NTNU
Norwegian University of
Science and Technology

Finite Element Analysis of Fretting Fatigue

Gaute Sundstrøm Slettestøl

Mechanical Engineering
Submission date: June 2018
Supervisor: Bjørn Haugen, MTP

Norwegian University of Science and Technology
Department of Mechanical and Industrial Engineering

Preface

This master's thesis was written at NTNU at the Department of Mechanical and Industrial Engineering. The thesis was written in the spring of 2018, with a workload of 30 credits.

I want to thank my supervisor Bjørn Haugen and co-supervisor Steffen Loen Sunde for support and guidance throughout the thesis.

Trondheim 11.06.2018

Gaute Sundstrøm Slettestøl

Gaute Sundstrøm Slettestøl

Abstract

In this thesis it is conducted a literature study which looks into the subcategory of fatigue, fretting fatigue. Based on the literature study, different analyses are performed with different geometries, different contact formulations, plane strain and plain stress conditions, and finally post-processing the results by analysing them and try to use *pfat* as a analysis tool.

The first analysis was performed on a *Dog-bone* geometry, where the goal was to look for convergence in the results depending on the element size in the mesh. For this a *Python* script was made, in order to automate the analyses. After this was done, new analyses of the same geometry were performed in order to obtain the stick/slip behaviour with its belonging stress distribution.

Afterwords a new geometry were analysed, this was the main study of the thesis. The *Dovetail* is a frequently used fixing for turbine blades in the aerospace industry, and has some challenges regarding what is believed to be fretting fatigue. This geometry is different from the previous one studied, but the goal was to use the first study to compare with the results for the *Dovetail*. It was important to identify the stick/slip behaviour here as well, and the previous work was important in order to understand the *Dovetail*. The results obtained gave more understanding of the stick/slip behaviour, about where the slip amplitude has peak values and the influence of the coefficient of friction regarding the slip amplitude.

Sammendrag

I denne masteroppgaven ble det utført et litteraturstudie som ser på underkategorien av utmatting, fretting utmatting. Basert på litteraturstudiet, ble forskjellige analyser utført med flere forskjellige geometrier, kontakt formuleringer, plane strain og plane stress. Til slutt ble resultatene analysert og det ble gjort forsøk på å bruke *pfat* som analyse program.

Den første analysen ble utført på en *Dog-bone* geometri, hvor målet var å se etter konvergens i resultatene, basert på element størrelsen i meshet. Til dette ble et *Python* script laget, slik at analysene gikk automatisk. Etter at dette var utført, ble nye analyser med den samme geometrien utført, dette for å forstå stick/slip oppførselen med dets tilhørende spenningsdistribusjon.

Videre ble en ny geometri analysert, *Dovetail*, dette var hoveddelen av oppgaven. *Dovetail* geometrien er en hyppig brukt innfesting for turbinblader i flyfarts industrien, og har utfordringer med det som sannsynligvis er fretting utmatting. Denne geometrien er forskjellig fra den som tidligere har blitt studert, målet var Ãě bruke resultatene fra foregående studie, med resultatene observert i *Dovetail* studiet. Det var også hær viktig å identifisere stick/slip oppførselen, og det foregående arbeidet var viktig for å kunne forstå *Dovetail* geometrien. Resultatene som ble oppnådd, gav en bede forståelse av stick/slip oppførselen, om hvor slip amplituden har høye verdier og hva friksjon har å si for slip amplituden.

Table of Contents

List of Figures

1 Introduction

1.1 Problem Description

Fretting fatigue is a phenomenon that occurs in regions of contact between two surfaces, with small relative displacement during cyclic loading. The displacement is often of the order of tens of micrometers or less. This small relative displacement is difficult to reproduce and observe under laboratory conditions. How can the Finite Element Method be used to better understand fretting fatigue in constructions? This thesis aims to perform Finite Element Analyses on two geometries which are subjected to fretting fatigue, a test specimen (*Dog-bone*) and a *Dovetail* fixing.

1.2 Background

Fatigue is an important design criteria in many industries, mechanical engineering, civil engineering, marine engineering, aerospace engineering and many more. The topic fatigue has been researched for many years, but there are still many unanswered questions and many uncertainties regarding fatigue and life prediction of constructions. Fatigue is a wide concept with many subcategories, depending on what kind of fatigue a construction could undergo. Fretting fatigue is one of them, and this thesis will look closer into this area of fatigue.

In the aerospace industry, fretting fatigue has been a challenge for as long as the industry has existed. Even so this area of fatigue has only been studied for the last 60 years. And the topic is still young, with many questions unanswered. Fretting fatigue may occur when two components in a structure have surfaces in contact with each other, and that there are a small relative displacement between them, often only in order of tens of micrometers or less. When the relative displacement is this small, it is difficult to obtain in tests, therefore it could be difficult to obtain the influence this has on the fatigue life. The challenges related to fretting fatigue are significant, so further research on the topic is necessary in order to understand and make good estimations on life prediction on constructions exposed for this. A critical component which will be studied in this thesis is the *Dovetail* fixing of the turbine blades in aeroplane engines. These fixings are exposed to small vibrations, centrifugal force and other loads and displacements, hence small relative displacement with fretting fatigue as a possible result.

1.3 Aim

The goal of this thesis is to perform finite element analysis(FEA), on two different geometries. And use the results to analyse the stress distribution, the relative displacement and the slip amplitude, in order to understand the importance of these regarding fretting fatigue. To create valid and realistic results for the *Dovetail* without a hertzian contact, but with two parallel contact surfaces is a challenging task, since this is a geometry with little to compare with in the literature. Therefore this part is extra interesting to analyse. And to look at the correlation to the more documented hertzian contact in order to better understand the complex fretting fatigue in *Dovetail* fixings.

Further this thesis aims to look at the post-processing analysis tool *pfat*, to have some life prediction estimates for the structures, which could be used in further work with testing and analysis. It will be a bonus if such results could be useful, since the main goal of the thesis are those described above.

2 | Methodology

2.1 Literature study

The first part of the thesis was to gather information and documentation on previous fretting fatigue studies. During the literature study several articles were found, a few books and the *Abaqus users manual* were also used for documentation and study on how problems could and should be solved. For articles *ScienceDirect*, *Elsevier Science*, *Oria* and *Google Scholar* were used as search engines.

The literature study gave a good base for the further work, and it gave a basic understanding of the fretting fatigue topic. This was a very important part of the study because the knowledge about fretting fatigue was very limited in the beginning. Therefore literature study has been important not just in the initial phase, but continuously during the whole period of this thesis. There have been challenges throughout the whole thesis where articles, books and the *Abaqus users guide* have been important resources in order to understand the challenges related to fretting fatigue. And to perform finite element analysis in a good way, with reliable results which can be related to theory.

2.2 Finite Element Analysis

The finite element analysis (FEA) is the most important part of this thesis, the goal for the thesis is to create good results for two different geometries, *Dog-bone* and *Dovetail*, which will be described in detail later. At first the important thing to do was to learn how to set up contact formulations in *Abaqus* and how fretting fatigue studies are performed. Here, the literature study was important in order to use the correct contact formulations and simulation method. After getting more comfortable with the analysis, it was time to begin on the *Dog-bone* study, getting results and analyse these. For the *Dog-bone* analysis there were created a *Python* script. The *Python* script was implemented in *Abaqus*, generating mesh automatically around the contact surfaces and text files of the contact stresses in the contact area. By this method it is possible to obtain convergence in the results, and by that decide the approximate element size of the model. This was important regarding streamlining the analysis process and reduce the time spent on analysing.

When the element size was satisfactory, more results were collected and analysed, trying to see a correlation between the different parameters studied. This was the core of the thesis, looking at the results and try to draw conclusions based on the them. This is the challenging part, since there are few test results to compare with, and the fact that fatigue analysis is a complex area to understand, especially the area covering fretting fatigue.

The *Dovetail* study was based on the results from the previous study, there was no need of doing a new study on convergence of the results based on the element size. The size was the same as for the *Dog-bone*. Although the contact surfaces are different in the *Dovetail* than the *Dog-bone*, it is comparable and results from the first study can be used as a guideline for the *Dovetail* study. The same procedure was followed as before with generating results and analysing them. As noted earlier this is the most important part, but also the most difficult.

2.3 Post-processing and discussion

After the finite element analysis was made, post-processing and discussion were important steps in getting a good understanding of the results. The post-processing was about making plots for different parameters to study, analyse these and finally discuss around them. First the *Dog-bone* study was analysed, with graphs showing the stick/slip behaviour, shear traction along the contact surface, along with Von Mises stress and stress in x-direction, both along the contact surface. The most important part was to analyse and discuss around the stick/slip behaviour, with the slip amplitude as a core parameter. This will be described in the theory chapter. The same analysis steps were repeated for the *Dovetail* geometry.

3 | Theory

3.1 Fatigue

When a construction is subjected to cyclic loading, the possibilities of fatigue failure has to be considered. Fatigue is weakening of a material after a number of cyclic loading, and is an important lifetime parameter for constructions. When materials experience cyclic stresses over time, they can undergo fatigue damage which finally leads to failure. The number cycles before fracture depends on the material, surface roughness, the stress applied, environmental conditions such as corrosive environment, temperature etc. The easiest way to describe fatigue in materials is by the stress intensity factor K, which is the most important factor of the power law discovered by Paris and Erdogan. The power law describes the crack growth of an already initiated crack. They introduced the equations we now know as the Paris law [14];

$$\frac{da}{dN} = C\Delta K^m \tag{3.1}$$

where

- $\frac{da}{dN}$: crack growth per cycle
- C and m : material constants determined experimentally
- ΔK : variation in stress intensity factor

The equation describes the linear section in figure 3.1. Where the ΔK_{th} is the fatigue threshold value, this value indicates what stress intensity factor that gives no crack growth. From figure 3.1 one can see that when the ΔK value reaches a certain value, the crack grows rapidly and fracture occurs. This approach is highly analytic, meaning that either the stress intensity factor or the crack growth needs to be known. The stress intensity can be calculated by this equation;

$$K_I = Y\sigma\sqrt{\pi a} \tag{3.2}$$

Y is here a geometry factor, for an edge crack the value is 1,12. In this case, a is the crack length and not the same as the semi width in the stick/slip zone described later in this chapter.

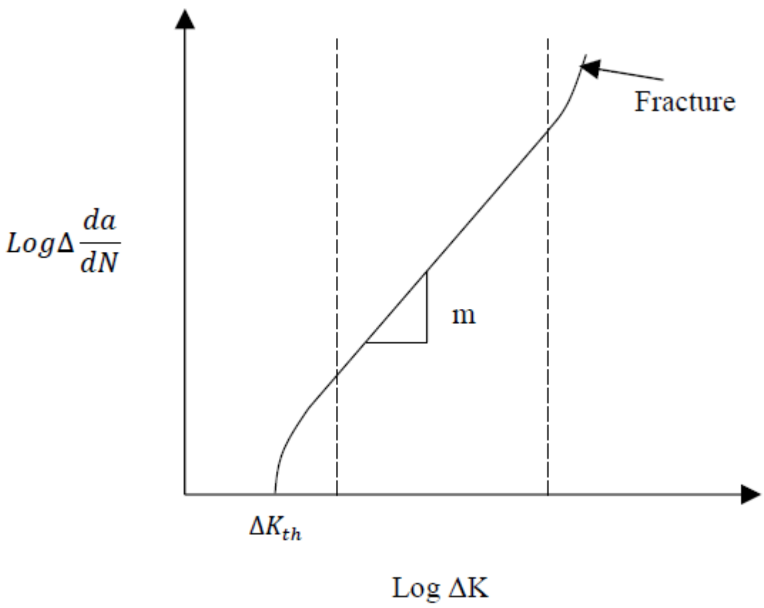

Figure 3.1: Fatigue crack growth behaviour

3.1.1 Notch fatigue

Geometric discontinuities such as holes, fillets and grooves are stress raisers often referred to as notches. And requires extra attention when designing geometries and performing life prediction on the component. Notches could, if not treated well, give devastating results for a geometry regarding fatigue life. This is due to the high increase in the local stress around the notch, the nominal applied stress could be greatly intensified when a notch is present. The tress concentration depends on the radius of the notch, the smaller the radius is the higher the stress concentration gets [6].

3.2 Fretting fatigue

Fretting fatigue is a phenomenon that occurs in regions of contact between two surfaces, with small relative displacement during cyclic loading. The displacement is often of the order of tens of micrometers or less. This small relative displacement is difficult to reproduce and observe under laboratory conditions [7][3].

The contact surface can be in full sliding across their surfaces during the cyclic loading. This is called *gross slip*, the damage caused by this is often identified as *fretting wear*. Another possibility is that the contact surfaces could be in *partial slip* loading condition with small slip zones near the edges of contact. The last condition that can occur is a combination of the two mentioned. This is *mixed-mode fretting* and may occur if the contact surfaces have initially a low friction coefficient resulting in gross slip [7]

3.2.1 Mechanics of fretting fatigue

Fretting has been a problem in mechanical engineering for at least a century, however, the mechanics of the problem and the influence of fretting on fatigue life, has been studied only for the last 60 years [11]. Fretting induced cracks usually grow normal to the free surface, it ultimately result in a brittle fracture of the component. The relative tangential displacement amplitude has a great influence on the fretting fatigue, this will be referred to as the slip amplitude. From a solid mechanics point of view it is not obvious why this has such a great influence, but it is shown in numbers of experiments that the surface is dependent on the slip amplitude. There are other variables as well that are of importance regarding fretting fatigue [8]:

- Variables controlling contact stress field
  - History of shearing force exerted and magnitude
  - Contact pressure distribution and intensity
- Bulk stress field and its variation
- Variables controlling asperity-scale stresses
  - Surface roughness
  - Amount of plasticity
  - Relative tangential displacement

Fretting loading also affects crack propagation, and it is the crack growth itself that gives failure. The surface degradation on the other hand is often quite mild. When the crack has grown, it could by described by normal linear elastic fracture mechanics. And the crack growth could relate to stress intensity factor and a power law, such as Paris equation described under section 3.1 [8][14]. Since the principal stress directions often are perpendicular or parallel to the free surface, the stress intensity factor will be mode I and mode II respectively[8]. The stress intensity factor is likely to be significant due to the localised stress concentration at the contact. The stress gradient is usually much higher than those induced by notches and holes. This is particularly important, when it most likely lead to a stronger size effect than in plain fatigue [5].

In fretting fatigue, the contact stresses, pressure and displacement depends on the loading history. The variation of displacement and stress during a load and displacement cycle changes the stick zone. The stick zone of a cylindrical hertzian contact formulation, c, can be calculated as follows [9];

$$c = a\sqrt{1 - \frac{Q_{max}}{fP}} \tag{3.3}$$

where

- c = stick zone
- a = semi contact width
- Q_{max} = maximum tangential load
- fP = Normal load multiplied with friction coefficient

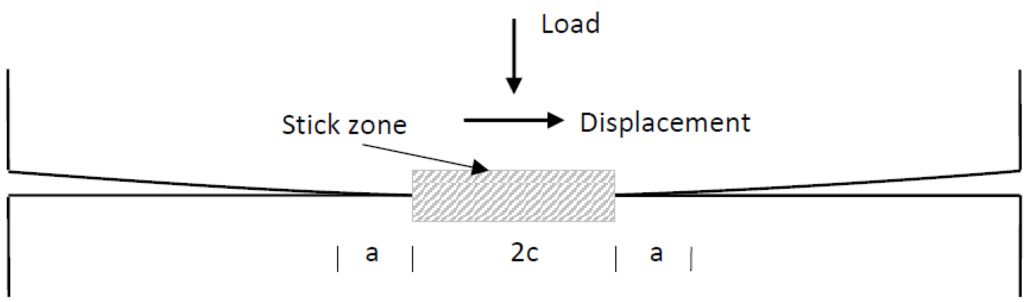

Figure 3.2: Illustration of the stick zone

This formulation could further be used in a new equation which describes the shear traction along the contact surfaces [10];

$$q(x) = -fP_0\sqrt{1-(\frac{x}{a})^2} \qquad ,c' <| x |\leq a \tag{3.4}$$

$$q(x) = -fp_0\sqrt{1-\frac{x}{a}^2} + 2(\frac{c'}{a})\sqrt{1-(\frac{x}{c'})^2} \qquad ,c \leq| x |\leq c' \tag{3.5}$$

$$q(x) = -fp_0\sqrt{1-(\frac{x}{a})^2} + 2(\frac{c'}{a})\sqrt{1-(\frac{x}{c'})^2} - (\frac{c'}{a})\sqrt{1-(\frac{x}{c})^2} \qquad ,| x |\leq c \tag{3.6}$$

As we can see, the shear traction varies with the position along the contact surface. X in these equations represent the position along the contact surface. The value of c' could be found by [10];

$$(\frac{c'}{a})^2 = 1 - \frac{1}{2fP}(Q_{max} - Q) \tag{3.7}$$

where

- c' = the new stick zone
- x = the distance from centre of the contact area
- q(x) = shear traction

During a load and displacement cycle the stick zone will change, therefore it is important to identify the new stick zone in order to obtain the correct result.

3.2.1.1 Stick/slip behaviour

The slip amplitude has shown to be a important part of fretting fatigue [5]. The fatigue life decreases rapidly as the slip amplitude increases up to a threshold value of approximately 50μm, and then increases again when the slip amplitude increases above this. For higher values of the slip amplitude, the wear rate increases so the embryo cracks gets closed before they can propagate. Figure 3.3 illustrates the behaviour of fretting fatigue life prediction depending on the slip amplitude[15]. As one could read from the figure, the threshold value is the worst case. It is also critical for values both above and below the threshold, a slip amplitude of approximately 10μm to 80μm will also give a rather low life prediction.

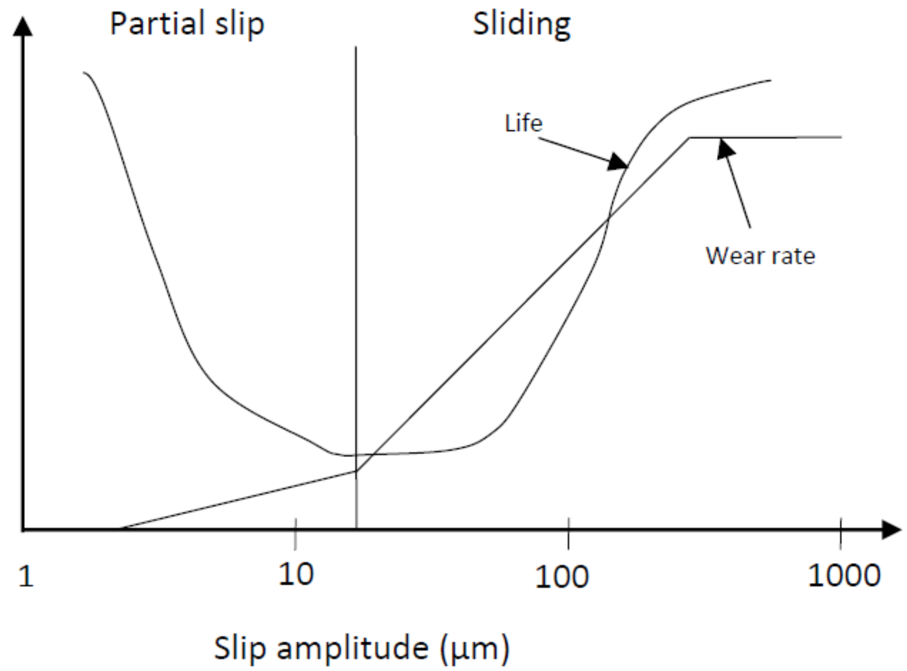

Figure 3.3: Graph of fretting fatigue life with slip amplitude

3.2.1.2 Friction

The friction force is a result of the normal load multiplied with the coefficient of friction. The coefficient increases as fretting occurs [4].

3.3 Finite Element Method

The finite element method (FEM) is a numerical method for solving mathematical physics and engineering problems. Depending on which solver or program one uses, the FEM can solve problems such as structural analysis, heat transfer, fluid flow, hydro dynamic problems and many more. In this thesis *Abaqus/CAE* is used as a FEM tool.

Abaqus is a powerful finite element analysis tool, with a wide range of opportunities, and is therefore frequently used in science and academia. In this thesis, *Abaqus/Standard* is used, which is suitable for static and low-speed problems[2]. In analysis of fretting fatigue it is important to use the correct properties and parameters to obtain good results, these will be described below.

3.3.1 Contact problems

When defining a contact problem in *Abaqus* it is important to use the correct form of contact. This application in *Abaqus* is called interactions, where there are several choices regarding the contact form. The standard is to use either "General contact" or "Surface to surface contact".

The "general contact" is not a good choice in problems like this. For practical reasons the contact pair is easy to use when the contact surfaces are known and as it is here, few contact pairs. The "surface to surface" contact is therefor the preferred choice. In this contact formulation there are two different discretizations; "surface-to-surface" and "node-to-surface".

With "node-to-surface" discretization the contact conditions are established so that the "slave nodes" on one part effectively interacts with a point on the "master surface" on the other component. The contact conditions are a single slave node and a few nearby "master nodes" which values are interpolated to the projection point [2].

- The "slave nodes" are constrained so they don´t penetrate into the "master surface"
- The contact direction is normal to the "master surface", this is illustrated in figure 3.4

With "surface-to-surface" discretization both "master and slave shapes" are considered as surfaces in the region of contact[2].

- The formulation enforces contact conditions in an average sense over regions nearby "slave nodes". The averaging regions are approximately centred on "slave nodes". Some penetration could be observed at individual nodes
- The contact direction is based on an average normal of the "slave surface" in the region surrounding a "slave node"

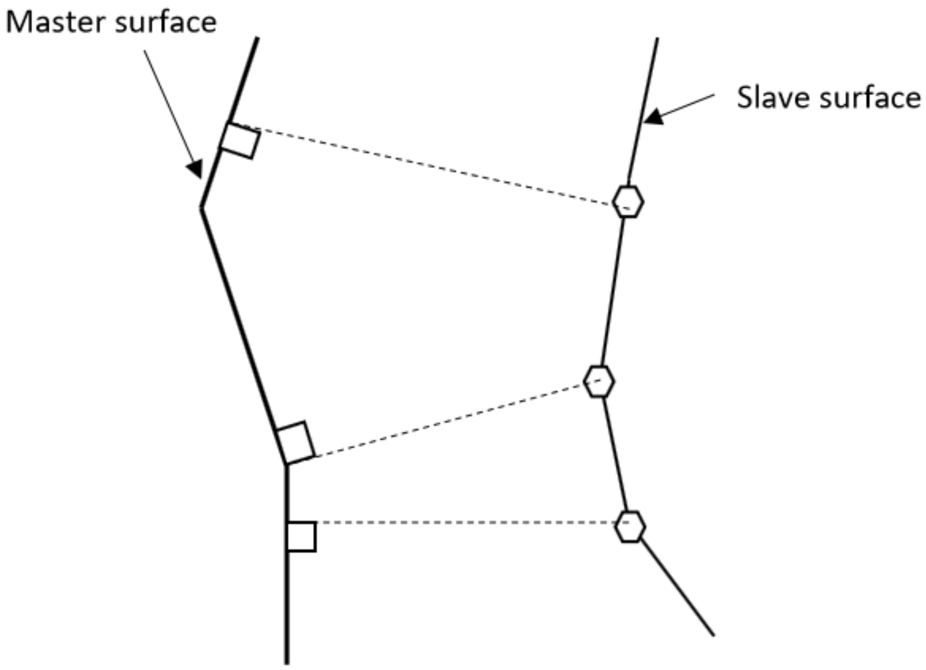

Figure 3.4: Node-to-surface connection

Generally "surface-to-surface" contact is more accurate regarding stress and pressure results than "node-to-surface". In "node-to-surface" the forces concentrates around the "slave nodes". This concentration leads to peaks in the stress distribution and will therefore give a overestimation [2].

After choosing one of the contact options, the contact properties have to be specified. The main different properties to define are whether it is "small" or "finite" sliding and what kind of tangential and normal behaviour the system has. Important parameters to define is the mechanical contact properties, they are as follows:

- Tangential behaviour
  - Friction formulation
  - Friction coefficient
- Normal behaviour
  - Pressure-overclosure
  - Constraint enforcement method

3.3.1.1 Tangential behaviour

When deciding what kind of friction formulation it is convenient to use, it is important to identify the problem properly. The standard choice to make is either "Penalty", "Frictionless" or "Lagrange multiplier". "Frictionless" is not the correct choice in this case, since friction is known to be high in fretting. It is possible to use either "penalty" or "Lagrange multiplier". The difference between the two methods is that the "Lagrange multiplier" is more accurate when there is a stick/slip problem to analyse.

By using the "Lagrange multiplier" implementation in *Abaqus/Standard* it is possible to enforce exactly the interface between two surfaces. This method increases the computational cost and by that also the analysis time. This is due to the added degrees of freedom and the increased number of iterations required to obtain convergence. It can in some cases prevent convergence if many points are iterating between stick and slip conditions. Even though the "Lagrange multiplier" gives this extra cost, it is recommended to use this method were the problem to analyse is stick/slip behaviour. Fretting between two bodies is an example of this.[2]

The choice of coefficient of friction depends on whether the component are coated or not, the surface roughness of the material and what materials are used. Even when these parameters are known, it is difficult to set the correct value. Most likely the coefficient would change during the cyclic loading due to the sticking of the surfaces. Therefore it is often used a coefficient as high as 0.9 [1], but also as low as 0.1 [13]. It is therefore difficult to make a clear decision whether the coefficient should be high or low. However it is reasonable to assume that a high coefficient would give the most correct result since the components will undergo stick/slip and therefore a high coefficient of friction in the stick zone.

3.3.1.2 Normal behaviour

The "pressure-overclosure" decides how the surfaces behave on each other, whether it allows penetration or not. There are two main formulations, hard contact and soft contact. Soft contact is divided into several different "formulations"; *explicit, implicit and linear*. They are used most frequently when the materials are soft or when simulating for example metal forming applications[2].

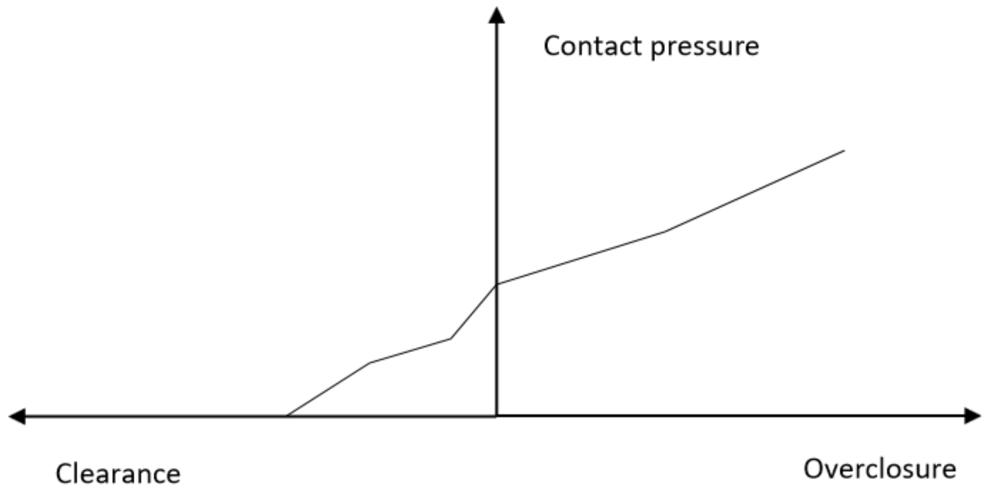

Figure 3.5: Soft contact formulation

Hard contact gives a behaviour where the surfaces are not allowed to penetrate each other, how strict the "no penetration" is, depends on the constrained enforced method used. Normally it allows a very small amount of penetration. When surfaces are in contact, any contact pressure can be transmitted between them. The surfaces separate if the contact pressure reduces to zero. Separated surfaces come into contact when the clearance between them reduces to zero [2].

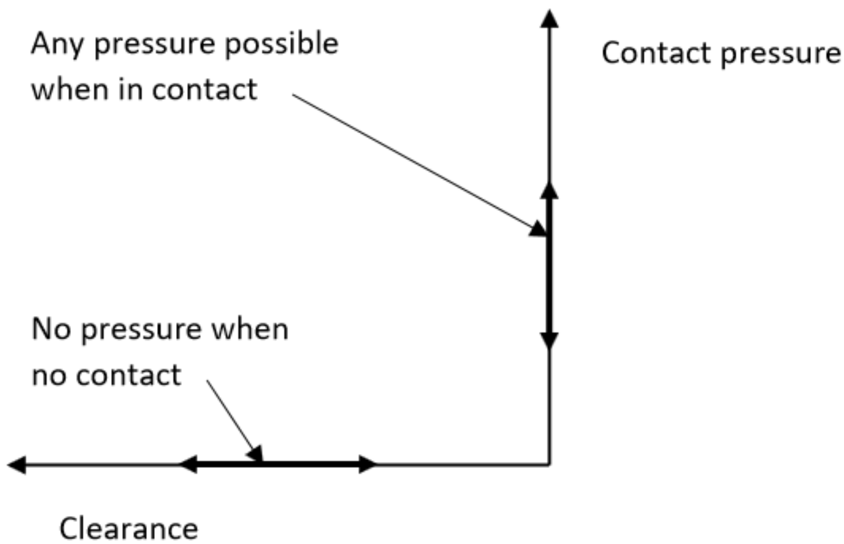

Figure 3.6: Hard contact formulation

The available constrained enforced methods for hard contact are as follows;

- Direct method
- Penalty method
- Augmented Lagrange method

The "direct method" gives a very strict "pressure-overclosure" behaviour. When combined with hard contact the "pressure-overclosure" is very strict. In order to obtain sufficient accuracy "Lagrange multiplier" is always used. An important limitation by using the direct method combined with hard contact is problem with overconstraint model, this is due to the strict overclosure between the contact pairs. The direct method is therefor best in combination with soft contact [2].

The "Penalty method" can be implemented such that no "Lagrange multiplier" needs to be used, this reduces the computational cost and thereby reduce the simulation time. This allows for some penetration to occur. There are two different types of penalties, linear and nonlinear. The linear penalty has a constant stiffness and therefor a linear behaviour between overclosure and pressure. The nonlinear penalty stiffness gives a nonlinear behaviour between the overclosure and pressure.

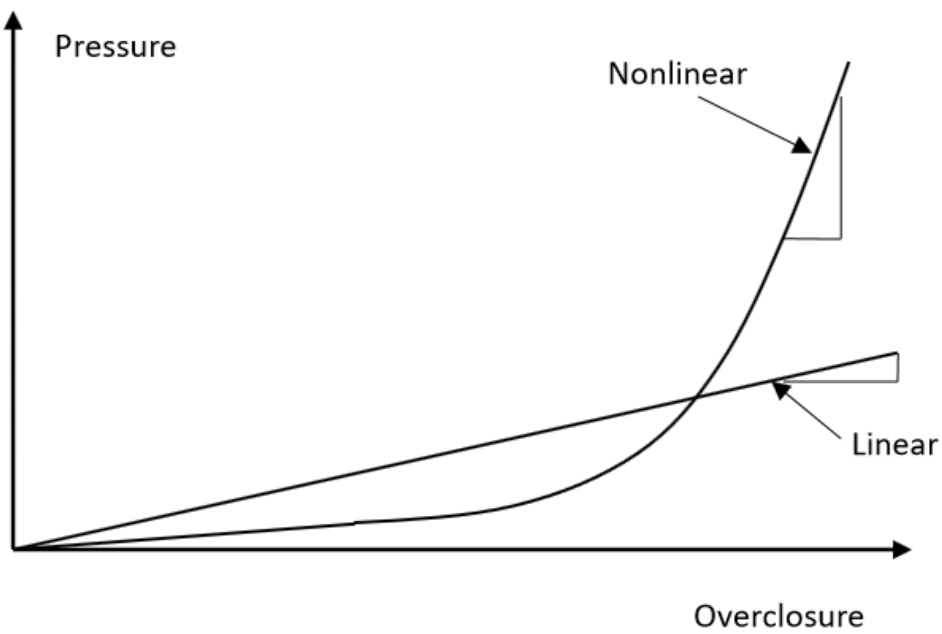

Figure 3.7: Linear vs Nonlinear stiffness

The "augmented Lagrange" method requires additional iterations in most cases and thereby additional solution time. But this approach gives a very accurate solution and it avoids problems regarding overconstraint. The "augmented Lagrange" method is actually an linear penalty method with an augmentation that reduces the penetration distance. Only with hard pressure overclosure relation this method could be used. There are three possible ways of finding the solution with this method;

- *Abaqus* uses the penalty method to find a converged solution
- If a "slave node or surface" penetrates the master surface with more than the specified, the contact pressure is augmented and more iterations is executed in order to obtain convergence
- *Abaqus* continues to augment the contact pressure until a converged solution occur

When the default settings are used the "augmented Lagrange" method do not use "Lagrange multipliers", this is due to the penalty stiffness, which for the "augmented Lagrange" method is 1000 times the underlying element stiffness. And the "Lagrange multiplier" is used only when the stiffness exceeds 1000 times the underlying stiffness.

3.4 Mesh

In *Abaqus* there are several different types of element formulations available, "triangular" or "quad", "linear" or "nonlinear", "plane strain" or "plane stress". The main differences are the accuracy and the behaviour of the elements during a simulation. The first thing to decide is whether the elements should be "triangular" or "quad" elements. In general the "quad" elements has better convergence rate than the "triangular". For complex geometries the "triangular" elements are better if the element get distorted [2]. The "triangular" mesh of first order are usually overly stiff, therefore a very fine mesh is required. In general it is therefore better to use better shaped elements in critical areas.

In the mesh option there are other futures to specify as well, full and reduced integration elements, "hourglassing" and "plane strain" or "plane stress" elements.

- Full or reduced integration elements; the reduced integration reduces the simulation time, especially for 3D analysis. For first order elements the simulation time is not reduced, and the accuracy of the solution depends on the problem. Normally the full integration is more accurate for first order elements.

- Hourglassing can be a problem with first order element, especially in stress/ displacement analysis.
- Plane stress and plane strain is suitable for different problems. In general plane stress should be used for thin models whilst plane strain should be used in models who has a large thickens compared to the rest of the model

4 | Dog-bone study

4.1 Dog-bone with fretting pad

The aim of the *Dog-bone* study is to perform finite element analysis to find the stresses and strains to perform post-processing analysis. The output from this study could be validated with reported results in the literature. The fretting-pad slides a small distance perpendicular to the specimen. This is to initiate a fretting induced fatigue crack in the specimen. Another important purpose of this study is to look at convergence in the results in order to decide an approximate element size for further studies.

4.1.1 Material properties

Both the fretting-pad and the specimen have the same material properties, both with the properties of Aluminium 2024-T351. The material is the same for both parts for simplicity reasons, since the stress distribution is the most interesting thing to study, the material could be the same for both. If comparing to a physical experiment, one could look at different materials in order to compare the test results with the finite element results. Material properties [12];

- Young‘s modulus; 74.1 GPa
- Shear modulus; 28 GPa
- Poisson‘s ratio; 0.33
- Yield strength; 324 MPa
- Ultimate tensile strength; 469 Mpa
- Fatigue strength; 138 MPa
- Shear strength; 283 MPa

4.1.2 Finite element model

The model is an assembly made of two parts, one fretting-pad and one specimen. The specimen is modelled with a 13,5mm diameter in the middle and 18mm diameter at the end on each side. The length is 115mm. The complete model is illustrated in figure 4.1.

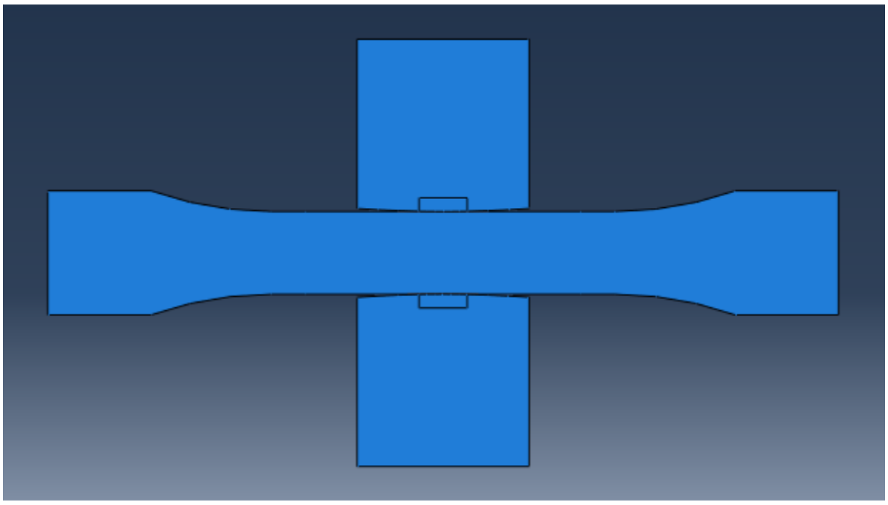

Figure 4.1: Illustration of Dog-bone

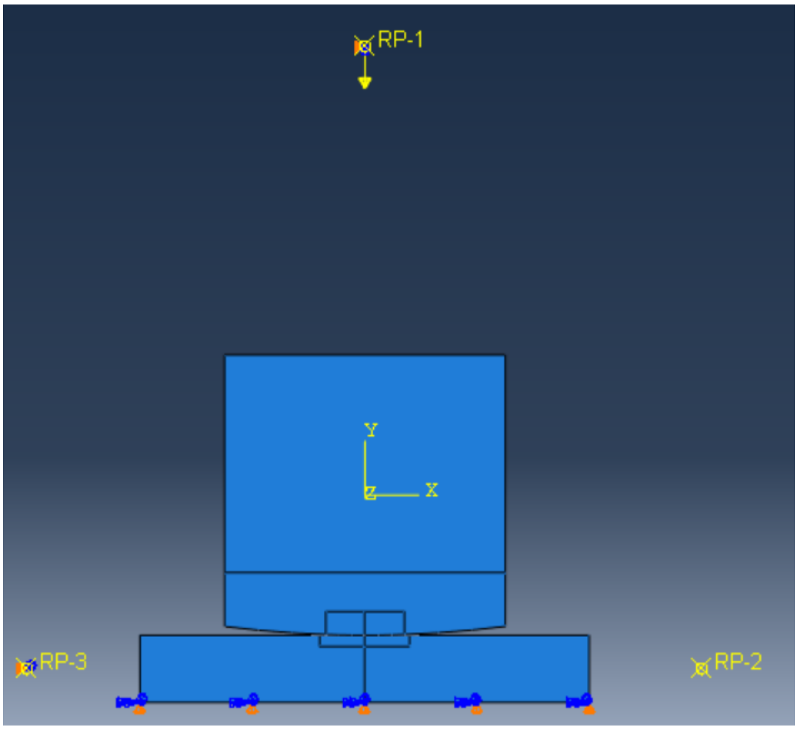

Figure 4.2: 2D model of test assembly

In order to reduce the simulation time, the original *Dog-bone* assembly is reduced by introducing symmetry to the model. Since both the model, load and displacement are symmetric about the horizontal centre of the model, it is safe to use symmetry. To achieve identical results for the reduced model, different boundary conditions are introduced. These will be described later under the section Model setup. Figure4.2 shows how the model is designed, the fretting pad is 25x25 mm with a radius of 100 mm, the specimen is 6,75x40 mm.

4.1.2.1 Model setup

	Name	Initial	Boundary con	Load	Movement
✔	DispPad				Created
✔	RP-left	Created	Propagated	Propagated	Propagated
✔	RP-right	Created	Inactive	Inactive	Inactive
✔	RP-topRotDisp	Created	Propagated	Propagated	Inactive
✔	Rp-top				Created
✔	Symmetry	Created	Propagated	Propagated	Propagated

Figure 4.3: Boundary conditions diagram

- DispPad; displacement of the fretting pad is sett to 0,05 mm in the x-direction. The displacement is created in the Movement-step, that means it is not activated in previous steps.
- RP-left; Reference Point is fixed in x, y and z direction
- RP-rigth; Reference Point is fixed in x, y and z direction
- RP-topRotDisp; the top reference point is fixed in x and z direction. This boundary condition is made inactive in the movement step. This is to allow the displacement of the fretting pad
- RP-top; the top reference point is fixed in z direction and rotation around x, y and z axis. The boundary condition is activated in the movement step
- Symmetry; In order to reduce simulation time, symmetry is introduced about the x-axis. Since the load and displacement conditions are symmetric it is possible to reduce the model.

The load is created in the step called Load. The load is set to 2000 N in negative y-direction and is initiated in the Reference point above the model, RP-1 in figure 4.2. In the main study, the load is reduced to 1000 N.

4.1.2.2 Mesh

In a fretting fatigue analysis, the element size is of great importance. In order to obtain the stick/slip effects, the elements have to be small enough. Smaller elements gives more equations to solve and by that increases the simulation time and the computational cost. Therefore it is important to find the element size small enough to be accurate, but big enough to give a reasonable simulation-time. Even though it is desirable to reduce the simulation time, the important part, without doubt, is the accuracy. In order to obtain a satisfying element size, a *Python* script were made to run multiple analyses with different element size in each analysis. The *Python* script makes these analyses automatic and there is no need for manual update. The element sizes that were tested using the *Python* script were; 1, 0.5, 0.1, 0.05, 0.03, 0.015. Not all the results will be presented, since the largest element sizes were known to be to large. In addition to updating the element size, the *Python* script generated text files with values of Von Mises stress and slip amplitude for every element size.

The mesh is divided into sections in order to divide the elements into different sizes. The purpose of this is to have a global mesh with as large elements as possible, whilst the local mesh near the contact area are of a smaller size. It is also important to control the transition between small and large elements, and that they don't get deformed. Figure 4.4 illustrates the outer contact surfaces, they are defined in order to give the mesh-update the correct direction, and give a smooth transition from local to global size. Figure 4.5 illustrates the area where the mesh is of local size. The areas are of great importance regarding the accuracy of the test model. The finer local mesh size gives a more accurate and precise test-result of the stress distribution in the two parts.

Figure 4.4: The outer contact surfaces

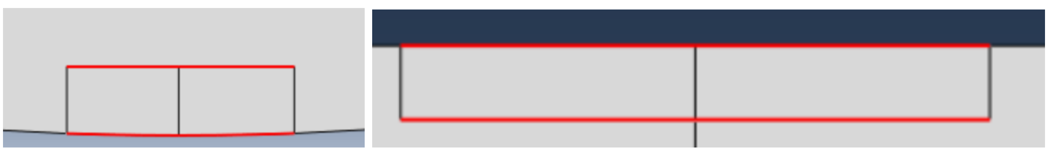

Figure 4.5: Contact surface

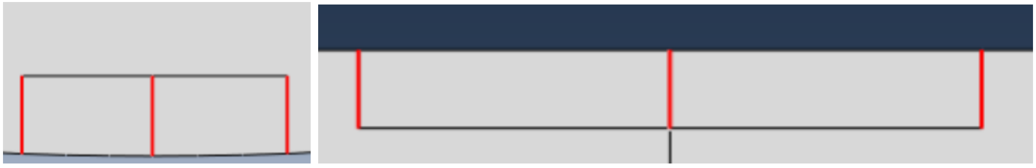

Figure 4.6: Cross section

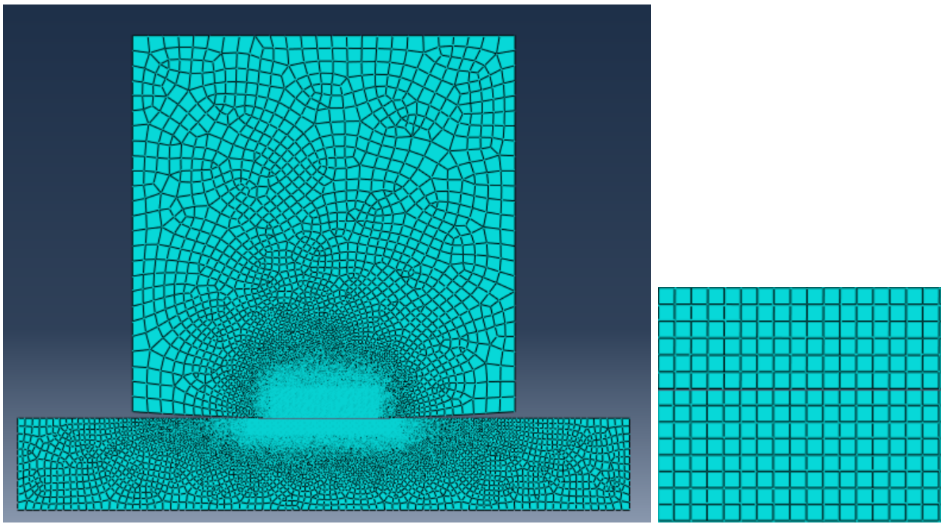

Figure 4.7: Mesh of the Dog-bone and detail to the right

Figure 4.7 shows the mesh of the assembly and the detailed figure to the right shows the mesh around the contact surfaces, which is of a very fine element size.

4.1.3 Analysis

The first analysis were performed with "Lagrange Multiplier" as tangential friction behaviour, and "Augmented Lagrange" as normal behaviour. This contact properties gave a rather slow simulation with very small step-time, down to 0.001 seconds. The analysis completed without any error, but with warnings about convergence difficulties due to the combination of "Lagrange friction" and "augmented Lagrange". As mentioned in the theory chapter, this is the recommended contact properties for a fretting fatigue analysis [2]. Even though the computational cost increased with the use of "Lagrange Multiplier" and "Augmented Lagrange", this were still the preferred contact formulation based on the theory. And since the analysis were done with the *Python* script, the analysis were created automatically, and could be run whilst other tasks were done. The first analyses were performed in order to obtain convergence in the results, and by that decide the element size for further analyses.

The second analysis were performed with the same contact formulations as the first one, but with a predetermined element size of 0.015 mm in the contact area, and a larger global size. In this analysis more outputs were to bee analysed; slip amplitude, frictional shear stress, Von Mises stress and stress in the x-direction. All stresses is taken from the path of the contact surface on the *Dog-bone*. These analyses were made with plane strain elements, CPE4. In addition, one analysis with plane stress elements were performed in order to compare the results.

The third analysis were performed with "Penalty" friction as tangential friction behaviour, and "Penalty" stiffness as normal behaviour. This was done in order to compare the results from the different contact formulations, and by that have the opportunity to use this formulation on the larger model in the *Dovetail* study.

4.1.4 Results

4.1.4.1 Pre-study

From the pre-study of the *Dog-bone*, the results gives an indication on how to solve the main task of the thesis. From figure 4.8 to figure 4.10 one can see that the results for Von Mises stresses converges around 0.03 and 0.015 mm element size. This indicates that the element size in the *Dovetail* study later, should possibly be around 0.015 mm in the contact area. The 0.015 mm gives a slightly higher stress in the area near the contact surface, but the difference is almost negligible. If we look at the computational cost, it is reasonable to assume that 0.015 mm will be sufficient enough.

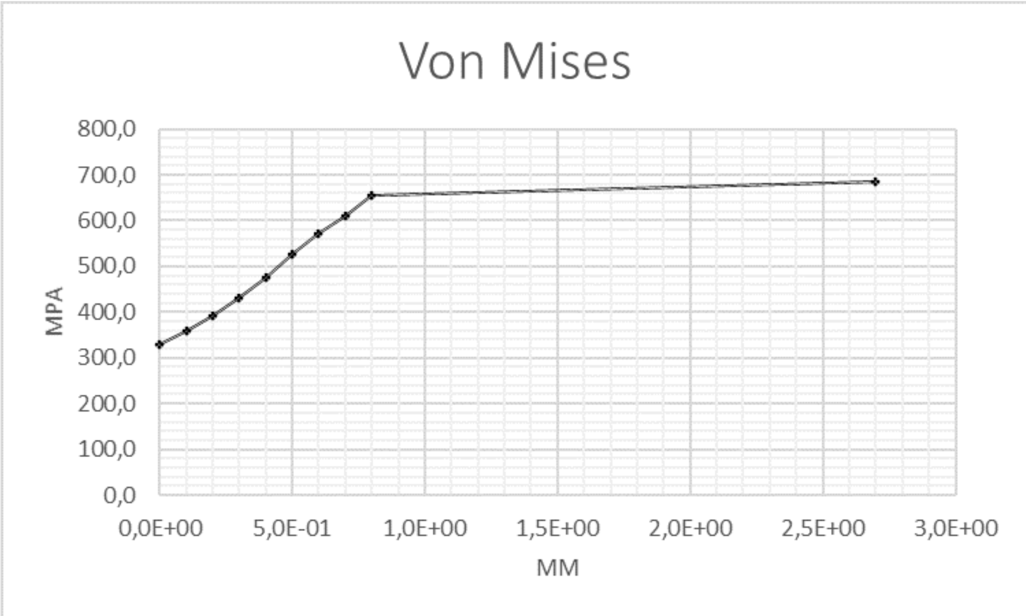

Figure 4.8: Von Mises stress 0.1mm element size

The analysis with element size 0.1mm around the contact area gives a result of approximately 680 MPa close to the contact surface. This is illustrated in figure 4.8 where the y-axis is the Von Mises stress in MPa and the x-axis is the distance in y-direction from the contact surface of the specimen, 0.0 is actually 2.5 mm from the surface, whilst 2.5 mm is at the surface.

From figure 4.9 and 4.10 one can see that the Von Mises stress for element size 0.03mm and 0.015mm is almost identical, both the maximum around the contact surfaces and the graphs itself are very similar. As mentioned earlier in this section, this is a good indication that the element size probably should be around 0.015mm. Another important parameter to include when deciding the element size is the contact slip amplitude. This gives a indication of the stick/slip behaviour of the surfaces, and are therefore important in this case. From the figures below it is obtained that the results are very much alike for all the element sizes, but with a slightly increased value for element size 0.03mm and 0.015mm. By that one can assume that the result converges someplace around 0.03mm element size. If combining the Von Mises and the contact slip amplitude, it is safe to conclude that a element size of 0.015mm will give results accurate enough for a fretting analysis.

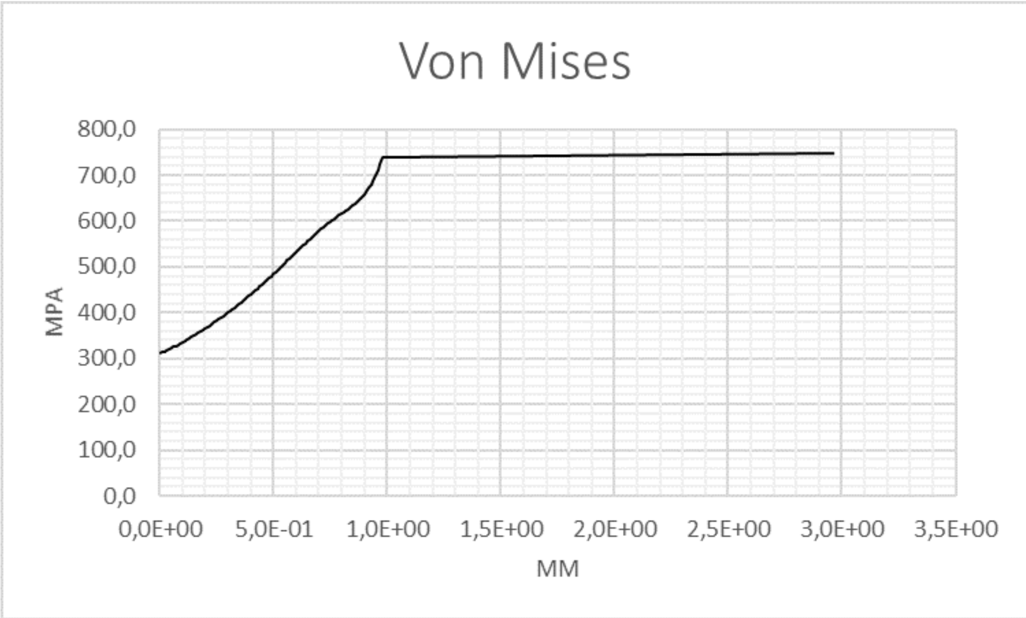

Figure 4.9: Von Mises stress 0.03mm element size

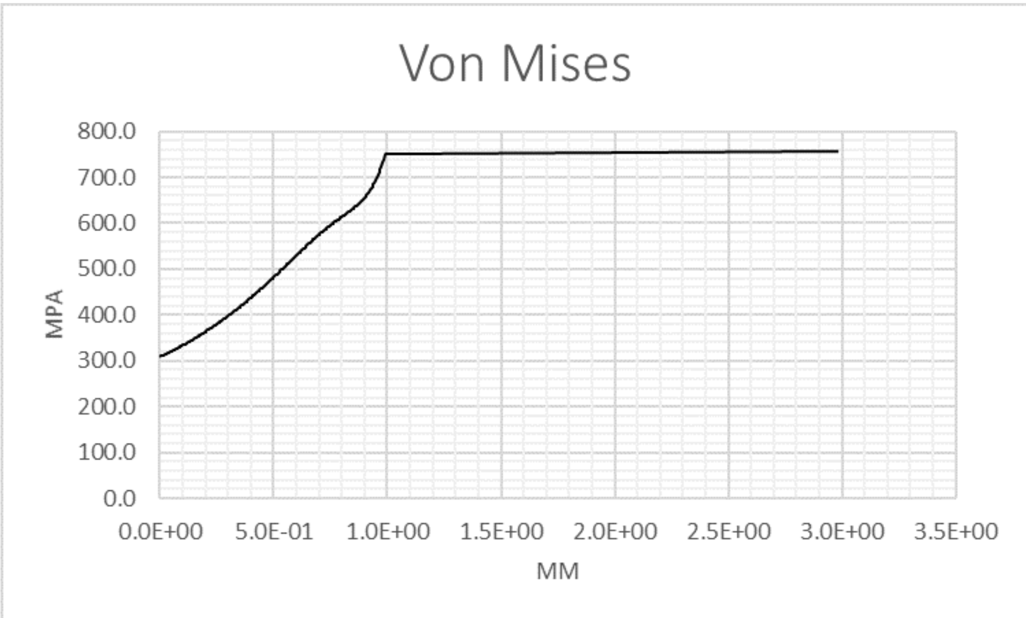

Figure 4.10: Von Mises stress 0.015mm element size

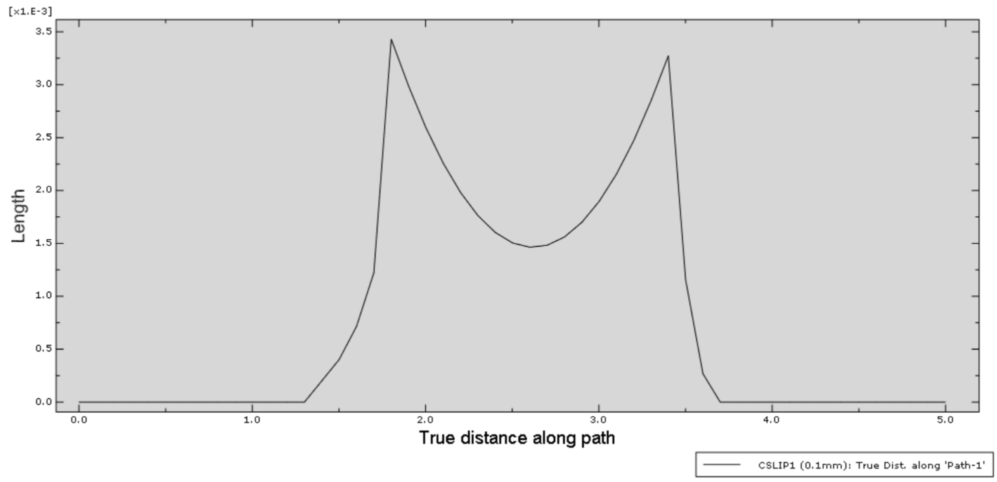

Figure 4.11: Contact slip for 0.1mm element size

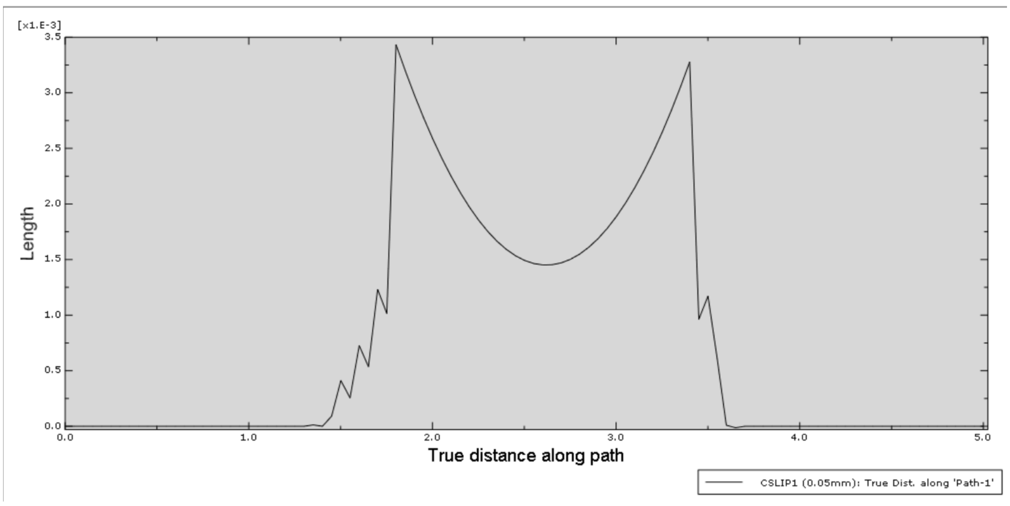

Figure 4.12: Contact slip for 0.05mm element size

Figure 4.11 to 4.14 shows the slip amplitude for element sizes 0.1, 0.05, 0.03 and 0.015 mm. The graphs are from 100% (0.05mm) displacement and with a time-step 0.1 second, with a total of 1 second, therefore this analysis is not as accurate as in the main study, but it is as mentioned only for deciding the element size. And these plots give very similar results for all element sizes, with a small increase in the slip amplitude from 0.05 to 0.03 mm element size. This along with the convergence in the Von Mises stress from 0.03 to 0.015 mm element size, further strengthens the assumption that 0.015 mm element size should be sufficient for the main study, both for the *Dog-bone* and the *Dovetail* study. Later an analysis with smaller time-steps and other parameters like shear and stress distribution will be presented.

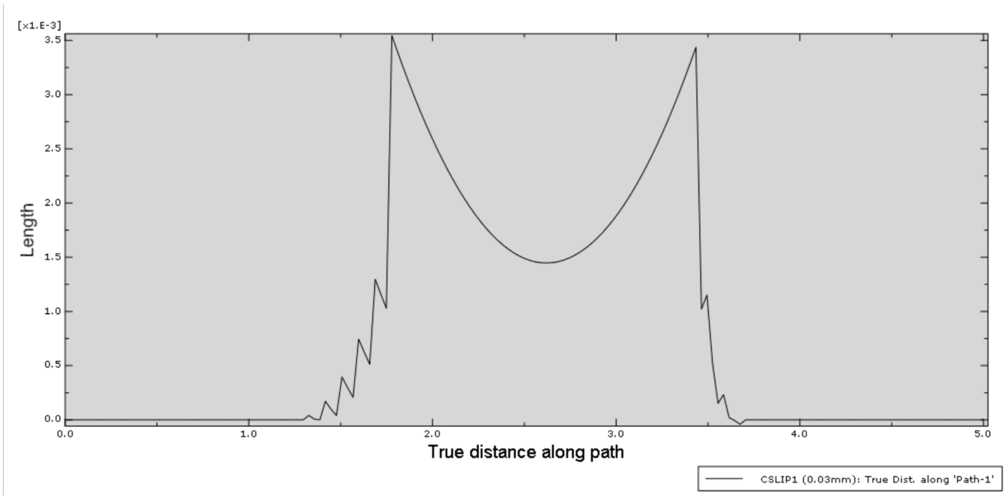

Figure 4.13: Contact slip for 0.03mm element size

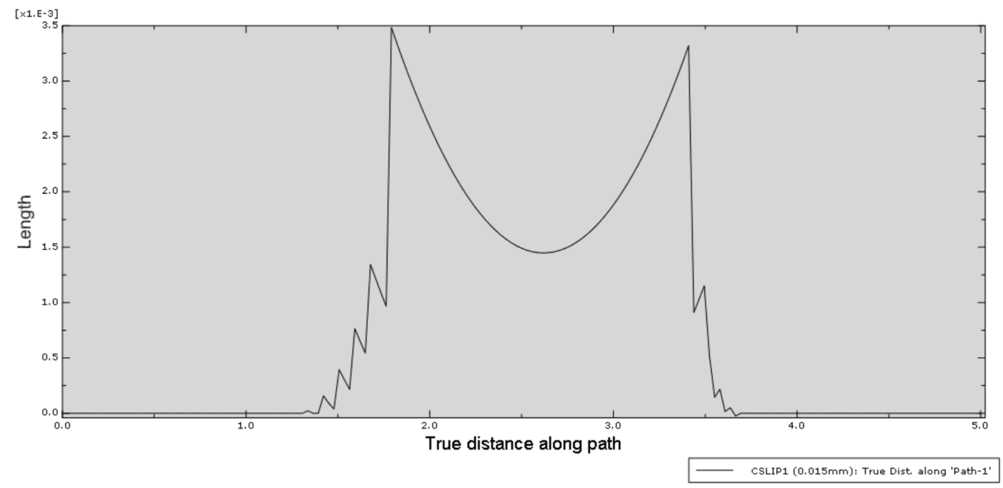

Figure 4.14: Contact slip for 0.015mm element size

4.1.4.2 Main study

The main study of the *Dog-bone* with fretting pad is set up very similar to the pre-study as described earlier. The difference is mainly the step-time which is decreased from 0.1 second to 0.01 second for the displacement-step. This is in order to get a better plot for analysing the results and to better understand the importance of the relative displacement between the two surfaces. The parameters to be analysed are as follows;

- Relative tangent at surface nodes
- Frictional shear stress at surface nodes

- Von Mises stress at surface nodes
- Stress in x-direction at surface nodes
- Contact pressure

The relative tangent at surface nodes describes the stick/slip behaviour of the two surfaces. This parameter is important in order to understand why there will be a stress gradient where the behaviour goes from sticking to slipping, and in what area this typically occurs. This is the parameter that is highly dependent on the element size as described earlier.

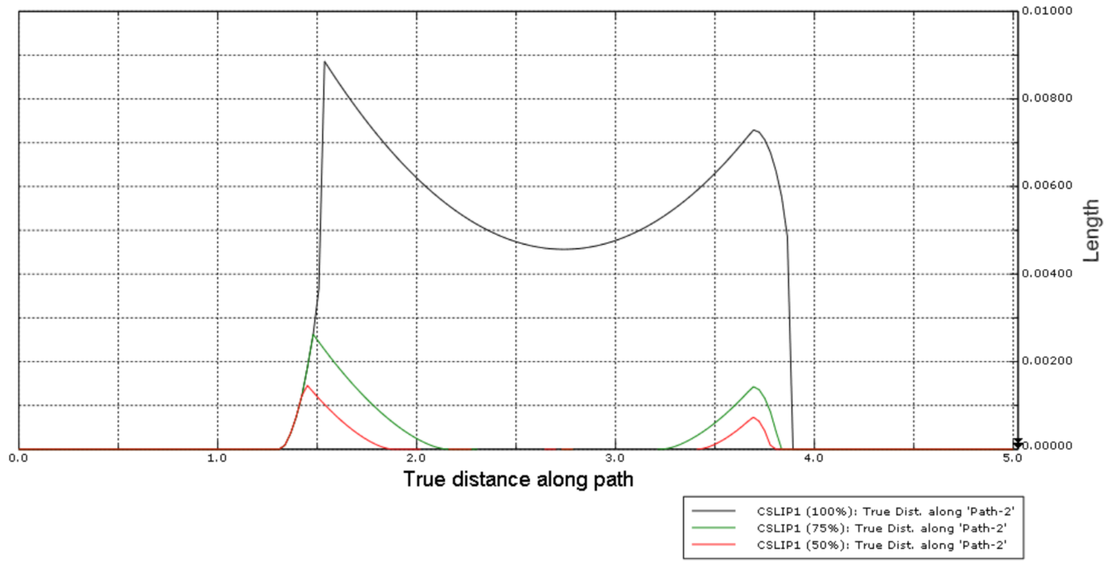

Figure 4.15: Stick/slip behaviour

The graphs in figure 4.15 shows the stick/slip behaviour from three different positions in the displacement. The red, green and black graph shows the behaviour after 50% (0.025mm), 75% (0.0375mm) and 100% (0.05mm) displacement respectively. As we can see the stick/slip behaviour is highly dependent on the displacement. In this analysis the displacement is 0.05mm or 50μm, and there is relatively much difference in 25μm and 50μm displacement. The stick zone decreases as the displacement increases, as long as the load is kept constant, as it is in this case, the whole contact zone will undergo partial slip after a certain amount of displacement. This will increase the stress in and around the contact area, both regarding the shear and the von mises stress.

Figure 4.16 shows the slip amplitude for the plane stress analysis. The plane stress analysis gave a slightly decrease in the slip amplitude, especially after 100% displacement. the peak slip amplitude goes from 0.0088 in plane strain to 0.0064 in plane stress. But after 50% and 75% displacement, the graphs are identical. This could indicate that the plane stress condition are more dependent on the relative displacement.

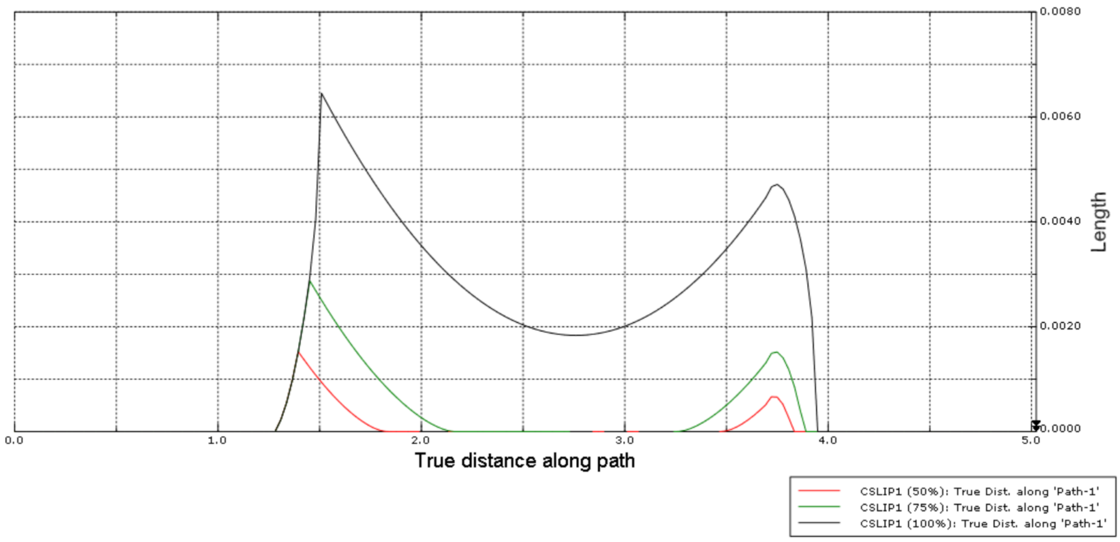

Figure 4.16: Stick/slip behaviour plane stress

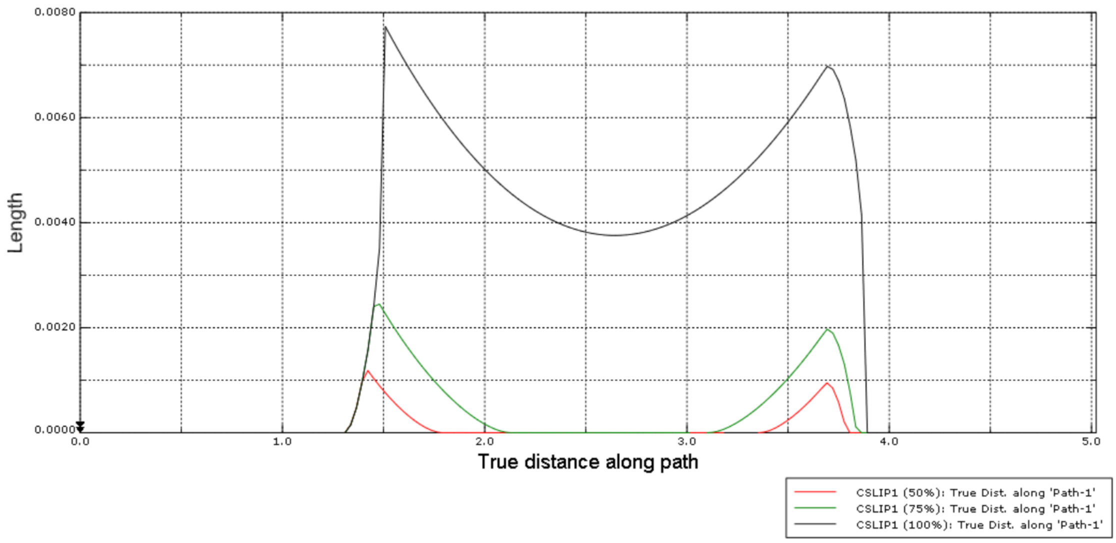

Figure 4.17: Stick/slip behaviour Lagrange Multiplier

Figure 4.17 shows the slip amplitude with Lagrange multiplier. The peak value is slightly lower than for the penalty in figure 4.15. After 50% and 75% displacement the values are similar.

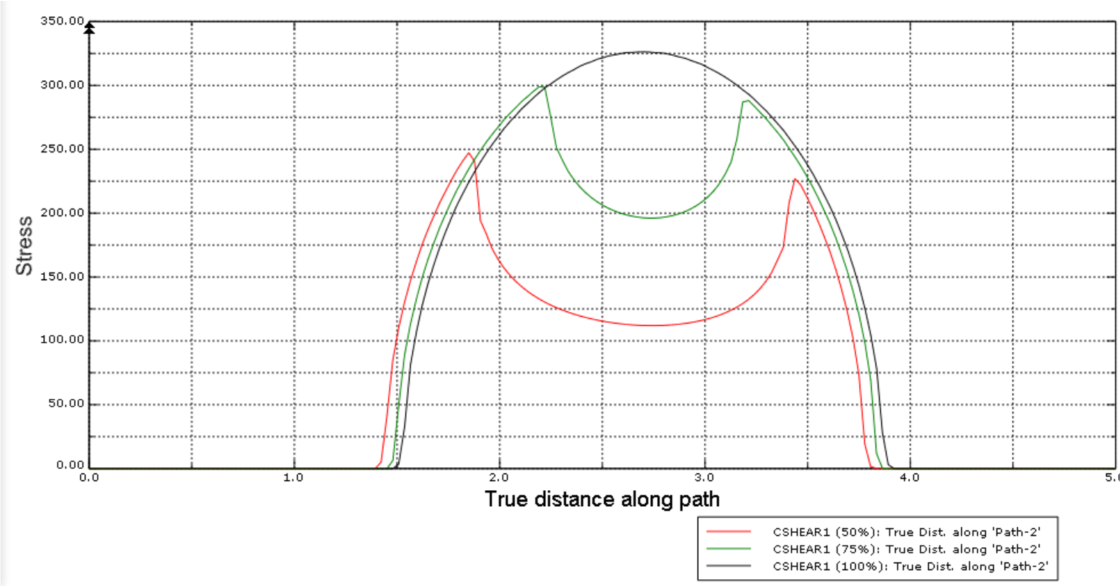

Figure 4.18: Frictional shear stress

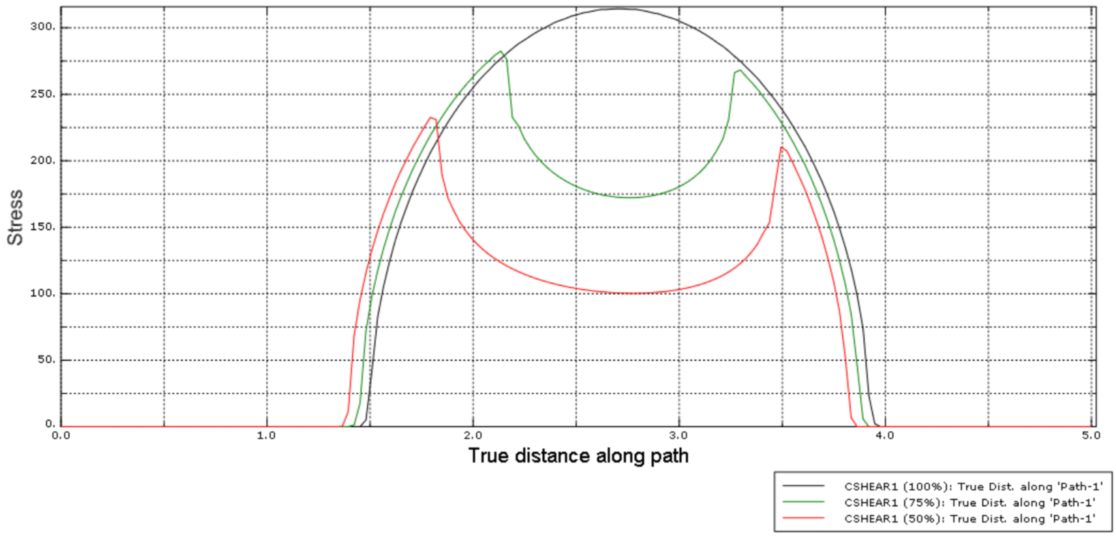

Figure 4.19: Frictional shear stress, plane stress

As figure 4.18 shows the shear stress is also dependent on the displacement. This is as expected since the stick/slip graph indicates more slip at the end of the contact zone than in the middle. This is especially clear after 50% displacement where the area in the middle does not undergo slip, and the shear stress is significantly lower in this area. Whilst after 100% displacement the shear stress peeks in the middle of the contact surface.

Figure 4.19 shows the shear distribution in the *Dog-bone* with plane stress conditions. The peak stress after 100% is very similar to the one for plane strain conditions in figure 4.18, whilst the stresses after 50% and 75% displacement are slightly increased.

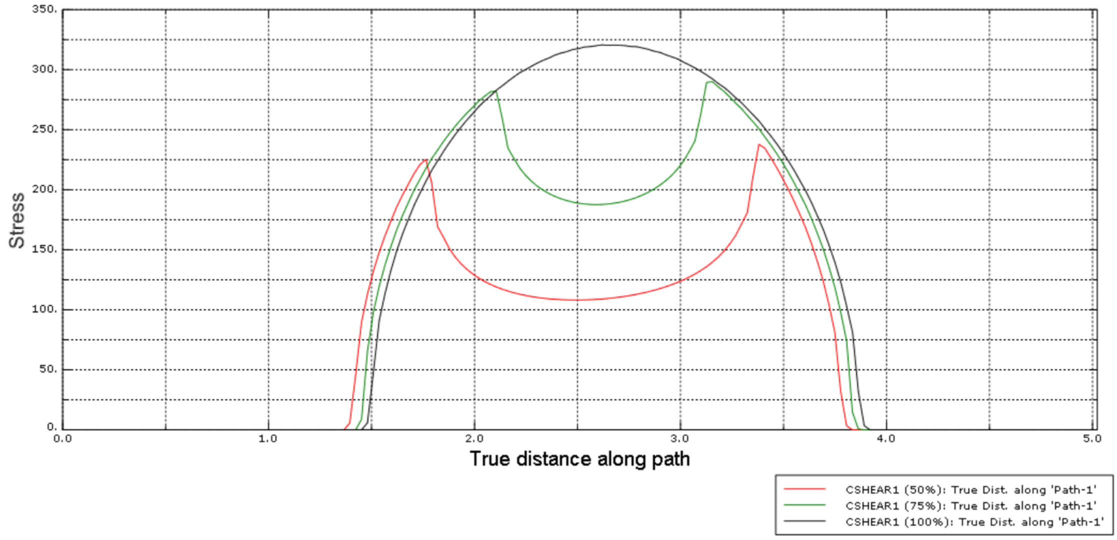

Figure 4.20: Frictional shear stress, Lagrange Multiplier

Figure 4.20 shows that the Lagrange Multiplier configuration gives a identical shear stress when the whole contact surface undergoes partial slip, whilst for the peak stress around the stick/slip transition, the Lagrange Multiplier gives a slightly decreased shear stress.

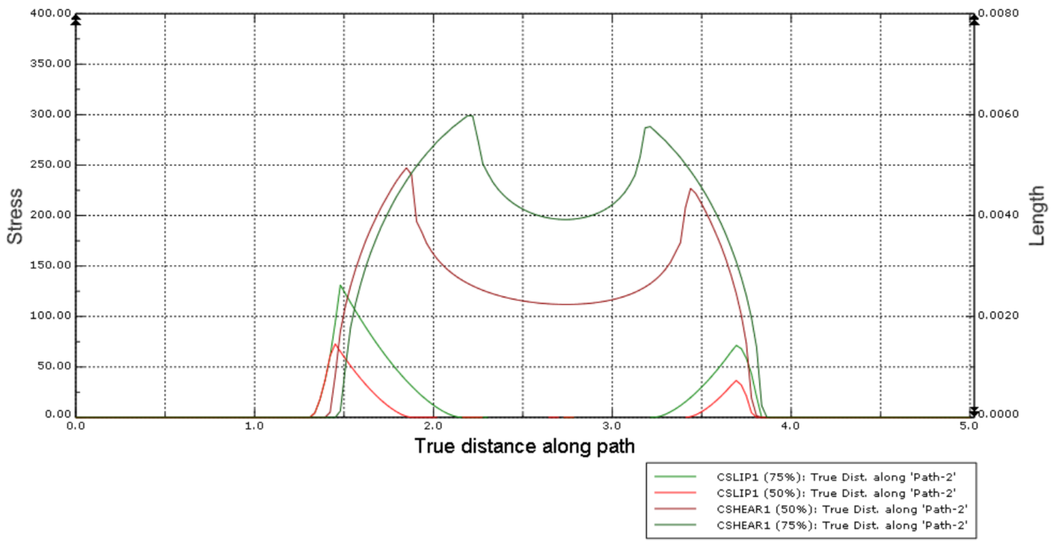

Figure 4.21: Shear stress and slip along path

The correlation between slip and shear stress along the contact surface for the plane strain conditions with penalty formulation are shown in figure 4.21. The upper two graphs are the shear stress after 50% and 75% displacement, whilst the lower two are the slip amplitude after 50% and 75% displacement. The graphs shows that the shear stress is at its peak where the contact goes from stick to partial slip. Here the slip amplitude of 0.0025mm is below the critical threshold value of 50μm, as described in the theory chapter figure 3.3. This indicates that the stress raiser finds place around the area where the behaviour changes from stick to partial slip, at least for as long as part of the geometry is sticking. As mentioned above the shear stress is at its highest when the whole surface undergoes partial slip.

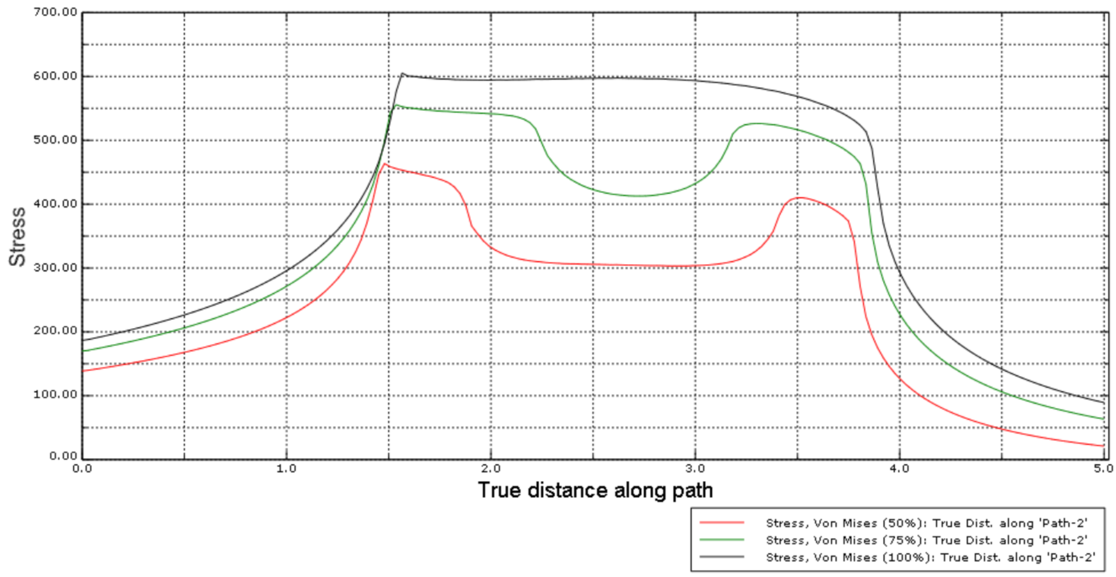

Figure 4.22: Von Mises stress

The Von Mises stress is as the shear stress highest at the end of the contact zone as long as the displacement is 50% and 75%. This is due to the stick/slip behaviour in the contact surface. As described for the correlation between the shear stress and slip amplitude above, the highest stress values are in the area where the surfaces goes from stick to partial slip. Although after 0.05mm displacement, 100%, the Von Mises stress is equally high over the whole contact zone, this is due to the partial slip in this area.

The stress in the x-direction along the contact surface has the same behaviour as the stresses described above. The peek stresses are around the ends of the contact zone, but here the indicator changes from positive to negative. Whether the stress is negative or positive has no influence on either the shear stress or the Von Mises stress. The stress is not as dependent as the above mentioned regarding the displacement of the fretting-pad, even though the stresses are slightly higher for the 100% displacement than the 75% and the 50%. This is because this stress is mostly induced by the friction force, which comes from the normal load and the coefficient of friction.

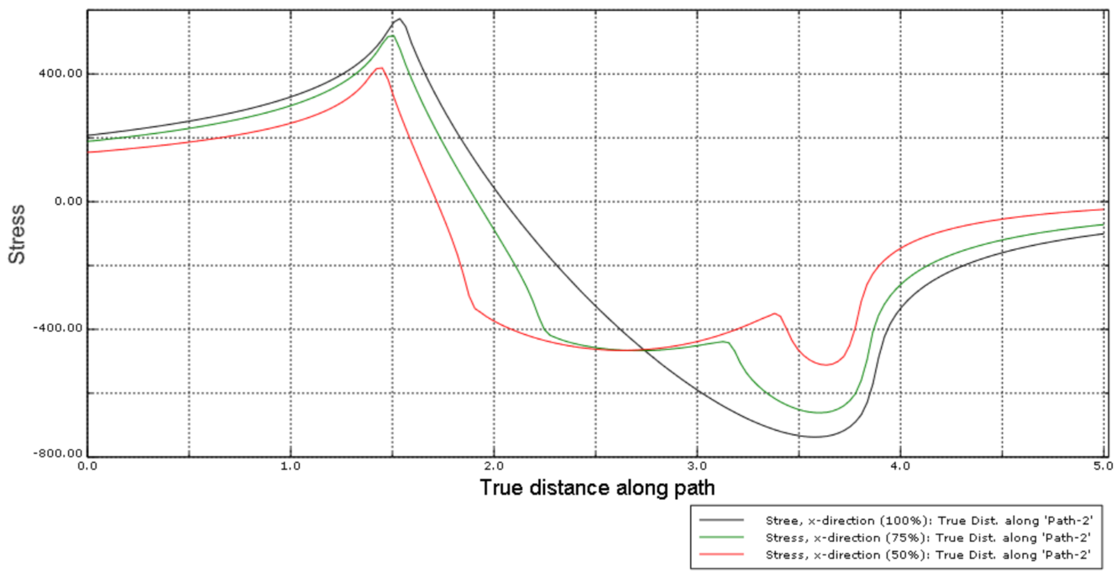

Figure 4.23: Stress in the x-direction along the contact surface

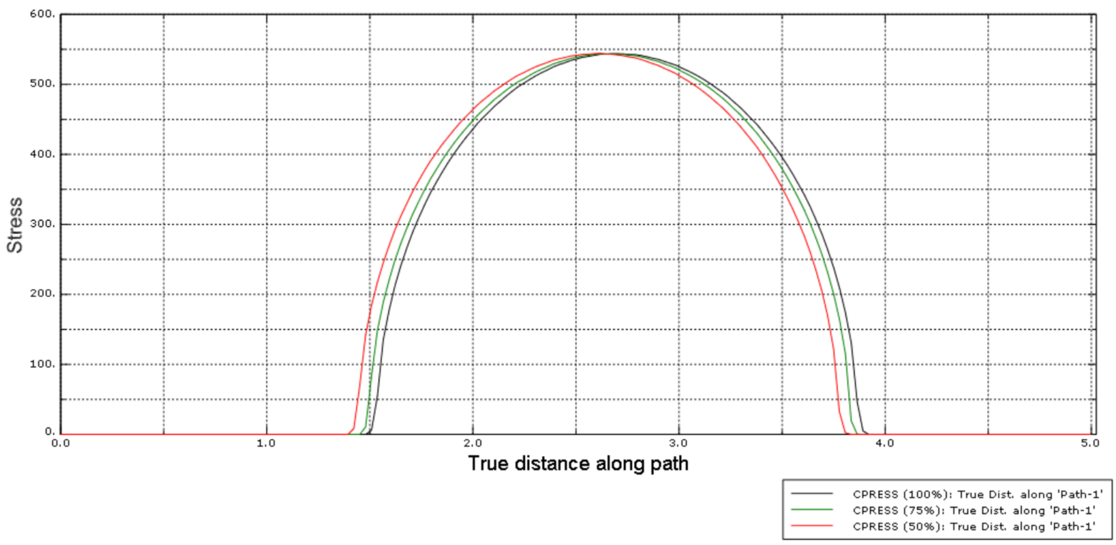

Figure 4.24: Contact pressure

The contact pressure is given in figure 4.24, the pressures are the same for every kind of displacement. The only difference between the grade of displacement are that the pressure moves with the relative displacement.

5 | Dovetail study

Dovetail root fixings are frequently used to attach blades to disks in gas turbine engines and aeroplane engines. Under the action of centrifugal loading and vibration, fretting fatigue may occur between the blade root and the disc and the stresses in the neighbourhood of the contact need to be accurately estimated for fatigue life prediction [13]. The *Dovetail* study is performed with two slightly different geometries, one with two flat surfaces in contact and the other with one surface where there are a radii on the disk surface. The radius is equal to the radius of the fretting-pad in the *Dog-bone* study. Hopefully this can be comparable and give results that could be compared.

5.1 Finite element model

The geometry for the *Dovetail* study is modelled in *Abaqus*. The dimensions are not the most important parameter in the study, but it is important regarding the relative displacement of the turbine-blade, as well as for the element size which will be described later. Figure 5.1 and 5.2 shows the full assembly, the dimension of the disk and turbine-blade respectively. The dimensions are relative small compared to a real life example, but the important part of this thesis, is the understanding of fretting fatigue. And the size of the geometry is not the most important part.

Figure 5.3 shows the radii of the contact surface of the disk. The radii is set to 100 mm as for the fretting pad in the *Dog-bone* study. Other than the radius, the models are identical and the set up for the analyses are the same. The important dimensions of the model is the angle in the contact surface, which is set to 35 deg. The radii are made such that there will be no stress gradients in these areas, and avoid fatigue in these in a physical test that might be based on the geometry.

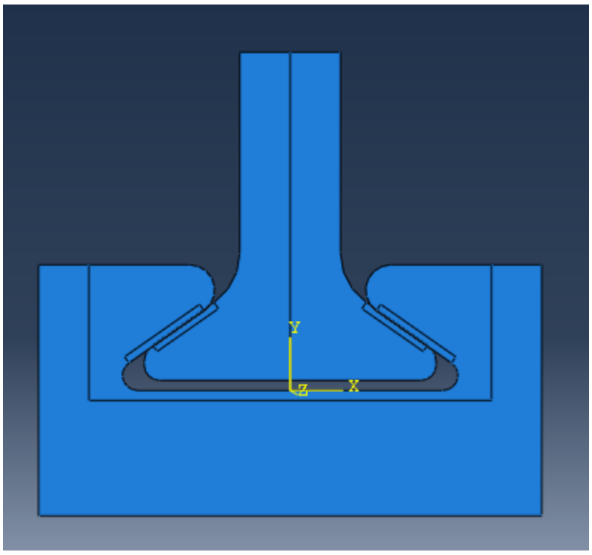

Figure 5.1: Illustration of dovetail assembly

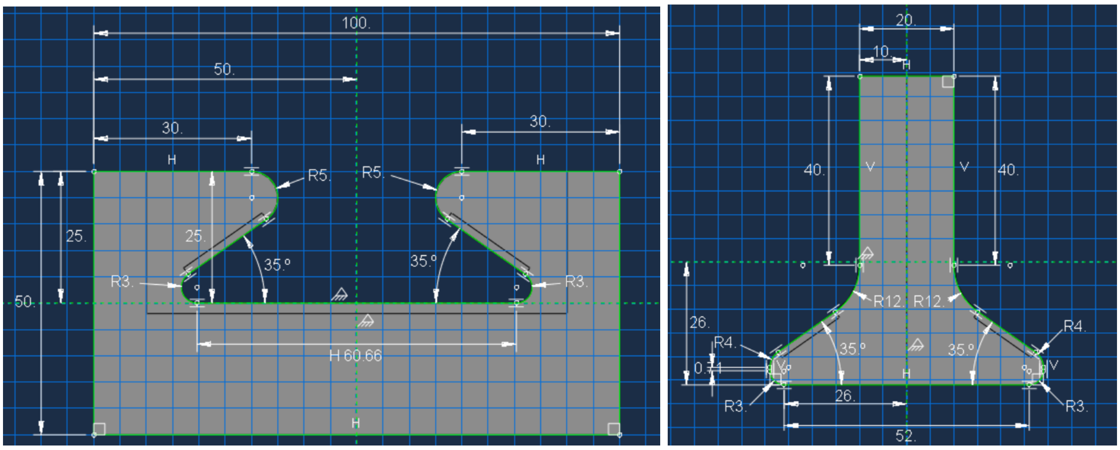

Figure 5.2: Dimensions of dovetail assembly

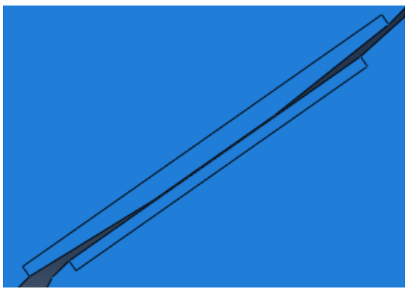

Figure 5.3: Radius of the dovetail

5.1.1 Model setup

Symmetry is introduced in order to reduce simulation time. By that boundary conditions needs to be introduced, figure 5.4 shows the different boundary conditions. In order to avoid yielding in the disk, the thickness of the model is set to 2mm whilst the turbine blade is 1mm.

	Name	Initial	Load
✔	Displacement		Created
✔	Fixed	Created	Propagated
✗	Load		Created
✔	Symmetri	Created	Propagated

Figure 5.4: Boundary conditions dovetail

- Displacement is set to 0.1mm in positive y-direction, and is initiated in the RP-top reference point under the Load step
- Fixed is a ENCASTRE conditions which mean that there are no degrees of freedom for the line in the bottom of the model
- Symmetry is on the right side of the model, and there are no displacement allowed in x-direction and no rotation around y-axis
- Load is disabled in this analysis, but is used for the *pfat* analyses, and is set to 100 N.

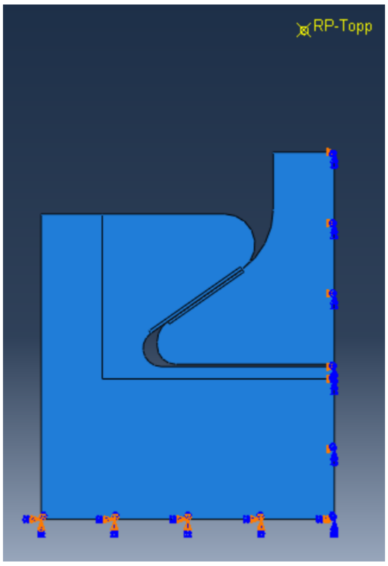

Figure 5.5: Illustration of 2D symmetry model

Figure 5.5 shows the 2D model with boundary conditions as it is modelled for analysis.

5.1.2 Mesh

As mentioned previously in chapter 4, the element size is important regarding the accuracy of the results that are to be obtained. In the *Dog-bone* study the element size was as small as 0.015mm, it is chosen to use the same element size for the *Dovetail* study. On this particular model it is not used a *Python* script to update the element size and check for convergence in the results. From the previous study, the knowledge about the element size is continued into this study, and therefor the element size is selected based on a better basis than what was possible for the *Dog-bone* study. The element type used in this study is as before CPE4, standard linear plane strain elements with four nodes in each element.

The mesh is divided into sections as for the *Dog-bone* study. The purpose of this is, as described in the previous chapter, to have a small element size at and near the contact surfaces, whilst the global size remain a larger size in order to reduce the simulation time. The section is illustrated in figure 5.6 and a illustration of the global mesh and the more detailed local mesh is shown in figure 5.7. The mesh is identical for the model with radius in the contact area, same element size and the same parameters for the mesh.

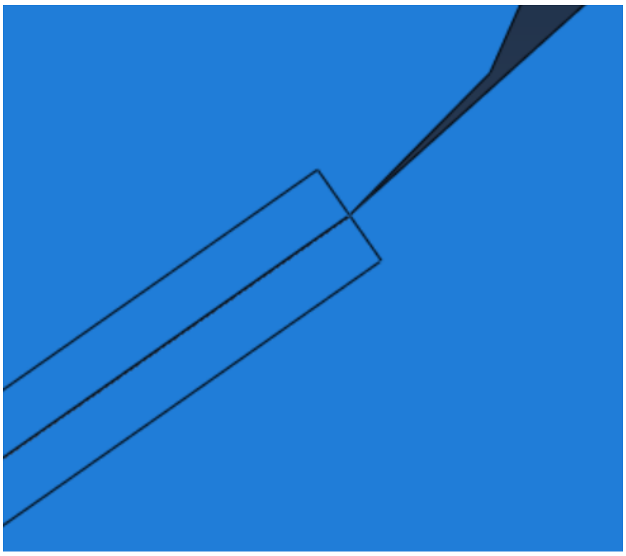

Figure 5.6: Detail of section

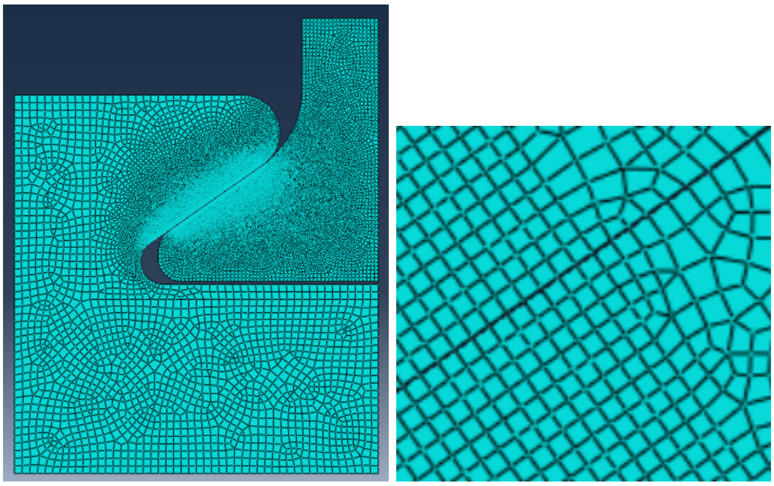

Figure 5.7: Mesh on dovetail, detailed for mesh around contact surfaces to the right

5.2 Analysis

For this study as well, there were performed two different analysis regarding the contact formulation, first "Augmented Lagrange" as normal behaviour and "Lagrange Multiplier" as tangential friction behaviour, then "Penalty" for both tangential friction and normal behaviour. Here the coefficient of friction is set to 0.6. Even though the *Dog-bone* study gave very similar results for the two contact formulations, it is necessary to do both here as well because of the difference in the geometry. The path it will be referred to later in the results is the red line illustrated in figure 5.8. In all graphs presented the left side is the beginning of the path which goes from bottom left, to top right. This path represents all the surface nodes of the contact area for the turbine blade.

Based on the results from the two first analysis, the "Penalty" friction and "Penalty" normal behaviour were chosen for the further analysis. In those analyses different coefficient of friction were used, in order to obtain the different stick/slip behaviour and thereby different stress distribution. 0.3, 0.6 and 0.9 as coefficient of friction, as described in the theory chapter there have been used many different values in previous studies, it is therefor interesting to study the differences in the results obtained.

The analysis for the model with radius in the contact area has the same setup as for the analysis with different coefficients of friction described above.

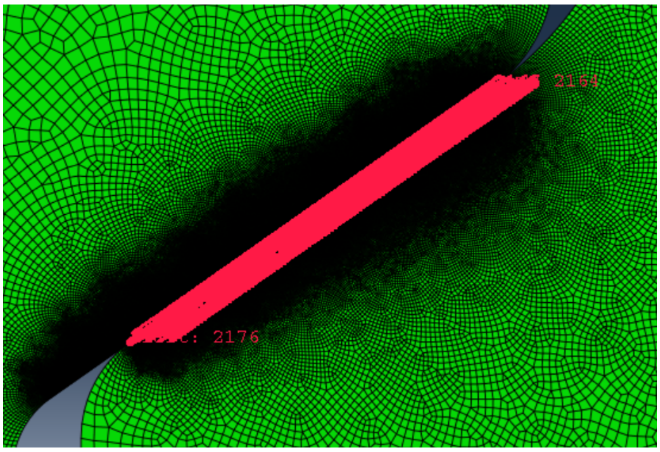

Figure 5.8: Illustration of the path in Dovetail study

The last analysis performed, was fatigue analysis in *pfat*, which is a post-processing analysis tool. In *pfat* the result files from *Abaqus* analyses are imported, and it is possible to perform life prediction analysis on them. The challenge with *pfat*, is that the program is based on linear behaviour of stresses and strains. It is therefore not very suitable for analysing fretting fatigue, but could be interesting to compare the results here with possible test results in the future.

5.3 Results

The results will be presented in different categories depending on both contact formulation used and the coefficient of friction. For all the graphs presented below, the red graph represents the results for 50% (0.05mm) displacement and the green and black represents the results for 75% (0.075mm) and 100% (0.1mm) displacement respectively. It is important to notice that the displacement induced here is only one direction, and in a real case scenario, as for a turbine engine, the displacement will be cyclic with vibrations as well on the blade.

5.3.1 Lagrange multiplier

As mentioned above the first analysis were performed using Lagrange multiplier as tangential friction behaviour and Augmented Lagrange as normal behaviour. This is the recommended formulation as mentioned in the theory chapter. The problem with the Lagrange Multiplier is the convergence problems that can occur, in this analysis, *Abaqus* used three days and finally the job was aborted at 90% completion. This formulation is therefore not used further in the *Dovetail* study, but some results are presented below.

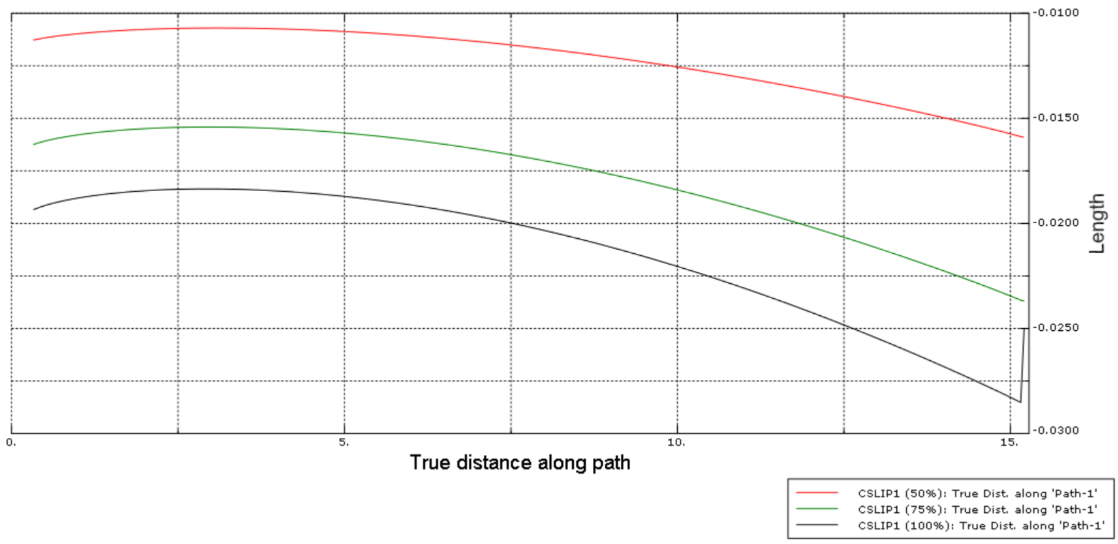

Figure 5.9: Slip amplitude with Lagrange multiplier

In figure 5.9 the slip amplitude along the path varies with the displacement and along the path. The slip amplitude for the "Lagrange Multiplier" are reaching values from 0.011-0.016mm for 50% displacement to 0.018-0.028mm for 100% displacement. These are values that lies in the critical area for partial slip based on figure 3.3.

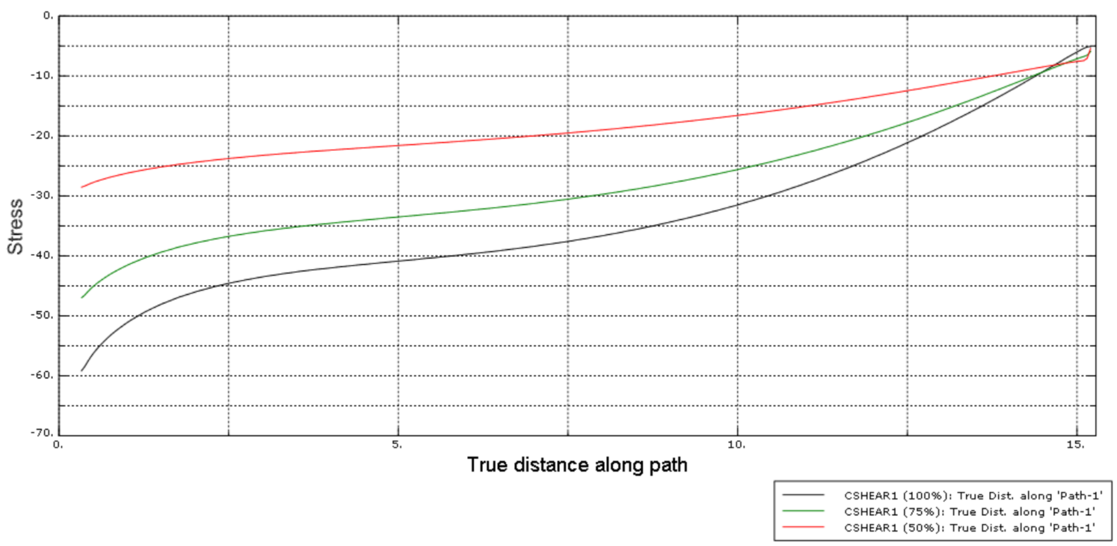

Figure 5.10: Shear stress with Lagrange multiplier

The shear stress along the path with Lagrange Multiplier gives a peak stress of approximately 60 MPa, this would have been higher if the analysis had finished instead of been aborted at 90%. The shear stress reduces along the path, highest at the lowest point of the turbine blade.

5.3.2 Coefficient of friction 0.3

The frictional coefficient set to 0.3 for the study is in the lower area of the studies done by others as mentioned in the theory chapter.

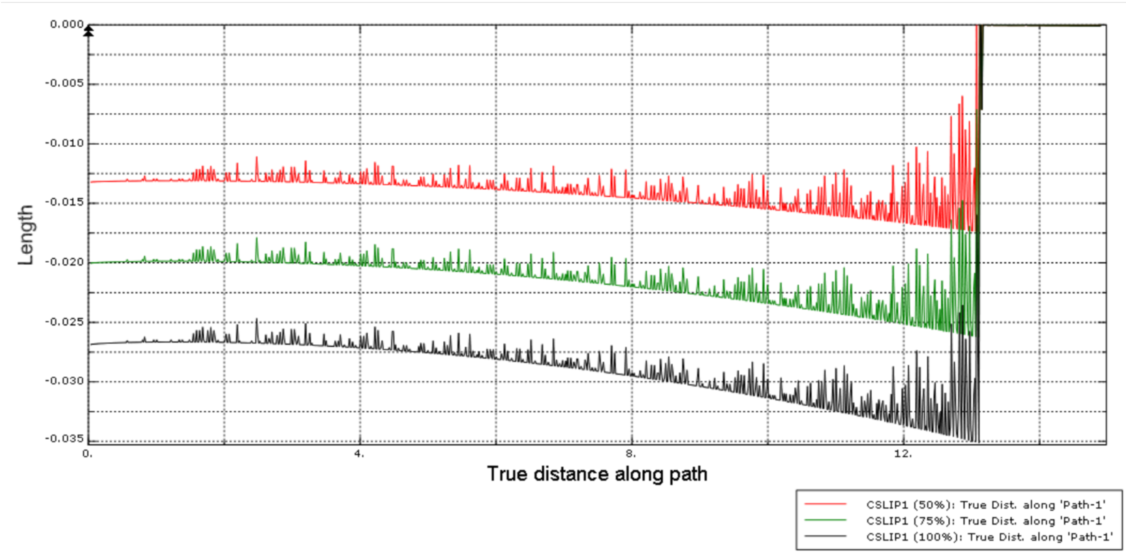

Figure 5.11: Stick/slip behaviour with 0.3 friction

In figure 5.11 we can see that the slip amplitude varies along the path of the contact surface of the turbine blade. The slip increases slightly along the path, this goes for all three graphs. With this low friction the slip amplitude reaches 0.035 mm at its maximum after 100% displacement in the y-direction. From figure 3.3 in the theory chapter this value is highly critical regarding the lifetime of parts exposed to fretting fatigue. Also after 50% displacement the value of 0.015 is critical based on figure 3.3.

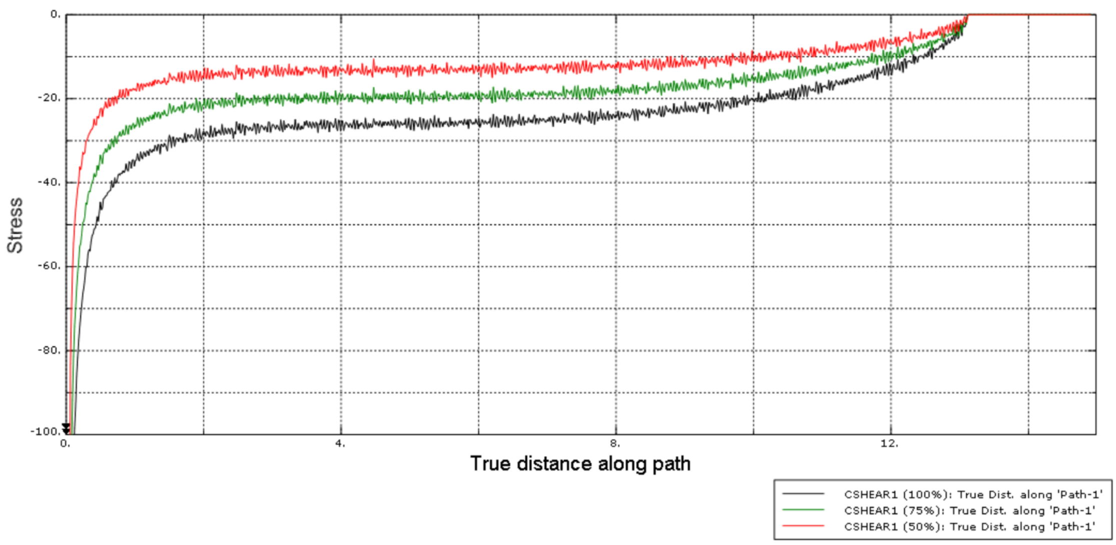

Figure 5.12: Frictional shear stress 0.3 friction

The frictional shear stress is shown in figure 5.12, where the shear stress naturally increases as the displacement increases. The very high stresses at the beginning of the contact surface is most likely a singularity in the FE model, which is unlikely to be real. The Shear stresses are at its highest where the slip is at its lowest. The peak frictional shear stress at 100% displacement is approximately 40 MPa.

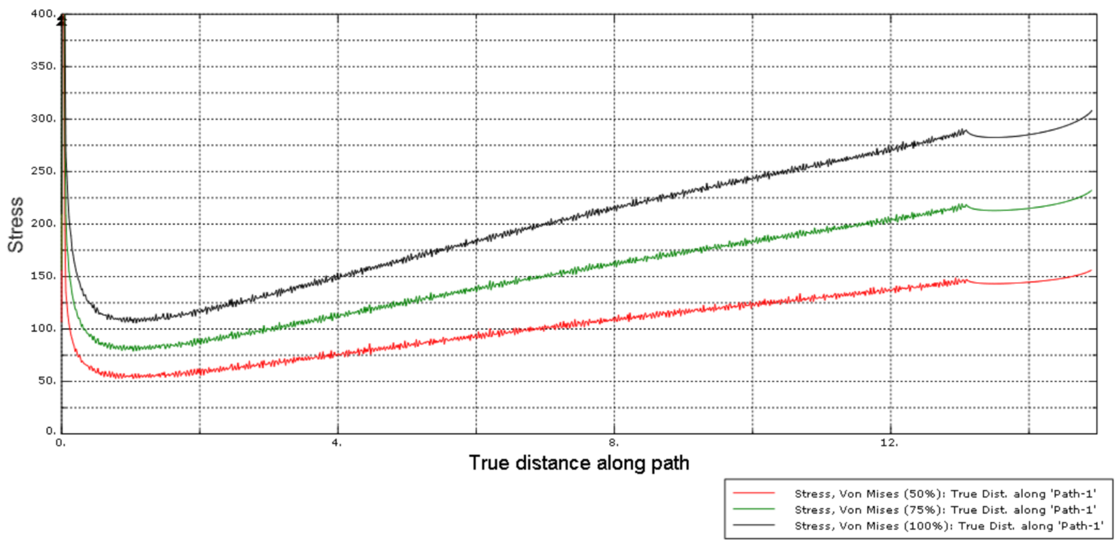

Figure 5.13: Von mises stress 0.3 friction

The Von Mises stress has a different stress distribution than the shear above. Here the peak stress is at the other end of the contact area, this is due to the geometry and the stress increase in the radius of the turbine blade. As for the sear stress, the high stress is most likely because of an singularity in the FE model.

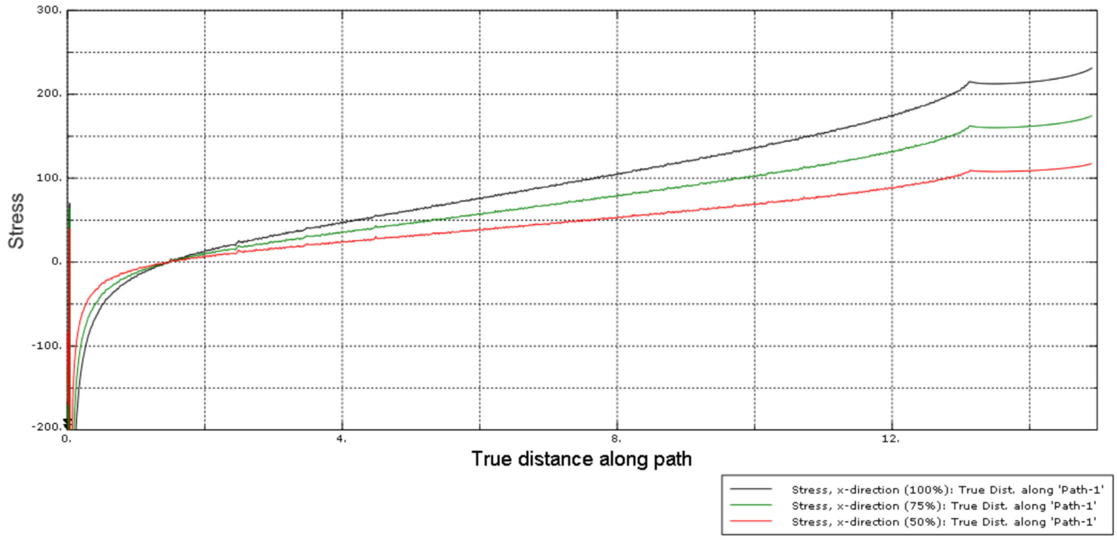

Figure 5.14: Stress in the x-direction along the contact surface 0.3 friction

The stress along the path in x-direction has a similar behaviour as the Von Mises stress, and the negative peak stress is the singularity.

5.3.3 Coefficient of friction 0.6

0.6 as coefficient of friction is in the middle of the area used for tests in fretting fatigue. This is mentioned in the theory chapter.

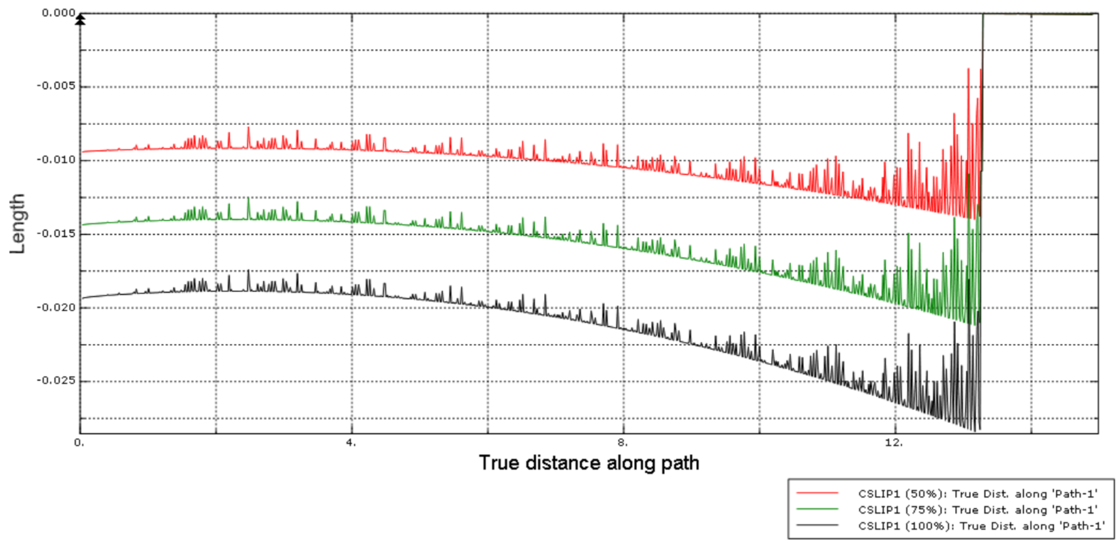

Figure 5.15: Stick/slip behaviour

As expected the slip length decreases as the coefficient of friction increases.

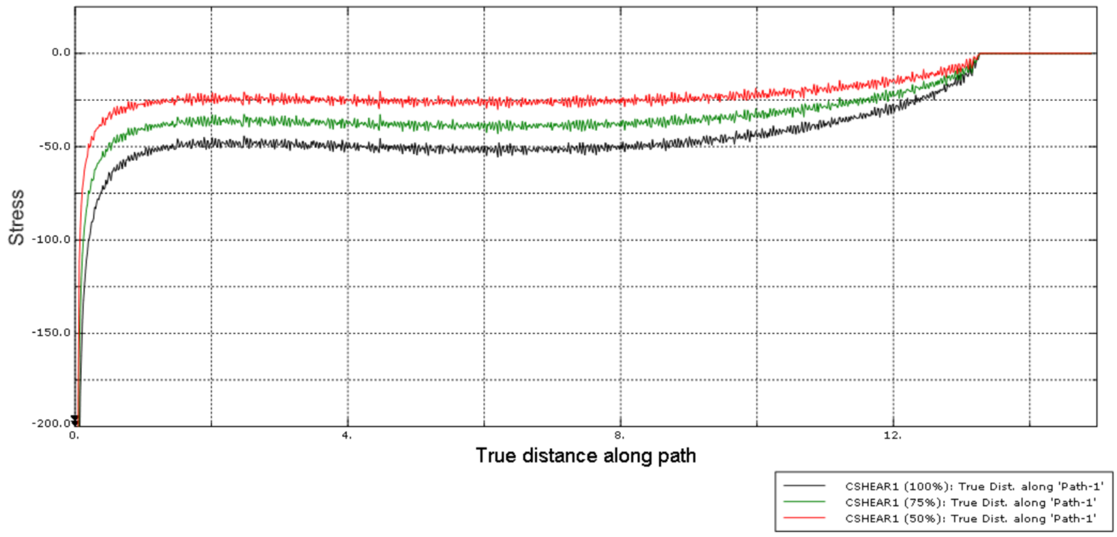

Figure 5.16: Frictional shear stress

The frictional shear stress along the path is slightly higher for 0.6 friction than for 0.3. This make sense since the increased coefficient of friction gives higher friction force, and thereby increased shear stress. The shear stress reaches -60 MPa, which is an increase of 50% from 0.3 friction shear stress.

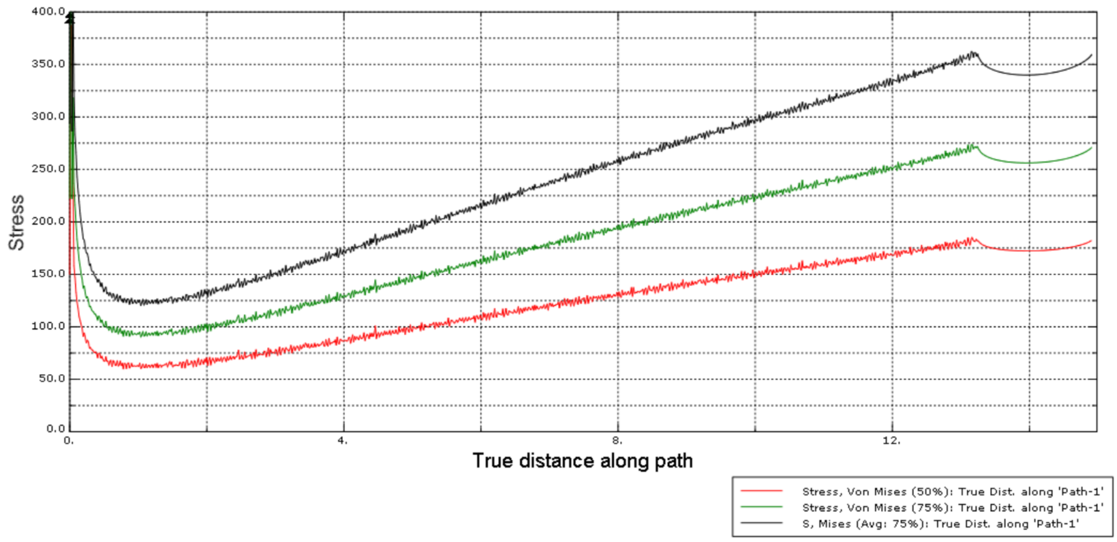

Figure 5.17: Von Mises stress

The Von Mises stress increases as well, close up to 400 MPa, as before the steep peak is the singularity in the FE model.

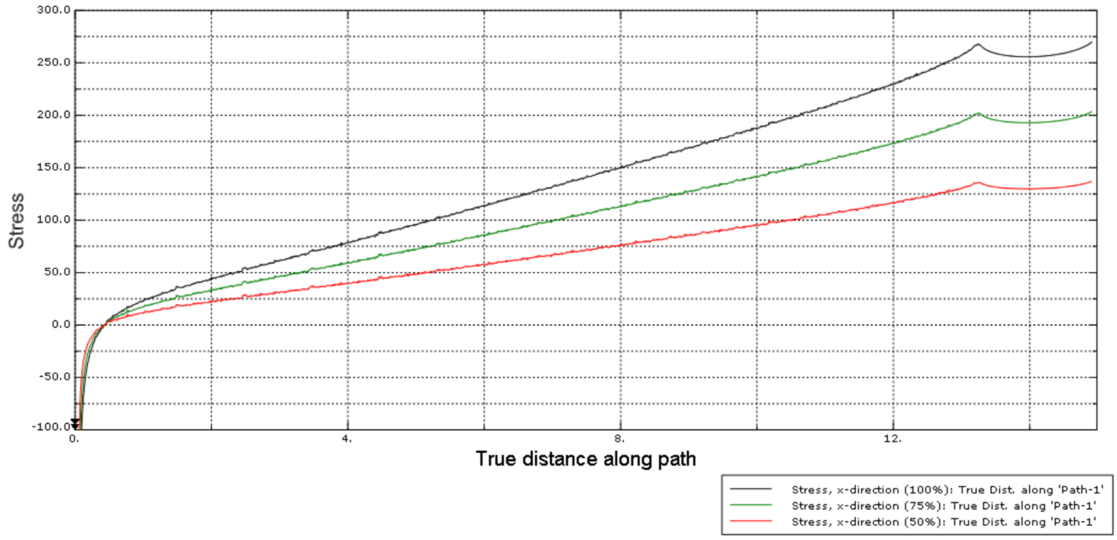

Figure 5.18: Stress in the x-direction along the contact surface

5.3.4 Coefficient of friction 0.9

With a high coefficient of friction like 0.9, the results is expected to show the behaviour of the beginning of the actual fatigue induced by fretting. The stick/slip behaviour could with this high friction begin to undergo sticking.

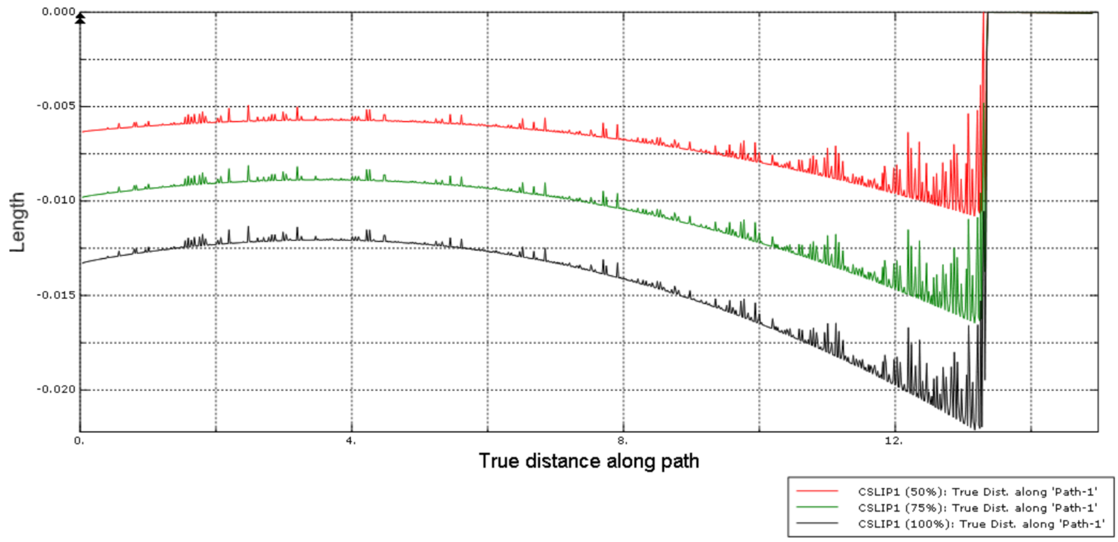

Figure 5.19: Stick/slip 0.9 coefficient of friction

The stick/slip behaviour is as shown in figure 5.19 similar to the analysis with a lower coefficient of friction, but the values are lower, at least for the 50% displacement. It also indicates a slightly lower value closer to the middle of the contact zone, which indicates a possible stick zone. The value of 0.006 mm is in the critical area for slip regarding fretting fatigue, see the theory chapter for a description of the stick/slip behaviour.

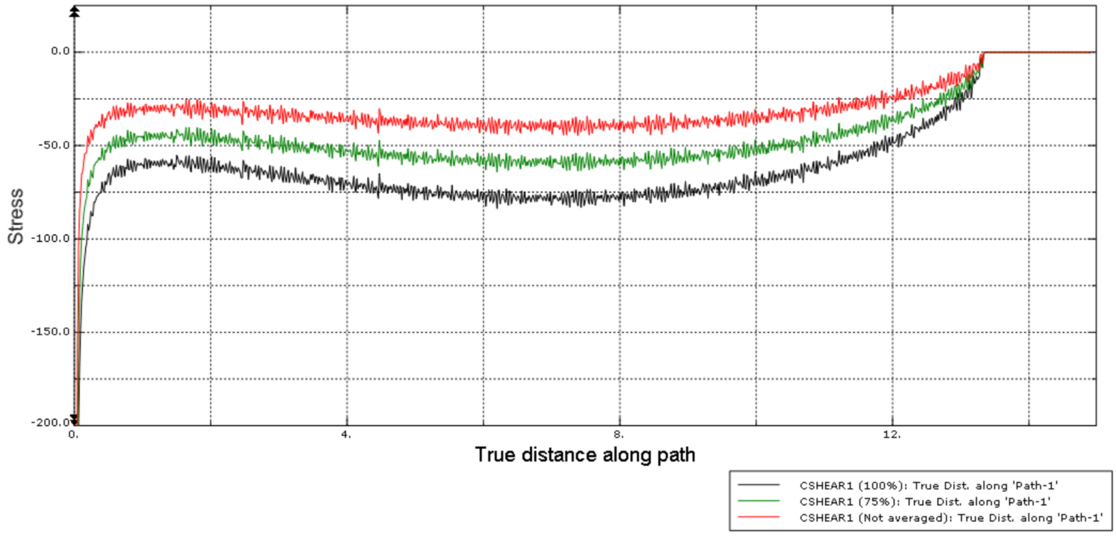

Figure 5.20: Frictional shear stress 0.9 coefficient of friction

The frictional shear stress in figure 5.20 shows a even higher stress than before. This is expected due to the increased friction force. The value of -75 MPa is an increase of 25% from the 0.6 friction, and an increase of 87.5% from the 0.3 friction.

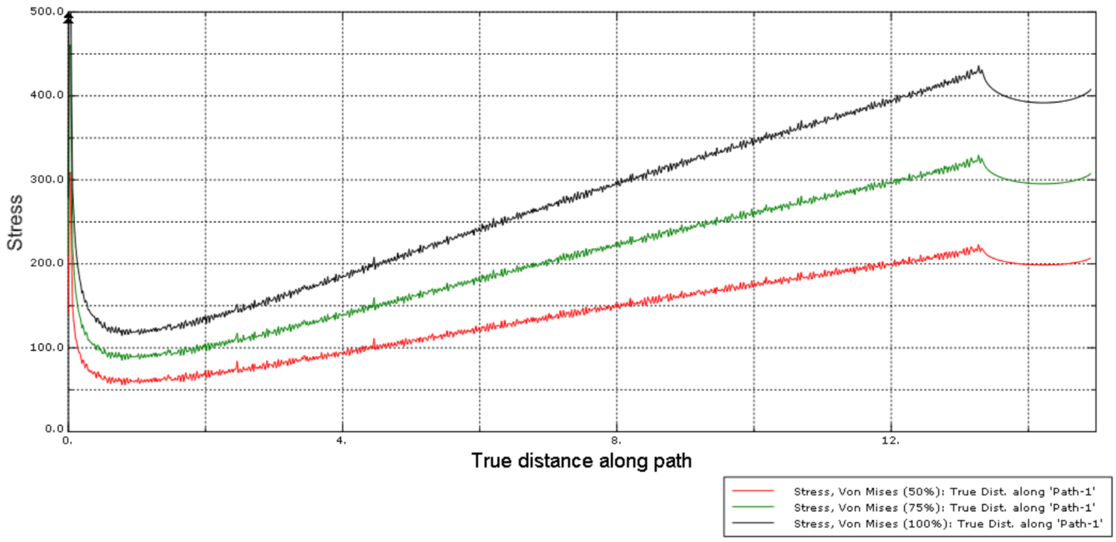

Figure 5.21: Von Mises stress 0.9 coefficient of friction

The Von Mises stress is slightly increased at the end of the contact zone, but it is decreased in the beginning. This is due to the increase in negative shear stress in this zone.

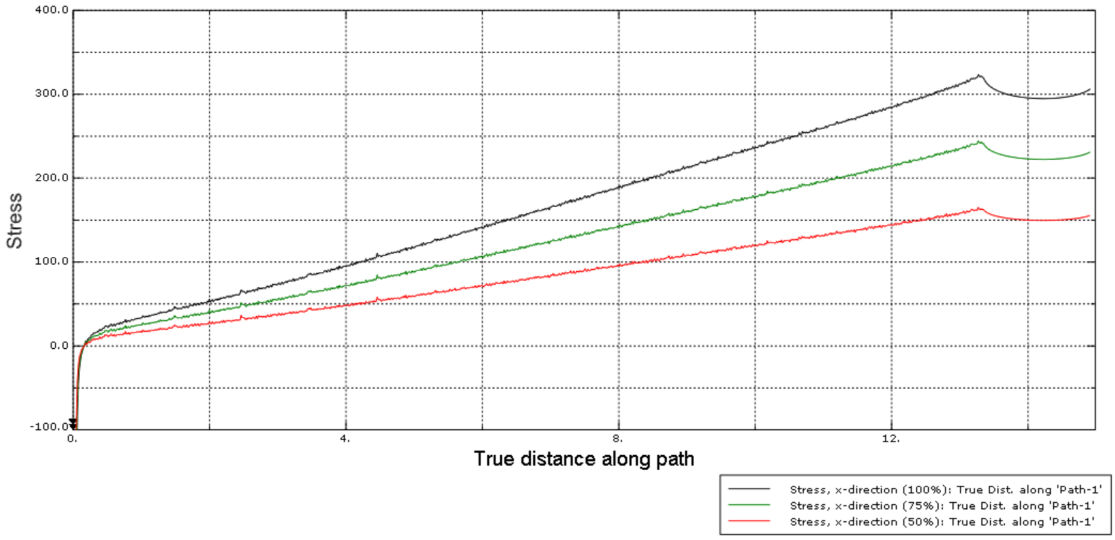

Figure 5.22: Stress in the x-direction along the contact surface, 0.9 coefficient of friction

5.3.5 Geometry with radius

The radius in this analysis has the purpose of comparing the results in this *Dovetail* study with the results from the *Dog-bone* study. Since the radius is equal, hopefully it will be possible to compare, even though the displacement will be different and hence the stress distribution and the stick/slip behaviour will be as well, but some comparing should be possible.

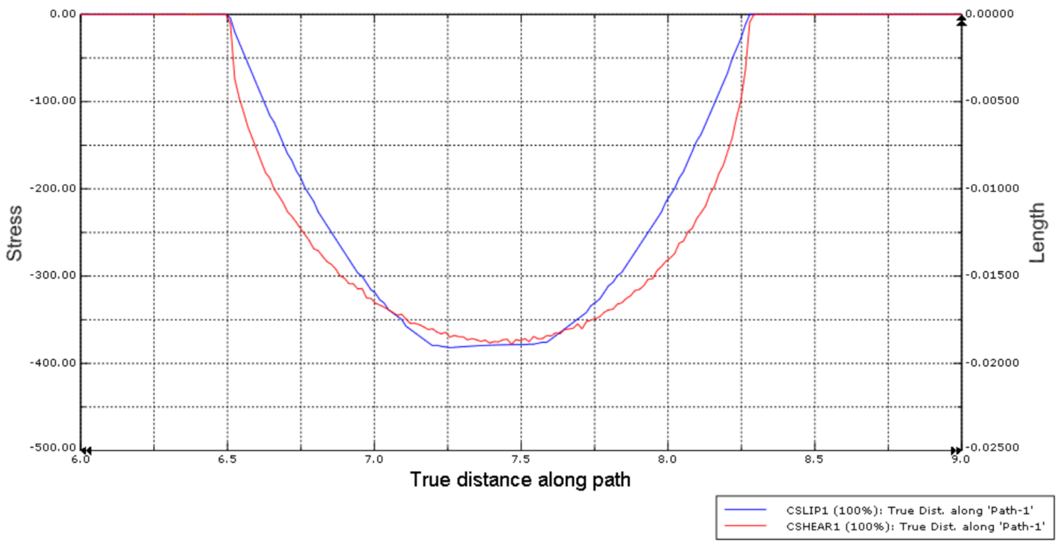

Figure 5.23: Shear stress and slip amplitude

The Stick/slip behaviour versus shear stress with the radius in the *Dovetail* gives a different result, as expected, than the *Dovetail* with two flat surfaces, this can be seen in figure 5.23. Where the red graph is the shear traction along the path and the blue graph is the slip amplitude. The shear stress increases as the slip amplitude increases, and they correlate very well. This is very much similar to the *Dog-bone* when the whole surface undergoes partial slip.

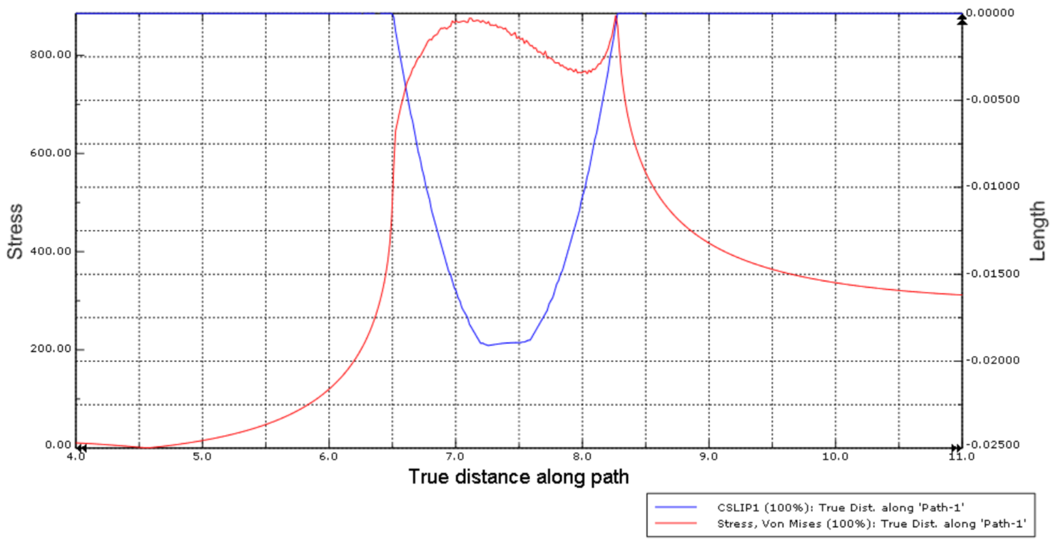

Figure 5.24: Von Mises stress and slip amplitude

The Von Mises stress has a peak where the slip zone ends. The stress here is as high as almost 900 MPa, this is due to the radius and the displacement in y-direction of 0.1 mm will give high stresses when the contact area is this small.

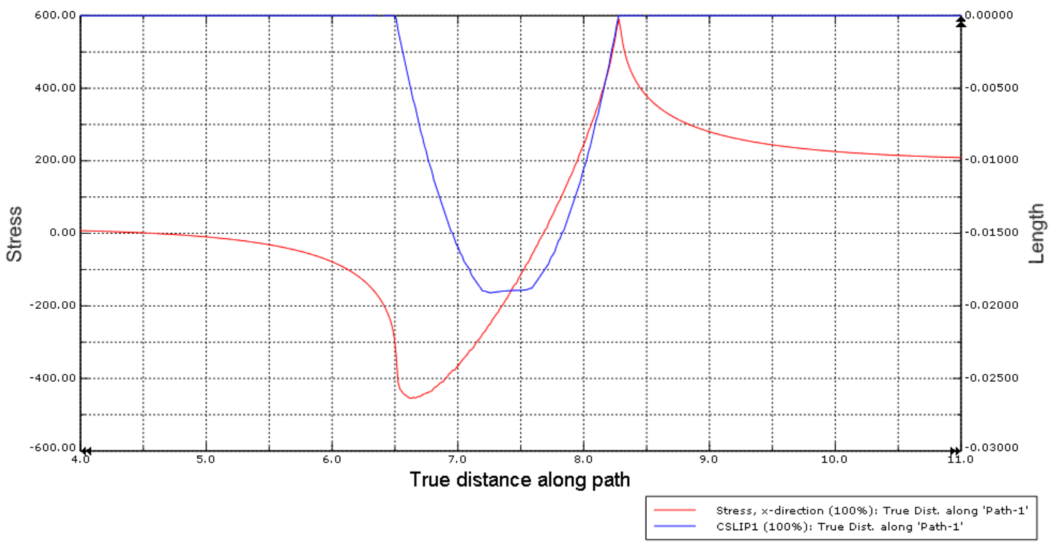

Figure 5.25: Stress in x-direction and slip amplitude

As for the *Dog-bone* study, the stress in x-direction is at its highest at the ends of the contact zone, but with opposite indicators.

5.3.6 Pfat analyses

The post-processing analysis tool *pfat* gave results which indicates a lifetime in the area between 3.9×10^6 and 2.9×10^7 cycles. In the first analysis, the mean load was set to 47 Newton with a amplitude of 100 Newton, which gave the result of 3.9×10^6. The second analysis had a mean load of 47 Newton with a amplitude of 47 Newton. This gave a result of 2.9×10^7. A third analysis was performed with minimum and maximum load, with 0 Newton and 100 Newton respectively. This gave a result of 2.15×10^7 cycles.

6 Discussion

6.1 Dog-bone

The *Dog-bone* study was based on work done previously by other researchers but with slightly different properties regarding load case. The material properties were of aluminium for both the specimen and the fretting pad. This could also have been done with other materials, like titanium or steel. This would have been materials more equal to the ones used in turbine blades and aerospace engines. But for simplicity reasons aluminium were chosen. It was assumed that these properties would be sufficient for the study, and that the stress distribution and the stick/slip behaviour would be very equal regardless of the material properties. It was assumed that the load and displacement and coefficient of friction would be more important regarding especially the stick/slip behaviour, which anyways is the most important result to analyse in fretting fatigue cases. The dimensions are the same as for previous research done by others. This was done in order to have something to compare the results with.

When the analysis began, a *Python* script were made. This was to automate the analysis, with automatic update of the element size around the contact surfaces of both components, and generating text files for the Von Mises stress and the slip amplitude. The *Python* scripting of analysis is clearly a very help-full tool in finite element analysis. In this thesis *Python* could have been used even more. Scripts could have been used for generating plots from *Abaqus*, also for the main study, this would have given many opportunities with modifying graphs and correlate different results with each other. The graphs in the report are sufficient, but with *Python* they could have been even better.

The result from the pre-study was as expected, with converging results as the mesh was refined. The study were made simple, with few results analysed. Only the Stick/slip behaviour at the end of the displacement and the Von Mises stress in y-direction of the specimen were analysed. The reason for this, was that the purpose of the pre-study was to look for convergence in the results, and with the stick/slip behaviour as the most important parameter. The stick/slip is, as described in the theory chapter, very important regarding fretting fatigue. By that it was regarded as sufficient to look at only these two parameters for this study. The results gave a convergence around 0.03mm and 0.015mm. A element size of 0.015mm were therefore chosen, even though the slip amplitude were equal for 0.03mm and 0.015mm, which indicates that the result might have converged at or before 0.03mm. To be sure that the result would be accurate enough, the 0.015mm element size was preferred.

In the main study, the results of slip amplitude and the shear stresses gives good results regarding the stick/slip behaviour. The shear stress follows the slip amplitude the way it is expected to do. Where the shear stress increases significantly when approaching the area where the stick/slip transition zone, the area where the contact surfaces undergoes partial slip. This is something to use in the *Dovetail* study as well. Regarding the different contact formulations and the plain stress instead of plain strain, the results indicates similar stick/slip behaviour. There are some differences especially in the peak value for the slip amplitude. And the plain strain with "Penalty" friction gives the highest values. Therefore this will be the most conservative approach, whether it is the correct formulation or not is hard to tell without testing. Although in the *Abacus user manual* the "Lagrange multiplier" is described as the most accurate one. But as discussed in the *Dovetail* section below, the Lagrange formulation increased the computational cost significantly. And it was regarded as sufficient to use "Penalty", especially when it is the most conservative formulation.

6.2 Dovetail

The *Dovetail* fixing is a frequently used configuration in aerospace industry and in gas turbines, and is therefore interesting to study. The study performed in this thesis is based both on previous work and on the *Dog-bone* study. The material is chosen based on the same arguments as for the previous study, and the material is aluminium here as well. In order to avoid yielding in the material of the disk, the thickness were twice of the turbine blade, 2mm and 1mm respectively. One could of course have two different materials or different E-modulus instead. It is hard to tell what would have given the most correct results, but the difference should not be to significant. There are also other aspects a real life scenario of a rotating engine with many *Dovetail* fixings have. Such as creep, micro vibrations, force in other directions than x and y-directions. This is not given any attention in this thesis, the reason is that one wanted to begin with a simple configuration in order to analyse the stick/slip behaviour with as few unknown as possible.

In the literature the *Dovetail* has been studied with similar contact contact geometry as for the *Dog-bone*, with an hertzian contact. Fretting pads have been introduced in the disk, with a contact radius and by that different contact surfaces than in this thesis. Here it is used a flat contact surface for both the turbine blade and the disk. One could have argued for the use of hertzian contact also, and that it would have been more comparable to the literature. But if looking ahead the configuration chosen for this thesis, the flat contact surfaces, there are a lot of interesting researches that could be done, regarding both finite element analysis and physical tests.

The results of the *Dovetail* study shows that the coefficient of friction is of great importance regarding the stick/slip behaviour. For all analysis the vertical displacement are identical, but the slip amplitude varies with the coefficient of friction. The lower coefficient gives a significantly higher slip amplitude than the higher coefficient. With 0.3 as coefficient the maximum slip amplitude is 0.035mm whilst with 0.9 the maximum slip amplitude is 0.022mm. From the literature we know that as long as the slip amplitude increases towards the threshold value of approximately 0.05mm, the life prediction of the components decreases. On the other hand, the stresses, Von Mises, shear and in x-direction, increases as the coefficient of friction changes from 0.3 up to 0.9, this is expected since the stress induced by the normal force multiplied with the coefficient of friction will increase. Still the slip amplitude have such a significant impact on the lifetime, that it is regarded as more critical. This is most likely due to the high stress gradient induced by the partial slip.

The contact formulation is done in two different ways, one as described with "Lagrange multiplier" and one with "Penalty". As mentioned in the theory chapter, the "Lagrange multiplier" are the most accurate one and are recommended for analyses such as fretting fatigue. One can see the difference in the two analysis in figure 5.9 and figure 5.10 for the "Lagrange multiplier, and figure 5.15 and figure 5.16 for the "Penalty", both with a coefficient of friction 0.6. The graphs for the "Lagrange multiplier" are smoother than for the "Penalty". But the values are very much similar. And the fact that the "Lagrange multiplier" had a simulation time of more than three days, made the decision of which contact formulation to be used easier. The time spent on each simulation would not have justified the better accuracy, at least not for a master thesis. If further work are to be done, this must be considered.

The last analysis were made with a radius for one of the contact surfaces. This was in order to compare it with both the results from the *Dog-bone* study, with a hertzian contact, and to the other *Dovetail* study with two parallel contact surfaces. The results of this configuration gave very different result than for the first configuration. The slip amplitude has a shape that looks more like the for the *Dog-bone* study. The contact configuration is different, hence different slip amplitude, as expected. The slip amplitude has a lower value but the stresses, shear, Von Mises and x-direction, are significantly higher than previous. This is due to the smaller contact area and thereby higher stress concentration. This was done mostly in order to have comparable results, since a actual *Dovetail* fixing has the geometry of the first *Dovetail* study with parallel contact surfaces.

6.3 Correlation

The two different studies, *Dog-bone* and *Dovetail*, are performed with equal parameters regarding the contact formulations. Both with the main study in plane strain conditions and "Penalty" frictional behaviour. The difference between them are the angle versus applied load/displacement. The *Dog-bone* has a force perpendicular to the contact surface of the specimen and a displacement parallel to it, whilst the *Dovetail* has a displacement in the positive vertical direction and contact surfaces 35 deg. to the horizontal line. This gives different stress distributions and different stick/slip behaviour, hence the differences in the graphs presented. The correlation between the results for the two studies are difficult to compare, since the relative displacement between the components in each study are different and the geometries are different.

One interesting correlation or more lack of correlation, is the slip amplitude for the two geometries. Where the *Dog-bone* has some sticking in the contact area especially when the displacement is small, whilst the *Dovetail* seems to have non zones that sticks together. And by that the slip amplitude behaves differently on the two geometries. For both the geometries the slip amplitude increases as it approaches the edge of the contact. This indicates that it will be in this area that a crack will occur after a for now unknown number of cycles. The increase of slip amplitude is as described in the theory chapter, negative for the life of the component, all the time the slip amplitude is below approximately 80μm. This might be the most important correlation to draw between the two analysis, the increase of slip amplitude as it approaches the edge of the contact area.

The last analysis for the *Dovetail*, with Hertzian contact, has a slip amplitude that looks more like the one for *Dog-bone*. At least as long as the fretting-pad has a relative displacement of minimum 50μm. This relation is not of main interest in this thesis because the parallel contact described above is the one that is comparable to a real case in a aeroplane *Dovetail* fixing. Although it could be interesting regarding identical contact but with different load and displacement directions. It needs further studies and ideally testing to compare the two analysis.

6.4 Pfat

The *pfat* analysis gives results that are difficult to say whether they are reliable or not. The challenge with fretting fatigue life prediction, is the high grade of nonlinear behaviour, and since *pfat* is a post-processing tool which support only linear behaviour, it is very difficult to use these results for other than comparing with possible tests in the future. The analyses were also performed with load control instead of displacement control, this was in order to have better control on the applied load. And might therefore be more suitable as results to compare with.

7 | Conclusion

The *Dog-bone* study gives results which correlates well with the literature, with similar graphs regarding the Stick/slip behaviour and the slip amplitude, along with the belonging stress distributions. This indicates that the contact formulations, "Penalty friction", are reliable and that the mesh size is sufficient for this geometry. The results in the *Dog-bone* study gives a good indication that this is the correct formulation to use in the *Dovetail* study as well. In the *Dovetail* study the results are different from the *Dog-bone* regarding the slip amplitude and the stress distribution. The interesting thing to conclude with is that the slip amplitude increases as the edge of the contact approaches. And based on the theory about a slip amplitude in the area between 10μm and 80μm is the worst area regarding fretting fatigue. With a threshold value of 50μm, an increase in the slip amplitude up to the threshold value will give a lower number of cycles for life prediction before crack initiation and brittle fracture occurs.

Further the different coefficient of friction will give a significant difference in the results for the *Dovetail* study. Where a low coefficient of 0.3 will give a increased slip amplitude with belonging decreased stresses. In the other end of the scale, a coefficient of friction of 0.9 gives a decreased slip amplitude and increased stresses. This is as expected, since a higher coefficient gives increased frictional stress and by that less slip and higher stresses. This is the most interesting part in fretting fatigue, where in other areas of fatigue, such as notch fatigue, the higher the stress is the shorter life prediction. Whilst for constructions where there are small finite sliding between the components, the slip amplitude is the critical parameter. At least as long as the stresses are way below critical values such as yield stress and fatigue strength.

When considering these factors, the worst case scenario in this particular case, a low coefficient of friction with its belonging slip amplitude might be the most critical aspect. This is due to the high stress raisers in the area which undergoes partial slip. Based on the analysis and the theory about the mechanics of fretting fatigue, the area in the *Dovetail* fixing which most likely will have a crack initiation is illustrated in figure 7.1. This needs to be studied further and testing should be done to validate this.

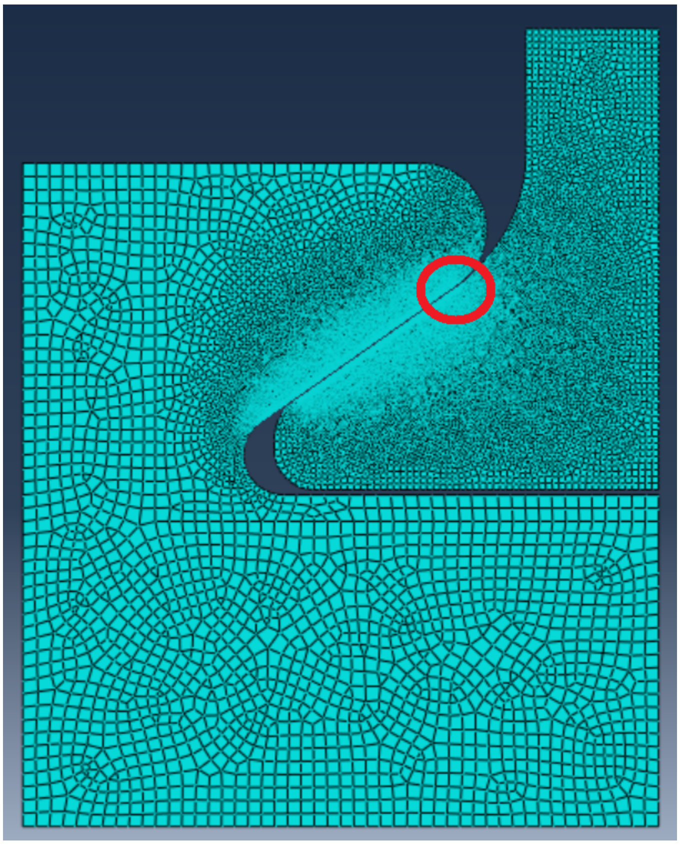

Figure 7.1: Possible crack initiation

To learn more about the stick/slip behaviour for the *Dovetail* geometry was one of the main goals for this study. Further studies needs to be done, but this study shows that there are some correlations between the *Dog-bone* and the *Dovetail* regarding the increase of slip at the edges of the contact surfaces.

The post-processing analysis tool *pfat* gave results which indicates a lifetime in the area between $3.9x10^6$ and $2.9x10^7$ depending on load conditions are not reliable. It could only be concluded that this should be used as a guiding and something to compare eventual tests with. Further research is needed on this topic before any clear conclusions could be drawn.

8 Further work

In this thesis there are unanswered questions, especially regarding lifetime prediction of the Dovetail geometry. A few analyses with *pfat* have been performed, but with the conclusion that they are not reliable. Further work needs to be done in order to validate results from *pfat* and the results of slip amplitude in *Abaqus*. The most important studies to be done in the future, will be testing of the geometry and compare them with analysis in the thesis. And by that further improve the analysis done here. Since these analyses, especially the Dovetail geometry, needs more validation, there are room for improvement in geometric details as well as the contact formulations. And only by comparing with tests these could be improved such that they are more reliable. In addition, the results especially from the *Dog-bone* study could be validated against numerical solutions.

Bibliography

[1] B. Berthel A. de Pannemaecker S. Fouvry and J. Y. Buffiere. "Numerical methods for stress intensity factor ΔK calculations of fretting cracked interface". In: *Tribology International* (2018).

[2] Abaqus/CEA. *Abaqus Analysis User's Guide*. URL: `http://abaqus.software.polimi.it/v6.14/books/usb/default.htm`.

[3] B. P. Conner and T. Nicholas. "Using a Dovetail Fixture to Study Fretting Fatigue and Fretting Palliatives". In: *Journal of Engineering Materials and Technology* (2006).

[4] D. Nowell D. A. Hills and J. J. O'Conner. "On the Mechanics of Fretting Fatigue". In: *Elsevier Sequoia* (1988).

[5] D. Dini D. Nowell and D. A. Nowell. "Recent development in the understanding of fretting fatigue". In: *Science Direct* (2005).

[6] Norman E. Dowling. *Mechanical Behaviour of Materials - Engineering Methods for Deformation, Fracture and Fatigue*. Fourth edition. Pearson Education Limited, 2013, pp. 491–495. ISBN: 978-0-273-76455-7.

[7] P. J. Golden and T. Nicholas. "The effect of angle on dovetail fretting experiments in Ti-6Al-4V". In: *N/A* (2005).

[8] D. A. Hills. "Mechanics of fretting fatigue". In: *Elsevier Science S.A.* (1993).

[9] D. A. Hills and D. Nowell. *Mechanics of Fretting Fatigue*. First edition. Springer-science+Business Media, B.V., 1994, p. 53. ISBN: 978-94-015-8281-0.

[10] D. A. Hills and D. Nowell. *Mechanics of Fretting Fatigue*. First edition. Springer-science+Business Media, B.V., 1994, p. 55. ISBN: 978-94-015-8281-0.

[11] D. A. Hills and D. Nowell. "Mechanics of fretting fatigue - Oxford's contribution". In: *Tribology International* (2013).

[12] ASM Aerospace Specification Metals. *Data sheet*. URL: `http://asm.matweb.com/search/SpecificMaterial.asp?bassnum=ma2024t4`.

[13] R. Rajasekaran and D. Nowell. “Fretting fatigue in dovetail blade roots: Experiment and analysis”. In: *Science Direct* (2006).

[14] Ph.D T. L. Anderson. *Fracture Mechanics - fundamentals and applications*. Third edition. Taylor and Francis, 2005, pp. 453, 454. ISBN: 978-0-8493-1656-2.

[15] Olof Vingsbo and Steffan Soderberg. “On Fretting Maps”. In: *Elsevier Sequoia* (1987).