MASTER

Biaxial stresses in steel fibre reinforced concrete
modelling the pull out-behaviour of a single steel fibre using FEM

van der Aa, P.J.

Award date:
2014

Link to publication

Disclaimer
This document contains a student thesis (bachelor's or master's), as authored by a student at Eindhoven University of Technology. Student theses are made available in the TU/e repository upon obtaining the required degree. The grade received is not published on the document as presented in the repository. The required complexity or quality of research of student theses may vary by program, and the required minimum study period may vary in duration.

General rights
Copyright and moral rights for the publications made accessible in the public portal are retained by the authors and/or other copyright owners and it is a condition of accessing publications that users recognise and abide by the legal requirements associated with these rights.

• Users may download and print one copy of any publication from the public portal for the purpose of private study or research.
• You may not further distribute the material or use it for any profit-making activity or commercial gain
Biaxial Stresses in Steel Fibre Reinforced Concrete

Modelling the Pull-Out Behaviour of a single Steel Fibre using FEM

P.J. van der Aa
January 2014
A-2014.61
Biaxial Stresses in Steel Fibre Reinforced Concrete
Modelling the Pull-Out Behaviour of a single Steel Fibre using FEM

Final Version
29 - 01 - 2014
A-2014.61

Name: PJ (Pim) van der Aa
Student number: 0636279
E-mail address: p.j.v.d.aa@student.tue.nl
                        pimvanderaa@hotmail.com
Address: De Sitterlaan 56
Zip code: 5505 AD Veldhoven

Chairman/Supervisor: Prof. Dr. Ir. T.A.M. Salet
Chair: Material related Structural Design - Concrete Structures
Position: Full Professor - Concrete Structures

Supervisor: Prof. Dr. Ir. A.S.J. Suiker
Chair: Applied Mechanics and Design
Position: Full Professor - Mechanics

Supervisor: Ir. F.J.M. Luijten
Chair: Material related Structural Design - Concrete Structures
Position: Assistant Professor

Supervisor: Ing. A. Hoekstra
Company: NV Bekaert SA
Position: Technical Manager Building Products
PREFACE

In this study, the behaviour of steel fibre reinforced concrete under biaxial stresses is investigated. A finite element model has been developed, which describes the pull-out behaviour of a single steel fibre. The finite element model has been compared to experimental research.

This graduation project has been carried out as a part of the master Architecture, Building and Planning with the specialization Structural Design. It has been performed at the department of Built Environment at Eindhoven University of Technology, the Netherlands.

I would like to thank Sven van den Bulck and the personnel of the Pieter van Musschenbroek Laboratory (TU/e) for their support concerning the experimental research.

Finally, I would like to take my family and friends for their support. In particular, I would like to thank my parents and my brother for their interest and support during good and bad times throughout the whole study.

Pim van der Aa
Eindhoven, January 2014
Steel fibres are often added to concrete to improve the overall material performance. Typical test results of Steel Fibre Reinforced Concrete (SFRC) show no significant improvement in tensile strength in comparison to plain concrete. However, major improvements in ductility are witnessed. The fibres become active after cracking of the concrete. Therefore, the fibres contribute to the post-cracking behaviour of SFRC by bridging the crack and providing resistance to the crack opening.

Testing and calculation methods for SFRC can be found in building codes and the scientific literature. These methods focus upon the uniaxial material behaviour of SFRC, and therefore it is unclear whether these methods also are applicable for investigating the biaxial material behaviour. Multiple experimental researches have been performed on biaxial stresses in SFRC. These researches show conflicting results. The research performed by Tschegg (2009), who carried out biaxial splitting test, showed that the ductile behaviour of SFRC almost completely disapears in a biaxial stress situation. This is in contrast to other studies on biaxial stresses in SFRC, which show an increase in ductility. More research on the effect of the biaxial stress in SFRC is needed in order to better understand biaxial material behaviour.

In order to gain insight in the material characteristics of SFRC, the pull-out behaviour of a single steel fibre is investigated. Two types of fibres are treated: a straight fibre and a hooked-end fibre. The pull-out behaviour of a straight fibre is governed by three characteristics: adhesive bonding, debonding and friction. All these characteristics relate to the fibre-matrix interface. The pull-out behaviour of a hooked-end fibre is governed by adhesive bonding, debonding and friction at the interface (similar to the straight fibre), as well as the nonlinear deformation behaviour of the fibre and the matrix.

A finite element model of the pull-out behaviour of a fibre is made with Abaqus. All previously mentioned phenomena, which determine the pull-out behaviour, should be included in the FEM model. Therefore, the fibre-matrix interface of the FEM model has to be able to model adhesive bonding, debonding and friction. This is possible with a surface-based contact simulation, which uses surface properties to model the behaviour at the fibre-matrix interface. The non-linear behaviour of the steel is modelled using a Plasticity model and concrete is modelled using a Concrete Damaged Plasticity model.

The FEM model is used to numerically simulate a pull-out test for a straight fibre and a hooked-end fibre. Also, the influence of imperfections on the pull-out behaviour for the straight fibre is investigated with the FEM model. It is shown that imperfections could lead to an increase of the frictional resistance of the pull-out behaviour. Furthermore, the influence of lateral normal stresses on the pull-out behaviour for the straight fibre is investigated with the FEM model. In this manner, a biaxial stress situation in SFRC is simulated, which gains insight in the biaxial material behaviour of SFRC. It is shown that lateral normal stresses lead to an increase of the pull-out force.
Experimental research has been performed to qualitatively verify the FEM model. Pull-out tests have been performed with straight and hooked-end fibres, with or without the addition of lateral loading.

The experimental results of the pull-out tests without lateral loading for both fibre types are in good correspondence with the numerical results. The experimental results of the pull-out tests with lateral loading for the straight fibre show similar pull-out behaviour compared to the numerical results. However, two differences are observed: The change of elastic stiffness and the change of debonding energy in the experiments do not correspond with the numerical model.
# TABLE OF CONTENTS

PREFACE ................................................................................................................................. i

ABSTRACT .............................................................................................................................. iii

TABLE OF CONTENTS ........................................................................................................... v

NOTATIONS ........................................................................................................................... vii

CHAPTER 1: INTRODUCTION ............................................................................................... 1

CHAPTER 2: PULL-OUT BEHAVIOUR .................................................................................. 5
  2.1 Straight fibre .................................................................................................................. 5
  2.2 Hooked-end fibre ......................................................................................................... 6

CHAPTER 3: FEM MODEL .................................................................................................. 7
  3.1 Fibre-matrix interface ............................................................................................... 7
  3.1.1 Adhesive bond ..................................................................................................... 7
  3.1.2 Debonding ........................................................................................................ 11
  3.1.3 Friction ............................................................................................................. 18
  3.1.4 Combining debonding and friction ..................................................................... 23
  3.2 Fibre .......................................................................................................................... 24
  3.2.1 Plasticity ........................................................................................................... 24
  3.2.2 Parameters for plasticity ..................................................................................... 26
  3.3 Matrix ........................................................................................................................ 27
  3.3.1 Concrete Damaged Plasticity ........................................................................... 27
  3.3.2 Parameters for CDP ........................................................................................... 30
  3.3.3 Shrinkage ......................................................................................................... 31

CHAPTER 4: STRAIGHT FIBRE ......................................................................................... 33
  4.1 Pull-out test ................................................................................................................ 33
  4.1.1 FEM model ................................................................................................------- 33
  4.1.2 Mesh study ......................................................................................................... 34
  4.1.3 Discussion of the results ...................................................................................... 35
  4.2 The influence of imperfections on the pull-out behaviour ....................................... 37
  4.2.1 2D FEM analysis .............................................................................................. 37
  4.2.1 3D FEM analysis ............................................................................................ 43
  4.3 The influence of lateral normal stress on the pull-out behaviour ............................... 45
  4.3.1 FEM model ...................................................................................................... 45
  4.3.2 Mesh study ....................................................................................................... 46
  4.3.3 Discussion of the results .................................................................................... 47

CHAPTER 5: HOOKED-END FIBRE ................................................................................. 51
  5.1 Pull-out test ................................................................................................................ 51
  5.1.1 FEM model ....................................................................................................... 51
  5.1.2 Mesh study ....................................................................................................... 52
  5.1.3 Discussion of the results .................................................................................... 53
<table>
<thead>
<tr>
<th>CHAPTER 6: EXPERIMENTAL RESEARCH</th>
<th>................................................................. 59</th>
</tr>
</thead>
<tbody>
<tr>
<td>6.1 Experimental program</td>
<td>.......................................................................... 59</td>
</tr>
<tr>
<td>6.1.1 Purpose</td>
<td>.......................................................................... 59</td>
</tr>
<tr>
<td>6.1.2 Test specimen</td>
<td>.......................................................................... 59</td>
</tr>
<tr>
<td>6.1.3 Test setup</td>
<td>.......................................................................... 60</td>
</tr>
<tr>
<td>6.1.4 Testing procedure</td>
<td>.......................................................................... 61</td>
</tr>
<tr>
<td>6.2 Experimental results</td>
<td>.......................................................................... 62</td>
</tr>
<tr>
<td>6.2.1 Straight fibre</td>
<td>.......................................................................... 62</td>
</tr>
<tr>
<td>6.2.2 Hooked-end fibre</td>
<td>.......................................................................... 63</td>
</tr>
<tr>
<td>6.3 Conclusions</td>
<td>.......................................................................... 65</td>
</tr>
<tr>
<td>CHAPTER 7: FEM vs. EXPERIMENTS</td>
<td>.......................................................................... 67</td>
</tr>
<tr>
<td>7.1 Straight fibre</td>
<td>.......................................................................... 67</td>
</tr>
<tr>
<td>7.1.1 Pull-out test</td>
<td>.......................................................................... 67</td>
</tr>
<tr>
<td>7.1.2 The influence of lateral forces on the pull-out behaviour</td>
<td>68</td>
</tr>
<tr>
<td>7.2 Hooked-end fibre</td>
<td>.......................................................................... 69</td>
</tr>
<tr>
<td>7.2.1 Pull-out test</td>
<td>.......................................................................... 69</td>
</tr>
<tr>
<td>CHAPTER 8: CONCLUSIONS &amp; RECOMMENDATIONS</td>
<td>.......................................................................... 71</td>
</tr>
<tr>
<td>8.1 Conclusions</td>
<td>.......................................................................... 71</td>
</tr>
<tr>
<td>8.2 Recommendations</td>
<td>.......................................................................... 73</td>
</tr>
<tr>
<td>BIBLIOGRAPHY</td>
<td>.......................................................................... 75</td>
</tr>
<tr>
<td>APPENDIX</td>
<td>.......................................................................... 79</td>
</tr>
<tr>
<td>Appendix A: Bruggeling’s tensile bar model</td>
<td>.......................................................................... 79</td>
</tr>
<tr>
<td>Appendix B: Extension of Bruggeling’s tensile bar model</td>
<td>.......................................................................... 82</td>
</tr>
<tr>
<td>Appendix C: Input files Abaqus</td>
<td>.......................................................................... 84</td>
</tr>
</tbody>
</table>
NOTATIONS

Symbols

Greek Symbols
\( \alpha \) = coefficient of expansion \[ - \]
\( \alpha \) = rate of damage evolution \[ - \]
\( \alpha \) = pull-out angle \[ ° \]
\( \delta \) = separation \[ mm \]
\( \delta \) = displacement/slip \[ mm \]
\( \varepsilon \) = strain \[ - \]
\( \theta \) = temperature \[ K \]
\( \lambda \) = multiplier \[ - \]
\( \mu \) = coefficient of friction \[ - \]
\( \sigma \) = stress \[ N/mm^2 \]
\( \tau \) = shear stress \[ N/mm^2 \]
\( \psi \) = perimeter \[ mm \]
\( \psi \) = dilatation angle \[ ° \]

Latin Symbols
\( A \) = area \[ mm^2 \]
\( D \) = damage variable \[ - \]
\( D \) = stiffness \[ N/mm^2 \]
\( E \) = Young’s modulus \[ N/mm^2 \]
\( F \) = force \[ N \]
\( F \) = yield function \[ N/mm^2 \]
\( G \) = fracture energy \[ N/mm \]
\( G \) = plastic potential \[ N/mm^2 \]
\( K \) = elastic stiffness \[ N/mm^2 \]
\( N \) = force \[ N \]
\( a \) = distance between force introduction and support \[ mm \]
\( d \) = internal lever arm \[ mm \]
\( d \) = stiffness degradation variable \[ - \]
\( d \) = detection distance \[ mm \]
\( f \) = strength \[ N/mm^2 \]
\( h \) = overclosure distance \[ mm \]
\( k \) = elastic stiffness \[ N/mm^2 \]
\( l \) = length \[ mm \]
\( p \) = pressure \[ N/mm^2 \]
\( t \) = traction \[ N/mm^2 \]
\( v \) = Poisson’s ratio \[ - \]
\( x \) = local position \[ mm \]
Indices

$X_1$ = first principal direction
$X_2$ = second principal direction
$X_3$ = third principal direction
$X_c$ = concrete
$X_{cr}$ = compression
$X_{el}$ = elastic
$X_f$ = fibre
$X_f$ = fracture
$X_m$ = mixed-mode direction
$X_{max}$ = maximum
$X_{min}$ = minimum
$X_n$ = normal direction
$X_{pl}$ = plastic
$X_s$ = steel
$X_s$ = first shear direction
$X_{shr}$ = shrinkage
$X_t$ = second shear direction
$X_t$ = tension
$X_y$ = yield

Abbreviations

CDP: Concrete Damaged Plasticity
CMOD: Crack Mouth Opening Displacement
FEM: Finite Element Method
HF: Hooked-end Fibre
LVDT: Linear Voltage-Displacement Transducer
SFRC: Steel Fibre Reinforced Concrete
SF: Straight Fibre
CHAPTER 1: 
INTRODUCTION

This chapter describes the need to perform an investigation on the effect of biaxial stresses in Steel Fibre Reinforced Concrete (SFRC). A research proposal has been set up for this investigation.

Concrete has a relatively low tensile strength. Steel bars are added to the concrete in order to increase the tensile strength, which is what characterizes reinforced concrete. These reinforcement bars are placed inside the mould and subsequently the mould is filled with concrete. Placing the reinforcement is very labour-intensive work. SFRC is a relative new alternative for reinforced concrete. SFRC is a composite material, which consists of a concrete matrix complemented with randomly oriented and distributed steel fibres. SFRC is manufactured by adding steel fibres to the concrete mixture, which reduces the labour intensity in comparison to the use of reinforcement bars.

Testing and calculation methods (RILEM TC 162-TDF 2002 and CUR 111) for SFRC have been developed, which focus on bending in structural elements. Typical test results of these methods show no significant improvement in bending strength compared to plain concrete. However, major improvements in ductility are witnessed (Fig. 1.1). This can be explained by the fact that concrete needs to crack in order for the fibres to become active. When cracking occurs, the fibres contribute to the post-cracking behaviour of SFRC by bridging the cracks and providing resistance to crack opening.

Fig. 1.1 - Schematic view of the behaviour of plain concrete (a) and SFRC (b) in a bending test

Fig. 1.2 shows a four point bending test. The stresses within the beam have been drawn schematically. The beam is divided into a mid section (Beam zone) and two side sections (Disturbed zone). In the Beam zone (B-zone) there is a linear variation in strain over the section of the beam, which leads to a uniaxial stress situation. In the Disturbed zone (D-zone) there is a complex variation in strain over the section of the beam, which leads to a biaxial stress situation. Whether failure occurs in the B-zone or in the D-zone of the concrete beam depends on the ratio a/d.
As mentioned before, the testing and calculation methods of SFRC focus on bending, which means these methods relate to the B-zone. It is unclear whether these methods are also applicable for the D-zone.

Multiple experimental researches (Tschegg, 2009; Demeke and Tegos, 1994; Kragh-Poulsen et al., 2010; Narayanan and Kareem-Palanjian, 1983) have been performed on biaxial stresses (D-zone) in SFRC. These researches show conflicting results, and clear explanations or theories for the difference in behaviour are seldomly presented.

E.K. Tschegg (2009) carried out experimental research using a biaxial splitting test, which showed two worrying results. Firstly, by introducing a compression force perpendicular to the tensile force, the maximum tensile strength of SFRC showed a 10% decrease in comparison to normal concrete (Fig. 1.3a). Secondly, the ductile behaviour of SFRC almost disappears for the most part, in case a normalized compression force from 15% is introduced (Fig. 1.3b). The main benefit of SFRC is the improvement of the post-cracking behaviour. According to Tschegg’s research this benefit would disappear in case SFRC is loaded biaxially.

In contrast, Demeke and Tegos (1994) show an increase of the maximum tensile strength in case a compression force perpendicular to the tensile force is introduced. The article does not mention anything about a change in post-cracking behaviour. Experimental research about shear failure (Kragh-Poulsen et al., 2010) and torsion failure (Narayanan and Kareem-Palanjian, 1983) shows an
increase in the deformation capacity and a more ductile failure mode, which indicates an increase in the maximum tensile strength and an improvement of the post-cracking behaviour.

In order to better understand these controversial experimental results, the research objective of the present work is:

“To gain insight in the material behaviour of steel fibre reinforced concrete under biaxial stress conditions”

In order to fulfill the research objective three research questions have been formulated:

- How to model the pull-out behaviour of a fibre using FEM?
- What is the influence of lateral normal stresses on the pull-out behaviour of a fibre?
- Is the uniaxial material behaviour that follows from CUR 111 or RILEM TC 162-TDF 2002 valid for structural elements that will fail on a biaxial stress combination?

For gaining insight in material behaviour of SFRC, the pull-out behaviour of a single steel fibre is investigated. The pull-out behaviour can be measured from pull-out tests, where a single fibre is pulled out of a matrix. By adding a lateral load on the matrix during the pull-out of a fibre, the system can be subjected to biaxial stress conditions. This may lead to insight on the material behaviour of SFRC under biaxial stresses. The pull-out behaviour is simulated with a FEM model and is validated with experiments. The focus of this research is on the pull-out behaviour of straight fibres. However it will be shown that the FEM model is also applicable for the study of hooked-end fibres.

The thesis is divided into 8 chapters. The pull-out behaviour of a straight fibre and a hooked-end fibre is discussed in chapter 2. All the phenomena, which determine the pull-out behaviour, are discussed.

Chapter 3 discusses the FEM model. In a typical pull-out problem three components have to be considered: the fibre, the matrix and the fibre-matrix interface. First, the fibre-matrix interface is discussed. The behaviour of the fibre-matrix interface is governed by adhesive bond, debonding and friction. Subsequently, the nonlinear material models of the fibre and the matrix are discussed.

Chapter 4 discusses the results of the numerical model for the pull-out of a straight fibre. The first part treats the results of a standard pull-out test, the second part shows the influence of imperfections on the pull-out behaviour and the last part discusses the influence of lateral forces on the pull-out behaviour. Chapter 5 treats the results of the numerical model for a pull-out of a hooked-end fibre. A typical pull-out test is modelled in 2D.

Experimental research has been done in the Pieter van Musschenbroek Laboratory (TU/e) on the pull-out behaviour of fibres as support for this research. Pull-out tests have been performed with and without the addition of lateral loading. Chapter 6 discusses the experimental program and the test results. Chapter 7 provides a qualitative comparison between the FEM results and the experimental results. Chapter 8 presents the conclusions and recommendations for further research.
CHAPTER 2: PULL-OUT BEHAVIOUR

In this chapter the pull-out behaviour of a straight fibre and a hooked-end fibre are discussed. All phenomena, determining the pull-out behaviour of a single, are treated in detail.

2.1 Straight fibre

The pull-out process of a straight fibre can be divided into three steps (See Fig. 2.1). The first step considers the stresses in the fibre and the concrete matrix due to the adhesive bond (0-A). This step is fully elastic. The second step describes the debonding of the fibre (A-B). At point A the maximum shear stress is reached at the top of the interface (between fibre and matrix) and the fibre starts to debond. As the slip keeps increasing, the fibre debonds from the top of the interface to the bottom of the interface (point B). As the fibre starts to debond, the stresses due to friction start to increase until the fibre is fully debonded. The final step (B-F) considers the pull-out of the fully debonded fibre. The remaining fibre pull-out occurs under frictional slip.

The area under the load-displacement curve is equal to the total energy dissipated during the pull-out process (see Fig. 2.1b). The pull-out energy (both debonding and friction) is directly related to the embedded fibre length, up to the length at which the pull-out load becomes higher than the maximum fibre strength and rupture of the fibre will occur. (Löfgren, 2005)

Fig. 2.1c shows the global pull-out curve in which a debonding and friction contribution can be distinguished. During the debonding of the fibre, the frictional stress increases, which leads to a local shear stress characterized by debonding and friction. The size of both contributions is unknown, because only the combined response can be measured in a pull-out test. It is complicated to differentiate between both phenomena.

In order to correctly simulate a pull-out test in a computational analysis, it is important that adhesive bonding, debonding and friction are included properly in the model.
2.2 Hooked-end fibre

Fig. 2.2 shows the pull-out process of the hooked-end fibre. The first step (0-A) describes the adhesive bonding between the fibre and the matrix, which is similar to that of the straight fibre. The second step (A-B) describes the debonding of the fibre. In contrast to the straight fibre, the pull-out load increases until point C, due to mechanical anchorage of the end hook of the fibre. At point C the bottom of the fibre passes the first corner in the surrounding concrete matrix. Plastic deformation of the fibre has already started. From C-E the fibre heavily deforms until the end hook of the fibre is straight. Between D and E an increase of the pull-out load can be observed which corresponds to the fibre bottom passing the second corner in the surrounding concrete matrix. From E to F the fibre is pulled out of the concrete matrix under frictional resistance. The hooked-end fibre is not completely straight, which leads to a higher frictional resistance than for the straight fibre (Cunha et al., 2007).

![Fig. 2.2 - Typical pull-out behaviour for a straight and a hooked-end fibre (Löfgren, 2005)](image)

For hooked-end fibres it is important to adjust the size of the end hook to the typical steel behaviour and concrete behaviour in order to achieve an optimal pull-out behaviour. For steel the strength and the deformation behaviour (plastic deformation) are important to prevent rupture of the fibre. For concrete the strength of the concrete is important to avoid failure of the concrete matrix.

The overall pull-out behaviour of a hooked-end fibre is fairly similar to the pull-out behaviour of a straight fibre. The pull-out behaviour for a hooked-end fibre is also described by adhesive bonding, debonding and friction at the fibre-matrix interface. However, due to nonlinear deformation of the end hook and the concrete matrix local differences in the force-displacement curve are observed.
CHAPTER 3: FEM MODEL

This chapter discusses the FEM model developed for simulating the pull-out behaviour. In a typical pull-out problem three components have to be considered: the fibre, the matrix and the fibre-matrix interface (Fig. 3.1). First, the behaviour of the fibre-matrix interface is discussed, which is governed by adhesive bonding, debonding and friction (see ch. 2). Subsequently, the nonlinear material models of the fibre and the matrix are discussed. The numerical model is developed within the FEM program Abaqus.

Fig. 3.1 - Three components of a typical pull-out problem

3.1 Fibre-matrix interface

3.1.1 Adhesive bond

The first step of the pull-out behaviour is determined by the adhesive bond. Adhesion is characterized as the bonding between two different materials. The pulling of a fibre out of a matrix can be modelled as the pulling of a rod out of a solid. (Fig. 3.2)

Fig. 3.2 - Schematic stress distribution in a pull-out problem

During pull-out, shear stresses are transferred from the rod to the solid through the interfacial bond. These shear stresses lead to stresses in the solid. The stress distributions in the rod, solid and interface depend on the relationship between the shear stress and the relative slip. This is called the bond-slip behaviour \((\tau - \delta)\), which is governed by the strength and stiffness of the interface.
Modelling adhesive bond

In a reinforced concrete structure cracking occurs. Nearby these cracks stresses are transferred from the concrete to the reinforcement bar within the crack (Fig. 3.3). This mechanism is similar as described before for the pull-out problem. Extensive research has been performed on adhesive bonding in reinforced concrete. Multiple analytical and numerical models (Noakowski, 1978; Tassios and Yannopoulos, 1981; Bruggeling, 1986) have been developed to calculate the local stress distributions near reinforcement bars. Also for the pull-out of a steel fibre analytical models have been developed to calculate the stress distribution (Lawrence, 1972; Naaman, 1989). These calculation methods for reinforced concrete and the steel fibre are all based on the same principles, as described below.

In order to calculate the stresses near the fibre a bond-slip relation \((\tau - \delta)\) between fibre and matrix has to be assumed. This relation is used to calculate the shear stress in the interface.

![Fig. 3.3 - Schematic distributions of stresses and slip along the cracked element (Tassios and Yannopoulos, 1981)](image)

FEM model

Before the bond-slip behaviour can be modelled with FEM, first an interface element/simulation for the FEM has to be selected. This is done in the next section.

Fibre-matrix interface

As discussed earlier, the bond-slip of a fibre-matrix interface can be compared with the bond-slip of a reinforcement bar in concrete. Various elements for modelling the bond-slip of reinforced concrete have been proposed in literature (Casanova et al., 2012). The most common elements are the spring, the interface/joint and the embedded element (see Fig. 3.4). Spring elements have originally been proposed by Ngo and Scordelis (1967) using a linear constitutive law. Nowadays it is more common to use non-linear spring elements (Davalos, 2008). Joint/interface elements (with and without thickness) have been proposed with a nonlinear constitutive law to model the adhesive bond of the
interface (Lowes et al., 2004). In addition, embedded elements have been developed, which combine the material behaviour and the bond effects in one element (Monti et al. 1997).

Another way to model bond-slip is through the use of a surface-based contact simulation (Abaqus, 2011). This simulation uses master and slave surfaces to model the behaviour in the interface. (See Fig. 3.5).

In case of selecting a proper model to model bond-slip in the interface, one should keep in mind the fact that the model should be able to mimic adhesive bonding, debonding and friction at the interface.

This is not possible using embedded elements, because these elements cannot model all three phenomena. For springs it is possible to model all the three mechanisms, by combining different springs. Modelling adhesive bonding and debonding can be accomplished using nonlinear springs (Liu et al., 2012) Modelling friction for large deformations using springs leads to inaccurate results, because of the excessive distortion of the spring element (Huang et al., 2006). Joint/interface elements are suitable for modelling adhesive bond and debonding. They are widely applied in fracture mechanics (Cerioni, 2008; Alfaro et al., 2010). The combination of the three mechanisms within a single element is not possible in most commercial FEM packages, which can be solved by either combining multiple elements (cohesive elements with contact or gap elements) or implementing this type of element as a ‘user-supplied subroutine’ within a commercial FEM package. A surface-based contact formulation has the possibility to combine the three mechanisms in one surface. This option is available in the FEM program Abaqus, which simplifies the modelling procedure. Accordingly, a surface-based contact simulation is chosen in this thesis work.

Fig. 3.4 - Bond elements (Casanova et al., 2012)

Fig. 3.5 - Surface-based contact simulation
**Adhesive bond in FEM model**

Under uniaxial conditions, the traction-separation behaviour is described by equation 3.1, where shear stress $t$ equals the relative displacement (slip) $\delta$ times the elastic stiffness $K$,

$$ t = K\delta $$  \hspace{1cm} (3.1)

Under multi-axial conditions, this behaviour needs to be extended to equation 3.2. The elastic stiffness $K$ relates the traction $t$ (stress in the interface) to the separation $\delta$ (relative displacement between the two opposite faces of the interface). The traction-separation relation is described in terms of three components (normal direction $n$ and two shear directions $s$ and $t$).

$$ t = \begin{bmatrix} t_n \\ t_s \\ t_t \end{bmatrix} = \begin{bmatrix} K_{nn} & K_{ns} & K_{nt} \\ K_{ns} & K_{ss} & K_{st} \\ K_{nt} & K_{st} & K_{tt} \end{bmatrix} \begin{bmatrix} \delta_n \\ \delta_s \\ \delta_t \end{bmatrix} = \mathbf{K} \delta $$ \hspace{1cm} (3.2)

**Verification of the FEM model**

In order to verify the bond-slip relation in the FEM model, a comparison with Bruggeling's tensile bar model (Bruggeling, 1986) is made (more information about Bruggeling's model can be found in appendix A). A simple axisymmetric FEM model (Fig. 3.6) is used. The concrete deformation is not taken into account in order to simplify the comparison. Therefore the young's modulus of the concrete matrix is assumed to be infinitely large. The interface has a length $l_f = 30$ mm and the fibre diameter $D_f = 0.9$ mm. The fibre is loaded with a stress $\sigma_p = 100$ N/mm$^2$.

![Fig. 3.6 - Axisymmetric FEM model](image)

Fig. 3.7 shows the local stresses along the fibre for various shear stiffnesses ($K = 100$, 1000 and 10000 N/mm$^2$) calculated with Bruggeling’s model and with the FEM model. Both models show similar results, proving the their accuracy.
3.1.2 Debonding

The second step is determined by the debonding of the fibre. As the fibre is pulled out the matrix, the bond shear stress increases in the fibre-matrix interface. At a certain stage, the adhesive bond in the fibre-matrix interface start to fail. When the maximum bond strength is reached, the fibre-matrix interface does not fail at once; a certain amount of stress can still be transferred through the fibre-matrix interface. After the maximum bond stress has been reached, the transfer of stresses through the interface decreases as the slip increases. Complete failure of the interface is reached, when no stresses are transferred through the interface. The process from of the maximum strength until complete failure of the interface is called “debonding”.

Modelling debonding

This chapter describes how to model debonding at the interface. There are two differences compared to modelling adhesive bonding. Firstly, a different material model is used for interfacial debonding. Secondly, the possibility for large displacements needs to be accounted for when debonding modelled. The following section discusses the material model for debonding. The next chapter discusses corresponding in the FEM formulation and how to cope with large displacements.

Material model

Debonding can be modelled with a traction-separation law (Needleman, 1987; Tvergaard, 1989). A traction-separation law (see Fig. 3.8) describes the relation between the local stress (traction) and the local displacement (separation) between two points or two surfaces. This is similar to a bond-slip relation. As the separation increases, the traction first increases linear-elasticly (adhesive bond) until the maximum traction is reached. From here damage starts to occur and the traction decreases linearly, exponentially or by any given function to zero. The area under the traction-separation diagram equals the fracture energy.
The traction-separation law should be defined for the three types of fracture modes, named mode I, II and III. Mode I relates to fracture in the normal direction and mode II and III relate to fracture in the two tangential directions (see Fig. 3.9).

Although the three modes can be defined independently, during a simulation a combination of failure modes may occur, as characterized by a mixed-mode definition (Camanho and Dávila, 2002). Fig. 3.10 shows the interaction between mode I and II. A criterion should be set to determine the combined maximum traction or maximum displacement (Damage initiation criterion) and a criterion should be set to determine the combined softening behaviour (Damage evolution criterion).
**FEM model**

**Debonding in FEM model**

This section describes the debonding behaviour at the interface in Abaqus (2011), where a mixed-mode definition is given by introducing a damage initiation criterion and a damage evolution criterion.

Damage initiation refers to the beginning of the degradation of the traction-separation law. Debonding starts at this point. Various damage initiation criteria can be selected in Abaqus. A quadratic stress criterion is used in the present study in accordance with

\[
\left( \frac{t_n}{t_n^o} \right)^2 + \left( \frac{t_s}{t_s^o} \right)^2 + \left( \frac{t_t}{t_t^o} \right)^2 = 1
\]

(3.3)

Normal traction \( t_n \) has been put inside Macaulay brackets, which indicates that pure compression does not lead to the initiation of damage.

Damage evolution describes the degradation of the stiffness once the damage initiation criterion has been fulfilled. Modelling damage is done with the damage variable \( D \), which ranges from 0 (no damage) to 1 (full damage). The affected stress components can be calculated with eq. 3.4, 3.5 and 3.6. \( \bar{t}_n, \bar{t}_s, \bar{t}_t \) describe the full elastic behaviour without damage. The normal component \( \bar{t}_n \) has been put inside Macaulay brackets, again indicating that pure compression does not influence the damage evolution.

\[
t_n = (1 - D) \cdot \langle t_n \rangle
\]

(3.4)

\[
t_s = (1 - D) \cdot \bar{t}_s
\]

(3.5)

\[
t_t = (1 - D) \cdot \bar{t}_t
\]

(3.6)

To describe the evolution of damage under a mixed-mode separation across the interface, an effective separation is introduced.

\[
\delta_m = \sqrt{\langle \delta_n \rangle^2 + \delta_s^2 + \delta_t^2}
\]

(3.7)

Pull-out tests have been performed in this research, which show exponential softening during debonding (Van den Bulck, 2013). Equation 3.8 gives the function of the damage variable for exponential softening. The evolution is based on the effective separation \( \delta_m \), where \( \alpha \) determines the damage evolution rate.

\[
D = 1 - \left\{ \frac{\delta_m}{\delta_m^0} \right\} \cdot \left[ 1 - \exp\left(-\alpha\left( \frac{\delta_m - \delta_m^0}{\delta_m^0 - \delta_m^0} \right) \right) \right]\]

\[
\frac{1}{1 - \exp(-\alpha)}
\]

(3.8)
The mode mixity at a point is defined by the relative proportions of normal and shear separations. The mode mix ratio ($\phi$) is calculated by equation 3.9, which is based on the traction values. The ratio $\phi$ can vary from 0 (tensile failure) to 1 (shear failure).

$$\phi = \frac{2}{\pi} \tan^{-1} \left( \frac{\sqrt{t_s^2 + t_t^2}}{t_n} \right)$$

(3.9)

**Large displacements in FEM model**

Debonding in the interface leads to large displacements. In order to explain how the FEM model treats large displacements, first the surface-based contact simulation in Abaqus is discussed. It is thereby emphasized under which specific conditions a surface-based contact simulation can be used.

A surface-based contact simulation (Abaqus, 2011) prescribes how surfaces that may get into contact with each other during a simulation. It uses a master surface (target surface) and a slave surface (contact surface) to model contact (see Fig. 3.11). During the simulation, the individual nodes on the slave surface try to make contact with the closest associated point on the master surface (contact detection). Subsequently, a discretization of the contact area is performed between the master surface and the slave surface. Subsequently, the analysis is continued in accordance with the chosen contact law.

![Fig. 3.11 - Master-Slave surface definition (Abaqus, 2011)](image)

**Contact detection**

The first step in a contact simulation is to check whether the slave surface is close enough to the master surface. This is determined by the maximal detection distance ($d_{\text{max}}$). In case a slave node falls within the contact detection zone of the master surface, this slave node is considered as a possible contact node at the current load step (see Fig. 3.12).

![Fig. 3.12 - Contact detection using $d_{\text{max}}$ (Yastrebov, 2010)](image)
Fig. 3.12 shows a solid with a master surface (round nodes) and a solid with a slave surface (triangular nodes). Four nodes (green triangles) of the slave surface fall within the contact detection zone. These slave nodes find the closest master surface nodes.

**Discretization**

The discretization describes the partitioning of the contact area into elementary units responsible for the transfer of contact stresses between contact surfaces. In the present thesis work node-to-surface discretization is used. The slave surface consists of surface nodes and the master surface consists of surfaces between the nodes. Each slave node is trying to make contact to an elementary master surface. (see Fig. 3.13).

There is a drawback to this discretization technique. The slave surface can be penetrated by the master surface, because the slave surface only consists of nodes (see Fig. 3.14). This leads to inaccurate results of the FEM model. These inaccuracies can be minimized by choosing the correct surface as master surface and/or by refining the mesh of the slave surface.

Finite sliding (Abaqus, 2011) has been used to model large displacements in the surface-based contact simulation. Finite sliding is used is suitable for large deformations, because the slave node can transfer load to any node on the master surface and the point of interaction on the master surface is updated every increment. (Kings and Richards, 2013)
**Condition of use**

The previous section explained how large deformations are modelled with a surface-based contact simulation. In this section it is demonstrated that the accuracy of the results depends for the greater part on the ratio between the maximum separation of the bond and the total surface length.

Schoenmakers (2013) discusses the surface-based contact simulation (cohesive surfaces). He does an analysis in mode II that consists of two bodies connected by surfaces. The top surface consists of slave nodes and the bottom surface is the master surface. The bodies in the analysis consist of one element (Fig. 3.15). A traction-separation law is used to model debonding, with a maximal separation of two times the total surface (separation to surface length ratio = 2:1).

![Fig. 3.15 - Mode II: 2x (1x1) elements (Schoenmakers, 2013)](image)

The result is plotted in Fig. 3.16 in a normalized traction-separation diagram. The red line shows the exact value and the blue line shows the calculated value by Abaqus. The numerical result indeed is very inaccurate.

![Fig. 3.16 - Normalized t-δ diagram for 2x (1x1) elements](image)

Fig. 3.16 shows two instantaneous drops of the traction. Both drops correspond to the loss of contact between the slave node and the master surface. As soon as the slave node is outside the contact zone, the traction between the slave node and the master surface reduces to zero. In case both slave nodes are outside the contact zone, the top body experiences a rigid body displacement.

The analysis above is repeated with different ratios of separation to surface length (1:1, 1:2, 1:5 and 1:10), to show the influence on the accuracy of the results. The results are plotted in Fig. 3.17.
It can be seen that lower ratios yield more accurate results. The ratio 1:10 provides the exact answer, because both slave nodes remain within the contact zone of the master surface and therefore no sudden drops in traction occur. It is decided to use a surface-based contact simulation for small ratios of separation to surface length, in order to prevent or to minimize inaccuracies.

Verification of the FEM model
To verify the debonding in the FEM model, a comparison is made with a model, based on Bruggeling’s tensile bar model (more information can be found in appendix B).

Bruggeling’s model needs as input a \( \tau - \delta \) relationship that describes the fibre-matrix interface. The pull-out of the fibre is performed displacement controlled (Appendix B).

In the FEM simulation an axisymmetric model is used (Fig. 3.18). For a proper comparison, the concrete is modelled as rigid. The interface has a length \( l_f = 30 \) mm and the fibre diameter \( D_f = 0.9 \) mm. In the comparison the pull-out behaviour is performed using two different \( \tau - \delta \) relationships, which are described in Fig. 3.19. The results are plotted in Fig. 3.20.
The FEM model and the modified model of Bruggeling give similar results for the pull-out behaviour of the fibre (Fig. 3.20).

### 3.1.3 Friction

The last phase of the pull-out response is determined by friction. When the interface is fully debonded, the fibre is no longer connected to the matrix. Experiments (Naaman et al., 1989; Van den Bulck, 2013) show that after full debonding of the fibre, a pull-out force remains to exist until the fibre is fully pulled out. This phenomenon can be explained by frictional stresses in the fibre-matrix interface.

According to Coulomb’s law of friction (Coulomb, 1785), a frictional force arises in case there is a compressive force between the contacting surfaces. This would relate to a compressive force in the interface, but the reason for this frictional resistance is seldomly described in literature. Naaman et al. (1989) and Li et Mobasher (1998) ascribe the activation of frictional resistance to shrinkage of the concrete matrix. The possibility of friction caused by imperfections of the fibre has not been discussed in literature.
The following section discusses friction caused by shrinkage and subsequently discusses the possibility of friction caused by the imperfection of the fibre.

**Friction due to shrinkage**

At the fibre, shrinkage of the concrete is prevented by the fibre, leading to compressive stresses between the fibre and the matrix (see Fig. 3.21).

![Fig. 3.21 - Shrinkage of the concrete matrix](image)

To obtain a better understanding of concrete shrinkage, a short overview of the different types of concrete shrinkage that could occur in a pull-out test is given below.

*Chemical shrinkage*

Chemical shrinkage refers to an absolute volume reduction associated with the chemical reaction between water and cement. The volume of water and cement is bigger than its chemical product. The volume reduction is mostly internal, but the bulk shrinkage is not negligible (Lura, 2003). Chemical shrinkage will not lead to significant stresses between fibre and matrix, because the Young's modulus is still low and the relaxation parameter is high in the early stage of the curing process.

*Thermal shrinkage*

Thermal shrinkage describes the expansion or shrinkage due to a temperature increase or decrease. During the chemical process the concrete heats up after which the temperature decreases again and the concrete matrix will shrink. Because the thermal expansion coefficient of concrete and steel are approximately the same, shrinkage of the concrete matrix will not lead to stresses between fibre and matrix.

*Autogenous shrinkage*

Autogenous shrinkage is an external volume change where, regardless of the mechanism provided there is no moisture transferred to the surrounding environment (Altoubat et Lange, 2001). In case there is not enough water in the paste for the cement to hydrate, the water in the pores is used for the cement hydration. This will increase the capillary action in the pores, leading to external shrinkage of the concrete. This is also known as self-desiccation. Autogenous shrinkage starts to occur in case the w/c ratio is below 0.42. The size of the shrinkage is dependent of the w/c ratio, but also the use of mineral admixtures (for example silica fume) and the type of aggregate influence the autogenous
shrinkage. (Holt, 2001). Autogenous shrinkage in the hardening phase (after 1 day) could lead to significant stresses between fibre and matrix. Whether it occurs and to what extent, depends on the concrete mixture.

**Drying shrinkage**
Drying shrinkage is an external volume change associated with the water exchange with the surrounding environment. Due to evaporation of the water at the surface, a non-uniform shrinkage pattern will occur throughout the cross-section. (Mors, 2011). Drying shrinkage is a long term process and is dependent on the weather condition. Drying shrinkage leads to significant stresses between fibre and matrix. The magnitude of shrinkage is mostly time dependent.

**Conclusion**
Both autogenous shrinkage and drying shrinkage could result in stresses between the fibre and the matrix. Whether or not these phenomena actually lead to stresses and to what extent depends on many factors, as described earlier.

**Friction due to imperfections**
Fig. 3.22 shows two geometrical imperfections of the fibre, which will lead to friction between fibre and matrix. Both imperfections cause a normal force between matrix and fibre in case a pull-out force is applied to the fibre, which induces to frictional forces. The exact influence of imperfections on the pull-out behaviour is unknown.

![Fig. 3.22 - Imperfections](image)

**Conclusion**
Both shrinkage and imperfections of the fibre could cause a frictional resistance. The magnitude of these frictional forces depends on many factors as described in this section. In this thesis shrinkage is used to model frictional pull-out. The influence of imperfections is treated in chapter 4.2.

**Modelling friction**
Friction is modelled using Coulomb’s law of friction (Coulomb, 1785). Coulomb’s law of friction states that the friction force \( F_f \) is a fraction \( \mu \) of the normal force \( F_n \) (see Fig. 3.23).
FEM model
This section discusses friction in the FEM model. In order to model friction in Abaqus, the tangential behaviour and the normal behaviour are defined. Tangential behaviour describes the shear deformation between two surfaces. This is modelled by using Coulomb’s law of friction. The normal behaviour needs to be defined to prevent or minimize penetration of the surfaces. This is modelled with a pressure-overclosure contact model.

Tangential behaviour
Coulomb’s law of friction determines when two contacting surfaces stick to each other and when two surfaces slide relatively to each other. The maximum allowable shear stress ($\tau_{\text{crit}}$) between two contacting surfaces is a function of the coefficient of friction ($\mu$) and the contact pressure ($p$):

$$\tau_{\text{crit}} = \mu p$$  \hspace{1cm} (3.10)

In case a three-dimensional simulation is performed, the shear stress should be defined in two directions. These two stress components can be combined into an equivalent shear stress ($\tau$), assuming an isotropic coefficient of friction ($\mu$):

$$\tau = \sqrt{\tau_1^2 + \tau_2^2}$$  \hspace{1cm} (3.11)

Fig. 3.24 shows the equivalent shear stress as a function of contact pressure. The stick region is defined as the area under the critical shear stress.

---

**Fig. 3.23 - Coulomb’s law of friction**

**Fig. 3.24 - Stick region for Coulomb’s law of friction (Abaqus, 2011)**
Abaqus provides two methods to enforce the tangential friction constraint in the contact analysis: the Penalty method and the Lagrange multiplier method. For the penalty method some relative motion of the surfaces is permitted when the surfaces should be sticking. The magnitude of sliding is limited to the elastic slip, which can be set. The Langrange multiplier method does not allow relative motion in the stick region and is therefore more accurate than the penalty method. Due to adding a multiplier to the contact formulation, the number of variables increases. Therefore this method is computationally somewhat more expensive than the penalty method. The penalty method is used in the calculations.

**Normal behaviour**

The normal behaviour is described by the pressure-overclosure contact model. This model minimizes penetration of the master surface by slave nodes. This constraint induces pressure when the surfaces are in contact and sets the pressure to zero when the surfaces separate.

![Fig. 3.25 - Pressure-overclosure model (Abaqus, 2011)](image)

The contact pressure $p$ is a function of the overclosure distance $h$, which is the penetration distance during the iterations before equilibrium in the increment is achieved. The basic model is described by the two conditions below:

$$
p = 0 \text{ for } h < 0 \text{ (open)}
$$

$$
\hat{h} = 0 \text{ for } p > 0 \text{ (closed)}
$$

Also for normal behaviour Abaqus allows you to choose between the Penalty method and the (Augmented) Langrange multiplier method. Because of the high convergence rate and the expectation that the normal forces will have relatively low values (minor penetration), the penalty method is used to calculate the contact pressure.

**Verification friction model**

In order to verify the friction model, a simple sliding test is carried out (see Fig. 3.26). The test consists of two rigid solids, which slide relatively to each other. The top solid is loaded with $\sigma = 100 \text{ N/mm}^2$ in vertical direction and will slide in horizontal direction ($\delta = 4,0 \text{ mm}$) with $\mu = 0,5$ and $A_{\text{contact}} = 1,0 \text{ mm}^2$. According to Coulomb's friction law this will lead to a sliding force of $50 \text{ N}$. 


The FEM model uses the penalty method to model the frictional behaviour. This means an elastic slip is introduced. The magnitude of the elastic slip has been set to 0.10 mm. Fig. 3.27 shows an exact match between the FEM model and analytical value, which verifies the FEM model.

3.1.4 Combining debonding and friction

The interaction behaviour in tangential direction consists of two phases. At first, the model describing debonding is fully active and the friction model is inactive. The slip is fully elastic and will be resisted by the adhesive bond. As soon as the damage initiation criterion is satisfied the friction model becomes active. As damage keeps increasing, both models are partly contributing to the shear stresses. Once full debonding has occurred, the debonding model does not contribute to the shear stress anymore and the frictional model is fully active. Fig. 3.28 shows the interaction of the two models for linear and exponential softening.
3.2 Fibre

This chapter describes the nonlinear material model, which is used to model the material behaviour of the fibre.

Fig. 3.29 - Elasto-plastic behaviour (Krabbenhøft, 2002)

3.2.1 Plasticity

To model the plastic behaviour of the fibre the yield behaviour (yield criterion and yield surface) and post-yield behaviour (hardening and flow rule) have to be defined.

Yield behaviour

The Von Mises yield criterion (Von Mises, 1913) is used for defining the elastic threshold above which plasticity occurs. This criterion (yield function \( F \)) states that yielding of the elasto-plastic material occurs when the effective stress \( \sigma \) reaches the yield stress \( \sigma_0 \):

\[
F = \bar{\sigma} - \sigma_0 \leq 0
\]  

(3.12)

where:

\[
\bar{\sigma} = \left[ \frac{1}{2} (\sigma_1 - \sigma_2)^2 + \frac{1}{2} (\sigma_2 - \sigma_3)^2 + \frac{1}{2} (\sigma_3 - \sigma_1)^2 \right]^{\frac{1}{2}}
\]

(3.13)

where \( \sigma_1, \sigma_2, \) and \( \sigma_3 \) are the principal stresses, \( \sigma_{xx}, \sigma_{yy} \) and \( \sigma_{zz} \) are the normal stresses in the x-y-z reference system, and \( \tau_{xy}, \tau_{yz}, \) and \( \tau_{zx} \) are the shear stresses.

The von Mises yield criterion can be represented in the principal stress space by an infinite long cylinder (Fig. 3.30). The surface of the cylinder is called the yield surface, which corresponds to \( F = 0 \). Points which lie within the yield surface represent elastic behaviour and points that lie on the surface represent plastic behaviour. Points outside the yield surface are not permitted.
Post-yield behaviour

Within the yield surface the behaviour is elastic. But as the loading increases, the yield surface will be reached and plastic yielding will occur. Most metals exhibit hardening when yielding. Hardening leads to a modification in shape and/or position of the yield surface. Isotropic hardening has been used in de FEM simulation (Fig. 3.31). With isotropic hardening the yield surface increases in size uniformly in all directions as plastic straining occurs.

When the stress reaches the yield surface, non-linear deformation will occur. The increment of the deformation can be divided into an elastic part and a plastic part:

\[ \varepsilon = \varepsilon^{el} + \varepsilon^{pl} \]  

(3.14)

The plastic strain increment can be calculated with the plastic flow rule (see eq. 3.15). For most metals the associated flow rule (normality rule) is valid (Abaqus, 2011). The parameter \( d \lambda \) is a positive scalar of proportionality and \( G \) is called the plastic potential. The plastic potential is equal to the yield criterion for associated plasticity. This rule implies that the plastic strain is in the direction of the normal of the yield surface. This leads to a flow rule where the yield criterion is directly coupled to the plastic strain.

\[ d\varepsilon^{pl} = d\lambda \frac{\partial G}{\partial \sigma} \]  

(3.15)

Where:

\[ G = F \]  

(3.16)
3.2.2 Parameters for plasticity

Plasticity requires a relationship between yield stress ($\sigma_0$) and the elastic strain ($\varepsilon^{pl}$). This relation is shown in Fig. 3.32, and has been derived from steel fibre tensile tests (Van den Bulck, 2013).

Fig. 3.32 - $\sigma_0$ - $\varepsilon^{pl}$ relation
3.3 Matrix

This chapter describes the nonlinear material model of concrete and the shrinkage of the matrix. Concrete is an anisotropic material, it behaves different when loaded in tension than when loaded in compression. To model the behaviour accurately, a concrete material model is used. Shrinkage of the concrete matrix is simulated using a thermal analogy.

3.3.1 Concrete Damaged Plasticity

The Concrete Damaged Plasticity model (Lee and Fenves, 1998) is used to model the concrete material behaviour. CDP is a continuum, plasticity-based, damage model for concrete, which has been derived from the Drucker-Prager yield criterion. The main two failure mechanisms of concrete are combined in one model; cracking (tension) and crushing (compression).

**Stress-strain relationship**

The CDP model is based on a stress-strain relationship with scalar damage:

\[
\sigma = (1 - d) D^0 (\varepsilon - \varepsilon^{pl}) = D^{pl} (\varepsilon - \varepsilon^{pl})
\]  
(3.17)

The scalar stiffness degradation variable \(d\) can vary between 0 (undamaged) and 1 (fully damaged). Note that \(D^0\) is the initial (undamaged) elastic stiffness of the material and \(D^{pl} = (1 - d)D^0\) is the damaged stiffness. An increase in damage leads to a reduction of the elastic stiffness. The evolution of the damage variable \(d\) is governed by the effective stress \(\bar{\sigma}\) and the hardening (softening) variable \(\bar{\varepsilon}^{pl}\):

\[
d = d(\bar{\sigma}, \bar{\varepsilon}^{pl})
\]  
(3.18)

Cracking and crushing in concrete are represented by increasing values of the hardening variables:

\[
\bar{\varepsilon}^{pl} = \begin{bmatrix} \bar{\varepsilon}^{pl}_t \\ \bar{\varepsilon}^{pl}_c \end{bmatrix}
\]

(3.19)

The evolution of the hardening variable is characterized by the two hardening variables, the equivalent plastic strains in tension \(\bar{\varepsilon}^{pl}_t\) and in compression \(\bar{\varepsilon}^{pl}_c\). These variables control the evolution of the yield surface and the degradation of the elastic stiffness.

The equivalent plastic strains are determined through the uniaxial behaviour of concrete either in tension (eq. 3.20 and Fig. 3.33) or in compression (eq. 3.21 and Fig. 3.34). The user can define the stress-strain behaviour of concrete as a tabular function of the cracking strain \(\varepsilon^{cr}_t\) or the crushing strain \(\varepsilon^{cr}_c\):

\[
\bar{\varepsilon}^{pl}_t = \varepsilon^{cr}_t - \frac{d_t}{(1 - d_t) E_0} \sigma_t
\]  
(3.20)

\[
\bar{\varepsilon}^{pl}_c = \varepsilon^{cr}_c - \frac{d_c}{(1 - d_c) E_0} \sigma_c
\]  
(3.21)
Yield behaviour

The yield function (eq. 3.22) represents a stress surface, which determines the states of failure or damage (see Fig. 3.35). The yield function is governed by the effective stress and the hardening variable:

\[
F(\bar{\sigma}, \dot{\varepsilon}^p_l) = \frac{1}{1 - \alpha} \left( \bar{q} - 3\alpha\bar{p} + \beta(\dot{\varepsilon}^p_l) \left( \hat{\sigma}_{\text{max}} - \gamma \dot{\varepsilon}_{\text{max}} - \bar{\sigma}_c \right) \right) \leq 0 \tag{3.22}
\]

Where:

\[
\alpha = \frac{\sigma_{b0} - \sigma_{c0}}{2\sigma_{b0} - \sigma_{c0}} \tag{3.23}
\]

\[
\beta(\dot{\varepsilon}^p_l) = \frac{\bar{\sigma}_c(\dot{\varepsilon}^p_l)}{\bar{\sigma}_c(\dot{\varepsilon}^p_l)} (1 - \alpha) - (1 + \alpha) \tag{3.24}
\]
\[ \gamma = \frac{3(1 - K_c)}{2K_c - 1} \]  

(3.25)

where \( \bar{p} \) is the effective hydrostatic pressure, \( \bar{q} \) the Mises equivalent effective stress and \( \hat{\sigma}_{\text{max}} \) the maximum eigenvalue \( \hat{\sigma} \). Further, \( \alpha, \beta \) and \( \gamma \) are dimensionless material parameters, which determine the shape of the yield surface. The parameter \( \alpha \) (and indirectly \( \beta \)) is governed by the ratio of the biaxial compressive stress and the uniaxial compressive stress \( (\sigma_{\text{so}} / \sigma_{\text{so}}) \), which can be defined in Abaqus. This value has been set at 1.16. The parameter \( \gamma \) is based on triaxial stress results, and is governed by \( K_c \), which can be defined in Abaqus. \( K_c = 0.667 \) is recommended (Abaqus, 2011).

**Flow rule**

The plastic strain can be determined with the flow rule (eq. 3.26). The flow rule is governed by the hyperbolic Drucker-Prager flow potential (eq. 3.27). The yield function and the flow potential do not coincide, which can be characterized as nonassociated potential flow. The plastic flow therefore develops perpendicular to the plastic flow potential, but not perpendicular to the yield surface.

\[ d\varepsilon^{pl} = d\lambda \frac{\partial G}{\partial \sigma} \]  

Where;

\[ G = \sqrt{(\varepsilon \sigma_{\text{so}} \tan \psi)^2 + \bar{q}^2} - \bar{p} \tan \psi \]  

(3.27)

The dilation angle \( (\psi) \) determines the angle of the flow potential in the \( \bar{p} - \bar{q} \) plane. The dilation angle can be defined in Abaqus. A standard value is 38° for concrete (Abaqus, 2011). The eccentricity parameter \( (\varepsilon) \) characterizes the tension cut-off in the \( \bar{p} - \bar{q} \) space. An eccentricity of 0.0 corresponds to the dashed line in Fig. 3.36. In the present study, a value of 0.1 is chosen, which is realistic for concrete.
3.3.2 Parameters for CDP

Table 3.1 shows 5 parameters, required for the CDP model. Fig. 3.37 shows the stress-strain relation of the tensile behaviour, which is based on Hordijk’s experimental results (1991). Fig. 3.38 shows the stress-strain relation in compressive behaviour, which can be calculated with the Eurocode 2 (2011).

Table 3.1: Parameters for concrete damaged plasticity

<table>
<thead>
<tr>
<th>Parameter(s)</th>
<th>Value(s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>CDP</td>
<td></td>
</tr>
<tr>
<td>Dilatation angle</td>
<td>$\psi = 38^\circ$</td>
</tr>
<tr>
<td>Eccentricity</td>
<td>$\epsilon = 0,1$</td>
</tr>
<tr>
<td>Ratio biaxial/uniaxial</td>
<td>$\sigma_b/\sigma_c = 1,16$</td>
</tr>
<tr>
<td>Parameter $K_c$</td>
<td>$K_c = 0,667$</td>
</tr>
<tr>
<td>Viscosity parameter</td>
<td>$\mu = 0,0001$</td>
</tr>
</tbody>
</table>

Fig. 3.37 - Input $\sigma$-$\varepsilon_t^{th}$ relation in tension
3.3.3 Shrinkage

Thermal shrinkage of the matrix is characterized by the coefficient of thermal expansion ($\alpha$). In the present study isotropic shrinkage is assumed, for which the coefficient $\alpha$ is the same in the three spatial directions.

![Graph showing input $\sigma$-strain $\varepsilon_c$ relation in compression](image)
CHAPTER 4:
STRAIGHT FIBRE

This chapter shows the numerical results for the pull-out of a single straight fibre. The first part shows the results of a standard pull-out test, while the second part shows the influence of geometrical imperfections on the pull-out behaviour, and the last part shows the influence of lateral forces.

4.1 Pull-out test

4.1.1 FEM model

As a first step, an axi-symmetric model is used to simulate the pull-out behaviour of a steel fibre from a concrete matrix. Fig. 4.2 shows the geometry of the FEM model.

The model uses axisymmetric elements (CAX4: A 4-node bilinear axisymmetric quadrilateral). For simplicity, both the steel and the concrete material models are initially assumed as linear elastic. An initial state of friction is assumed to be caused by autogenous shrinkage. Shrinkage of the concrete
matrix is assumed to occur uniformly along the fibre length and is simulated using the analogy with thermal shrinkage. The shrinkage strain is given in table 4.1. The behaviour in the fibre-matrix interface is modelled nonlinear. The interaction between the fibre and the matrix is modelled with a surface contact formulation. Two contact surfaces are defined: a master surface (concrete surface) and a slave surface (fibre surface).

The material properties, which are used in the FEM model, are shown in Table 4.1. The parameters, which determine the traction-separation law \( (K, \tau, \delta, \text{and } \alpha) \), are based on experimental pull-out tests, which have been performed within a different research project as a support for the present research. The experimental research is discussed in chapter 6. The parameters, influencing the frictional pull-out behaviour \( (\varepsilon_{\text{shr}} \text{ and } \mu) \), are taken from literature (Mangat and Azari, 1984; Lura, 2003).

**Table 4.1: Material properties**

<table>
<thead>
<tr>
<th>Parameter(s)</th>
<th>Value(s)</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Fibre</strong></td>
<td></td>
</tr>
<tr>
<td>Young’s modulus</td>
<td>( E = 210000 , \text{N/mm}^2 )</td>
</tr>
<tr>
<td>Poisson’s ratio</td>
<td>( \nu = 0,3 )</td>
</tr>
<tr>
<td><strong>Matrix</strong></td>
<td></td>
</tr>
<tr>
<td>Young’s modulus</td>
<td>( E = 33000 , \text{N/mm}^2 )</td>
</tr>
<tr>
<td>Poisson’s ratio</td>
<td>( \nu = 0,2 )</td>
</tr>
<tr>
<td>Shrinkage strain</td>
<td>( \varepsilon_{\text{shr}} = 1,5\times10^{-4} )</td>
</tr>
<tr>
<td><strong>Fibre-matrix interface</strong></td>
<td></td>
</tr>
<tr>
<td>Elastic stiffness</td>
<td>( K = 11 , \text{N/mm}^2 )</td>
</tr>
<tr>
<td>Maximum traction</td>
<td>( \tau_n = \tau_s = \tau_t = 0,7 , \text{N/mm}^2 )</td>
</tr>
<tr>
<td>Maximum separation</td>
<td>( \delta_n = \delta_s = \delta_t = 0,5 , \text{mm} )</td>
</tr>
<tr>
<td>Exponential parameter</td>
<td>( \alpha = 6 )</td>
</tr>
<tr>
<td>Coefficient of friction</td>
<td>( \mu = 0,04 )</td>
</tr>
</tbody>
</table>

### 4.1.2 Mesh study

A mesh refinement study is done to find the optimal mesh size for the FEM model, as a balance between the accuracy of the results and the computational time. Three different meshes have been defined, named Coarse, Intermediate and Fine (Fig. 4.3). The length of the elementary slave surfaces (fibre) are taken a quarter of the length of the elementary master surfaces (matrix) to minimize the chance of penetration between the surfaces.

![Fig. 4.3 - Three different meshes: Coarse, Intermediate and Fine](image-url)
Fig. 4.4 shows the pull-out behaviour for the three different meshes. Only minor differences can be observed between the three responses. Hence, the coarse mesh is chosen for the remaining computations, since this mesh results in the shortest computational time of the three meshes investigated.

4.1.3 Discussion of the results

Fig. 4.5 shows the pull-out behaviour ($\delta = 0$-1 mm) calculated with the FEM model. In this diagram the most important mechanisms, which determine the pullout behaviour, are shown separately: (de)bonding (blue) and friction (red). These results are drawn schematically and only give a qualitative image of the two phenomena. The dashed line indicates the displacement at which the fibre starts to debond. From this moment the frictional resistance along the fibre length is mobilised. Four points have been highlighted in the diagram. These points are discussed.
Point 1 ($\delta = 0$ mm) describes the initial state of the FEM model. The stresses and strains are determined by shrinkage of the concrete matrix. The stress state in x-direction (S11) is shown in Fig. 4.6. Shrinkage of the matrix in x-direction leads to compressive stresses in x-direction close to the interface, due to restrained deformation of the matrix caused by the fibre.

At point 2 ($\delta = 0.07$ mm) the maximum pull-out force (58.0 N) is reached. Decohesion has already started at this point. At point 3 ($\delta = 0.20$ mm) both the surface-based cohesive model and the frictional model are active, and at point 4 ($\delta = 1.0$ mm) the pull-out force is only determined by the frictional model.
4.2 The influence of imperfections on the pull-out behaviour

Paragraph 2.1.2 discussed the frictional pull-out of a straight fibre. This paragraph indicates that shrinkage and imperfections of the fibre could lead to frictional resistance. In chapter 4.1 it was assumed that friction was caused by shrinkage of the matrix. This chapter treats the influence of imperfections on the frictional resistance. First a 2D FEM analysis is performed to gain understanding of the pull-out behaviour and subsequently a 3D FEM analysis is carried out.

4.2.1 2D FEM analysis

FEM model

In this research the skewed fibre is treated. Fig. 4.7 shows the FEM model for this imperfection. The fibre is modelled straight and is pulled out under an angle.

![2D geometry FEM model](image)

Fig. 4.7 - 2D geometry FEM model

The model uses plain stress elements (CPS4: A 4-node bilinear plane stress quadrilateral). Both the steel and the concrete material models are initially assumed as linear elastic. The shrinkage strain is assumed zero. The behaviour in the fibre-matrix interface is modelled non-linear. The interaction is modelled with a surface-based contact formulation. Two contact surfaces are defined: a master surface (concrete surface) and a slave surface (fibre surface). The material properties, which are used in the FEM model, are shown in Table 4.2.
Table 4.2: Material properties

<table>
<thead>
<tr>
<th>Parameter(s)</th>
<th>Value(s)</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Fibre</strong></td>
<td></td>
</tr>
<tr>
<td>Young’s modulus</td>
<td>$E = 210000 , \text{N/mm}^2$</td>
</tr>
<tr>
<td>Poisson’s ratio</td>
<td>$v = 0.3$</td>
</tr>
<tr>
<td>Pull-out angle</td>
<td>$\alpha = 0, 1, 2 \text{ and } 4^\circ$</td>
</tr>
<tr>
<td><strong>Matrix</strong></td>
<td></td>
</tr>
<tr>
<td>Young’s modulus</td>
<td>$E = 33000 , \text{N/mm}^2$</td>
</tr>
<tr>
<td>Poisson’s ratio</td>
<td>$v = 0.2$</td>
</tr>
<tr>
<td>Shrinkage strain</td>
<td>$\varepsilon_{shr} = 0$</td>
</tr>
<tr>
<td><strong>Fibre-matrix interface</strong></td>
<td></td>
</tr>
<tr>
<td>Elastic stiffness</td>
<td>$K = 11 , \text{N/mm}^3$</td>
</tr>
<tr>
<td>Maximum traction</td>
<td>$t_x = t_z = t_t = 0.7 , \text{N/mm}^2$</td>
</tr>
<tr>
<td>Maximum separation</td>
<td>$\delta_x = \delta_z = \delta_t = 0.5 , \text{mm}$</td>
</tr>
<tr>
<td>Exponential parameter</td>
<td>$\alpha = 6$</td>
</tr>
<tr>
<td>Coefficient of friction</td>
<td>$\mu = 0.04$</td>
</tr>
</tbody>
</table>

Mesh study and validation of the model

The coarse mesh from the standard pull-out test is used to model imperfection in 2D (Fig. 4.8). The result of the pull-out behaviour is shown in Fig. 4.9 with $\alpha = 4^\circ$.

![Coarse mesh for imperfection model](image)

Fig. 4.9 shows a graph with large saw teeth. These are caused by penetration of the slave surface by the master surface. Fig. 4.10 shows the stress distribution in x-direction (S11) in point 1 (no penetration) and point 2 (penetration) in the graph. Due to loss of contact between the slave node (fibre) and the master surface (matrix) a penetration occurs. This leads to an immediate drop in contact pressure, which leads to an immediate drop in the pull-out force. This problem is solved by exchanging the master and the slave surface, as a result of which no penetration occurs at that point.
Imperfect pull-out locally leads to local high stresses in the top of the matrix nearby the fibre. Due to the large size of the elements, high stress discontinuities occur. By refining the element size at the top of the matrix, stress discontinuities can be reduced.

Furthermore, an automatic step refinement is done in Abaqus. Abaqus by default does a check on the residual forces after four equilibrium iterations ($I_0 = 4$). When the residual forces increase in two consecutive iterations, the calculation within the time increment stops and a new calculation starts with a reduced time increment. Furthermore, Abaqus does a convergence check after eight equilibrium iterations ($I_R = 8$). To prevent premature cutbacks analysis the number of equilibrium iterations for both checks have been increased ($I_0 = 8$ and $I_R = 10$).
As described before, the concrete material model has been assumed as linear elastic. As a result, the stresses in the concrete matrix are not very realistic. Locally high stresses occur, which lead to crushing of the concrete, which influences the pull-out behaviour. The concrete damaged plasticity model can be used to model this behaviour. For a better understanding of the influence of imperfections on the pull-out behaviour, the elastic model is used in a first FEM calculation and subsequently the CDP model is used.

This leads to a new simulation, with a new mesh (Fig. 4.11) and an exchange of the master and slave surface. The concrete material model is still elastic. The results of the pull-out behaviour with $\alpha = 4^\circ$ are plotted in Fig. 4.12. It shows that both a mesh refinement and an exchange of the surfaces is needed to find a plausible result.

**Fig. 4.11 - New mesh for imperfection model**

**Fig. 4.12 - Pull-out behaviour for different FEM models**
Discussion of the results with elastic material model

Fig. 4.13 shows the stress distribution during pull-out of a fibre in x-direction (S11). Two points (A and B) determine the frictional force. The rest of the fibre is not subjected to friction. Point A is subjected to a much higher force than point B. Therefore the magnitude of the friction is mostly determined by point A.

Fig. 4.13 - Stress distribution S11

Fig. 4.14 shows the pull-out behaviour for \( \alpha = 0, 1, 2 \) and \( 4^\circ \). For \( \alpha = 0^\circ \) no friction occurs. This is in line with paragraph 2.1.2, which states that friction can only be caused by shrinkage of the matrix and imperfections of the fibre. The graph for \( \alpha = 1, 2 \) and \( 4^\circ \) show a similar shape, with a higher residual force for a larger \( \alpha \).

Fig. 4.14 - Pull-out behaviour with linear elastic material model

The curvature directly determines the frictional force. The magnitude of the curvature is determined by the ratio between the horizontal displacement of the top of the fibre and the distance from the top of the fibre to the top of the matrix. This means that the initial fibre length, which is above the matrix \( (l_i) \), has a big influence on the size of the frictional force. If a large \( l_i \) is chosen, the curvature will be small and therefore the frictional force will be low and vice versa. The influence of this length is not further investigated in this research. The \( l_i \) is kept at 2.0 mm.
Discussion of the results with CDP model

Fig. 4.15 shows the influence of the CDP model on the pull-out behaviour in comparison to the elastic model. The diagram shows a non-proportional reduction of the frictional resistance for a decreasing imperfection. The reduction of frictional resistance is caused by the increase of strain in the concrete matrix (Fig. 4.16).

![Fig. 4.15 - Pull-out behaviour with CDP model](image)

![Fig. 4.16 - Principal strain for α=4° (a) elastic concrete model (b) CDP model](image)
4.2.1 3D FEM analysis

The 2D model is extended towards a 3D model. For good understanding of the FEM model, two analyses are done. The concrete material model is elastic in the first analysis and the CDP model is used in the second analysis. The result of the 2D model cannot be compared quantitatively. Therefore the results are compared qualitatively.

Mesh

The 3D mesh has a similar discretization fineness as the 2D mesh. Fig. 4.17 shows the 3D mesh. The model uses 3D brick elements (C3D8: an 8-node linear brick). The material properties of the 2D model are used (Table 4.2).

![3D mesh for imperfections](image)

Fig. 4.17 - 3D mesh for imperfections

Discussion of the results with elastic material model

Fig. 4.18 shows the results the pull-out behaviour with an elastic material model for the 3D model. The results of the 3D FEM model show good qualitative resemblance with the results of the 2D FEM model. The shape of the curves is very similar.

![Pull-out behaviour with elastic model for 3D model](image)

Fig. 4.18 - Pull-out behaviour with elastic model for 3D model
Discussion of the results with CDP model

Fig. 4.19 shows the influence of the CDP material model in a 3D FEM calculation.

Clearly, the pull-out behaviour is influenced by the magnitude of the imperfection. To gain more information about the influence of imperfections on the fibre pull-out, an imperfection study should be done. In this thesis no further research is done on the influence of imperfections of the fibre.
4.3 The influence of lateral normal stress on the pull-out behaviour

By introducing a lateral normal stress on the concrete matrix in a pull-out test, a biaxial stress situation for the SFRC can be simulated.

4.3.1 FEM model

Fig. 4.20 shows the geometry of the FEM model. The specimen is subjected to a lateral normal stress during the pull-out. The magnitude of the lateral stress equals 2.8 N/mm$^2$ and 14.0 N/mm$^2$.

![FEM model diagram](image)

**Fig. 4.20 - 3D geometry FEM model**

<table>
<thead>
<tr>
<th>Table 4.4: Material properties</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Parameter(s)</strong></td>
</tr>
<tr>
<td>Fibre</td>
</tr>
<tr>
<td>Young’s modulus</td>
</tr>
<tr>
<td>Poisson’s ratio</td>
</tr>
<tr>
<td>Matrix</td>
</tr>
<tr>
<td>Young’s modulus</td>
</tr>
<tr>
<td>Poisson’s ratio</td>
</tr>
<tr>
<td>Shrinkage strain</td>
</tr>
<tr>
<td>Lateral stress on matrix</td>
</tr>
<tr>
<td>Fibre-matrix interface</td>
</tr>
<tr>
<td>Elastic stiffness</td>
</tr>
<tr>
<td>Maximum traction</td>
</tr>
<tr>
<td>Maximum separation</td>
</tr>
<tr>
<td>Exponential parameter</td>
</tr>
<tr>
<td>Coefficient of friction</td>
</tr>
</tbody>
</table>
The model uses 3D brick elements (C3D8: a 8-node linear brick). Friction is assumed to be caused by autogenous shrinkage of the matrix. The matrix surface is defined as the master surface and the fibre surface is defined as the slave surface. The material properties, which are used in the FEM model, are shown in Table 4.4.

4.3.2 Mesh study

The axisymmetric FEM model cannot be used for asymmetric loading situations. Therefore the axisymmetric model is transferred to a 3D model. Fig. 4.21 shows the mesh of the 3D model. Due to symmetry only a quarter is modelled.

The 3D model is compared to the axisymmetric model in Fig. 4.22. The pull-out response of the two models is in good correspondence.
4.3.3 Discussion of the results

For better understanding of the FEM model, first a FEM analysis with the linear elastic concrete model is performed and subsequently a FEA with the CDP model is performed.

Linear elastic concrete model

Fig. 4.23 shows the pull-out behaviour for a standard pull-out test (shrinkage) and two laterally loaded pull-out tests ($\sigma_{\text{lat},1} = 2.8$ and $\sigma_{\text{lat},2} = 14 \text{ N/mm}^2$). The maximum pull-out force of the pull-out test loaded with $\sigma_{\text{lat},1}$ barely increases in comparison to the standard pull-out test. The frictional force on the other hand does increase significantly. The pull-out behaviour of $\sigma_{\text{lat},2}$ shows a minor increase of the maximum pull-out force and a major increase in the frictional force.

![Graph showing pull-out behaviour subjected to lateral stresses](image)

To gain insight in the change of the two mechanisms ((de)bond and frictional resistance), the contribution to the pull-out behaviour is drawn separately for each pull-out test for $\delta = 0-1 \text{ mm}$. The (de)bond contribution is blue and the friction contribution is red. These results are drawn schematically and only give a qualitative image of the two phenomena. The dashed line indicates the moment of damage initiation (see Fig. 4.24, Fig. 4.25 and Fig. 4.26).
Fig. 4.24 - Debonding and frictional contribution to pull-out behaviour

Fig. 4.25 - Debonding and frictional contribution to pull-out behaviour for $\sigma_{a,1} = 2.8 \text{ N/mm}^2$

Fig. 4.26 - Debonding and frictional contribution to pull-out behaviour for $\sigma_{a,2} = 14 \text{ N/mm}^2$
The figures show what is happening in the FEM model when a lateral normal stress is applied. The (de)bonding behaviour in shear direction is not influenced by normal compression in the interface. The damage initiation point remains the same, because compression is not taken into account in the damage initiation condition (Eq. 3.2 (par. 3.1.4)). Therefore the damage initiation is fully determined by the traction in shear direction. Also the damage evolution in shear direction is not influenced by compression in the interface (Eq. 3.3 and Eq 3.6 (par. 3.1.4)), which means that the (de)bonding energy is equal in all three figures. The friction behaviour is directly influenced by compression in the interface. The magnitude of the friction is directly determined by the compression in the interface (see Coulomb’s law of friction Eq. 3.8).

**CDP model**

Fig. 4.27 shows the influence of the CDP on the pull-out behaviour. The pull-out behaviour of $\sigma_{lat,2} = 14,0 \text{ N/mm}^2$ shows a significant decrease in residual pull-out force compared to the elastic case. The pull-out behaviour of $\sigma_{lat,1} = 2,8 \text{ N/mm}^2$ shows a minor decrease in pull-out force. The frictional force reduces, due to crushing (negative plastic strain) of the concrete matrix nearby the interface, which leads to a reduction in the compressive stress in the interface.

![Graph showing pull-out behaviour](image-url)

*Fig. 4.27 - Pull-out behaviour subjected to lateral stresses with CDP model*
CHAPTER 5: HOOKED-END FIBRE

This chapter shows the results of the numerical model for a pull-out of a hooked-end fibre. A standard pull-out test is modelled in 2D.

5.1 Pull-out test
5.1.1 FEM model

A typical pull-out test is modelled. Fig. 5.1 shows the 2D geometry of the FEM model.

The model uses plain stress elements (CPS4: A 4-node bilinear plane stress quadrilateral). Steel and concrete are both assumed to behave nonlinearly. The material model of steel includes plasticity and the material model of concrete is represented by the CDP model. Shrinkage of the concrete matrix is not taken into account. The interaction is modelled with a surface-based contact formulation. Two contact surfaces are defined: a master surface (concrete surface) and a slave surface (fibre surface). In order to model large deformations of the fibre correctly, nonlinear geometric effects need to be taken into account; therefore a geometrically non-linear analysis is performed. The material properties, used in the FEM simulation, are listed in Table 4.5.
When determining the optimal mesh for the FEM model, the concrete material is assumed as linear elastic. Local refinement of the mesh is done nearby the contact surfaces to minimize penetration of the slave surface (fibre) by the master surface (concrete). Fig. 5.2 shows the three different meshes, named Coarse, Intermediate and Fine. In each mesh in the mesh size nearby the contact surfaces has a different refinement.

<table>
<thead>
<tr>
<th>Parameter(s)</th>
<th>Value(s)</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Fibre</strong></td>
<td></td>
</tr>
<tr>
<td>Young’s modulus</td>
<td>$E = 210000 \text{ N/mm}^2$</td>
</tr>
<tr>
<td>Poisson’s ratio</td>
<td>$\nu = 0.3$</td>
</tr>
<tr>
<td><strong>Matrix</strong></td>
<td></td>
</tr>
<tr>
<td>Young’s modulus</td>
<td>$E = 33000 \text{ N/mm}^2$</td>
</tr>
<tr>
<td>Poisson’s ratio</td>
<td>$\nu = 0.2$</td>
</tr>
<tr>
<td>Shrinkage strain</td>
<td>$\varepsilon_{shr} = 0$</td>
</tr>
<tr>
<td><strong>Fibre-matrix interface</strong></td>
<td></td>
</tr>
<tr>
<td>Elastic stiffness</td>
<td>$K = 11 \text{ N/mm}^3$</td>
</tr>
<tr>
<td>Maximum traction</td>
<td>$t_n = t_s = t_t = 0.7 \text{ N/mm}^2$</td>
</tr>
<tr>
<td>Maximum separation</td>
<td>$\delta_n = \delta_s = \delta_t = 0.5 \text{ mm}$</td>
</tr>
<tr>
<td>Exponential parameter</td>
<td>$\alpha = 6$</td>
</tr>
<tr>
<td>Coefficient of friction</td>
<td>$\mu = 0.04$</td>
</tr>
</tbody>
</table>

**5.1.2 Mesh study**

Fig. 5.2 - Three different meshes: Coarse, Intermediate and Fine
Fig. 5.3 shows results of the pull-out behaviour for the three different meshes. The three graphs are not smooth; they show small sudden drops in pull-out force. Local examination of the model showed that these drops are caused by a combination of penetration of the slave surface by the master surface and by sequential stick-slip events. The magnitude of drops in force is the largest for the Coarse mesh. The magnitude in drops is similar for the Intermediate and Fine meshes. Therefore the Intermediate mesh will be used in this simulation.

5.1.3 Discussion of the results

For better understanding of the FEM model two analyses are done: an analysis with the linear elastic concrete model and an analysis with the CDP model.

Linear elastic concrete model

Fig. 5.4 and Fig. 5.5 show the pull-out behaviour for a hooked-end fibre calculated with the FEM model. In this diagram the most important mechanisms, determining the pull-out behaviour are shown separately: deformation of the fibre (green), (de)bonding (blue) and friction (red). These results are drawn schematically and only give a qualitative image of the three phenomena. The figures show that the pull-out behaviour is mainly determined by the deformation of the fibre. The (de)bonding behaviour has an almost negligible influence on the pull-out of a hooked fibre. Five points have been highlighted in the diagram. These points are discussed.
**Point 1 ($\delta = 0.1 \text{ mm}$)**

At this point the steel fibre experiences linear elastic behaviour. There is no plastic deformation. Tensile forces are starting to occur just above the inside of both corners due to bending of both corners of the steel fibre.
Fig. 5.6 - Max. principal stress [N/mm²] distribution at point 1

Point 2 (δ = 0.45 mm)
This point describes the maximum pull-out force. The fibre is already deforming plastically and large tensile forces appear at the bottom of the fibre due to bending.

Fig. 5.7 - Max. principal stress [N/mm²] and max. principal plastic strain [-] distribution at point 2

Point 3 (δ = 2.0 mm)
At this stage substantial decohesion has occurred, which leads to a significant lower pull-out load. Also, substantial plastic deformations have developed in the fibre (Fig. 5.8).
Point 4 (δ = 3,0 mm)
Between point 3 and point 4 the pull-out decreases slightly. This is explained by the fact that the angle between the end hook and the fibre itself remains equal. Only the length of the end hook changes. Therefore the deformation energy remains equal, which leads to an equal pull-out load.

Point 5 (δ = 5,2 mm)
At this point the fibre is fully deformed and the fibre is pulled out purely by friction. The frictional resistance is caused by the fact the fibre is not fully straight.
Fig. 5.10 - Max. principal stress [N/mm²] and max. principal plastic strain [-] distribution at point 5

CDP model

Fig. 5.11 shows what happens when the CDP model is used as a concrete material model in this 2D analysis. Substantial failure in the matrix occurs, which is not realistic for a three-dimensional situation. Therefore it is recommended to use a linear elastic material for qualitatively simulating the three-dimensional pull-out behaviour.

Fig. 5.11 - Max. principal plastic strain distribution and deformation in y-direction
CHAPTER 6:
EXPERIMENTAL RESEARCH

Experimental research has been done in the Pieter van Musschenbroek Laboratory (TU/e) on the pull-out behaviour of fibres as support for this research. Pull-out tests have been done with and without lateral loading. The experimental program and the test results are discussed in this chapter (Van den Bulck, 2013).

6.1 Experimental program

6.1.1 Purpose

The primary goals of the experimental program are to qualitatively verify the FEM model and to get an unambiguous image of the pull-out behaviour either with or without lateral loading. Also multiple properties have to be defined: traction-separation law (derive from pull-out test of straight fibre), the concrete properties (concrete compression tests) and the steel properties (steel fibre tensile test).

6.1.2 Test specimen

Each test specimen consists of a concrete matrix and a steel fibre (see Fig. 6.1b). For practical reasons the concrete matrix is casted in a standard cube mould of 150 x 150 x 150 mm³. Concrete type is C28/35 (cement type: blast-furnace-slag (BFS) CEM III). A standard mixture from Beamix (Beamix Constructie Beton 100) is used. Between 2,25-2,5 litre water is added to one bag of 25 kg. The water/cement ratio is unknown, because the ratio of the ingredients of the concrete mixture is unknown. The largest grain size is 12 mm.

Two types of steel fibres were used: the straight fibre and hooked-end fibre (Dramix 3D 65/60BG of Bekaeert). Both fibres have the same steel properties; the straight fibres were made by cutting off the hooks of the Dramix 3D fibre. The fibres have a diameter of 0,9 mm and a tensile strength of 1160 N/mm². The fibres are placed 30,0 mm inside the concrete matrix. To prevent inaccurate test results, the fibre needs to be exactly perpendicular to the concrete surface and exactly 30,0 mm inside the concrete matrix. To secure this, a wooden slat with a small hole is placed on the concrete mould (See
Fig. 6.1a). The part of the fibre above the slat is set to the correct length. Subsequently, the fibre is secured with a clip.

6.1.3 Test setup

The tensile test machine (capacity = 250 kN) has been used for the pull-out tests. Relative small tensile forces are expected (F < 1.0 kN), therefore the measuring equipment of the tensile machine is not used. The tensile force of the tensile machine is determined by a load cell (capacity = 2.0 kN) placed between the jack and the clamps. Two LVDT’s are used to determine the displacement of the fibre. The LVDT’s are attached on each side of the outer jaws and placed on top of the concrete matrix. Four different setups were used depending on the boundary conditions. Tests have been done with and without lateral loading and with either a straight fibre (SF) or a hooked fibre (HF). Fig. 6.2 shows all four test setups.

The pull-out of a straight fibre uses a wire connector to clamp the fibre (Fig. 6.2a,b). The wire connector is placed inside the outer jaws. By using a wire connector the pull-out force is exactly in line with the fibre. The wire connector has been tested for slip. The wire connector does not comply for the hooked fibre. Due to a higher pull-out force, the wire connector is unable to prevent slipping of the fibre. The hooked-end fibre uses clamping jaws to clamp the fibre (Fig. 6.2c,d). The drawback is the low mobility of the clamping jaws. Therefore it is much harder to set the pull-out force exactly in line with the fibre.

Different test setups have been used for the pull-out of a fibre with and without lateral loading. For the tests without lateral loading (Fig. 6.2a,c) the test specimen are supported downwards on the edges of the top surface of the concrete matrix. For the tests with lateral loading (Fig. 6.2b,d) a hydraulic jack is used. Due to friction between the specimen and the steel plates caused by the lateral loading it is not necessary to support the top surface of the concrete matrix.
6.1.4 Testing procedure

To prevent inaccurate testing results it is important to follow the procedure outlined below:

1. Place test specimen below the steel plate (no lateral force) or between the hydraulic jack and the angular profile (with lateral force).
2. Dropping the tensile jack to just above the concrete matrix.
3. Placing the fibre in the middle of the outer jaws.
4. Tightening the studs by hand of the angular profile (with lateral force).
5. Setting the LVDT’s.
6. Placing the wire connector (straight fibre) or the clamping jaws (hooked-end fibre).
7. No forces acting on the test specimen. Set LVDT’s and Load cell on 0.
8. Tightening the wire connector (straight fibre) or the clamping jaws (hooked-end fibre).
10. Setting the lateral force if necessary.
11. Start pull-out of the fibre
12. After full pull-out of the fibre, stop the measuring.
6.2 Experimental results

6.2.1 Straight fibre

Fig. 6.3 shows the results for a straight fibre without lateral loading. The results are shown for $\delta = 0$-3 mm. A large variety of maximum pull-out loads and initial stiffness is observed.

Fig. 6.3 - Pull-out behaviour for a straight fibre

Fig. 6.4 shows the results for a straight fibre subjected to a lateral normal stress of 0, 2.8 and 14 N/mm$^2$. All the results have been plotted in one diagram to get a good indication of the effect of the lateral normal stresses. An increase in maximum pull-out force and friction is witnessed. Also a decrease is witnessed in the stiffness of the adhesive bond.

Fig. 6.4 - Pull-out behaviour for a straight fibre with $\sigma_{lat} = 0, 2.8$ and 14 N/mm$^2$
6.2.2 Hooked-end fibre

Fig. 6.5 shows the results for the pull-out behaviour of a hooked-end fibre without lateral normal stress.

Fig. 6.6 and Fig. 6.7 show the pull-out behaviour of a hooked-end fibre with a lateral compressive stress of respectively 2,8 and 14 N/mm². When comparing the three figures, it can be observed that the maximum pull-out force has a similar average value of about 200 N, and that the amount of energy dissipated during pull-out (which is represented by the area under the $F_p - \delta$ curve) increases for a higher lateral compressive stress. This can be ascribed to the increase in frictional resistance as a result of a higher contact pressure between the fibre and the matrix.

![Fig. 6.5 - Pull-out behaviour for hooked-end fibre](image1)

![Fig. 6.6 - Pull-out behaviour for hooked-end fibre with $\sigma_{lat} = 2,8$ N/mm²](image2)
Fig. 6.7 - Pull-out behaviour for hooked-end fibre with $\sigma_{\text{ult}} = 14 \text{ N/mm}^2$
6.3 Conclusions

The results of the pull-out of the straight fibre show a high variance of maximum pull-out loads and initial stiffness, which indicates a high variance in adhesive bond. The variance in frictional force is acceptable and is most likely a combination of shrinkage of the matrix and imperfections of the fibre.

The results of the pull-out of a straight fibre subjected to lateral loading, show an increase in maximum pull-out force and an increase in frictional resistance is witnessed. Also a decrease is witnessed in stiffness of the adhesive bond. However, due to the limited amount of test is hard to draw any scientific conclusions.

A relative low variance of the results is witnessed for the hooked-end fibre. As shown in chapter 5, the pull-out behaviour of a hooked-end fibre is strongly determined by the deformation of the fibre. Since the variety in shape of a hooked-end fibre is relatively low, the variety in pull-out behaviour is also relatively low.

The results of the pull-out behaviour of the hooked fibre show higher variance as the lateral force is increased. When the lateral normal stress increases, the nonlinear failure behaviour of the concrete becomes more important. For the test series a relatively heterogenic concrete mixture has been used. Therefore the spread in pull-out results is most likely explained by to the heterogeneity of the concrete.

The test specimens had dried for 28 days in the laboratory. This means the influence of the drying shrinkage on the pull-out response is probably negligible. However evaporation of the water at the surface is possible, because the specimens were not sealed. This would mean drying shrinkage at the top of the specimen may occur, and therefore stresses between the fibre and the matrix may appear at the top of the specimen.

The w/c ratio of the concrete mixture and information about mineral admixtures are not known. When assuming a standard w/c ratio of 0.4, autogenous shrinkage will occur and will lead to stresses between fibre and matrix. However, the size of the initial stress caused by the autogenous shrinkage is unknown.
CHAPTER 7:
FEM vs. EXPERIMENTS

This chapter provides a qualitative comparison between the FEM results and the experimental results. The first part discusses the pull-out response of a straight fibre (standard pull-out test and influence of lateral forces on the pull-out behaviour) and the second part discusses the pull-out behaviour of the hooked-end fibre.

7.1 Straight fibre
7.1.1 Pull-out test

Fig. 7.1 shows the FEM result (Coarse mesh) and Fig. 7.2 shows the experimental results of a standard pull-out test of a straight fibre. When comparing both results, response curves are observed. Both results (FEM and experimental) show first a linear branch, subsequently an exponential softening branch and finally an approximately horizontal plateau.

![Fig. 7.1 - Pull-out behaviour of a straight fibre (FEM)](image-url)
7.1.2 The influence of lateral forces on the pull-out behaviour

Fig. 7.3 shows the FEM results and Fig. 7.4 shows the experimental results of a pull-out test subjected to a lateral force.
When comparing the results, two important differences can be observed. Firstly, the maximum pull-out force of the FEM model increases little when lateral stresses are added. As discussed in chapter 4.3, the (de)bonding energy in the FEM model does not depend on the compressive stress. However, the experiments show a significant increase in maximum pull-out force when lateral stresses are added, which is most likely caused by an increase in (de)bonding energy. This would imply that the (de)bonding energy in shear direction is affected by the magnitude of the compressive stress. The damage initiation and the damage evolution criterion of the FEM model should be adapted accordingly.

Secondly, in the test results the initial stiffness varies significantly for the different tests, which is not the case for the numerical simulations.

Because the variety in experimental results is large with little test results, it is hard to draw adequate conclusions.

### 7.2 Hooked-end fibre

#### 7.2.1 Pull-out test

Fig. 7.5 shows the FEM result (Intermediate mesh) and Fig. 7.6 shows the experimental results of the pull-out behaviour of a hooked-end fibre. When comparing the results, similar trends for the force-displacement response are observed. Differences in magnitude can be observed, which are mainly due to comparing a 2D-response with a 3D-response.
Fig. 7.5 - Pull-out behaviour of a hooked-end fibre (FEM)

Fig. 7.6 - Pull-out behaviour of a hooked-end fibre (experiments)
CHAPTER 8: CONCLUSIONS & RECOMMENDATIONS

In this thesis a FEM model is presented for modelling the pull-out behaviour of a single steel fibre. The main conclusions are presented below and recommendations for further research are given.

8.1 Conclusions

How to model the pull-out behaviour of a fibre using FEM?

To correctly model the pull-out behaviour, all the mechanisms which influence the pull-out of a fibre should be included in a model. In the fibre-matrix interface the pull-out behaviour is governed by adhesive bonding, debonding and friction. Furthermore, the failure and deformation behaviour of the steel fibre and the concrete matrix should be modelled with nonlinear material models.

To model the failure mechanisms in the fibre-matrix interface, a surface-based contact simulation is used. The surface-based contact formulation adequately accounts for surfaces that might get in contact during the simulation. The formulation uses a master surface (target surface) and a slave surface (contact surface) to model contact. The individual nodes on the slave surface try to contact the closest associated point on the master surface. Subsequently, there is a discretization of the contact area between the master surface and the slave surface. Finally, the contact interaction is described by the mechanical contact law. Three contact models were used: pressure-overclosure contact model (to minimize or prevent penetration of elements), surface-based cohesive behaviour (to model adhesive bond and debonding) and frictional behaviour (to model friction). Furthermore, the fibre uses plasticity to model the nonlinear behaviour of steel and the matrix uses CDP to model nonlinear behaviour of concrete. Shrinkage of the matrix is simulated using a thermal analogy.

This model leads to good qualitative results for a straight fibre and a hooked-end fibre when standard pull-out tests are simulated.

What is the influence of lateral normal stresses on the pull-out behaviour of a fibre?

For the straight fibre this leads to a positive change in pull-out behaviour; the maximum pull-out force increases and the frictional pull-out resistance increases. Both the FEM model and the experiments show these changes, however the sizes of increase differ between the FEM model and the experiments. The FEM model shows a minor increase in maximum pull-out force, because the (de)bonding energy is not dependent of the compressive stress. The experiments show a substantial variation in maximum pull-out force, but it is hard to draw scientific conclusions, because the number of tests is low.
Is the uniaxial material behaviour that follows from CUR 111 or RILEM TC 162-TDF 2002 valid for structural elements that will fail on a biaxial stress combination?

The material behaviour of SFRC is a combination of the behaviour of concrete and the pull-out behaviour of the fibre. Therefore, a change in pull-out behaviour is directly related to a change in the concrete material behaviour.

The maximum pull-out force and the residual force of a straight fibre increase under biaxial stress conditions, which mean the material behaviour of SFRC improves under biaxial stress condition. Therefore, it is safe to use the CUR 111 and RILEM TC 162-TDF 2002 in biaxial stress conditions.
8.2 Recommendations

More experimental research is necessary to gain further insight on the influence of lateral stress on the pull-out behaviour. It is recommended to use a more homogeneous concrete mixture to minimize the high variety of the experimental results. Furthermore, it is recommended to prevent a skewed shape of the fibre and measure the exact initial shape of the fibre. Also the introduction of the lateral force should be improved. It is recommended to use brushes or frictionless foils at the location where the lateral stress is introduced. Furthermore, shrinkage tests (Altoubat et Lange, 2001; Lura, 2003) should be performed where the initial stresses in the matrix should be measured.

Paragraph 7.1.2 discusses the fact that (de)bonding energy in shear direction increases when a compressive stress is added. This phenomenon is not implemented in the FEM model. This should be added to the FEM model. To correctly model this behaviour a user subroutine in Abaqus should be used to define surface interaction behaviour for the surface-based contact simulation (UIINTER).

In this thesis the hooked-end fibre was modelled in 2D. To gain quantitative results and to model the influence of lateral stress on the pull-out behaviour, the FEM model should be extended to 3D.

Modelling the material behaviour of SFRC is a combination of discrete cracking of concrete and the pull-out behaviour of steel fibres. The modelling of discrete cracking can be done with a cohesive zone model (Cerioni, 2008; Alfaro et al., 2010; Schoenmakers, 2013). The model used in this thesis work can be used for the pull-out behaviour. By combining both models SFRC can be modelled, see Fig. 8.1.

![Fig. 8.1 - Steps for modelling the material behaviour SFRC](image-url)


BRUGGELING, A.S.J., DE BRUIJN, W.A., Theorie en praktijk van het gewapend beton, deel 1, Den Bosch, 1986


DAVALOS, J., CHEN, Y., RAY, I., Numerical Modeling for the Interface Bond of Concrete and FRP bar, Earth & Space (2008), 1-11, 2008.


KING, S., RICHARDS, T., Solving Contact Problems with Abaqus (Lecture notes), Dessault Systèmes, UK, 2013.

KRABBENHØFT, K., Basic computational plasticity, TU Denmark, 2002.


MITTELSTÄDT, J., Numerische Untersuchungen zur Verbundwirkung bei betongefüllten Hohlprofilstützen, TU Hamburg, Germany, 2008.


Appendix A: Bruggeling’s tensile bar model

Bruggeling uses an incremental approach to calculate the transfer length and the crack widths of a reinforcement concrete bar loaded in tension. Fig. A.1 shows the principle of the model of Bruggeling. The model simplifies the axial loaded concrete bar to a pull-out problem of a reinforcement bar out of a concrete matrix. This pullout is similar to the pullout of a fibre out of a concrete matrix.

![Fig. A.1 - Principle of the Bruggeling’s tensile bar model (Bruggeling, 1986)](image)

The reinforced concrete bar is divided in finite segments with thickness $dx$. As input for the model a $\tau_{cs} - \delta$ relationship needs to be determined, which describes the shear-displacement behaviour. At the left side of the model the strains are equal, the relative displacement is zero and the shear stress is zero. Calculation starts on the left side of the model at the first segment. At this point the steel and the concrete strain are equal. Fig. A.2 shows the flow chart for the Bruggeling’s tensile model. In this calculation the concrete deformation is neglected, which means step 3 lapses.
The first step is to calculate the shear stress ($\tau_{cs}$) by using the $\tau_{cs} - \delta_s$ relationship. The shear stress and the slip are often related to each other by the shear stiffness ($k$). The benefit of this method is that any mathematical function that describes the $\tau_{cs} - \delta_s$ relationship can be used. Only drawback is that the shear stress cannot be zero if the slip is zero, because this will lead to an endless loop in the model. So an initial value is obligatory.

Second step uses horizontal equilibrium of one segment of the fibre (Fig. A.3) to calculate the increment of the steel stress ($d\sigma_{sx}$).

$$d\sigma_{sx} = \frac{V}{A_s} \tau_{cs} \, dx \quad (3.11)$$

The final step is to calculate the relative displacement at the end of the first segment. In this case the relative displacement equals the steel displacement, because the concrete deformation is neglected. The steel stress is known, which means that the elongation over this segment and the displacement at the end of the segment can be calculated.

The first step can be repeated on the second segment, using the relative displacement at the end of the first segment. These steps are repeated until external equilibrium is found for the whole fibre:

$$(\sigma_{sx} + \sum d\sigma_{sx}) \cdot A_s = N_s \quad (3.12)$$

The transfer length is $l_s = n \cdot dx$, with $n$ is the number of segments (see Fig. A.1).
Fig. A.4 shows the local stresses along the fibre for different starting values calculated with the tensile bar model. Parameters are $k = 1000 \text{ N/mm}^2$ and $\sigma_p = 100 \text{ N/mm}^2$. The initial value is an important parameter. The initial value of the $\tau_{cs} - \delta_x$ relationship determines the starting gradient of the graph (see Fig. A.4) and therefore highly influences the transfer length. A high starting value leads to a low transfer length and vice versa.

![Graph of normal stresses along the fibre with different starting values](image1)

**Fig. A.4 - Normal stresses along the fibre with different starting values**

In order to calculate the stresses in a fibre of 30 mm, a starting value is chosen, which leads to a transfer length of 30 mm. Fig. A.5 shows the local stresses along the fibre ($l_f = 30 \text{ mm}$) varied in shear stiffness ($k = 100, 1000$ and $10000 \text{ N/mm}^2$) calculated with the FDM model. The fibre is loaded with $\sigma_p = 100 \text{ N/mm}^2$.

![Graph of local stress distribution in a fibre of 30 mm](image2)

**Fig. A.5 - Local stress distribution in a fibre of 30 mm**
Appendix B: Extension of Bruggeling’s tensile bar model

Bruggeling’s model has the possibility to use any $\tau_{cs} - \delta_x$ relationship as input. Therefore a traction-separation law can be used. In order to model the pull-out behaviour correctly with the tensile bar model of Bruggeling, some modifications are needed. The pull-out of a fibre is displacement controlled. Bruggeling’s model is force controlled and the relative displacement is assumed zero at the first segment of the fibre. By controlling the relative displacement at the first segment instead of controlling the pull-out force, the model becomes displacement controlled. The first step is setting an initial displacement at the first segment. In this case no starting value is needed for the $\tau_{cs} - \delta_x$ relationship.

The next step is to follow the standard flowchart of Bruggeling’s model, but in this case looping until the fibre length has been reached instead of horizontal equilibrium. After these steps the relative displacement and the normal stress at the last segment are known. This data is the input for the pull-out diagram. This total process is looped until the normal stresses are zero in the last segment, which lead to a pull-out behaviour. Fig. B.1 shows the flow chart for the modified model of Bruggeling.

![Fig. B.1 - Flow chart modified Bruggeling’s tensile bar model](image)

A calculation of the pull-out behaviour is performed using two different $\tau - \delta$ relationships, which are described by a traction-separation law (Fig. B.2). The results are plotted in Fig. B.3.
Fig. B.2 - Traction-separation law

Fig. B.3 - Pull-out behaviour
Appendix C: Input files Abaqus

C1 Straight fibre

C1.1 Pull-out test

*Heading
** Job name: Job1-axi-surf-shr-coarsest Model name: Model-1_axi_surface-shrink-coarsest
** Generated by: Abaqus/CAE 6.11-2
*Preprint, echo=NO, model=NO, history=NO, contact=NO
**
** PARTS
**
*Part, name=concrete
*Node
  1, 30., 0.
  2, 0.45, 0.
  3, 0., 0.
  .................................
  .................................
  320, 27., 0.
*Element, type=CAX4
  1, 1, 9, 95, 56
  2, 9, 10, 96, 95
  3, 10, 11, 97, 96
  ..................................
  ..................................
  282, 320, 65, 7, 66
*Nset, nset=_PickedSet3, internal, generate
  1, 320, 1
*Elset, elset=_PickedSet3, internal, generate
  1, 282, 1
*Nset, nset=_PickedSet32, internal, generate
  1, 320, 1
*Elset, elset=_PickedSet32, internal, generate
  1, 282, 1
*Elset, elset=_Conc_surf_S4, internal, generate
  73, 262, 21
*Surface, type=ELEMENT, name=Conc_surf_S4, S4
*Orientation, name=Ori-1
  1., 0., 0., 1., 0.
  3, 0.
** Section: conc_sec
*Solid Section, elset=_PickedSet3, orientation=Ori-1, material=Concrete
,* End Part
**
*Part, name=fibre
*Node
  1, 0., 0.
  2, 0.15 0.
  3, 0.3, 0.
  ..................................
  ..................................
  176, 0.45, 32.
*Element, type=CAX4
  1, 1, 2, 6, 5
  2, 2, 3, 7, 6
  3, 3, 4, 8, 7
  ..................................

84 |
129, 171, 172, 176, 175
*Nset, nset=_PickedSet2, internal, generate
1, 176, 1
*Elset, elset=_PickedSet2, internal, generate
1, 129, 1
*Elset, elset=_fibre_surf_S2, internal, generate
3, 129, 3
*Surface, type=ELEMENT, name=fibre_surf
_fibre_surf_S2, S2
** Section: steel_sec
*Solid Section, elset=_PickedSet2, material=Steel

** ASSEMBLY
**
*Assembly, name=Assembly
**
*Instance, name=concrete-1, part=concrete
*End Instance
**
*Instance, name=fibre-1, part=fibre
*End Instance
**
*Nset, nset=_PickedSet9, internal, instance=fibre-1, generate
173, 176, 1
*Elset, elset=_PickedSet9, internal, instance=fibre-1, generate
127, 129, 1
*Nset, nset=_PickedSet16, internal, instance=concrete-1
4, 5, 6, 33, 34, 35, 36, 37, 38, 39, 40, 41, 42, 43, 44, 45
46, 47, 48, 49, 50, 51, 52, 53, 54
*Elset, elset=_PickedSet16, internal, instance=concrete-1, generate
49, 72, 1
*Nset, nset=_PickedSet17, internal, instance=concrete-1, generate
1, 320, 1
*Elset, elset=_PickedSet17, internal, instance=concrete-1, generate
1, 282, 1
*End Assembly
**
** MATERIALS
**
*Material, name=Concrete
*Elastic
33000., 0.2
*Expansion
0.00015,
*Material, name=Steel
*Elastic
210000., 0.3
**
** INTERACTION PROPERTIES
**
*Surface Interaction, name=cohesive_prop
1.,
*Friction, elastic slip=0.01
0.04,
*Surface Behavior, penalty
*Cohesive Behavior, eligibility=ORIGINAL CONTACTS
*Damage Initiation, criterion=QUADS
0.7, 0.7, 0.
*Damage Evolution, type=DISPLACEMENT, softening=EXPONENTIAL, mixed mode behavior=TABULAR, mode mix ratio=TRACTION
0.5, 6.0, 0.0
0.5, 6.1, 0.0.
*Damage Stabilization
0.
**
** BOUNDARY CONDITIONS
**
** Name: BC-1 Type: Displacement/Rotation
*Boundary
_PickedSet16, 2, 2
**
** PREDEFINED FIELDS
**
** Name: Predefined Field-1 Type: Temperature
*Initial Conditions, type=TEMPERATURE
_PickedSet17, 0.
**
** INTERACTIONS
**
** Interaction: cohesive
*Contact Pair, interaction=cohesive_prop
fibre-1.fibre_surf, concrete-1.Conc_surf
**
** STEP: shrinkage
**
*Step, name=shrinkage
*Static
1., 1., 1e-05, 1.
**
** PREDEFINED FIELDS
**
** Name: Predefined Field-1 Type: Temperature
*Temperature
_PickedSet17, -1.
**
** OUTPUT REQUESTS
**
*Restart, write, frequency=0
*Output, field, frequency=0
*Output, history, frequency=0
*End Step
**
** STEP: Displ
**
*Step, name=Displ, inc=100000
*Static
0.005, 1., 1e-20, 0.005
**
** BOUNDARY CONDITIONS
**
** Name: displ Type: Displacement/Rotation
*Boundary
_PickedSet9, 2, 2, 0.5
** OUTPUT REQUESTS
**
*Restart, write, frequency=0
**
** FIELD OUTPUT: F-Output-1
**
*Output, field, variable=PRESELECT
**
** HISTORY OUTPUT: H-Output-1
**
*Output, history, variable=PRESELECT
*End Step
**  
** STEP: Displ 2
**
*Step, name="Displ 2", inc=10000
*Static
  0.005, 1., 1e-10, 0.005
**
** BOUNDARY CONDITIONS
**
** Name: displ Type: Displacement/Rotation
*Boundary
  PickedSet9, 2, 2, 4.
**
** OUTPUT REQUESTS
**
*Restart, write, frequency=0
**
** FIELD OUTPUT: F-Output-1
**
*Output, field, variable=PRESELECT
**
** HISTORY OUTPUT: H-Output-1
**
*Output, history, variable=PRESELECT
*End Step
C1.2 Influence of imperfection on the pull-out behaviour

C1.2.1 2D elastic material model

*Heading
** Job name: Imperfection_Mesh1-both Model name: Model-3-recht-imperfection-Mesh1-both
** Generated by: Abaqus/CAE 6.11-2
*Preprint, echo=NO, model=NO, history=NO, contact=NO
**
** PARTS
**
*Part, name=Fibre
*Node
  1,  25.4500008,  42.
  2,  25.2999992,  42.
  3,  25.1499996,  42.
  .......................................................... 588,  24.5499992,  10.
*Element, type=CPS4
  1,  1,  2,  9,  8
  2,  2,  3, 10,  9
  3,  3,  4, 11, 10
  .......................................................... 498, 580, 581, 588, 587
*Nset, nset=_PickedSet4, internal, generate
  1,  588,      1
*Elset, elset=_PickedSet4, internal, generate
  1,  498,      1
*Elset, elset=_Surf-fibre_left_S2, internal, generate
  6,  498,      6
*Surface, type=ELEMENT, name=Surf-fibre_left
  _Surf-fibre_left_S2, S2
*Elset, elset=_Surf-fibre_right_S4, internal, generate
  1,  493,      6
*Surface, type=ELEMENT, name=Surf-fibre_right
  _Surf-fibre_right_S4, S4
** Section: Section_fibre
*Solid Section, elset=_PickedSet4, material=Steel

*Part, name=Matrix
*Node
  1,          50.,  34.
  2,  25.4500008,  34.
  3,  25.4500008,  10.
  .......................................................... 1236, 47.099205,  39.7000008
*Element, type=CPS4
  1,  1, 18, 275,  87
  2, 18, 19, 276, 275
  3, 19, 20, 277, 276
  .......................................................... 1140, 1236, 247,  16, 248
*Nset, nset=_PickedSet4, internal, generate
  1, 1236,      1
*Elset, elset=_PickedSet4, internal, generate
*Elset, elset=_Surf-Matrix_left_S4, internal, generate 337, 652, 21
*Elset, elset=_Surf-Matrix_left_S2, internal, generate 819, 966, 21
*Surface, type=ELEMENT, name=Surf-Matrix_left_S4, S4_Surf-Matrix_left_S2, S2
*Elset, elset=_Surf-Matrix_right_S2, internal, generate 21, 336, 21
*Elset, elset=_Surf-Matrix_right_S4, internal, generate 973, 1120, 21
*Surface, type=ELEMENT, name=Surf-Matrix_right_S2, S2_Surf-Matrix_right_S4, S4
** Section: Section-concrete
*Solid Section, elset=_PickedSet4, material=Concrete
*End Part
**
** ASSEMBLY
**
*Assembly, name=Assembly
**
*Instance, name=Fibre-1, part=Fibre
*End Instance
**
*Instance, name=Matrix-1, part=Matrix
*End Instance
**
*Nset, nset=_PickedSet11, internal, instance=Matrix-1
  9, 10, 11, 12, 15, 160, 161, 162, 163, 164, 165, 166, 167, 168, 169, 170
  171, 172, 173, 174, 175, 176, 177, 178, 179, 186, 187, 188, 189, 190, 191, 192
  193, 194, 195, 196, 197, 198, 199, 200, 201, 202, 203, 204, 205
*Elset, elset=_PickedSet11, internal, instance=Matrix-1
  675, 678, 681, 684, 687, 690, 693, 696, 699, 702, 705, 708, 711, 714, 717, 720
  723, 726, 729, 732, 735, 736, 739, 742, 745, 748, 751, 754, 757, 760, 763, 766
  769, 772, 775, 778, 781, 784, 787, 790, 793, 796, 969, 972
*Nset, nset=_PickedSet13, internal, instance=Fibre-1
  4,
*Elset, elset=_PickedSet15, internal, instance=Matrix-1
  1, 4, 6, 7, 10, 12, 14, 16, 73, 74, 75, 76, 77, 78, 79, 80
  117, 118, 119, 120, 121, 122, 180, 181, 184, 185, 233, 234, 235, 236, 237, 238
  239, 241, 242, 243, 244, 245, 246, 247
*Elset, elset=_PickedSet15, internal, instance=Matrix-1
  1, 22, 43, 64, 85, 106, 127, 148, 169, 190, 211, 232, 253, 274, 295, 316
  357, 378, 399, 420, 441, 462, 483, 504, 525, 546, 567, 588, 609, 630, 651, 672
  733, 734, 735, 796, 797, 798, 799, 820, 841, 862, 883, 904, 925, 946, 993, 1014
  1035, 1056, 1077, 1098, 1119, 1140
*End Assembly
**
** MATERIALS
**
*Material, name=Concrete
*Elastic 33000. 0.2
*Material, name=Steel
*Elastic
** INTERACTION PROPERTIES **

*Surface Interaction, name=_Int-left-Prop*
 1.,
*Friction, elastic slip=0.01  0.04,*
*Surface Behavior, penalty*
*Cohesive Behavior, eligibility=ORIGINAL CONTACTS*
 11.,11.,11.*
*Damage Initiation, criterion=QUADS*
 0.7, 0.7, 0.7*
*Damage Evolution, type=DISPLACEMENT, softening=EXPONENTIAL, mixed mode behavior=TABULAR*
 0.5,6.,0.,0.
 0.5,6.,1.,0.*
*Damage Stabilization*
 0.*
*Surface Interaction, name="left properties"
 1.,
*Friction, elastic slip=0.01  0.04,*
*Surface Behavior, penalty*
*Cohesive Behavior, eligibility=ORIGINAL CONTACTS*
 11.,11.,11.*
*Damage Initiation, criterion=QUADS*
 0.7, 0.7, 0.7*
*Damage Evolution, type=DISPLACEMENT, softening=EXPONENTIAL, mixed mode behavior=TABULAR*
 0.5,6.,0.,0.
 0.5,6.,1.,0.*
*Damage Stabilization*
 0.*
*Surface Interaction, name="right properties"
 1.,
*Friction, elastic slip=0.01  0.04,*
*Surface Behavior, penalty*
*Cohesive Behavior, eligibility=ORIGINAL CONTACTS*
 11.,11.,11.*
*Damage Initiation, criterion=MAXS*
 0.7, 0.7, 0.7*
*Damage Evolution, type=DISPLACEMENT, softening=EXPONENTIAL, mixed mode behavior=TABULAR*
 0.5,6.,0.,0.
 0.5,6.,1.,0.*
*Damage Stabilization*
 0.

** BOUNDARY CONDITIONS **

** Name: BC-2 Type: Displacement/Rotation
*Boundary
_PickedSet11, 2, 2
** Name: BC-3 Type: Displacement/Rotation
*Boundary
_PickedSet15, 1, 1

** INTERACTIONS
** Interaction: Int-left
*Contact Pair, interaction=_Int-left-Prop, adjust=0.0
** Interaction: Int-right
*Contact Pair, interaction="right properties", adjust=0.0
Matrix-1.Surf-Matrix_right, Fibre-1.Surf-fibre_right
**

** STEP: Displ
**
*Step, name=Displ, inc=100000
*Static
0.001, 1., 1e-20, 0.01
**

** BOUNDARY CONDITIONS
**
** Name: displ Type: Displacement/Rotation
*Boundary
_PickedSet13, 1, 1, 0.20978
_PickedSet13, 2, 2, 3.
**

** INTERACTIONS
**
** Interaction: Int-left
**
*Change Friction, interaction=_Int-left-Prop
*Friction, elastic slip=0.01
0.04,
**

** CONTROLS
**
*Controls, reset
*Controls, analysis= discontinuous
**

** OUTPUT REQUESTS
**
*Restart, write, frequency=0
**

** FIELD OUTPUT: F-Output-1
**
*Output, field
*Node Output
CF, RF, U
*Element Output, directions=YES
LE, PE, PEEQ, PEMAG, S, SE
*Contact Output
CDISP, CSTRESS
**

** HISTORY OUTPUT: H-Output-1
**
*Output, history, variable=PRESELECT
*End Step
C1.2.2 2D CDP model

*Heading
** Job name: Imperfection_Mesh1-CDP Model name: Model-3-recht-imperfection-Mesh1-CDP
** Generated by: Abaqus/CAE 6.11-2
*Preprint, echo=NO, model=NO, history=NO, contact=NO
**
** PARTS
**
*Part, name=Fibre
*Node
  1, 25.4500008, 42.
  2, 25.2999992, 42.
  3, 25.1499996, 42.
  ................................
  ................................
  588, 24.5499992, 10.
*Element, type=CPS4
  1, 1, 2, 9, 8
  2, 2, 3, 10, 9
  3, 3, 4, 11, 10
  ................................
  ................................
  498, 580, 581, 588, 587
*Nset, nset=_PickedSet4, internal, generate
  1, 588, 1
*Elset, elset=_PickedSet4, internal, generate
  1, 498, 1
*Elset, elset=_Surf-fibre_left_S2, internal, generate
  6, 498, 6
*Surface, type=ELEMENT, name=Surf-fibre_left_S2, S2
*Elset, elset=_Surf-fibre_right_S4, internal, generate
  1, 493, 6
*Surface, type=ELEMENT, name=Surf-fibre_right_S4, S4
** Section: Section_fibre
*Solid Section, elset=_PickedSet4, material=Steel
*
*End Part
**
*Part, name=Matrix
*Node
  1, 50., 34.
  2, 25.4500008, 34.
  3, 25.4500008, 10.
  ................................
  ................................
  972, 47.099205, 39.7000008
*Element, type=CPS4
  1, 1, 18, 251, 71
  2, 18, 19, 252, 251
  3, 19, 20, 253, 252
  ................................
  ................................
  888, 972, 221, 16, 222
*Nset, nset=_PickedSet4, internal, generate
  1, 972, 1
*Elset, elset=_PickedSet4, internal, generate
  1, 888, 1
*Elset, elset=_Surf-Matrix_left_S4, internal, generate

92 |
169, 316, 21
*Elset, elset=_Surf-Matrix_left_S2, internal, generate 483, 672, 21
*Surface, type=ELEMENT, name=Surf-Matrix_left
_Surf-Matrix_left_S4, S4
_Surf-Matrix_left_S2, S2
*Elset, elset=_Surf-Matrix_right_S2, internal, generate 21, 168, 21
*Elset, elset=_Surf-Matrix_right_S4, internal, generate 679, 868, 21
*Surface, type=ELEMENT, name=Surf
_Matrix_right_S2, S2
_Matrix_right_S4, S4
** Section: Section-concrete
*Solid Section, elset=_PickedSet4, material=Concrete

** End Part
**
** ASSEMBLY
**
*Assembly, name=Assembly
**
*Instance, name=Fibre-1, part=Fibre
*End Instance
**
*Instance, name=Matrix-1, part=Matrix
*End Instance
**
*Nset, nset=_PickedSet11, internal, instance=Matrix-1
  9, 10, 11, 12, 15, 128, 129, 130, 131, 132, 133, 134, 135, 136, 137, 138
  139, 140, 141, 142, 143, 144, 145, 146, 147, 154, 155, 156, 157, 158, 159, 160
  161, 162, 163, 164, 165, 166, 167, 168, 169, 170, 171, 172, 173
*Elset, elset=_PickedSet11, internal, instance=Matrix-1
  387, 390, 393, 396, 399, 400, 403, 406, 409, 412, 415, 418, 421, 424, 427, 430
  433, 436, 439, 442, 445, 448, 451, 454, 457, 460, 675, 678
*Nset, nset=_PickedSet13, internal, instance=Fibre-1
  4,
*Nset, nset=_PickedSet15, internal, instance=Matrix-1
  1, 4, 6, 7, 10, 12, 14, 16, 65, 66, 67, 68, 69, 70, 71, 92
  93, 94, 95, 96, 97, 98, 148, 149, 152, 153, 203, 204, 205, 206, 207, 208
  209, 210, 211, 213, 214, 215, 216, 217, 218, 219, 220, 221
*Elset, elset=_PickedSet15, internal, instance=Matrix-1
  1, 12, 43, 64, 85, 106, 127, 148, 189, 210, 231, 252, 273, 294, 315, 336
  397, 398, 399, 460, 461, 462, 463, 484, 505, 526, 547, 568, 589, 610, 631, 652
  699, 720, 741, 762, 783, 804, 825, 846, 867, 888
*End Assembly
**
** MATERIALS
**
*Material, name=Concrete
*Elastic
33000., 0.2
*Concrete Damaged Plasticity
38., 0.1, 1.16, 0.666667, 0.0001
*Concrete Compression Hardening
8.46, 0.
22.26, 7.2e-05
27.89, 0.000146
37., 0.000397
39.95, 0.000579
41.62, 0.0008
41.96, 0.00099
41.61, 0.0012
39.74, 0.00154
31.26, 0.00234
6.26, 0.00436
3.76, 0.0047
2.51, 0.005
1.26, 0.0055
*Concrete Tension Stiffening
3.4, 0.
2., 0.00015
0.8, 0.0005
0.4, 0.0012
0.2, 0.002
*Material, name=Steel
Elastic
210000., 0.3
**
** INTERACTION PROPERTIES
**
*Surface Interaction, name=_Int-left-Prop
1.,
*Friction, elastic slip=0.01
0.04,
*Surface Behavior, penalty
*Cohesive Behavior, eligibility=ORIGINAL CONTACTS
11.,11.,11.
*Damage Initiation, criterion=QUADS
0.7, 0.7, 0.7
*Damage Evolution, type=DISPLACEMENT, softening=EXPONENTIAL, mixed mode behavior=TABULAR
0.5,6.,0.,0.
0.5,6.,1.,0.
*Damage Stabilization
0.
*Surface Interaction, name="left properties"
1.,
*Friction, elastic slip=0.01
0.04,
*Surface Behavior, penalty
*Cohesive Behavior, eligibility=ORIGINAL CONTACTS
11.,11.,11.
*Damage Initiation, criterion=QUADS
0.7, 0.7, 0.7
*Damage Evolution, type=DISPLACEMENT, softening=EXPONENTIAL, mixed mode behavior=TABULAR
0.5,6.,0.,0.
0.5,6.,1.,0.
*Damage Stabilization
0.
*Surface Interaction, name="right properties"
1.,
*Friction, elastic slip=0.01
0.04,
*Surface Behavior, penalty
*Cohesive Behavior, eligibility=ORIGINAL CONTACTS
11.,11.,11.
*Damage Initiation, criterion=MAXS
0.7, 0.7, 0.7
*Damage Evolution, type=DISPLACEMENT, softening=EXPONENTIAL, mixed mode
behavior=TABULAR
0.5,6.,0..0.
0.5,6.,1..0.
*Damage Stabilization 0.
**
** BOUNDARY CONDITIONS
**
** Name: BC-2 Type: Displacement/Rotation
*Boundary
_PickedSet11, 2, 2
** Name: BC-3 Type: Displacement/Rotation
*Boundary
_PickedSet15, 1, 1
**
** INTERACTIONS
**
** Interaction: Int-left
*Contact Pair, interaction=_Int-left-Prop, adjust=0.0
**
** Interaction: Int-right
*Contact Pair, interaction="right properties", adjust=0.0
Matrix-1.Surf-Matrix_right, Fibre-1.Surf-fibre_right
**
** STEP: Displ
**
*Step, name=Displ, inc=100000
*Static
0.001, 1., 1e-20, 0.05
**
** BOUNDARY CONDITIONS
**
** Name: displ Type: Displacement/Rotation
*Boundary
_PickedSet13, 1, 1, 0.020978
_PickedSet13, 2, 2, 0.3
**
** INTERACTIONS
**
** Interaction: Int-left
*Change Friction, interaction=_Int-left-Prop
*Friction, elastic slip=0.01
0.04,
**
** CONTROLS
**
*Controls, reset
*Controls, analysis=discontinuous
**
** OUTPUT REQUESTS
**
*Restart, write, frequency=0
**
** FIELD OUTPUT: F-Output-1
**
*Output, field, variable=PRESELECT
C1.2.3 3D elastic material model

** HISTORY OUTPUT: H-Output-1
**
*Output, history, variable=PRESELECT
*End Step
** ---------------------------------------------------------------
**
** STEP: Displ 2
**
*Step, name="Displ 2", inc=1000000
*Static
0.01, 1., 1e-20, 0.01
**
** BOUNDARY CONDITIONS
**
** Name: displ Type: Displacement/Rotation
*Boundary
_PickedSet13, 1, 1, 0.20978
_PickedSet13, 2, 2, 3.
**
** OUTPUT REQUESTS
**
*Restart, write, frequency=0
**
** FIELD OUTPUT: F-Output-1
**
*Output, field, variable=PRESELECT, frequency=10
**
** HISTORY OUTPUT: H-Output-1
**
*Output, history, variable=PRESELECT, frequency=10
*End Step

**
** PARTS
**
*Part, name=Fibre
*Node
  1, 0., 0.449999988, 24.
  2, 0., 0., 24.
  3, 0., 0., 30.
    ...........................................
  768, -0.142053023, 0.141492426, 3.
*Element, type=C3D8
  1, 272, 38, 215, 566, 17, 1, 37, 177
  2, 566, 215, 216, 567, 177, 37, 36, 178
  3, 567, 216, 217, 568, 178, 36, 35, 179
    ..........................................
  496, 531, 166, 542, 749, 540, 167, 544, 762
*Nset, nset=_PickedSet4, internal, generate
  1, 768, 1
*Elset, elset=_PickedSet4, internal, generate
1, 496, 1
*Elset, elset=_Fibre_S3, internal
---------------------------------------------------------------------
491, 492, 493, 494, 495, 496
*Elset, elset=_Fibre_S4, internal
1, 2, 3, 4, 5, 6, 7, 8, 9, 10, 11, 12, 13, 14, 15, 16
---------------------------------------------------------------------
461, 462, 463, 464
*Elset, elset=_Fibre_S2, internal
301, 302, 303, 304, 305, 306, 307, 308, 309, 310, 311, 312, 313, 314, 315, 316
317, 318, 319, 320, 401, 402, 403, 404, 405, 406, 407, 408, 431, 432
*Surface, type=ELEMENT, name=Fibre
 _Fibre_S3, S3
 _Fibre_S4, S4
 _Fibre_S2, S2
** Section: Fibre_sec
*Solid Section, elset=_PickedSet4, material=Steel
.
*End Part
**
*Part, name=Matrix
*Node
  1, 0.449999988, 0., 0.
  2, 0., 0.449999988, 0.
  3, 0., 0., 0.
---------------------------------------------------------------------
16482, -12.5173759, 10.9254351, 29.8999996
*Element, type=C3D8
  1, 38, 647, 32, 1, 1359, 5212, 5207, 1358
  2, 647, 645, 33, 32, 5212, 5210, 5208, 5207
  3, 38, 37, 648, 647, 1359, 1360, 5213, 5212
---------------------------------------------------------------------
14656, 16383, 16382, 16384, 16481, 4657, 4656, 4658, 4755
*Nset, nset=_PickedSet3_#4, internal, generate
  1, 16482, 1
*Elset, elset=_PickedSet3_#4, internal, generate
  1, 14656, 1
*Elset, elset=_Matrix_S3, internal
1053, 1056, 1383, 1386, 1713, 1716, 2043, 2046, 2373, 2376, 2703, 2706, 3033, 3036,
---------------------------------------------------------------------
12691, 12694, 13025, 13028, 13359, 13362, 13693, 13696, 14027, 14030, 14361, 14364
*Elset, elset=_Matrix_S4, internal
1058, 1060, 1388, 1390, 1718, 1720, 2048, 2050, 2378, 2380, 2708, 2710, 3038, 3040,
---------------------------------------------------------------------
12696, 12698, 13030, 13032, 13364, 13366, 13698, 13700, 14032, 14034, 14366, 14368
*Surface, type=ELEMENT, name=Matrix
 _Matrix_S3, S3
 _Matrix_S4, S4
** Section: Matrix_sec
*Solid Section, elset=_PickedSet3_#4, material=Concrete
.
*End Part
**
** ASSEMBLY
**
*Assembly, name=Assembly
*Instance, name=Matrix0-1, part=Matrix
*End Instance
**
*Instance, name=Fibre0-1, part=Fibre
*End Instance
**
*Nset, nset=_PickedSet27, internal, instance=Matrix0-1
  8, 9, 140, 141, 142, 143, 144, 145, 146, 147, 148, 149, 150, 151, 152, 153
  154, 155, 156, 157, 158, 159, 160, 161, 162, 163, 164, 165, 166
*Elset, elset=_PickedSet28, internal, instance=Matrix0-1
  681, 682, 683, 688, 692, 695, 705, 706, 709, 710, 712, 713, 714, 715, 718, 720
  722, 724, 730, 731, 732, 733, 736, 739, 741, 926, 952, 992, 7973, 7975, 7976, 7980
  8015, 8019, 8023, 8025, 8026, 8054, 8055, 8056, 8062, 8065, 8066, 8069, 8070, 8071, 8073
  8074, 8076, 8078, 8080, 8085, 8162, 8187
*Nset, nset=_PickedSet28, internal, instance=Matrix0-1
  1, 3, 4, 7, 8, 11, 12, 17, 18, 21, 23, 24, 27, 28, 29, 31
  37, 38, 81, 82, 83, 84, 85, 86, 87, 88, 89, 90, 91, 92, 93, 94
  5011, 5012, 5013, 5014, 5015, 5016, 5017, 5018, 5019, 5020, 5021, 5022, 5023, 5024, 5025, 5026
*Elset, elset=_PickedSet28, internal, instance=Matrix0-1
  1, 3, 6, 47, 49, 70, 71, 75, 76, 88, 89, 90, 96, 97, 99, 100
  14352, 14353, 14354, 14356, 14357, 14358, 14359, 14360, 14362, 14363, 14368, 14559, 14588, 14629
*Nset, nset=_PickedSet30, internal, instance=Matrix0-1
  7, 8, 9, 10, 24, 25, 113, 114, 115, 116, 117, 118, 119, 120, 121, 122
  123, 124, 125, 126, 127, 128, 129, 130, 131, 132, 133, 134, 135, 136, 137, 138
  3747, 3748, 3749, 3750, 3751, 3752, 3753, 3754, 3755
*Elset, elset=_PickedSet30, internal, instance=Matrix0-1
  693, 694, 695, 696, 697, 698, 699, 700, 701, 702, 703, 704, 705, 706, 707, 708
  8287, 8288, 8289, 8290, 8291, 8292, 8293, 8294, 8295, 8296, 8297, 8298, 8299, 8300, 8301, 8302
  8303, 8304, 8305, 8306, 8307, 8308, 8309, 8310
*Nset, nset=_PickedSet32, internal, instance=Fibre0-1
  9
*Nset, nset=_PickedSet33, internal, instance=Fibre0-1
  3, 6, 8, 9, 11, 15, 64, 65, 93, 94, 119, 120, 123, 124, 128, 129
  158, 159, 163, 164, 431, 432, 433, 434, 498, 499, 500, 501
*Elset, elset=_PickedSet33, internal, instance=Fibre0-1
  430, 432
*End Assembly
**
** MATERIALS
**
*Material, name=Concrete
*Elastic
  33000., 0.2
*Material, name=Steel
*Elastic
210000., 0.3
**
** INTERACTION PROPERTIES
**
*Surface Interaction, name=IntProp-1
1.,
*Friction, elastic slip=0.01
0.04,
*Surface Behavior, penalty
*Cohesive Behavior, eligibility=ORIGINAL CONTACTS
11.,11.,11.
*Damage Initiation, criterion=QUADS
0.7, 0.7, 0.7
*Damage Evolution, type=DISPLACEMENT, softening=EXPONENTIAL, mixed mode
behavior=TABULAR
0.5,6.,0.,0.
0.5,6.,1.,0.
*Damage Stabilization
0.
**
** BOUNDARY CONDITIONS
**
** Name: BC-1 Type: Displacement/Rotation
*Boundary
_PickedSet27, 1, 1
** Name: BC-2 Type: Displacement/Rotation
*Boundary
_PickedSet28, 2, 2
** Name: BC-2F Type: Displacement/Rotation
*Boundary
_PickedSet33, 2, 2
** Name: BC-3 Type: Displacement/Rotation
*Boundary
_PickedSet30, 3, 3
**
** INTERACTIONS
**
** Interaction: Int-1
*Contact Pair, interaction=IntProp-1, adjust=0.0
Matrix0-1.Matrix, Fibre0-1.Fibre
** ---------------------------------------------------------------
**
** STEP: Displ
**
*Step, name=Displ, inc=10000
*Static
0.001, 1., 1e-15, 0.01
**
** BOUNDARY CONDITIONS
**
** Name: Displ Type: Displacement/Rotation
*Boundary
_PickedSet32, 1, 1, 0.20978
_PickedSet32, 3, 3, 3.
**
** CONTROLS
**
*Controls, reset
*Controls, analysis=discontinuous
*Controls, parameters=time incrementation
  ; ; ; ; 20 ,
** OUTPUT REQUESTS
**
*Restart, write, frequency=0
**
** FIELD OUTPUT: F-Output-1
**
*Output, field, variable=PRESELECT
**
** HISTORY OUTPUT: H-Output-1
**
*Output, history, variable=PRESELECT
*End Step

C1.2.4 3D CDP model

*Heading
** Job name: StraightFibre_imperf_noshr_cdp4 Model name: StraightFibre_imperf_noshr_cdp4
** Generated by: Abaqus/CAE 6.11-2
*Preprint, echo=NO, model=NO, history=NO, contact=NO
**
** PARTS
**
*Part, name=Fibre
*Node
  1, 0., 0.449999988, 24.
  2, 0., 0., 24.
  3, 0., 0., 30.
  ......................................
  ......................................
  768, -0.142053023, 0.141492426, 3.
*Element, type=C3D8
  1, 272, 38, 215, 566, 17, 1, 37, 177
  2, 566, 215, 216, 567, 177, 37, 36, 178
  3, 567, 216, 217, 568, 178, 36, 35, 179
  ..................................................
  ..................................................
  496, 531, 166, 542, 749, 540, 167, 544, 762
*Nset, nset=_PickedSet4, internal, generate
  1, 768, 1
*Elset, elset=_PickedSet4, internal, generate
  1, 496, 1
*Elset, elset=_Fibre_S3, internal
  117, 118, 119, 120, 221, 222, 223, 224, 225, 226, 227, 228, 229, 230, 231, 232
  233, 234, 235, 236, 237, 238, 239, 240, 421, 422, 449, 450, 451, 452, 453, 454
  455, 456, 481, 482, 483, 484, 485, 486, 487, 488, 489, 490, 491, 492, 493, 494
  495, 496
*Elset, elset=_Fibre_S4, internal
  321, 322, 323, 324, 325, 326, 327, 328, 329, 330, 331, 332, 334, 345, 346, 347, 348
---

```
365, 366, 367, 368, 369, 370, 371, 372, 373, 374, 375, 376

*Surface, type=ELEMENT, name=Fibre
  _Fibre_S3, S3
_ Fibre_S4, S4
_ Fibre_S2, S2
** Section: Fibre_sec
*Solid Section, elset=_PickedSet4, material=Steel

*End Part
**
*Part, name=Matrix
*Node
  1, 0.449999988, 0., 24.
  2, 30., 0., 24.
  3, 30., 30., 24.

7620, 0.232719854, 0.651185453, -6.66666651
*Element, type=C3D8
  1, 638, 99, 100, 654, 3514, 878, 879, 3530
  2, 654, 100, 1, 32, 3530, 879, 101, 1066
  3, 637, 638, 654, 655, 3513, 3514, 3530, 3531

6280, 7510, 7487, 7488, 7620, 3296, 3273, 3274, 3406
*nset, nset=_PickedSet3_#4, internal, generate
  1, 7620, 1
*Elset, elset=_PickedSet3_#4, internal, generate
  1, 6280, 1
*Elset, elset=Matrix_S5, internal
  147, 164, 311, 328, 475, 492, 639, 656, 803, 820, 967, 984, 1131, 1148, 1295, 1312
  1459, 1476, 1623, 1640, 1787, 1804, 1951, 1968, 2115, 2132, 2279, 2296, 2443, 2460, 2607, 2624
  2625, 2626, 2789, 2790, 2953, 2954, 3117, 3118, 3281, 3282, 3445, 3446, 3609, 3610, 3773, 3774
  4453, 4454, 4617, 4618, 4781, 4782, 4945, 4946, 5109, 5110, 5273, 5274, 5437, 5438, 5601, 5602
*Elset, elset=Matrix_S4, internal
  1, 2, 165, 166, 329, 330, 493, 494, 657, 658, 821, 822, 985, 986, 1149, 1150
  1313, 1314, 1477, 1478, 1641, 1642, 1805, 1806, 1969, 1970, 2133, 2134, 2297, 2298, 2461, 2462
  2771, 2788, 2935, 2952, 3099, 3116, 3263, 3280, 3427, 3444, 3591, 3608, 3755, 3772, 3919, 3936
  4599, 4616, 4763, 4780, 4927, 4944, 5091, 5108, 5255, 5272, 5419, 5436, 5583, 5600, 5747, 5764
*Surface, type=ELEMENT, name=Matrix
  _Matrix_S5, S5
_ Matrix_S4, S4
** Section: Matrix_sec
*Solid Section, elset=_PickedSet3_#4, material=Concrete

*End Part
**
** ASSEMBLY
**
*Assembly, name=Assembly
**
*Instance, name=Matrix0-1, part=Matrix
*End Instance
**
*Instance, name=Fibre0-1, part=Fibre
*End Instance
**
*nset, nset=_PickedSet27, internal, instance=Matrix0-1
```
446, 447, 448, 449, 450, 451, 452, 453, 454, 455, 456, 457, 458
*Elset, elset=_PickedSet27, internal, instance=Matrix0-1
4285, 4287, 4288, 4290, 4292, 4294, 4296, 4298, 4300, 4302, 4304, 4306, 4308, 4310, 4312, 4314
4316, 4318, 4320, 4322, 4323, 4324, 4325, 4326, 4327, 4328, 4329, 4330, 6113, 6114, 6115, 6232
6234, 6236, 6238, 6240, 6242, 6244, 6246, 6248, 6249, 6250, 6251, 6252, 6253, 6254, 6255
6256, 6257, 6258, 6259, 6260, 6261, 6262, 6263
*Nset, nset=_PickedSet28, internal, instance=Matrix0-1
  1, 2, 7, 8, 12, 13, 14, 16, 19, 20, 22, 24, 25, 27, 29, 30
..............................................................
3477, 3478, 3479, 3480, 3481, 3482, 3483, 3484, 3485, 3486, 3487, 3488, 3489, 3490, 3491, 3492
3493, 3494, 3495, 3496
*Elset, elset=_PickedSet28, internal, instance=Matrix0-1
  2, 4, 6, 8, 10, 12, 14, 16, 18, 20, 22, 24, 26, 28, 30, 32
  34, 35, 36, 37, 38, 39, 40, 41, 42, 166, 168, 170, 172, 174, 176, 178
..............................................................
6120, 6122, 6124, 6126, 6128, 6130, 6132, 6134, 6136, 6138, 6140, 6142, 6144, 6146, 6148, 6150
6151, 6152, 6153, 6154, 6155, 6156, 6157, 6158
*Nset, nset=_PickedSet30, internal, instance=Matrix0-1
  23, 24, 25, 26, 30, 31, 432, 433, 434, 435, 436, 437, 438, 439, 440, 441
..............................................................
3382, 3383, 3384, 3385, 3386, 3387, 3388, 3389, 3390, 3391, 3392, 3393, 3394, 3395, 3396, 3397
3398, 3399, 3400, 3401, 3402, 3403, 3404, 3405, 3406
*Elset, elset=_PickedSet30, internal, instance=Matrix0-1
4281, 4282, 4283, 4284, 4285, 4286, 4287, 4288, 4289, 4290, 4291, 4292, 4293, 4294, 4295, 4296
..............................................................
6257, 6258, 6259, 6260, 6261, 6262, 6263, 6264, 6265, 6266, 6267, 6268, 6269, 6270, 6271, 6272
6273, 6274, 6275, 6276, 6277, 6278, 6279, 6280
*Nset, nset=_PickedSet32, internal, instance=Fibre0-1
  11,
*Nset, nset=_PickedSet33, internal, instance=Fibre0-1
  3, 6, 8, 9, 11, 15, 64, 65, 112, 113, 119, 120, 124, 125, 128, 129
158, 159, 160, 161, 431, 432, 433, 434, 492, 493, 494, 495
*Elset, elset=_PickedSet33, internal, instance=Fibre0-1
423, 424
*End Assembly
**
** MATERIALS
**
*Material, name=Concrete
*Elastic
33000., 0.2
*Concrete Damaged Plasticity
38., 0.1, 1.16, 0.666667, 0.0001
*Concrete Compression Hardening
8.46, 0
22.26, 7.2e-05
27.89, 0.000146
37., 0.000397
39.95, 0.000579
41.62, 0.0008
41.96, 0.00099
41.61, 0.0012
39.74, 0.00154
31.26, 0.00234
6.26, 0.00436
3.76, 0.0047
2.51, 0.005
1.26, 0.0055
*Concrete Tension Stiffening
3.4, 0
2., 0.00015
0.8, 0.0005
0.4, 0.0012
0.2, 0.002
*Material, name=Steel
*Elastic
210000., 0.3
***
** INTERACTION PROPERTIES
**
*Surface Interaction, name=IntProp-1
1.,
*Friction, elastic slip=0.01
0.04,
*Surface Behavior, penalty
*Cohesive Behavior, eligibility=ORIGINAL CONTACTS
11.,11.,11.
*Damage Initiation, criterion=QUADS
0.7, 0.7, 0.7
*Damage Evolution, type=DISPLACEMENT, softening=EXPONENTIAL, mixed mode behavior=TABULAR
0.5,0.5,0.0,0.
0.5,0.5,1.0,0.
*Damage Stabilization
0.
***
** BOUNDARY CONDITIONS
**
** Name: BC-1 Type: Displacement/Rotation
*Boundary
_PickedSet27, 1, 1
** Name: BC-2 Type: Displacement/Rotation
*Boundary
_PickedSet28, 2, 2
** Name: BC-2F Type: Displacement/Rotation
*Boundary
_PickedSet33, 2, 2
** Name: BC-3 Type: Displacement/Rotation
*Boundary
_PickedSet30, 3, 3
**
** INTERACTIONS
**
** Interaction: Int-1
*Contact Pair, interaction=IntProp-1, adjust=0.0
Matrix0-1.Matrix, Fibre0-1.Fibre
** -----------------------------------------------
**
** STEP: Displ
**
*Step, name=Displ, inc=10000
*Static
0.001, 1., 1e-15, 0.01
**
** BOUNDARY CONDITIONS
**
** Name: Displ Type: Displacement/Rotation
*Boundary
  _PickedSet32, 1, 1, 0.020978
  _PickedSet32, 3, 3, 0.3
**
** CONTROLS
**
*Controls, reset
*Controls, analysis=discontinuous
*Controls, parameters=time incrementation
  , , , , 20, ,
**
** OUTPUT REQUESTS
**
*Restart, write, frequency=0
**
** FIELD OUTPUT: F-Output-1
**
*Output, field, variable=PRESELECT
**
** HISTORY OUTPUT: H-Output-1
**
*Output, history, variable=PRESELECT
*End Step
**
------------------------------------------------
**
** STEP: Displ 2
**
*Step, name="Displ 2", inc=100000
*Static
  0.01, 1., 1e-15, 0.01
**
** BOUNDARY CONDITIONS
**
** Name: Displ Type: Displacement/Rotation
*Boundary
  _PickedSet32, 1, 1, 0.20978
  _PickedSet32, 3, 3, 3.
**
** OUTPUT REQUESTS
**
*Restart, write, frequency=0
**
** FIELD OUTPUT: F-Output-1
**
*Output, field, variable=PRESELECT, frequency=5
**
** HISTORY OUTPUT: H-Output-1
**
*Output, history, variable=PRESELECT, frequency=5
*End Step
C1.3 Influence of lateral forces on the pull-out behaviour

C1.3.1 Elastic material model

*Heading
** Job name: StraightFibre_quart_shr-LF2 Model name: StraightFibre_quarter_shr-LatForc2
** Generated by: Abaqus/CAE 6.11-2
** Preprint, echo=NO, model=NO, history=NO, contact=NO
**
** PARTS
**
*Part, name=Fibre
*Node
  1, 0., 0.449999988, 30.
  2, 0.449999988, 0., 30.
  3, 0.449999988, 0., 0.
  ...................................................
  ...................................................
  616, 0.141668797, 0.141851112, 30.666666
*Element, type=C3D8
  1, 275, 276, 494, 473, 29, 30, 155, 154
  2, 276, 277, 495, 494, 30, 31, 156, 155
  3, 277, 278, 496, 495, 31, 32, 157, 156
  ...................................................
  ...................................................
  344, 611, 452, 450, 609, 448, 149, 148, 446
*Nset, nset=_PickedSet3, internal, generate
  1, 616, 1
*Elset, elset=_PickedSet3, internal, generate
  1, 344, 1
*Elset, elset=_Fibre_S4, internal
  329, 330, 331, 332, 339, 340, 341, 342
*Elset, elset=_Fibre_S3, internal
  333, 334
*Elset, elset=_Fibre_S2, internal
  1, 2, 3, 4, 5, 6, 7, 8, 9, 10, 11, 12, 13, 14, 15, 16
  ...................................................
  ...................................................
  145, 146, 147, 148, 149, 150, 151, 152, 153, 154, 155, 156, 157, 158, 159, 160
  343, 344
*Surface, type=ELEMENT, name=Fibre
  _Fibre_S4, S4
  _Fibre_S3, S3
  _Fibre_S2, S2
** Section: Fibre_sec
*Solid Section, elset=_PickedSet3, material=Steel
  *
  *End Part
  **
*Part, name=Matrix
*Node
  1, 30., 0., 0.
  2, 30., 30., 0.
  3, 0., 30., 0.
  ...................................................
  ...................................................
  2584, 8.60946751, 3.06718206, -6.66666651
*Element, type=C3D8
  1, 267, 69, 70, 288, 1303, 503, 504, 1324
  2, 288, 70, 71, 289, 1324, 504, 505, 1325
  3, 289, 71, 72, 290, 1325, 505, 506, 1326
1948, 2477, 1279, 1280, 2584, 1059, 202, 203, 1166
*Nset, nset=_PickedSet3, internal, generate
  1, 2584, _ 1
*Elset, elset=_PickedSet3, internal, generate
  1, 1948, _ 1
*Elset, elset=_Matrix_S5, internal
  109, 122, 257, 270, 405, 418, 553, 566, 701, 714, 849, 862, 997, 1010, 1145, 1158
  1293, 1306, 1441, 1454
*Elset, elset=_Matrix_S4, internal
  123, 124, 271, 272, 419, 420, 567, 568, 715, 716, 863, 864, 1011, 1012, 1159, 1160
  1307, 1308, 1455, 1456
*Surface, type=ELEMENT, name=Matrix
  _Matrix_S5, S5
  _Matrix_S4, S4
** Section: Matrix_sec
*Solid Section, elset=_PickedSet3, material=Concrete
  ...
*End Part
**
** ASSEMBLY
**
*Assembly, name=Assembly
**
*Instance, name=Matrix-1, part=Matrix
*End Instance
**
*Instance, name=Fibre-1, part=Fibre
*End Instance
**
*Nset, nset=_PickedSet12, internal, instance=Matrix-1
  3, 4, 6, 10, 11, 13, 14, 34, 35, 36, 37, 38, 39, 40, 41, 42
  43, 44, 45, 46, 47, 48, 49, 50, 51, 52, 53, 95, 96, 97, 98, 99
  1217, 1218, 1219, 1220, 1221, 1222, 1223, 1224, 1225, 1226, 1227, 1228, 1229, 1230, 1231, 1232
  1233, 1234, 1235, 1236, 1237, 1238, 1239, 1240, 1241, 1242, 1243, 1244, 1245, 1246, 1247, 1248
*Elset, elset=_PickedSet12, internal, instance=Matrix-1
  82, 84, 86, 88, 90, 92, 94, 96, 97, 98, 99, 100, 101, 102, 103, 104
  1752, 1753, 1796, 1797, 1798, 1882, 1884, 1886, 1888, 1890, 1892, 1894, 1896, 1897, 1898, 1899
  1900, 1901, 1902, 1903, 1904, 1905, 1906, 1907, 1908, 1909
*Nset, nset=_PickedSet13, internal, instance=Matrix-1
  1, 5, 7, 8, 11, 12, 13, 57, 58, 59, 60, 61, 62, 63, 64, 65
  66, 67, 68, 69, 70, 71, 72, 73, 74, 75, 76, 77, 78, 79, 80, 81
  1263, 1264, 1265, 1266, 1267, 1268, 1269, 1270, 1271, 1272, 1273, 1274, 1275, 1276, 1277, 1278
  1279, 1280, 1281, 1282, 1283, 1284, 1285, 1286, 1287, 1288, 1289, 1290, 1291, 1292, 1293, 1294
*Elset, elset=_PickedSet13, internal, instance=Matrix-1
  1, 2, 3, 4, 5, 6, 7, 8, 124, 126, 128, 130, 132, 134, 136, 138
  140, 142, 144, 146, 148, 149, 150, 151, 152, 153, 154, 155, 156, 272, 274, 276
  1790, 1792, 1793, 1798, 1800, 1801, 1802, 1803, 1804, 1805, 1806, 1807, 1808, 1924, 1926, 1928
*Nset, nset=_PickedSet14, internal, instance=Matrix-1
  12, 13, 14, 15, 195, 196, 197, 198, 199, 200, 201, 202, 203, 204, 205, 206
  207, 208, 209, 210, 211, 212, 213, 214, 215, 216, 217, 218, 219, 220, 221, 222

*Nset, nset=_PickedSet14, internal, instance=Matrix-1
  1152, 1153, 1154, 1155, 1156, 1157, 1158, 1159, 1160, 1161, 1162, 1163, 1164, 1165, 1166

*Elset, elset=_PickedSet14, internal, instance=Matrix-1
  generate
  1793, 1948, 1

*Nset, nset=_PickedSet17, internal, instance=Matrix-1
  1, 2584, 1

*Elset, elset=_PickedSet17, internal, instance=Matrix-1
  generate
  1, 2584, 1

*Nset, nset=_PickedSet18, internal, instance=Fibre-1
  7, 8, 9, 145, 146, 147, 148, 149, 150, 151, 449, 450, 451, 452

*Elset, elset=_PickedSet18, internal, instance=Fibre-1
  generate
  1, 1, 1

*Nset, nset=_PickedSet19, internal, instance=Fibre-1
  2, 3, 5, 6, 7, 8, 54, 55, 56, 57, 58, 59, 60, 61, 62, 63

*Elset, elset=_PickedSet19, internal, instance=Fibre-1
  333, 334, 335, 336, 337, 338, 343, 344

*Nset, nset=_PickedSet19, internal, instance=Fibre-1
  64, 65, 66, 67, 68, 69, 70, 71, 72, 73, 74, 75, 76, 77, 78, 79

*Nset, nset=_PickedSet19, internal, instance=Fibre-1

*Elset, elset=_PickedSet19, internal, instance=Fibre-1
  42, 44, 46, 48, 50, 52, 54, 56, 58, 60, 62, 64, 66, 68, 70, 72

*Nset, nset=_PickedSet20, internal, instance=Fibre-1
  2, 3, 5, 6, 7, 8, 54, 55, 56, 57, 58, 59, 60, 61, 62, 63

*Elset, elset=_PickedSet20, internal, instance=Fibre-1
  273, 274, 275, 276, 277, 278, 279, 280, 321, 322, 323, 324, 329, 330, 333, 335

*Nset, nset=_PickedSet20, internal, instance=Fibre-1
  336,

*Elset, elset=_PickedSet20, internal, instance=Fibre-1
  1, 4, 5, 6, 8, 9, 10, 11, 12, 13, 14, 15, 16, 17, 18, 19

*Nset, nset=_PickedSet21, internal, instance=Fibre-1
  17, 18, 19, 20, 21, 22, 23, 24, 25, 26, 27, 28, 29, 30, 31, 32, 33, 34, 35

*Nset, nset=_PickedSet21, internal, instance=Fibre-1
  341, 342, 343, 344, 345, 346, 347, 348, 349, 350, 351, 352, 439, 440, 441, 442

*Elset, elset=_PickedSet21, internal, instance=Fibre-1
  1, 2, 3, 4, 5, 6, 7, 8, 9, 10, 11, 12, 13, 14, 15, 16

*Surface, type=ELEMENT, name=_PickedSurf21, internal

*Surface, type=ELEMENT, name=_PickedSurf21_S4, internal

*Surface, type=ELEMENT, name=_PickedSurf21_S5, internal

*End Assembly

** MATERIALS

**

*Material, name=Concrete
*Elastic
33000., 0.2
*Expansion
0.00015,
*Material, name=Steel
*Elastic
210000., 0.3
**
** INTERACTION PROPERTIES
**
*Surface Interaction, name=IntProp-1
1.,
*Friction, elastic slip=0.01
0.04,
*Surface Behavior, penalty
*Cohesive Behavior, eligibility=ORIGINAL CONTACTS
11.,11.,11.
*Damage Initiation, criterion=QUADS
0.7, 0.7, 0.7
*Damage Evolution, type=DISPLACEMENT, softening=EXPONENTIAL, mixed mode behavior=TABULAR
0.5,6.,0.,0.
0.5,6.,1.,0.
*Damage Stabilization
0.
**
** BOUNDARY CONDITIONS
**
** Name: BC-1 Type: Displacement/Rotation
*Boundary
  _PickedSet12, 1, 1
** Name: BC-1F Type: Displacement/Rotation
*Boundary
  _PickedSet20, 1, 1
** Name: BC-2 Type: Displacement/Rotation
*Boundary
  _PickedSet13, 2, 2
** Name: BC-2F Type: Displacement/Rotation
*Boundary
  _PickedSet19, 2, 2
** Name: BC-3 Type: Displacement/Rotation
*Boundary
  _PickedSet14, 3, 3
**
** PREDEFINED FIELDS
**
** Name: Predefined Field-1 Type: Temperature
*Initial Conditions, type=TEMPERATURE
  _PickedSet17, 0.
**
** INTERACTIONS
**
** Interaction: Int-1
*Contact Pair, interaction=IntProp-1, adjust=0.0
  Fibre-1.Fibre, Matrix-1.Matrix
**
** STEP: Shrinkage
**
*Step, name=Shrinkage
*Static  
1., 1., 1e-05, 1.  
**  
** PREDEFINED FIELDS  
**  
** Name: Predefined Field-1  Type: Temperature  
*Temperature  
_PickedSet17, -1.  
**  
** OUTPUT REQUESTS  
**  
*Restart, write, frequency=0  
**  
** FIELD OUTPUT: F-Output-1  
**  
*Output, field, variable=PRESELECT  
**  
** HISTORY OUTPUT: H-Output-1  
**  
*Output, history, variable=PRESELECT  
*End Step  
**  
** STEP: Lateral load  
**  
*Step, name="Lateral load"  
*Static  
1., 1., 1e-05, 1.  
**  
** LOADS  
**  
** Name: Pressure  Type: Pressure  
*Dload  
_PickedSurf21, P, 14.  
**  
** OUTPUT REQUESTS  
**  
*Restart, write, frequency=0  
**  
** FIELD OUTPUT: F-Output-1  
**  
*Output, field, variable=PRESELECT  
**  
** HISTORY OUTPUT: H-Output-1  
**  
*Output, history, variable=PRESELECT  
*End Step  
**  
** STEP: Displ  
**  
*Step, name=Displ, inc=10000  
*Static  
0.005, 1., 1e-15, 0.005  
**  
** BOUNDARY CONDITIONS  
**  
** Name: Displ Type: Displacement/Rotation  
*Boundary  
_PickedSet18, 3, 3, 0.5
** OUTPUT REQUESTS
**
*Restart, write, frequency=0
**
** FIELD OUTPUT: F-Output-1
**
*Output, field, variable=PRESELECT
**
** HISTORY OUTPUT: H-Output-1
**
*Output, history, variable=PRESELECT
*End Step
**
** STEP: Displ 2
**
*Step, name="Displ 2", inc=100000
*Static
0.005, 1., 1e-15, 0.005
**
** BOUNDARY CONDITIONS
**
** Name: Displ Type: Displacement/Rotation
*Boundary
*PickedSet18, 3, 3, 3.
**
** OUTPUT REQUESTS
**
*Restart, write, frequency=0
**
** FIELD OUTPUT: F-Output-1
**
*Output, field, variable=PRESELECT, frequency=5
**
** HISTORY OUTPUT: H-Output-1
**
*Output, history, variable=PRESELECT, frequency=5
*End Step

C1.3.2 CDP model

*Heading
** Job name: StraightFibre_quart_shr-LF2-CDP Model name: StraightFibre_quarter_shr-LatForc2-CDP
** Generated by: Abaqus/CAE 6.11-2
*Preprint, echo=NO, model=NO, history=NO, contact=NO
**
** PARTS
**
*Part, name=Fibre
*Node
  1, 0., 0.449999988, 30.
  2, 0.449999988, 0., 30.
  3, 0.449999988, 0., 0.

........................................
........................................
  616, 0.141668797, 0.141851112, 30.666666
*Element, type=C3D8
  1, 275, 276, 494, 473, 29, 30, 155, 154
2, 276, 277, 495, 494, 30, 31, 156, 155
3, 277, 278, 496, 495, 31, 32, 157, 156

344, 611, 452, 450, 609, 448, 149, 148, 446
*Nset, nset=_PickedSet3, internal, generate
1, 616, 1
*Elset, elset=_PickedSet3, internal, generate
1, 344, 1
*Elset, elset=_Fibre_S4, internal
329, 330, 331, 332, 339, 340, 341, 342
*Elset, elset=_Fibre_S3, internal
333, 334
*Elset, elset=_Fibre_S2, internal
1, 2, 3, 4, 5, 6, 7, 8, 9, 10, 11, 12, 13, 14, 15, 16

145, 146, 147, 148, 149, 150, 151, 152, 153, 154, 155, 156, 157, 158, 159, 160
343, 344
*Surface, type=ELEMENT, name=Fibre
_Fibre_S4, S4
_Fibre_S3, S3
_Fibre_S2, S2
** Section: Fibre_sec
*Solid Section, elset=_PickedSet3, material=Steel

*End Part

*Part, name=Matrix
*Node
1, 30., 0., 0.
2, 30., 30., 0.
3, 0., 30., 0.

2584, 8.60946751, 3.06718206, -6.66666651
*Element, type=C3D8
1, 267, 69, 70, 288, 1303, 503, 504, 1324
2, 288, 70, 71, 289, 1324, 504, 505, 1325
3, 289, 71, 72, 290, 1325, 505, 506, 1326

1948, 2477, 1279, 1280, 2584, 1059, 202, 203, 1166
*Nset, nset=_PickedSet3, internal, generate
1, 2584, 1
*Elset, elset=_PickedSet3, internal, generate
1, 1948, 1
*Elset, elset=_Matrix_S5, internal
109, 122, 257, 270, 405, 418, 553, 566, 701, 714, 849, 862, 997, 1010, 1145, 1158
1293, 1306, 1441, 1454
*Elset, elset=_Matrix_S4, internal
123, 124, 271, 272, 419, 420, 567, 568, 715, 716, 863, 864, 1011, 1012, 1159, 1160
1307, 1308, 1455, 1456
*Surface, type=ELEMENT, name=Matrix
_Matrix_S5, S5
_Matrix_S4, S4
** Section: Matrix_sec
*Solid Section, elset=_PickedSet3, material=Concrete

*End Part
** ASSEMBLY
**
*Assembly, name=Assembly
**
*Instance, name=Matrix-1, part=Matrix
*End Instance
**
*Instance, name=Fibre-1, part=Fibre
*End Instance
**
*Nset, nset=_PickedSet12, internal, instance=Matrix-1
  3, 4, 6, 10, 11, 13, 14, 34, 35, 36, 37, 38, 39, 40, 41, 42
  1233, 1234, 1235, 1236, 1237, 1238, 1239, 1240, 1241, 1242, 1243, 1244, 1245, 1246, 1247, 1248
*Elset, elset=_PickedSet12, internal, instance=Matrix-1
  82, 84, 86, 88, 90, 92, 94, 96, 97, 98, 99, 100, 101, 102, 103, 104
  1752, 1753, 1796, 1797, 1798, 1882, 1884, 1886, 1888, 1890, 1892, 1894, 1896, 1897, 1898, 1899
  1900, 1901, 1902, 1903, 1904, 1905, 1906, 1907, 1908, 1909
*Nset, nset=_PickedSet13, internal, instance=Matrix-1
  1, 5, 7, 8, 11, 12, 13, 57, 58, 59, 60, 61, 62, 63, 64, 65
  66, 67, 68, 69, 70, 71, 72, 73, 74, 75, 76, 77, 78, 79, 80, 81
  1263, 1264, 1265, 1266, 1267, 1268, 1269, 1270, 1271, 1272, 1273, 1274, 1275, 1276, 1277, 1278
  1279, 1280, 1281, 1282, 1283, 1284, 1285, 1286, 1287, 1288, 1289, 1290, 1291, 1292, 1293, 1294
*Elset, elset=_PickedSet13, internal, instance=Matrix-1
  1, 2, 3, 4, 5, 6, 7, 8, 124, 126, 128, 130, 132, 134, 136, 138
  140, 142, 144, 146, 148, 149, 150, 151, 152, 153, 154, 155, 156, 272, 274, 276
  1790, 1792, 1793, 1798, 1800, 1801, 1802, 1803, 1804, 1805, 1806, 1807, 1808, 1924, 1926, 1928
*Nset, nset=_PickedSet14, internal, instance=Matrix-1
  12, 13, 14, 15, 195, 196, 197, 198, 199, 200, 201, 202, 203, 204, 205, 206
  207, 208, 209, 210, 211, 212, 213, 214, 215, 216, 217, 218, 219, 220, 221, 222
  1152, 1153, 1154, 1155, 1156, 1157, 1158, 1159, 1160, 1161, 1162, 1163, 1164, 1165, 1166
*Elset, elset=_PickedSet14, internal, instance=Matrix-1, generate
  1793, 1948, 1
*Nset, nset=_PickedSet17, internal, instance=Matrix-1, generate
  1, 2584, 1
*Elset, elset=_PickedSet17, internal, instance=Matrix-1, generate
  1, 1948, 1
*Nset, nset=_PickedSet18, internal, instance=Fibre-1
  7, 8, 9, 145, 146, 147, 148, 149, 150, 151, 449, 450, 451, 452
*Elset, elset=_PickedSet18, internal, instance=Fibre-1
  333, 334, 335, 336, 337, 338, 343, 344
*Nset, nset=_PickedSet19, internal, instance=Fibre-1
  2, 3, 5, 6, 7, 8, 54, 55, 56, 57, 58, 59, 60, 61, 62, 63
  64, 65, 66, 67, 68, 69, 70, 71, 72, 73, 74, 75, 76, 77, 78, 79
*Elset, elset=_PickedSet19, internal, instance=Fibre-1

112 |
42, 44, 46, 48, 50, 52, 54, 56, 58, 60, 62, 64, 66, 68, 70, 72

273, 274, 275, 276, 277, 278, 279, 280, 321, 322, 323, 324, 329, 330, 333, 335
336
*Nset, nset=_PickedSet20, internal, instance=Fibre-1
  1,  4,  5,  6,  8,  9, 10, 11, 12, 13, 14, 15, 16, 17, 18, 19

341, 342, 343, 344, 345, 346, 347, 348, 350, 351, 352, 439, 440, 441, 442
*Elset, elset=_PickedSet20, internal, instance=Fibre-1
  1,  2,  3,  4,  5,  6,  7,  8,  9, 10, 11, 12, 13, 14, 15, 16

343,
*Elset, elset=_PickedSurf21_S4, internal, instance=Matrix-1
  17,  18,  19,  20,  21,  22,  23,  24,  165, 166, 167, 168, 169, 170, 171, 172
  313, 314, 315, 316, 317, 318, 319, 320, 461, 462, 463, 464, 465, 466, 467, 468
  609, 610, 611, 612, 613, 614, 615, 616, 757, 758, 759, 760, 761, 762, 763, 764
  905, 906, 907, 908, 909, 910, 911, 912, 1053, 1054, 1055, 1056, 1057, 1058, 1059, 1060
1201, 1202, 1203, 1204, 1205, 1206, 1207, 1208, 1349, 1350, 1351, 1352, 1353, 1354, 1355, 1356
1496, 1504, 1652, 1660, 1808, 1816
*Elset, elset=_PickedSurf21_S5, internal, instance=Matrix-1
  8,  16, 156, 164, 304, 312, 452, 460, 600, 608, 748, 756, 896, 904, 1044, 1052
1192, 1200, 1340, 1348, 1505, 1506, 1507, 1508, 1509, 1510, 1511, 1512, 1661, 1662, 1663, 1664
1665, 1666, 1667, 1668, 1817, 1818, 1819, 1820, 1821, 1822, 1823, 1824
*Surface, type=ELEMENT, name=_PickedSurf21, internal
  __PickedSurf21_S4, S4
  __PickedSurf21_S5, S5
*End Assembly

** MATERIALS
**

*Material, name=Concrete
*Elastic
33000., 0.2
*Expansion
0.00015,
*Concrete Damaged Plasticity
38.,  0.1,  1.16, 0.666667, 0.0001
*Concrete Compression Hardening
  8.46,  0.
  22.26,  7.2e-05
  27.89,  0.000146
  37.,  0.000397
  39.95,  0.000579
  41.62,  0.0008
  41.96,  0.00099
  41.61,  0.0012
  39.74,  0.00154
  31.26,  0.00234
  6.26,  0.00436
  3.76,  0.0047
  2.51,  0.005
  1.26,  0.0055
*Concrete Tension Stiffening
  3.4,  0.
  2.,  0.00015
  0.8,  0.0005
0.4, 0.0012
0.2, 0.002
*Material, name=Steel
*Elastic
210000., 0.3
**
** INTERACTION PROPERTIES
**
*Surface Interaction, name=IntProp-1
1.,
*Friction, elastic slip=0.01
0.04,
*Surface Behavior, penalty
*Cohesive Behavior, eligibility=ORIGINAL CONTACTS
11.,11..11.
*Damage Initiation, criterion=QUADS
0.7, 0.7, 0.7
*Damage Evolution, type=DISPLACEMENT, softening=EXPONENTIAL, mixed mode behavior=TABULAR, mode mix ratio=TRACTION
0.5,6..0.,0.
0.5,6..1.,0.
*Damage Stabilization
0.
**
** BOUNDARY CONDITIONS
**
** Name: BC-1 Type: Displacement/Rotation
*Boundary
_PickedSet12, 1, 1
** Name: BC-1F Type: Displacement/Rotation
*Boundary
_PickedSet20, 1, 1
** Name: BC-2 Type: Displacement/Rotation
*Boundary
_PickedSet13, 2, 2
** Name: BC-2F Type: Displacement/Rotation
*Boundary
_PickedSet19, 2, 2
** Name: BC-3 Type: Displacement/Rotation
*Boundary
_PickedSet14, 3, 3
**
** PREDEFINED FIELDS
**
** Name: Predefined Field-1 Type: Temperature
*Initial Conditions, type=TEMPERATURE
_PickedSet17, 0.
**
** INTERACTIONS
**
** Interaction: Int-1
*Contact Pair, interaction=IntProp-1, adjust=0.0
Fibre-1.Fibre, Matrix-1.Matrix
**
** STEP: Shrinkage
**
*Step, name=Shrinkage
*Static
1., 1., 1e-05, 1.
** PREDEFINED FIELDS
**
** Name: Predefined Field-1  Type: Temperature
  *Temperature  
PickedSet17, -1.
**
** OUTPUT REQUESTS
**
  *Restart, write, frequency=0
**
** FIELD OUTPUT: F-Output-1
**
  *Output, field, variable=PRESELECT
**
** HISTORY OUTPUT: H-Output-1
**
  *Output, history, variable=PRESELECT
  *End Step
  **
  ** STEP: Lateral load
  **
  *Step, name="Lateral load"
  *Static
  1., 1., 1e-05, 1.
  **
  ** LOADS
  **
  ** Name: Pressure  Type: Pressure
  *Dsload  
PickedSurf21, P, 14.
  **
  ** OUTPUT REQUESTS
  **
  *Restart, write, frequency=0
  **
  ** FIELD OUTPUT: F-Output-1
  **
  *Output, field, variable=PRESELECT
  **
  ** HISTORY OUTPUT: H-Output-1
  **
  *Output, history, variable=PRESELECT
  *End Step
  **
  ** STEP: Displ
  **
  *Step, name=Displ, inc=10000
  *Static
  0.005, 1., 1e-15, 0.005
  **
  ** BOUNDARY CONDITIONS
  **
  ** Name: Displ Type: Displacement/Rotation
  *Boundary
  PickedSet18, 3, 3, 0.5
  **
  ** OUTPUT REQUESTS
**
*Restart, write, frequency=0
**
** FIELD OUTPUT: F-Output-1
**
*Output, field, variable=PRESELECT
**
** HISTORY OUTPUT: H-Output-1
**
*Output, history, variable=PRESELECT
*End Step
**
** STEP: Displ 2
**
*Step, name="Displ 2", inc=100000
*Static
0.005, 1., 1e-15, 0.005
**
** BOUNDARY CONDITIONS
**
** Name: Displ Type: Displacement/Rotation
*Boundary
PickedSet18, 3, 3, 3.
**
** OUTPUT REQUESTS
**
*Restart, write, frequency=0
**
** FIELD OUTPUT: F-Output-1
**
*Output, field, variable=PRESELECT, frequency=5
**
** HISTORY OUTPUT: H-Output-1
**
*Output, history, variable=PRESELECT, frequency=5
*End Step
C2  Hooked-end fibre

C2.1  Pull-out test

*Heading
** Job name: MESH-coh+fri Model name: Hooked_Fibre-nl-steel+coh+fri-MESH
** Generated by: Abaqus/CAE 6.11-2
*Preprint, echo=NO, model=NO, history=NO, contact=NO
**
** PARTS
**
*Part, name=fibre
*Node
   1,  0.449999988,  3.25071596e-08
   2,  0.675000012,  3.25071596e-08
   3,  0.899999976,  3.25071596e-08
   ......................................
   620, 0.431968778, 32.
*Element, type=CPS4
   1,   1,   2,   7,   6
   2,   2,   3,   8,   7
   3,   3,   4,   9,   8
   .................
   492, 614, 615, 620, 619
*Nset, nset=_PickedSet2, internal, generate
   1, 620, 1
*Elset, elset=_PickedSet2, internal, generate
   1, 492, 1
*Elset, elset=_Surf-L_S4, internal, generate
   1, 489, 4
*Surface, type=ELEMENT, name=Surf-L_S4, S4
*Elset, elset=_Surf-R_S2, internal, generate
   4, 492, 4
*Surface, type=ELEMENT, name=Surf-R_S2, S2
** Section: steel_sec
*Solid Section, elset=_PickedSet2, material=Steel

*End Part
**
*Part, name=matrix
*Node
   1,  0.449999988,  1.45000005
   2,  0.449999988,  3.25071596e-08
   3,  0.260268807,  1.94859695
   ...................................
   4152, 5.13050461, 11.7795906
*Element, type=CPS4
   1,  544,  543,  570,  573
   2, 1032,  803,  805, 1253
   3, 1423, 1612, 1606, 1421
   .................................
   3988, 4144, 2687, 4139, 4143
*Nset, nset=_PickedSet2, internal, generate
   1, 4152, 1
*Elset, elset=_PickedSet2, internal, generate
$1, 3988, 1$

*Elset, elset= Surf-L_S4, internal
459, 461, 462, 463, 464, 465, 466, 467, 469, 470, 471, 472, 473, 474, 475, 476
477, 478, 479, 480, 481, 483, 485, 486, 487, 489, 490, 492, 493, 494, 495, 496
497, 499, 500, 501, 502, 504, 505, 506, 507, 508, 509, 510, 729, 786
*Elset, elset= Surf-L_S1, internal
38, 311, 444, 446, 448, 451, 452, 635, 638, 641, 645, 871, 3827
*Elset, elset= Surf-L_S3, internal
36, 37, 41, 441, 443, 445, 449, 513, 639, 642
*Elset, elset= Surf-L_S2, internal
42, 80, 102, 453, 482, 488, 503, 695, 713, 730, 749, 768, 783, 799, 812, 818
*Surface, type=ELEMENT, name=Surf
_Surf-L_S4, S4
_Surf-L_S1, S1
_Surf-L_S2, S2
_Surf-L_S3, S3
*Elset, elset= Surf-R_S3, internal
221, 222, 233, 234, 242, 245, 558, 573, 574, 575, 576, 577, 580, 582, 584, 588
590, 593, 595, 598, 603, 604, 605, 609, 611, 615, 776
*Elset, elset= Surf-R_S1, internal
596, 597, 599, 600, 601, 602, 606, 607, 608, 610, 612, 613, 614, 616, 617, 619
621, 623, 624, 629, 701, 710, 719, 728, 738, 748, 759, 766, 775, 794, 807, 821
925,
*Elset, elset= Surf-R_S2, internal
223, 224, 225, 251, 260, 261, 262, 265, 557, 572, 625, 626, 649, 654, 659
*Elset, elset= Surf-R_S4, internal
250, 259, 622, 627, 628, 629, 630, 664, 668
*Surface, type=ELEMENT, name=Surf
_Surf-R_S3, S3
_Surf-R_S1, S1
_Surf-R_S2, S2
_Surf-R_S4, S4
** Section: conc_sec
*Solid Section, elset= PickedSet2, material=concrete

, *End Part
**

** ASSEMBLY
**

*Assembly, name=Assembly
**

*Instance, name=fibre-1, part=fibre
*End Instance
**

*Instance, name=matrix-1, part=matrix
*End Instance
**

*Nset, nset= PickedSet9, internal, instance=matrix-1
24, 25, 26, 27, 28, 29, 30, 31, 32, 33, 34, 48, 49, 50, 51, 52
53, 54, 55, 56, 57, 58, 402, 403, 404, 436, 437, 438
*Elset, elset= PickedSet9, internal, instance=matrix-1
55, 56, 57, 58, 110, 111, 120, 128, 132, 138, 143, 146, 162, 176, 177, 182
184, 186, 188, 191, 192, 195, 267, 268, 269, 380
*Nset, nset= PickedSet11, internal, instance=fibre-1, generate
616, 620, 1
*Elset, elset= PickedSet11, internal, instance=fibre-1, generate
489, 492, 1
*Nset, nset= PickedSet13, internal, instance=matrix-1

*Elset, elset=_PickedSet13, internal, instance=matrix-1
6, 7, 67, 69, 118, 131, 137, 141, 143, 144, 150, 176, 180, 181, 529, 530
531, 532, 533, 534, 535, 536, 537, 538, 539, 540, 541, 542, 543, 544, 545, 546
547, 548, 549, 550, 551, 552, 553, 554, 555, 556, 830, 3722, 3723

*Elset, elset=_PickedSurf10_S3, internal, instance=fibre-1, generate
489, 492, 1

*Surface, type=ELEMENT, name=_PickedSurf10, internal
_PickedSurf10_S3, S3

*End Assembly

** MATERIALS

** Material, name=Steel
*Elastic
210000., 0.3
*Plastic
1000., 0.
1061., 0.02
1096., 0.05
1136., 0.1
1166., 0.15
1192., 0.2
1214., 0.25
1235., 0.3
1254., 0.35
1271., 0.4
1288., 0.45
1303., 0.5

*Material, name=concrete
*Elastic
33000., 0.2

** INTERACTION PROPERTIES

** Surface Interaction, name=IntProp-L
1.,
*Friction, elastic slip=0.01
0.04,
*Surface Behavior, penalty
*Cohesive Behavior, eligibility=ORIGINAL CONTACTS
11.,11.,11.
*Damage Initiation, criterion=QUADS
0.7, 0.7, 0.7
*Damage Evolution, type=DISPLACEMENT, softening=EXPONENTIAL, mixed mode behavior=TABULAR
0.5,6.,0.,0.
0.5,6.,1.,0.
*Damage Stabilization
0.
*Surface Interaction, name=IntProp-R
1.,
*Friction, elastic slip=0.01
0.04,
*Surface Behavior, penalty
*Cohesive Behavior, eligibility=ORIGINAL CONTACTS
11.,11.,11.
*Damage Initiation, criterion=QUADS
0.7, 0.7, 0.7
*Damage Evolution, type=DISPLACEMENT, softening=EXPONENTIAL, mixed mode
behavior=TABULAR, mode mix ratio=TRACTION
0.5, 6, 0.5, 0.0
0.5, 6, 1.0, 0.0
*Damage Stabilization
0.
**
** BOUNDARY CONDITIONS
**
** Name: Bottom Type: Displacement/Rotation
*Boundary
_PickedSet13, 2, 2
** Name: Side Type: Displacement/Rotation
*Boundary
_PickedSet9, 1, 1
**
** INTERACTIONS
**
** Interaction: Int-L
*Contact Pair, interaction=IntProp-L, adjust=0.0
** Interaction: Int-R
*Contact Pair, interaction=IntProp-R, adjust=0.0
**
** STEP: Loadstep
**
*Step, name=Loadstep, nlgeom=YES, extrapolation=PARABOLIC, inc=100000
*Static
0.0001, 1., 1e-30, 0.01
**
** BOUNDARY CONDITIONS
**
** Name: displ Type: Displacement/Rotation
*Boundary
_PickedSet11, 2, 2, 6.
**
** CONTROLS
**
*Controls, reset
*Controls, analysis=discontinuous
*Controls, parameters=time incrementation
,, , , , , 20,,
*Controls, parameters=line search
10,, , , 0.01
**
** OUTPUT REQUESTS
**
*Restart, write, frequency=0
**
** FIELD OUTPUT: F-Output-1
**
*Output, field, variable=PRESELECT
**
** HISTORY OUTPUT: H-Output-1
**
*Output, history, variable=PRESELECT
*End Step