ESCUELA TÉCNICA SUPERIOR DE INGENIEROS INDUSTRIALES Y DE TELECOMUNICACIÓN

Titulación:

INGENIERO INDUSTRIAL

Título del proyecto:

MODELING CRACKS WITH ABAQUS

Miguel Arbeloa Ronco

Pablo Sanchis Gurpide

Pamplona, 22 de Julio del 2010

Author's Declaration

Statement 1

This work has not previously been presented in any form to Glyndŵr University or at any other institutional body whether for assessment or for any other purposes. Save for any express acknowledgements, references, and/or bibliographies cited in the work, I confirm that the intellectual content of the work is the result of my own efforts and no other person.

Statement 2

It is acknowledged that the author of this work shall own the copyright. However, by submitting this copyright work for assessment, the author grants to the University a perpetual royalty-free license to do all or any of those things referred to in section 16(I) of the copyright, designs, and patents act 1988 (viz: to copy work; to issue copies to the public; to perform or show or play the work in public, to broadcast the work or make an adaptation of the work).

Signed: _____ .

Date: _____ .

ACKNOWLEDGEMENTS

The acknowledgement of this project must be dedicated to all the people who have helped and supported me.

First of all, to my supervisor Dr Richard Grant for his dedication in this project and all his knowledge in stress analysis, cracks behaviour and Abaqus software, that has been very helpful to me.

I want also thank Ms Ann Lambert for all her knowledge in Abaqus software.

ABSTRACT

This project arises from the fact that nowadays the engineering process is becoming more experimental, so the knowledge on design software is very important in order to save money and time.

From this point of view, new mathematical methods are applied to design software to develop new tools for solving a huge number of design analyses.

In this case Abaqus software is based on the Finite Element Method, which is a numerical technique for solving models in differential form. For a given design, the FEM requires the entire design, including the surrounding region, to be modelled with finite elements.

The final purpose of this project is to develop techniques to model cracks in 2-D and 3-D structures using Abaqus CAE. To develop this project the techniques proposed have to be compared and the data obtained validated with published data.

TABLE OF CONTENTS

1. INTRODUCTION

The analysis of the behaviour of cracks is very important nowadays due to the need to know because the components of a structure fail to avoid or prevent a subsequent failure. The failures caused by cracks may be due to two types of situations. Firstly, the material is subjected to a fixed load and therefore the crack opens and grows due to this effort. And secondly, the piece is submitted to cyclical loads that open and close the crack while it grows. Learn how a crack behaves in a material submitted to a certain effort or to a few cyclical loads helps us to be able to anticipate possible accidents.

This project arises from this need to know the behaviour of a crack and the intensity factor submitted to a fixed load. The modelling of cracks by finite elements allows us to estimate the behaviour of the crack with amazing accuracy, and this information can be exported then to the practice to obtain the correct prediction.

As this method could improve stress analyses it is important to have the necessary tools to perform them. So the development and the purpose of making know this method to new engineers acquire in this project the same importance than the stress analysis.

The number of complete engineering software (Electric, magnetic, thermal, structural…) that uses finite element method is very large. In this project the software selected is Abaqus 6.9-1.

The concrete program used is the Abaqus 6.9-1, specifically Abaqus CAE. Abaqus is a finite element program that approximates the solution by dividing the problem in a finite number of points, called nodes. Abaqus obtains the solution in these points and interpolate the solution for the rest of the points.

To do this project, it is very important to have fracture mechanics knowledge, because I will model cracks subjected to a big load. Fracture mechanics refers to a phenomenon where the breaking of materials occurs under big loads. Other important way of collapse is fatigue. Fatigue of materials occurs when the materials collapse under cycle dynamics loads.

To sum up, the principal aims of this project are two: the first is make more common the use of software based on finite element method and the second is make an exhaust modelling of 2D and 3D cracks.

It is important to have these two goals clearly defined because both can develop the skills to design pieces safest and without doing practical test with real pieces, what means less waste of time and money.

2. AIMS OF THE PROJECT

As it is known this project arises from the need to develop techniques to model cracks in 2D and 3D with Abaqus.

The most immediate objective is to and shape cracks in different geometries both in 2-D and 3-D. Develop different techniques of meshing and analysis depending on the type of geometry of the piece and on the type of crack.

The other main objective is to initially understand and install single edge cracks in plates and compare S.I.F. values with published data. Once created and installed the cracks, the data obtained with the program should be contrasted with published data to be able to validate them. This is a very important point because if the data does not match up the analysis will have to be realized again.

To further this work to more complexes published geometric and compare. Once finished with simple geometries I will go on to more complicated geometries, both of piece and of crack. In this step as in the previous one the data obtained will have to be also validated.

As personal experience, this project has helped me go deeper into the design of pieces from a different angle to the common. It means, not only the design of a concrete piece, it has been the design with a view to its use and functionality. So I have to develop skills to analyze a piece form the point of view of a complete structure, whole scaffolding instead of only a plate whit a crack.

Finally, this project has helped me in the aspect of collecting and interpreting data. This is important because the behaviour of a crack can be described and what is more important, predicted.

3. REVIEW OF LITERATURE

3.1. INTRODUCTION

In this section I will make a brief statement of the theoretical foundations that must be clear to make a project of this nature. I will treat from the theoretical foundations of the Fracture mechanics up to the operation of the finite elements method that use programs like ABAQUS.

3.2. FRACTURE MECHANICS

- **Introduction**

Fracture mechanics and particularly the linear elastic fracture mechanics is a science that studies the mechanisms and processes of crack propagation in solids, as well as the distribution of stresses and strains that occur in a cracked material or material with discontinuities, subjected to certain external tension. This science arose with Griffith's fundamental works on criteria of spread of cracks in solid, based on concepts of transformation of elastic energy in surface energy and for this reason it is known as an energetic formulation of fracture mechanics.

In simple terms, the energetic formation of fracture mechanics consists of comparing the available energy for propagation of a crack in a structure with the energy required to produce its cracking. The available energy for the advance of crack per unit area is called the energy release rate (G) and the energy required for cracking is called the critical rate of energy release or energy of cracking (G_{IC}).

Irwin introduced a breakthrough in fracture mechanics and raised the analysis in terms of tensions, which had not been able to do before due to the fact that theoretically at the tip of a crack the stress tend to infinity, independently of the magnitude of the applied load. For his formulation, Irwin argues that the fracture process cannot be concentrated on a single point, as deducted from a purely elastic analysis, but if appears in a small but finite zone, which names plastic zone in which the tensions stop being infinite because part of the elastic energy is consumed in the plastic deformation of the material near the tip. The main concept introduced here is that the stress intensity factor

(K) of great application in the mechanics of fracture. The parameter K is related to the parameter G by the elastic properties of the material and for the condition of crack initiation is considered that K becomes a critical value and for such a reason is called critical stress intensity factor or fracture toughness K_{IC}.

One of the important aspects that make it different from a conventional analysis of resistance of materials to a fracture mechanics analysis is that in the first case the resistance or tensile strength is independent from the size of the structure or element under load, whereas in the second one, the resistance can change depending on the size or the structure. This type of behaviour is studied by the so-called size effect and allows to know if a certain material adjust of not to the criteria of the LEFM of if it is necessary to introduce alterations for effects of size to the parameters G_{IC} and K_{IC} in order to be considered as real material constants, independently of its size.

- **Principles of fracture mechanics**

The resistance to the fracture of a solid material is a function of the cohesive forces that exist between the atoms. On this base, the theoretical cohesive strength of a brittle elastic solid has been estimated and is approximately equal to E/10 where E is the modulus of elasticity. The resistance to the experimental fracture of the majority of the materials in engineering normally is between 10 and 1000 times lower than the theoretical value. Griffith suggested that this discrepancy between the cohesive theoretical resistance and the resistance of the observed fracture could be explained by the presence of microscopic cracks, which exist in normal conditions on the surface and inside a piece of material. These flaws go the detriment of the resistance to the fracture due to the fact that an applied tension can be amplified or concentrated at the tip of the defect, in a degree that depends on the orientation of the crack and on the geometry. This phenomenon is shown in the figure below which shows the magnitude of the stress across the section containing a crack. As illustrated is the profile, the magnitude of the located tension decreases with the distance from the tip of the crack. At positions far removed, the stress is exactly the rated tension σ_0, or the load divided by the area of the section. Due to the ability to amplify in their surroundings an applied stress, these defects are sometimes called stress concentrators or stress raisers.
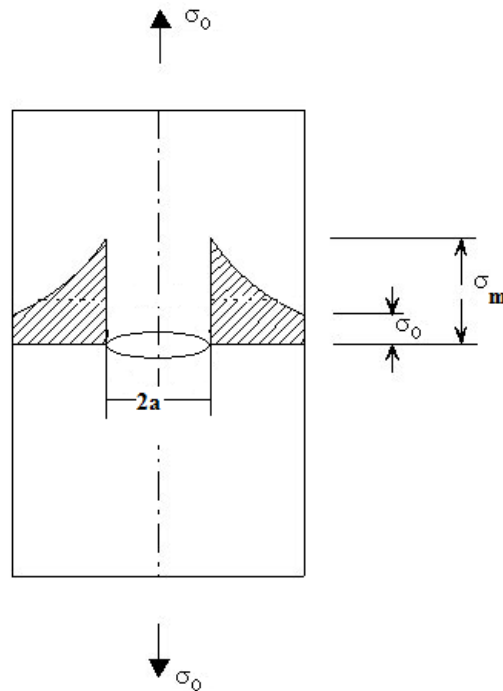
Figure 3.1

Assuming that the crack has an elliptical shape and is orientated with its axis perpendicular to the applied stress, the maximum stress at the end of the crack, σ_m, it can be approximated by

$$\sigma_m = 2\sigma_0 \left(\frac{a}{\rho_t}\right)^{1/2} \qquad \textbf{Equation 3.1}$$

Where σ_0 is the magnitude of the rated stress applied, ρ_t is the radius of curvature of the tip of the crack and represents the length of a surface crack, or half the length of an internal crack. So for a relatively long micro-crack has a radius of curvature small, the factor $\left(\frac{a}{\rho_t}\right)^{1/2}$ can be very large. This will give a value of σ_m many times greater than σ_0.

Often the coefficient σ_m/σ_0 is called stress concentration factor K_t:

$$K_t = \frac{\sigma_m}{\sigma_0} = 2 \left(\frac{a}{\rho_t}\right)^{1/2} \quad \textbf{Equation 3.2}$$

That is simply a measure of the degree to which an external voltage is amplified at the tip of a small crack.

- ## **Griffith theory of Brittle Fracture**

During the propagation of a crack, there is a release of what is termed the elastic strain energy, some of the energy that is stored in the material as it is elastically deformed. Furthermore, during the crack extension process, new free surfaces are created at the faces of a crack, which give rise to an increase in surface energy of the system. Griffith developed a criterion for crack propagation of an elliptical crack by performing an energy balance using these two energies. He demonstrated that the critical stress σ_c required for crack propagation in a brittle material is described by

$$\sigma_c = \left(\frac{2E\gamma_s}{\pi a}\right)^{1/2} \qquad \textbf{Equation 3.3}$$

Where

E = modulus of elasticity

γ_s = specific surface energy

a = one half the length of an internal crack

The previous development is applied to materials completely fragile which has no plastic deformation. The majority of the metals and many polymers experience some plastic deformation before the fracture; this produces an increase in the radius of the tip of the crack therefore increase resistance to fracture. Mathematically, this can be taken into account by replacing γ_s for $\gamma_s + \gamma_p$ where γ_p represents the plastic formation energy associated with the extension of the crack.

Irwin then suggested grouping the two terms in a single term, G, so

$$G = 2\left(\gamma_s + \gamma_p\right) \qquad \textbf{Equation 3.4}$$

and it is called the energy available for fracture. The crack extension occurs when G exceeds a critical value Gc.

- ## **Stress analysis of cracks**

There are three ways of applying a force to enable a crack to propagate:

➢ Mode I crack: Opening mode (a tensile stress normal to the plane of the crack)

➢ Mode II crack: Sliding mode (a shear stress acting parallel to the plane of the crack and perpendicular to the crack front)

➢ Mode III crack: Tearing mode (a shear stress acting parallel to the plane of the crack and parallel to the crack front)

Mode I: Mode II: Mode III:
Opening   In-plane shear   Out-of-plane shear
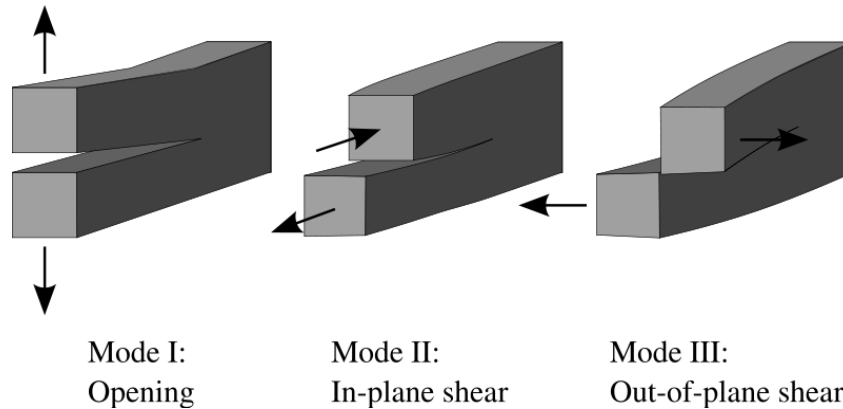
Figure 3.2

The model I is the most frequently occurring.

For the model I configuration, the stresses acting on an element of material appear in the following figure. Using the principles of the theory of elasticity, the tensile (σ_x, σ_y) and shear (τ_{xy}) stresses are functions of distance r and angle θ.
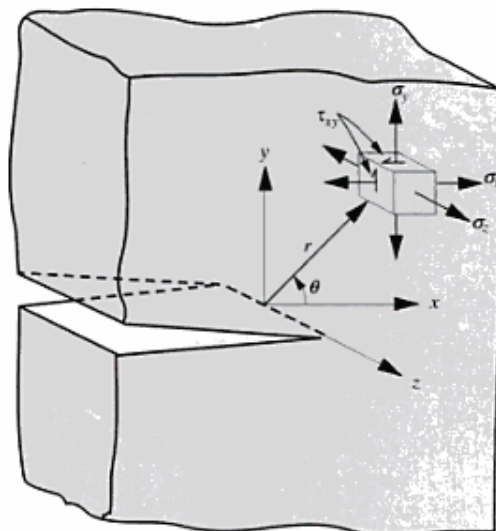
Figure 3.3

$$\sigma_x = \frac{K}{\sqrt{2\pi r}} f_x(\theta) \qquad \textbf{Equation 3.5}$$

$$\sigma_y = \frac{K}{\sqrt{2\pi r}} f_y(\theta) \qquad \textbf{Equation 3.6}$$

$$\tau_{xy} = \frac{K}{\sqrt{2\pi r}} f_{xy}(\theta) \qquad \textbf{Equation 3.7}$$

If the plate is enough thin compared with the dimensions of the crack, then σ_z=0, and it is said that there exist conditions of plane stress. In the other hand, (in a case of a relatively thick plate), σ_z=υ (σ_x +σ_y) and the condition is called plane strain (since ε_z=0); in this expression υ is the Poisson's ratio.

In the previous equations, the parameter K is called the stress intensity factor; determines the magnitude of the stress distribution around a crack. It should be noted that the stress intensity factor and the stress concentration factor K_t of the previous equation, though similar, they are not equivalent.

- **Fracture toughness**

The fracture occurs when the level of the stress applied is greater than a critical value σ_c. Similarly, since the tensions in the proximities of a crack tip are defined in terms of stress intensity factor, there exists a critical value of this parameter. This critical value is called fracture toughness K_C and may be used to specify the conditions of brittle fracture. In general, it can be expressed as

$$K_c = Y\sigma\sqrt{\pi a} \qquad \textbf{Equation 3.8}$$

Where Y is a parameter without dimension that depends on the geometry of the piece and the crack.

By definition, the fracture toughness is a property that is a measure of material resistance to brittle fracture when a crack is present.

For relatively thin specimens, the value of K_C will depend on the thickness of the specimens and decrease as show in the Figure 3.4. From a certain thickness, K_C is independent of the thickness, is at that point when we are in plane strain conditions. This value of K_C is called strain fracture toughness K_{IC}, which follows the expression

$$K_{Ic} = Y\sigma\sqrt{\pi a} \qquad \textbf{Equation 3.9}$$

The K_{IC} value is always less than that of K_C. The subscript I indicates that is for the mode I displacement of the crack.
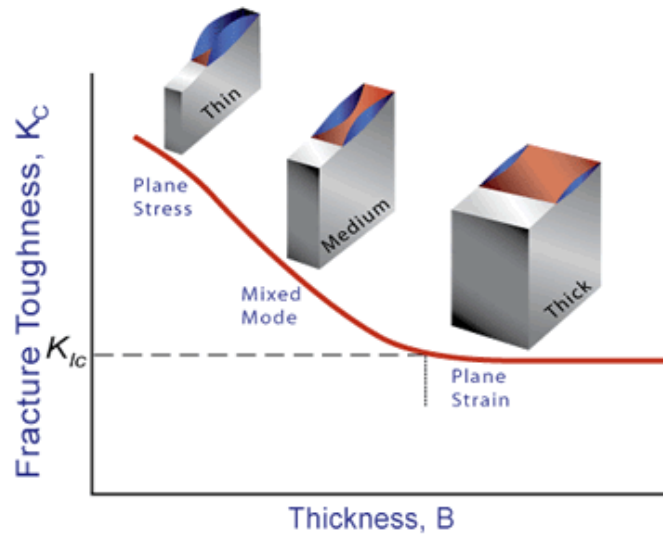
Figure 3.4 Fracture Toughness vs. Thickness **[1]**

- **Plastic yielding at crack tip**

To determinate the plastic zone at the crack tip, Irwin presented a simple mode assuming the material is elastic-perfectly plastic. Consider the distribution of tensile stress σ_{yy} acting across a line extending, ahead of and in the same direction as the crack. The local y-stress near the crack tip is,

$$\sigma_{yy} = \frac{K_I}{\sqrt{2\pi r}} \quad \textbf{Equation 3.10}$$

Where r is the distance from the crack tip. As a first approximation, we can assume that the boundary between elastic and plastic behaviour occurs when the stress given by the above equation satisfies a yield criterion. For plane stress conditions, yielding occurs when $\sigma_{yy} = \sigma_{ys}$, the uniaxial yield strength of the material. Then the distance ahead of the crack tip over which this happens is

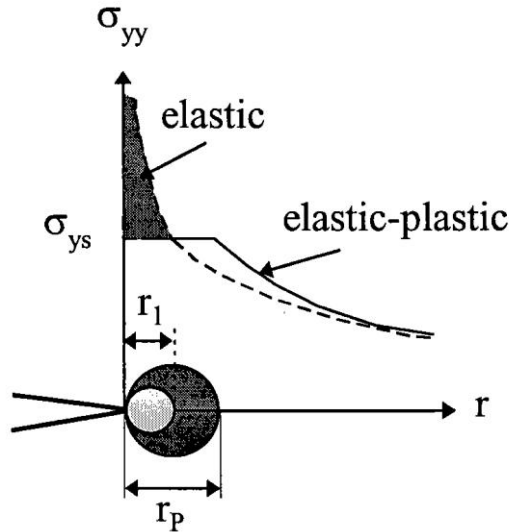$$r_I = \frac{K_I^2}{2\pi\sigma_{ys}^2} \quad \textbf{Equation 3.11}$$

Figure 3.5 Estimated plastic zones **[2]**

- **J-integral**

In the analysis that I will realize with Abaqus, the program will give back a value called J-integral.

The J-integral represents a way to calculate the strain energy release rate, or work (energy) per unit fracture surface area, in a material. The theoretical concept of J-integral was developed in 1967 by Cherepanov and in 1968 by Jin Rice independently, who showed that the energetic contour path integral (called J) was independent of the path around a crack.

The form of the J-integral for two-dimension is:

$$J = \int_{\tau}\left(W dx_2 - t\frac{\partial u}{\partial x_1}ds\right) \qquad \textbf{Equation 3.12}$$

Where W(x1,x2) is the strain energy density, x1,x2, are the coordinate directions, t=n ·σ is the traction vector, n is the normal to the curve Γ, σ is the Cauchy stress tensor, and u is the displacement vector. The strain energy density is given by

$$W = \int_{0}^{\varepsilon} \sigma d\varepsilon \qquad \textbf{Equation 3.13}$$

$$\varepsilon = \frac{1}{2}[\nabla u + (\nabla u)^T] \qquad \textbf{Equation 3.14}$$
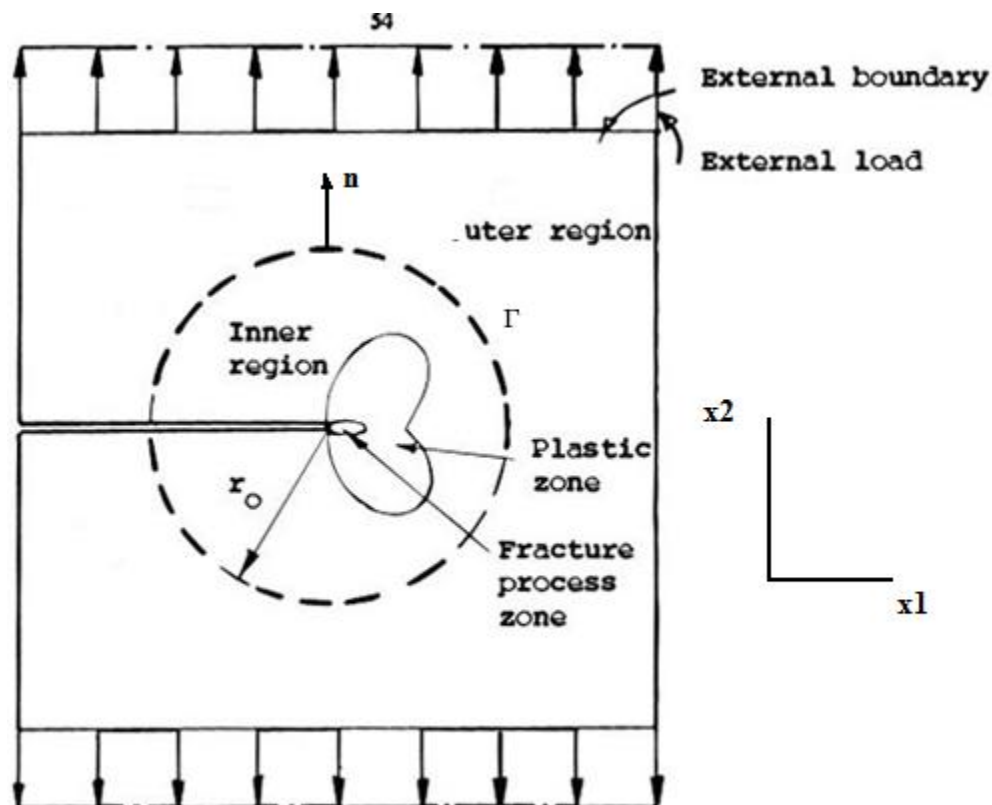
Figure 3.6

The J-integral is equal to the strain energy release rate for a crack in a body subjected to monotonic loading. This is true, under quasistatic conditions, both for linear elastic materials and for materials that experience small-scale yielding at the crack tip. And both are related to the stress intensity factor "K" in the following fashion:

- Plane stress

$$J = \frac{K^2}{E} \quad \textbf{Equation 3.15}$$

- Plane strain

$$J = \frac{K^2}{E}(1 - v^2) \quad \textbf{Equation 3.16}$$

3.3. <u>FINITE ELEMENT METHOD (FEM)</u>

The employment of numerical analysis technique known as Finite Element Method (FEM) has been spreading in the last decades at the same rate that was imposing the use of computers due to the capacity and speed of calculation.

This technique allows us to solve common problems in engineering such as mechanical problems, thermal, etc. These entail the integration of complex systems of partial differential equations (PDE's), which until then were unapproachable, except in very simplified cases and, in most cases, slightly representative of the reality.

In outline, the FEM manages to turn the PDE's system into another system of algebraic linear equations (or not), whose resolution in computers, or even by hand if they are few, is relatively simple. As expected, this changes results in an error, which we always assume.

Initially, the procedure to follow consists of replace a domain formed by an infinite number of points, for a finite number that the user considers to be enough under his criteria. This process is called discretization that gives place to a mesh formed by nodes and elements.

On that domain, there exist a few equations of government that describe the behaviour of the physical phenomenon in question (mechanical balance, transmission of heat, water flow, etc.). As it happens in the analytical resolution, in order that solution exists and this one is unique, we must impose boundary conditions.

The unknown function (displacement, temperature, etc.) can be interpolated through the points obtained in the discretization which support each elementary function, called shape functions. These functions may be linear, parabolic, etc.

The weak of the variational form of the governing equations allows us to obtain the above mentioned system of equations that we will have to solve once imposed the boundary conditions, which allow us to invert the system matrix.

All this is done in a first step in each element then go on to describe the global behaviour through a process called assembly.

In mechanical problems the fundamental unknown are the displacements. The weak form of governing equations (the equation of internal equilibrium) is called the Principle of Virtual Works that not only involves the deformations (easily deducible of the displacements) but also depends on the tensions. Here appears the constitutive equation of the medium in which we are. This equation relates in a linear way or not, the tensions with the deformations. Thus, after application of the law, we are able to formulate the problem exclusively in terms of the displacements. Then, from the displacements, we can obtain the deformations and the tensions.

- **Finite element formulation for plane stress/strain analysis**

For plane stress and strain elasticity there are three equations which are:

Kinematic relationship:

$$\boldsymbol{\varepsilon} = \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \end{Bmatrix} = \begin{Bmatrix} \partial u/\partial x \\ \partial v/\partial x \\ \partial u/\partial y + \partial v/\partial x \end{Bmatrix} \quad \textbf{Equation 3.17}$$

Constitutional law:

$$\boldsymbol{\sigma} = \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = \begin{bmatrix} E & -vE & 0 \\ -vE & E & 0 \\ 0 & 0 & \frac{1}{2}(1+v)E \end{bmatrix} \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \end{Bmatrix} = D\varepsilon \quad \textbf{Equation 3.18}$$

Equilibrium equation:

$$\partial\sigma_x/\partial x + \partial\tau_{xy}/\partial y + q_x = 0 \quad \textbf{Equation 3.19}$$

$$\partial\tau_{xy}/\partial_x + \partial\sigma_y/\partial y + q_y = 0 \quad \textbf{Equation 3.20}$$

Where u is the displacement in the x direction and v is the displacement in the y direction, q_x and q_y are distributed body forces in the x and y directions, respectively; ε_x and ε_y are extensional strains in the x and y directions and ε_z is the shear strain; σ_x and σ_y are extensional stresses in the x and y directions and τ_{xy} is the shear stress; E is the Young's modulus, and υ is the Poisson's ratio. The boundary conditions that I have supposed are:

Displacement boundary conditions:

$$u = \bar{u} \qquad v = \bar{v} \qquad \textbf{Equation 3.21}$$

Force boundary conditions:

$$t_n = \bar{t}_n \qquad t_s = \bar{t}_s \qquad \textbf{Equation 3.22}$$

Where t_n and t_s are tractions in the normal and tangential directions of the boundary, and an overbar represents the prescribed value.

I will only derive a four-node quadrilateral element for plane stress or strain analysis. The geometrical domain of the element is defined in terms of shape functions and its nodal coordinates, as follows:

$$\boldsymbol{x} = \begin{Bmatrix} x \\ y \end{Bmatrix} = \sum_{i=1}^{4} N_i(\xi,\eta) \begin{Bmatrix} x_i \\ y_i \end{Bmatrix} = \boldsymbol{N}(\xi,\eta)\boldsymbol{X} \qquad \textbf{Equation 3.23}$$

Where $\mathbf{x} = (x, y)^T$ is the coordinate system, (x_i, y_i) (i = 1,2,3,4) is the coordinate of the ith node of the element, $\mathbf{X} = [x_1, y_1, x_2, y_2, x_3, y_3, x_4, y_4]^T$ is the vector of element nodal coordinates, $\mathbf{N}(\xi,\eta)$ is the shape function matrix defined as

$$\boldsymbol{N}(\xi,\eta) = \begin{bmatrix} N_1(\xi,\eta) & 0 & N_2(\xi,\eta) & 0 & N_3(\xi,\eta) & 0 & N_4(\xi,\eta) & 0 \\ 0 & N_1(\xi,\eta) & 0 & N_2(\xi,\eta) & 0 & N_3(\xi,\eta) & 0 & N_4(\xi,\eta) \end{bmatrix}$$

$$N_i(\xi,\eta) = \tfrac{1}{4}(1 + \xi\xi_i)(1 + \eta\eta_i), \quad i = 1,2,3,4 \qquad \textbf{Equation 3.24}$$

Where ξ and η consist of the local coordinate system associated with the element, and (ξ_i, η_i) is the local coordinate of the ith node of the element. The

displacements in the element are interpolated by the same expressions as for the coordinates:

$$\boldsymbol{u} = \begin{Bmatrix} u \\ v \end{Bmatrix} = \sum_{i=1}^{4} N_i(\xi, \eta) \begin{Bmatrix} u_i \\ v_i \end{Bmatrix} = \boldsymbol{N}(\xi, \eta)\delta \qquad \textbf{Equation 3.25}$$

Where (u_i, v_i) $(i = 1,2,3,4)$ is the displacement of the ith node of the element, and $\delta = [u_1, v_1, u_2, v_2, u_3, v_3, u_4, v_4]^T$ is the vector of the element nodal degrees of freedom. The element with the same interpolation expressions for the displacements and the coordinates is called the isoparametric element.

Substituting equation 3.25 into equation 3.17 gives the strain vector in terms of the shape functions and the nodal degrees of freedom:

$$\epsilon = \boldsymbol{B}\delta \qquad \textbf{Equation 3.26}$$

Where

$$\boldsymbol{B} = [B_1, B_2, B_3, B_4] \quad \boldsymbol{B}_i = \begin{bmatrix} \partial N_i/\partial x & 0 \\ 0 & \partial N_i/\partial y \\ \partial N_i/\partial y & \partial N_i/\partial x \end{bmatrix} \quad i=1,2,3,4 \qquad \textbf{Equation 3.27}$$

$\boldsymbol{B}$ is called the strain matrix. The principle of minimum potential energy applied to the element requires

$$\pi = \int_A \frac{1}{2}\varepsilon^T \boldsymbol{D}\varepsilon\, dxdy - \int_A q^T \boldsymbol{u}\, dxdy - \int_{\partial A}(t^0 + t^i)^T \boldsymbol{u}\, ds = min \qquad \textbf{Equation 3.28}$$

Where A is the area of the element, ∂A is the boundary of A, $\mathbf{q} = (q_x, q_y)^T$ is the vector of body forces, and t^0 and t^i are boundary tractions due to the externally applied load and the inner forces between adjacent element. Substituting equation 3.23 and equation 3.26 into equation 3.28 gives

$$\pi = \frac{1}{2}\delta^T \boldsymbol{k}\delta\delta - (\boldsymbol{p}^0 + \boldsymbol{p}^i)^T \delta \qquad \textbf{Equation 3.29}$$

Where $\mathbf{k}$ is the element stiffness matrix, and $\mathbf{p}^0$ and $\mathbf{p}^i$ are equivalent nodal forces of externally applied load and boundary tractions between adjacent elements, respectively. They read,

$$\boldsymbol{k} = \int_A \boldsymbol{B}^T \boldsymbol{D}\boldsymbol{B}\, dxdy \quad \textbf{Equation 3.30}$$

$$p^0 = \int_A N^T q \, dx \, dy + \int_{\partial A} N^T t^0 \, ds \quad \text{Equation 3.31}$$

$$p^i = \int_{\partial A} N^T t^i \, ds \qquad \text{Equation 3.32}$$

Minimizing equation 3.29 results in the following element-level finite element equation:

$$k\delta = p^0 + p^i \qquad \text{Equation 3.33}$$

Again, after assembling all element equilibrium equations, the inner force terms are eliminated from each other due to the principle of action and reaction between adjacent elements. The global finite element equilibrium equation is then written as

$$KU = F \qquad \text{Equation 3.34}$$

Where **K** is the global stiffness matrix, **U** is the global vector of all nodal degrees of freedom, and **F** is the global vector of all equivalent nodal applied forces.

4. DESIGN PROCESS

4.1. INTRODUCTION

In the following section I will introduce how I have realized the modelling of cracks in both 2D and 3D. It shows the way of creating cracks on plates and meshes in 2D and 3D. This section is very important as it is showed how to degenerate mesh elements to analyse the behaviour of the cracks. The following sections have form of tutorial as they have been written in this form to send it to ABAQUS since the program lacks a tutorial about cracks.

In the modelling of cracks, at the crack tip, four –side elements (in 2D) are degenerated down to triangles as we can see in Figure 4.1. Note that three nodes occupy the same point in space. Figure 4.2 shows the situation for 3D, where a brick element is degenerated to a wedge.
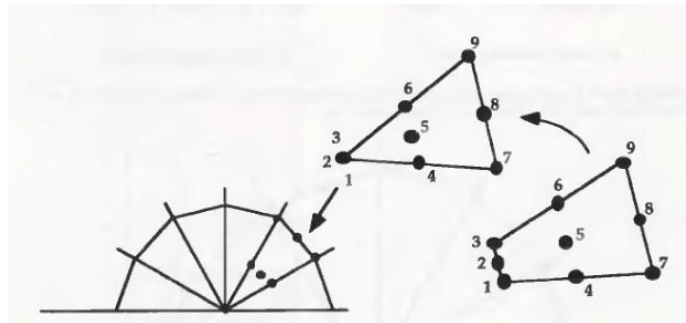
Figure 4.1
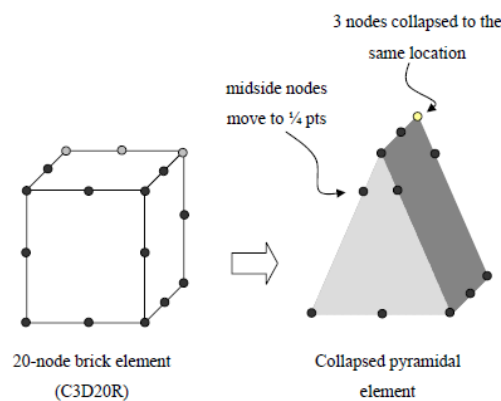
Figure 4.2 [3]

For elastic problems, the nodes at the crack tip are normally tied, and the mid-side nodes moved to the 1/4 points as we can see in Figure 4.1. Such a modification results in a $1/\sqrt{r}$ strain singularity in the element, which improve numerical accuracy.

When the plastic zone appears, the $1/\sqrt{r}$ singularity no longer exists at the crack tip and elastic singular elements are not appropriate for these analyses. Now, the element is degenerated to a triangle as before, but the nodes at the crack are not tied and the location of the mid-side nodes still being at 1/4 point . This element geometry produces a $1/r$ strain singularity, which corresponds to the actual crack tip strain field for fully plastic materials.

The most benefit of the plastic singular element derives from the fact that allows the crack tip opening displacement to be computed from the deformed mesh as it is shown in Figure 4.3. The untied nodes stay initially in the same point but then, they move as the element deforms.
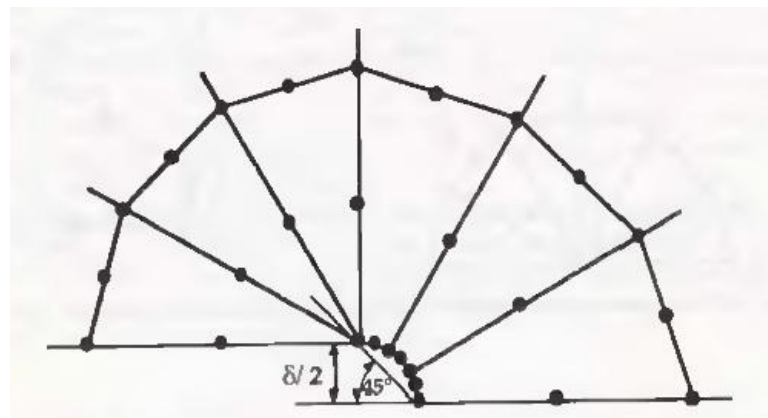
Figure 4.3

For mainly problems, the most efficient mesh design for the crack tip region belongs to the "spider web" configuration. This configuration consists of concentric rings of four sided elements that are focused toward the crack tip. The close ring is formed by quadrangular elements degenerated to triangles. Since the crack tip region contains steep stress and strain gradients, the mesh refinement should be greatest at the crack tip. Whit this configuration we can obtain a smooth transition from the fine mesh near the crack tip to a rude mesh far away to the tip.

In the modelling of cracks that I will show then, I have degenerated quadrilaterals in 2D and hexahedron in 3D until triangles and triangular prisms respectively.

This degeneration of the elements is performed to exhibit a $1/\sqrt{r}$ strain singularity, which improves the accuracy and reduces the need for a high degree of mesh refinement at the crack tip.

4.2. BUILDING 2D CRACKS

- ### CREATING AND ANALYZING A 2D CRACK

The following section leads you through the ABAQUS/CAE modelling process by visiting each of the modules and showing you the basic steps to create and analyze a 2D crack. To illustrate each of the steps, you will first create a model of a steel sheet with an edge crack and load its top and bottom surface as it is shown in the figure.
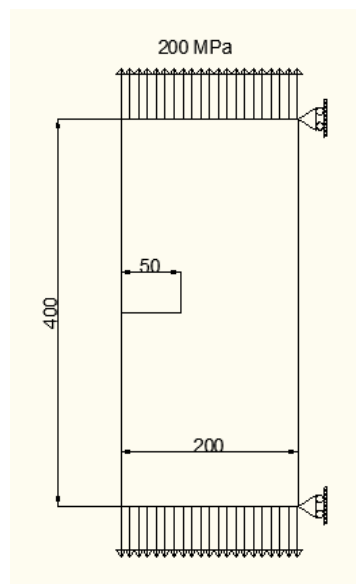
Figure 4.2.1: A loaded sheet with a crack edge

1) Creating a part

To create the sheet:

1) In the Model Tree, double-click the **Parts** container to create a new part.

2) Name the part *Sheet* Accept the default settings of a two-dimensional, deformable body and a solid, extruded base feature. In the **Approximate size** text field, type 1000. And click Continue.

3) Draw a rectangle of size 200 x 400 mm using

   You should obtain these figures:
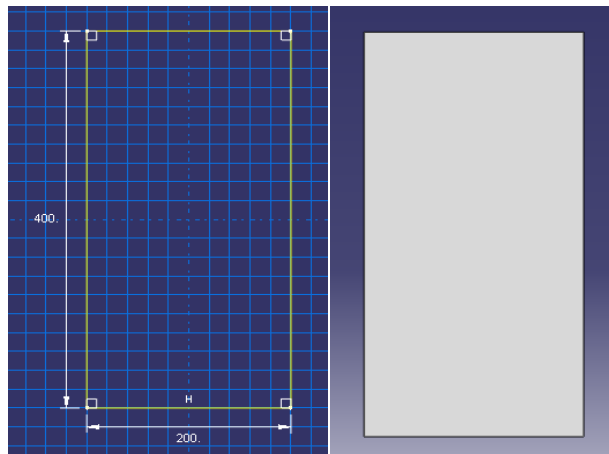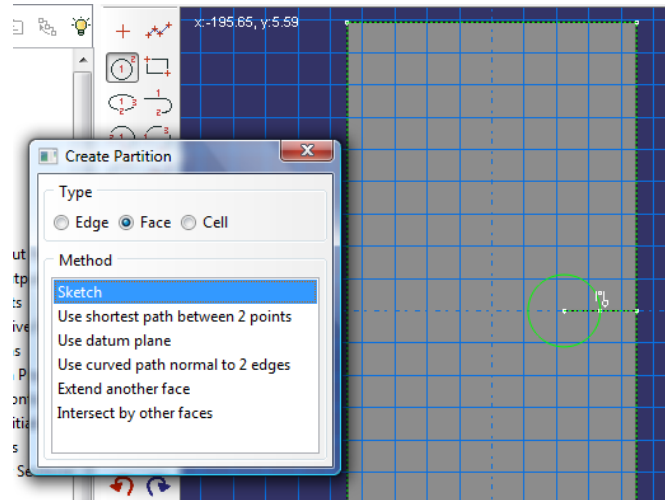

Figure 4.2.2: Sketch of the rectanle

4) Then you must draw the size of the crack you want to analyze. For do it go click on and draw a line of length 50mm.
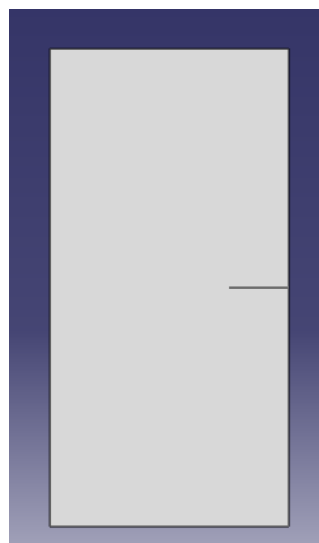
Figure 4.2.3: View of the sheet with the crack

5) After drawing the crack, you must draw a circle around the crack tip. Inside this circle is where you will create the degenerated mesh.

Tools → Partition → Face → Sketch

Figure 4.2.4: Sketch of the circle

As a result of doing this, you should obtain:
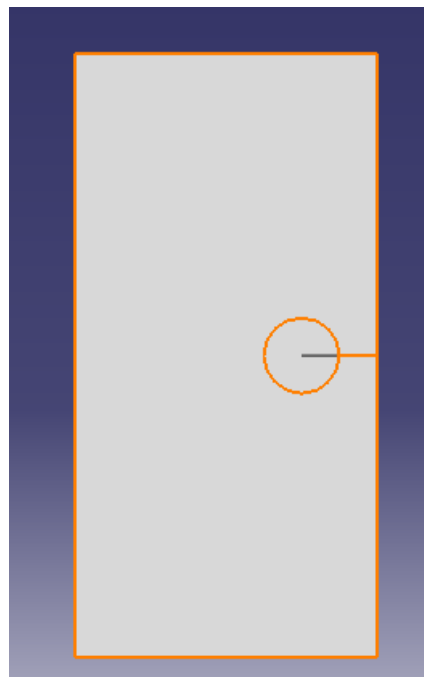
Figure 4.2.5: View of the sheet with the crack and the circle

2) <u>**Creating a material**</u>

1) In the Model Tree, double-click the **Materials** container to create a new material.

2) Name the material `Steel`.
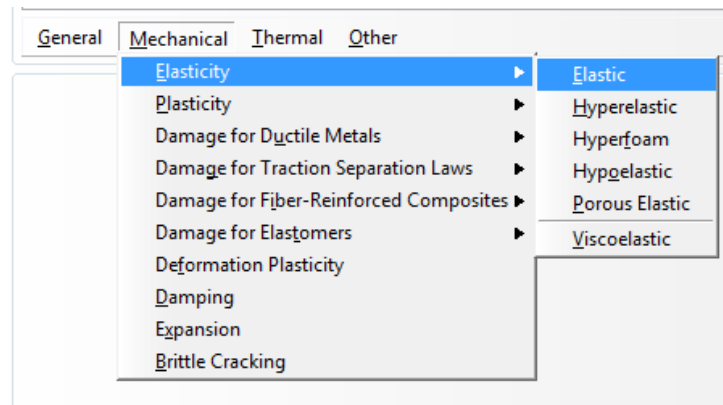
Figure 4.2.6: Submenus available under the Mechanical menu.

3) From the material editor's menu bar, select **Mechanical ⟶ Elasticity ⟶ Elastic**.

4) Type a value of `200` for Young's modulus and a value of `0.3` for Poisson's ratio in the respective fields, as shown in Figure 7.

Figure 4.2.7: Entering data values for the elastic material properties.

5) Click **OK** to exit the material editor.

3) Defining and assigning section properties

    i) Defining a homogeneous solid section

1) In the Model Tree, double-click the **Sections** container to create a section.

2) In the **Crate Section** dialog box:

    a. Name the section *SheetSection*.

    b. In the **Category** list, accept **Solid** as the default category selection.

    c. In the **Type** list, accept **Homogeneous** as the default type selection.

    d. Click **Continue**

3) In the dialog box:

    a. Accept the default selection of Steel for the **Material** associated with the section.

    b. Accept the default value of 1 for the Plane stress/strain thickness.

    c. Click **OK**.

  i) Assigning the section to the sheet.

1) From the main menu bar, select **Assign → Section**

2) Click on the different regions to select the region to which the section will be applied.

3) Click **Done** in the prompt area to accept the selected geometry

4) Accept the default selection of *SheetSection* as the section, and click **OK**.

4) Assembling the model

1) In the **Module** list locate under the toolbar, click **Assembly** to enter the Assembly module

2) From the main menu bar, select **Instance → Create**

3) In the dialog box, select **Sheet** and make sure that you are in **Independent instance** type. Then click **OK**.

5) Defining your analysis steps

  i. Creating an analysis step

To create a general, static analysis step:

1) In the **Module** list located under the toolbar, click **Step** to enter the Step module.

2) From the main menu bar, select **Step → Create** to create a step

3) Name the step *Sheetcrack*

4) From the list of available general procedures in the **Create Step** dialog box, select **Static, General** and **Continue.**

5) The **Basic** tab is selected by default. In the **Description** field, type *Crack of the sheet.*

6) Click **OK** to create the step and to exit the **Edit Step** dialog box.

ii.   Requesting data output

You should do this step after doing the point 6 of this tutorial because you need to create the crack before requesting the data output in the **History Output Request**.

To examine your output requests:

1) From the main menu bar, select Output → Field Output Request → Manager.
2) Review the default output request that ABAQUS/CAE generates for the **Static, General** step you created and named *Sheetcrack*.
3) On the right side of the **Field Output Request Manager**, click **Edit** to view more detailed information about the output request.
4) On the **Edit History Output Request** dialog box select **Contour Integral** in the Domain Tree and select the Crack you have created.
5) Type 3 in the Number of contours and J-integral as Type.
6) Click **OK** to close the field output editor and save the changes.
7) Click **Dismiss** to close the **History Output Request Manager**.

6) Creating the crack and the seam

1) In the **Module** list located under the toolbar, click **Interaction** to enter the Interaction module.
2) From the main menu bar, select **Special → Crack → Create.**
3) Name the crack *Crack* and click **Continue**.
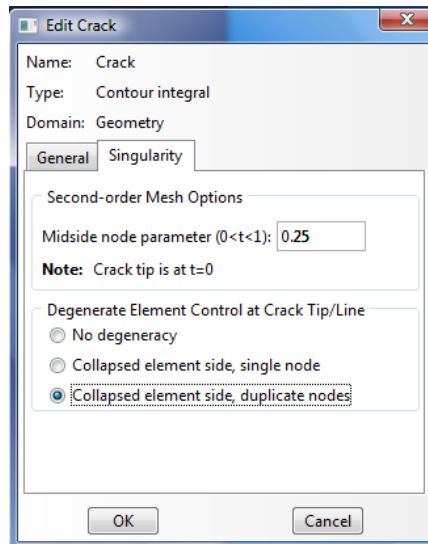4) Pick the end of the crack and then **Done.**
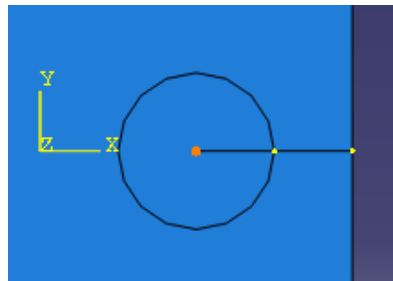
Figure 4.2.8: View of the crack tip.

5) Select **q-vector** as method to specify the crack extension direction.
6) Select the beginning of the crack length and the crack tip.
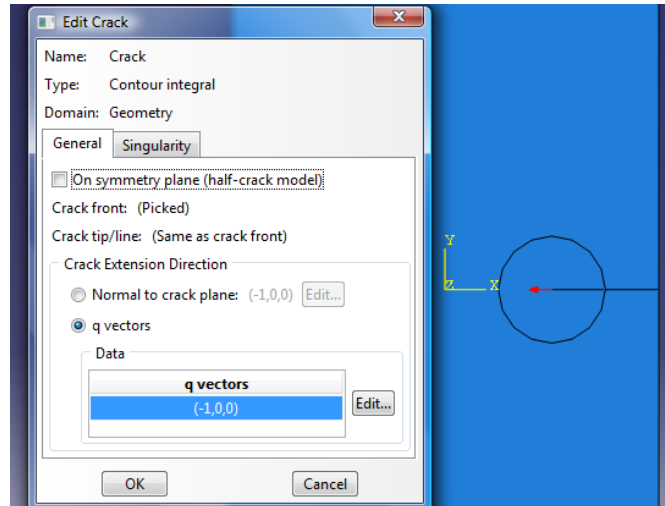
Figure 4.2.9: View of the Edit crack box

7) Then click Singularity and type 0.25 as Midside node parameter and select Collapsed element side, duplicate nodes. Click **OK**.

Figure 4.2.10: Entering singularity properties of the crack

8) From the main menu bar select **Special → Crack →Assign Seam…**

9) Select the crack line and click **Done** two times.
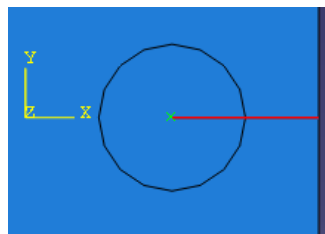
Figure 4.2.11: View of the crack seam

10) Remember to come back to step 5.ii.

7) <u>Applying a boundary condition and a load to the model</u>

      i.   Applying a boundary condition

1) In the **Module** list located under the toolbar, click **Load** to enter the Load module.

2) From the main menu bar, select **BC → Create**

3) In the **Create Boundary Condition** dialog box:

    a. Name the boundary condition *Articulated*

    b. From the list of steps, select *Sheetcrack* as the step in which the boundary condition will be activated.

    c. In the **Category** list, accept **Mechanical** as the default category selection.

    d. In the **Types for Selected Step** list, select **Displacement/Rotation** and click **Continue**

4) Select the two corners in which you will put the boundary conditions and click **Done**

5) Type U1 value equal zero to restrict X direction movement.
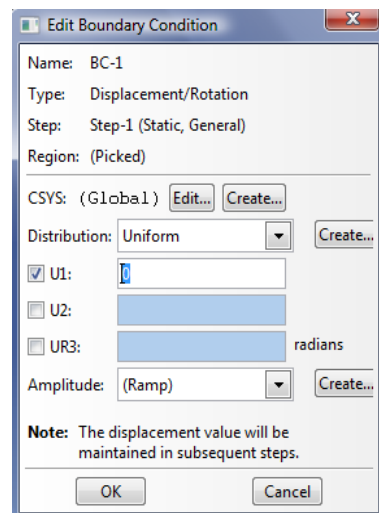
Figure 4.2.12: Entering Boundary Conditions

6) Click **OK**.

7) Now you have to repeat the steps before to restring the displacement in Y direction.

8) From the main menu bar, select **BC → Create**

9) In the **Create Boundary Condition** dialog box:

    a. Name the boundary condition *Articulated y*

b. From the list of steps, select *Sheetcrack* as the step in which the boundary condition will be activated.

c. In the **Category** list, accept **Mechanical** as the default category selection.

d. In the **Types for Selected Step** list, select **Displacement/Rotation** and click **Continue**

10) Select the middle point of the edge opposite to the crack in which you will put the boundary conditions and click **Done**

11) Type U2 value equal zero to restrict Y direction movement.

12) Click **OK**.

ii.   Applying loads

1) From the main menu bar, select **Load → Create**

2) In the **Create Load** dialog box:

a. Name the load *Stress*

b. From the list of steps, select *Sheetcrack* as the step in which the load will be applied.

c. In the **Category** list, accept **Mechanical** as the default category selection.

d. In the **Types for Selected Step** list, select **Pressure**, and click **Continue.**

3) In the viewport, select the top and bottom edge of the sheet as the surface to which the load will be applied. Click **Done**.

4) In the **Edit Load** dialog box:

a. Enter a magnitude of -200 for the load. The sign minus is because it is the way of put and stress using pressure. The direction of the load is opposite to the pressure.

b. Accept the default Amplitude selection

c. Click **OK** to create the load and to close the dialog box.
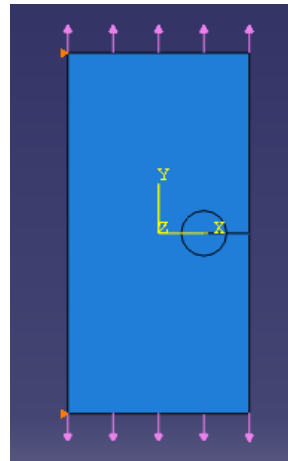
**Figure 4.2.13:** View of the finished model

8) <u>Meshing the model</u>

i. Assigning mesh controls

1) In the **Module** list located under the toolbar, click **Mesh** to enter the Mesh module

2) From the main menu bar, select **Mesh → Controls**

3) Select firs the circular region to assign Mesh controls and click **Done**

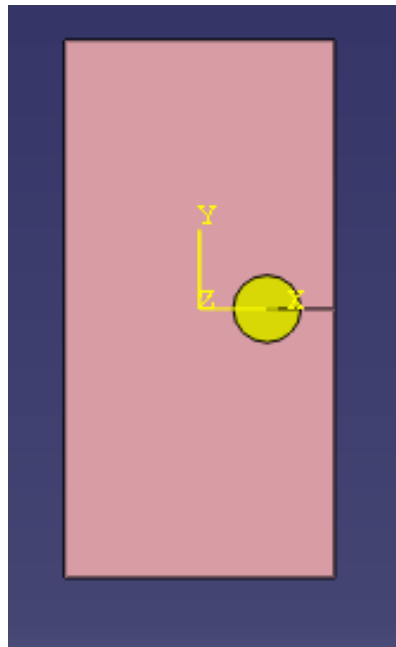4) In the dialog box, select **Quad-dominated** as element shape and **Sweep** as Technique. Click **OK**.
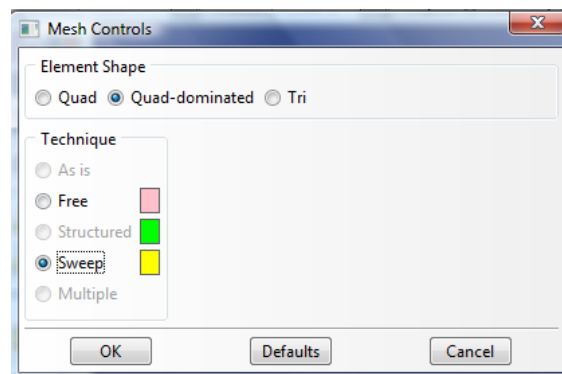
Figure 4.2.14: Assigning mesh controls

5) Select now the other region and click **Done**.

6) In the dialog box, select **Quad** as element shape, **Free** as Technique and **Medial axis** as algorithm. Click **OK**.
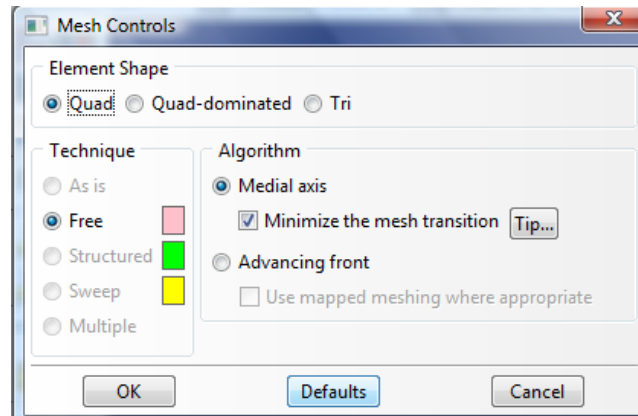
**Figure 4.2.15:** Assigning mesh controls

7) You should obtain the following figure

Figure 4.2.16: View of the sheet with the assigned mesh controls

ii.   Assigning an ABAQUS element type

1) From the main menu bar, select **Mesh → Element Type**. Select both regions and click **Done**.

2) In the dialog box, select **Quadratic** as **Geometric Order** click off **Reduced integration**. You should obtain **CPS8** elements (An 8-nodes biquadratic plane stress quadrilateral).
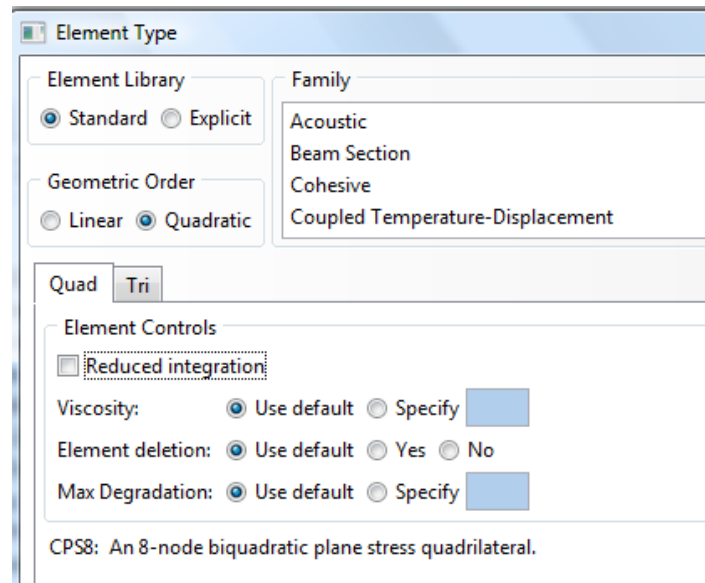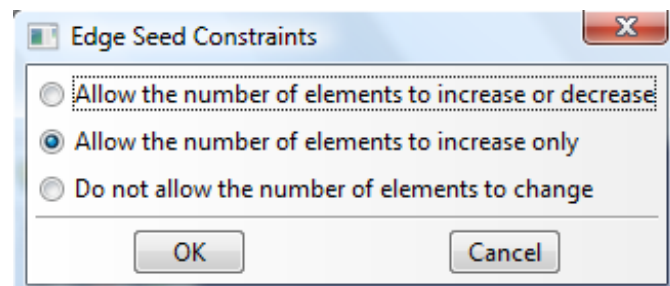
Figure 4.2.17: Assigning element type

3) Click **OK** and then **Done**.

iii.   Creating the mesh

1) From the main menu bar, select **Seed → Edge** by number or click .

2) Select the circle from the viewport and click **Done**

3) Type 20 as Number of elements along the edges

4) Click **Constrains** and put **Allow the number of element to increase only**.

Figure 4.2.18: Assigning edge seed constraints

5) Click **OK** and the puss **Enter** and then **Done**.

6) From the main menu bar, select **Mesh → Instance**

7) Click **Yes**.
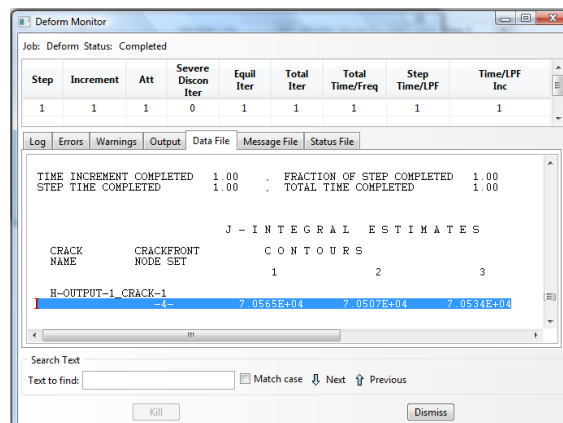
You should obtain this kind of mesh:
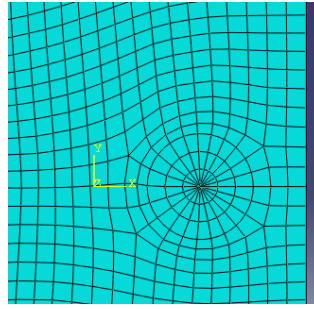
Figure 4.2.19: View of the special mesh around the crack tip

9) <u>Creating and submitting an analysis job</u>

1) In the **Module** list located under the toolbar, click **Job** to enter the Job module.

2) From the main menu bar, select **Job → Create** to create a job

3) Name the job *Deform*.

4) Click **Continue** to create the job

5) In the **Description** field, type *Crack tutorial*.

6) Click **OK** to accept all the default job settings and to close the dialog box

7) From the main menu bar, select **Job → Manager** to start the **Job Manager**.

8) From the buttons on the right edge of the **Job Manager**, click **Submit** to submit your job for analysis.

9) When the job completes successfully, you are ready to view the results of the analysis.

10) Click **Monitor**.

11) Click **Data File** and down you can read the J-values.

Figure 4.2.20: View of the J-integral values

12) Click **Dismiss**

13) Click now **Results** and you will see the deformed sheet.
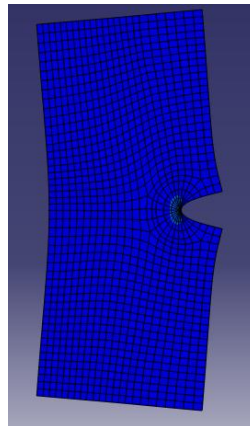
Figure 4.2.21: View of the deformed sheet

4.3. <u>BUILDING 3D CRACKS</u>

- ## CREATING AND ANALYZING A 3D CRACK

The following section leads you through the ABAQUS/CAE modelling process by visiting each of the modules and showing you the basic steps to create and analyze a 3D crack. To illustrate each of the steps, you will first create a model of a steel sheet with an edge crack and load its top and bottom surface as it is shown in the figure.
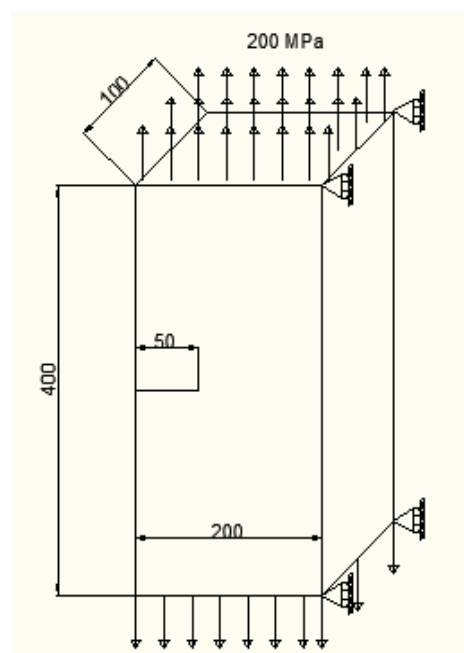

Figure 4.3.1: A loaded sheet with a crack edge

4) <u>Creating a part</u>

To create the sheet:

1) In the Model Tree, double-click the **Parts** container to create a new part.

2) Name the part *Brick*. Accept the default settings of a three-dimensional, deformable body and a solid, extruded base feature. In the **Approximate size** text field, type 1000. And click Continue.

3) Draw a rectangle of size 200 x 400 mm using

4) Type a thickness of 100 mm. Click **Done**.

You should obtain these figures:
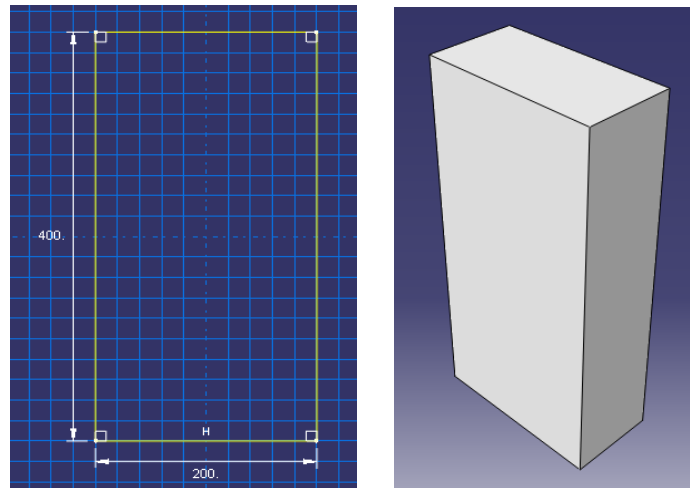
Figure 4.3.2: Sketch of the rectanle

5) Then you must draw the size of the crack you want to analyze and the circle in which you will create the special mesh. For do it go to **shape → shell → extrude** or click on .

6) Select the plane for the shell extrusion

7) Select an edge or axis that will appear

8) Draw the crack and the circle at the same time.

9) Put the **Extrude direction** in the right direction. Type the thickness of your sheet and click **Keep internal boundaries**.
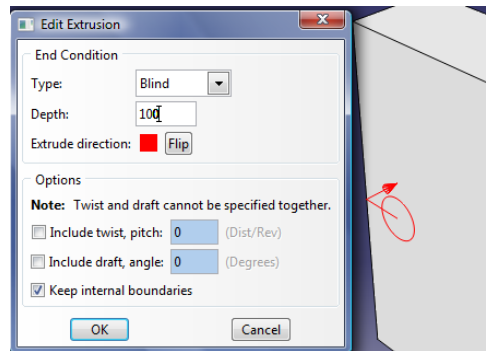
Figure 4.3.3: View of the Edit Extrusion dialog box and the Extrude direction.

As a result of doing this, you should obtain:
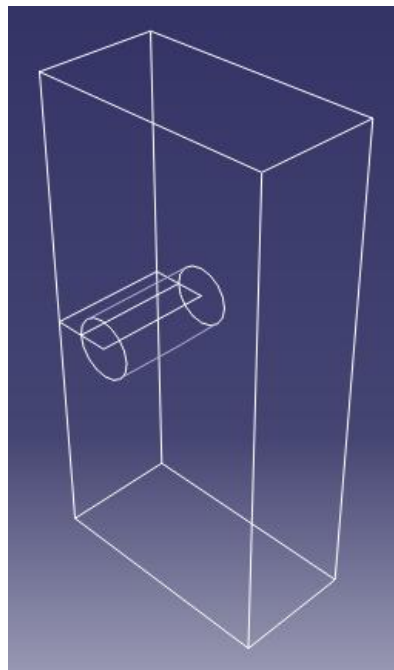
Figure 4.3.4: View of the sheet with the crack and the circle

5) <u>**Creating a material**</u>

1) In the Model Tree, double-click the **Materials** container to create a new material.

2) Name the material *Steel*.

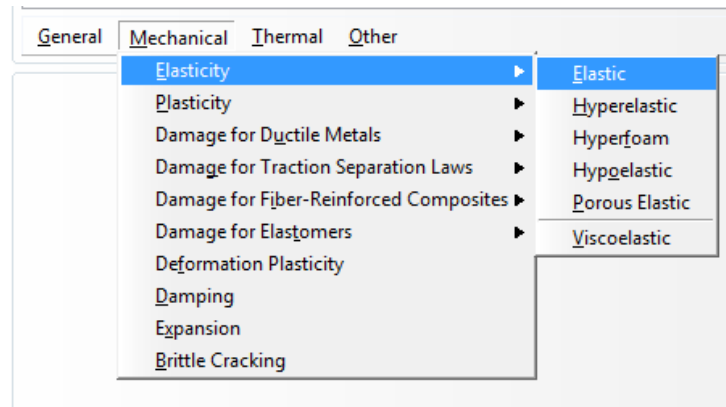**Figure 4.3.5:** Submenus available under the Mechanical menu.

3) From the material editor's menu bar, select **Mechanical ⟶ Elasticity ⟶ Elastic**.

4) Type a value of `200` for Young's modulus and a value of `0.3` for Poisson's ratio in the respective fields, as shown in Figure 6

Figure 4.3.6: Entering data values for the elastic material properties.

5) Click **OK** to exit the material editor.

6) <u>**Defining and assigning section properties**</u>

   i) **Defining a homogeneous solid section**

1) In the Model Tree, double-click the **Sections** container to create a section.

2) In the **Create Section** dialog box:

   a. Name the section *BrickSection*

   b. In the **Category** list, accept **Solid** as the default category selection

   c. In the **Type** list, accept **Homogeneous** as the default type selection.

   d. Click **Continue.**

3) In the dialog box:

    a. Accept the default selection of *Steel* for the **Material** associated with the section.

    b. Accept the default value of 1 for the Plane stress/strain thickness.

    c. Click **OK.**

ii) Assigning the section to the sheet.

1) From the main menu bar, select **Assign → Section**

2) Click on the different regions to select the region to which the section will be applied.

3) Click **Done** in the prompt area to accept the selected geometry.

4) Accept the default selection of *BrickSection* as the section, and click **OK**.

7) Assembling the model

1) In the **Module** list locate under the toolbar, click **Assembly** to enter the Assembly module

2) From the main menu bar, select **Instance → Create**

3) In the dialog box, select **Sheet** and make sure that you are in **Independent instance** type. Then click **OK**.

8) Defining your analysis steps

iii. Creating an analysis step

To create a general, static analysis step:

1) In the **Module** list located under the toolbar, click **Step** to enter the Step module.

2) From the main menu bar, select **Step → Create** to create a step

3) Name the step *Brickcrack*

4) From the list of available general procedures in the **Create Step** dialog box, select **Static, General** and **Continue.**

5) The **Basic** tab is selected by default. In the **Description** field, type *Crack of the Brick.*

6) Click **OK** to create the step and to exit the **Edit Step** dialog box.

iv. Requesting data output

You should do this step after doing the point 6 of this tutorial because you need to create the crack before requesting the data output in the **History Output Request**.

To examine your output requests:

1) From the main menu bar, select Output → Field Output Request → Manager.

2) Review the default output request that ABAQUS/CAE generates for the **Static, General** step you created and named *Brickcrack.*

3) On the right side of the **Field Output Request Manager**, click **Edit** to view more detailed information about the output request.

4) On the **Edit History Output Request** dialog box select **Contour Integral** in the Domain Tree and select the Crack you have created.

5) Type 3 in the Number of contours and J-integral as Type.

6) Click **OK** to close the field output editor and save the changes.

7) Click **Dismiss** to close the **History Output Request Manager**.

9) <u>Creating the crack and the seam</u>

1) In the **Module** list located under the toolbar, click **Interaction** to enter the Interaction module.

2) From the main menu bar, select **Special → Crack → Create.**

3) Name the crack *Crack* and click **Continue**.

4) Pick the edge of the crack and then **Done.**
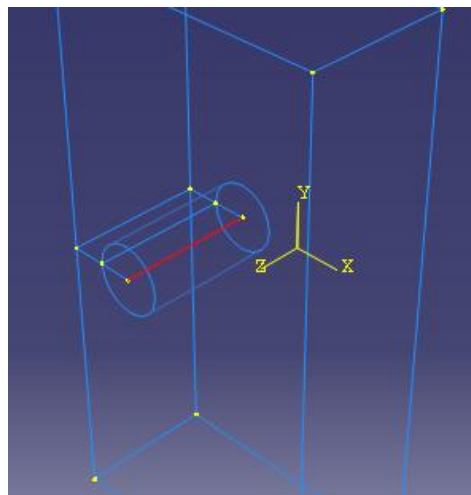
Figure 4.3.7: View of the crack edge.

5) Select **q-vector** as method to specify the crack extension direction.

6) Select the beginning of the crack length and the crack tip.
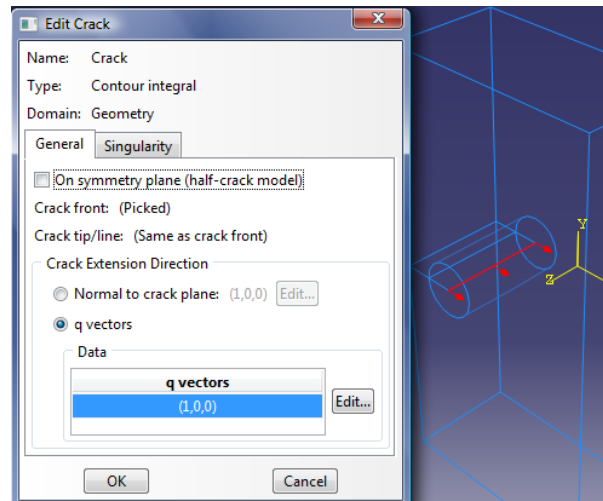
Figure 4.3.8: View of the Edit crack box

7) Then click Singularity and type 0.25 as Midside node parameter and select Collapsed element side, duplicate nodes. Click **OK**.
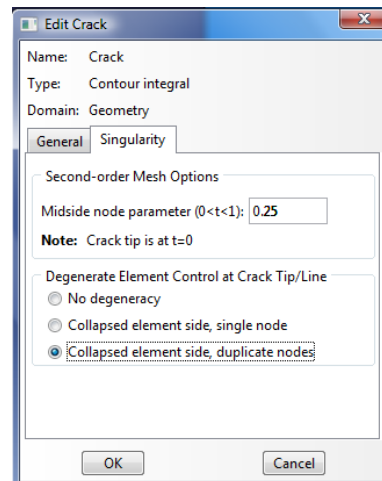
Figure 4.3.9: Entering singularity properties of the crack

8) From the main menu bar select **Special → Crack →Assign Seam…**

9) Select the crack surface and click **Done** two times.
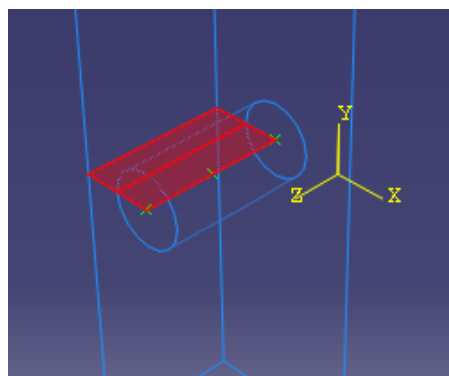
Figure 4.3.10: View of the crack seam

10) Remember to come back to step 5.ii.

10) **Applying a boundary condition and a load to the model**

iii.   **Applying a boundary condition**

1) In the **Module** list located under the toolbar, click **Load** to enter the Load module.

2) From the main menu bar, select **BC → Create**

3) In the **Create Boundary Condition** dialog box:

   a. Name the boundary condition *Articulated*

   b. From the list of steps, select *Brickcrack* as the step in which the boundary condition will be activated.

   c. In the **Category** list, accept **Mechanical** as the default category selection.

   d. In the **Types for Selected Step** list, select **Displacement/Rotation** and click **Continue**

4) Select the four corners of the surface in which you will put the boundary conditions and click **Done**

5) Type U1 value equal zero to restrict X direction movement.
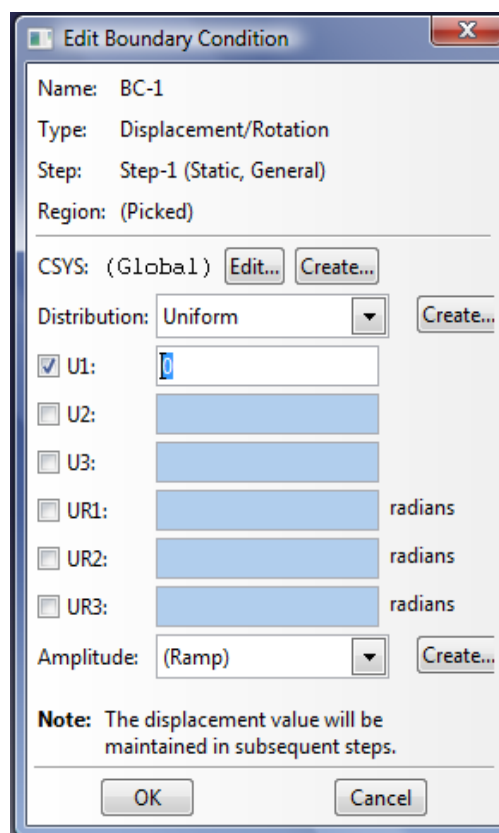
Figure 4.3.11: Entering Boundary Conditions

6) Click **OK**.

7) Now you have to repeat the steps before to restring the displacement in Y direction.

8) From the main menu bar, select **BC → Create**

9) In the **Create Boundary Condition** dialog box:

    a. Name the boundary condition *Articulated y*

    b. From the list of steps, select *Brickcrack* as the step in which the boundary condition will be activated.

    c. In the **Category** list, accept **Mechanical** as the default category selection.

    d. In the **Types for Selected Step** list, select **Displacement/Rotation** and click **Continue**

10) Select the middle point of the edges opposite to the crack in which you will put the boundary conditions and click **Done**

11) Type U2 value equal zero to restrict Y direction movement.

12) Click **OK**.

iv.   Applying loads

1) From the main menu bar, select **Load → Create**

2) In the **Create Load** dialog box:

    a. Name the load *Stress*

    b. From the list of steps, select *Brickcrack* as the step in which the load will be applied.

    c. In the **Category** list, accept **Mechanical** as the default category selection.

    d. In the **Types for Selected Step** list, select **Pressure**, and click **Continue.**

3) In the viewport, select the top and bottom surface of the sheet as the surface to which the load will be applied. Click **Done**.

4) In the **Edit Load** dialog box:

    a. Enter a magnitude of -200 for the load. The sign minus is because it is the way of put and stress using pressure. The direction of the load is opposite to the pressure.

    b. Accept the default Amplitude selection

    c. Click **OK** to create the load and to close the dialog box.
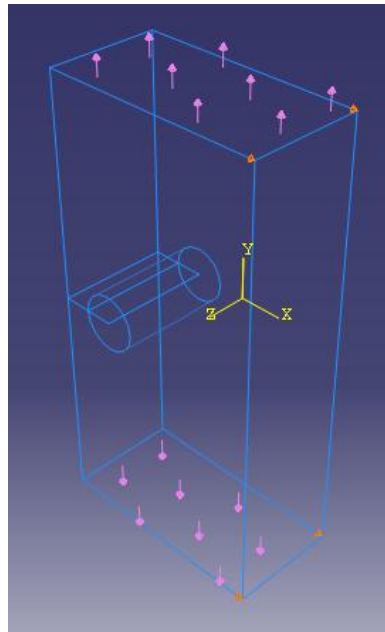
Figure 4.3.12: View of the finished model

11) <u>Meshing the model</u>

iv.   Assigning mesh controls

1) In the **Module** list located under the toolbar, click **Mesh** to enter the Mesh module

2) From the main menu bar, select **Mesh → Controls**

3) Select firs the circular region to assign Mesh controls and click **Done**

4) In the dialog box, select **Wedge** as element shape and **Sweep** as Technique. Click **OK**.
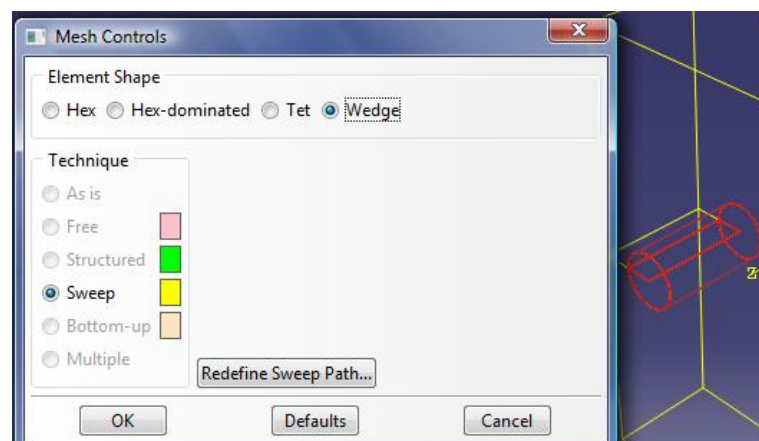
Figure 4.3.13: Assigning mesh controls

Note: In a 3D model, when you try to create the deformed mesh around the crack tip, you must use Wedge Element Shape. ABAQUS know that you are

working with quadrangular elements and will analyze it as degenerated quadrangular elements. Also, with 3D geometry, it is difficult to get the mesh to sweep around the singularity and then sweep from one face to another. To force this you need to make sure that there is just one element along the length of the line within the circular region. As it is a swept mesh you only need to assign an element number along this line on one side as the seeding scheme will propagate to the other side.

5) Select now the other region and click **Done**.

6) In the dialog box, select **Hex** as element shape, **Sweep** as Technique and **Medial axis** as algorithm. Click **OK**.
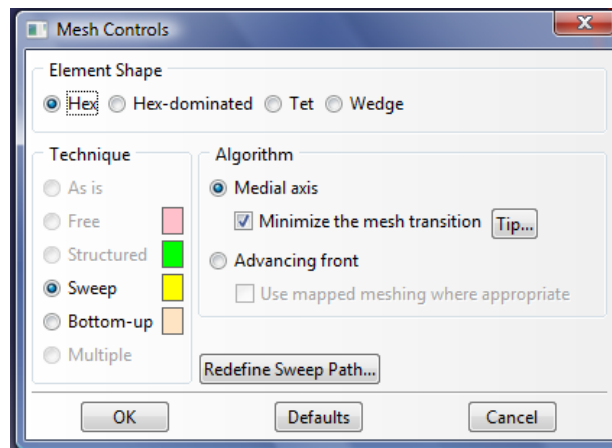
Figure 4.3.14: Assigning mesh controls

7) You should obtain the following figure
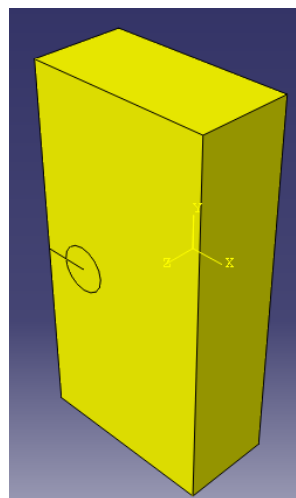
Figure 4.3.15: View of the sheet with the assigned mesh controls

v.   Assigning an ABAQUS element type

1) From the main menu bar, select **Mesh → Element Type**. Select both regions and click **Done**.

2) In the dialog box, select **Quadratic** as **Geometric Order** click off **Reduced integration**. You should obtain **C3D20** elements (A 20-node quadratic brick).
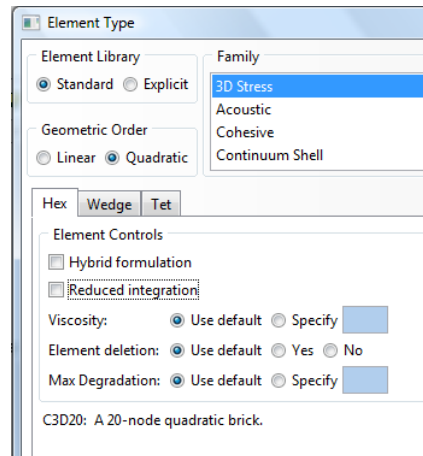
Figure 4.3.16: Assigning element type

3) Click **OK** and then **Done**.

vi.   Creating the mesh

1) From the main menu bar, select **Seed → Edge** by number or click .

2) Select the circle from the viewport and click **Done**

3) Type 20 as Number of elements along the edges

4) Click **Constrains** and put **Allow the number of element to increase only**.

Figure 4.3.17: Assigning edge seed constraints

5) Then select the line inside the circular region and click **Done**.

6) Type 1 as Number of elements along the edge

7) Click **Constrains** and put **Do not allow the number of elements to change**.
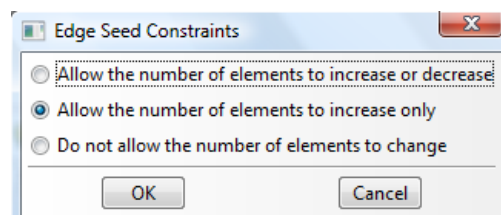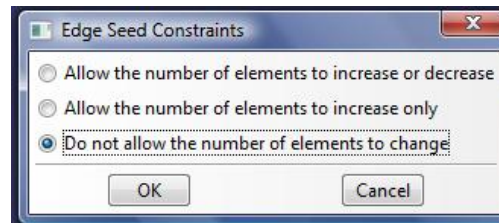
Figure 4.3.18: Assigning edge seed constrains

8) Click **OK** and the puss **Enter** and then **Done**.

9) From the main menu bar, select **Mesh → Instance**

10) Click **Yes**.

You should obtain this kind of mesh:
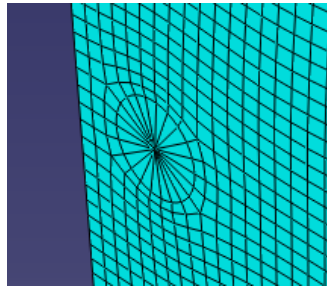
Figure 4.3.19: View of the special mesh around the crack tip

12) **Creating and submitting an analysis job**

1) In the **Module** list located under the toolbar, click **Job** to enter the Job module.

2) From the main menu bar, select **Job → Create** to create a job

3) Name the job *Deform*.

4) Click **Continue** to create the job

5) In the **Description** field, type *Crack tutorial*.

6) Click **OK** to accept all the default job settings and to close the dialog box

7) From the main menu bar, select **Job → Manager** to start the **Job Manager**.

8) From the buttons on the right edge of the **Job Manager**, click **Submit** to submit your job for analysis.

9) When the job completes successfully, you are ready to view the results of the analysis.

10) Click **Monitor**.

11) Click **Data File** and down you can read the J-values.

Figure 4.3.20: View of the J-integral values

12) Click **Dismiss**

13) Click now **Results** and you will see the deformed sheet.
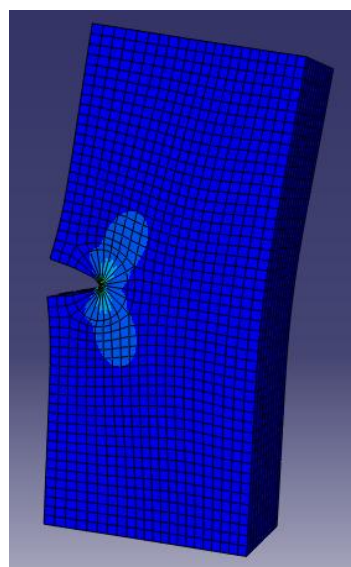
Figure 4.3.21: View of the deformed sheet

5. RESULTS AND DISCUSSION

5.1. <u>INTRODUCTION</u>

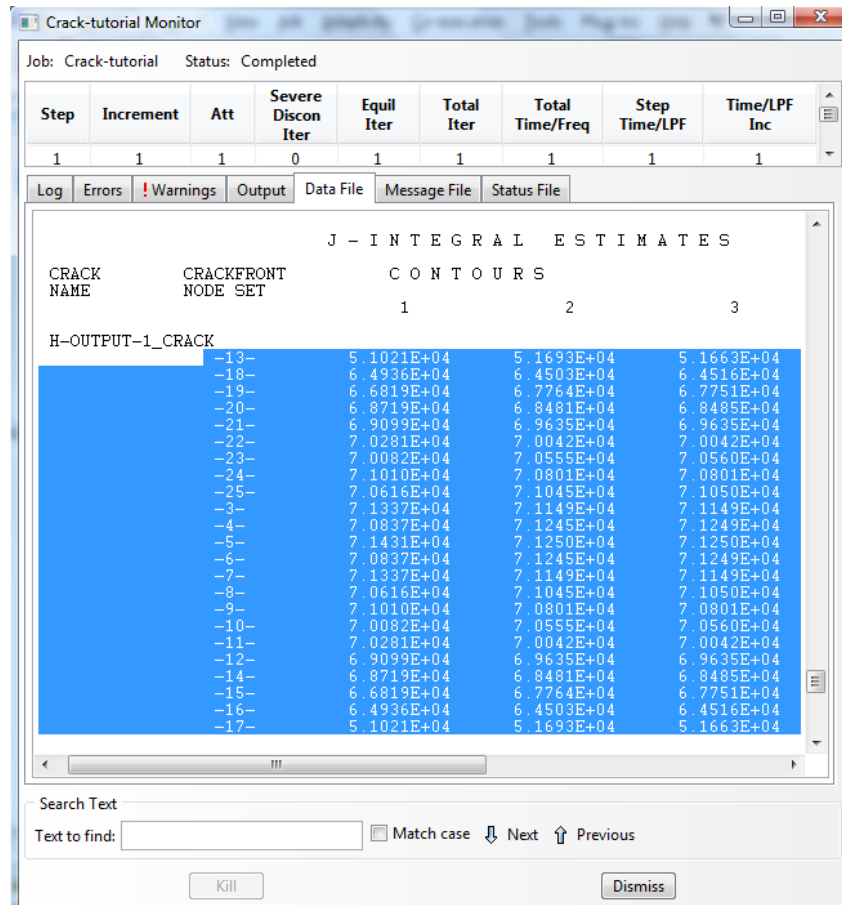The program that I have used to develop the project, Abaqus, is an extremely sophisticated finite element program, designed first and foremost to model the behaviour of solids and structures under externally applied loading. Abaqus includes the following characteristics:

- Capabilities for static and dynamic problems

- The ability to model very large shape changes in solids, in 2D and 3D.

- A very large element library, with a lot of continuum elements, beam elements, shell and plate elements, among others.

- A sophisticated capability to model contact between solids

- An advanced material library, including the usual elastic and elastic-plastic solids; models for foams, concrete, soils, piezoelectric materials, and many other.

- Capabilities to model a number of phenomena of interest, including vibrations, coupled fluid/ structure interactions, acoustics, buckling problems, and so on.

The main strength of Abaqus, however, is that it is based on a very sound theoretical framework. As practicing engineer, you may be called upon to make crucial decisions based on the results of computer simulations. While no computer program can ever be guaranteed free of bugs, Abaqus is among the more trustworthy codes. For this reason, Abaqus is used by a wide range of industries, including aircraft manufactures, automobile companies, oil companies and microelectronics industries, as well as national laboratories and research universities.

5.2. <u>2D CRACKS</u>

- **EDGE CRACK IN A FINITE WIDTH SHEET: UNIAXIAL TENSILE STRESS WITH TENSILE STRESS WITHOUT BENDING CONSTRAINS**

The first modelling of a 2D crack is a sheet of width b and height 2h containing a crack of length a in the middle of one side. It is subjected, at the ends, to a uniform tensile stress σ perpendicular to the direction of the crack.

The published data of this kind of crack are take from <u>Stress intensity factors</u> book and shows the results for K_I as a polynomial in a/b which is accurate to within 1% for all h/b≥ 1.0 and a/b ≤ 0.6; it is

$$\frac{K_I}{K_0} = 1.12 - 0.23\left(\frac{a}{b}\right) + 10.6(a/b)^2 - 21.7(a/b)^3 + 30.4(a/b)^4 \qquad \textbf{Equation 5.1}$$

Where K_0 is the stress intensity factor for an isolated crack of length 2a in a sheet subjected to a uniform stress σ, and is given by

$$K_0 = \sigma\sqrt{\pi a} \quad \textbf{Equation 5.2}$$

To develop this analysis I have created several sheets of different sizes. To analyse this crack I have to assume some boundary conditions to create the same conditions as the published data. The boundary conditions that I have assumed are:

- The right corners, top and base have the restriction of no movement in x direction
- The middle point of the right edge has the restriction of no movement in y direction.

With the first boundary conditions I let the sheet to open freely but I do not let the right corners to move along the y direction. It is needed because the sheet does not have to rotate. And the second boundary condition is needed to do not let the sheet to move up or down.
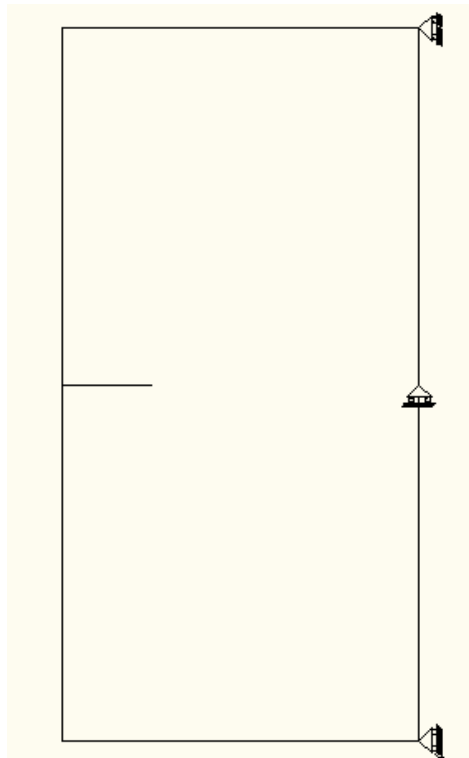
Figure 5.1

The conditions that I have assumed that are the same for all the performed tests are:

- Uniform tensile stress σ of 200 MPa perpendicular to the crack length
- Common engineering material with Young's Modulus E=200 GPa and Poisson's ratio υ=0.3.

- **200 x 400 mm sheet with a crack of 50 mm**

For this kind of sheet and crack, the published data shows that Y=1.5 ± 0.025 and following the equation for the stress intensity factor,

$$K_I = Y\sigma\sqrt{\pi a} \qquad \textbf{Equation 5.3}$$

Where Y=1.5, σ=200 N/mm^2 and a=50mm is the crack length. So K_I takes the value of,

$$\textbf{K}_\textbf{I} = \textbf{3759.94} \pm \textbf{62.67 N/mm}^\textbf{3/2}$$

Modelling now the crack with Abaqus, I have used a circle around the crack with 25mm of ratio where I want to create the degenerated mesh.

Figure 5.2

Figure 5.3

As we can see in the picture, the values of the J-integral are 7.0549e+04, 7.0522e+04 and 7.0539e+04 N/m. Following the equation showed in the literature,

$$K_I = \sqrt{E\left[\frac{N}{mm^2}\right] xJ\left[\frac{N}{mm}\right]} \qquad \textbf{Equation 5.4}$$

I have taken the last value because it belongs to the third integral and takes more elements to do the approximation. So the result that I have obtained is,

$$\textbf{K}_\textbf{I} = \textbf{3756.04 N/mm}^{\textbf{3/2}}$$

As we can see the values are very close, the error is smaller than 1% so we can conclude it is a great approximation.

- **100 x 300 mm sheet with a crack of 40 mm**

For this kind of sheet and crack, the published data shows that Y=2.11 ± 0.025 and following the equation for the stress intensity factor,

$$K_I = Y\sigma\sqrt{\pi a} \qquad \textbf{Equation 5.5}$$

Where Y=2.11, σ=200 N/mm^2 and a=40mm is the crack length. So K_I takes the value of,

$$\textbf{K}_\textbf{I} = \textbf{4730.61 ± 56.05 N/mm}^{\textbf{3/2}}$$

Modelling now the crack with Abaqus, I have used a circle around the crack with 20mm of ratio where I want to create the degenerated mesh.
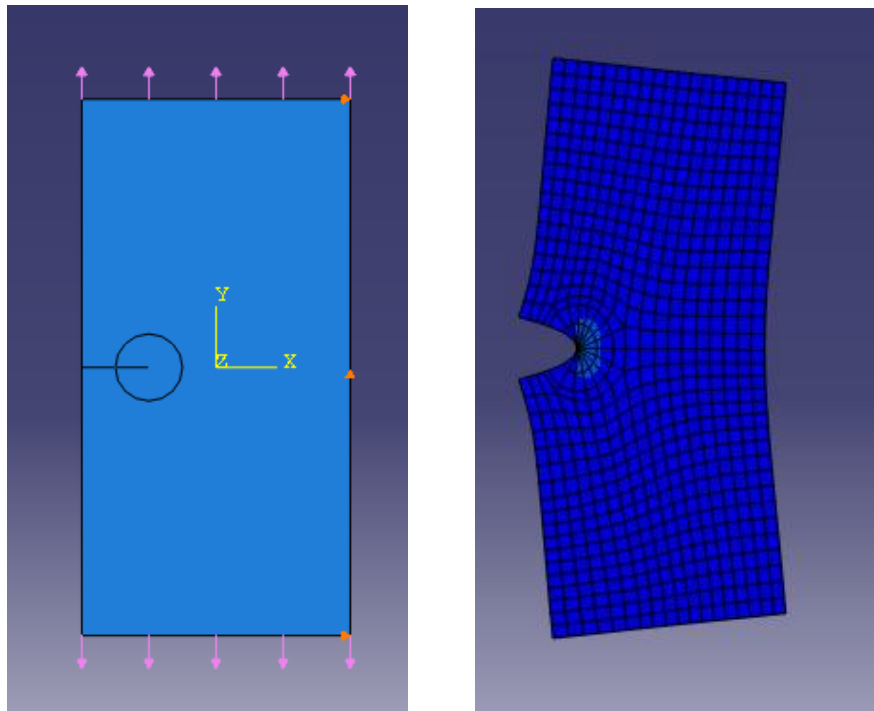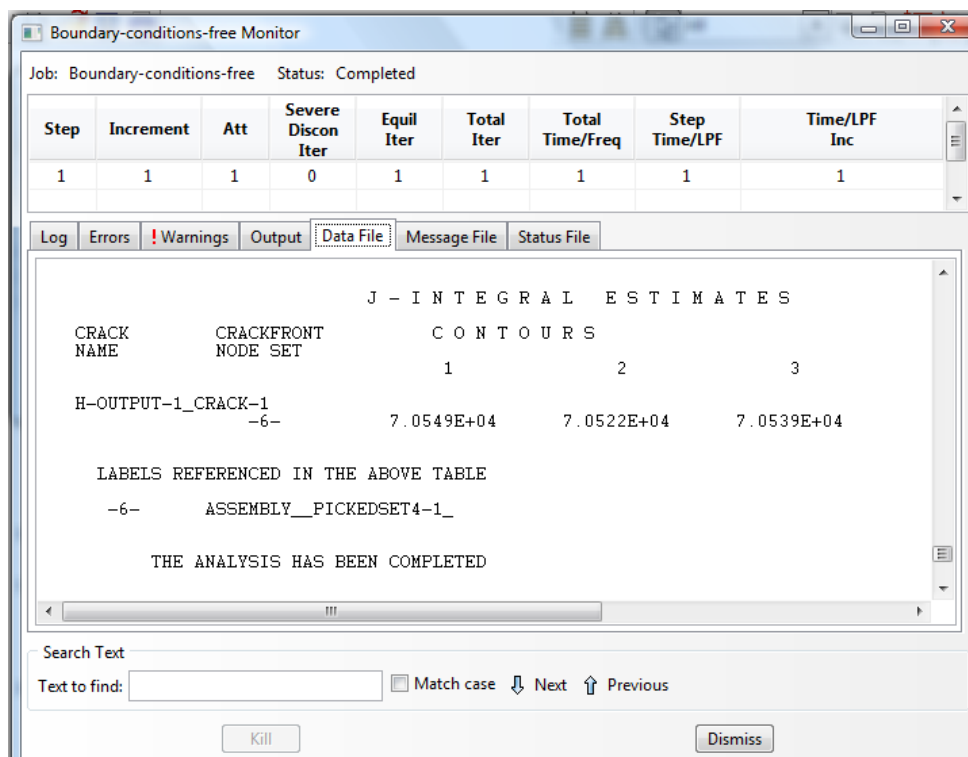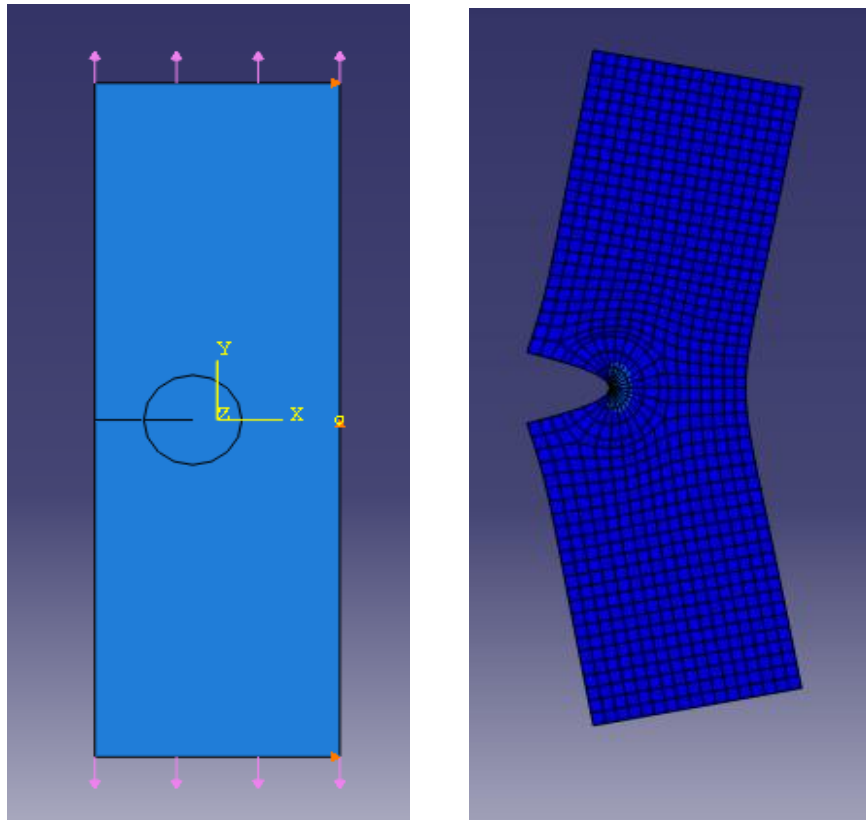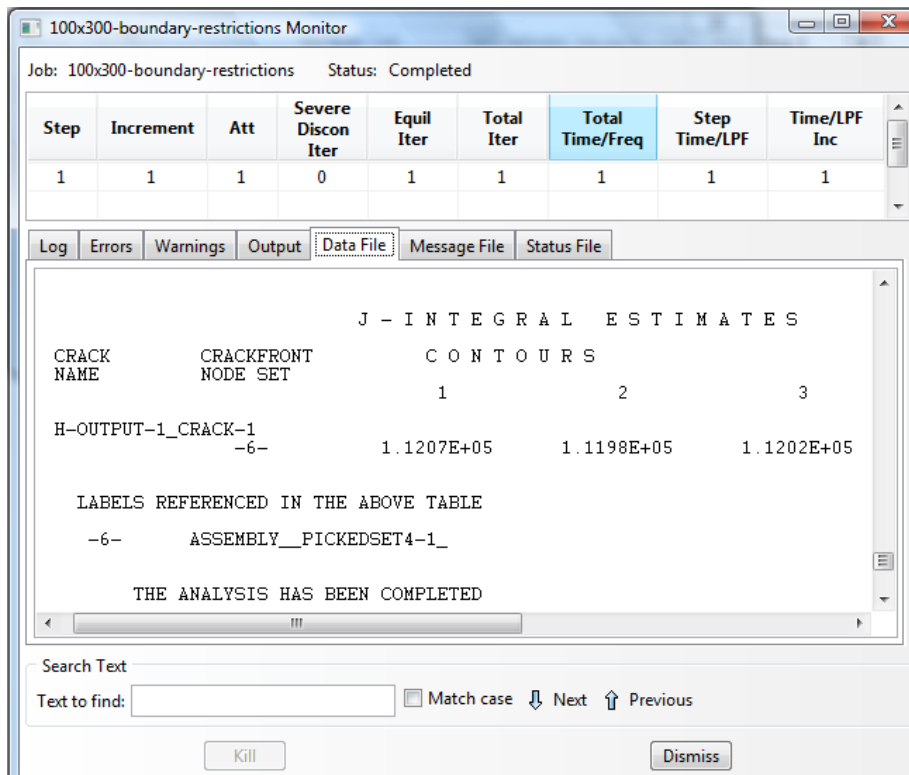
Figure 5.4

Figure 5.5

As we can see in the picture, the values of the J-integral are 1.1207e+05, 1.1198e+05 and 1.1202e+05 N/m. Following the equation showed in the literature,

$$K_I = \sqrt{E\left[\frac{N}{mm^2}\right] x J\left[\frac{N}{mm}\right]} \qquad \textbf{Equation 5.6}$$

I have taken the last value because it belongs to the third integral and takes more elements to do the approximation. So the result that I have obtained is,

$$\textbf{K}_I = \textbf{4733.27 N/mm}^{3/2}$$

As we can see the values are very close, the error is smaller than 1% so we can conclude, like in the previous test, that it is a great approximation.

The results that I have obtained in this first modelling are really good. The data obtained are in truth close to the published one, with errors smaller than 1%. So I can conclude that this is the way to be followed to analyse this kind of crack.

- **EDGE CRACK IN A RECTANGULAR SHEET: UNIFORM NORMAL DISPLACEMENT**

In the second analysis a sheet of width b and height 2h containing a crack of length a in the middle of one side is subjected at the ends to a uniform displacement in a direction perpendicular to the crack line. No rotation of the ends of the sheet is assumed.

The published data are obtained for the <u>Stress intensity factors</u> book that obtained the result using conformal mapping techniques and the results for the stress intensity factor are shown as curves K_I/K_0 vs. a/b in **Figure 5.7** for various values of h/b. K_0 is given by

$$K_0 = \sigma\sqrt{\pi a} \quad \textbf{Equation 5.7}$$

where σ is the average stress on the ends of the sheet.

To develop this analysis I have created several sheets of different sizes. To analyse this model I have to assume some boundary conditions to create the same

conditions as the published data. Because of the sheet must move only in the perpendicular direction, I have created two plates that have two functions. The plates have also high Young's Modulus because I want they not affect the results. Therefore,

- The plates with the rails does not allow the sheet to bend
- The plates also simulate the towing hooks that probably were used in the real experiments.
- The middle point of the right edge has the restriction of no movement in y direction. Consequently, the sheet will not translate in this direction.
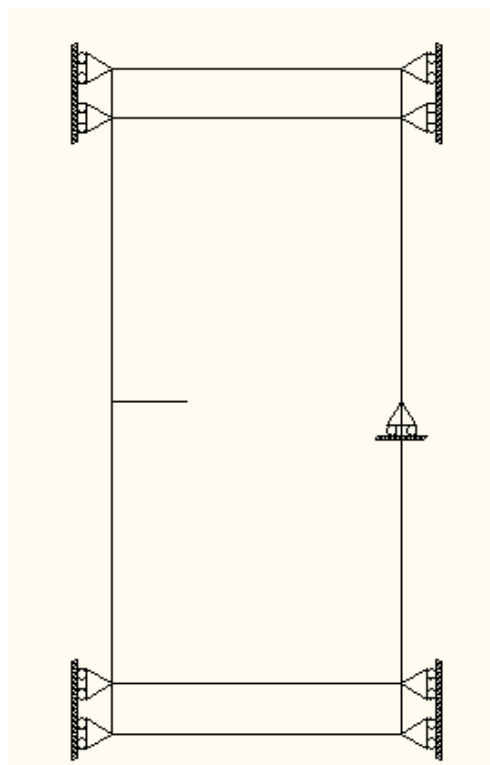
Figure 5.6

The conditions that I have assumed that are the same for all the performed tests are:

- Uniform tensile stress σ of 200 MPa perpendicular to the crack length
- Common engineering material with Young's Modulus E=200 GPa and Poisson's ratio υ=0.3 for the sheet part.
- Material with Young's Modulus E=20000 GPa and Poisson's ratio υ=0.3 for the plates.

Figure 5.7

- **200 x 184 mm sheet with a crack of 40 mm**

For this kind of sheet and crack, the published data shows that Y=1 ± 0.02 and following the equation for the stress intensity factor,

$$K_I = Y\sigma\sqrt{\pi a} \qquad \textbf{Equation 5.8}$$

Where Y=1, σ=200 N/mm^2 and a=40mm is the crack length. So K_I takes the value of,

K_I = 2241.99 ± 44.84 N/mm$^{3/2}$

Modelling now the crack with Abaqus, I have used a circle around the crack with 20mm of ratio where I want to create the degenerated mesh.
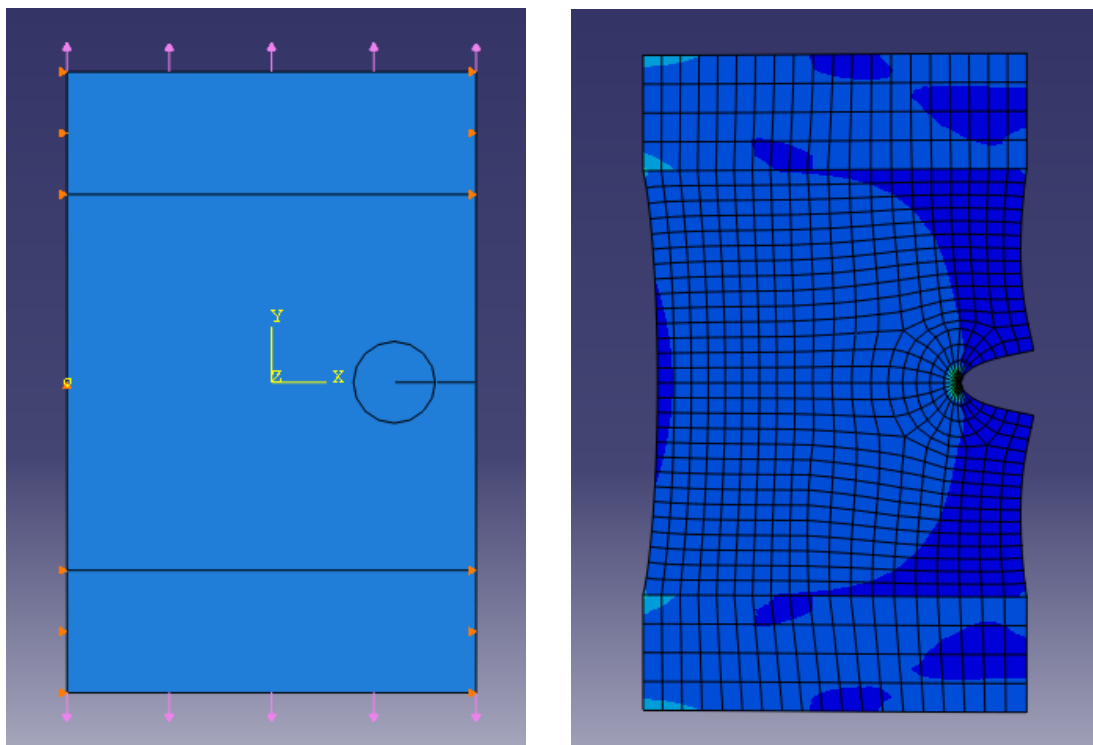
Figure 5.8

Figure 5.9

As we can see in the picture, the values of the J-integral are 2.241e+04, 2.239e+04 and 2.2398e+04 N/m. Following the equation showed in the literature,

$$K_I = \sqrt{E\left[\frac{N}{mm^2}\right] x J\left[\frac{N}{mm}\right]} \qquad \textbf{Equation 5.9}$$

I have taken the last value because it belongs to the third integral and takes more elements to do the approximation. So the result that I have obtained is,

$$\textbf{K}_I = \textbf{2116.51 N/mm}^{3/2}$$

As we can see the values are close but not as close as the model before. The error that I have obtained is around 4%. However, this could be because of the edge effects, so in the next analysis I will introduce more material around the crack.
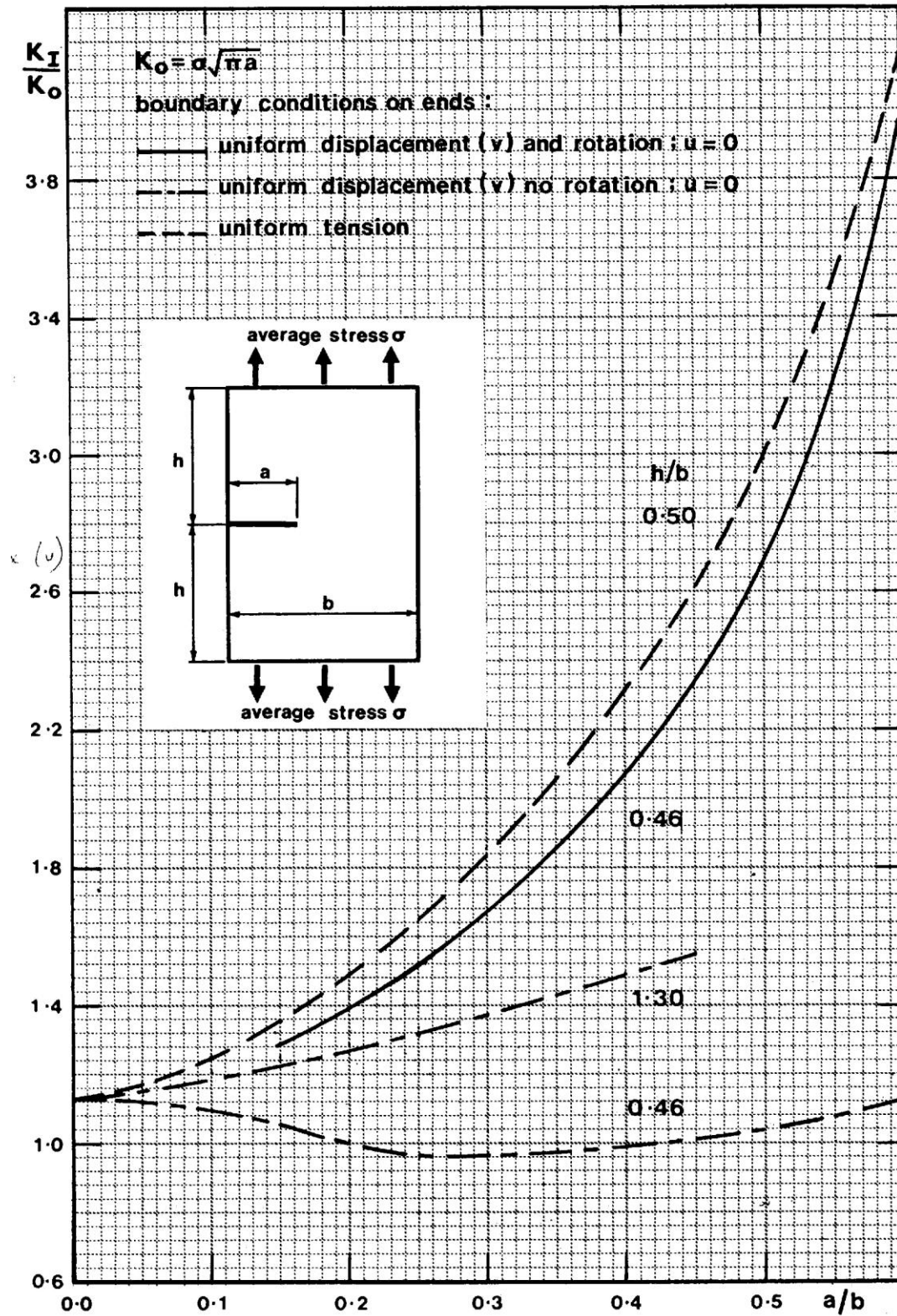
- **200 x 520 mm sheet with a crack of 50 mm.**

For this kind of sheet and crack, the published data shows that Y=1.32 ± 0.02 and following the equation for the stress intensity factor,

$$K_I = Y\sigma\sqrt{\pi a} \qquad \textbf{Equation 5.10}$$

Where Y=1.32, σ=200 N/mm^2 and a=50mm is the crack length. So K_I takes the value of,

$$K_I = 3308.75 \pm 66.18 \text{ N/mm}^{3/2}$$

Modelling now the crack with Abaqus, I have used a circle around the crack with 25mm of ratio where I want to create the degenerated mesh.
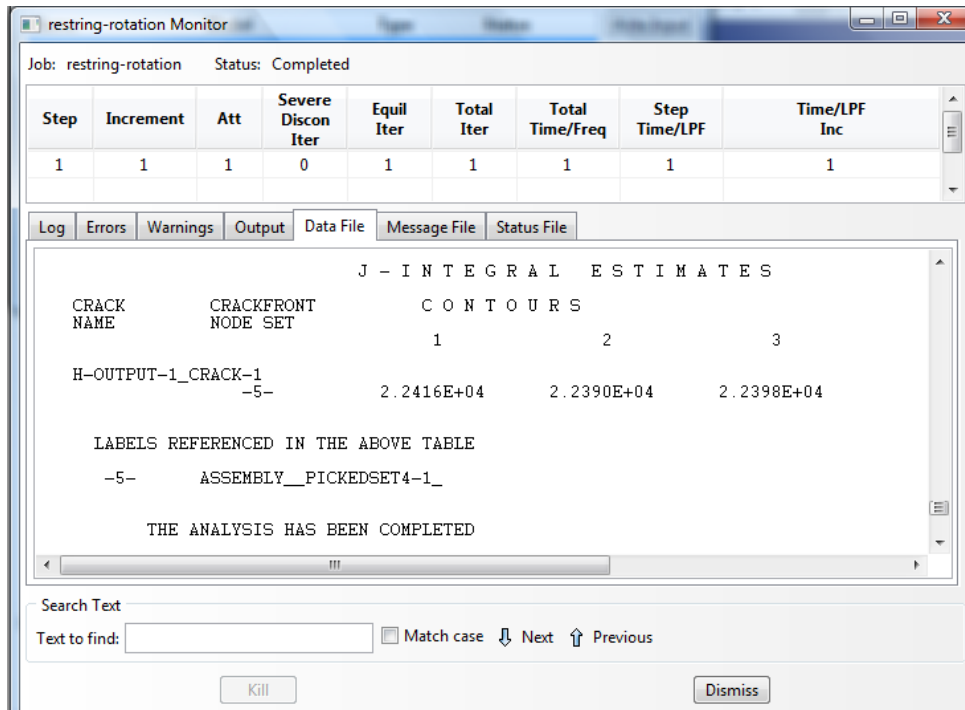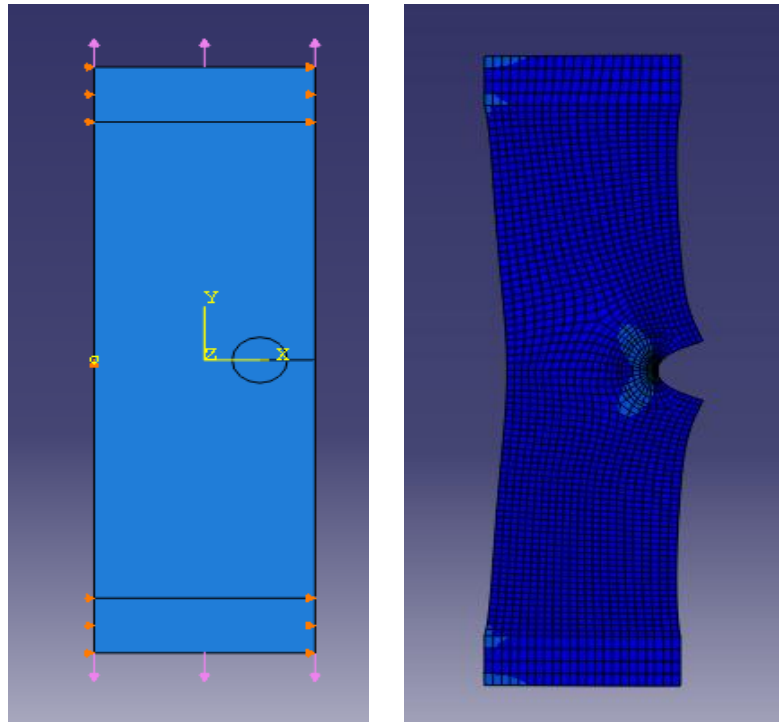

Figure 5.10
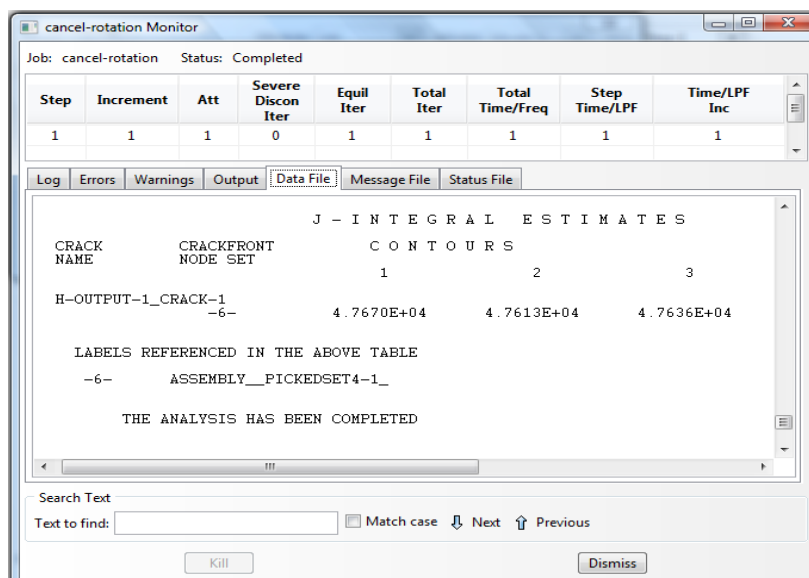
Figure 5.11

As we can see in the picture, the values of the J-integral are 4.7670e+04, 4.7613e+04 and 4.7636e+04 N/m. Following the equation showed in the literature,

$$K_I = \sqrt{E\left[\frac{N}{mm^2}\right] xJ\left[\frac{N}{mm}\right]} \qquad \textbf{Equation 5.11}$$

I have taken the last value because it belongs to the third integral and takes more elements to do the approximation. So the result that I have obtained is,

$$K_I = 3086.62 \text{ N/mm}^{3/2}$$

As we can see the values are close but not as close as the model before. The error that I have obtained is around 5%. This is quite interesting because I have introduced more material around the rack so the errors owing to the effect of the approximation that I have done with the plate with the rails should be lower; but the error is bigger than the analysis before.

This could be due to the fact that the published data are not too recent or that the effect of the plates is bigger than what I had thought before. Apart from that the result are quite good.

- **SLANT EDGE CRACK IN A RECTANGULAR SHEET: UNIFORM UNIAXIAL TENSILE STRESS**

A rectangular sheet of width b and length 2.5b, containing an edge crack of length a, is subjected to a uniform uniaxial tensile stress σ at the ends. The crack is located eccentrically a distance b from one end and inclined at an angle β towards the other end.

The published data are obtained for the Stress intensity factors book that obtained the result using boundary collocation and the results for the stress intensity factor are shown as curves K_I/K_0 vs. a/b in **Figure 5.13** for several values of β. K_0 is the stress intensity factor for an isolated crack in a sheet under a uniform uniaxial tensile stress σ and is given by

$$K_0 = \sigma\sqrt{\pi a} \qquad \textbf{Equation 5.12}$$

To develop this analysis I have created several sheets of different sizes. To analyse this crack I have to assume some boundary conditions to create the same conditions as the published data. The boundary conditions that I have assumed are:

- The top corners have the restriction of no movement along y direction
- The middle point of the top edge has the restriction of no movement along x direction.

With the first boundary conditions I let the sheet to open freely but I do not let the tops corners to move along the y direction. It is needed because the sheet does not have to rotate. And the second boundary condition is needed to do not let the sheet to move left or right.
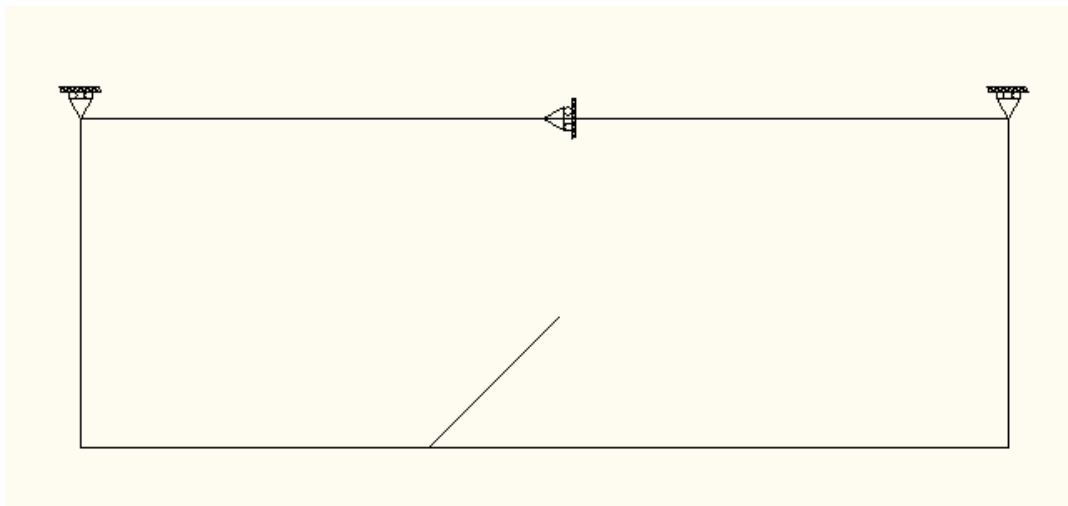
Figure 5.12

The conditions that I have assumed that are the same for all the performed tests are:

- Uniform tensile stress σ of 200 MPa perpendicular to the crack length
- Common engineering material with Young's Modulus E=200 GPa and Poisson's ratio υ=0.3.

Figure 5.13

- **200 x 80 mm sheet with a 40 mm crack with an inclination of 45˚.**

For this kind of sheet and crack, the published data shows that Y=1.2 ± 0.025 and following the equation for the stress intensity factor,

$$K_I = Y\sigma\sqrt{\pi a} \qquad \textbf{Equation 5.13}$$

Where Y=1.2, σ=200 N/mm^2 and a=40mm is the crack length. So K_I takes the value of,

$$\textbf{K}_\textbf{I} = \textbf{2690.40} \pm \textbf{56.05 N/mm}^{\textbf{3/2}}$$

Modelling now the crack with Abaqus, I have used a circle around the crack with 20mm of ratio where I want to create the degenerated mesh.
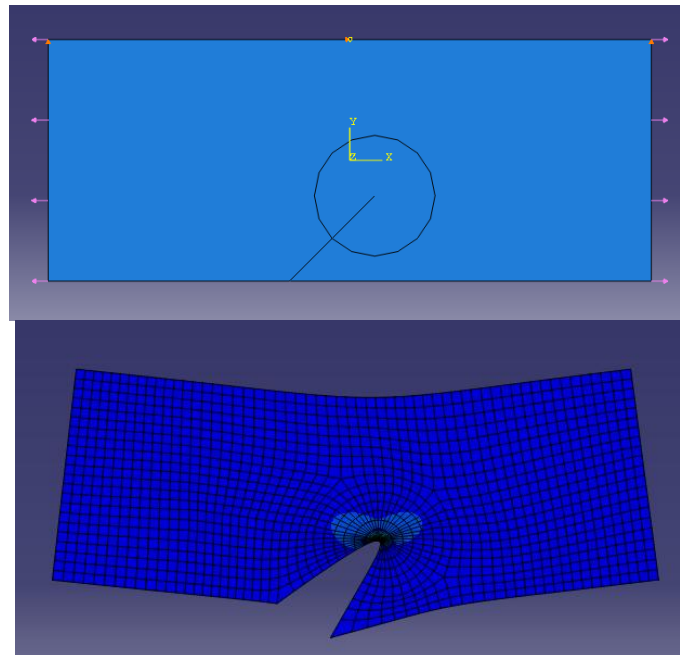
Figure 5.14

Figure5.15

As the figure shows, the values of the J-integral are 4.4903e+04, 4.4851e+04 and 4.4872e+04 N/m. Following the equation showed in the literature,

$$K_I = \sqrt{E \left[\frac{N}{mm^2}\right] x J \left[\frac{N}{mm}\right]} \qquad \textbf{Equation 5.14}$$

I have taken the last value because it belongs to the third integral and takes more elements to do the approximation. So the result that I have obtained is,

$$K_I = 2995.73 \ N/mm^{3/2}$$

The error that I obtain with this modelling is around 9%, which is a not to big error but larger that one I expected. In the next analysis I will put more material around the crack to see what change.

- **250 x 100 mm sheet with a 40 mm crack with an inclination of 45˚.**

For this kind of sheet and crack, the published data shows that Y=1.02 ± 0.025 and following the equation for the stress intensity factor,

$$K_I = Y\sigma\sqrt{\pi a}$$ **Equation 5.15**

Where Y=1.02, σ=200 N/mm^2 and a=40mm is the crack length. So K_I takes the value of,

$$\mathbf{K_I = 2286.84 \pm 56.05 \ N/mm^{3/2}}$$

Modelling now the crack with Abaqus, I have used a circle around the crack with 20mm of ratio where I want to create the degenerated mesh.
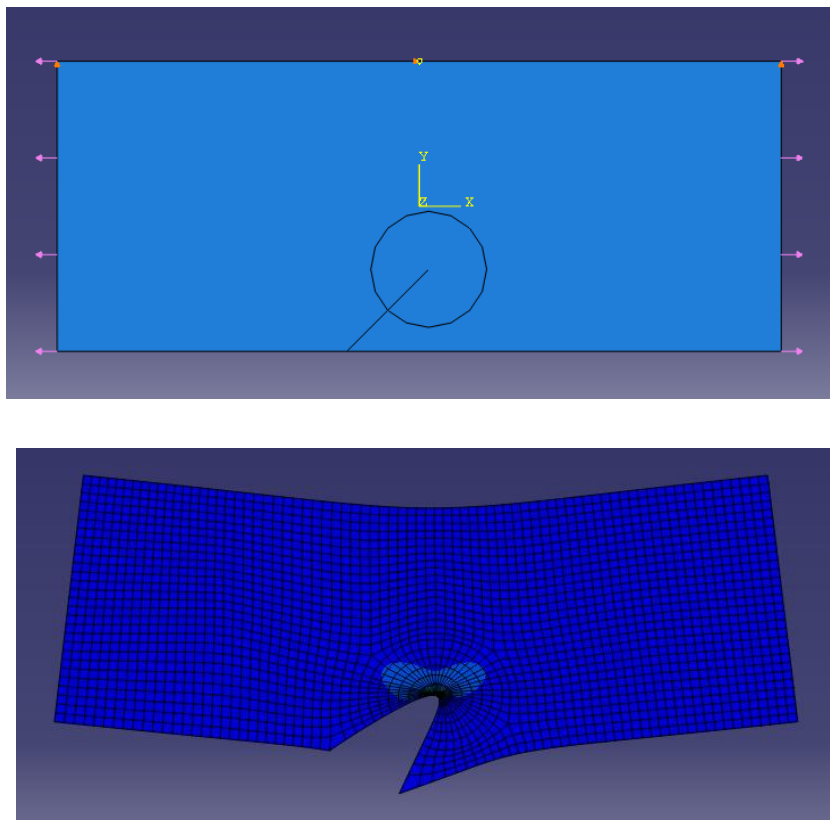
Figure 5.16

Figure 5.17

As we see in the picture, the values of the J-integral are 3.2457e+04, 3.2418e+04 and 3.2434e+04 N/m. Following the equation showed in the literature,

$$K_I = \sqrt{E\left[\frac{N}{mm^2}\right] x J\left[\frac{N}{mm}\right]} \qquad \textbf{Equation 5.16}$$

I have taken the last value because it belongs to the third integral and takes more elements to do the approximation. So the result that I have obtained is,

$$K_I = 2546.92 \text{ N/mm}^{3/2}$$

The error that I obtain with this modelling is around 9% again. It still being large but at least it is as big as the modelling before, that it is a good sign. Because the error is always the same, the problem may lie in the published data or in the Abaqus iterations.
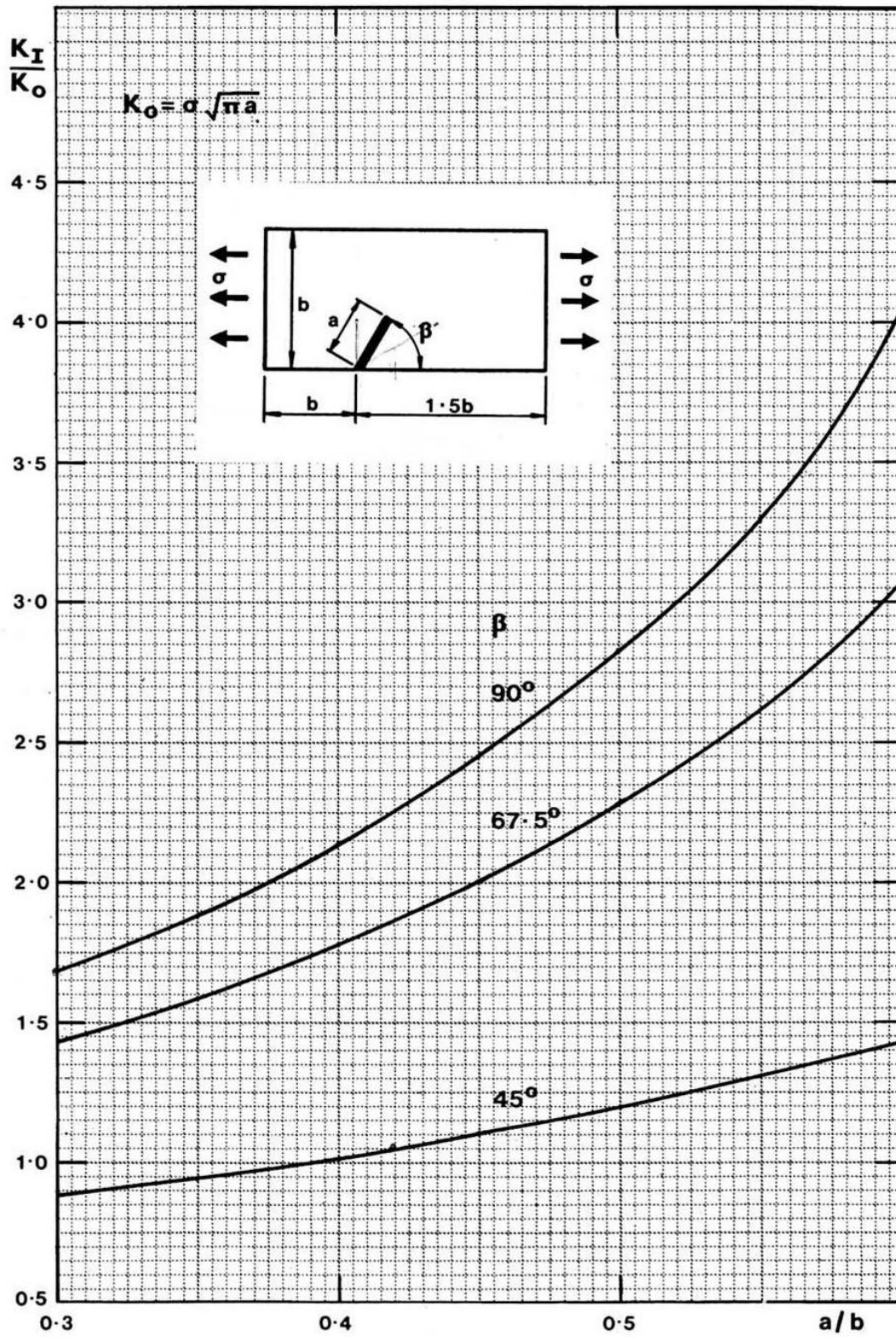
- **200 x 80 mm sheet with a 40 mm crack with an inclination of 67.5˚.**

For this kind of sheet and crack, the published data shows that Y=2.27 ± 0.025 and following the equation for the stress intensity factor,

$$K_I = Y\sigma\sqrt{\pi a}\qquad\textbf{Equation 5.17}$$

Where Y=2.27, σ=200 N/mm^2 and a=40mm is the crack length. So K_I takes the value of,

$$\textbf{K}_\textbf{I} = \textbf{5089.33} \pm \textbf{56.05 N/mm}^{\textbf{3/2}}$$

Modelling now the crack with Abaqus, I have used a circle around the crack with 20mm of ratio where I want to create the degenerated mesh.
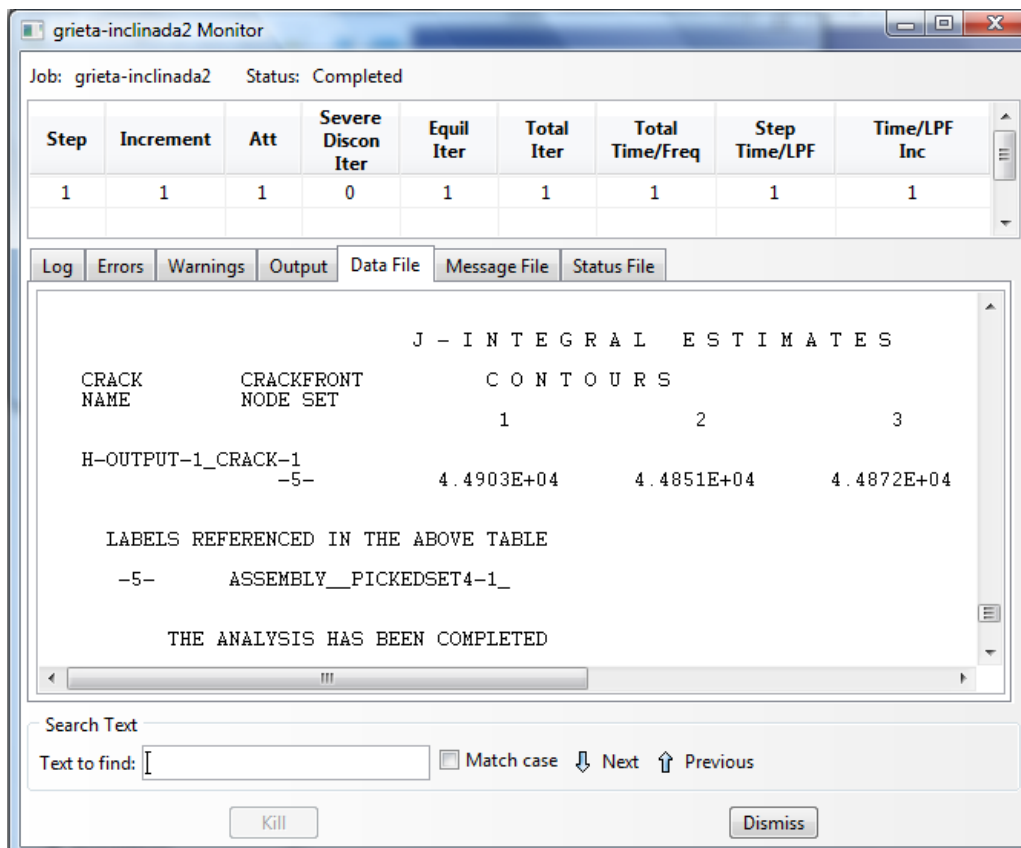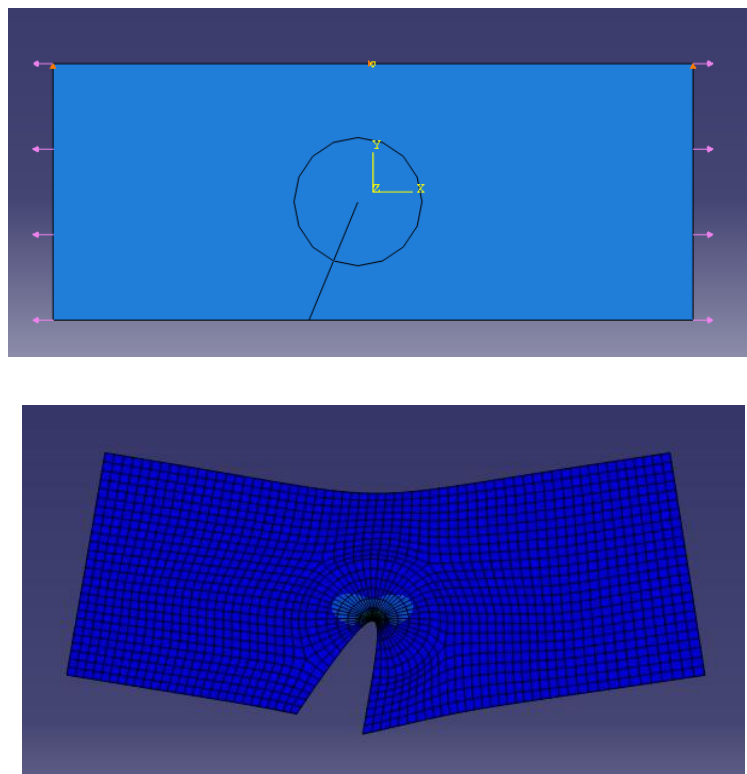

Figure 5.18
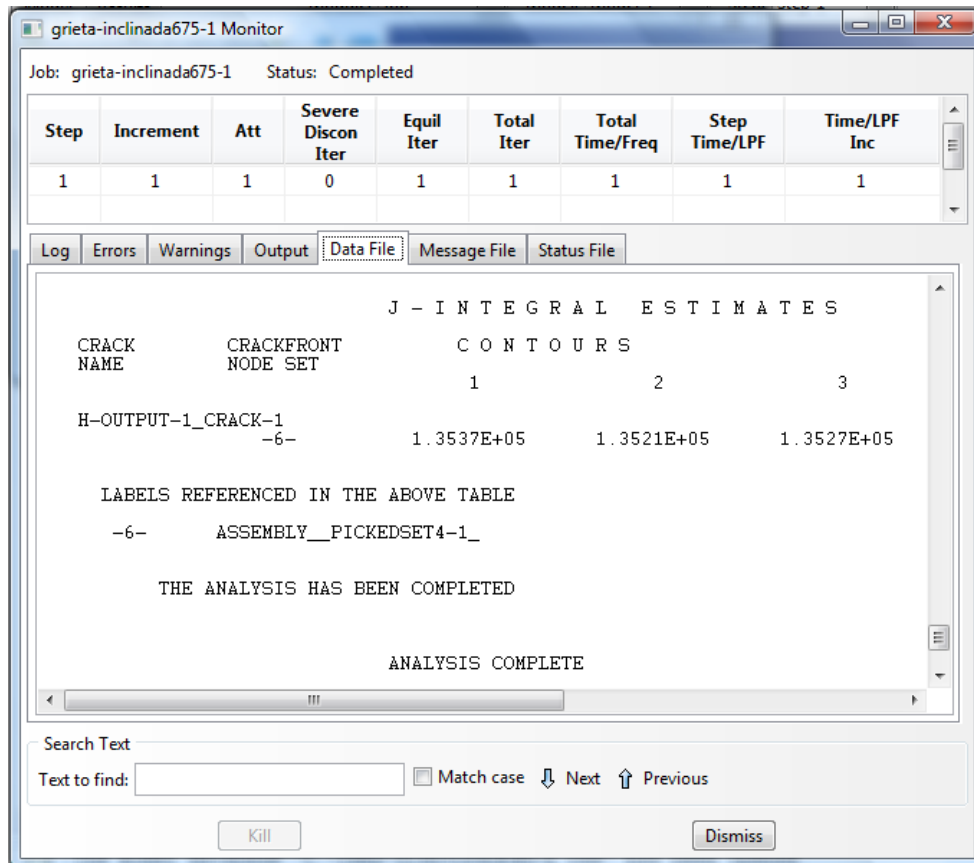
Figure 5.19

As we see in the picture, the values of the J-integral are 1.3537e+05, 1.3521e+05 and 1.3527e+05 N/m. Following the equation showed in the literature,

$$K_I = \sqrt{E\left[\frac{N}{mm^2}\right] x J\left[\frac{N}{mm}\right]} \qquad \textbf{Equation 5.18}$$

I have taken the last value because it belongs to the third integral and takes more elements to do the approximation. So the result that I have obtained is,

$$\mathbf{K_I = 5201.35 \ N/mm^{3/2}}$$

In this case, the error that I have obtained is near 2%, which is a great value. As in the previous analysis, I will introduce more material around the crack, and observe what happen.

- **250 x 100 mm sheet with a 40 mm crack with an inclination of 67.5˚.**

For this kind of sheet and crack, the published data shows that Y=1.775 ± 0.025 and following the equation for the stress intensity factor,

$$K_I = Y\sigma\sqrt{\pi a} \qquad \textbf{Equation 5.19}$$

Where Y=1.775, σ=200 N/mm^2 and a=40mm is the crack length. So K_I takes the value of,

$$\textbf{K}_I = \textbf{3979.54} \pm \textbf{56.05 N/mm}^{3/2}$$

Modelling now the crack with Abaqus, I have used a circle around the crack with 20mm of ratio where I want to create the degenerated mesh.
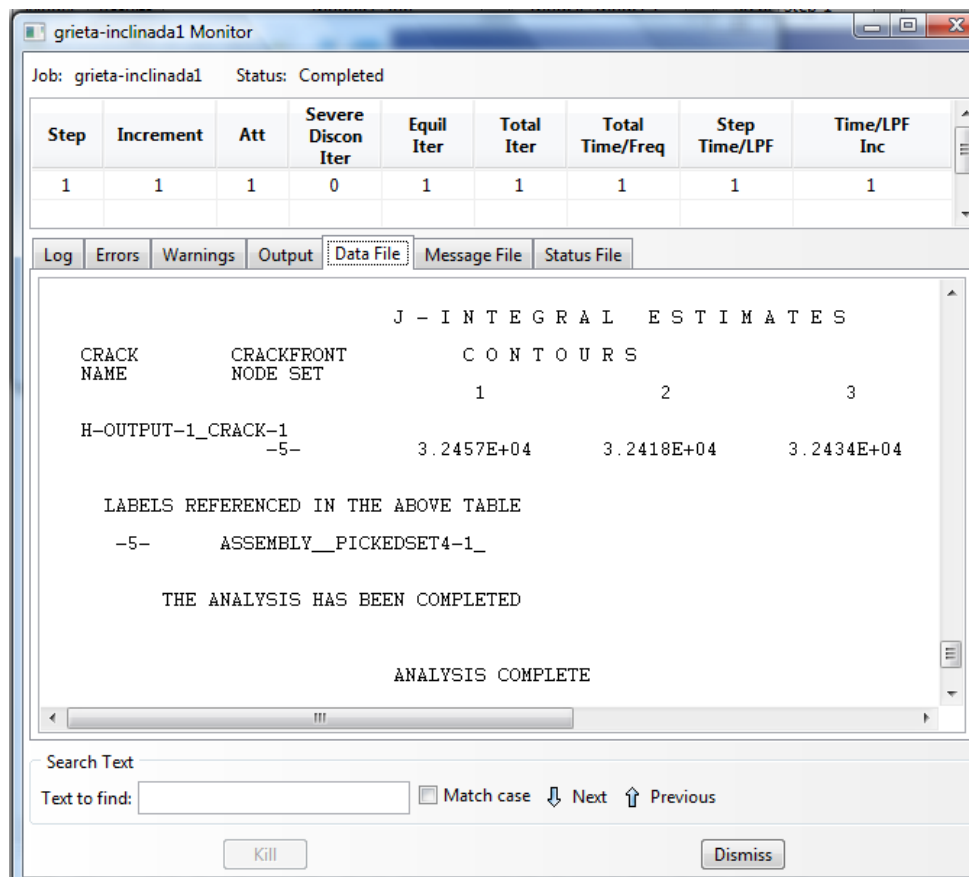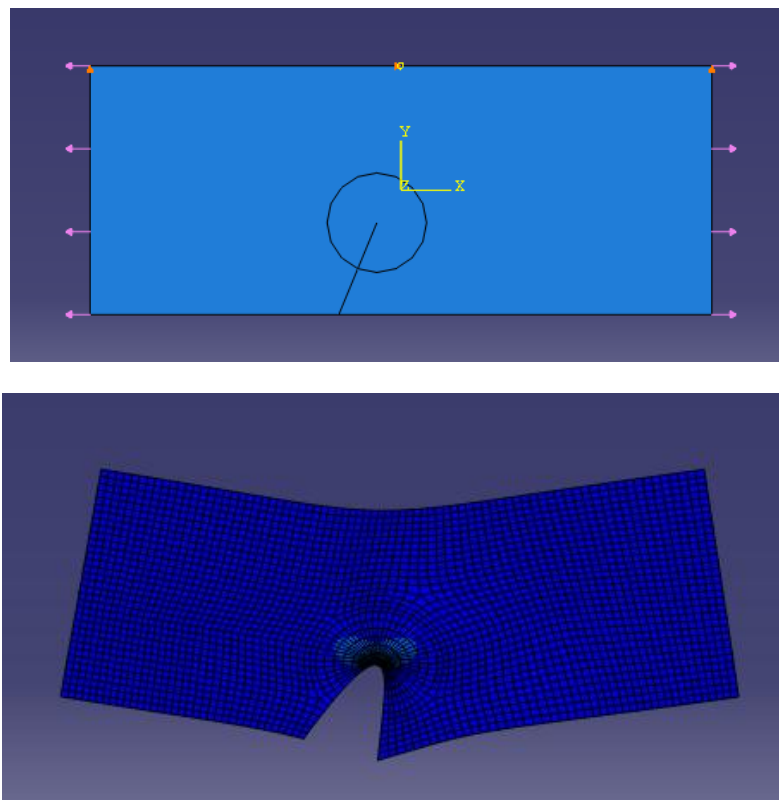

Figure 5.20
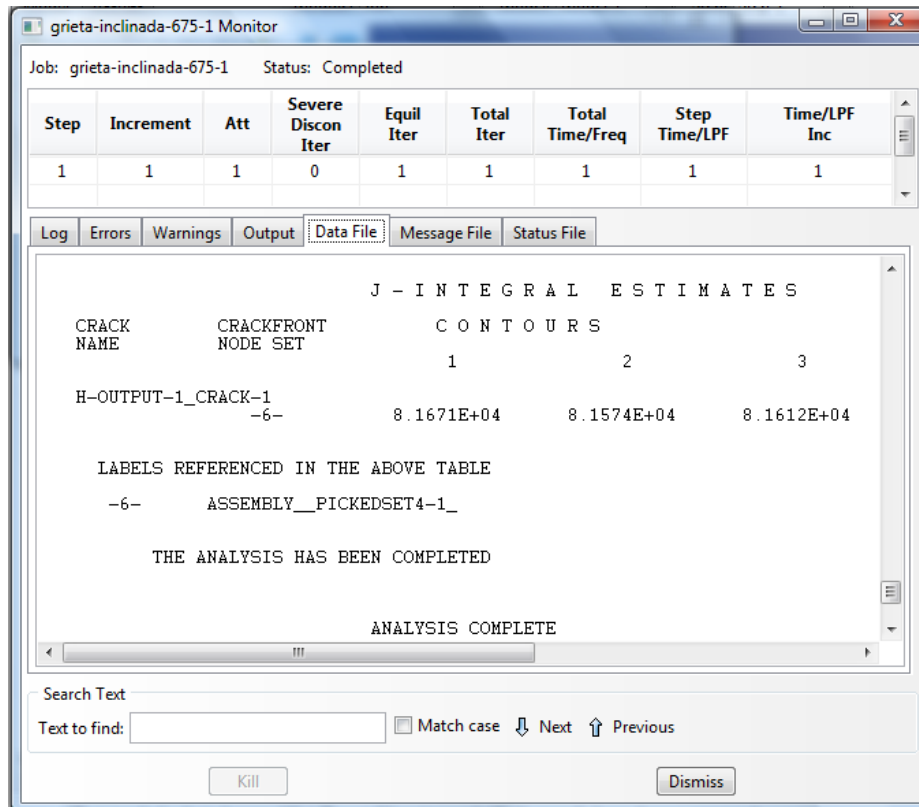
Figure 5.21

As we see in the picture, the values of the J-integral are 8.1671e+04, 8.1574e+04 and 8.1612e+04 N/m. Following the equation showed in the literature,

$$K_I = \sqrt{E\left[\frac{N}{mm^2}\right] x J\left[\frac{N}{mm}\right]} \qquad \textbf{Equation 5.20}$$

I have taken the last value because it belongs to the third integral and takes more elements to do the approximation. So the result that I have obtained is,

$$\textbf{K}_\textbf{I} = \textbf{4040.10 N/mm}^{\textbf{3/2}}$$

In this case, the error that I have obtained is near 2% again, which is also a great value and happens the same as the two previous analysis, where the error remains constant. This result affirms the thing supposed previously. The error may be due to the fact that the published data are too little recent or that Abaqus iteration method for this kind or cracks is not so accurate.

The last two analyses have more accurate results. That could be due to the fact that the conditions of this two analysis are nearer to the conditions of the first analysis (Edge crack in a finite width sheet: uniaxial tensile stress with tensile stress without bending constrains), in which I have obtained great results.

- **ACCURACY DUE TO THE KIND OF MESH**

The following analysis that I have done is to determinate the appropriate mesh that should be used in the analysis of cracks.

I have taken into account two parameters that can affect the accuracy of the results.

- The number of elements along the crack
- The number of elements around the circle

➢ **Accuracy due to the number of elements along the crack**

To develop this analysis I have chosen two of the tests made before. I have changed the number of elements along the crack in either case and studied the change on the accuracy that occurs with these changes.

1. Edge crack in a finite width sheet: uniaxial tensile stress with tensile stress without bending constrains

As we can see in the following graph, I have needed 10 elements to converge to the final solution. The differences between the values are rather small and the value obtained with 6 elements and the value obtained with 10 elements are more or less the same. The differences in the time needed to solve the problem are quite similar. So for this case, with 6 elements we obtain the accuracy needed.

Accuracy due to the number of elements along the crack line
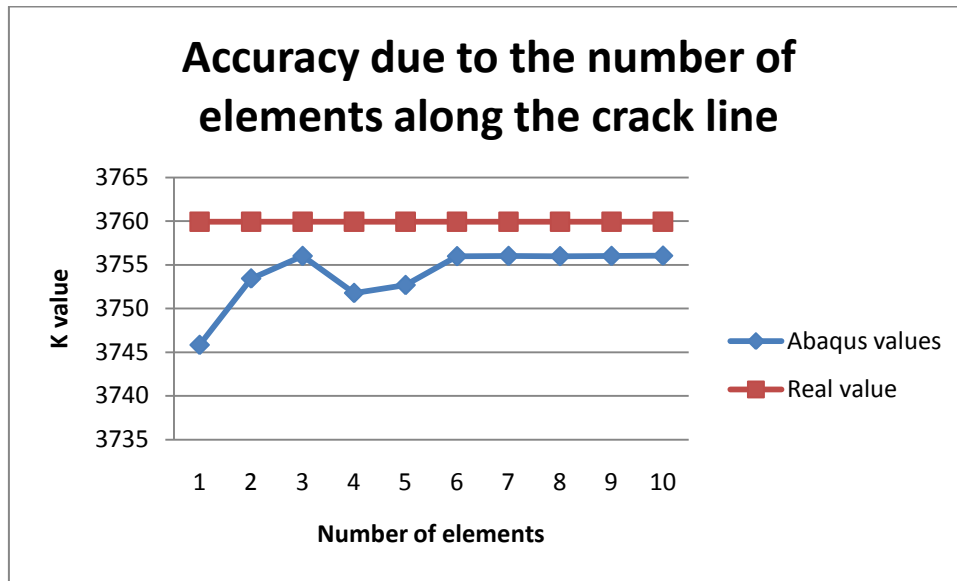
Graph 5.1

Number of elements	Abaqus K value [N/mm$^{3/2}$]	Real K value [N/mm$^{3/2}$]	Distance [%]
1	3745,82	3759,94	0,38
2	3753,43	3759,94	0,17
3	3756,01	3759,94	0,10
4	3751,77	3759,94	0,22
5	3752,68	3759,94	0,19
6	3755,98	3759,94	0,11
7	3756,01	3759,94	0,10
8	3755,98	3759,94	0,11
9	3756,01	3759,94	0,10
10	3756,04	3759,94	0,10

Table 5.1

2. Edge crack in a rectangular sheet: uniform normal displacement

In this case, the number of elements needed to converge to the finally solution are more than in the previous one. This could be due to the fact that the model is a little bit more complicated.

The data show that the differences between the values in each case are rather short and the time needed to solve the problems is also quite similar in every case. In this case, with 6 or 9 nodes we can obtain the accuracy needed.
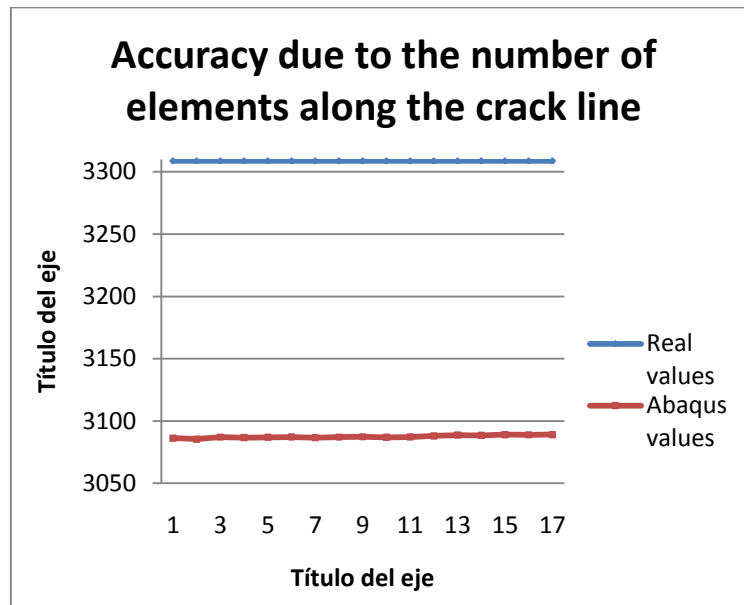
Graph 5.2

Number of elements	Abaqus K value [N/mm$^{3/2}$]	Real K value [N/mm$^{3/2}$]	Distance [%]
1	3086,29227	3308,75	5,72
2	3085,48213	3308,75	5,75
3	3087,13459	3308,75	5,7
4	3086,71346	3308,75	5,71
5	3086,94023	3308,75	5,7
6	3087,19938	3308,75	5,7
7	3086,64867	3308,75	5,71
8	3087,06981	3308,75	5,7
9	3087,39372	3308,75	5,69
10	3086,94023	3308,75	5,7
11	3087,19938	3308,75	5,7
12	3088,07383	3308,75	5,67
13	3088,75379	3308,75	5,65
14	3088,49478	3308,75	5,66
15	3089,07753	3308,75	5,64
16	3088,94804	3308,75	5,64
17	3089,07753	3308,75	5,64

Table 5.2

> ➢ **Accuracy due to the number of elements around the circle**
1. Edge crack in a finite width sheet: uniaxial tensile stress with tensile stress without bending constrains

This case is different because I started the analysis with 8 nodes because Abaqus cannot create the degenerated mesh around the tip with fewer elements. As the data show, I have needed 18 elements to converge to the final solution. However, the differences since the 14 elements and the time needed are very small so with 14 elements it is enough to obtain a good result.
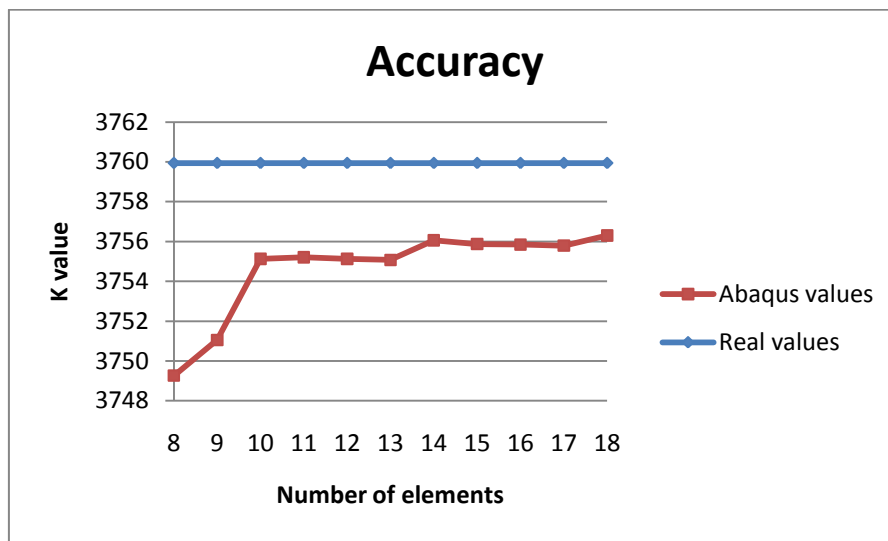
Graph 5.3

Number of elements	Real K value [N/mm$^{3/2}$]	Abaqus K value [N/mm$^{3/2}$]	Distance [%]
8	3759,94	3749,27	0,28
9	3759,94	3751,05	0,24
10	3759,94	3755,13	0,13
11	3759,94	3755,21	0,13
12	3759,94	3755,13	0,13
13	3759,94	3755,08	0,13
14	3759,94	3756,06	0,10
15	3759,94	3755,88	0,11
16	3759,94	3755,85	0,11
17	3759,94	3755,80	0,11
18	3759,94	3756,30	0,10

Table 5.3

2. Edge crack in a rectangular sheet: uniform normal displacement

As in the case before, I have started the analysis with 8 elements around the crack tip and I have needed 24 elements instead of 18 to converge to the solution. In this case, the differences between the values in all the cases are extremely short and the time needed to solve problem is approximately the shame. Therefore in this case, with 11 elements is enough.
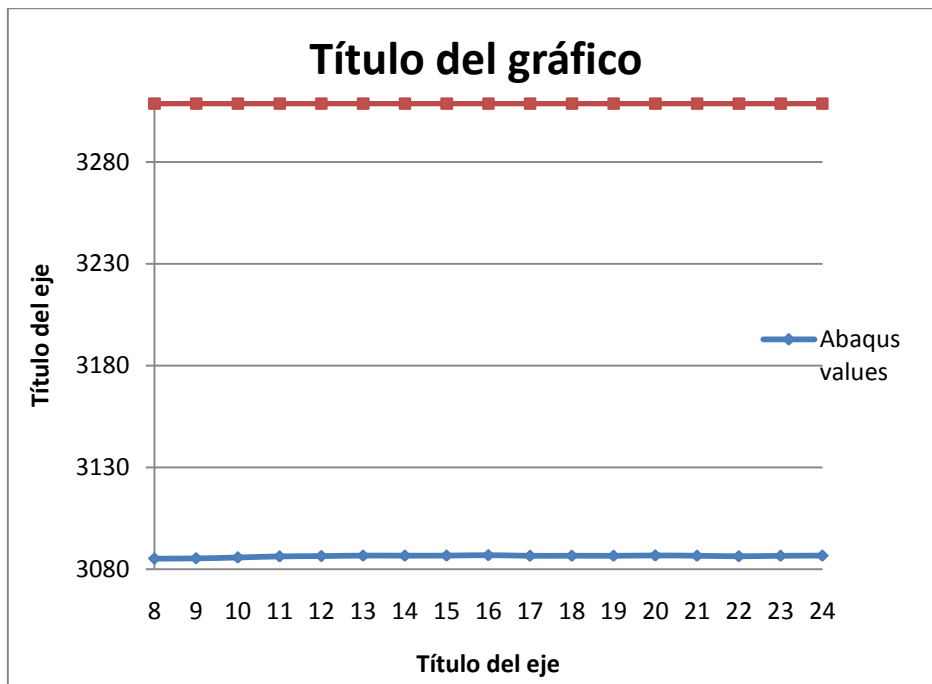
Graph 5.4

Number of elements	Real K value [N/mm$^{3/2}$]	Abaqus K value [N/mm$^{3/2}$]	Distance [%]
8	3308,75	3085,16	5,76
9	3308,75	3085,29	5,75
10	3308,75	3085,71	5,74
11	3308,75	3086,23	5,73
12	3308,75	3086,39	5,72
13	3308,75	3086,62	5,71
14	3308,75	3086,58	5,71
15	3308,75	3086,62	5,71
16	3308,75	3086,81	5,71
17	3308,75	3086,52	5,72

18	3308,75	3086,55	5,72
19	3308,75	3086,52	5,72
20	3308,75	3086,68	5,71
21	3308,75	3086,55	5,72
22	3308,75	3086,29	5,72
23	3308,75	3086,52	5,72
24	3308,75	3086,62	5,71

Table 5.4

In the analyzed cases, the K factor values do not change very much when I change the number of elements. This is due to the fact that the analyzed cases are quite simple. If we rely on the theory, if we increase the number of elements, the solution will be more accurate because it takes into account many more points. It is therefore advisable to increase the number of elements when you want more precision. However, it is necessary to bear in mind that the more nodes we put more resources need the program to solve the problem and therefore need more time to obtain the solution. As in all cases of engineering, this is a problem of making decisions depending on what more we are interested in every moment, if more precision or to finish earlier.

5.3. <u>3D CRACKS</u>

- **EDGE CRACK IN A FINITE WIDTH PLATE: UNIAXIAL TENSILE STRESS WITH TENSILE STRESS WITHOUT BENDING CONSTRAINS**

The first modelling of a 3D crack is a plate of width b, height 2h and thickness t containing a crack of length a in the middle of one side. It is subjected, at the ends, to a uniform tensile stress σ perpendicular to the direction of the crack.

The published data of this kind of crack are take from <u>Stress intensity factors</u> book and shows the results for K_I as a polynomial in a/b which is accurate to within 1% for all h/b≥ 1.0 and a/b ≤ 0.6; it is

$$\frac{K_I}{K_0} = 1.12 - 0.23\left(\frac{a}{b}\right) + 10.6(a/b)^2 - 21.7(a/b)^3 + 30.4(a/b)^4 \quad \textbf{Equation 5.21}$$

Where K_0 is the stress intensity factor for an isolated crack of length 2a in a sheet subjected to a uniform stress σ, and is given by

$$K_0 = \sigma\sqrt{\pi a} \qquad \textbf{Equation 5.22}$$

This equation of K_I/K_0 is only suitable for Plane Stress conditions where the edge effects not affect to the J value.

To develop this analysis I have create a plate with the same conditions of one of the 2D sheets that I have analyse before so I can compare both data and determinate where start the Plane stress conditions and where is Plane strain conditions.

The boundary conditions that I have assumed for my analysis are the same than in the 2D analysis but now in 3D.

- The right corners, top and base have the restriction of no movement in x direction
- The middle points of the edges of the right surface have the restriction of no movement in y direction.

With the first boundary conditions I let the plate to open freely but I do not let the right corners to move along the y direction. It is needed because the sheet does not have to rotate. And the second boundary condition is needed to do not let the sheet to move up or down.
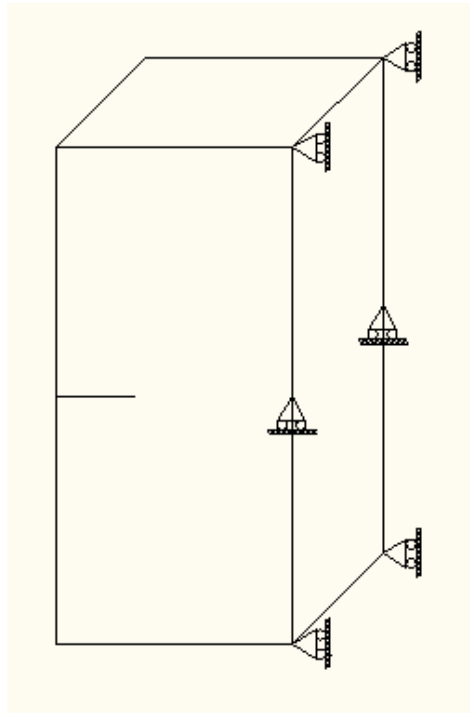
Figure 5.22

The conditions that I have assumed are:

- Uniform tensile stress σ of 200 MPa perpendicular to the crack length
- Common engineering material with Young's Modulus E=200 Gpa and Poisson's ratio υ=0.3.

- **200 x 400 x 100 mm plate with a crack of 50 mm**

In Plane stress conditions and for this kind of plate and crack, the published data shows that Y=1.5 ± 0.025 and following the equation for the stress intensity factor,

$$K_I = Y\sigma\sqrt{\pi a}$$ **Equation 5.23**

Where Y=1.5, σ=200 N/mm^2 and a=50mm is the crack length. So K_I takes the value of,

K_I = 3759.94 ± 62.67 N/mm$^{3/2}$

Modelling now the crack with Abaqus, I have used a circle around the crack with 25mm of ratio where I have created the degenerated mesh. Ç

In the first test that I have done to model this crack, I have used a uniform distributed mesh along the thickness. I have done this to analyse to have a initially approximation of how changes the intensity factor along the thickness and to can do later a better mesh. I have put fifteen elements along the thickness.
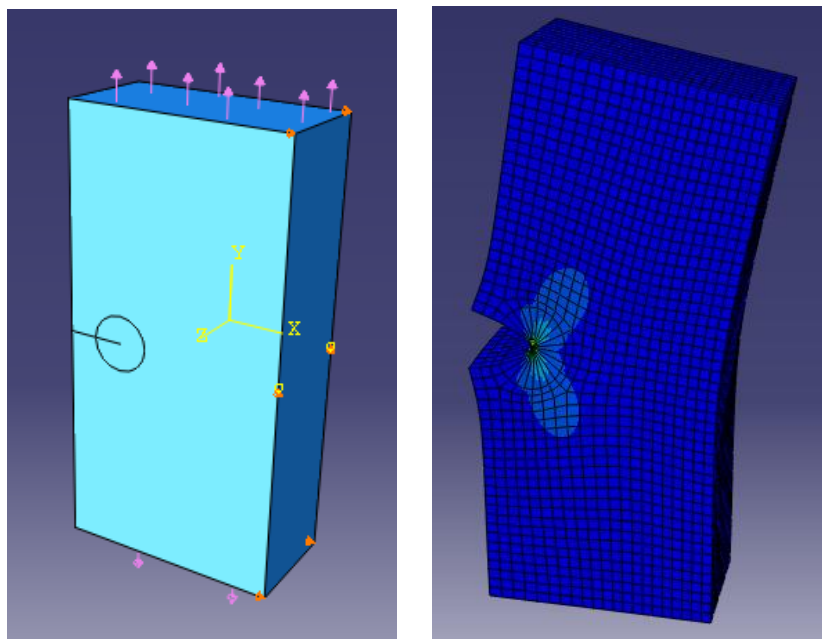
Figure 5.23

Thickness [mm]	Abaqus K value [N/mm$^{3/2}$]	2D K value [N/mm$^{3/2}$]	Distance [%]
0	3107,31	3756,04	17,27
3,33	3493,68	3756,04	6,99
6,66	3701,24	3756,04	1,46
9,99	3691,15	3756,04	1,73
13,32	3732,96	3756,04	0,61
16,65	3732,48	3756,04	0,63
19,98	3745,96	3756,04	0,27
23,31	3749,16	3756,04	0,18
26,64	3755,69	3756,04	0,01
29,97	3758,27	3756,04	-0,06
33,3	3762,53	3756,04	-0,17
36,63	3764,09	3756,04	-0,21
39,96	3766,86	3756,04	-0,29
43,29	3767,47	3756,04	-0,30
46,62	3768,95	3756,04	-0,34
49,95	3768,55	3756,04	-0,33
53,28	3768,95	3756,04	-0,34
56,61	3767,47	3756,04	-0,30
59,94	3766,86	3756,04	-0,29
63,27	3764,09	3756,04	-0,21
66,6	3762,53	3756,04	-0,17
69,93	3758,27	3756,04	-0,06
73,26	3755,69	3756,04	0,01
76,59	3749,16	3756,04	0,18
79,92	3745,96	3756,04	0,27
83,25	3732,48	3756,04	0,63
86,58	3732,96	3756,04	0,61
89,91	3691,15	3756,04	1,73
93,24	3701,24	3756,04	1,46
96,57	3493,68	3756,04	6,99
100	3107,31	3756,04	17,27

Table 5.5
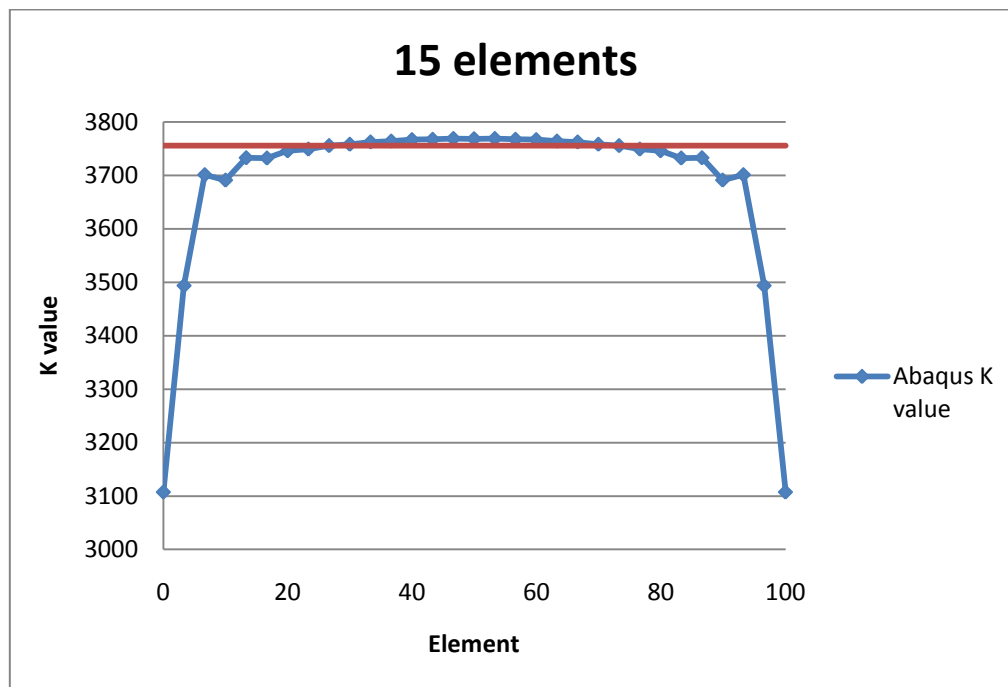
Graph 5.5

The graph shows the distribution of the stress intensity factor along the thickness. We can realise that there is a part in which the K value changes very fast. In the next test I will create a special mesh to focus on this part. In the middle of the plate, we can see that there is Plane strain conditions and that the values are very close to those obtained in 2D analysis, which is a great result.

In my second analysis I have used an especial kind of mesh to differentiate the parts in which there are Plane stress conditions from those in which there are Plane strain conditions. It is known that the plane stress conditions comes up to the 10% of the thickness, thus I have created a finer mesh in this zone and a rude mesh in the middle of the plate.
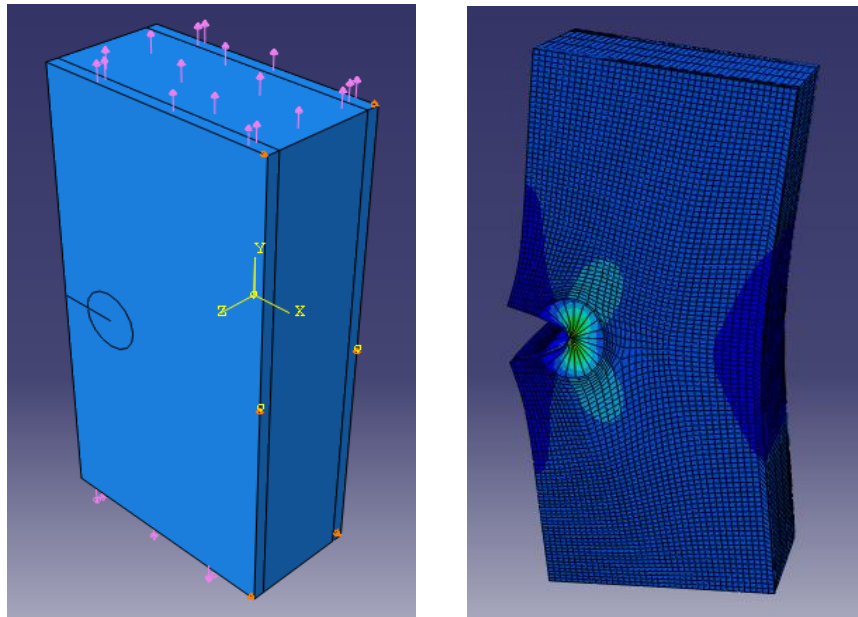
Figure 5.24

Thickness [mm]	Abaqus K value [N/mm$^{3/2}$]	2D K value [N/mm$^{3/2}$]	Distance [%]
0	2.907,13	3756,04	22,60
1	3.226,48	3756,04	14,10
2	3.426,84	3756,04	8,76
3	3.521,70	3756,04	6,24
4	3.577,65	3756,04	4,75
5	3.618,20	3756,04	3,67
6	3.650,07	3756,04	2,82
7	3.668,46	3756,04	2,33
8	3.680,41	3756,04	2,01
9	3.698,57	3756,04	1,53
10	3.710,88	3756,04	1,20
23	3.744,57	3756,04	0,31
37	3.772,43	3756,04	-0,44
50	3.766,67	3756,04	-0,28
63	3.772,43	3756,04	-0,44
77	3.744,57	3756,04	0,31
90	3.710,88	3756,04	1,20
91	3.698,57	3756,04	1,53
92	3.680,41	3756,04	2,01
93	3.668,46	3756,04	2,33
94	3.650,07	3756,04	2,82
95	3.618,20	3756,04	3,67
96	3.577,65	3756,04	4,75
97	3.521,70	3756,04	6,24

98	3.426,84	3756,04	8,76
99	3.226,48	3756,04	14,10
100	2.907,13	3756,04	22,60

Table 5.6
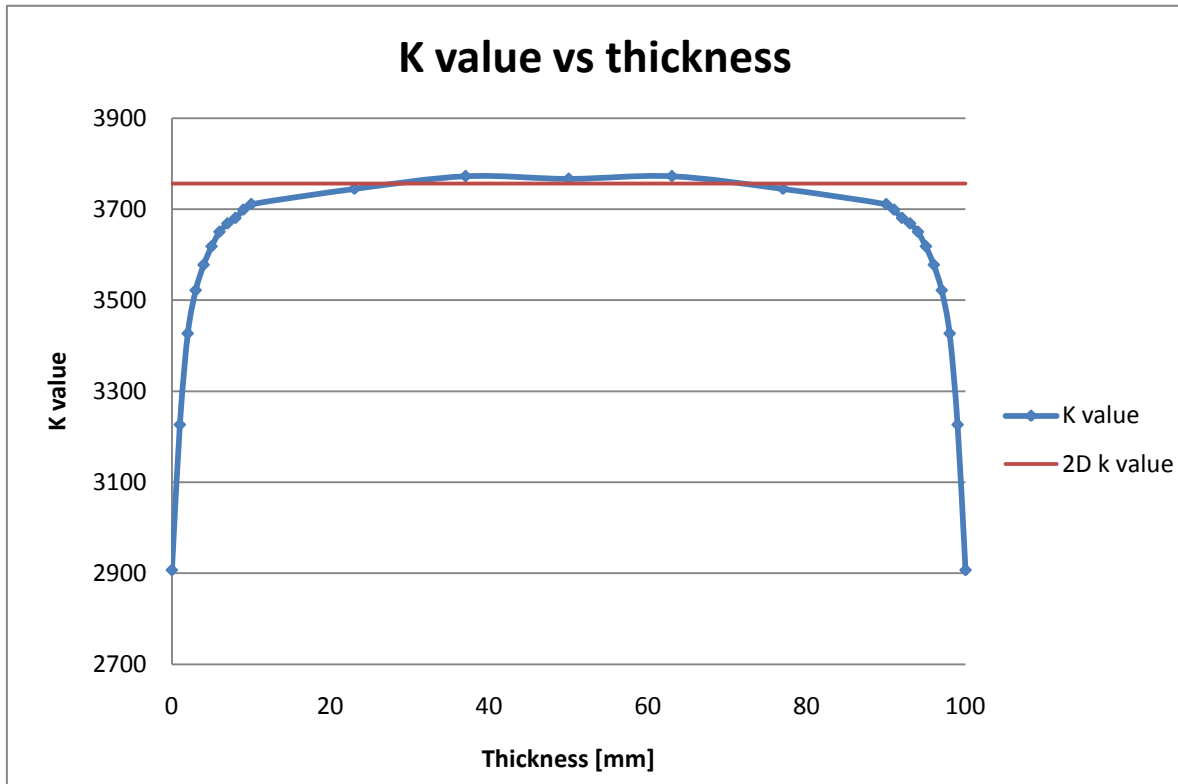
Graph 5.6

The graph shows the distribution of the Intensity factor along the thickness. As I have supposed, the part in which there is Plane Stress conditions is around 10 % of the thickness and in the rest of the plate there is Plane Strain conditions, where the K value is more or less constant.

If I compare the K value in 2D with the K value en 3D, we can see that the closer value appears in the middle of the plate, that it is something obvious. But if I take the first point, in which I have considered that the Plane strain conditions star, we can see that the distance is 1.2 %, so we can consider that the K value is constant in the zone with Plane Strain conditions and approximately the same value as the 2D model.

6. CONCLUSIONS

This project has been carried out with lot of software problems that have slowed the analysis and it is more difficult to reach a complete conclusion without a great number of analyses. However, these problems have help to deepen in this part of the study that is the development of cracks installing and cracks meshing in Abaqus.

The first part of the project was to develop the techniques to create cracks and mesh these cracks with Abaqus. It has been done perfectly, as prove of this are the tutorials both 2D and 3D. It was a really hard work because there is no tutorial about how to create a crack or how to create the deformed mesh, so they have to be developed.

The second part of the project was to develop techniques of analysis for different kind of cracks and different geometries. Several cracks have been tested and modelled with great accuracy.

Prove of it is that the first 2D crack modelled has an error smaller that 1%, that shows that approximation of the solution with the finite element method is really huge. The other analyses that have been done in 2D are also excellent.

In the second one, where the displacement of the sheet is restricted, and only can move in the perpendicular direction of the crack length, the result obtained are also fine. The error estimated is around 4%, which is not as great as the analysis before but I have proved that the error is always constant, so the problem could lie on the published data.

In the slant one, the two first analyses where the crack is very slant the error is lightly large, but it is constant as happen in the analysis before. As we are removing the crack inclination and we are coming up to the first analysis, the error is getting smaller and I have obtained errors smaller that 2%.

As a conclusion of this part, it must be said that the analysis of the accuracy doing after shows the ideal mesh to use in both case. Although these cases are simple geometries and the variations in the precision is not very large, in subsequent cases with more complicated geometries, the mesh type to use means that we obtain high accuracy but also can lead that the computer needs more resources and the time needed increase.

The last part of the project was to develop techniques of analysis for different kind of cracks in 3D models. As I have not enough time I could only do one analysis. However, the results obtained of this analysis are great and shows the way of continuing with my work.

ABAQUS software is widely used in engineering but the resources to create models and perform analysis are complicated to understand and it is very difficult to create them. But what makes ABAQUS an interesting toll is that, as a lot of software based on Finite Element Method, can give us very many information about analyses.

Although computational methods are very useful in fracture mechanics, they cannot replace experiments. A numerical fracture simulation of a cracked body can compute crack tip parameters, but such an analysis alone cannot predict when fracture will occur. Finite element analysis relies on continuum theory, and it does not contain voids, microcracks, second-phase particles, dislocation, or any of the other microscopic or submicroscopic features that control fracture behaviour in engineering materials.

As a conclusion, it is important to take into account that finite element analyses, as boundary element analyses, are just tools to help the engineer, because the engineer must know how to interpret the results given by these programs.

7. RECOMMENDATIONS

First of all, I would like to remark that this project has provided the basis and foundations of an exhaust 3D crack analysis. This project can be used to acquire the necessary acknowledge to perform a complete study.

A initial analysis of the crack behaviour in 2D and 3D is already done, but to improve the analysis some changes could be done, this includes new geometries or different kind of mesh could be apply, to make the mesh more accurate for complex geometries. A part from that, new processes should be run, this means that for example analyses with different loads or different geometries could have help predicting much better the behaviour of the cracks inside the plates.

8. REFERENCES

[1] NDT 200? *Fracture Toughness* (online), retrieved via Mozilla Firefox http://www.ndt-ed.org/EducationResources/CommunityCollege/Materials/Mechanical/FractureToughness.htm

[2] 2002 *Zona plastica* (online), retrieved via Internet Explorer http://www.edicionsupc.es/ftppublic/pdfmostra/EM03604M.pdf

[3] 200? *Steel Reinforcement Effectiveness to Combat Reflective Cracking in Rehabilitation Applications* (online), retrieved via Internet Explorer http://scholar.lib.vt.edu/theses/available/etd-03242003-163736/unrestricted/CHAPTER_5.pdf

Rooke, D.P. & Cartwright, D.J. *Stress intensity factors* (HSMO Procurement Executive, Ministry of Defence).

Anderson, T.L., *Fracture mechanics: Fundamentals and applications*, (Taylor & Francis Group).

Gudas & Joyce & Hackett, *Fracture Mechanic,* (ASTM)

Underwood, J.H. & Macdonald, B.D. & Mitchell, M.R., *Fatigue and Fracture mechanics,* (ASTM)

Belytschko, T. 2005, *International Journal for Numerical Methods in Engineering,* (John Wiley & Sons).

Aliabadi, M.H. & Rooke, D.P. 1991 *Numerical Fracture Mechanics*, (Computational Mechanics Publications, Southampton and Kluwer Academic Publishers).

Zienkiewicz, O.C. & Taylor, R.L. & Zhu, J.Z. 2005 *The finite element method: Its basis & fundamentals,* (ELSEVIER).

Callister, W.D. 1996, *Introducción a la ciencia e ingeniería de los materiales,* (U.S.A., John Wiley & Sons).

Knight, C.E., 1993 *The Finite Element Method in Mechanical Design,* (Boston, PWS-KENT Publishing Company)

Corten, H.T. 1972 *Fracture Toughness,* (Philadelphia, ASTM)

Alcalá, J. & Llanes, L.M. 2002 *Fractura de materiales,* (Barcelona, Ediciones UPC)

Kim, J.K. 2000 *Virginia Tech. Materials Science and Engineering* (online), retrieved via Mozilla Firefox
http://www.sv.vt.edu/classes/MSE2094_NoteBook/97ClassProj/anal/kim/intensity.html

De Mecánica 2002 *Breve reseña del MEF (Método de los elementos finitos)* (online), retrieved via Mozilla Firefox http://www.demecanica.com/TeoriaEst/TeoriaEst.htm

Vega, M. 2005 *Técnicas de predicción del crecimiento de grietas y tolerancia al daño* (online), retrieved via Internet Explorer
http://www.dim.udec.cl/aero/fondef/fractomecanica.pdf

Sukumar, N. & Möes, N. & Moran, B. & Belytschko, T. 2000 *Extended finite element method for three-dimensional crack modelling* (online), retrieved via Internet Explorer
http://www.tam.northwestern.edu/X-FEM/papers/3d.pdf

2002 *Zona plastica* (online), retrieved via Internet Explorer
http://www.edicionsupc.es/ftppublic/pdfmostra/EM03604M.pdf

NDT 200? *Fracture Toughness* (online), retrieved via Mozilla Firefox http://www.ndt-ed.org/EducationResources/CommunityCollege/Materials/Mechanical/FractureToughness.htm

Buscaglia, G. & Dari, E. & Zamonsky, O. 2004 *Aplicación de la integral J de dominio al análisis tridimensional de grietas en sólidos termoelásticos* (online), retrieved via Internet Explorer http://www.cab.cnea.gov.ar/enief/dirjobs/S23/RBalderrama_final.pdf