{\,i}(x, y))^2 = \min \textstyle\sum_i (P_i\, * a - Z_{\,i})^2 \tag{19}$$

In order to obtain the minimum of this function, one must derivate with respect to A and equal to zero. After operating, the final expression is in equation number 20, where all parameters are known except A.

$$(\sum_i (P_i^T * P_i - Z_{\,i}) * A = \sum_i P_i * Z_i \tag{20}$$

9.2. AXIAL FORCE

The attention is now focused on Matlab and programming the chores before mentioned. The first function to be created is function ***Polynomial.*** This function simply has the mission of telling the main function (ahead mentioned) which type of polynomial to adjust the results to. It must receive as inputs X and Y values and it will create a matrix P, just like in equation 16. The first column is filled up with a column vector of one, satisfying P`s size.

```
function    P=Polynomial(X,Y)

tam=size(X,1);
P=[ones(tam,1) X Y X.^2 X.*Y Y.^2];
```

Illustration 92. Polynomial function

The second function created is going to be the inputs for the main function, d/D, di/D and Kt. Matlab must be able to operate with these values and process them but for that, user must introduce information in the correct way. It was said at the end of point 8.1 that for axial force case, the most appropriate criterion was 1st Principle Stress as well for bending moment case. Then, for the torsional moment case Von Misses presented the most adequate results for these analyses. The way the parameters are introduced are in matrix form because it`s an efficient and ordinated manner to treat information in Matlab.

The matrix created will be called Data and will contain 3 columns and as many rows as analyses (64). The first column will be for d/D going from 0.1 to 0.8 in steps of 0.05. The second column will bear di/D values starting at 0.2 and ending at 0.9 in steps of 0.1. Finally, the third row will contemplate Kt values for the combination of the d/D and di/D values in that row. To make this data introducing chore easier, user can go to the excel files in the CD of this project or refer to the addendums in this report and copy values from the results presented. For the axial force under 1st Principle Stress criterion, data function is shown in previous illustrations. Because the function is so large, only a portion of it is shown in this illustration. This function is only copied information in rows, so the mechanics are simple and remain the same. Showing a few is enough to capture the idea of process.

```
Data = [...

%d/D    di/D        Kt
0.1     0.2     3.42578862920074;...
0.1     0.3     3.33136839594031;...
0.1     0.4     3.30008369030137;...
0.1     0.5     3.22127159363702;...
0.1     0.6     3.21285892111840;...
0.1     0.7     3.19228509588888;...
0.1     0.8     3.20337311997469;...
0.1     0.9     3.01541527897359;...
0.15    0.3     3.48101718478099;...
0.15    0.4     3.38355378466163;...
```

Illustration 93. Portion of Data function for axial force and 1st Principle stress

The last function in all this process is going to be the 'master' function. This function will receive data from Data and process it to output A vector, which are the values of the desired equation.

This function is called Coefficients and needs as an input Data from the analyses. The first operation done in this function is to order the information from Data and this is done by telling Matlab that first column belongs to d/D, the second one to di/D and the third one to Kt. Then X and Y are defined as dD and diD respectively. The lines that follow this are all due to the least square method explained in 9.1 previous concepts. Here is where vector A is obtained and some other variables are declared for plotting them further on.

In order to evaluate how good the fitting is, there`ll be two different methods. One will be by means of obtaining a residue (R). the smaller this residue is, the better approximation. The second checking method is to plot initial information and then plot on top the polynomial. As it can seem logical, if both plots are identical then the residue will be null and the fitting will be perfect.

```
function    a=Coefficients(Data)

close all

dD=Data(:,1);
diD=Data(:,2);
Kt0=Data(:,3);

X=dD;
Y=diD;

P=Polynomial(X,Y);
M=P'*P;
B=P'*Kt0;

a=M\B;

Z=P*a;

R=sum(((Z-Kt0).^2)./(Kt0.^2))
plot3(dD,diD,Kt0,'rx');
hold on
box on;

Z=Polynomial(X,Y)*a;
plot3(dD,diD,Z,'bo');

xlabel('d/D');
ylabel('di/D');
zlabel('Kt');
title('Polynomial Fitting');
```

Illustration 94. Coefficients function

The first plot is related to the initial information given by Data and to differentiate it from the other plot, it will be represented as red x marks. The polynomial will be plotted blue circles and to give the result some coherence, all three axis receive a name, as well as title for the entire graph. This function can be seen in illustration 94.

As it was said before, the first trial will be trying to adjust to a quadratic polynomial using the information from axial force under 1st Principle Stress. This information is stored in a file called ***DataAxialForce1stPrincipleStress.***

To carry out the fitting, enter Matlab software and enter the folder where all functions have been created. Secondly, type in the command window he name of the file where Data has been saved, in this case, type in ***DataAxialForce1stPrincipleStress.*** After pressing intro, in the workspace there must be a variable called data being a matrix of dimensions 64x3. Then, the last step is to call function Coefficients in the command window just like: ***a = Coefficients (Data)***. Automatically, An R value will be printed on screen (residue) followed by an A vector (coefficients of polynomial) and a secondary window in Matlab will appear with the two plotted graphs.

For this first trial, surprisingly, the residue is R = 0.060667434074019 and the plot is shown in illustration 95.

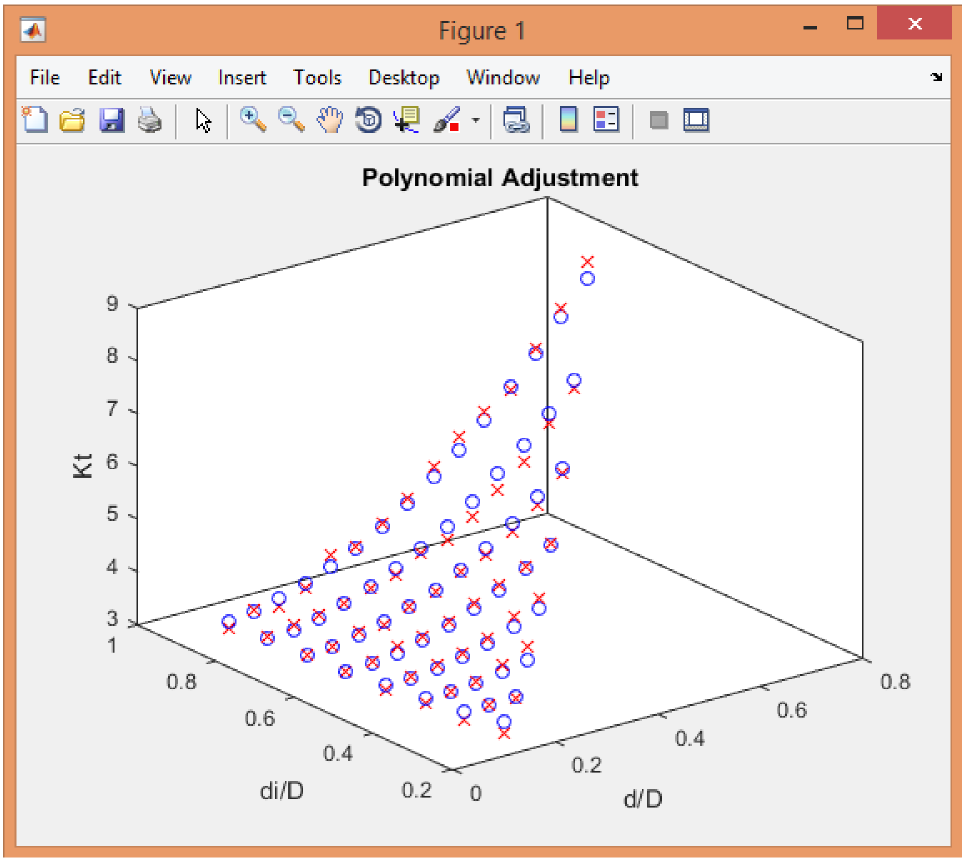


Illustration 95. First quadratic polynomial fitting

As it can be seen, the residue is quite low and the blue circle (polynomial) are very close to the initial information (red x marks). But at the extreme points where d/D and di/D are high, the fitting isn`t that good so this leads to another trial.

For the second trial, the information is going to be stretched so that the fitting is better. This is done by introducing Neperian logarithms. For times being, only Kt will be held under Neperian logarithms to see the effect. So, the polynomial function remains untouched as well as the Data function but the ***Coefficients*** function is slightly changed.

The polynomial that would result from this operation would have the following appearance:

$$Ln\, K_t = a_0 + a_1 \frac{d}{D} + a_2 \frac{di}{D} + a_3 \left(\frac{d}{D}\right)^2 + a_4 \frac{d}{D}\frac{di}{D} + a_5 \left(\frac{di}{D}\right)^2 \tag{21}$$

```
function    a=Coefficients(Data)
close all
dD=Data(:,1);
diD=Data(:,2);
Kt0=Data(:,3);
Kt1=log(Kt0);

X=dD;
Y=diD;

P=Polynomial(X,Y);
M=P'*P;
B=P'*Kt1;

a=M\B;

Z=P*a;

R=sum(((exp(Z)-Kt0).^2)./(Kt0.^2))
plot3(dD,diD,Kt0,'rx');
hold on
box on;

Z=Polynomial(X,Y)*a;
plot3(dD,diD,exp(Z),'bo');

xlabel('d/D');
ylabel('di/D');
zlabel('Kt');
title('Polynomial Fitting');
```

Illustration 96. Second quadratic trial with Ln Kt

Introduce the following command in the command window to execute fitting ***a = Coefficients (Data)*** and obtain the results just like before. In this new test, the residue value descends a little down to R = 0.041397629405568 and the figure emerged is visualized in illustration 97. Some of the points have adjusted better to this polynomial, specially, in the top corner, where d/D and di/D values are.

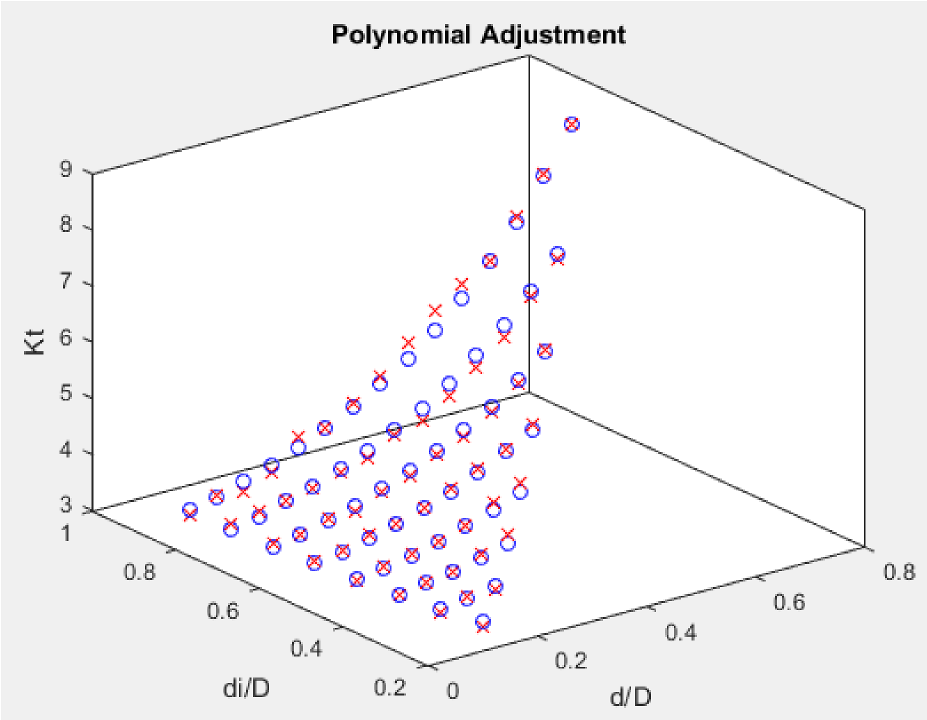


Illustration 97. Second trial plot

Once reached this point, trying putting X and Y (d/D and di/D) in Neperian logarithms was tested by the fitting was only worse and the graph didn`t tend to the initial information so this alternative was thrown away. But then came an idea to improve the fitting. One must remember point 8. Of this report where all graphs were shown. Here, some of these curves had ups and downs and this method is trying to adjust a polynomial to that paraboloid form. Instead, it was tested with a cubic function that could capture in a better way these S shapes in the function. For this change, ***Coefficients*** function isn`t changed at all and the change needs to be made in the ***Polynomials*** function, changing from a quadratic to a cubic function. The desired function will have the following appearance:

$$Ln\, K_t = a_0 + a_1 \frac{d}{D} + a_2 \frac{di}{D} + a_3 \left(\frac{d}{D}\right)^2 + a_4 \frac{d}{D}\frac{di}{D} + a_5 \left(\frac{di}{D}\right)^2 + a_6 \left(\frac{d}{D}\right)^3 + a_7 \left(\frac{d}{D}\right)^2 \frac{di}{D} + a_8 \frac{d}{D}\left(\frac{di}{D}\right)^2 + a_9 \left(\frac{d_i}{D}\right)^3 \tag{22}$$

This equation has more terms but it`s more accurate. ***Coefficients*** function is the same as illustration 96 but now, the ***Polynomial*** function is as in illustration 98.

```
function     P=Polynomial(X,Y)
tam=size(X,1);
P=[ones(tam,1)  X  Y  X.^2  X.*Y  Y.^2  X.^3  X.^2.*Y  X.*Y.^2  Y.^3];
```

Illustration 98. Cubic polynomial function

After running the Matlab program as before, the residue is stated at R = 0.021169505385467 and the plot emerged is shown in illustration 99.

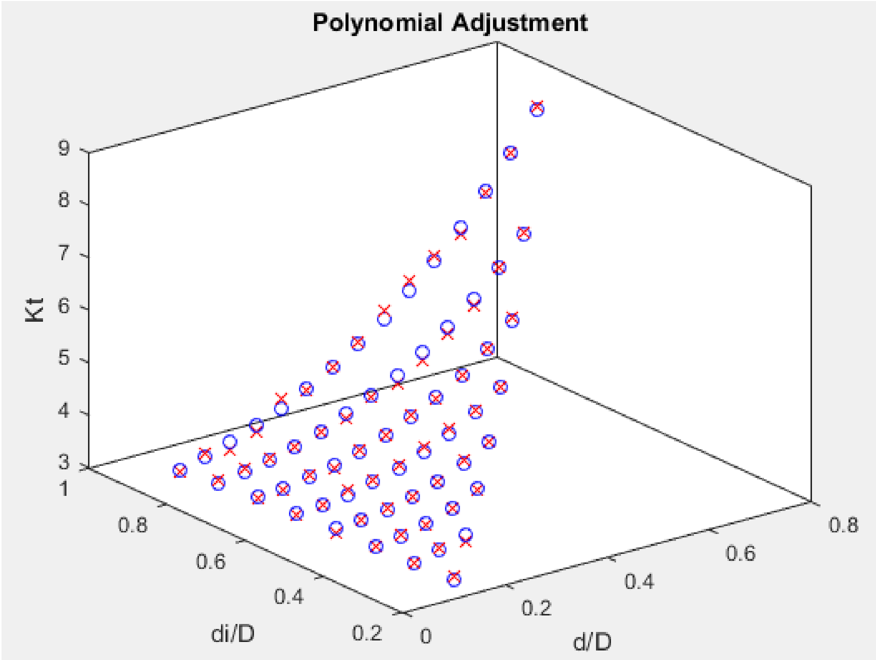


Illustration 99. Cubic polynomial fitting with Ln Kt

Seen the results and that every modification only improves the residue in 0.02, which isn`t much it`s concluded that this will be the final approach for all three load cases. The first fitting was good enough and only a few modifications were needed to adjust some points.

It also to mention that a better residue is very difficult to reach because of the fluctuations of the maximum stress being at the top layers and then at the bottom ones. Many other approaches have been tested and not mentioned in order to improve the polynomial fitting but all not mentioned are effective, for instance, inverting Kt to see the effect doesn`t help.

For the axial load, the final plot has been already shown in illustration 88, the functions have also been shown and the only aspect left is the equation. From the fitting in Matlab the A vector is:

```
a =

   1.067296579553410
   1.293605482338741
   0.481361027162074
   4.350762491358120
  -6.804363623852792
  -0.173281542267865
  -0.499302975066403
  -3.256499394073978
   6.563925358037619
  -0.430320338146333
```

The equation that represents Kt, function of d/D and di/D, for any geometry, attending to an axial force is:

$$Ln\,K_t = 1.0672965795 + 1.2936054823\,\frac{d}{D} + 0.4813610271\,\frac{di}{D} + 4.3507624913\,(\frac{d}{D})^2 - 6.8043636238\,\frac{d}{D}\frac{di}{D} - 0.1732815422\left(\frac{di}{D}\right)^2 - 0.4993029751\,(\frac{d}{D})^3 - 3.2564993941\left(\frac{d}{D}\right)^2\frac{di}{D} + 6.563925358\,\frac{d}{D}\left(\frac{di}{D}\right)^2 - 0.4303203381(\frac{d_i}{D})^3 \quad (22)$$

9.3. BENDING MOMENT

Once the axial force case is solved, the Matlab functions are the same. ***Coefficients*** function is shown in illustration 96 and ***Polynomials*** is represented in illustration 98. The only function that will change from each case is the Data, obviously. For the bending moment case, data will be introduced in the same format but values will change. Now, the results from the bending moment under 1st Principle Stress criterion need to be introduced into ***Data*** function. For convenience this file is named as ***DataBendingMoment1stPrincipleStress*** and can be found in the CD with this project. Call he Data function in the command window to have in workspace a 64x3 matrix. Then call ***Coefficients*** function just like before and the results will appear on screen.

The plotted fitting is shown in illustration 89 with a residue of R = 0.024684416439989. This is a similar value to axial load case, which is a good fitting.

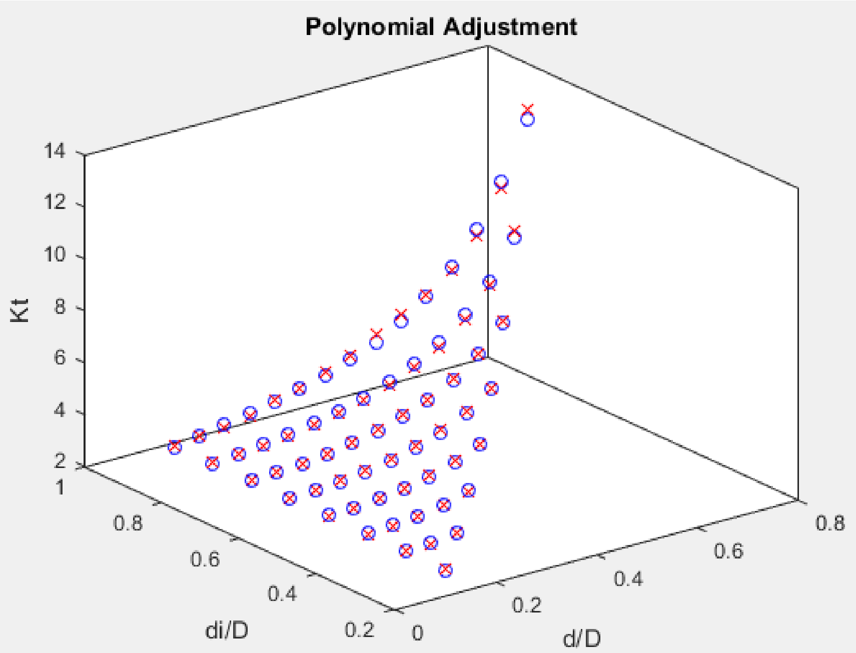


Illustration 100. Plot for cubic fitting bending moment

The A vector, coefficients of the equation, are in the next image, so the equation for bending moment case will be like equation 23.

```
a =

   0.938695220372350
   0.011653870704430
   1.069715631993602
   6.555765951870157
  -4.291749782771163
  -1.502458625417739
   2.679165958810125
  -8.770460134000366
   6.296190321219241
   0.375755448995840
```

$$Ln\ K_t = 0.938695220 + 0.011653870\ \frac{d}{D} + 1.069715631\ \frac{di}{D} + 6.555765951\ (\frac{d}{D})^2 - 4.291749782\ \frac{d}{D}\frac{di}{D} - 1.502458625\ \left(\frac{di}{D}\right)^2 + 2.679165958(\frac{d}{D})^3 - 8.770460134\left(\frac{d}{D}\right)^2\frac{di}{D} + 6.296190321\ \frac{d}{D}\left(\frac{di}{D}\right)^2 + 0.375755448(\frac{d_i}{D})^3 \quad (23)$$

9.4. TORSIONAL MOMENT

As before, change the Data function so that it holds the torsional moment under Von Misses criterion and call that function. In this case the file is named as ***DataTorsionalMomentVonMisses***. Execute the process and the results should be like illustration 101. This process responds with a residue of R = 0.049192617506024 and the A vector is directly incorporated in equation 24.

$$Ln\ K_t = 0.755811763 + 0.269550772\ \frac{d}{D} + 4.137873857\ \frac{di}{D} + 9.958183592\ (\frac{d}{D})^2 - 11.214476146\ \frac{d}{D}\frac{di}{D} - 7.385989094\ \left(\frac{di}{D}\right)^2 + 1.086258909(\frac{d}{D})^3 - 14.0948401262\left(\frac{d}{D}\right)^2\frac{di}{D} + 16.9520199396\ \frac{d}{D}\left(\frac{di}{D}\right)^2 + 3.4891291581 \quad (24)$$

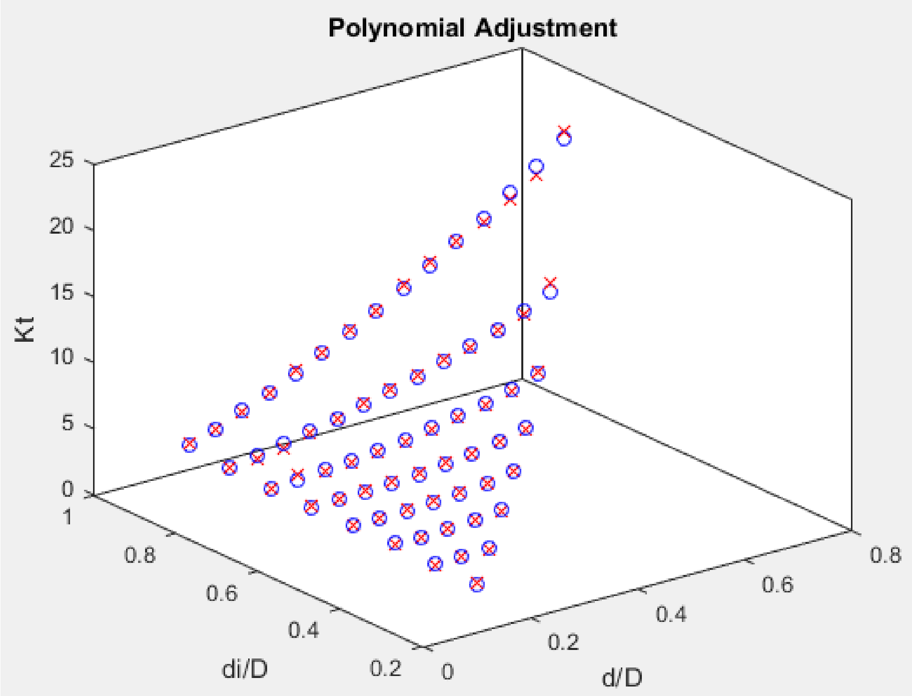


Illustration 101. Cubic fitting for torsional moment Tresca criterion

10. CONCLUSIONS

This project has established a process to calculate the stress concentration factor of a circular section pipe with a transverse hole, no matter the geometry or load applied. Some graphs have been presented but also an equation for each load case has been deducted, function of two basic parameters of the component.

The results of many different combinations of components have been calculated by means of a macro in a finite element method program, obtaining the essential information of the stress concentration factor, Kt. Posteriorly, this information has been post processed in Matlab to obtain an equation that reflects all these results. This equation is a polynomial fitting to the initial data extracted from ANSYS.

An important factor to mention as a conclusion is that this project has been done with an academic version of a finite element method software. This limits the designer's calculus power but even though, the results are convincing, achieving the final goal. Acquiring a professional version software would certainly improve the results here shown, allowing a further and deeper research on stress factors.

Viewing the results from this project and comparing them to the main bibliography of this report, being Peterson`s Stress Concentration Factors 3rd edition, there are some differences. First of all, comparing charts reveals that the method is correct because the shape and tendency is the same but in some areas, both graphs differ. It was concluded that Peterson obtained his charts via experimentation using strain gages. These were located at the exterior layers of the pipe because he predicted that maximum stress would be at the stress raiser. This assumption was correct in some ways. Strain gages were at the correct area of component but not at the correct location. As it was proven in this report, maximum stress is not always located at the exterior, contemplating the possibility of maximum stress changing from the exterior layers to the inner ones. Here is a main difference between both models. Peterson only represented stress at the location where strain gages were, holding a difference from obtaining maximum stress at any possible point of component. This statement reaffirms the use of the finite element method, avoiding problems of this type, impossible to detect experimentally.

Also, industrial processes realized on Peterson`s essay tubes can have a vast influence, such as manufacturing processes or numerical control techniques applied. All these factors can be eliminated from scene by adopting the finite element method, proven by the results of this report which models any component avoiding these issues. It must be held in mind formulation for the finite element method, guaranteeing a convergence to an exact solution (this is true while mesh gets smaller). The element used to mesh is able capture all information, leading to a precise solution, while Peterson didn`t take account for any possible internal defects, or manufacturing techniques.

This project has shown the effectiveness of the finite element method, producing satisfactory results against an uncertain experimental method that cannot bear all factors for a precise stress concentration factor calculus.

11. BIBLIOGRAPHY

[1] CÁLCULO ESTRUCTURAL "Método de los Elementos Finitos" Javier Fuenmayor Fernández Juan José Ródenas García José Enrique Tarancón Caro Manuel Tur Valiente Ana Vercher Martínez.

[2] Peterson`s Stress Concentration Factors. Third edition. Walter D. Pilkey Deborah F. Pilkey. Hoboken, New Jersey. USA. ISBN 978-0-470-04824-5.

[3] CÁLCULO ESTRUCTURAL "Método de los Elementos Finitos Prácticas de Laboratiorio" Javier Fuenmayor Fernández Juan José Ródenas García José Enrique Tarancón Caro Manuel Tur Valiente Ana Vercher Martínez.

[4] Anderson, T. L. Fracture Mechanics. Fundamentals and applications. Third edition. Boca Ratón, Florida. USA 2005. ISBN 10:0-8493-1656-1.

[5] Budynas R.G. and Nisbett J.K. Diseño en Ingeniería Mecánica de Shigley. Octava edición en español, Mac Graw Hill, México, 2008.

[6] El Método de los Elementos Finitos Volúmen I y II. O.C.Zienkiewicz, R.L. Taylor. Ed. McGraw-Hill.

[7] Técnicas Computacionales en la Ingeniería Mecánica. Método de los elementos finitos. Juan José Ródenas García José Enrique Tarancón Caro. Departamento de Ingeniería Mecánica y de Materiales. Universidad Politécnica de Valencia.

12. ACKNOWLEDGEMENTS

Above all I would like to thank Dr. Juan José Ródenas García, my teacher and tutor for giving me the opportunity to carry out this project and for sharing his knowledge and time with me. It`s always a pleasure to learn from the best and special gratitude for being humble in every aspect.

Furthermore, I would like to mention my brother Sam for being a constant drive in my life and for supporting me no matter what. We both deeply know that without him, I wouldn`t be where I am now and certainly, wouldn`t be the same man.

In addition, I want to thank my family and specially my mother and uncles. My mother for guiding and advising me and my uncles for giving me the opportunity to study in a good university. Thankfulness for being a pillar to lean on whenever needed.

Last but not least, thank my girlfriend Ana María for supporting me in difficult moments and sharing many others, and her family for being by my side in every decision.

This project has a bit of everyone, although they might not never know it.

13. ANNEXES

13.1. MACRO

```
! ANSYS START UP !
/GRA,POWER
/GST,ON
/PLO,INFO,3
/GRO,CURL,ON
/CPLANE,1
/REPLOT,RESIZE
! DEFINING ESSENTIAL PARAMETERS !
D = ARG1
DD = ARG2
DID = ARG3
! USEFUL VARIABLES !
LENG = 3.67*D
DSR = DD*D
DI = DID*D
LOAD = 121.24
EXTCYL = (DSR/2)+0.03
ALPHA = ASIN((EXTCYL)/(DI/2))*(180/3.1415926535)
BETA = ALPHA + 5
! LOGICAL CONDITION CASE 1 !
*IF,ALPHA,LE,60,THEN
! WRITING TITLE !
WPSTYLE,,,,,,,,0
/TITLE, CASE 1: D = %D%, d/D = %DD%, di/D = %DID%
! PREFERENCES !
/NOPR
KEYW,PR_SET,1
KEYW,PR_STRUC,1
KEYW,PR_THERM,0
KEYW,PR_FLUID,0
KEYW,PR_ELMAG,0
KEYW,MAGNOD,0
KEYW,MAGEDG,0
KEYW,MAGHFE,0
KEYW,MAGELC,0
KEYW,PR_MULTI,0
/GO
/PREP7
! ELEMENT TYPE DEFINITION !
ET,1,SOLID187
ET,2,MASS21
R,1,0,0,0,0,0,0,
MPTEMP,,,,,,,,
MPTEMP,1,0
! MATERIAL PROPERTIES !
MPDATA,EX,1,,2.1e11
MPDATA,PRXY,1,,0.3
! KEYPOINT CREATION !
K,1,0,0,0,
K,2,LENG,0,0,
K,3,0,DI/2,0,
K,4,0,D/2,0,
K,5,LENG,D/2,0,
K,6,LENG,DI/2,0,
K,7,(LENG/2)(DSR/2)(0.15),DI/2,0,
K,8,(LENG/2)(DSR/2)(0.15),D/2,0,
K,9,(LENG/2)+(DSR/2)+(0.15),D/2,0,
K,10,(LENG/2)+(DSR/2)+(0.15),DI/2,0,
! CREATING COORDINATE SYSTEM 11 !
LOCAL,11,0,0,0,0, ,(90BETA), ,1,1,
```

```
WPCSYS,1,11,
! KEYPOINTS AT CS 11 !
K,11,LENG,0,DI/2,
K,12,LENG,0,D/2,
K,13,LENG,0,DI/2,
K,14,LENG,0,D/2,
! CREATING COORDINATE SYSTEM 12 !
LOCAL,12,0,0,0,0, ,90BETA, ,1,1,
WPCSYS,1,12,
! KEYPOINTS AT CS 12 !
K,15,LENG,0,DI/2,
K,16,LENG,0,D/2,
K,17,LENG,0,DI/2,
K,18,LENG,0,D/2,
! LINES FOR DIVIDING PIPE !
LSTR,     11,     12
LSTR,     17,     18
LSTR,     13,     14
LSTR,     15,     16
! LINES FOR HOLLOW PIPE !
LSTR,      1,      2
LSTR,      3,      4
LSTR,      4,      8
LSTR,      8,      7
LSTR,      3,      7
LSTR,      7,     10
LSTR,     10,      9
LSTR,      8,      9
LSTR,      9,      5
LSTR,     10,      6
LSTR,      5,      6
! AREAS FOR HOLLOW PIPE CREATION !
FLST,2,4,4
FITEM,2,9
FITEM,2,6
FITEM,2,7
FITEM,2,8
AL,P51X
FLST,2,4,4
FITEM,2,10
FITEM,2,8
FITEM,2,12
FITEM,2,11
AL,P51X
FLST,2,4,4
FITEM,2,14
FITEM,2,15
FITEM,2,13
FITEM,2,11
AL,P51X
! AREAS FOR DIVIDING HOLLOW PIPE !
ADRAG,     1, , , , , ,     5
ADRAG,     2, , , , , ,     5
ADRAG,     4, , , , , ,     5
ADRAG,     3, , , , , ,     5
! REVOLUTE AREAS AROUND AXIS !
FLST,2,3,5,ORDE,2
FITEM,2,1
FITEM,2,3
FLST,8,2,3
FITEM,8,1
FITEM,8,2
VROTAT,P51X, , , , , ,P51X, ,360,4,
! DIVIDING PIPE INTO 8 SECTIONS !
FLST,2,12,6,ORDE,2
FITEM,2,1
FITEM,2,12
FLST,3,4,5,ORDE,2
FITEM,3,4
FITEM,3,7
```

```
VSBA,P51X,P51X
! CREATING COORD SYSTEM FOR STRESS RAISER !
LOCAL,13,0,LENG/2,(D/2)+((D/2)(DI/2)),0, ,90, ,1,1,
WPCSYS,1,13,
! STRESS RAISER CYLINDER !
CYL4,0,0,DSR/2, , , ,D+(D/2)
FLST,2,24,6,ORDE,2
FITEM,2,13
FITEM,2,36
VSBV,P51X,     1
! SURROUNDING CYLINDER !
CYL4,0,0,EXTCYL, , , ,D+(D/2)
VDELE,     1
FLST,2,8,6,ORDE,6
FITEM,2,2
FITEM,2,5
FITEM,2,21
FITEM,2,22
FITEM,2,33
FITEM,2,34
FLST,3,4,5,ORDE,3
FITEM,3,2
FITEM,3,4
FITEM,3,6
VSBA,P51X,P51X
 ! DIVIDING ALL VOLUME BY WORKPLANE !
wpro,,,90.000000
wpstyle,0.05,0.1,10,10,0.003,0,1,,5
FLST,2,28,6,ORDE,7
FITEM,2,1
FITEM,2,6
FITEM,2,12
FITEM,2,15
FITEM,2,26
FITEM,2,29
FITEM,2,36
VSBW,P51X
wpro,,,90.000000
wpstyle,0.05,0.1,10,10,0.003,0,2,,5
! RETURNING TO COORDINATE SYSTEM 0 !
WPSTYLE,,,,,,,,0
WPCSYS,1,0,
csys,0
! MESHING !
! MESHING ATTRIBUTES !
FLST,5,110,3,ORDE,2
FITEM,5,1
FITEM,5,110
CM,_Y,KP
KSEL, , , ,P51X
CM,_Y1,KP
CMSEL,S,_Y
CMSEL,S,_Y1
KATT,     1,     1,   1,     0
CMSEL,S,_Y
CMDELE,_Y
CMDELE,_Y1
CM,_Y,KP
KSEL, , , ,     2
CM,_Y1,KP
CMSEL,S,_Y
CMSEL,S,_Y1
KATT,     1,     1,   2,     0
CMSEL,S,_Y
CMDELE,_Y
CMDELE,_Y1
! END SECTION KEYPOINT SIZE !
KSEL,S,LOC,X,0
KSEL,A,LOC,X,LENG
KPLOT
```

```
FLST,5,34,3,ORDE,13
FITEM,5,1
FITEM,5,6
FITEM,5,11
FITEM,5,27
FITEM,5,30
FITEM,5,33
FITEM,5,35
FITEM,5,38
FITEM,5,41
FITEM,5,43
FITEM,5,46
FITEM,5,49
FITEM,5,50
CM,_Y,KP
KSEL, , , ,P51X
CM,_Y1,KP
CMSEL,S,_Y
CMSEL,S,_Y1
KESIZE,ALL,(DD)
CMSEL,S,_Y
CMDELE,_Y1
CMDELE,_Y
! STRESS RAISER LINE SIZE !
FLST,5,24,4,ORDE,22
FITEM,5,10
FITEM,5,18
FITEM,5,23
FITEM,5,26
FITEM,5,35
FITEM,5,39
FITEM,5,45
FITEM,5,57
FITEM,5,62
FITEM,5,63
FITEM,5,69
FITEM,5,200
FITEM,5,202
FITEM,5,204
FITEM,5,210
FITEM,5,212
FITEM,5,214
FITEM,5,216
FITEM,5,222
FITEM,5,223
FITEM,5,226
FITEM,5,227
CM,_Y,LINE
LSEL, , , ,P51X
CM,_Y1,LINE
CMSEL,,_Y
LESIZE,_Y1,(DSR/32.5), , , , , , ,1
! EXTERIOR CYLINDER LINE SIZE !
FLST,5,24,4,ORDE,24
FITEM,5,12
FITEM,5,20
FITEM,5,24
FITEM,5,27
FITEM,5,40
FITEM,5,54
FITEM,5,58
FITEM,5,68
FITEM,5,81
FITEM,5,88
FITEM,5,187
FITEM,5,201
FITEM,5,205
FITEM,5,207
FITEM,5,209
FITEM,5,211
```

```
FITEM,5,213
FITEM,5,217
FITEM,5,219
FITEM,5,221
FITEM,5,224
FITEM,5,225
FITEM,5,228
FITEM,5,229
CM,_Y,LINE
LSEL, , , ,P51X
CM,_Y1,LINE
CMSEL,,_Y
LESIZE,_Y1,(DSR/24.5), , , , , , ,1
! AREA SIZES !
FLST,5,64,5,ORDE,25
FITEM,5,1
FITEM,5,3
FITEM,5,12
FITEM,5,20
FITEM,5,25
FITEM,5,33
FITEM,5,38
FITEM,5,46
FITEM,5,67
FITEM,5,84
FITEM,5,86
FITEM,5,89
FITEM,5,91
FITEM,5,99
FITEM,5,107
FITEM,5,109
FITEM,5,112
FITEM,5,114
FITEM,5,122
FITEM,5,130
FITEM,5,132
FITEM,5,134
FITEM,5,136
FITEM,5,143
FITEM,5,148
ASEL,U, , ,P51X
FLST,2,16,5,ORDE,8
FITEM,2,196
FITEM,2,197
FITEM,2,199
FITEM,2,204
FITEM,2,206
FITEM,2,207
FITEM,2,209
FITEM,2,214
AESIZE,P51X,(DSR/17.5),
 FLST,2,16,5,ORDE,16
FITEM,2,165
FITEM,2,166
FITEM,2,169
FITEM,2,170
FITEM,2,173
FITEM,2,174
FITEM,2,177
FITEM,2,178
FITEM,2,181
FITEM,2,182
FITEM,2,185
FITEM,2,186
FITEM,2,190
FITEM,2,191
FITEM,2,193
FITEM,2,209
AESIZE,P51X,(DD/3),
! MESHING KEYPOINT 2 (CREATING NODE 1) !
```

```
ALLSEL,ALL
KMESH,      2
MSHAPE,1,3D
MSHKEY,0
! MESHING ALL VOLUMES !
FLST,5,40,6,ORDE,8
FITEM,5,2
FITEM,5,5
FITEM,5,13
FITEM,5,20
FITEM,5,23
FITEM,5,32
FITEM,5,35
FITEM,5,52
CM,_Y,VOLU
VSEL, , , ,P51X
CM,_Y1,VOLU
CHKMSH,'VOLU'
CMSEL,S,_Y
VMESH,_Y1
CMDELE,_Y
CMDELE,_Y1
CMDELE,_Y2
! BOUNDARY CONDITIONS !
! EMBEDMENT !
ASEL,S,LOC,X,0
FLST,2,8,5,ORDE,8
FITEM,2,75
FITEM,2,79
FITEM,2,83
FITEM,2,87
FITEM,2,106
FITEM,2,110
FITEM,2,129
FITEM,2,133
/GO
DA,P51X,ALL,
! RIGID REGION !
NSEL,S,LOC,X,LENG
NPLOT
CERIG,1,ALL,ALL, , , ,
ALLSEL,ALL
! SETTING LOADS !
FLST,2,1,3,ORDE,1
FITEM,2,2
! AXIAL FORCE !
/GO
FK,P51X,FX,LOAD
LSWRITE,1,
FLST,2,1,3,ORDE,1
FITEM,2,2
FKDELE,P51X,ALL
FLST,2,1,1,ORDE,1
FITEM,2,1
FDELE,P51X,ALL
FLST,2,1,3,ORDE,1
FITEM,2,2
! BENDING MOMENT !
/GO
FK,P51X,MZ,LOAD
LSWRITE,2,
FLST,2,1,3,ORDE,1
FITEM,2,2
FKDELE,P51X,ALL
FLST,2,1,1,ORDE,1
FITEM,2,1
FDELE,P51X,ALL
FLST,2,1,3,ORDE,1
FITEM,2,2
! TORSIONAL MOMENT !
```

```
/GO
FK,P51X,MX,LOAD
LSWRITE,3,
FLST,2,1,3,ORDE,1
FITEM,2,2
FKDELE,P51X,ALL
FLST,2,1,1,ORDE,1
FITEM,2,1
FDELE,P51X,ALL
ALLSEL,ALL
FINISH
! SOLVE !
/SOL
LSSOLVE,1,3,1,
FINISH
! DEFINING EACH LOAD CASE !
/POST1
LCDEF,1,1
LCDEF,2,2
LCDEF,3,3
! SAVING DATA FOR POST PROCESSING !
VPLOT
FLST,5,10,6,ORDE,10
FITEM,5,13
FITEM,5,15
FITEM,5,23
FITEM,5,24
FITEM,5,28
FITEM,5,30
FITEM,5,39
FITEM,5,40
FITEM,5,47
FITEM,5,52
VSEL,U, , ,P51X
ESLV,S
EPLOT
ESLV,S
/VIEW,1,1,1,1
/ANG,1
/REP,FAST
/DIST,1,0.924021086472,1
/REP,FAST
! AXIAL FORCE !
LCASE,1
/EFACET,1
PLNSOL, S,1, 0,1.0
/UI,COPY,SAVE,JPEG,FULL,COLOR,NORM,PORTRAIT,NO,
100
/RENAME,SolveTube000,jpg,,C:\Users\BenJordan\Docu
ments\University\TFG\ANSYS\Results\AxialForce\1stPri
ncipleStress/%D% %DD% %DID%,jpg
/EFACET,1
PLNSOL, S,INT, 0,1.0
/UI,COPY,SAVE,JPEG,FULL,COLOR,NORM,PORTRAIT,NO,
100
/RENAME,SolveTube000,jpg,,C:\Users\BenJordan\Docu
ments\University\TFG\ANSYS\Results\AxialForce\StressI
ntensity/%D% %DD% %DID%,jpg
/EFACET,1
PLNSOL, S,EQV, 0,1.0
/UI,COPY,SAVE,JPEG,FULL,COLOR,NORM,PORTRAIT,NO,
100
/RENAME,SolveTube000,jpg,,C:\Users\BenJordan\Docu
ments\University\TFG\ANSYS\Results\AxialForce\VonMi
ssesStress/%D% %DD% %DID%,jpg
! BENDING MOMENT !
LCASE,2
/EFACET,1
PLNSOL, S,1, 0,1.0
/UI,COPY,SAVE,JPEG,FULL,COLOR,NORM,PORTRAIT,NO,
100
/RENAME,SolveTube000,jpg,,C:\Users\BenJordan\Docu
ments\University\TFG\ANSYS\Results\BendingMoment\
1stPrincipleStress/%D% %DD% %DID%,jpg
/EFACET,1
```

```
PLNSOL, S,INT, 0,1.0

/UI,COPY,SAVE,JPEG,FULL,COLOR,NORM,PORTRAIT,NO,
100

/RENAME,SolveTube000,jpg,,C:\Users\BenJordan\Docu
ments\University\TFG\ANSYS\Results\BendingMoment\
StressIntensity/%D% %DD% %DID%,jpg

/EFACET,1

PLNSOL, S,EQV, 0,1.0

/UI,COPY,SAVE,JPEG,FULL,COLOR,NORM,PORTRAIT,NO,
100

/RENAME,SolveTube000,jpg,,C:\Users\BenJordan\Docu
ments\University\TFG\ANSYS\Results\BendingMoment\
VonMissesStress/%D% %DD% %DID%,jpg

! TORSIONAL MOMENT !

LCASE,3

/EFACET,1

PLNSOL, S,1, 0,1.0

/UI,COPY,SAVE,JPEG,FULL,COLOR,NORM,PORTRAIT,NO,
100

/RENAME,SolveTube000,jpg,,C:\Users\BenJordan\Docu
ments\University\TFG\ANSYS\Results\TorsionalMoment
\1stPrincipleStress/%D% %DD% %DID%,jpg

/EFACET,1

PLNSOL, S,INT, 0,1.0

/UI,COPY,SAVE,JPEG,FULL,COLOR,NORM,PORTRAIT,NO,
100

/RENAME,SolveTube000,jpg,,C:\Users\BenJordan\Docu
ments\University\TFG\ANSYS\Results\TorsionalMoment
\StressIntensity/%D% %DD% %DID%,jpg

/EFACET,1

PLNSOL, S,EQV, 0,1.0

/UI,COPY,SAVE,JPEG,FULL,COLOR,NORM,PORTRAIT,NO,
100

/RENAME,SolveTube000,jpg,,C:\Users\BenJordan\Docu
ments\University\TFG\ANSYS\Results\TorsionalMoment
\VonMissesStress/%D% %DD% %DID%,jpg

SAVE

FINISH

! LOGICAL CONDITION CASE 2 !

*ELSEIF,DD,LE,0.3,THEN

! WRITING TITLE !

WPSTYLE,,,,,,,,0

/TITLE, CASE 2: D = %D%, d/D = %DD%, di/D = %DID%

! PREFERENCES !

/NOPR

KEYW,PR_SET,1

KEYW,PR_STRUC,1

KEYW,PR_THERM,0

KEYW,PR_FLUID,0

KEYW,PR_ELMAG,0

KEYW,MAGNOD,0

KEYW,MAGEDG,0

KEYW,MAGHFE,0

KEYW,MAGELC,0

KEYW,PR_MULTI,0

/GO

/PREP7

! ELEMENT TYPE DEFINITION !

ET,1,SOLID187

ET,2,MASS21

R,1,0,0,0,0,0,0,

MPTEMP,,,,,,,,

MPTEMP,1,0

! MATERIAL PROPERTIES !

MPDATA,EX,1,,2.1e11

MPDATA,PRXY,1,,0.3

! KEYPOINT CREATION !

K,1,0,0,0,

K,2,LENG,0,0,

K,3,0,DI/2,0,

K,4,0,D/2,0,

K,5,LENG,D/2,0,

K,6,LENG,DI/2,0,

K,7,(LENG/2)(DSR/2)(0.15),DI/2,0,

K,8,(LENG/2)(DSR/2)(0.15),D/2,0,

K,9,(LENG/2)+(DSR/2)+(0.15),D/2,0,
```

```
K,10,(LENG/2)+(DSR/2)+(0.15),DI/2,0,
! LINES FOR HOLLOW PIPE !
LSTR,       1,       2
LSTR,       3,       4
LSTR,       4,       8
LSTR,       8,       7
LSTR,       7,       3
LSTR,       8,       9
LSTR,       9,      10
LSTR,      10,       7
LSTR,      10,       6
LSTR,       6,       5
LSTR,       5,       9
! AREAS CREATION !
FLST,2,4,4
FITEM,2,5
FITEM,2,2
FITEM,2,3
FITEM,2,4
AL,P51X
FLST,2,4,4
FITEM,2,8
FITEM,2,4
FITEM,2,6
FITEM,2,7
AL,P51X
FLST,2,4,4
FITEM,2,9
FITEM,2,10
FITEM,2,11
FITEM,2,7
AL,P51X
! AREAS REVOLUTION !
FLST,2,3,5,ORDE,2
FITEM,2,1
FITEM,2,3
FLST,8,2,3
FITEM,8,1
FITEM,8,2
VROTAT,P51X, , , , , ,P51X, ,360,4,
! CREATING COORD SYSTEM FOR STRESS RAISER !
LOCAL,11,0,LENG/2,(D/2)+((D/2)(DI/2)),0, ,90, ,1,1,
WPCSYS,1,11,
! STRESS RAISER CYLINDER !
CYL4,0,0,DSR/2, , , ,D+(D/2)
FLST,2,13,6,ORDE,2
FITEM,2,1
FITEM,2,13
VSBV,P51X,      13
! SURROUNDING CYLINDER !
CYL4,0,0,EXTCYL, , , ,1.125
 VDELE,       2
FLST,2,12,6,ORDE,10
FITEM,2,1
FITEM,2,3
FITEM,2,4
FITEM,2,6
FITEM,2,7
FITEM,2,9
FITEM,2,10
FITEM,2,12
FITEM,2,14
FITEM,2,17
FLST,3,4,5,ORDE,4
FITEM,3,2
FITEM,3,9
FITEM,3,10
FITEM,3,22
VSBA,P51X,P51X
! DIVIDING ALL VOLUME BY WORKPLANE !
```

```
wpstyle,0.05,0.1,10,10,0.003,0,1,,5
wpro,,,90.000000
FLST,2,16,6,ORDE,4
FITEM,2,1
FITEM,2,13
FITEM,2,18
FITEM,2,20
VSBW,P51X
wpro,,,90.000000
wpstyle,0.05,0.1,10,10,0.003,0,2,,5
! RETURNING TO COORDINATE SYSTEM 0 !
WPSTYLE,,,,,,,,0
WPCSYS,1,0,
CSYS,0
! MESHING !
! MESHING ATTRIBUTES !
FLST,5,70,3,ORDE,2
FITEM,5,1
FITEM,5,70
CM,_Y,KP
KSEL, , , ,P51X
CM,_Y1,KP
CMSEL,S,_Y
CMSEL,S,_Y1
KATT,      1,       1,   1,       0
CMSEL,S,_Y
CMDELE,_Y
CMDELE,_Y1
CM,_Y,KP
KSEL, , , ,       2
CM,_Y1,KP
CMSEL,S,_Y
CMSEL,S,_Y1
KATT,      1,       1,   2,       0
CMSEL,S,_Y
CMDELE,_Y
CMDELE,_Y1
! END SECTION KEYPOINT SIZE !
KSEL,S,LOC,X,0
KSEL,A,LOC,X,LENG
KPLOT
KSEL,A,LOC,X,2.7525
FLST,5,18,3,ORDE,14
FITEM,5,1
FITEM,5,6
FITEM,5,12
FITEM,5,13
FITEM,5,17
FITEM,5,18
FITEM,5,20
FITEM,5,21
FITEM,5,25
FITEM,5,26
FITEM,5,28
FITEM,5,29
FITEM,5,33
FITEM,5,34
CM,_Y,KP
KSEL, , , ,P51X
CM,_Y1,KP
CMSEL,S,_Y
CMSEL,S,_Y1
KESIZE,ALL,(DD)
CMSEL,S,_Y
CMDELE,_Y1
CMDELE,_Y
! MIDDLE SECTION AREA SIZES !
FLST,5,32,5,ORDE,14
FITEM,5,1
FITEM,5,3
```

```
FITEM,5,6
FITEM,5,8
FITEM,5,13
FITEM,5,19
FITEM,5,21
FITEM,5,26
FITEM,5,32
FITEM,5,34
FITEM,5,39
FITEM,5,45
FITEM,5,50
FITEM,5,52
ASEL,U, , ,P51X
 FLST,2,76,5,ORDE,29
FITEM,2,2
FITEM,2,7
FITEM,2,9
FITEM,2,11
FITEM,2,20
FITEM,2,22
FITEM,2,25
FITEM,2,33
FITEM,2,37
FITEM,2,46
FITEM,2,48
FITEM,2,49
FITEM,2,53
FITEM,2,57
FITEM,2,58
FITEM,2,60
FITEM,2,63
FITEM,2,65
FITEM,2,66
FITEM,2,68
FITEM,2,69
FITEM,2,71
FITEM,2,72
FITEM,2,79
FITEM,2,80
FITEM,2,83
FITEM,2,84
FITEM,2,91
FITEM,2,134
AESIZE,P51X,(DSR/3.5),
! EXTERNAL CYLINDER AREA SIZE !
FLST,5,40,5,ORDE,20
FITEM,5,2
FITEM,5,9
FITEM,5,10
FITEM,5,22
FITEM,5,23
FITEM,5,25
FITEM,5,48
FITEM,5,53
FITEM,5,57
FITEM,5,58
FITEM,5,60
FITEM,5,63
FITEM,5,65
FITEM,5,66
FITEM,5,68
FITEM,5,69
FITEM,5,71
FITEM,5,72
FITEM,5,91
FITEM,5,110
ASEL,S, , ,P51X
FLST,2,40,5,ORDE,20
FITEM,2,2
FITEM,2,9
```

```
FITEM,2,10
FITEM,2,22
FITEM,2,23
FITEM,2,25
FITEM,2,48
FITEM,2,53
FITEM,2,57
FITEM,2,58
FITEM,2,60
FITEM,2,63
FITEM,2,65
FITEM,2,66
FITEM,2,68
FITEM,2,69
FITEM,2,71
FITEM,2,72
FITEM,2,91
FITEM,2,110
AESIZE,P51X,(DSR/9),
! STRESS RAISER AREA SIZE !
FLST,5,8,5,ORDE,8
FITEM,5,22
FITEM,5,62
FITEM,5,68
FITEM,5,92
FITEM,5,93
FITEM,5,98
FITEM,5,103
FITEM,5,107
ASEL,S, , ,P51X
FLST,2,8,5,ORDE,8
FITEM,2,22
FITEM,2,62
FITEM,2,68
FITEM,2,92
FITEM,2,93
FITEM,2,98
FITEM,2,103
FITEM,2,107
AESIZE,P51X,(DSR/12),
! MESHING KEYPOINT 2 (CREATING NODE 1) !
ALLSEL,ALL
KMESH,     2
MSHAPE,1,3D
MSHKEY,0
! MESHING ALL VOLUMES !
FLST,5,24,6,ORDE,12
FITEM,5,1
FITEM,5,3
FITEM,5,4
FITEM,5,6
FITEM,5,7
FITEM,5,9
FITEM,5,10
FITEM,5,12
FITEM,5,14
FITEM,5,17
FITEM,5,21
FITEM,5,32
CM,_Y,VOLU
VSEL, , , ,P51X
CM,_Y1,VOLU
CHKMSH,'VOLU'
CMSEL,S,_Y
VMESH,_Y1
CMDELE,_Y
CMDELE,_Y1
CMDELE,_Y2
/VIEW,1,1,1,1
/ANG,1
```

```
/REP,FAST
! BOUNDARY CONDITIONS !
! EMBEDMENT !
ASEL,S,LOC,X,0
FLST,2,4,5,ORDE,4
FITEM,2,5
FITEM,2,18
FITEM,2,31
FITEM,2,44
/GO
DA,P51X,ALL,
! RIGID REGION !
NSEL,S,LOC,X,LENG
NPLOT
CERIG,1,ALL,ALL, , , ,
ALLSEL,ALL
! SETTING LOADS !
FLST,2,1,3,ORDE,1
FITEM,2,2
! AXIAL FORCE !
/GO
FK,P51X,FX,LOAD
LSWRITE,1,
FLST,2,1,3,ORDE,1
FITEM,2,2
FKDELE,P51X,ALL
FLST,2,1,1,ORDE,1
FITEM,2,1
FDELE,P51X,ALL
FLST,2,1,3,ORDE,1
FITEM,2,2
! BENDING MOMENT !
/GO
FK,P51X,MZ,LOAD
LSWRITE,2,
FLST,2,1,3,ORDE,1
FITEM,2,2
FKDELE,P51X,ALL
FLST,2,1,1,ORDE,1
FITEM,2,1
FDELE,P51X,ALL
FLST,2,1,3,ORDE,1
FITEM,2,2
! TORSIONAL MOMENT !
/GO
FK,P51X,MX,LOAD
LSWRITE,3,
FLST,2,1,3,ORDE,1
FITEM,2,2
FKDELE,P51X,ALL
FLST,2,1,1,ORDE,1
FITEM,2,1
FDELE,P51X,ALL
ALLSEL,ALL
FINISH
! SOLVE !
/SOL
LSSOLVE,1,3,1,
FINISH
! DEFINING EACH LOAD CASE !
/POST1
LCDEF,1,1
LCDEF,2,2
LCDEF,3,3
! SAVING DATA FOR POST PROCESSING !
FLST,5,6,6,ORDE,6
FITEM,5,3
FITEM,5,6
FITEM,5,17
FITEM,5,24
```

```
FITEM,5,26

FITEM,5,28

VSEL,U, , ,P51X

VPLOT

/VIEW,1,1,1,1

/ANG,1

/REP,FAST

ESLV,S

EPLOT

ESLV,S

/DIST,1,0.924021086472,1

/REP,FAST

! AXIAL FORCE !

LCASE,1

/EFACET,1

PLNSOL, S,1, 0,1.0

/UI,COPY,SAVE,JPEG,FULL,COLOR,NORM,PORTRAIT,NO,
100

/RENAME,SolveTube000,jpg,,C:\Users\BenJordan\Docu
ments\University\TFG\ANSYS\Results\AxialForce\1stPri
ncipleStress/%D% %DD% %DID%,jpg

/EFACET,1

PLNSOL, S,INT, 0,1.0

/UI,COPY,SAVE,JPEG,FULL,COLOR,NORM,PORTRAIT,NO,
100

/RENAME,SolveTube000,jpg,,C:\Users\BenJordan\Docu
ments\University\TFG\ANSYS\Results\AxialForce\StressI
ntensity/%D% %DD% %DID%,jpg

/EFACET,1

PLNSOL, S,EQV, 0,1.0

/UI,COPY,SAVE,JPEG,FULL,COLOR,NORM,PORTRAIT,NO,
100

/RENAME,SolveTube000,jpg,,C:\Users\BenJordan\Docu
ments\University\TFG\ANSYS\Results\AxialForce\VonMi
ssesStress/%D% %DD% %DID%,jpg

! BENDING MOMENT !

LCASE,2

/EFACET,1

PLNSOL, S,1, 0,1.0

/UI,COPY,SAVE,JPEG,FULL,COLOR,NORM,PORTRAIT,NO,
100

/RENAME,SolveTube000,jpg,,C:\Users\BenJordan\Docu
ments\University\TFG\ANSYS\Results\BendingMoment\
1stPrincipleStress/%D% %DD% %DID%,jpg

/EFACET,1

PLNSOL, S,INT, 0,1.0

/UI,COPY,SAVE,JPEG,FULL,COLOR,NORM,PORTRAIT,NO,
100

/RENAME,SolveTube000,jpg,,C:\Users\BenJordan\Docu
ments\University\TFG\ANSYS\Results\BendingMoment\
StressIntensity/%D% %DD% %DID%,jpg

/EFACET,1

PLNSOL, S,EQV, 0,1.0

/UI,COPY,SAVE,JPEG,FULL,COLOR,NORM,PORTRAIT,NO,
100

/RENAME,SolveTube000,jpg,,C:\Users\BenJordan\Docu
ments\University\TFG\ANSYS\Results\BendingMoment\
VonMissesStress/%D% %DD% %DID%,jpg

! TORSIONAL MOMENT !

LCASE,3

/EFACET,1

PLNSOL, S,1, 0,1.0

/UI,COPY,SAVE,JPEG,FULL,COLOR,NORM,PORTRAIT,NO,
100

/RENAME,SolveTube000,jpg,,C:\Users\BenJordan\Docu
ments\University\TFG\ANSYS\Results\TorsionalMoment
\1stPrincipleStress/%D% %DD% %DID%,jpg

/EFACET,1

PLNSOL, S,INT, 0,1.0

/UI,COPY,SAVE,JPEG,FULL,COLOR,NORM,PORTRAIT,NO,
100

/RENAME,SolveTube000,jpg,,C:\Users\BenJordan\Docu
ments\University\TFG\ANSYS\Results\TorsionalMoment
\StressIntensity/%D% %DD% %DID%,jpg

/EFACET,1

PLNSOL, S,EQV, 0,1.0

/UI,COPY,SAVE,JPEG,FULL,COLOR,NORM,PORTRAIT,NO,
100

/RENAME,SolveTube000,jpg,,C:\Users\BenJordan\Docu
ments\University\TFG\ANSYS\Results\TorsionalMoment
\VonMissesStress/%D% %DD% %DID%,jpg
```

```
SAVE
FINISH
! LOGICAL CONDITION CASE 3 !
*ELSE
! WRITING TITLE !
WPSTYLE,,,,,,,,0
/TITLE, CASE 3: D = %D%, d/D = %DD%, di/D = %DID%
! PREFERENCES !
/NOPR
KEYW,PR_SET,1
KEYW,PR_STRUC,1
KEYW,PR_THERM,0
KEYW,PR_FLUID,0
KEYW,PR_ELMAG,0
KEYW,MAGNOD,0
KEYW,MAGEDG,0
KEYW,MAGHFE,0
KEYW,MAGELC,0
KEYW,PR_MULTI,0
/GO
/PREP7
! ELEMENT TYPE DEFINITION !
ET,1,SOLID187
ET,2,MASS21
R,1,0,0,0,0,0,0,
MPTEMP,,,,,,,,
MPTEMP,1,0
! MATERIAL PROPERTIES !
MPDATA,EX,1,,2.1e11
MPDATA,PRXY,1,,0.3
! KEYPOINT CREATION !
K,1,0,0,0,
K,2,LENG,0,0,
K,3,0,DI/2,0,
K,4,0,D/2,0,
K,5,LENG,D/2,0,
K,6,LENG,DI/2,0,
K,7,(LENG/2)(DSR/2)(0.15),DI/2,0,
K,8,(LENG/2)(DSR/2)(0.15),D/2,0,
K,9,(LENG/2)+(DSR/2)+(0.15),D/2,0,
K,10,(LENG/2)+(DSR/2)+(0.15),DI/2,0,
! LINES FOR HOLLOW PIPE !
LSTR,    1,    2
LSTR,    3,    4
LSTR,    4,    8
LSTR,    8,    7
LSTR,    7,    3
LSTR,    8,    9
LSTR,    9,   10
LSTR,   10,    7
LSTR,   10,    6
LSTR,    6,    5
LSTR,    5,    9
! AREAS CREATION !
FLST,2,4,4
FITEM,2,5
FITEM,2,2
FITEM,2,3
FITEM,2,4
AL,P51X
FLST,2,4,4
FITEM,2,8
FITEM,2,4
FITEM,2,6
FITEM,2,7
AL,P51X
FLST,2,4,4
FITEM,2,9
FITEM,2,10
FITEM,2,11
```

```
FITEM,2,7
AL,P51X
! AREAS REVOLUTION !
FLST,2,3,5,ORDE,2
FITEM,2,1
FITEM,2,3
FLST,8,2,3
FITEM,8,1
FITEM,8,2
VROTAT,P51X, , , , , ,P51X, ,360,4,
! CREATING COORD SYSTEM FOR STRESS RAISER !
LOCAL,11,0,LENG/2,(D/2)+((D/2)(DI/2)),0, ,90, ,1,1,
WPCSYS,1,11,
! STRESS RAISER CYLINDER !
CYL4,0,0,DSR/2, , , ,D+(D/2)
FLST,2,13,6,ORDE,2
FITEM,2,1
FITEM,2,13
VSBV,P51X,      13
! DIVIDING ALL VOLUME BY WORKPLANE !
wpstyle,0.05,0.1,10,10,0.003,0,1,,5
wpro,,,90.000000
FLST,2,12,6,ORDE,10
FITEM,2,1
FITEM,2,3
FITEM,2,4
FITEM,2,6
FITEM,2,7
FITEM,2,9
FITEM,2,10
FITEM,2,12
FITEM,2,14
FITEM,2,17
VSBW,P51X
wpstyle,0.05,0.1,10,10,0.003,0,2,,5
wpro,,,90.000000
! RETURNING TO COORDINATE SYSTEM 0 !
WPSTYLE,,,,,,,,0
WPCSYS,1,0,
CSYS,0
! MESHING !
! MESHING ATTRIBUTES !
FLST,5,54,3,ORDE,2
FITEM,5,1
FITEM,5,54
CM,_Y,KP
KSEL, , , ,P51X
CM,_Y1,KP
CMSEL,S,_Y
CMSEL,S,_Y1
KATT,      1,       1,   1,       0
CMSEL,S,_Y
CMDELE,_Y
CMDELE,_Y1
CM,_Y,KP
KSEL, , , ,       2
CM,_Y1,KP
CMSEL,S,_Y
CMSEL,S,_Y1
KATT,      1,       1,   2,       0
CMSEL,S,_Y
CMDELE,_Y
CMDELE,_Y1
! END SECTION KEYPOINT SIZE !
KSEL,S,LOC,X,0
KSEL,A,LOC,X,LENG
KPLOT
FLST,5,18,3,ORDE,14
FITEM,5,1
FITEM,5,6
```

```
FITEM,5,12
FITEM,5,13
FITEM,5,17
FITEM,5,18
FITEM,5,20
FITEM,5,21
FITEM,5,25
FITEM,5,26
FITEM,5,28
FITEM,5,29
FITEM,5,33
FITEM,5,34
CM,_Y,KP
KSEL, , , ,P51X
CM,_Y1,KP
CMSEL,S,_Y
CMSEL,S,_Y1
KESIZE,ALL,(DD/2)
CMSEL,S,_Y
CMDELE,_Y1
CMDELE,_Y
! STRESS RAISER LINE SIZE !
FLST,5,24,4,ORDE,17
FITEM,5,39
FITEM,5,74
FITEM,5,75
FITEM,5,80
FITEM,5,81
FITEM,5,86
FITEM,5,89
FITEM,5,94
FITEM,5,97
FITEM,5,104
FITEM,5,108
FITEM,5,112
FITEM,5,114
FITEM,5,119
FITEM,5,123
FITEM,5,126
FITEM,5,127
CM,_Y,LINE
LSEL, , , ,P51X
CM,_Y1,LINE
CMSEL,,_Y
LESIZE,_Y1,(DSR/37), , , , , , ,1
! MIDDLE SECTION AREA SIZE !
FLST,5,36,5,ORDE,16
FITEM,5,2
FITEM,5,9
FITEM,5,10
FITEM,5,22
FITEM,5,23
FITEM,5,25
FITEM,5,35
FITEM,5,36
FITEM,5,47
FITEM,5,48
FITEM,5,53
FITEM,5,58
FITEM,5,62
FITEM,5,63
FITEM,5,73
FITEM,5,90
ASEL,S, , ,P51X
FLST,2,36,5,ORDE,16
FITEM,2,2
FITEM,2,9
FITEM,2,10
FITEM,2,22
FITEM,2,23
```

```
FITEM,2,25
FITEM,2,35
FITEM,2,36
FITEM,2,47
FITEM,2,48
FITEM,2,53
FITEM,2,58
FITEM,2,62
FITEM,2,63
FITEM,2,73
FITEM,2,90
AESIZE,P51X,(DSR/20),
! STRESS RAISER AREA SIZE !
FLST,5,8,5,ORDE,8
FITEM,5,9
FITEM,5,35
FITEM,5,53
FITEM,5,56
FITEM,5,76
FITEM,5,81
FITEM,5,84
FITEM,5,88
ASEL,S, , ,P51X
FLST,2,8,5,ORDE,8
FITEM,2,9
FITEM,2,35
FITEM,2,53
FITEM,2,56
FITEM,2,76
FITEM,2,81
FITEM,2,84
FITEM,2,88
AESIZE,P51X,(DSR/37),
! MESHING KEYPOINT 2 (CREATING NODE 1) !
ALLSEL,ALL
KMESH,    2
MSHAPE,1,3D
MSHKEY,0
! MESHING ALL VOLUMES !
FLST,5,16,6,ORDE,4
FITEM,5,1
FITEM,5,13
FITEM,5,18
FITEM,5,20
CM,_Y,VOLU
VSEL, , , ,P51X
CM,_Y1,VOLU
CHKMSH,'VOLU'
CMSEL,S,_Y
VMESH,_Y1
CMDELE,_Y
CMDELE,_Y1
CMDELE,_Y2
! BOUNDARY CONDITIONS !
! EMBEDMENT !
ASEL,S,LOC,X,0
FLST,2,4,5,ORDE,4
FITEM,2,5
FITEM,2,18
FITEM,2,31
FITEM,2,44
/GO
DA,P51X,ALL,
! RIGID REGION !
NSEL,S,LOC,X,LENG
NPLOT
CERIG,1,ALL,ALL, , , ,
ALLSEL,ALL
! SETTING LOADS !
FLST,2,1,3,ORDE,1
```

```
FITEM,2,2
! AXIAL FORCE !
/GO
FK,P51X,FX,LOAD
LSWRITE,1,
FLST,2,1,3,ORDE,1
FITEM,2,2
FKDELE,P51X,ALL
FLST,2,1,1,ORDE,1
FITEM,2,1
FDELE,P51X,ALL
FLST,2,1,3,ORDE,1
FITEM,2,2
! BENDING MOMENT !
/GO
FK,P51X,MZ,LOAD
LSWRITE,2,
FLST,2,1,3,ORDE,1
FITEM,2,2
FKDELE,P51X,ALL
FLST,2,1,1,ORDE,1
FITEM,2,1
FDELE,P51X,ALL
FLST,2,1,3,ORDE,1
FITEM,2,2
! TORSIONAL MOMENT !
/GO
FK,P51X,MX,LOAD
LSWRITE,3,
FLST,2,1,3,ORDE,1
FITEM,2,2
FKDELE,P51X,ALL
FLST,2,1,1,ORDE,1
FITEM,2,1
FDELE,P51X,ALL
ALLSEL,ALL
FINISH
! SOLVE !
/SOL
LSSOLVE,1,3,1,
FINISH
! DEFINING EACH LOAD CASE !
/POST1
LCDEF,1,1
LCDEF,2,2
LCDEF,3,3
! SAVING DATA FOR POST PROCESSING !
 FLST,5,4,6,ORDE,4
FITEM,5,3
FITEM,5,5
FITEM,5,6
FITEM,5,11
VSEL,U, , ,P51X
/VIEW,1,1,1,1
/ANG,1
/REP,FAST
/DIST,1,0.924021086472,1
/REP,FAST
ESLV,S
EPLOT
ESLV,S
! AXIAL FORCE !
LCASE,1
/EFACET,1
PLNSOL, S,1, 0,1.0
/UI,COPY,SAVE,JPEG,FULL,COLOR,NORM,PORTRAIT,NO,
100
/RENAME,SolveTube000,jpg,,C:\Users\BenJordan\Docu
ments\University\TFG\ANSYS\Results\AxialForce\1stPri
ncipleStress/%D% %DD% %DID%,jpg
/EFACET,1
```

```
PLNSOL, S,INT, 0,1.0

/UI,COPY,SAVE,JPEG,FULL,COLOR,NORM,PORTRAIT,NO,
100

/RENAME,SolveTube000,jpg,,C:\Users\BenJordan\Documents\University\TFG\ANSYS\Results\AxialForce\StressIntensity/%D%  %DD%  %DID%,jpg

/EFACET,1

PLNSOL, S,EQV, 0,1.0

/UI,COPY,SAVE,JPEG,FULL,COLOR,NORM,PORTRAIT,NO,
100

/RENAME,SolveTube000,jpg,,C:\Users\BenJordan\Documents\University\TFG\ANSYS\Results\AxialForce\VonMissesStress/%D%  %DD%  %DID%,jpg

!  BENDING MOMENT  !

LCASE,2

/EFACET,1

PLNSOL, S,1, 0,1.0

/UI,COPY,SAVE,JPEG,FULL,COLOR,NORM,PORTRAIT,NO,
100

/RENAME,SolveTube000,jpg,,C:\Users\BenJordan\Documents\University\TFG\ANSYS\Results\BendingMoment\1stPrincipleStress/%D%  %DD%  %DID%,jpg

 /EFACET,1

PLNSOL, S,INT, 0,1.0

/UI,COPY,SAVE,JPEG,FULL,COLOR,NORM,PORTRAIT,NO,
100

/RENAME,SolveTube000,jpg,,C:\Users\BenJordan\Documents\University\TFG\ANSYS\Results\BendingMoment\StressIntensity/%D%  %DD%  %DID%,jpg

/EFACET,1

PLNSOL, S,EQV, 0,1.0

/UI,COPY,SAVE,JPEG,FULL,COLOR,NORM,PORTRAIT,NO,
100

/RENAME,SolveTube000,jpg,,C:\Users\BenJordan\Documents\University\TFG\ANSYS\Results\BendingMoment\VonMissesStress/%D%  %DD%  %DID%,jpg

!  TORSIONAL MOMENT  !

LCASE,3

/EFACET,1

PLNSOL, S,1, 0,1.0

/UI,COPY,SAVE,JPEG,FULL,COLOR,NORM,PORTRAIT,NO,
100

/RENAME,SolveTube000,jpg,,C:\Users\BenJordan\Documents\University\TFG\ANSYS\Results\TorsionalMoment\1stPrincipleStress/%D%  %DD%  %DID%,jpg

/EFACET,1

PLNSOL, S,INT, 0,1.0

/UI,COPY,SAVE,JPEG,FULL,COLOR,NORM,PORTRAIT,NO,
100

/RENAME,SolveTube000,jpg,,C:\Users\BenJordan\Documents\University\TFG\ANSYS\Results\TorsionalMoment\StressIntensity/%D%  %DD%  %DID%,jpg

/EFACET,1

PLNSOL, S,EQV, 0,1.0

/UI,COPY,SAVE,JPEG,FULL,COLOR,NORM,PORTRAIT,NO,
100

/RENAME,SolveTube000,jpg,,C:\Users\BenJordan\Documents\University\TFG\ANSYS\Results\TorsionalMoment\VonMissesStress/%D%  %DD%  %DID%,jpg

SAVE

FINISH

*ENDIF
```

13.2. RESULTS

AXIAL FORCE 1 ST PRINCIPLE STRESS					
D	d/D	di/D	Gross Stress (Pa)	Maximum Stress (Pa)	Kt
0.75	0.1	0.2	285.866	979.316	3.4257886292
0.75	0.1	0.3	301.573	1004.65	3.3313683959
0.75	0.1	0.4	326.704	1078.15	3.3000836903
0.75	0.1	0.5	365.908	1178.69	3.2212715936
0.75	0.1	0.6	428.799	1377.67	3.2128589211
0.75	0.1	0.7	538.100	1717.77	3.1922850959
0.75	0.1	0.8	762.309	2441.96	3.2033731200
0.75	0.1	0.9	1444.375	4355.39	3.0154152790
0.75	0.15	0.3	301.573	1049.78	3.4810171848
0.75	0.15	0.4	326.704	1105.42	3.3835537847
0.75	0.15	0.5	365.908	1214.87	3.3201488271
0.75	0.15	0.6	428.799	1403.8	3.2737965939
0.75	0.15	0.7	538.100	1729.9	3.2148273560
0.75	0.15	0.8	762.309	2504.28	3.2851247510
0.75	0.15	0.9	1444.375	4659.41	3.2259008045
0.75	0.2	0.3	301.573	1043.87	3.4614199248
0.75	0.2	0.4	326.704	1121.82	3.4337521546
0.75	0.2	0.5	365.908	1251.64	3.4206384863
0.75	0.2	0.6	428.799	1465.56	3.4178268529
0.75	0.2	0.7	538.100	1817.93	3.3784213512
0.75	0.2	0.8	762.309	2542.93	3.3358259791
0.75	0.2	0.9	1444.375	4564.32	3.1600660942
0.75	0.25	0.4	326.704	1183.83	3.6235570886
0.75	0.25	0.5	365.908	1289.72	3.5247082776
0.75	0.25	0.6	428.799	1487.72	3.4695061039
0.75	0.25	0.7	538.100	1816.71	3.3761541164
0.75	0.25	0.8	762.309	2603.17	3.4148490576
0.75	0.25	0.9	1444.375	4872.96	3.3737502354
0.75	0.3	0.4	326.704	1254.06	3.8385224251
0.75	0.3	0.5	365.908	1336.77	3.6532924079
0.75	0.3	0.6	428.799	1558.34	3.6341987349
0.75	0.3	0.7	538.100	1928.13	3.5832158333
0.75	0.3	0.8	762.309	2730.99	3.5825238566
0.75	0.3	0.9	1444.375	5572.82	3.8582920416
0.75	0.35	0.5	365.908	1442.71	3.9428185026
0.75	0.35	0.6	428.799	1647.82	3.8428746996
0.75	0.35	0.7	538.100	2012.58	3.7401567953
0.75	0.35	0.8	762.309	2828.41	3.7103198112
0.75	0.35	0.9	1444.375	5617.66	3.8893366142
0.75	0.4	0.5	365.908	1517.65	4.1476239165
0.75	0.4	0.6	428.799	1749.44	4.0798623117
0.75	0.4	0.7	538.100	2133.79	3.9654121418
0.75	0.4	0.8	762.309	3031.14	3.9762618548
0.75	0.4	0.9	1444.375	6092.53	4.2181086079
0.75	0.45	0.6	428.799	1837.84	4.2860196125
0.75	0.45	0.7	538.100	2241.31	4.1652261410
0.75	0.45	0.8	762.309	3133.35	4.1103413510
0.75	0.45	0.9	1444.375	6569.04	4.5480160409
0.75	0.5	0.6	428.799	1976.17	4.6086184748
0.75	0.5	0.7	538.100	2414.1	4.4863371988
0.75	0.5	0.8	762.309	3372.81	4.4244659588
0.75	0.5	0.9	1444.375	7275.98	5.0374596216
0.75	0.55	0.7	538.100	2614.27	4.8583309510
0.75	0.55	0.8	762.309	3644.88	4.7813684981
0.75	0.55	0.9	1444.375	7868.79	5.4478863186
0.75	0.6	0.7	538.100	2855.19	5.3060540602
0.75	0.6	0.8	762.309	3969.59	5.2073244047
0.75	0.6	0.9	1444.375	8386.94	5.8066228328
0.75	0.65	0.8	762.309	4411	5.7863678489
0.75	0.65	0.9	1444.375	8778.05	6.0774043403
0.75	0.7	0.8	762.309	4823.72	6.3277756336
0.75	0.7	0.9	1444.375	9742.31	6.7450011196
0.75	0.75	0.9	1444.375	10633.100	7.3617316022
0.75	0.8	0.9	1444.375	11740.7	8.1285685474

AXIAL FORCE TRESCA					
D	d/D	di/D	Gross Stress (Pa)	Maximum Stress (Pa)	Kt
0.75	0.1	0.2	285.866	931.554	3.2587102638
0.75	0.1	0.3	301.573	964.187	3.1971951422
0.75	0.1	0.4	326.704	1040.56	3.1850253534
0.75	0.1	0.5	365.908	1162.62	3.1773534858
0.75	0.1	0.6	428.799	1374.67	3.2058626326
0.75	0.1	0.7	538.100	1692.51	3.1453421865
0.75	0.1	0.8	762.309	2421.11	3.1760220047
0.75	0.1	0.9	1444.375	4297.44	2.9752941152
0.75	0.15	0.3	301.573	1004.61	3.3312357580
0.75	0.15	0.4	326.704	1095.98	3.3546591132
0.75	0.15	0.5	365.908	1201.87	3.2846207996
0.75	0.15	0.6	428.799	1384.95	3.2298365812
0.75	0.15	0.7	538.100	1706.4	3.1711552115
0.75	0.15	0.8	762.309	2479.84	3.2530642590
0.75	0.15	0.9	1444.375	4646.97	3.2172880819
0.75	0.2	0.3	301.573	1030.77	3.4179809899
0.75	0.2	0.4	326.704	1121.29	3.4321298902
0.75	0.2	0.5	365.908	1241.93	3.3941017827
0.75	0.2	0.6	428.799	1456.52	3.3967447036
0.75	0.2	0.7	538.100	1811.41	3.3663046540
0.75	0.2	0.8	762.309	2525.58	3.3130661781
0.75	0.2	0.9	1444.375	4528.42	3.1352110506
0.75	0.25	0.4	326.704	1153.38	3.5303534079
0.75	0.25	0.5	365.908	1288.76	3.5220846694
0.75	0.25	0.6	428.799	1487.12	3.4681068462
0.75	0.25	0.7	538.100	1811.47	3.3664161574
0.75	0.25	0.8	762.309	2590.6	3.3983596802
0.75	0.25	0.9	1444.375	4863.94	3.3675053192
0.75	0.3	0.4	326.704	1247.65	3.8189022086
0.75	0.3	0.5	365.908	1323.5	3.6170264906
0.75	0.3	0.6	428.799	1558.92	3.6355513507
0.75	0.3	0.7	538.100	1923.39	3.5744070688
0.75	0.3	0.8	762.309	2723.41	3.5725803816
0.75	0.3	0.9	1444.375	5570.34	3.8565750358
0.75	0.35	0.5	365.908	1412.12	3.8592183210
0.75	0.35	0.6	428.799	1619.96	3.7779025005
0.75	0.35	0.7	538.100	2008.65	3.7328533260
0.75	0.35	0.8	762.309	2805.02	3.6796367135
0.75	0.35	0.9	1444.375	5612.81	3.8859787601
0.75	0.4	0.5	365.908	1513.76	4.1369928375
0.75	0.4	0.6	428.799	1726.01	4.0252212986
0.75	0.4	0.7	538.100	2135.7	3.9689616650
0.75	0.4	0.8	762.309	3022.58	3.9650328117
0.75	0.4	0.9	1444.375	6093.68	4.2189048001
0.75	0.45	0.6	428.799	1838.47	4.2874888331
0.75	0.45	0.7	538.100	2230.9	4.1458803102
0.75	0.45	0.8	762.309	3108.08	4.0771920616
0.75	0.45	0.9	1444.375	6560.5	4.5421034483
0.75	0.5	0.6	428.799	1972.41	4.5998497932
0.75	0.5	0.7	538.100	2413.98	4.4861141921
0.75	0.5	0.8	762.309	3344.48	4.3873025489
0.75	0.5	0.9	1444.375	7272.14	5.0348010320
0.75	0.55	0.7	538.100	2616.8	4.8630326755
0.75	0.55	0.8	762.309	3644.57	4.7809618388
0.75	0.55	0.9	1444.375	7873.27	5.4509880065
0.75	0.6	0.7	538.100	2849.33	5.2951638999
0.75	0.6	0.8	762.309	3972.21	5.2107613314
0.75	0.6	0.9	1444.375	8366.57	5.7925198457
0.75	0.65	0.8	762.309	4342.52	5.6965355047
0.75	0.65	0.9	1444.375	8753.25	6.0602342823
0.75	0.7	0.8	762.309	4797.75	6.2937080814
0.75	0.7	0.9	1444.375	9692.44	6.7104740716
0.75	0.75	0.9	1444.375	10506.400	7.2740119914
0.75	0.8	0.9	1444.375	11711.2	8.1081444865

AXIAL FORCE VON MISSES					
D	d/D	di/D	Gross Stress (Pa)	Maximum Stress (Pa)	Kt
0.75	0.1	0.2	285.866	889.341	3.1110431008
0.75	0.1	0.3	301.573	936.452	3.1052272902
0.75	0.1	0.4	326.704	993.774	3.0418191989
0.75	0.1	0.5	365.908	1117.67	3.0545084985
0.75	0.1	0.6	428.799	1303.08	3.0389078683
0.75	0.1	0.7	538.100	1618.03	3.0069293641
0.75	0.1	0.8	762.309	2331.7	3.0587336008
0.75	0.1	0.9	1444.375	4256.39	2.9468735152
0.75	0.15	0.3	301.573	963.922	3.1963164156
0.75	0.15	0.4	326.704	1040.93	3.1861578776
0.75	0.15	0.5	365.908	1147	3.1346651943
0.75	0.15	0.6	428.799	1340.47	3.1261049438
0.75	0.15	0.7	538.100	1668.35	3.1004435051
0.75	0.15	0.8	762.309	2432.51	3.1909765713
0.75	0.15	0.9	1444.375	4592.45	3.1795416479
0.75	0.2	0.3	301.573	999.281	3.3135650645
0.75	0.2	0.4	326.704	1076.52	3.2950944621
0.75	0.2	0.5	365.908	1196.12	3.2689064797
0.75	0.2	0.6	428.799	1422.39	3.3171502615
0.75	0.2	0.7	538.100	1777.66	3.3035840209
0.75	0.2	0.8	762.309	2483.94	3.2584426557
0.75	0.2	0.9	1444.375	4496.65	3.1132153755
0.75	0.25	0.4	326.704	1129.08	3.4559741159
0.75	0.25	0.5	365.908	1251.11	3.4191900360
0.75	0.25	0.6	428.799	1458.96	3.4024350182
0.75	0.25	0.7	538.100	1791.81	3.3298802271
0.75	0.25	0.8	762.309	2563.73	3.3631115042
0.75	0.25	0.9	1444.375	4831.71	3.3451911672
0.75	0.3	0.4	326.704	1212.72	3.7119858024
0.75	0.3	0.5	365.908	1312.17	3.5860624482
0.75	0.3	0.6	428.799	1532.92	3.5749168505
0.75	0.3	0.7	538.100	1894.2	3.5201606901
0.75	0.3	0.8	762.309	2677.15	3.5118963243
0.75	0.3	0.9	1444.375	5554.73	3.8457675920
0.75	0.35	0.5	365.908	1402.05	3.8316977643
0.75	0.35	0.6	428.799	1609.03	3.7524126894
0.75	0.35	0.7	538.100	1992.51	3.7028589255
0.75	0.35	0.8	762.309	2786.47	3.6553027476
0.75	0.35	0.9	1444.375	5591.58	3.8712803596
0.75	0.4	0.5	365.908	1492.89	4.0799566887
0.75	0.4	0.6	428.799	1714.85	3.9991951054
0.75	0.4	0.7	538.100	2109.87	3.9209594738
0.75	0.4	0.8	762.309	2986.24	3.9173618511
0.75	0.4	0.9	1444.375	6064.01	4.1983630412
0.75	0.45	0.6	428.799	1814.44	4.2314485623
0.75	0.45	0.7	538.100	2217.59	4.1211451509
0.75	0.45	0.8	762.309	3076.98	4.0363949543
0.75	0.45	0.9	1444.375	6510.4	4.5074171618
0.75	0.5	0.6	428.799	1958.4	4.5671771260
0.75	0.5	0.7	538.100	2397.91	4.4562498788
0.75	0.5	0.8	762.309	3333.22	4.3725316348
0.75	0.5	0.9	1444.375	7223.1	5.0008486270
0.75	0.55	0.7	538.100	2585.52	4.8049022635
0.75	0.55	0.8	762.309	3598	4.7198711222
0.75	0.55	0.9	1444.375	7818.41	5.4130061765
0.75	0.6	0.7	538.100	2822.67	5.2456192457
0.75	0.6	0.8	762.309	3922.91	5.1460893897
0.75	0.6	0.9	1444.375	8298.02	5.7450598668
0.75	0.65	0.8	762.309	4306.28	5.6489957244
0.75	0.65	0.9	1444.375	8625.77	5.9719746454
0.75	0.7	0.8	762.309	4771.71	6.2595487028
0.75	0.7	0.9	1444.375	9576.22	6.6300101949
0.75	0.75	0.9	1444.375	10374.000	7.1823460365
0.75	0.8	0.9	1444.375	11574.1	8.0132245288

BENDING MOMENT 1 ST PRINCIPLE STRESS					
D	d/D	di/D	Gross Stress (Pa)	Maximum Stress (Pa)	Kt
0.75	0.1	0.2	2931.958	8900.69	3.0357500060
0.75	0.1	0.3	2951.171	8992.62	3.0471363569
0.75	0.1	0.4	3004.173	9021.5	3.0029893058
0.75	0.1	0.5	3122.417	9420.9	3.0171814376
0.75	0.1	0.6	3363.128	10130.7	3.0122852415
0.75	0.1	0.7	3852.173	11505.7	2.9868075977
0.75	0.1	0.8	4958.107	14929.6	3.0111492192
0.75	0.1	0.9	8511.970	25320.4	2.9746816530
0.75	0.15	0.3	2951.171	8962.87	3.0370556122
0.75	0.15	0.4	3004.173	9197.92	3.0617142821
0.75	0.15	0.5	3122.417	9546.01	3.0572497505
0.75	0.15	0.6	3363.128	10210.1	3.0358942170
0.75	0.15	0.7	3852.173	11677.1	3.0313019633
0.75	0.15	0.8	4958.107	15275.8	3.0809742553
0.75	0.15	0.9	8511.970	26424.9	3.1044401041
0.75	0.2	0.3	2951.171	9396.83	3.1841023342
0.75	0.2	0.4	3004.173	9428.42	3.1384408835
0.75	0.2	0.5	3122.417	9767.72	3.1282556307
0.75	0.2	0.6	3363.128	10580	3.1458811193
0.75	0.2	0.7	3852.173	12061.9	3.1311936312
0.75	0.2	0.8	4958.107	15572.1	3.1407349665
0.75	0.2	0.9	8511.970	26490.1	3.1120999059
0.75	0.25	0.4	3004.173	9971.91	3.3193525565
0.75	0.25	0.5	3122.417	10255.7	3.2845383848
0.75	0.25	0.6	3363.128	11015.2	3.2752844712
0.75	0.25	0.7	3852.173	12483.5	3.2406383484
0.75	0.25	0.8	4958.107	16065.7	3.2402890908
0.75	0.25	0.9	8511.970	28006.1	3.2902020443
0.75	0.3	0.4	3004.173	10716	3.5670380093
0.75	0.3	0.5	3122.417	10890.4	3.4878103714
0.75	0.3	0.6	3363.128	11659.5	3.4668620899
0.75	0.3	0.7	3852.173	13135.7	3.4099453802
0.75	0.3	0.8	4958.107	16873.6	3.4032343442
0.75	0.3	0.9	8511.970	31113.4	3.6552526873
0.75	0.35	0.5	3122.417	11886.5	3.8068260100
0.75	0.35	0.6	3363.128	12454.9	3.7033681241
0.75	0.35	0.7	3852.173	14006.3	3.6359476829
0.75	0.35	0.8	4958.107	17723.8	3.5747110794
0.75	0.35	0.9	8511.970	32731	3.8452909585
0.75	0.4	0.5	3122.417	13011.9	4.1672518706
0.75	0.4	0.6	3363.128	13669.8	4.0646092367
0.75	0.4	0.7	3852.173	15125.2	3.9264071092
0.75	0.4	0.8	4958.107	19095.8	3.8514295935
0.75	0.4	0.9	8511.970	35822.5	4.2084853918
0.75	0.45	0.6	3363.128	15098.4	4.4893923905
0.75	0.45	0.7	3852.173	16459.3	4.2727311065
0.75	0.45	0.8	4958.107	20427.8	4.1200805125
0.75	0.45	0.9	8511.970	38939.8	4.5747108510
0.75	0.5	0.6	3363.128	17195	5.1128001745
0.75	0.5	0.7	3852.173	18579.9	4.8232255738
0.75	0.5	0.8	4958.107	22615.5	4.5613174611
0.75	0.5	0.9	8511.970	43674.6	5.1309628332
0.75	0.55	0.7	3852.173	21187.9	5.5002460258
0.75	0.55	0.8	4958.107	25277.5	5.0982159193
0.75	0.55	0.9	8511.970	47874.3	5.6243503997
0.75	0.6	0.7	3852.173	25097.2	6.5150758007
0.75	0.6	0.8	4958.107	29102.7	5.8697200448
0.75	0.6	0.9	8511.970	52489.4	6.1665398318
0.75	0.65	0.8	4958.107	34601.1	6.9786916761
0.75	0.65	0.9	8511.970	58063.4	6.8213823909
0.75	0.7	0.8	4958.107	43589.6	8.7915811545
0.75	0.7	0.9	8511.970	67085.9	7.8813603224
0.75	0.75	0.9	8511.970	80512.100	9.4586920711
0.75	0.8	0.9	8511.970	104478	12.2742448675

BENDING MOMENT TRESCA					
D	d/D	di/D	Gross Stress (Pa)	Maximum Stress (Pa)	Kt
0.75	0.1	0.2	2931.958	8879.03	3.0283624501
0.75	0.1	0.3	2951.171	8912.01	3.0198217743
0.75	0.1	0.4	3004.173	8812.81	2.9335226053
0.75	0.1	0.5	3122.417	9291.31	2.9756783389
0.75	0.1	0.6	3363.128	9942.24	2.9562481191
0.75	0.1	0.7	3852.173	11411.9	2.9624577057
0.75	0.1	0.8	4958.107	14799.7	2.9849497039
0.75	0.1	0.9	8511.970	25150.4	2.9547097773
0.75	0.15	0.3	2951.171	8956.62	3.0349378087
0.75	0.15	0.4	3004.173	9142.86	3.0433864440
0.75	0.15	0.5	3122.417	9332.94	2.9890109571
0.75	0.15	0.6	3363.128	10160.9	3.0212649778
0.75	0.15	0.7	3852.173	11591.8	3.0091586180
0.75	0.15	0.8	4958.107	15209.8	3.0676627233
0.75	0.15	0.9	8511.970	26415.1	3.1032887842
0.75	0.2	0.3	2951.171	9281.27	3.1449449944
0.75	0.2	0.4	3004.173	9426.7	3.1378683466
0.75	0.2	0.5	3122.417	9752.66	3.1234324448
0.75	0.2	0.6	3363.128	10540.9	3.1342550369
0.75	0.2	0.7	3852.173	12055.2	3.1294543532
0.75	0.2	0.8	4958.107	15401.9	3.1064073491
0.75	0.2	0.9	8511.970	26447.4	3.1070834406
0.75	0.25	0.4	3004.173	9947.73	3.3113037529
0.75	0.25	0.5	3122.417	10237.6	3.2787415943
0.75	0.25	0.6	3363.128	10995.4	3.2693970945
0.75	0.25	0.7	3852.173	12457	3.2337591146
0.75	0.25	0.8	4958.107	16077.2	3.2426085245
0.75	0.25	0.9	8511.970	27994.5	3.2888392575
0.75	0.3	0.4	3004.173	10766.1	3.5837148108
0.75	0.3	0.5	3122.417	10871.4	3.4817253426
0.75	0.3	0.6	3363.128	11661.1	3.4673378375
0.75	0.3	0.7	3852.173	13134.8	3.4097117458
0.75	0.3	0.8	4958.107	16848.2	3.3981114213
0.75	0.3	0.9	8511.970	31121.7	3.6562277847
0.75	0.35	0.5	3122.417	11891.3	3.8083632804
0.75	0.35	0.6	3363.128	12448	3.7013164625
0.75	0.35	0.7	3852.173	13983.7	3.6300808646
0.75	0.35	0.8	4958.107	17719.3	3.5738034749
0.75	0.35	0.9	8511.970	32712.6	3.8431292966
0.75	0.4	0.5	3122.417	13093.4	4.1933534413
0.75	0.4	0.6	3363.128	13668.3	4.0641632234
0.75	0.4	0.7	3852.173	15140.7	3.9304308120
0.75	0.4	0.8	4958.107	19070.1	3.8462461637
0.75	0.4	0.9	8511.970	35917.2	4.2196109014
0.75	0.45	0.6	3363.128	15160.8	4.5079465476
0.75	0.45	0.7	3852.173	16469.6	4.2754049220
0.75	0.45	0.8	4958.107	20419.8	4.1184669935
0.75	0.45	0.9	8511.970	38960.5	4.5771427206
0.75	0.5	0.6	3363.128	17151.3	5.0998063177
0.75	0.5	0.7	3852.173	18600.8	4.8286510828
0.75	0.5	0.8	4958.107	22623.2	4.5628704731
0.75	0.5	0.9	8511.970	43652	5.1283077486
0.75	0.55	0.7	3852.173	21192.4	5.5014141976
0.75	0.55	0.8	4958.107	25287.9	5.1003134940
0.75	0.55	0.9	8511.970	47930.2	5.6309176224
0.75	0.6	0.7	3852.173	25043.4	6.5011096580
0.75	0.6	0.8	4958.107	29127.8	5.8747824608
0.75	0.6	0.9	8511.970	52481	6.1655529862
0.75	0.65	0.8	4958.107	34569.3	6.9722779380
0.75	0.65	0.9	8511.970	57684.5	6.7768686045
0.75	0.7	0.8	4958.107	43660.9	8.8059616429
0.75	0.7	0.9	8511.970	67677.3	7.9508389534
0.75	0.75	0.9	8511.970	79534.200	9.3438067932
0.75	0.8	0.9	8511.970	105133	12.3511953297

BENDING MOMENT VON MISSES					
D	d/D	di/D	Gross Stress (Pa)	Maximum Stress (Pa)	Kt
0.75	0.1	0.2	2931.958	8675.76	2.9590333415
0.75	0.1	0.3	2951.171	8681.38	2.9416731304
0.75	0.1	0.4	3004.173	8596.85	2.8616359379
0.75	0.1	0.5	3122.417	9100.66	2.9146198794
0.75	0.1	0.6	3363.128	9750.54	2.8992476077
0.75	0.1	0.7	3852.173	11195.4	2.9062556628
0.75	0.1	0.8	4958.107	14443.4	2.9130876000
0.75	0.1	0.9	8511.970	24942	2.9302266074
0.75	0.15	0.3	2951.171	8772.65	2.9725998387
0.75	0.15	0.4	3004.173	8982.71	2.9900772673
0.75	0.15	0.5	3122.417	9244.42	2.9606611285
0.75	0.15	0.6	3363.128	10026.3	2.9812427095
0.75	0.15	0.7	3852.173	11496.8	2.9844972135
0.75	0.15	0.8	4958.107	15081.1	3.0417052359
0.75	0.15	0.9	8511.970	26126.2	3.0693483437
0.75	0.2	0.3	2951.171	9187.63	3.1132152150
0.75	0.2	0.4	3004.173	9315.57	3.1008764715
0.75	0.2	0.5	3122.417	9649.64	3.0904387784
0.75	0.2	0.6	3363.128	10363.3	3.0814470514
0.75	0.2	0.7	3852.173	11897.2	3.0884385436
0.75	0.2	0.8	4958.107	15365.6	3.0990860065
0.75	0.2	0.9	8511.970	26218.3	3.0801684011
0.75	0.25	0.4	3004.173	9830.48	3.2722747116
0.75	0.25	0.5	3122.417	10146.2	3.2494694033
0.75	0.25	0.6	3363.128	10904.2	3.2422794803
0.75	0.25	0.7	3852.173	12380.4	3.2138742347
0.75	0.25	0.8	4958.107	15991.7	3.2253640398
0.75	0.25	0.9	8511.970	27747	3.2597625561
0.75	0.3	0.4	3004.173	10655.2	3.5467994958
0.75	0.3	0.5	3122.417	10787.5	3.4548551368
0.75	0.3	0.6	3363.128	11593.6	3.4472672349
0.75	0.3	0.7	3852.173	13066.5	3.3919814940
0.75	0.3	0.8	4958.107	16767	3.3817342031
0.75	0.3	0.9	8511.970	31033.1	3.6458189131
0.75	0.35	0.5	3122.417	11807.8	3.7816211804
0.75	0.35	0.6	3363.128	12363.2	3.6761018388
0.75	0.35	0.7	3852.173	13920	3.6135447439
0.75	0.35	0.8	4958.107	17638.6	3.5575271017
0.75	0.35	0.9	8511.970	32583.5	3.8279624193
0.75	0.4	0.5	3122.417	12981.4	4.1574837981
0.75	0.4	0.6	3363.128	13580.6	4.0380863071
0.75	0.4	0.7	3852.173	15061	3.9097411916
0.75	0.4	0.8	4958.107	18989.9	3.8300706353
0.75	0.4	0.9	8511.970	35726.6	4.1972189043
0.75	0.45	0.6	3363.128	15050.9	4.4752686331
0.75	0.45	0.7	3852.173	16406.8	4.2591024356
0.75	0.45	0.8	4958.107	20334.9	4.1013435228
0.75	0.45	0.9	8511.970	38741.8	4.5514494899
0.75	0.5	0.6	3363.128	17060.7	5.0728671089
0.75	0.5	0.7	3852.173	18540.7	4.8130494995
0.75	0.5	0.8	4958.107	22531.2	4.5443150042
0.75	0.5	0.9	8511.970	43326.3	5.0900439844
0.75	0.55	0.7	3852.173	21096.7	5.4765710775
0.75	0.55	0.8	4958.107	25195.6	5.0816975182
0.75	0.55	0.9	8511.970	47556.5	5.5870147404
0.75	0.6	0.7	3852.173	24861.3	6.4538376394
0.75	0.6	0.8	4958.107	28991.6	5.8473122993
0.75	0.6	0.9	8511.970	51872.7	6.0940889158
0.75	0.65	0.8	4958.107	34444.6	6.9471272101
0.75	0.65	0.9	8511.970	56877.4	6.6820491877
0.75	0.7	0.8	4958.107	42949.5	8.6624794629
0.75	0.7	0.9	8511.970	66835.7	7.8519664206
0.75	0.75	0.9	8511.970	79150.200	9.2986938505
0.75	0.8	0.9	8511.970	104493	12.2760070919

TORSIONAL MOMENT 1 ST PRINCIPLE STRESS					
D	d/D	di/D	Gross Stress (Pa)	Maximum Stress (Pa)	Kt
0.75	0.1	0.2	1465.979	5914.35	4.0344036357
0.75	0.1	0.3	1475.585	5848.32	3.9633896459
0.75	0.1	0.4	1502.087	6096.25	4.0585209900
0.75	0.1	0.5	1561.209	6253.24	4.0053837006
0.75	0.1	0.6	1681.564	6820.23	4.0558852148
0.75	0.1	0.7	1926.087	7775.71	4.0370511495
0.75	0.1	0.8	2479.053	10152.2	4.0951919814
0.75	0.1	0.9	4255.985	18270.8	4.2929664260
0.75	0.15	0.3	1475.585	5935.44	4.0224306194
0.75	0.15	0.4	1502.087	6016.71	4.0055679845
0.75	0.15	0.5	1561.209	6306.82	4.0397032628
0.75	0.15	0.6	1681.564	6784.08	4.0343873694
0.75	0.15	0.7	1926.087	8497.53	4.4118110442
0.75	0.15	0.8	2479.053	10524.8	4.2454912794
0.75	0.15	0.9	4255.985	20351.2	4.7817839574
0.75	0.2	0.3	1475.585	5891.87	3.9929033557
0.75	0.2	0.4	1502.087	6060.14	4.0344810978
0.75	0.2	0.5	1561.209	6251.54	4.0042948008
0.75	0.2	0.6	1681.564	6791.37	4.0387226195
0.75	0.2	0.7	1926.087	7943.77	4.1243057946
0.75	0.2	0.8	2479.053	10842.2	4.3735240146
0.75	0.2	0.9	4255.985	23455.8	5.5112508425
0.75	0.25	0.4	1502.087	6292.54	4.1891992078
0.75	0.25	0.5	1561.209	6366.1	4.0776738421
0.75	0.25	0.6	1681.564	6875.26	4.0886107040
0.75	0.25	0.7	1926.087	8109.93	4.2105739834
0.75	0.25	0.8	2479.053	11976.3	4.8309969984
0.75	0.25	0.9	4255.985	27343.2	6.4246469546
0.75	0.3	0.4	1502.087	6479.59	4.3137259826
0.75	0.3	0.5	1561.209	6546.84	4.1934431154
0.75	0.3	0.6	1681.564	7201.6	4.2826800508
0.75	0.3	0.7	1926.087	8458.56	4.3915783087
0.75	0.3	0.8	2479.053	13018.8	5.2515203964
0.75	0.3	0.9	4255.985	32165.9	7.5578041881
0.75	0.35	0.5	1561.209	6895.54	4.4167956968
0.75	0.35	0.6	1681.564	7437.06	4.4227044683
0.75	0.35	0.7	1926.087	8934.99	4.6389347918
0.75	0.35	0.8	2479.053	14678.6	5.9210501191
0.75	0.35	0.9	4255.985	35607.1	8.3663597010
0.75	0.4	0.5	1561.209	7218.84	4.6238788330
0.75	0.4	0.6	1681.564	7841.62	4.6632900383
0.75	0.4	0.7	1926.087	9500.68	4.9326339479
0.75	0.4	0.8	2479.053	16117.7	6.5015539291
0.75	0.4	0.9	4255.985	40647	9.5505509510
0.75	0.45	0.6	1681.564	8267.83	4.9167505283
0.75	0.45	0.7	1926.087	10347.8	5.3724480317
0.75	0.45	0.8	2479.053	17592.3	7.0963777206
0.75	0.45	0.9	4255.985	44589.3	10.4768465451
0.75	0.5	0.6	1681.564	8904.32	5.2952612795
0.75	0.5	0.7	1926.087	10898.4	5.6583126490
0.75	0.5	0.8	2479.053	19076.5	7.6950739578
0.75	0.5	0.9	4255.985	50836.9	11.9448029041
0.75	0.55	0.7	1926.087	11710.9	6.0801524628
0.75	0.55	0.8	2479.053	20289.7	8.1844542804
0.75	0.55	0.9	4255.985	56008.4	13.1599153169
0.75	0.6	0.7	1926.087	12561.4	6.5217214003
0.75	0.6	0.8	2479.053	21719.2	8.7610856448
0.75	0.6	0.9	4255.985	60283.6	14.1644301747
0.75	0.65	0.8	2479.053	23268.6	9.3860822422
0.75	0.65	0.9	4255.985	64126.3	15.0673234298
0.75	0.7	0.8	2479.053	24466.3	9.8692101786
0.75	0.7	0.9	4255.985	69149	16.2474733120
0.75	0.75	0.9	4255.985	70524.700	16.5707122458
0.75	0.8	0.9	4255.985	70490.1	16.5625825176

TORSIONAL MOMENT TRESCA					
D	d/D	di/D	Gross Stress (Pa)	Maximum Stress (Pa)	Kt
0.75	0.1	0.2	1465.979	5728.79	3.9078260847
0.75	0.1	0.3	1475.585	5551.44	3.7621949237
0.75	0.1	0.4	1502.087	5893.8	3.9237418103
0.75	0.1	0.5	1561.209	6103.3	3.9093427311
0.75	0.1	0.6	1681.564	6644.6	3.9514407723
0.75	0.1	0.7	1926.087	7587.38	3.9392725745
0.75	0.1	0.8	2479.053	9984.24	4.0274403172
0.75	0.1	0.9	4255.985	17995.7	4.2283280377
0.75	0.15	0.3	1475.585	5815.76	3.9413238276
0.75	0.15	0.4	1502.087	5826.49	3.8789308120
0.75	0.15	0.5	1561.209	6116.98	3.9181051725
0.75	0.15	0.6	1681.564	6664.09	3.9630311736
0.75	0.15	0.7	1926.087	8651.67	4.4918385998
0.75	0.15	0.8	2479.053	10397	4.1939393463
0.75	0.15	0.9	4255.985	20286.3	4.7665348429
0.75	0.2	0.3	1475.585	5774.91	3.9136399000
0.75	0.2	0.4	1502.087	6047.51	4.0260727943
0.75	0.2	0.5	1561.209	6221.17	3.9848419247
0.75	0.2	0.6	1681.564	6782.43	4.0334061399
0.75	0.2	0.7	1926.087	7859.53	4.0805694427
0.75	0.2	0.8	2479.053	10724.4	4.3260058791
0.75	0.2	0.9	4255.985	23374.4	5.4921248345
0.75	0.25	0.4	1502.087	5999	3.9937777189
0.75	0.25	0.5	1561.209	6341.86	4.0621474109
0.75	0.25	0.6	1681.564	6812.97	4.0515678052
0.75	0.25	0.7	1926.087	8038.65	4.1735663011
0.75	0.25	0.8	2479.053	11910.2	4.8043335964
0.75	0.25	0.9	4255.985	27469.4	6.4542993159
0.75	0.3	0.4	1502.087	6145.47	4.0912887412
0.75	0.3	0.5	1561.209	6529.94	4.1826181695
0.75	0.3	0.6	1681.564	7084.67	4.2131435897
0.75	0.3	0.7	1926.087	8381.38	4.3515074203
0.75	0.3	0.8	2479.053	13139	5.3000066433
0.75	0.3	0.9	4255.985	32113.3	7.5454451215
0.75	0.35	0.5	1561.209	6831.06	4.3754943649
0.75	0.35	0.6	1681.564	7432	4.4196953646
0.75	0.35	0.7	1926.087	8805.27	4.5715857941
0.75	0.35	0.8	2479.053	14663	5.9147573948
0.75	0.35	0.9	4255.985	35822.3	8.4169237910
0.75	0.4	0.5	1561.209	7795.04	4.9929518397
0.75	0.4	0.6	1681.564	7900.23	4.6981444982
0.75	0.4	0.7	1926.087	9505.78	4.9352818040
0.75	0.4	0.8	2479.053	16135.6	6.5087744268
0.75	0.4	0.9	4255.985	40730.5	9.5701703818
0.75	0.45	0.6	1681.564	8999.53	5.3518811928
0.75	0.45	0.7	1926.087	10289.9	5.3423870776
0.75	0.45	0.8	2479.053	17520.8	7.0675360679
0.75	0.45	0.9	4255.985	45015.4	10.5769643831
0.75	0.5	0.6	1681.564	10143.4	6.0321229765
0.75	0.5	0.7	1926.087	11621.6	6.0337890223
0.75	0.5	0.8	2479.053	18870.3	7.6118970516
0.75	0.5	0.9	4255.985	50732.1	11.9201787562
0.75	0.55	0.7	1926.087	13524.5	7.0217508461
0.75	0.55	0.8	2479.053	20144.4	8.1258432016
0.75	0.55	0.9	4255.985	55906	13.1358550808
0.75	0.6	0.7	1926.087	15704.2	8.1534237596
0.75	0.6	0.8	2479.053	22016	8.8808087571
0.75	0.6	0.9	4255.985	60184.9	14.1412393026
0.75	0.65	0.8	2479.053	26824.7	10.8205410004
0.75	0.65	0.9	4255.985	63885	15.0106267992
0.75	0.7	0.8	2479.053	32151.4	12.9692239586
0.75	0.7	0.9	4255.985	73005.3	17.1535620672
0.75	0.75	0.9	4255.985	85527.000	20.0957013110
0.75	0.8	0.9	4255.985	99228.6	23.3150736856

TORSIONAL MOMENT VON MISSES					
D	d/D	di/D	Gross Stress (Pa)	Maximum Stress (Pa)	Kt
0.75	0.1	0.2	1465.979	5414.35	3.6933345719
0.75	0.1	0.3	1475.585	5289.85	3.5849161329
0.75	0.1	0.4	1502.087	5592.45	3.7231208875
0.75	0.1	0.5	1561.209	5856.42	3.7512088473
0.75	0.1	0.6	1681.564	6351.48	3.7771268453
0.75	0.1	0.7	1926.087	7148.54	3.7114323482
0.75	0.1	0.8	2479.053	9487.08	3.8268960366
0.75	0.1	0.9	4255.985	17824.8	4.1881728194
0.75	0.15	0.3	1475.585	5530.57	3.7480513847
0.75	0.15	0.4	1502.087	5617.33	3.7396845130
0.75	0.15	0.5	1561.209	5815	3.7246781219
0.75	0.15	0.6	1681.564	6310.2	3.7525782683
0.75	0.15	0.7	1926.087	8001.7	4.1543823243
0.75	0.15	0.8	2479.053	9926.27	4.0040563927
0.75	0.15	0.9	4255.985	20259.3	4.7601908353
0.75	0.2	0.3	1475.585	5515.97	3.7381570067
0.75	0.2	0.4	1502.087	5868.34	3.9067920552
0.75	0.2	0.5	1561.209	5952	3.8124306417
0.75	0.2	0.6	1681.564	6564.61	3.9038719574
0.75	0.2	0.7	1926.087	7557.44	3.9237281019
0.75	0.2	0.8	2479.053	10443	4.2124948151
0.75	0.2	0.9	4255.985	23341	5.4842770622
0.75	0.25	0.4	1502.087	5841.57	3.8889701799
0.75	0.25	0.5	1561.209	6184.19	3.9611551496
0.75	0.25	0.6	1681.564	6636.77	3.9467843925
0.75	0.25	0.7	1926.087	7811.15	4.0554511532
0.75	0.25	0.8	2479.053	11877	4.7909413884
0.75	0.25	0.9	4255.985	27462.3	6.4526310769
0.75	0.3	0.4	1502.087	6101.45	4.0619828410
0.75	0.3	0.5	1561.209	6383.24	4.0886525151
0.75	0.3	0.6	1681.564	6920.99	4.1158056272
0.75	0.3	0.7	1926.087	8176.64	4.2452089791
0.75	0.3	0.8	2479.053	13116.3	5.2908499228
0.75	0.3	0.9	4255.985	32070.4	7.5353651984
0.75	0.35	0.5	1561.209	6586	4.2185262444
0.75	0.35	0.6	1681.564	7237.93	4.3042849395
0.75	0.35	0.7	1926.087	8687.64	4.5105137728
0.75	0.35	0.8	2479.053	14638.6	5.9049149287
0.75	0.35	0.9	4255.985	35692.5	8.3864255620
0.75	0.4	0.5	1561.209	7027.71	4.5014544599
0.75	0.4	0.6	1681.564	7561.74	4.4968497344
0.75	0.4	0.7	1926.087	9500.35	4.9324626160
0.75	0.4	0.8	2479.053	16073.3	6.4836438679
0.75	0.4	0.9	4255.985	40490.4	9.5137557071
0.75	0.45	0.6	1681.564	8066.21	4.7968502350
0.75	0.45	0.7	1926.087	10098.1	5.2428069222
0.75	0.45	0.8	2479.053	17458.9	7.0425668609
0.75	0.45	0.9	4255.985	44599.3	10.4791961776
0.75	0.5	0.6	1681.564	8785.91	5.2248446853
0.75	0.5	0.7	1926.087	10824.5	5.6199446954
0.75	0.5	0.8	2479.053	18738.1	7.5585702476
0.75	0.5	0.9	4255.985	50384.9	11.8385995181
0.75	0.55	0.7	1926.087	11712.9	6.0811908377
0.75	0.55	0.8	2479.053	19964.3	8.0531945071
0.75	0.55	0.9	4255.985	55694.9	13.0862543401
0.75	0.6	0.7	1926.087	13601.1	7.0615206058
0.75	0.6	0.8	2479.053	21656.6	8.7358340719
0.75	0.6	0.9	4255.985	60018.5	14.1021414189
0.75	0.65	0.8	2479.053	23236	9.3729320621
0.75	0.65	0.9	4255.985	63374.4	14.8906545671
0.75	0.7	0.8	2479.053	27847.8	11.2332388311
0.75	0.7	0.9	4255.985	68515.7	16.0986710900
0.75	0.75	0.9	4255.985	74068.800	17.4034454764
0.75	0.8	0.9	4255.985	85948.5	20.1947383181

Escuela Técnica Superior de Ingeniería del Diseño

Document 2

SPECIFICATION DOCUMENT

INDEX

1. INTRODUCTION

During the realization of this project, designer, engineer or whoever takes part of the work area is submitted to a series of work conditions that can or cannot affect some important aspects, just as health, efficiency, work rate and security etcetera. Due to the need of optimising these factor it`s necessary to establish and serve workers with assurances against work risks. These, obviously depend on the type of chores according to the job so in the case of this project, the main risks can be englobed under stress, depression and tiredness.

This project was fulfilled in Spain being the reason why it`s referenced to Spanish regulations. The basic rules that adjust to these situations is gathered in the *Real Decreto 488/1997*, 14 April, which specifies about minimum security and health conditions with electronic devices sharing visualization screens. According to article number 3, the employer has the obligation to make any modifications, if needed, so that worker bearing tasks where visualization screens are involved, doesn`t suffer any risk against his health and security, or in defect, reduce these effects down to minimums.

According to this statement, the following requests will be in consideration:

- Average use time of electronic equipment.
- Maximum continuous attention time required by electronic devices.
- Attention degree requested from jo in general.

The main risks that can emerge from these types of jobs are illumination, noise, ergonomics, visual, mental and physical fatigue. These are the main secondary effects of being in front of electronic visualization devices more time than recommended and therefore, they`ll be treated in a special way.

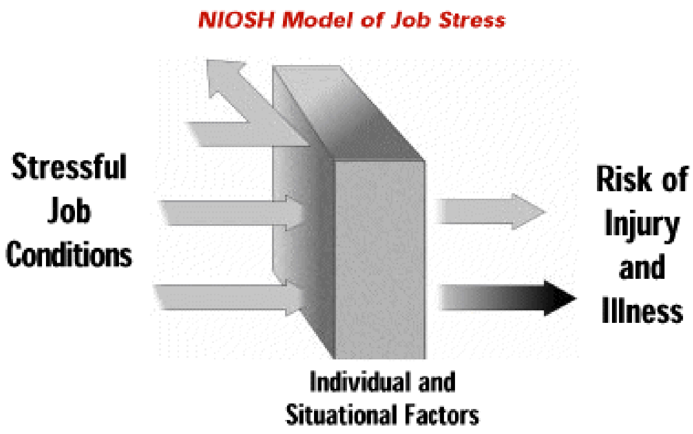


Figure 1. NIOSH Model of Job Stress

2. WORK PLACES

The location of where jobs are done is under the jurisdiction of *Real Decreto* 486/1997, in Spain, and it deals with health and security conditions and acts as complementary information for the previous *Real Decreto* about visualization devices. These conditions can be grouped in the following way:

2.1. *EMERGENCIES. EVACUATION EXITS*

The employing company must have adopted the pertinent modifications to foresee evacuation exits in case that an emergency occurs. Also as a mandatory, these conditions must be known by the employee from the employer. There are many regulations referred to emergencies and the main parts of them are focused on the constructive phase of the facility, i.e. when the building is projected and built. These regulations can be found in the *Código Técnico de la Edificación*, which is a series of rules towards building norms and security in industrial and non-industrial buildings in Spain.

2.2. *FIRE EXTINGUISHING PROTECTION*

In Spain, there are mandatory regulations and others than can be technical guides. The mandatory ones are defined by *Real Decreto*, which come from *El Ministerio de Fomento* and *El Ministerio de Industria*. From *Fomento* is where the *Código Técnico de la Edificación (CTE)* comes from and from *Industria* derives the *Reglamento de Seguridad contra Incendios en Establecimientos Industriales (RSCIEI).* As well, there are many UNE rules that have a mandatory application.

For non-industrial buildings, the regulation needed to follow is *Código Técnico de la Edificación 2006, Parte II. Documento Básico Seguridad contra Incendios.* For industrial buildings, refer to *RSCIEI.*

2.3. *ELECTRIC FACILITIES*

The electric installation must be projected, functioning and maintained by a company authorised by *El Ministerio de Industria, Energía y Turismo* or *Conserjería de Comunidad Autónoma competente.*

This facility should provide that, given the use that will be made of electricity, it can`t cause contact with people, fires or explosions, attending to the set out on tensions and security drivers, protection systems, etc. in the Regulations of Low and High voltage. From the perspective of electrical safety, provisions for workstations equipped with display data is indicated below.

- Meet the requirements of the directive on electromagnetic emissions, which requires that all electromagnetic radiation should be reduced to negligible levels from the point of view of security, safety and health of workers.

- Guarantee an appropriate maintaining of cables and connections.

- Maintain by separate electric cables and telephones.

- Use wire lengths sufficient to allow future changes, and while arranging them so that their maintenance is correct. Place them away from areas where they can be stepped on or subjected to adverse conditions.

- Ease access and cable maintenance without interruption of other parallel activities.

2.4. *ENVIRONMENTAL CONDITIONS*

Thermal comfort or the wellness situation is different for each worker, but it depends on factors such as temperature, humidity and air velocity, wall, floor and object temperature, the realised activity or dressing.

The *Real Decreto* 488/4997 on jobs with display data sets for these jobs that:

- The operating temperature of comfort must be maintained within the following range: in summer from 23 to 26 ° C, in winter time from 20 to 24 ° C and at all times should not exceed 26 ° C.

- The relative humidity should be maintained between 45 and 65% for any temperature, as this can prevent dry eyes and mucous membranes.

2.5. *ILLUMINATION*

Illumination can be natural or artificial. It is advisable to be natural, but its intensity varies during the day, with weather and seasons, which generally must be complemented by artificial lighting, being general or localized. The most usual is idea is general supplemented by localized.

In both types of illuminations, neither of them should produce glares or excessive contrast between light and shadow areas, as required by the specific regulations for these jobs.

There should be a general illumination where the visualisation screens are located. When using a source of supplementary individual lighting, this should not be used near the screen to prevent direct glare and reflections.

Regarding the location of work area and screen, regulations provide information on the most appropriate job placement to avoid glare and reflections. It is recommended that the screen is placed perpendicular to the windows, and never in front or behind them. Both cases would originate glare and reflections, either directly or by reflection. These modifications may be supplemented by using curtains or blinds to dim light.

2.6. *ERGONOMICS*

The design of the workplace is directly related to postural problems. If you consider that working with data display screens is characterized by prolonged static postures, one can deduct that the effects of these postures are worse when appropriate workplace design modifications are not taken account for.

The design must adapt to the anatomical and physiological conditions of the worker. This problem has been solved manufacturing, according to stipulated regulations, furniture that suits the job. Furthermore, furniture and surfaces susceptible to contact with worker should not be thermal transmitters, in order to prevent transmission to the user's skin.

- Seat.
  The seat height should be adjustable within the range necessary for the whole of the user. The backrest should have a smooth prominence to support lumbar region. Height and angle should be adjustable as well as the depth of the seat. All adjustment mechanisms should be easily manageable from a seated position.

- Workspace.
  Worktable should be of sufficient size and allow a flexible arrangement of the screen, keyboard, documents and related equipment. The space should be sufficient to allow workers a comfortable position. The job area must be large enough to allow working movements and posture change.

- User reference posture.
  Horizontal thighs and vertical legs. Vertical arms and horizontal forearms arms, forming a 90-degree angle at the elbow. Hands relaxed, without extension, keeping maximum straight forearm. Hence the recommendation to use palm rest.

  Straight spine. Line of the shoulders parallel to the frontal plane without twisting the back. Vision angle less than 60 degrees under the horizontal line. Distance from screen to the user's eyes should not be less than 40 cm. Moreover, the optimal distance from a visual comfort point of view should be 45 to 75 cm. Screen should be positioned so that its useful area can be seen under angles between the horizontal line of sight and the line drawn 60 below the horizontal. In the horizontal plane the screen should be positioned within the angle of 120 ° of field of view of the user, although it is advisable to place it within an angle of 70 °.

- Screen.
  The image on the screen should be stable, with no flickering or other forms of instability. The user data display screen brightness shall be easily adjusted between the characters and screen, and easily adapting to environmental conditions.
  Screen must allow position changes with the option of inclining at certain angles to adapt to user`s characteristics.

- Keyboard.
  Keyboard shall be inclined and separate from the screen so as to allow the worker to adopt a comfortable position, avoiding arm or hand fatigue.
  The height of the third row of keys should not exceed 30 mm from the support base of the keyboard, and the inclination of this must be in between 0 and 25 ° from the horizontal.

2.7. *NOISE*

According to the *Real Decreto 1316/1989 of October 27*, the employer shall assess the exposure of workers to noise, in order to determine whether the limits are exceeded in this standard, applying appropriate consequences.

Normally in these types of workplaces, usually, there isn`t a high noise level, but in this work environment there are levels that can disturb and disrupt the attention of operators. Do not forget that the computer user, often, needs concentration to carry out his work, so he`s more susceptible to being disturbed by noise or uncomfortable environment. To achieve this, one must use equipment with minimal noise emissions and optimize the acoustics of the workplace.

Regulations dictate that for difficult and complex tasks that require concentration, the equivalent continuous sound level should not exceed 55 dB.

3. INFORMATIC RESOURCE CONDITIONS

As an introduction to this chapter in the specification document, it`s known that in order to achieve decent results, just like the ones shown in this project, a medium level informatics resource base is needed that works side by side of designer or engineer.

In modern informatics, the first aspect to look at while accomplishing similar projects to this one is hardware and software, but software can be installed later. The main feature to look at will be hardware.

In this project, one computer has been used. It`s recommended to purchase a powerful PC because remember that first, all modelling, calculus and result extraction was done in a finite element program (ANSYS) which consumes many resources of our PC. After that, post-processing data was done via Excel sheets, Word documents and a powerful, numerical software like MATLAB. Finally, putting words to all these calculus and results was done with Microsoft Word documents, which also needs installing.

Luckily, these specifications can be found in any average computer on the market nowadays, due to a big informatics expansion. This project was done using the following computer: ASUS X550L. It`s technical specifications are detailed in the next lines.

Processor Type: Intel Core i7 - 4510U / BGA up to 3.1 GHz. 4th Generation latest Processor

Operating System Installed: Windows 8 64 bits, processor x64

Standard Memory: 4 GB DDR3

Maximum Memory: Up to 8 GB DDR 3

Internal Hard Disk Drive: 500 GB 5400R 4G

Memory Card Device: 2 in 1 card reader

Wireless Technologies: Wireless LAN 802.11b/g/n WLAN, Bluetooth

External I/O Ports: 2 x USB 3.0, 1 x USB 2.0, Headphone/Microphone Combo, LAN, VGA, HDMI

Display Size:15.6" HD WLED True-Life with (1366x768) Resolution

Video Adapter: Nvidia Ge-Force GT 820M

Video RAM: 2 GB

Keyboard: Full-size island-style keyboard with integrated numeric keypad

Weight: 2.4 Kg

This project was realised satisfactorily with this equipment, not needing further devices to enhance power ratio. It is to mention that this same work could have done using any other PC with similar characteristics.

Escuela Técnica Superior de Ingeniería del Diseño

UNIVERSITAT POLITÈCNICA VALÈNCIA

Document 3

BUDGET

INDEX

1. DESCRIPTION

A project budget is the estimated financial plan for a project, for which sometime funding is necessary. This document should include the expenses incurred for specified period of time, as well as income that is generated during the same period of time. It`s an important component of proposal, as it represents a financial picture of what the project will be. Depending in the type of project it can be a spreadsheet including small details or a one paged document where general expenses are covered.

Due to the nature of this project, it is very difficult, near to impossible, to determine the exact cost due to the fact that many software is used, as well as hardware which can`t be integrally counted for. Estimating software and hardware costs can be many times approximately estimated. For example, ANSYS has been used as a calculus program in this project and it requires for a licence. This licence can`t be integrally counted because it`s purchased once for a whole department of investigation, like Mechanical Engineering and Materials department from the UPV.

2. BUDGET

In order to reflect the effort put in this project in monetary value, the first thing to do is to quantify the amount of hours invested. The main activities done in this project have been:

- Objectives. Layout and approach.
- Previous research on the subject.
- Working ANSYS.
- Working Excel and Word documents.
- Working MATLAB.
- Polynomial adjustment.
- Writing report and other documents.

In the next table are reflected the amount of hours related to each activity.

Activity	Hours
Objectives. Layout and approach	10
Previous research on the subject	22
Working ANSYS	118
Working Excel and Word documents	12
Working MATLAB	10
Polynomial adjustment	20
Writing documents	112
Total	304

Suppose that the engineer or designer gets paid 28 euros per hour, that would lead us to an execution budget of 8512 €.

The next step is to evaluate all costs by software and hardware. As said before, this is a difficult chore because licences must be partially evaluated but in which proportion is a hard chore. First, the hardware is included, being the PC mentioned in the specification document (ASUS X550L). Then, an individual ANSYS licence is valued in 4000 euros. MATLAB software has an individual licence price of 2000 euros. Also, Microsoft Office Word and Excel have a price tag stipulated at 269 euros.

Product	**Cost (€)**
ASUS X550L	569
ANSYS licence	4,000
MATLAB licence	2,000
Microsoft Office package	269
Licence Cost	6,838

Now a total net execution budget is valuated adding the expenses of the engineer`s work.

Engineer`s activity	8,512 €
Licence Costs	6,838 €
Total execution budget	15,350 €
I.V.A (21%)	3,223.5 €
Total budget	18,573.5 €

Finally, the total budget adds up to **EIGHTEEN THOUSAND FIVE HUNDRED AND SEVENTY-THREE EUROS AND FIFTY CENTS**.

As mentioned previously, this price is overestimated due to the fact that a brand new computer has been included as well as all licences for every software used. These licences can be purchased for an entire department of investigation, in benefit of every worker there, so strictly, these costs would be reduced.