Thomas Pøtten Rakke
Jon Olav Lauvstad Røland

Numerical study on the influence of cover thickness and rebar spacing on effective areas in reinforced concrete ties

Master's thesis in Civil and Environmental Engineering
Supervisor: Max Hendriks and Jesus Reignard Medel Tan
June 2020

Master's thesis

NTNU
Norwegian University of Science and Technology
Faculty of Engineering
Department of Structural Engineering

NTNU
Norwegian University of
Science and Technology

Department of Structural Engineering
Faculty of Engineering
NTNU- Norwegian University of Science and Technology

ACCESSIBILITY
OPEN

MASTER THESIS 2020

SUBJECT AREA:	DATE:	NO. OF PAGES:
Crack width calculations in concrete structures	09.06.2020	12 + 49 + 27

TITLE:

Numerical study on the influence of cover thickness and rebar spacing on effective areas in reinforced concrete ties

Numerisk studie av påvirkningen overdekningsstørrelse og armeringsavstand har på det effektive arealet i strekkstaver i armert betong

BY:

Thomas Pøtten Rakke

Jon Olav Lauvstad Røland

SUMMARY:
Accurate predictions of crack propagation, distribution, width, and effect on structures are crucial to design for intended service life. The purpose of this thesis is to conduct finite element analyses on reinforced concrete ties to investigate the influence of cover thickness and rebar spacing on crack widths and crack spacing.

A literature review has been carried out in which numerical methods of estimating crack widths as well as finite element modelling of concrete has been studied. As the theory suggests, more refined and improved methods for estimating effective tensile areas and crack width calculations are necessary. The models have been validated by reconstructing the experiments of Yannopoulos and Bresler and Bertero, using them as benchmarks. Four models have been constructed with varying rebar spacing and cover thickness.

Numerical results from the finite element analyses support the idea in Eurocode 2 that an arbitrary cross section can be transformed into an equivalent axisymmetric section. Cover thickness and rebar spacing affect the effective tensile area around rebars. The effective area of two specimens have been overestimated by Eurocode 2, compared to tension zones indicated by numerical results.

A reduction factor ζ has been derived based on the diminution of bond stress that contribute to tensile stiffening. In practice, this reduces the total perimeter of all rebars contributing to bond. A proposition has been raised for the value of ζ based on the cover thickness and rebar spacing.

Keywords: Concrete, crack widths, crack spacing, finite element analysis, rebar spacing, cover thickness, bond slip.

SUPERVISOR: Professor Max Hendriks, NTNU and TU Delft

CO-SUPERVISOR: Associate professor Jesus Reignard Medel Tan, NTNU and Multiconsult

CARRIED OUT AT: The Department of Structural Engineering, NTNU

Preface

This thesis marks the final step of the two-year master's program in Civil and Environmental Engineering at the Norwegian University of Science and Technology (NTNU). During our time here we have been subject to the Department of Structural Engineering for which this thesis is written.

The main supervisor for this project has been Professor Max Hendriks with additional supervision from associate professor Reignard Tan. The work presented in this dissertation has been carried out in Trondheim during the period January to June and is equivalent to 30 ECTS per author.

We would like to thank Max and Reignard for guidance and encouragement during these months. Due to the coronavirus pandemic, meeting in-person has not been viable since March, but we will not forget the entertaining video conferences any time soon.

Thomas Rakke

Jon Olav L. Røland

Thomas Rakke

Jon Olav Lauvstad Røland

Trondheim
10.06.2020

Abstract

Accurate predictions of crack propagation, distribution, width, and effect on structures are crucial to design for intended service life. The purpose of this thesis is to conduct finite element analyses on reinforced concrete ties to investigate the influence of cover thickness and rebar spacing on crack widths and crack spacing.

A literature review has been carried out where numerical methods of estimating crack widths as well as finite element modelling of concrete has been studied. As the theory suggests, more refined and improved methods for estimating effective tensile areas and crack width calculations are necessary. The models have been validated by reconstructing the experiments of Yannopoulos and Bresler and Bertero and using them as benchmarks. Four models have been constructed with varying rebar spacing and cover thickness.

Numerical results from the finite element analyses support the idea in Eurocode 2 that an arbitrary cross section can be transformed into an equivalent axisymmetric section. Cover thickness and rebar spacing affect the effective tensile area around rebars. The effective area of two specimens have been overestimated by Eurocode 2, compared to tension zones indicated by numerical results.

A reduction factor ζ has been derived based on the diminution of bond stress that contribute to tensile stiffening. In practice, this reduces the total perimeter of all rebars contributing to bond. A proposition has been raised for the value of ζ based on the cover thickness and rebar spacing.

Sammendrag

Nøyaktige estimater av rissdannelse, -fordeling, -vidde og påvirkningen dette har på konstruksjoner er essensielt for å dimensjonere for tiltenkt levetid. Formålet med denne oppgaven er å gjennomføre endelige elementanalyser på strekkstaver i armert betong for å undersøke innflytelsen overdekning og senteravstand har på rissvidder og rissfordeling.

En litteraturstudie har blitt gjennomført med fokus på to områder: undersøke numeriske metoder for å estimere rissvidder og modellering av betong med elementanalyse. Som teorien tilsier er det behov for bedre metoder for rissviddeberegning og estimasjon av effektive strekksoner. Modellene har blitt validert ved å rekonstruere og bruke eksperimentene av Yannopoulos og Bresler og Bertero som referansetester. Fire modeller har blitt konstruert med ulik senteravstand og overdekning.

De numeriske resultatene fra elementanalysene støtter idéen i Eurokode 2 om at et tilfeldig tverrsnitt kan transformeres til et ekvivalent aksesymmetrisk tverrsnitt. Det er kommet frem til at overdekning og senteravstand påvirker de effektive strekksonene rundt armeringen. De effektive arealene i to av prøvene er blitt overestimert av Eurokode 2, sammenlignet med hva de numeriske resultatene indikerer.

En reduksjonsfaktor ζ er blitt utledet basert på reduksjon av heftspenninger som bidrar til strekkstivhet. I praksis reduserer faktoren den totale omkretsen av armering som bidrar til heft. Et forslag til hva ζ kan være er presentert, og er knyttet til de kjente størrelsene: overdekning og senteravstand.

Contents

Abbreviations

DMI	-	The Dutch Ministry of Infrastructure
DIANA	-	Displacement Analyzer version 10.3 and 10.4
EC2	-	Eurocode 2: Design of Concrete Structures
FEA	-	Finite Element Analysis
NLFEA	-	Non-Linear Finite Element Analysis
MC2010	-	*fib* Model Code 2010
RC	-	Reinforced concrete

Symbols

A_c	-	Concrete cross section area
$A_{c,eff}$	-	Effective tension area
A_e	-	Finite element area
$D_{tangent}$	-	Tangent stiffness matrix in local crack coordinate system
D	-	Tangent stiffness matrix
E_c	-	E-modulus of concrete
E_s	-	E-modulus of steel
E_{normal}	-	Interface normal stiffness
E_{shear}	-	Interface shear stiffness
F_c	-	Integrated bond stress force
F_{cr}	-	Cracking force
G_c	-	Compression fracture energy
G_f	-	Tensile fracture energy
K	-	System stiffness matrix
L	-	Length
ΔL	-	Change of length
R	-	Nodal forces
$S_{r,max}$	-	Crack distance
T	-	Transformation matrix
V	-	Element volume
W_k	-	Crack width
W_n	-	Newton Cotes integration point weight
a_{bar}	-	Rebar spacing
c	-	Cover thickness
d	-	Effective thickness of cross section
f_{ck}	-	Concrete cylinder compression strength
f_{cm}	-	Mean concrete compression strength
f_{ctm}	-	Mean concrete tensile strength
$f_{ct,eff}$	-	Mean value of concrete tensile strength at crack
f_t	-	Tensile strength
f_{yk}	-	Steel yield strength
h	-	Cross section height
$h_{c,eff}$	-	Effective height
h_{eq}	-	Element bandwidth

k	-	Influence of cover coefficient
k_1	-	Bond property coefficient
k_2	-	Strain distribution coefficient
k_3	-	EC2 coefficient in crack calculations
k_4	-	EC2 coefficient in crack calculations
k_t	-	Load duration coefficient
$l_{s,max}$	-	Length over which slip can occur
n	-	Integration point number
n, s, t	-	Local crack coordinate system
r	-	Nodal displacement
t_i	-	Interface thickness
u_{bottom}	-	Displacement at the rebar surface
$u_{interface}$	-	Critical interface opening
u_{top}		Displacement at the concrete cover surface
u_{xyz}	-	Total displacement
w_d	-	Crack width
x	-	Concrete compression zone
x_{cr}	-	Primary crack distance
$\mathit{\Delta}$	-	Delta
$\mathit{\Sigma}$	-	Epsilon, sum
α_c	-	Concrete strain at maximum compressive strength
$\alpha_{c/3}$	-	Concrete strain at one third of compressive strength
α_u	-	Ultimate concrete strain
β	-	Empirical coefficient for mean strain over $l_{s,max}$
$\Delta\varepsilon$	-	Total strain change of element
$\Delta\varepsilon^{co}$	-	Change in concrete strain
$\Delta\varepsilon^{cr}$	-	Change in crack strain
ε_{cm}	-	Mean concrete strain
ε_{cs}	-	Free shrinkage strain
ε_{sm}	-	Mean steel strain
ε_{sh}	-	Shrinkage strain
ε_u	-	Ultimate strain parameter
ζ	-	Bond stress reduction factor
η_r	-	Coefficient for shrinkage contribution
ν_c	-	Concrete Poisson ratio
ν_s	-	Steel Poisson ratio
$\rho_{s,eff}$	-	Reinforcement effective area ratio
σ_s	-	Steel stress
τ_{bm}	-	Mean bond stress
τ_{bms}	-	Mean bond strength
τ_{max}	-	Maximum bond stress
τ_n	-	Bond stress at specific integration point
ϕ	-	Reinforcement diameter

List of figures

List of tables

1 Introduction

1.1 Relevance of crack width calculations

As a part of the Ferry Free E39 project, the Norwegian Public Roads Administration is funding research on several topics that is of interest to the project. One of these topics are crack width calculations. Accurate predictions of crack propagation, distribution, width, and effect on structures are crucial to design for intended service life. Cracking, both internal and external, increases permeability, which hastens the damage onset of deteriorating mechanisms such as reinforcement corrosion. Local reinforcement corrosion in chloride-rich environments can be virtually impossible to detect and proves to be a severe threat to load bearing capacity of concrete structures. To prevent such mechanisms that have a detrimental effect in the serviceability and ultimate limit state, accurate crack width calculations are critical.

1.2 Thesis objective

The purpose of this thesis is to investigate the influence of cover thickness and rebar spacing on crack widths and crack spacing by conducting non-linear finite element analyses on reinforced concrete ties. The formulas in Eurocode 2 for predicting crack widths in structures assumes that an effective area surrounding the rebars contributes to tension stiffening. Hence, the main objectives of this thesis are to investigate how:

1. Cover thickness and rebar spacing affect the effective area and thus the bond stress contributing to tension stiffening.

2. The effective areas observed in this thesis conform with the ones suggested in Eurocode 2.

1.3 Research method and outline of thesis

A literature review of numerical modelling of concrete has been carried out to increase our knowledge within the topic. To study the behavior of concrete ties, the method of finite element modelling is quite suitable. It allows us to approximate a solution of the mathematical problem, and study results in every location of the model. To conduct these analyses in the software DIANA, time has been devoted to understanding the program and interpretation of results.

Chapter 2 is devoted to background theory for the thesis. Relevant theory on concrete cracking, analytical and numerical crack width calculations, and bond behavior in concrete ties will be presented. In chapter 3 the mechanical and numerical model is described, with a review of the verification and validation of the model. In chapter 4 numerical results from the analyses are presented. Chapter 5 is devoted to discussing the numerical results and accuracy of the finite element model. After reviewing and discussing the results, conclusions based upon the thesis objectives are presented in chapter 6.

2 Theory

2.1 Uni-axial behavior of concrete

A total crack strain model can describe the tensile and compressive behavior of a material with one stress-strain relationship. This constitutive model is recommended in the guidelines for nonlinear finite element analysis of concrete structures issued by The Dutch Ministry of Infrastructure, henceforth referred to as DMI. DMI advices to use exponential and parabolic softening diagrams for describing tensile and compressive behavior respectively.

Concrete in tension

The exponential tension softening curve presented by Hordijk [1] is based on fracture energy. Material inputs for the stress-strain relation are tensile strength (f_t) tensile fracture energy (G_f) and the crack bandwidth (h_{eq}). The shape of the tension softening diagram is dependent on the upper limit value of the ultimate strain parameter $\varepsilon_u = 5.136 \frac{G_f}{f_t \cdot h_{eq}}$. The concrete continues to transmit stresses until this strain level is reached. The area under the graph is equal to the total fracture energy, which in the smeared crack approach is evenly distributed over the crack bandwidth of the element shown in figure 2-1. According to DMI, this description of the tensile behavior results in more localized cracking [2], which is described further in chapter 2.4.

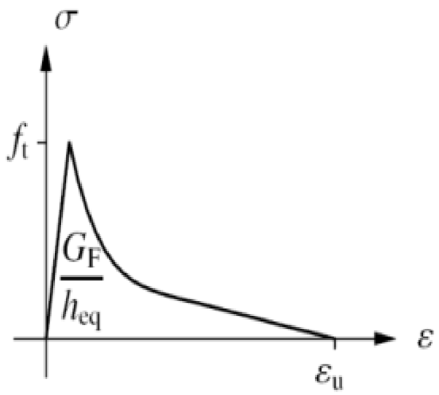


Figure 2-1 Tensile softening diagram by Hordijk [2]

Concrete in compression

The parabolic compression curve by Feenstra [3] is based on compressive fracture energy (G_c) and is described by the compressive strength (f_c) at the corresponding strain level (α_c). Linear-elastic behavior can be assumed up to a stress level of $\frac{1}{3}f_c$. Eventual micro cracks can be considered stable up to this stress level but will begin to propagate at higher stress levels. After passing f_c, micro cracks localize further eventually forming macro cracks. The compressive softening is a function of the compressive fracture energy and the crack bandwidth. Complete softening and failure occurs at $\alpha_u = \alpha_c - \frac{3}{2}\frac{G_c}{h_{eq} f_c}$. The parabolic compression curve is shown in figure 2-2.

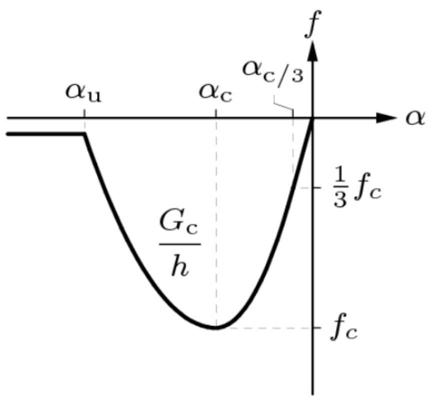


Figure 2-2 Parabolic compression diagram by Feenstra [2]

2.2 Concrete crack process

As stresses reach concrete tensile capacity, accumulation of energy initiates the propagation of a crack. Figure 2-3 shows the three stages of the crack process. If accumulated energy (G) exceeds fracture energy (G_f) of the concrete, equilibrium is not possible, and a crack will propagate. As the crack propagates, energy will flow into the crack tip and distributes into the fracture zone, forming equilibrium. It is at the tip of the crack and in the area around it that microcracking occurs [4]. The microcracking in the crack tip can be initiated at 90% of the tensile strength [5]. On a microscopic level, concrete is heterogeneous and cracks forming are highly tortuous. As microcracks grow, they eventually form larger discrete cracks. Crack planes form where the tensile capacity is low, which is in the interfacial transition zone between aggregates and matrix particles. Behind the crack tip, in the *bridging* zone, cracking occurs due to shearing action between matrix and aggregate particles. Depending on the energy level at the interface, the crack will either pass through the particle or debonding will occur. In the *traction-free* part of the crack, the crack process has completed. The opening is continuous, and no stresses can be transmitted across the crack plane [6].

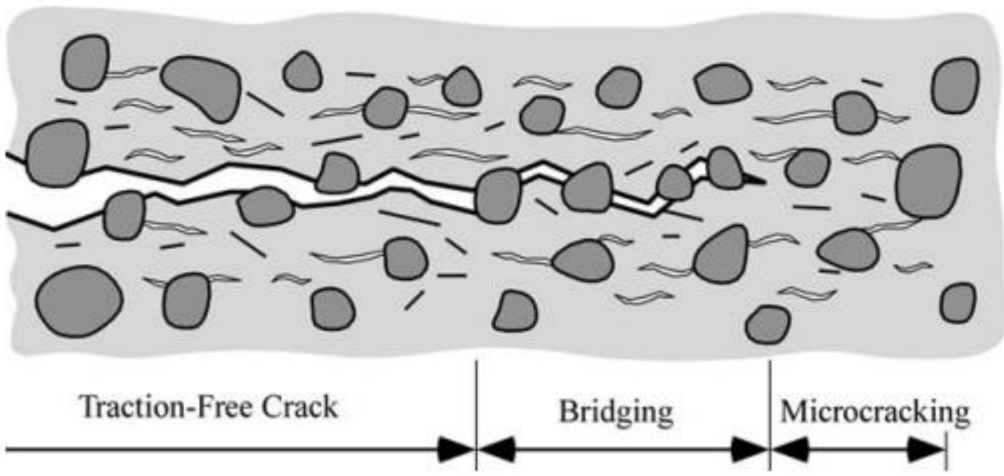


Figure 2-3 Crack development in concrete [6]

2.3 Analytical method for crack width control in Eurocode 2

The guidelines and methods devised in Eurocode 2, henceforth referred to as EC2, will be presented in this chapter.

$$W_k = S_{r,max}(\varepsilon_{sm} - \varepsilon_{cm}) \tag{2.1}$$

$$S_{r,max} = k_3 c + k_1 k_2 k_4 \frac{\phi}{\rho_{s,eff}} \tag{2.2}$$

$$\varepsilon_{sm} - \varepsilon_{cm} = \frac{\sigma_s - k_t \frac{f_{ct,eff}}{\rho_{s,eff}}\left(1 + \alpha_e \rho_{s,eff}\right)}{E_s} \geq 0.6 \frac{\sigma_s}{E_s} \tag{2.3}$$

The crack width (W_k) can be calculated from equation 2.1, and is a product of maximum crack distance ($S_{r,max}$) and the difference between mean strain in steel (ε_{sm}) and concrete (ε_{cm}). Crack distance and the difference in strain is shown in equation 2.2 and 2.3. $\rho_{s,eff}$ is an important factor in calculation of crack widths. It is the ratio between reinforcement area and effective concrete tension zone around the rebar. This effective area is explained in EC2 as; "the effective area of concrete in tension surrounding the reinforcement" [7, p. 121]. The zone is determined by an effective height ($h_{c,eff}$), given in equation 2.4 [7].

$$h_{c,eff} = \min\{\, 2.5(h - d); (h - x)/3; h/2 \,\} \tag{2.4}$$

Figure 2-4 shows how the effective area is determined for a beam, slab or member in tension as defined by EC2.

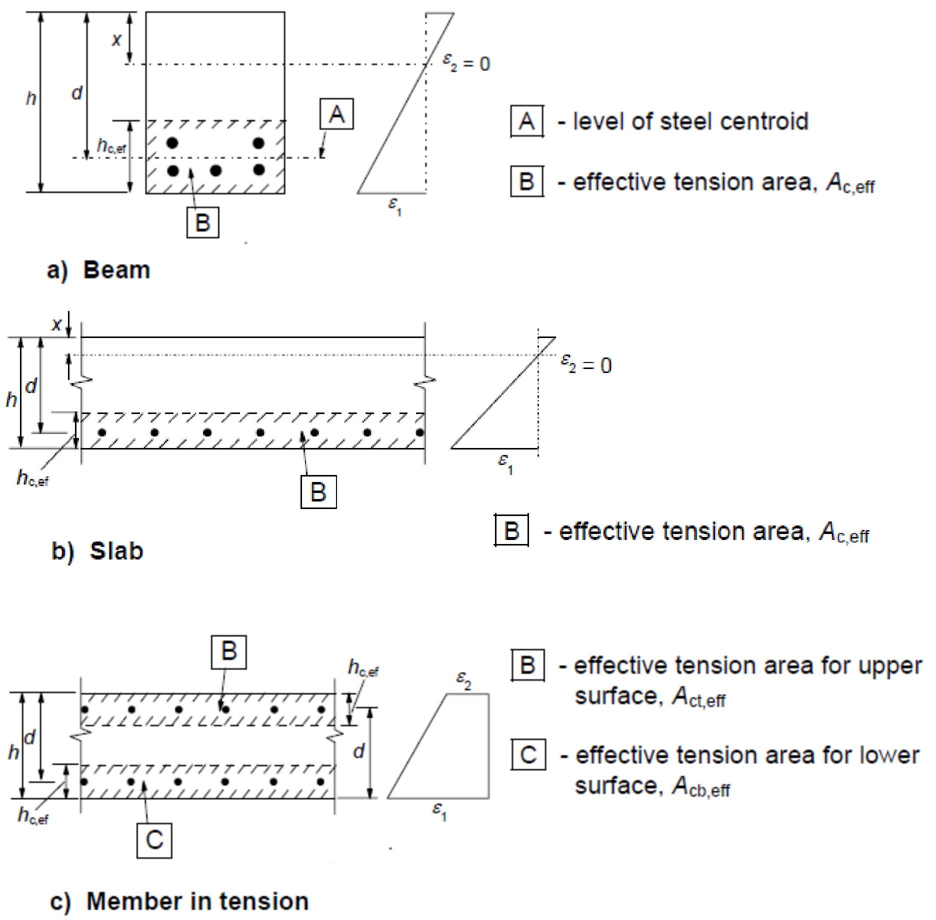


Figure 2-4 Effective heights described in Eurocode 2 [7]

Isolating a single rebar from a member in figure 2-4 and the corresponding effective area of concrete around it, defined by $h_{c,eff}$ and the rebar spacing, the area becomes rectangular. In equation 2.2 the factors $k_1 k_2 k_4$ are multiplied with $\frac{\phi}{\rho_{s,eff}}$, which implicitly transforms an arbitrary cross section into an equivalent axisymmetric cross section. It is therefore implied a constant bond stress, τ_{bm}, around each rebar [8].

The closely related fib Model Code 2010 utilizes a similar method, where a constant bond stress (τ_{bm}) is assumed as an alternative to the *k* factors in EC2. The code specifies that unless an effective tensile area can be explicitly determined, expressions for the effective height, which is comparable to $h_{c,eff}$ in Figure 2-4, can be applied [5].

The formulas in EC2 are derived from two conceptually different theories, the slip and no-slip theory. In slip theory, a certain slip is assumed to occur at the interface due to bond failure between concrete and steel. In no-slip theory, no slip is allowed at the interface. Any slip is therefore due to elastic shear deformation in the concrete section [9]. Tan et al. [10] explains that when incorporating two different theories, the concrete and steel strain experience a compatibility and incompatibility at the same time at the end of each transfer length.
As Tan notes:

> Such inconsistencies, including the fact that that equilibrium is violated at the end of transfer length, opposes the basic principles in solid mechanics and which further limits a generalization of the formulas [10, p. 7].

The formulas tend to yield fairly good results for small specimens, but when geometrical size increases, such as cover thickness or reinforcement diameter, the formulas tend to give inconsistent results [10].

As the theory suggests, better formulas for predicting crack widths are needed, and an important step to improving these is by investigating the effective area thoroughly.

2.4 Numerical methods for crack width calculations

The finite element method is a method for solving engineering and mathematical problems. The assembly process of elements into a system gives us the system stiffness relation shown in equation 2.5.

$$K \cdot r = R \tag{2.5}$$

In equation 2.5, K is the system stiffness matrix, r is the nodal point displacement and R is the nodal force. The mathematical problem has been transformed into a system of linear, algebraic equations. The system can be solved for the unknown nodal displacements r [11]. The relation between stiffness and nodal forces can in some cases be rendered void, such as in concrete cracking, and the system cannot be described linearly. A nonlinear relation must be established. In nonlinear problems, the stiffness matrix K and the nodal force vector R becomes functions of the displacement history r. The system stiffness relation is rewritten, shown in equation 2.6 [12].

$$K(r) \cdot r = R(r) \tag{2.6}$$

The system cannot immediately be solved for r, because the information needed to construct K and R is missing. An iterative process is needed to obtain r, and its corresponding K and R, such that $K(r)$ is in equilibrium with $R(r)$ [12]. Possible reasons for nonlinear effects in finite element analyses can be geometric nonlinearities, large deflections, contact forces, or nonlinear material behavior such as steel or concrete cracking.

When it comes to modelling the nonlinear effect of cracking in concrete, there are two main approaches: the *discrete* and *smeared* approach shown in figure 2-5 and figure 2-6. The *discrete* crack concept is based on the nodal separation between two adjacent elements. This prerequisite contradicts the requirement in the finite element formulation stating that every nodal point should have a unique correspondence with element and system degrees of freedom [11]. In addition, the approach also implies that cracks propagate along element edges. Discrete cracking could also be used in simulating debonding between steel and concrete.

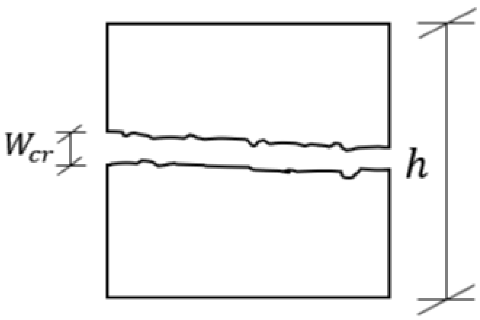


Figure 2-5 Discrete crack in an element

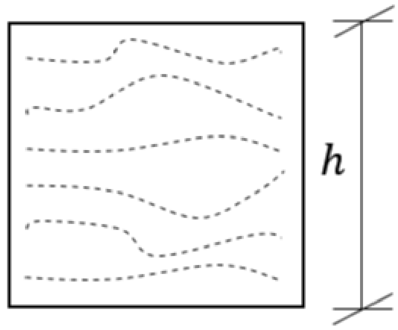


Figure 2-6 Smeared crack in an element

The *smeared* approach assesses a finite element and distributes the crack width (w_{cr}) as strain (ε) over the element width (h) given in equation 2.7.

$$\varepsilon = \frac{w_{cr}}{h} \tag{2.7}$$

The total strain of the element can be decomposed into concrete strain and crack strain to ease the incorporation of crack laws given in equation 2.8.

$$\Delta\varepsilon = \Delta\varepsilon^{co} + \Delta\varepsilon^{cr} \tag{2.8}$$

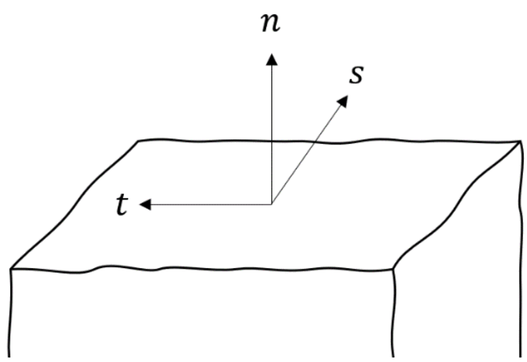


Figure 2-7 Crack plane coordinate system

Prior to crack formation, the element can be described using isotropic material properties. Upon crack formation, element properties are assumed to be orthotropic and the crack can co-rotate with the principle strain axes [13]. The orientation of the axes of orthotropy is determined by the condition of crack initiation. On a random crack plane such as in figure 2-7, local strains are computed in the n,s,t-coordinate which is unique for that plane. On a random crack plane such as in figure 2-7, local strains are computed in the n,s,t-coordinate which is unique for that plane. The relation between local crack plane and the system is given by a transformation matrix.

For solving the nonlinear system of equations in equation 2.6, an incremental-iterative process is used. Such an approach requires a system matrix that is used to update the incremental displacements. DIANA has two ways to define this stiffness: the secant stiffness matrix and tangent stiffness matrix. The latter has shown superiority where localized cracking and crack propagation are the most important phenomena. A new tangent stiffness matrix (D) is calculated after cracking by employing a transformation matrix (T) and a tangent stiffness matrix in the crack coordinate system ($D_{tangent}$). Equation 2.9 shows the transformation to the tangent stiffness matrix [14].

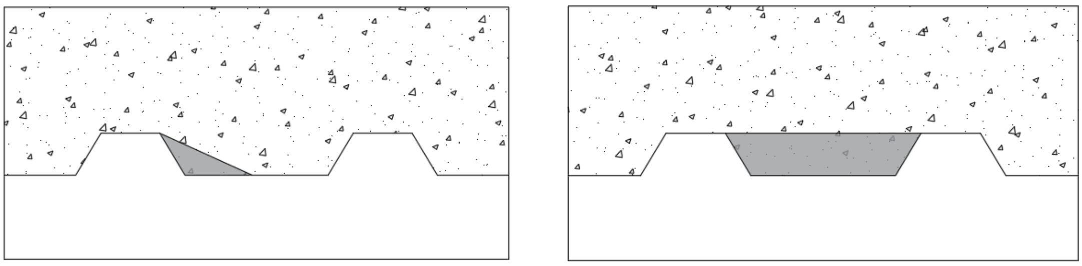
$$D = T^T D_{tangent} T \tag{2.9}$$

2.5 Bond behavior in reinforced concrete ties

The displacement between reinforcement and concrete is called bond slip, which leads to bond forces. Bond slip can be characterized into long and short slips, where slippage under 1 mm falls into the short category. There are three mechanisms that may contribute to bond forces: mechanical interaction, friction, and chemical adhesion. In the short slip category, the primary contributing mechanism is mechanical interaction between the rebar lug and concrete [15]. As a rebar slips, the contribution from cohesion to bond is small and eventually diminishes. When loading an anchored rebar, relative movement, or slip, will be mainly caused by crushing of the concrete in front of the rebar lugs [16].

As the chemical adhesion breaks down at low bond stress, bond will be maintained by bearing force on the rebar lug on the concrete. Diagonal microcracks will form in front of the rebar lug, which generates the largest part of the slip. These microcracks cause diagonal compression struts in front of the rebar lugs [17].

Based on the geometry and spacing of the rebar lug, research has observed two different pull-out failure modes: wedging action and concrete crushing. The two different failure modes are illustrated in figure 2-8. In most structural applications, a splitting failure due to wedging action is more common [16].



a) Concrete crushing in front of lug

b) Wedging action between lugs

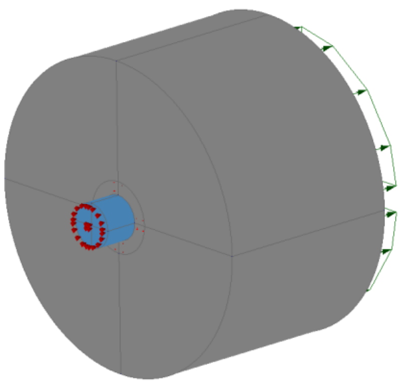
Figure 2-8 Pull-out failure modes

3 Numerical modelling

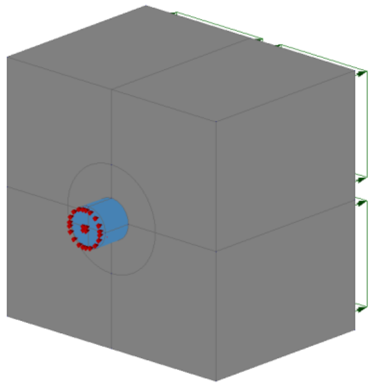
To perform virtual uniaxial tensile tests, the testing arrangements and the properties of the specimens can be explained in a mechanical and numerical model. The mechanical model is the physical description of the fundamental problem while the numerical model explains how the problem was solved using element analyses. Experiments have been conducted in the finite element program DIANA. Four different models have been constructed to study changes in bond stress related to concrete cover and rebar spacing. The material properties of the concrete have been set according to MC2010 with supplementary inputs from DMI [2]. To assess the validity of the finite element model, the experiments of Yannopoulos [18] and Bresler and Bertero [19] were reconstructed and used as benchmarks.

3.1 Mechanical model

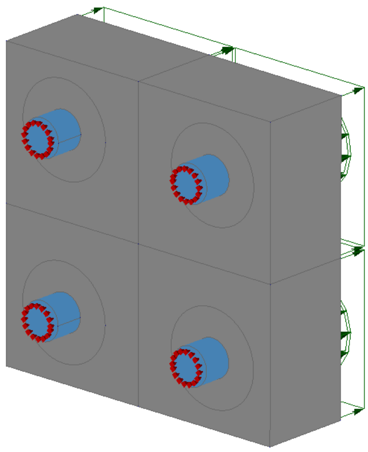
Figure 3-1 show the geometrical shapes of the four models. Sectional properties and measurements are listed in table 3-1. The models are meant to reflect the region between two primary cracks in a concrete tie. The end of the model with a protruding rebar is referred to as the *loaded end*, and the primary crack distance is measured from the face of the concrete on this side. The *symmetry section* is located on the opposite side, and is, due to the use of boundary conditions, located at $x = \frac{L}{2}$. The loading is uniformly applied in the symmetry section on the faces of both the concrete cover and rebar. At the loaded end, the face of the rebar is restrained from longitudinal displacement. The reinforcement is effectively tensioned by restraining the rebar in one end and pulling in the other, and stresses are thus transferred from the rebar to the concrete. The determination of primary crack distances is explained in chapter 3.3. The label of the models refers to their layout: Φ – rebar diameter, c – cover thickness, Q – quadratic cover.



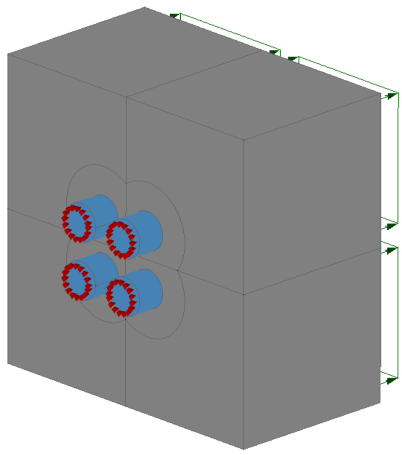
a) Singularly reinforced cylinder Φ20c60



b) Singularly reinforced square prism Φ20c52Q



c) Square prism with four spaced rebars 4Φ20c25Q



d) Square prism with four concentrated rebars 4Φ20c55Q

Figure 3-1 Overview of the short models

Units in mm	*Height*	*Width*	*Radius*	*Crack spacing*	*Cover*	*Rebar spacing*	*#Rebars*
Φ20c60	-	-	70	170	60	-	1Φ20
Φ20c52Q	62	62	-	175	52	-	1Φ20
4Φ20c25Q	160	160	-	95	25	70	4Φ20
4Φ20c55Q	160	160	-	180	55	10	4Φ20

Table 3-1 Sectional properties of the models

The material properties of the concrete have been set according to MC2010 with supplementary inputs from DMI. The material properties of concrete and steel are summarized in table 3-2 and table 3-3 respectively.

Concrete C35		
Characteristic compressive strength	f_{ck}	$35 \frac{N}{mm^2}$
Mean compressive strength	f_{cm}	$43 \frac{N}{mm^2}$
Mean tensile strength	f_{ctm}	$3.2 \frac{N}{mm^2}$
Elastic modulus[1]	E_c	$34962 \frac{N}{mm^2}$
Poisson ratio	ν_c	0.15
Tensile fracture energy	G_f	$0.144 \frac{N}{mm}$
Compressive fracture energy	G_c	$35.92 \frac{N}{mm}$

Table 3-2 Concrete properties

Reinforcement B500NC		
Elastic modulus	E_s	$200\ 000 \frac{N}{mm^2}$
Yield strength	f_{yk}	$500 \frac{N}{mm^2}$
Poisson ratio	ν_s	0.3

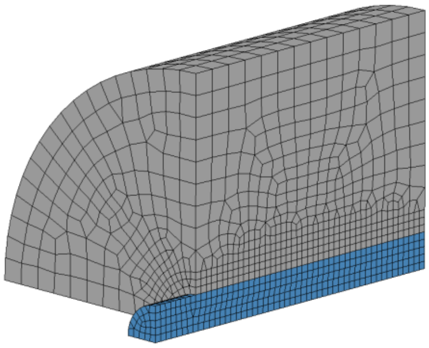
Table 3-3 Steel properties

[1] 28 days of curing.

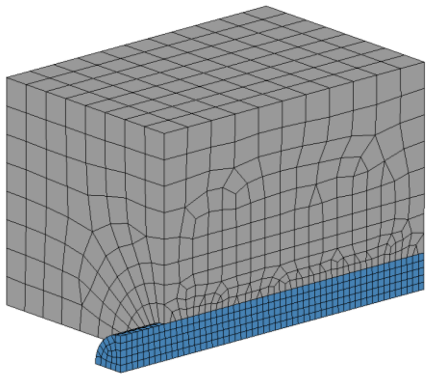
3.2 Finite element model

3.2.1 Discretization of elements

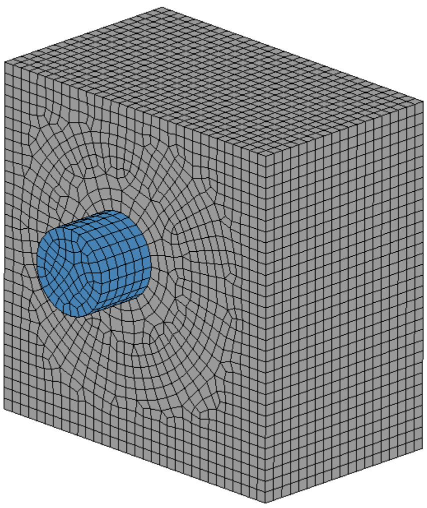
Figure 3-2 shows how the models were discretized. In the four experiments, half of the crack spacing x_{cr} was modeled. Due to double symmetry, a quart of each specimen was modeled by adding boundary conditions on the symmetry planes. The protruding end of the rebar is restrained against longitudinal displacement. A prescribed displacement is applied on the face of the rebar and concrete cover in the symmetry section. Geometrically accurate drawings of the finite element model are included in appendix A.



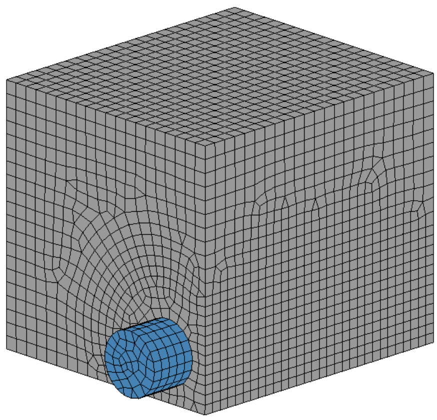
a) Singularly reinforced cylinder Φ20c60



b) Singularly reinforced square prism Φ20c52Q



d) Square prism with four spaced rebars 4Φ20c25Q



d) Square prism with four concentrated rebars 4Φ20c55Q

Figure 3-2 Element discretization of the models

The finite element model was discretized with volume elements and quadratic mesh order with linear interpolation. The size of the elements was adjusted to obtain approximately ten elements over the rebar diameter, and ten to fifteen elements over the cover thickness. The bond between the rebar and the concrete has been modeled by utilizing nonlinear plane quadrilateral interface elements. Table 3-4 shows the number of elements and the discretization of loads in the four experiments.

	Number of elements	*Deformation [mm]*	*Load increment*	*Number of load steps*
Φ20c60	10253	0.2	0.01	100
Φ20c52Q	4526	0.2	0.01	100
4Φ20c25Q	21233	0.1	0.03	50
4Φ20c55Q	16680	0.1	0.02	100

Table 3-4 Element and load discretization

3.2.2 Describing the bond slip model

As explained in chapter 2.5, adhesion breaks down as slip progresses and the bond is maintained through mechanical interaction between the rebar lug and concrete. Therefore, the transfer of bond stress at the interface between concrete and steel is assumed to be maintained even when the concrete separates radially from the rebar. By modeling the interaction between concrete and steel with interface elements, the mechanical bond generated by lugs are uniformly distributed over the rebar. Consequentially, the compression struts that are formed in front of the lugs, as explained in chapter 2.5, will not be explicitly replicated by the interface elements, but rather smeared out along the rebar.

The elastic material input of the interface is the Young modulus (E_{normal}) and the shear modulus (E_{shear}) in local x and y direction. The thickness of the interface was selected as relatively small, 0.1 mm, to ensure high interface stiffness. The moduli were calculated from equation 3.1 and 3.2.

$$E_{normal} = \frac{E_c}{t_i} \tag{3.1}$$

$$E_{shear} = \frac{E_c}{2(1+\nu_c)t_i} \tag{3.2}$$

The shear modulus was modeled with equal stiffness in x and y direction. This assumption is based on the skew geometry of the lugs. The skewness ensure that the rebar can neither slip nor rotate from the concrete.

The bond slip properties of the interface, as described by Tan *et al*. [20], have been incorporated in the finite element model. E_{normal} was reduced by a factor of 10^{-5} when the interface reached 80% of its tensile strain capacity, to secure a stable separation of interface and concrete elements. The reduction hinders numerical interference from microcracking that initiates at stress levels around 90% of the tensile strength capacity as described in chapter 2.4. This strain, multiplied with the interface thickness, is the critical interface opening ($u_{interface}$) and was calculated using equation 3.3. By keeping the shear modulus constant, the interface allows the rebar to radially separate from the concrete, but not slip nor rotate. Table 3-5 summarizes the interface properties used in the finite element model.

$$u_{interface} = 0.8 \cdot \frac{f_{ct}}{E_c} \cdot t_i \tag{3.3}$$

Interface

Thickness	t_i	0.1 mm
Elastic modulus of concrete	E_c	34962 $\frac{N}{mm^2}$
Elastic normal modulus	E_{normal}	349619 $\frac{N}{mm^2}$
Elastic shear modulus	E_{shear}	152008 $\frac{N}{mm^2}$
Critical interface opening	$u_{interface}$	7.32×10^{-6}
Reduction factor	$-$	10^{-5}

Table 3-5 Interface properties

3.2.3 Material behavior

The cracking behavior was modeled using the concept of rotating cracks with a crack bandwidth approach as described by Govindjee. The equivalent length of the crack bandwidth is determined as $h_{eq} = \sqrt[3]{V}$, where V is the volume of the element. Tension softening is described using Hordijk's exponential function. DIANA reduces the Poisson ratio using a damage-based formulation, since an elongated element with cracks will no longer experience contraction forces perpendicular to the elongated direction. The compressive behavior was described using a parabolic compression curve, with no reduction of compressive strength due to lateral cracking nor increase due to confinement. This assumption is based on related theory presented in chapter 2.1.

3.2.4 Achieving equilibrium

To achieve equilibrium, a modified Newton Rhapson method was chosen, in which the first tangent in the first iteration is set to tangential and line search is enabled. The maximum number of iterations per load increment was set to 30, and if an increment did not converge, the solver moved to the next step. In accordance with DMI, convergence norms were set to satisfy an energy tolerance of 0.001 and force tolerance of 0.01.

3.3 Determination of primary crack spacing

Preliminary finite element analyses were done on longer specimens to determine the mean crack spacing. The length on all specimens were set to 700 mm, as it was seen that the primary crack formed at a distance shorter than half the specimen length from the loaded end, as seen in figure 3-3 for the long square prism with four spaced rebars. The models created in these analyses were also carried out utilizing symmetry, identical to figure 3-2. The specimens were axially monotonically loaded until a primary crack was induced at the crack distance (x_{cr}) measured from the concrete face. The crack distances were determined by identifying the distances at which sudden changes in principal strain occurred through the cross section during the loading sequence. Half of these distances were used in the four short element models.

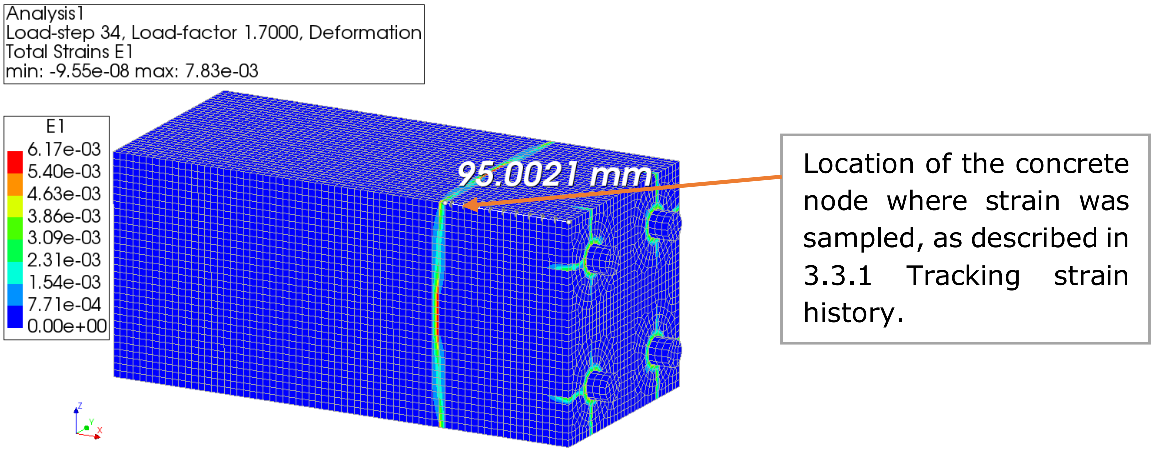


Figure 3-3 Primary crack formation in 4ϕ20c25Q

3.3.1 Verification of primary crack spacing

Bond stress resultant compared to crack force

To verify the results, bond stress was integrated over the crack distance. The bond stress was obtained by inspecting traction shear stress in the interface elements at the integration points. By summing the product of element area, integration point weight and traction shear stress; the bond stress resultant F_c was determined as shown in equation 3.4. The crack force, F_{cr}, is a product of the cross-sectional area multiplied by the mean tensile strength as shown in equation 3.5.

$$F_c = \sum_{1}^{n} A_e \cdot W_n \cdot \tau_n \qquad (3.4)$$

$$F_{cr} = f_{ctm} A_c \qquad (3.5)$$

The script used for integrating bond stress is included in appendix C.3. A 3x3 Newton Cotes integration scheme was applied to the interface elements, and weights from this scheme was used for the integration [21]. The resultant F_c should be approximately equal to the concrete cracking force F_{cr}, as steel stress is transferred to the concrete as bond stress in the interface. Table 3-6 shows the calculated crack distance for each specimen, as well as the bond stress resultant and the crack force.

	Applied steel stress [MPa]	*Crack distance [mm]*	*F_c [N]*	*F_{cr} [N]*
Φ20c60 – L	222.14	170	11016	12064
Φ20c52Q – L	227.05	175	11137	12049
4Φ20c25Q – L	141.86	95	18624	19475
4Φ20c55Q – L	131.86	180	18922	19475

Table 3-6 Results from bond stress integration in the long specimens

Tracking strain history

The primary crack distances were determined for each specimen by identifying the sections at which large changes in strain occurred during the loading sequence. A verification of this process of which the primary crack spacings were determined is shown in figure 3-4, which shows the strain history of a concrete node in the long square prism with four spaced rebars. The location of the node is at the outer corner of the specimen, adjacent to the primary crack, highlighted by an arrow seen in figure 3-3. The stress in the figure was sampled in the rebar at the loaded end. The plot shows a nearly linear relation between the stress in the rebar and strain in the node up to a stress level of 100 MPa, at which point the strain rapidly increase. The last registered stress level in the rebar, before a primary crack formed, was 141.86 MPa. Simultaneously the concrete node experienced a strain of 0.33‰. At the next converging load step, the steel stress is virtually identical, but the strain in the node is 0.047‰, clearly indicating that the primary crack has formed, and that the node is no longer strained.

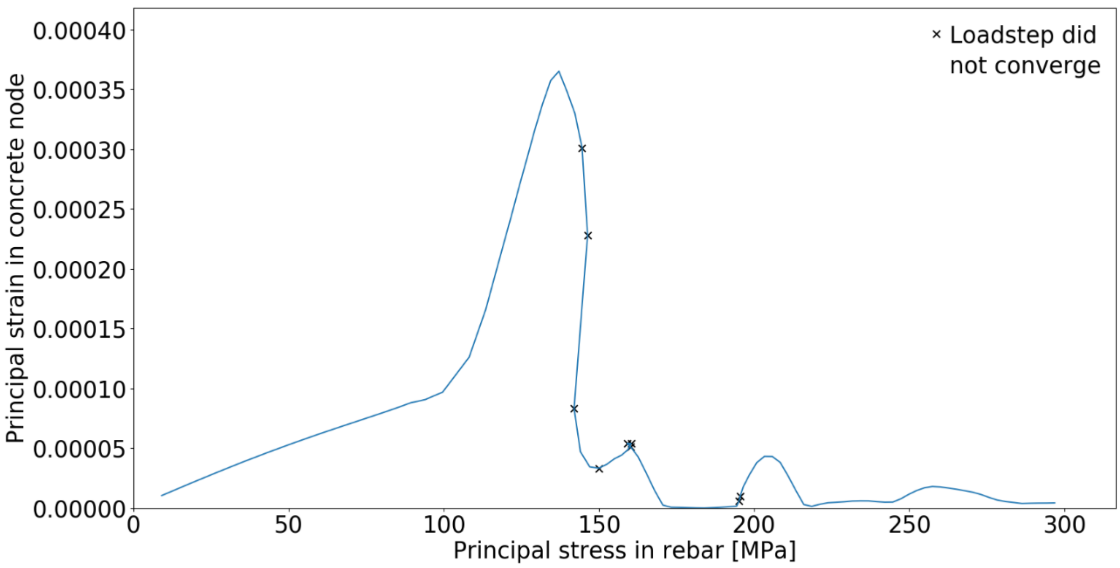


Figure 3-4 Strain history of concrete node adjacent to primary crack in 4ϕ20c25Q

3.4 Model verification

Equilibrium of forces

To verify that the finite element model does not violate the basic principles of mechanics, equilibrium of forces was checked at a random load step early in the loading sequence before major cracking behavior could be observed. Reaction forces were sampled in the nodes on each end of the specimen. The sum of forces in x, y and z direction are shown in table 3-7.

Units in N	ΣF_x	ΣF_y	ΣF_z
Loaded end	1778.11	-0.07	0.07
Symmetry section	-1778.11	-6.99	7.15
Difference	0.00	-7.06	7.22

Table 3-7 Equilibrium check of $\phi 20c60$

As table 3-7 shows, there is equilibrium in the longitudinal direction. There are some residual forces in y- and z-direction, which stem from the iteration process in DIANA. There can be some unbalanced forces even after a load step has converged, which are then allocated to the supports as external forces. The remaining forces are small compared to the forces in the longitudinal direction and does not affect the results.

Force-displacement diagram

Figure 3-5 shows the force-displacement diagram of the singularly reinforced cylinder, which indicate the nonlinear nature of the problem. Steel stress was sampled in the nodes on the face of the loaded end and multiplied with the rebar area to obtain the force. The displacement was sampled in the concrete node closest to the rebar surface. The red dot in figure 3-5 indicates the stress level where a primary crack formed at the symmetry section in the longer specimen.

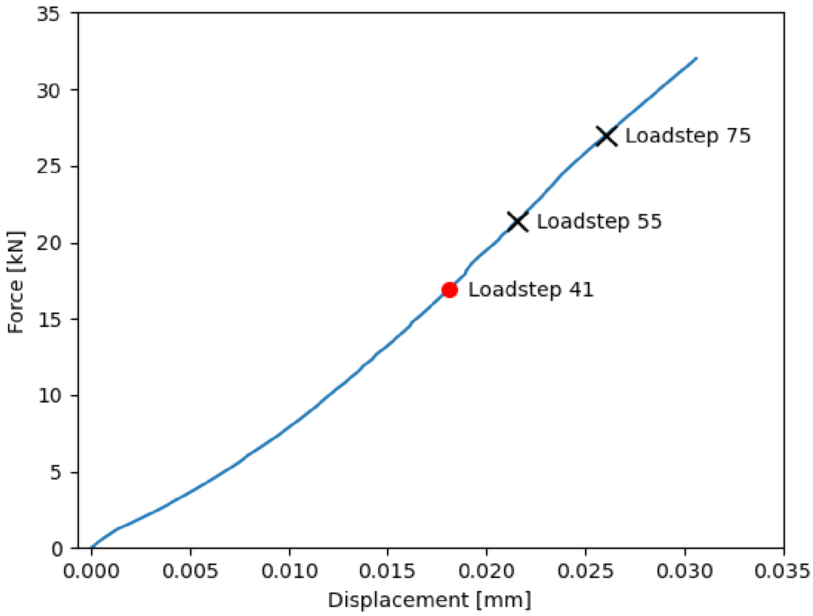


Figure 3-5 Force-displacement diagram of $\phi 20c60$

Load step 55 and 75 are two important load steps which are marked in the force displacement diagram in figure 3-5. At load step 55, vertical internal microcracks start to propagate along the surface of the rebar. The microcracks are shown in figure 3-6, where it can be observed that the cracks are approximately the same size. Upon further loading, a relatively large microcrack starts to form at $L/4$ as shown in figure 3-7. The development of microcracks from load step 55 to 75 corresponds well to the theory presented in chapter 2.2, where it is explained that as microcracks grow they will eventually form a larger discrete crack as observed in figure 3-7.

As explained in chapter 2.5, slip is manly caused by microcracking formed at the surface of the rebars. Even though the interface element does not explicitly model the rebar lugs, the compression struts and microcracking on the steel surface can be observed in specimen Φ20c60 shown in figure 3-6 and figure 3-7 and corresponds well to the theory.

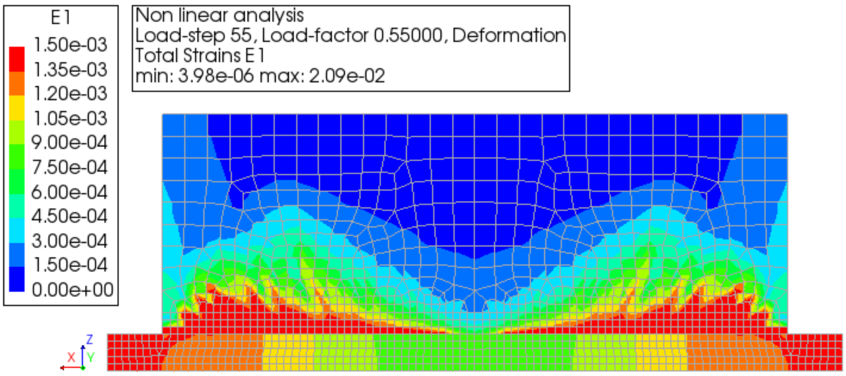


Figure 3-6 Vertical microcracks at rebar surface in $\phi 20c60$

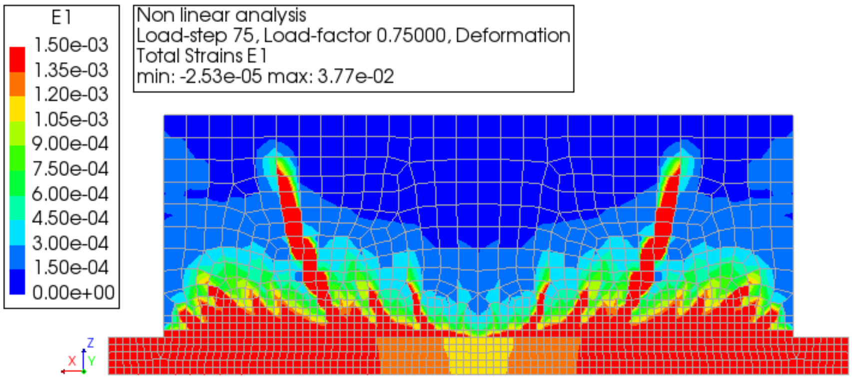


Figure 3-7 Discrete crack at rebar surface in $\phi 20c60$

The force-displacement diagram and vertical internal crack propagation indicate that the finite element model can replicate cracking behavior of a reinforced concrete tie accurately. The numerical results in figure 3-6 and figure 3-7 show that stresses are transferred from the rebar to the concrete through the interface elements in a realistic manner.

3.5 Model validation

To assess how well the finite element model represent the behavior of reinforced concrete ties in loading, two documented experiments on concrete ties by Yannopoulos [18] and Bresler and Bertero [19] have been modeled and used as benchmarks. The overall agreement in the results from the two benchmarking experiments confirms the validity of the model. The agreement in steel strain distribution and crack width estimation support the ability of a three-dimensional numerical element model to represent the behavior of a concrete specimen under tensile loading realistically.

3.5.1 Previous work on concrete ties

Variation of concrete crack widths through the concrete cover to reinforcement by Yannopoulos (1989)

Yannopoulos investigated axially loaded concrete cylinders by studying the relationship between the main crack at the steel surface to that at the external concrete surface. Measurements were taken of concrete surface strains, elongation of the rebar, and the slip between rebar and concrete. Tests on 800 mm long members were done to find the crack distance, which was determined to be 90 mm. Six cylindrical RC ties were investigated of which the lengths were 100 mm to avoid formation of a primary crack. The concrete cylinders had a diameter of 76 mm and were embedded with a steel rebar with a diameter of 16 mm. A sketch of the testing arrangements is shown in figure 3-8. The specimens were monotonically loaded while steel stress and crack widths were measured. Each specimen was gradually loaded to 80% of reinforcement yield load. Crack width was calculated as twice the difference in total deformation at the concrete surface and at the rebar surface, which means that slippage between steel and concrete noticeably governs the crack width estimation [18].

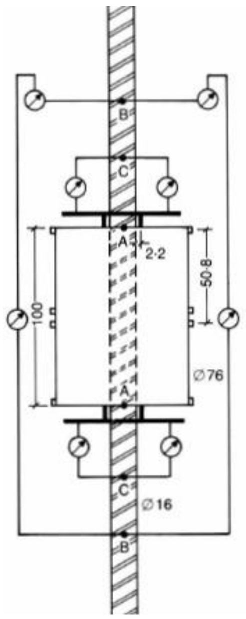


Figure 3-8 Sketch of experimental setup by Yannopoulos [18]

Figure 3-9 shows formation of two principal cracks and half the crack width from the measuring points at the top of the cover and steel surface.

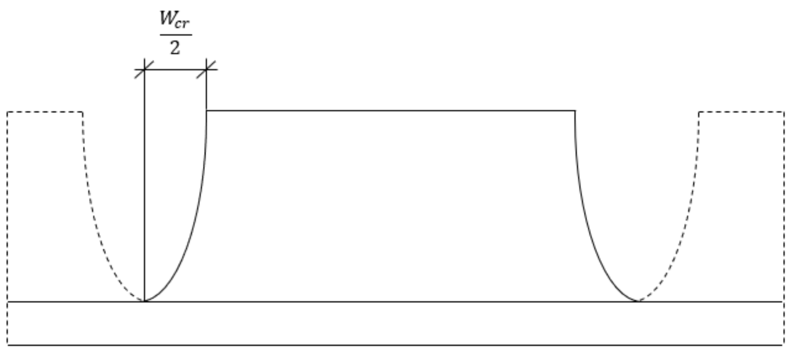


Figure 3-9 Principal crack width calculation from Yannopoulos

In 1968 Bresler and Bertero investigated how load and environmental history contributed to deterioration of concrete, and to develop design criterion to limit cumulative damage such as cracking. Tests were performed on uniaxially loaded concrete cylinders which were meant to represent the behavior in the tensile zone in beams. Prior to the study, pilot studies determined the crack spacing which was used in the specimens.

The specimen had a diameter of 152 mm and a length of 403 mm which was twice the crack spacing of the specimens in the pilot studies. The cylinders were concentrically reinforced with a single steel bar with a diameter of 28.7 mm. Strain gauges were installed in a milled groove which ran through the length of the steel bar with a radius of 5.6 mm in order to record the strain in the specimens during loading. The loading was applied in increments of approximately 9 kilonewtons until sightings of a surface crack were made, upon which the specimen was unloaded. Figure 3-10 shows a sketch of the experimental setup.

If the crack did not go through the entire specimen, loading would be reapplied until the crack was fully propagated through the cover or a prescribed steel stress was reached. The specimens underwent several load cycles with each cycle reaching a set steel stress level [19].

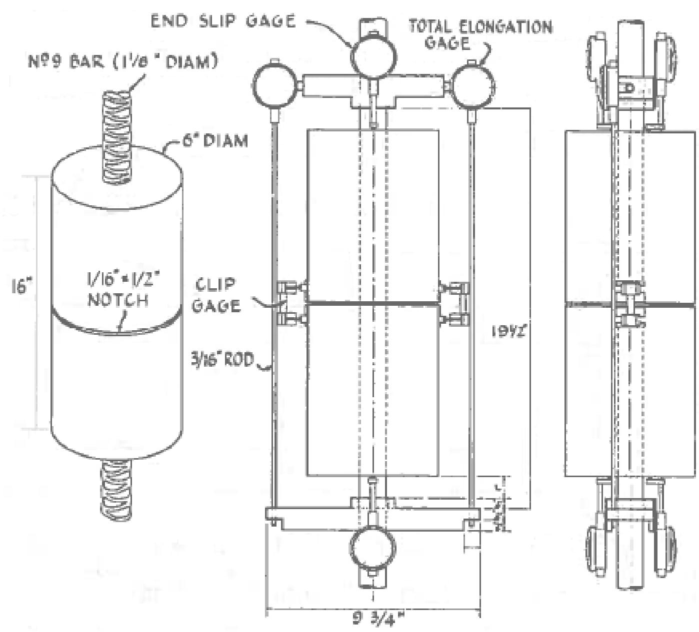


Figure 3-10 Sketch of experimental setup by Bresler and Bertero [19]

3.5.2 Numerical analyses of benchmarking experiments

Yannopoulos (1989)

Figure 3-11 show how the experiment of Yannopoulos was modeled. A non-linear analysis of the Yannopoulos experiment was done as explained in chapter 3.5.1 and [18]. The crack widths were determined by inspecting the difference in total displacement of the concrete nodes at the concrete surface and adjacent to the interface. Since only half the crack is modeled, the difference in displacement in equation 3.6 is multiplied with two.

$$w_{cr} = 2(u_{top} - u_{bottom}) \tag{3.6}$$

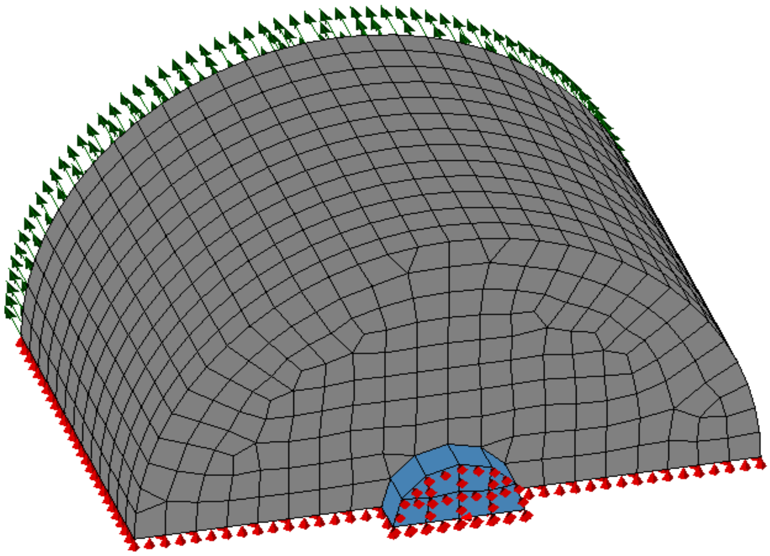


Figure 3-11 Finite element model of the Yannopoulos experiment

Figure 3-12 show a comparison between crack widths calculated from the finite element analysis and the crack widths reported by Yannopoulos. The comparison shows a general good agreement, but it should be noted that the finite element analysis estimate slightly higher crack widths for steel stresses below 300 MPa. It can be observed that around 300 MPa the crack widths from the numerical results intersect with the results of Yannopoulos. At stresses above 300 MPa, the crack widths from the finite element analysis become smaller for a given steel stress. This could be explained by that the finite element model underestimates the extent of internal cracking along the surface of the rebar, which could lead to a slightly lower calculated crack width. For loading in the serviceability area, the numerical model produces credible results.

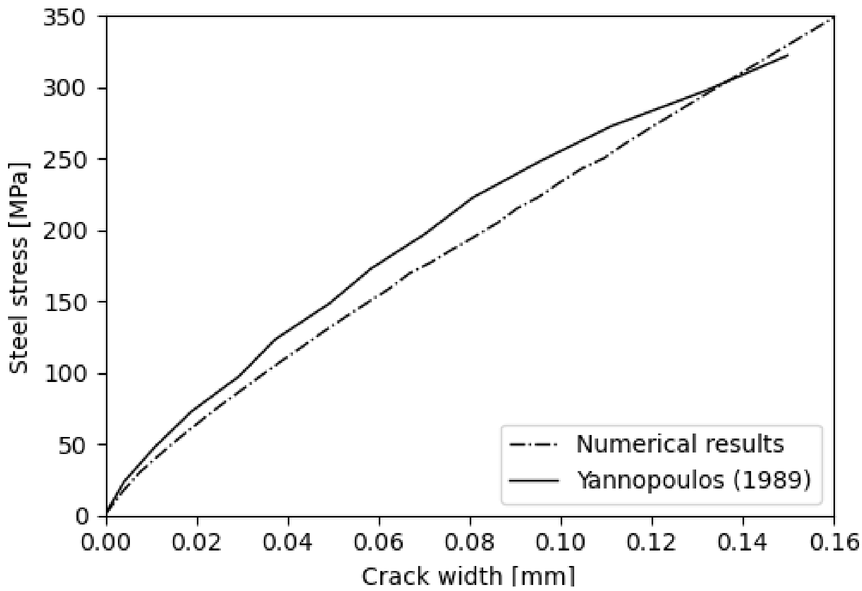


Figure 3-12 Comparison of crack width reported by Yannopoulos and from the finite element analysis

Bresler and Bertero

The finite element model, shown in figure 3-13, was created according to the testing arrangements as explained in 3.5.1 and [19], though the length of the finite element model was set to 201.5 mm, which is equal to the mean crack spacing found in the preliminary study of Bresler and Bertero. The milled groove, in which the strain gauges were placed, was implemented in the model to sample strain in a similar manner as Bresler and Bertero did.

The first loading cycle was carried out by increasing the prescribed deformation over ten load steps until steel stress reached 93 MPa. The next ten load steps gradually reduced the prescribed deformation until the specimen was unloaded. The second load cycle increased the prescribed deformation until the steel stress reached 273 MPa. Upon formation of a primary crack in the symmetry section, results were mirrored so that the numerical results could be compared to the results of Bresler and Bertero. The results were mirrored at 201.5 mm as shown in figure 3-14.

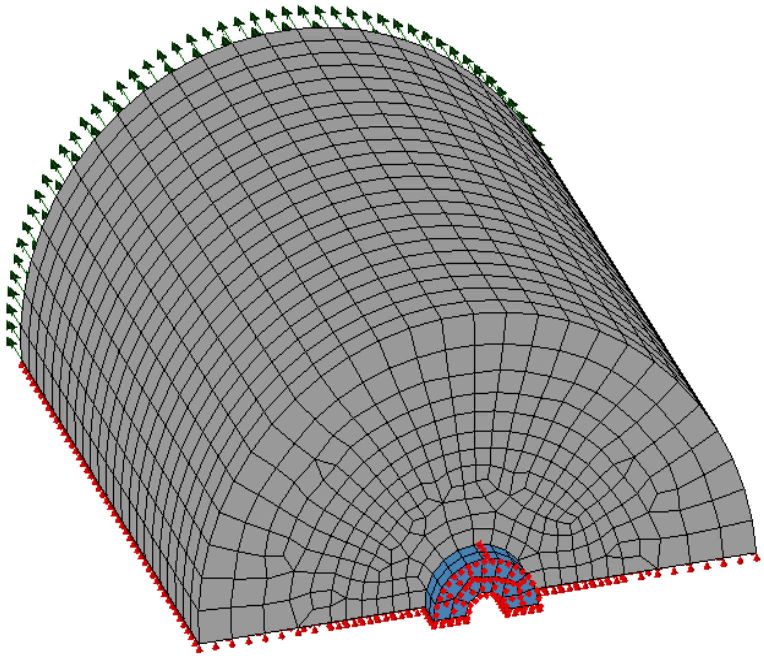


Figure 3-13 Finite element model of the experiment by Bresler and Bertero

Figure 3-14 shows the comparison of steel strain extracted from the two first load cycles of the experiments done in 1968 and from the finite element model. In the first load cycle the results from the two lower stress levels corresponds well to the behavior in the original experiment.

Figure 3-14 Comparison of steel strains reported by Bresler & Bertero and from the finite element analysis

In the second load cycle, the finite element model replicates the steel strain distribution well for the two stress levels but generally estimates lower strain along the reinforcement. This might be explained by how the interface elements represent the bond between steel and concrete. It is explained in MC2010 how cyclic loading can have a detrimental effect on the bond strength between rebars and concrete. Bond slip increases over time with constant load, reducing the bond stress. Repeated loading on the other hand produces a progressive increase in slip between rebar and concrete. This may lead to failure in the bond at stresses lower than the ultimate bond strength compared to monotonic loading. How much lower the failure bond stress depends on number and frequency of load cycles and load level [5]. In the finite element model, there is no mechanism that accurately represents this reduction in bond strength after a load cycle. This lack of representation might lead to an underestimation of slip, and a higher contribution to stiffness from the concrete cover. More contribution from concrete could result in lower steel strain as observed in figure 3-14. However, the results show very good agreement of general strain distribution, and the discrepancy is not important as cyclic loading is not the primary focus in this thesis.

4 Numerical results

The work presented in chapter 4 is divided into three different main phases. Determination of primary crack distances, the more thorough numerical analyses of short specimens and postprocessing of results from the finite element analyses.

To model the specimens, analyses on long subjects were needed to determine mean primary crack distance and the length of the short specimens. Chapter 4.1 presents the results from the initial experiments, consisting of principal strains and bond stress distributions of the four long specimens.

After the mean crack distance and consequently the length of the short specimens was determined, more thorough finite element analyses could be performed on the four short specimens. Stress-crack width diagrams of the specimens are shown in chapter 4.2, yielding information about important events in the loading sequence. Results in chapter 4.3 show axial stress from the symmetry section and cylindrical shear stress distribution plots from the middle of the symmetry section and loaded end.

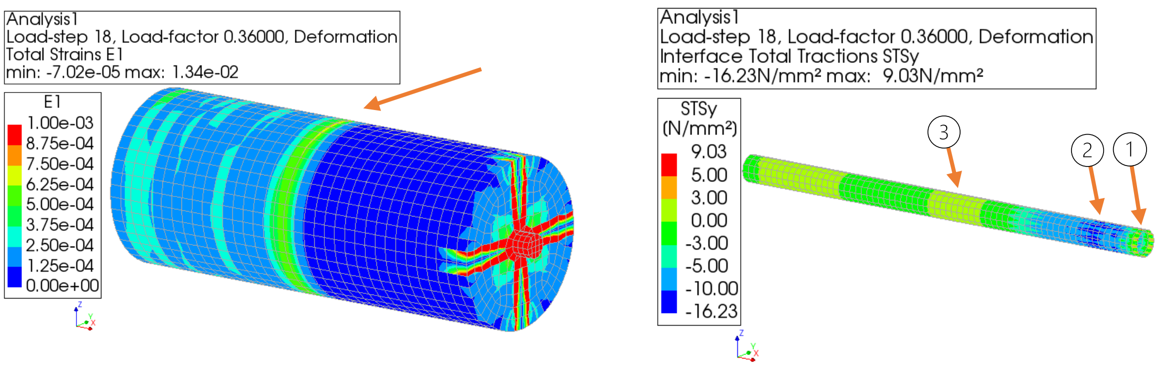
Results from the short specimens were postprocessed using the programming language Python. The post processed results are shown in chapter 4.4 as a bond stress plot and a plot of the bond stress reduction factor ζ along the rebar perimeter.

4.1 Primary crack distance of long specimens

Presented in the following subchapters are figures of distribution of principal strain and bond stress in each specimen at a load step close[2] to formation of a new primary crack. These figures are of interest to the procedure used in determining the crack distance x_{cr} as explained in chapter 3.3. The principal strain distribution indicates where and how the primary crack will form. The figure of bond stress distribution shows how stresses are transferred at the interface. The general behavior in the three regions indicated on the figures of bond stress distributions are:

1. In the region near the loaded end there is no longer bond between the concrete and rebar, since the concrete has radially separated from the rebar due to splitting cracks propagating near the surface of the rebar.
2. Slightly further from the loaded end the concrete has yet to separate from the rebar, and bond stresses in this region are high.
3. This region is centered in the primary crack x_{cr}. Upon formation of a primary crack, negative bond stress will appear just in front the crack.

4.1.1 Singularly reinforced cylinder



a) Principal strain at cracking

b) Bond stress distribution at cracking

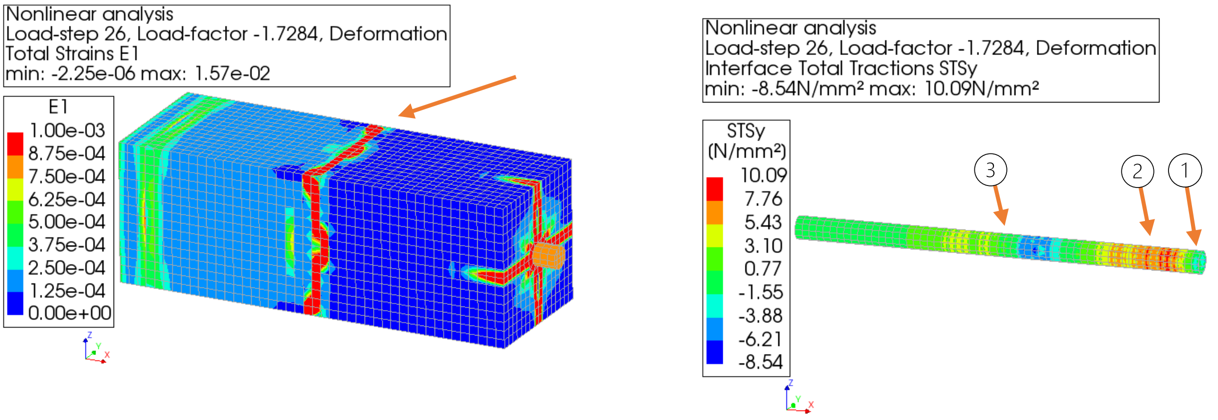
Figure 4-1 Primary crack and bond stress in ϕ20c60

Figure 4-1 shows the principal strain in the model and the bond stress in the interface elements, two load steps prior to formation of a primary crack. The orange arrow in a) marks the location where a primary crack forms 170 mm from the loaded end. In a) it is seen that strain localizes in the vicinity of where the primary crack will propagate, and towards the loaded end strain is decreasing. At the loaded end, a total of eight splitting cracks can be seen protruding from the rebar to the cover. The cracks which are close to each other can be regarded as one crack. The reason for this is that the specimen was modeled as a quart, as shown in figure 3-2. The crack forms adjacent to the symmetry section in the quart model, and due to mirroring appears as separate cracks.

In b) it is seen that the bond stress distribution is even, except for in region 2 where the distribution is uneven due to local cracking and initiation of debonding.

[2] Some of the specimens have been sampled prior to formation of a primary crack, others post formation. This is merely done to better illustrate how the strain concentrate around the primary crack distance x_{cr}.

4.1.2 Singularly reinforced square prism



a) Principal strain at cracking

b) Bond stress distribution at cracking

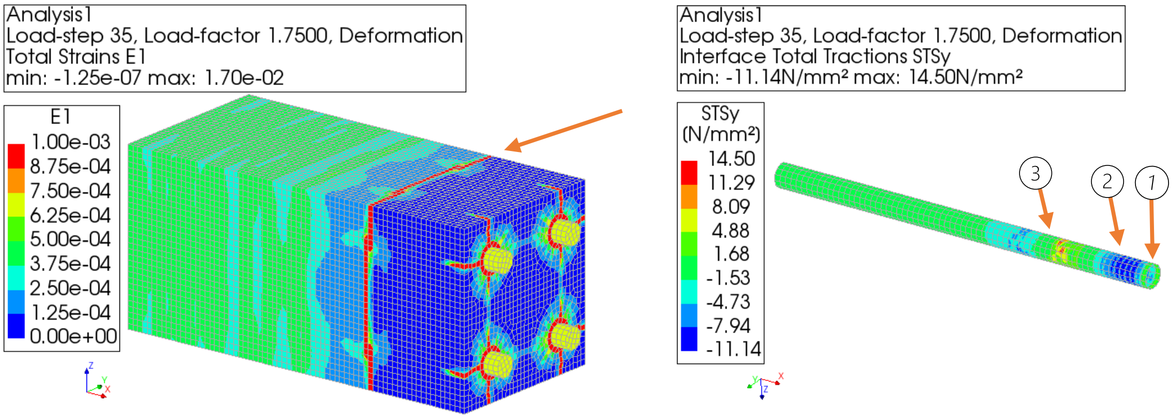
Figure 4-2 Primary crack and bond stress in $\phi 20c52Q$

Figure 4-2 is sampled four load steps after formation of a primary crack, 175 mm from the loaded end, as indicated by the orange arrow in a). The crack is even, which is to be expected since rebar placement and concrete cover is symmetric. Four splitting cracks propagating from the rebar towards the concrete cover can be seen in a), which is similar to the previous specimen. The cover thickness in this specimen is 8 mm smaller than the singularly reinforced cylinder, and the magnitude of strain at the concrete surface is larger.

The bond stress distribution shown in b)[3] is even, but there is a concentration of bond stress prior to region 3 indicated in blue which are negative. These bond stresses are negative because a new crack has formed, and equilibrium must be maintained within that section.

[3] Note that there is positive bond stress at the loaded end, which is opposite of the other three specimens. This is because local axis direction in the interface is switched.

4.1.3 Square prism with four spaced rebars



a) Principal strain at cracking

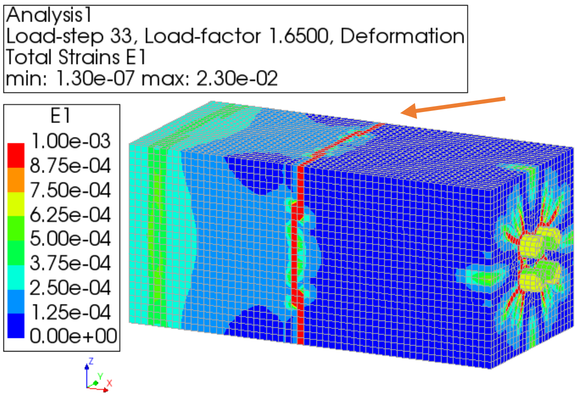
b) Bond stress distribution at cracking

Figure 4-3 Primary crack and bond stress in 4ϕ20c25Q

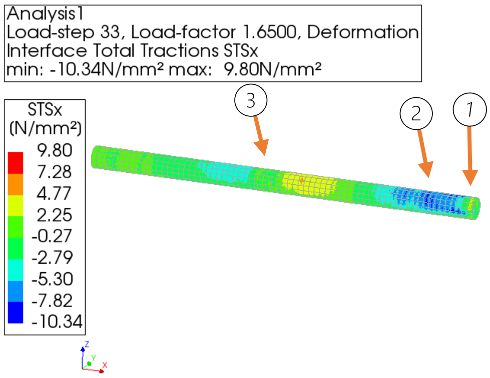
Figure 4-3 is sampled two load steps after the primary crack has formed shown with an orange arrow. In a) the crack has formed 95 mm from the loaded end. Splitting cracks propagate from the rebars towards the concrete cover. At the loaded end, there is a zone around each rebar that is experiencing increased levels of strain. The strain distributes evenly, except in the areas between the rebars where the strains meet.

In b) the figure is rotated 180 degrees, thus displaying the bond stress inwards to the center of the specimen. Even distribution of bond stress can be seen in region 2. A concentration of bond stress can be seen in region 3, indicated in red. This is stress that forms in front of the primary crack as explained for the previous specimen. Upon further loading the bond stress concentration will spread out around the rebar perimeter.

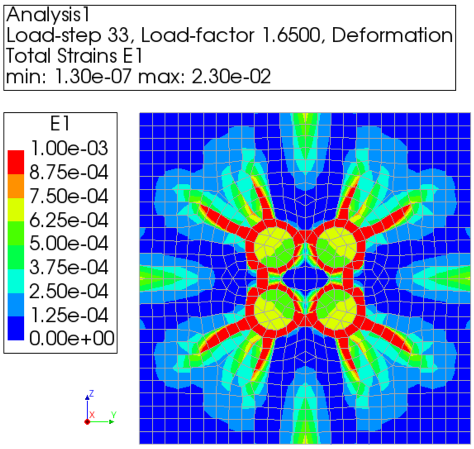
4.1.4 Square prism with four concentrated rebars



a) Principal strain at cracking



b) Bond stress distribution at cracking



c) Principal strain in the loaded

Figure 4-4 Primary crack and bond stress in 4ϕ20c55Q

Figure 4-4 a) shows a primary crack that has formed 180 mm from the loaded end, indicated by an orange arrow, two load steps after formation. In c) there is a better view of the loaded end, which shows splitting cracks protruding diagonally and cracking between rebars. The area in center of the rebars experiences far less strain than the other areas in the cross section. In b) the bond stress in region 2 concentrates on one side of the rebar. The bond stress reaches its largest value facing the outer corners of the specimen, where cover thickness is largest. Towards the center of the specimen, there is not much bond stress.

4.2 Stress-crack width diagram of short specimens

Figure 4-5 shows a plot of the steel stress and estimated crack width for the four short specimens. Principal stress was sampled in the loaded end, and displacements was sampled at the corner node of the concrete cover and on the surface of the rebar. The loadstep in which steel stress corresponding to primary cracking in the longer specimens is marked on the graph with a red dot. This is an notable step, as the stress distributions in chapter 4.3 were sampled at this loadstep.

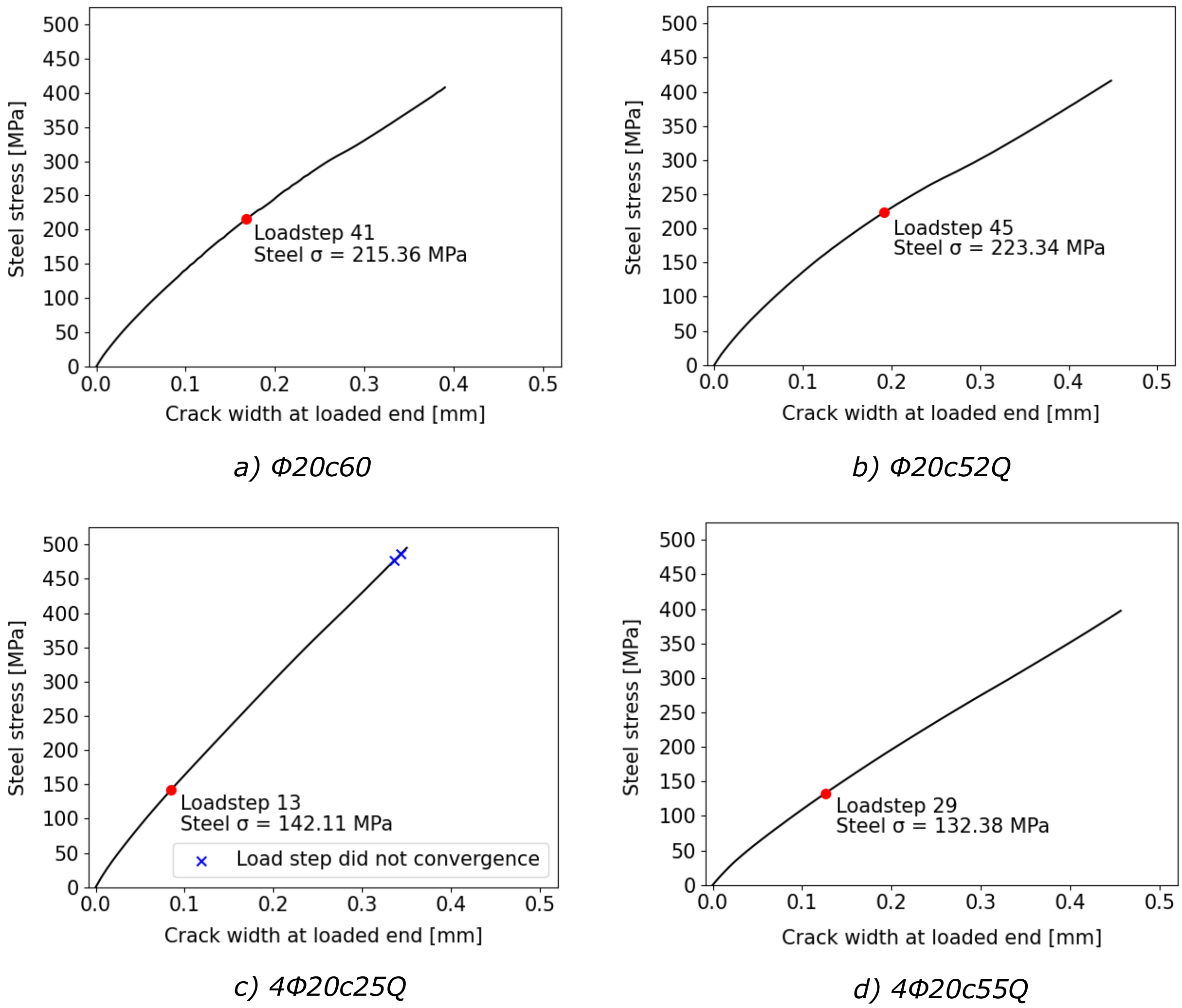


Figure 4-5 Stress-crack width diagram of the four short specimens

4.3 Tensile and shear stress distribution of short specimens

In the figures presenting tensile and shear stresses, stress below 1 MPa have been excluded from the plots and are shown as white areas. This is done to better highlight the larger stress levels. Tensile stress was sampled in the symmetry section, referred to as $L/2$. The shear stress was sampled in the middle between the symmetry section and loaded end, referred to as $L/4$.

4.3.1 Singularly reinforced cylinder

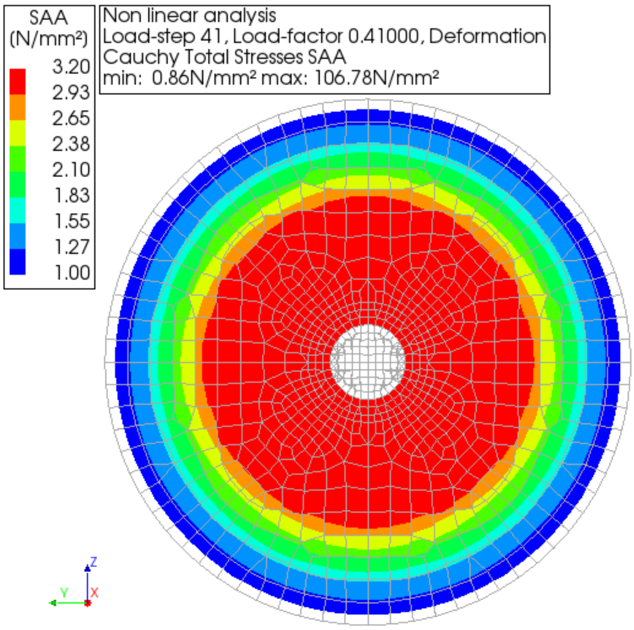


Figure 4-6 Axial tensile stress of $\phi 20c60$ at x=L/2

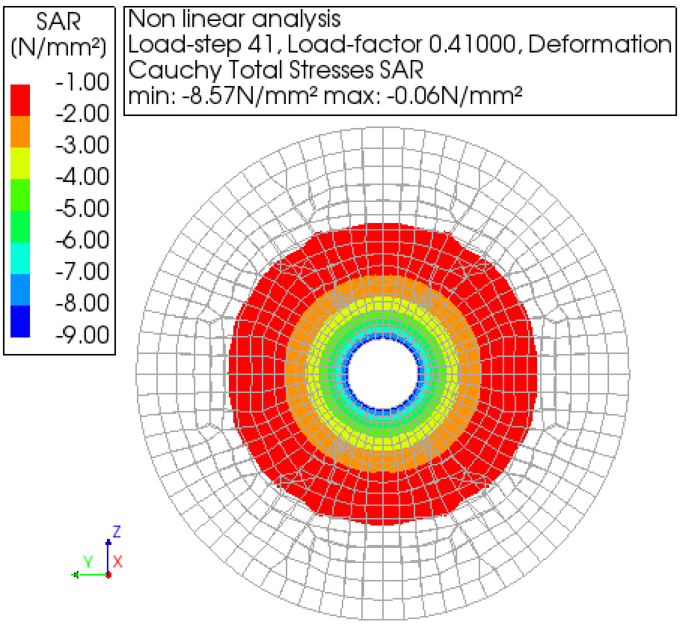


Figure 4-7 Cylindrical shear stress of $\phi 20c60$ at x=L/4

4.3.2 Singularly reinforced square prism

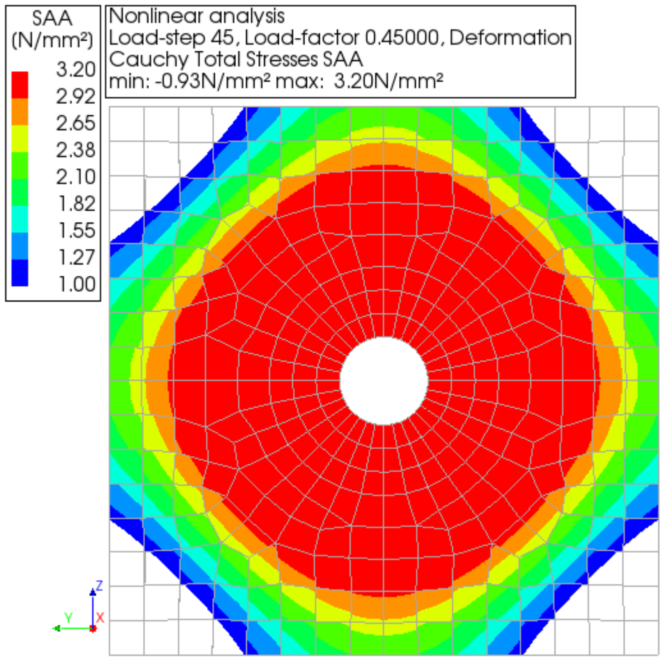


Figure 4-8 Axial tensile stress of $\phi 20c52Q$ at x=L/2

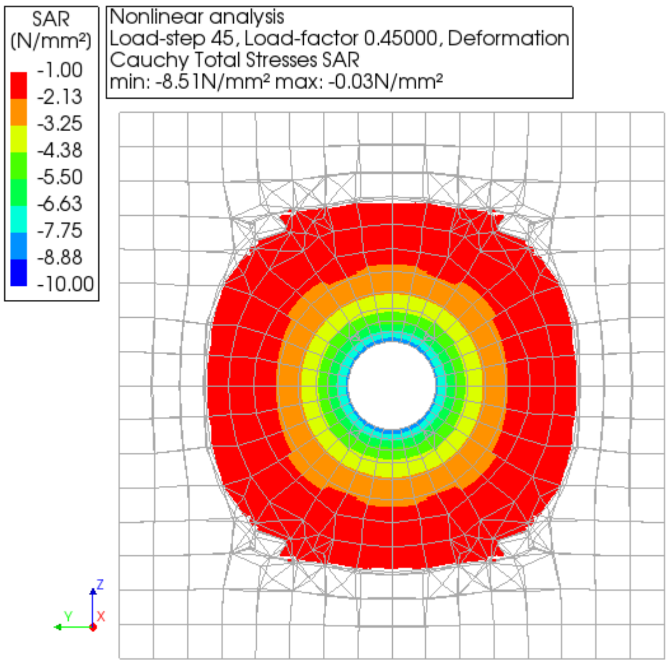


Figure 4-9 Cylindrical shear stress of $\phi 20c52Q$ at x=L/4

4.3.3 Square prism with four spaced rebars

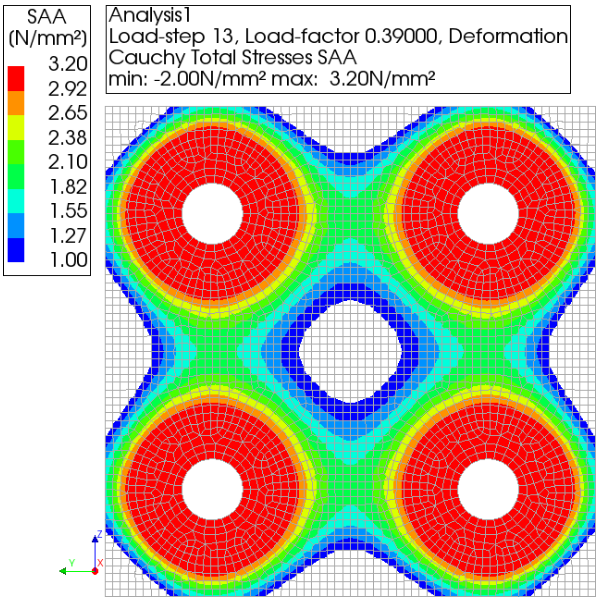


Figure 4-10 Axial tensile stress of 4ϕ20c25Q at x=L/2

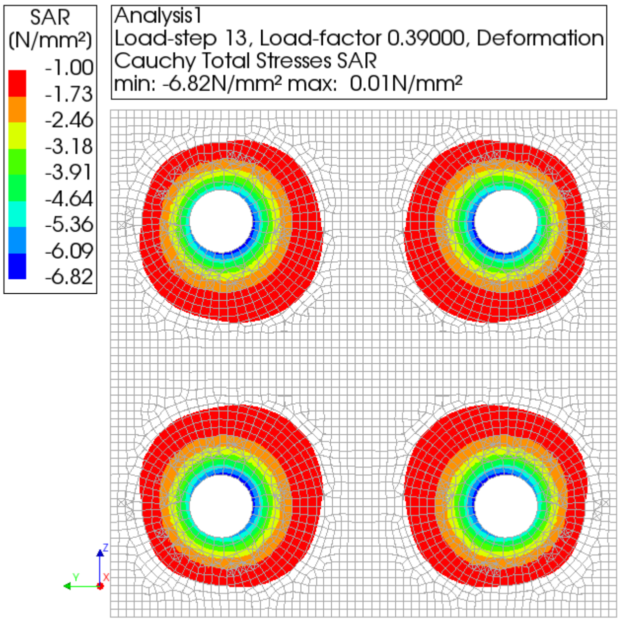


Figure 4-11 Cylindrical shear stress of 4ϕ20c25Q at x=L/4

4.3.4 Square prism with four concentrated rebars

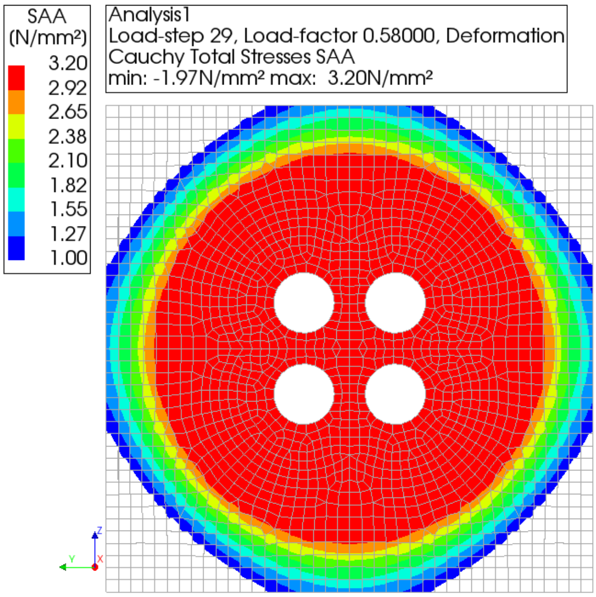


Figure 4-12 Axial tensile stress of 4ϕ20c55Q at x=L/2

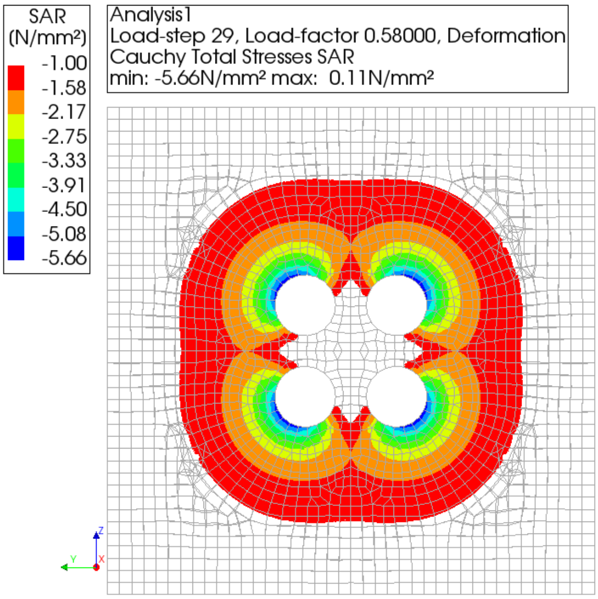


Figure 4-13 Cylindrical shear stress of 4ϕ20c55Q at x=L/4

4.4 Bond stress reduction factor ζ along perimeter of rebars

Figure 4-14 shows a plot of the bond stress at a cross section the middle between the symmetry section and loaded end, referred to as $L/4$. The load corresponds to the stress at the load step just before a primary crack occurs in the longer specimens as indicated in the stress-crack width diagrams in figure 4-5. Bond stress has been radially plotted around the rebars, so that a distance farther from the surface indicate higher bond stress.

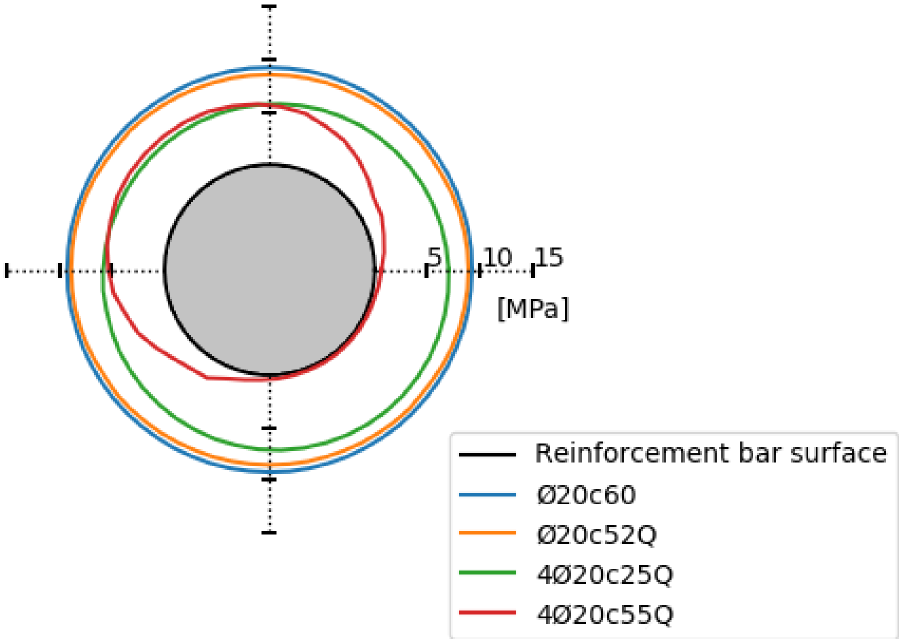


Figure 4-14 Bond stress distribution at x=L/4

The bond stress was sampled in the integration points on the interface elements, and an averaged value was calculated for each element.

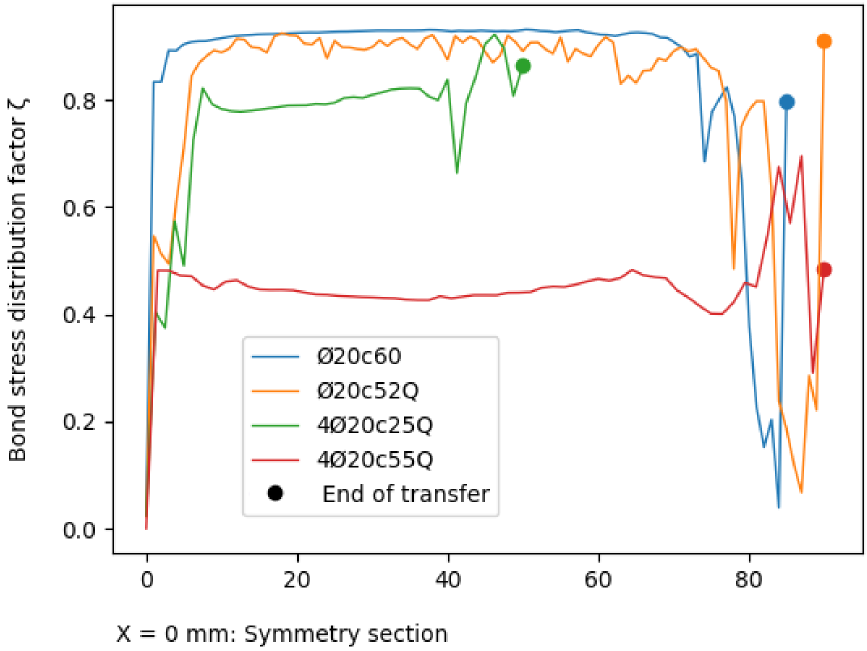


Figure 4-15 Bond stress reduction factor ζ along rebar surface

Based on the bond plot in figure 4-14, a parameter ζ has been derived, which is the ratio of the mean bond stress divided by the maximum bond stress in an arbitrary section, given by equation 4.1.

$$\zeta(x) = \frac{\tau_{bm}(x)}{\tau_{max}(x)} \tag{4.1}$$

The bond stress was sampled in the integration points on the interface elements, and a python script calculated the parameter ζ for positions along the length of the rebar, which were subsequently plotted in figure 4-15. Note that $x = 0$ represents the location of the symmetry section. The dots in the plot marks the end of transfer length for each specimen. The python script used to calculate ζ is included in appendix C.1.

5 Discussion

5.1 Accuracy of the finite element model

The accuracy and ability of the model to represent concrete ties have already been assessed in chapter 3.4 and 3.5, verification and validation of the numerical model. It is of interest, however, to bring up some topics and discuss how they might affect the numerical results.

Bond stress integration

Table 3-6 shows that the bond stress resultant was consistently lower than the crack force. It is a finite element model, with a simplified method of modeling the bond behavior between concrete and steel, and it is therefore not unreasonable to expect some discrepancy between the results. An explanation for the difference could be accredited to the crack force calculation. It is assumed that, when a primary crack forms, the entire cross section cracks. If the crack, or parts of it, is partially opened with tensile stress being transmitted across the crack, as explained in chapter 2.2, the calculation of the crack force would be overestimated. There is a general good agreement between the bond force and the crack force in all four specimens as documented in table 3-6.

Radial restraints at reinforcement symmetry section

Both the singularly reinforced cylinder and square prism are modeled as quarts, as shown in figure 3-2 and explained in 3.2.1. In the loaded end of the specimen, some of the first elements adjacent to each symmetry plane and the reinforcement, as highlighted in figure 5-1, exhibits an unusual concentration of bond stress. The concrete surrounding the rebar tries to separate evenly in a radial manner when the rebar is subjected to tension, but the boundary conditions in the model prohibits the concrete from doing so, and thus restraining forces appear in these elements.

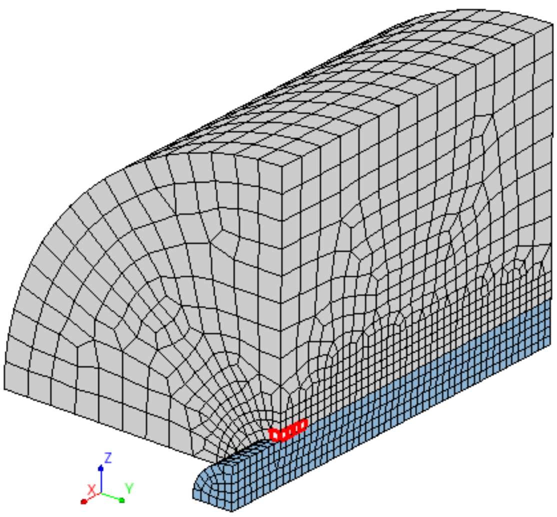


Figure 5-1 Restrained elements in $\phi20c60$

5.2 Transformation of square to equivalent axisymmetric effective tensile area

As explained in chapter 2.3, the effective area refers to concrete in tension surrounding a rebar. EC2 transforms this arbitrary effective area into an equivalent axisymmetric area. It is of interest to discuss the validity of this transformation by searching for indications of an effective area from the numerical results. The effective area is described as a tensile zone in EC2 and it is therefore pertinent to look at tensile and shear stress distributions from the numerical experiments to illustrate this area.

Singularly reinforced specimens

If comparing the axial tensile stress of the singularly reinforced cylinder and square prism, in figures figure 4-6 and figure 4-8, it can be seen that both specimens display similar circular stress distributions. By comparing the extent of tensile stress, it seems that the circular specimen has a slightly larger area than the square prism, but they are quite similar. The shear stress distribution in the cylindric and prismatic specimen, shown in figure 4-7 and figure 4-9, are both circular throughout the cover. This is expected as they have an equivalent concrete area but are shaped differently. The results seem to give a clear indication that the transformation is a valid part of the crack width calculation. However, when studying the stress distributions from the multiple reinforced specimens, the picture is not quite as clear.

Multiply reinforced specimens

The axial tensile stress distribution of the prism with spaced rebars in figure 4-10 shows that a circular tensile zone form around each rebar. The white areas in the center, on the edges and in the corners of the specimen, indicate that they contribute less or not at all to tension stiffening. The shear stress shown in figure 4-11 also concentrate around the rebars in circular areas.

The stress distributions in the prismatic specimen with concentrated rebars differ from the specimen with spaced rebars. The rebars are interacting which is clear from plots in figure 4-12 and figure 4-13, showing the axial tensile stress and shear stress respectively. The lack of shear stress towards the center of the specimen can be explained by the proximity of the rebars. Since they are close to each other there is less concrete between the rebars which leads to lower bond and shear stress. This explanation is supported by the plot in figure 4-14 which show the bond stress distributions around the rebars. The figure shows that the square prism with four concentrated rebars have almost zero bond stress on the surface facing other rebars. The bond stress distribution of the other specimens is almost uniform.

The stress distributions presented in chapter 4.3 and 4.4 show that there are indications of different effective areas that vary with rebar spacing and cover thickness. The results support the idea from EC2 that an arbitrary cross section can be transformed into an equivalent axisymmetric section. However, factors such as uneven cover size and rebar spacing should be considered in the transformation.

5.3 Influence of cover size and reinforcement spacing on the effective area

The European standard for design of concrete structures, EC2, gives clear guidelines on how to determine minimum cover thickness and rebar spacing. It is therefore of interest to discuss how the effective area and bond stress distribution are affected by these two factors.

Rebar spacing

Rebar proximity seems to have a significant effect on bond stress as shown in figure 4-14. Bond stress is practically zero at the surface close to the other rebars in the specimen with concentrated rebars. The spacing between rebars is only 10 mm, which can be considered unrealistic from a designer's perspective. In the specimen with four spaced rebars, the distribution is more even.

The reduction factor ζ introduced in chapter 4.4 shows the relationship between mean and maximum bond stress. This factor indicates how much of the rebar that contributes to bond. The plot in figure 4-15 shows that ζ converges to specific values for the different specimens. The square prism with spaced rebars converges to approximately $\zeta = 0.8$, and the concentrated rebars $\zeta = 0.5$. The reduction clearly shows that concentrated rebar spacing has a detrimental effect on ζ.

Rebar spacing also seems to influence the effective tensile area. If the proximity of rebars reduces the bond stress, it will also reduce the shear stress in the cross section. This can be seen in the square prism with concentrated rebars illustrated in figure 4-13, where the area in the middle has little or no shear stress.

To put rebar spacing in perspective, an example of minimum clear distance will be presented. EC2 advises that minimum clear distance between rebars should be set to the largest of $\{2\phi \,;\, d_g + 5 \,;\, 35\}\, mm$ [7]. If the largest aggregate size d_g is limited to 16 mm and the rebars have a diameter of 20 mm, minimum clear distance would be 40 mm. In the specimen with spaced rebars, which has a clear distance of 70 mm, ζ converges to approximately 0.8. Given that the clear distance, as defined by EC2, can be smaller than 70 mm, it is reasonable to expect that rebar spacing can be small enough to influence the bond stress distribution and effective area.

Cover thickness

Cover thickness seems to influence the bond stress distribution and ζ factor. In the specimen with four concentrated rebars, which has a cover thickness of 25 mm, the results show a small stress difference. The bond stress plot in figure 4-14, shows that the maximum bond stress is located towards the center of the specimen, while the minimum is located towards the outer edge. Studying the cylindrical shear stress in the same specimen in figure 4-11, the extent and magnitude of shear stress in the concrete cover is reduced towards the outer edge of the specimen, where the cover thickness is 25 mm. Given a slightly thicker cover it is reasonable to assume that the bond stress distribution would be more even, which would increase the reduction factor ζ.

By comparing the bond stress from the square prism with spaced and concentrated rebars, sampled at steel stresses of 141.11 and 132.38 MPa respectively, the difference in bond stress is miniscule towards the outer edge of the specimens. The results seem to indicate that a cover thickness of 25 and 55 mm will give the same bond stress. The discrepancy in steel stress at the sampled stage might be the explanation for the small difference in bond stress. It might be the case that if the specimens were loaded further, a larger difference in bond stress could have been observed, as the cover thickness is different.

Based on the results from the two specimens with four rebars, it seems that cover thickness can influence bond stress distribution and the effective area. It is challenging to draw conclusive remarks on the influence of cover thickness based on the results.

5.4 Comparison of stress distributions and effective area in Eurocode 2

EC2 has clear guidelines for determining the effective tensile area of concrete members. It is therefore of interest to see how these effective areas conforms with the ones observed from the numerical results. Two examples of how well the effective area of the square prisms with four rebars conform with the definition in EC2 are presented below. As presented in chapter 2.3, the effective area is defined as $A_{c,eff}$ and is dependent on the effective height of the cross section $h_{c,eff}$, as shown in Equation 2.4. The effective height is the smallest of $\{2.5(h-d)\ ;\ (h-x)/3\ ;\ h/2\}$.

Square prism with four spaced rebars

The dimensions are $h = 160\,mm$ and $d = 125\,mm$, with no compression zone, and the effective height becomes $h_{c,eff} = 70\,mm$. The effective area consists of two rectangular areas in the top and bottom of the cross section as shown in figure 5-2.

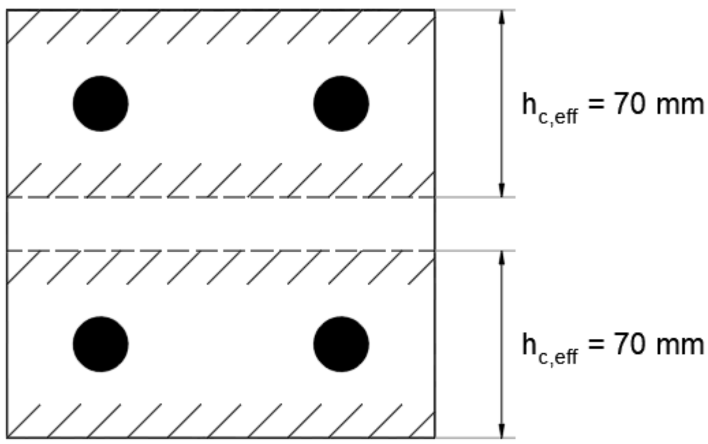


Figure 5-2 Effective area of 4ϕ20c25Q as defined by EC2

If we study the axial tensile stress distribution in figure 4-10, it can be observed that there is a non-contributing white area in the middle of the specimen and around the edges. The tensile stress is also reduced to about 2 MPa between the rebars. Figure 4-11 shows how the cylindrical shear stress concentrate around each rebar, while the non-contributing white area is quite protruding.

Square prism with four concentrated rebars

The dimensions are $h = 160\,mm$ and $d = \ 95\,mm$, with no compression zone, and the effective height becomes $h_{c,eff} = 80\,mm$. This means that the whole concrete cross section will contribute to tension stiffening as shown in figure 5-3.

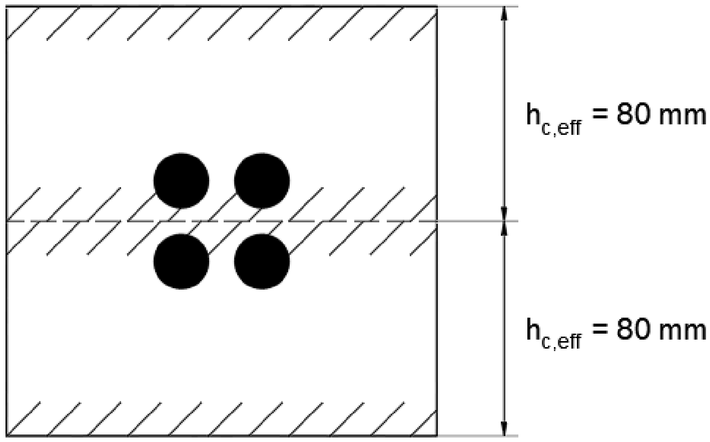


Figure 5-3 Effective area of 4$\phi 20c55Q$ as defined by EC2

If we study the axial tensile stress distribution from figure 4-12, it can be observed that the four rebars interact and form a circular tensile zone in the middle of the specimen. The white area in the corners indicate that there is little to no contribution to tensile stiffness. It can also be noticed that the tensile stress around the edges of the specimen have been considerably reduced. Figure 4-13 shows the shear stress concentrate around the cluster of rebars. In the center of the cluster, as well as in large areas around the edges of the specimen, there is little to no shear stress.

Comparing the effective areas indicated from the numerical results and those defined by EC2, it seems that EC2 overestimates the size of the effective area for the two multiple reinforced specimens. Based on the two comparisons presented, a reduction factor would be favorable to account for rebar spacing.

5.5 Determination of the reduction factor ζ

Based on the numerical results and the previous discussion, the effective area is influenced by cover thickness and rebar spacing. The fact that these two parameters change the effective area, means that some refinement to the definition in EC2 should be applied by introducing a reduction factor accounting for the effect of cover thickness and rebar spacing. A good candidate for this reduction is the factor ζ.

The factor ζ represents a reduction of bond stress, and not a redistribution, as can be observed in the bond stress plot in figure 4-14. By studying the maximum bond stress in the prism with spaced and concentrated rebars, it can be observed that with lower rebar spacing, maximum bond stress remains unchanged, while the mean is reduced.

The factor ζ reduces the total perimeter of rebars contributing to bond. Given a structural member reinforced with 10 bars and $\zeta = 0.5$; only five rebars would contribute to bond. This conforms to the shear distribution of the specimens with four rebars. Figure 4-11 show the prism with spaced rebars, where the whole perimeter of the rebar contributes to bond, while in the prism with concentrated bars in figure 4-13, only parts of the rebar perimeter contributes to bond.

In design of concrete structures and crack width estimations, it is of interest to determine the ζ value in a practical manner with widely known variables. Therefore, the relationship between ζ, rebar spacing a_{bar}, and cover thickness c is presented in equation 5.1.

$$\zeta = \begin{cases} 1.0, & if \quad a_{bar} > 2c \\ 0.75, & if \quad 2c > a_{bar} > c \\ 0.5, & if \quad a_{bar} < c \end{cases} \tag{5.1}$$

6 Conclusions

Numerical results from the finite element analyses show that the effective tensile area around rebars changes with varying cover height and rebar spacing. Small cover thickness and closely spaced rebars will lead to reduced tension stiffening. The results support the idea from EC2 that an arbitrary cross section can be transformed into an equivalent axisymmetric section. However, factors such as uneven cover thickness and rebar spacing should be considered in the transformation.

EC2 overestimates the effective area and tensile stiffening in the two specimens with multiple rebars which have been analyzed and in this thesis. A reduction factor would be favorable to account for the reduction of bond stress contributing to tensile stiffening.

A reduction factor has been derived based on the diminution of bond stress that contribute to tensile stiffening. In practice, this means that the factor ζ will reduce the total perimeter of all rebars contributing to bond.

Results from this thesis indicate that the reduction factor ζ can be estimated to:

$$\zeta = \begin{cases} 1.0, & if \quad a_{bar} > 2c \\ 0.75, & if \quad 2c > a_{bar} > c \\ 0.5, & if \quad a_{bar} < c \end{cases}$$

7 Recommendation for further research

Due to the limited amounts of specimens tested in this thesis, a conclusive remark on the influence of cover has not been achieved. Further studies on specimens with a broader range of cover thicknesses should be conducted to achieve this.

In this thesis the transformation of a square prism to an equivalent axisymmetric section was addressed. The results supported the idea that a square cross section could be transformed into an equivalent axisymmetric section. However, it would be interesting to study the transformation of non-square rectangular cross sections and investigate if the same conclusions can be drawn.

Further investigation of the reduction factor ζ, by extending the experimental setup to include realistic structural components. It would be interesting to see if the same conclusions can be drawn if analyzing a beam, slab or general member in tension. A study on the reduction factor ζ based on analyses of structural components would increase the utility value and usefulness of the factor in crack width calculations.

References

[1] D. Hordijk, "Local approach to fatigue of concrete," TU Delft, Delft, 1991.

[2] Rijkswaterstaat Centre for Infrastructure, "Guidelines for Non Linear Finite Element Analysis of Concrete Structures," The Dutch Ministry of Infrastructure, 2017.

[3] P. Feenstra, "Computational Aspects of Biaxial Stress in Plain and Reinforced Concrete," Delft University Press, Delft, 1993.

[4] American Concrete Institute, "Fracture mechanics of Concrete: Concepts, Models and Determination of Material Properties," American Concrete Institute, Detroit, 1989.

[5] International Federation for Structural Concrete (fib), "fib Model Code for Concrete Structures 2010," Wilhelm Ernst & Sohn, Lausanne, 2013.

[6] T. Anderson, Fracture Mechanics: Fundamentals and Applications, 3. ed., Boca Raton, , Florida: Taylor & Francis Group, 2005, pp. 282-293.

[7] Standard Norge, "Eurokode 2: Prosjektering av betongkonstruksjoner. Del 1-1: Allmenne regler og regler for bygninger," Standard Norge, Lysaker, 2004.

[8] T. Reignard, *Discussion on effective tensile area in EC2,* 2020.

[9] R. Tan, *Experimental and theoretical investigation of crack width calculation methods for RC ties,* Trondheim: Structural Concrete, 2017.

[10] R. Tan, M. A. Hendriks and T. Kanstad, "Evaluation of Current Crack Width Calculation Methods According to Eurocode 2 and fib Model Code 2010," Springer International Publishing, 2018.

[11] K. Bell, An engineering approach to Finite Element Analysis of linear structural mechanics problems, Trondheim: Fagbokforlaget, 2013, pp. 303-304.

[12] R. D. Cook, D. S. Malkus, M. E. Plesha and R. E. Witt, Concepts and Applications of Finite Element Analysis, New Jersey: John Wiley & Sons Inc, 2001, pp. 595-596.

[13] J. B. J.G Rots, "Crack models for concrete: Discrete or smeared? Fixed, multi-directional or rotating?," *HERON,* pp. 7-20, 1989.

[14] D. F. BV, "DIANA Material Library," 2010. [Online]. Available: https://dianafea.com/manuals/d942/MatLib/MatLib.html. [Accessed 13 May 2020].

[15] A. L. Lutz and P. Gergely, "Mechanics of Bond and Slip of Deformed Bars in Concrete," *Journal Proceedings,* pp. 711-721, 1 11 1967.

[16] O. C. Choi, "Effect of rib geometry on bond behavior and failure modes," *8th International Conference on Fracture Mechanics of Concrete and Concrete Structures, FraMCoS 2013,* pp. 517 - 524, 2013.

[17] Y. Goto and K. Otsuka, "Experimental studies on cracks formed in concrete around deformed tension bars," *Japan Society of Civil Engineers,* pp. 85-100, 1980.

[18] P. J. Yannopoulos, "Variation of concrete crack widths though the concrete cover to reinforcement," ICE Publishing Ltd., London, 1989.

[19] B. Bresler and V. Bertero, "Behavior of Reinforced Concrete Under Repeated Load," American Society of Civil Engineers, Virginia, 1968.

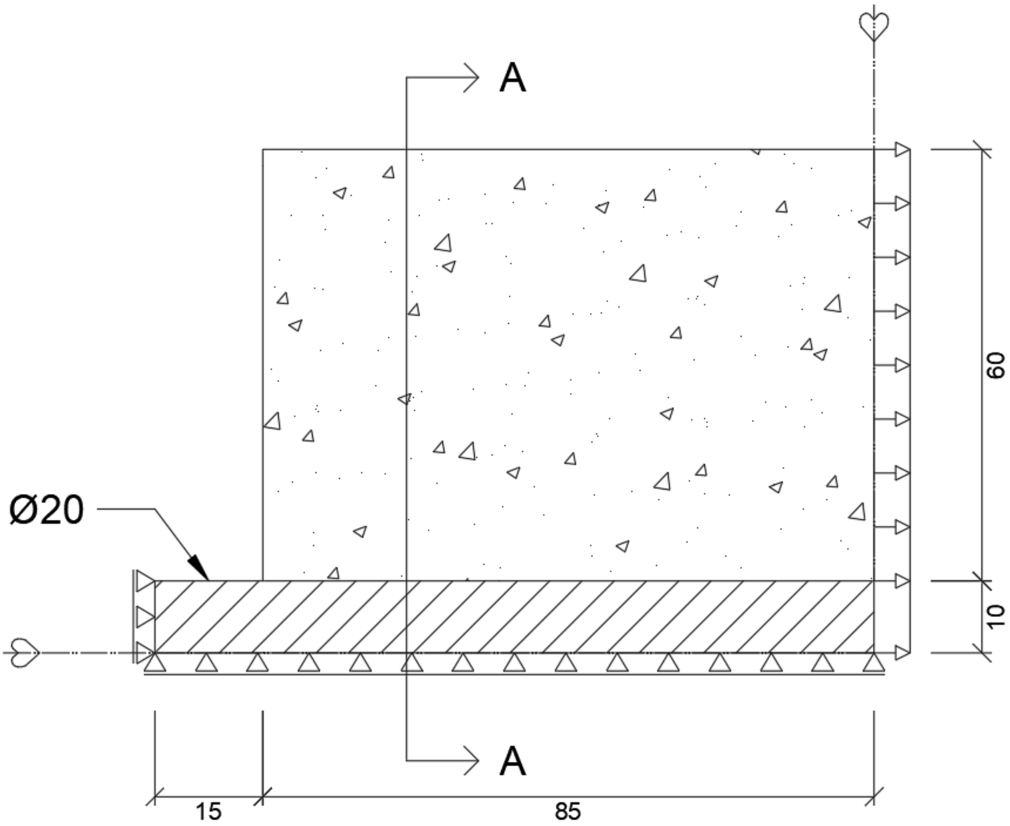
[20] r. Tan, H. Max, M. Geiker and T. Kanstad, "A numerical investigation of the cracking behaviour of reinforced-concrete tie elements," ice publishing, 2020.

[21] J. C. J. Shellekens and R. De Borst, *On the Numerical Integration of Interface Elements,* Delft: Internation Journal for Numerical Methods in Engineering, 1993.

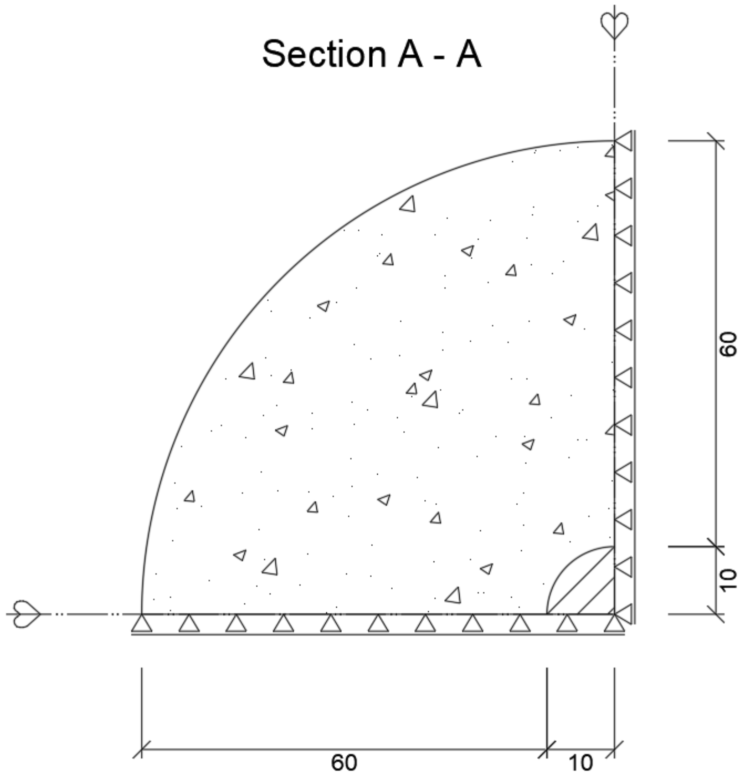
[22] D. FE, "DIANA Users Manual," [Online]. Available: https://dianafea.com/manuals/d100/MatLib/MatLib.html. [Accessed 03 02 2020].

Appendices

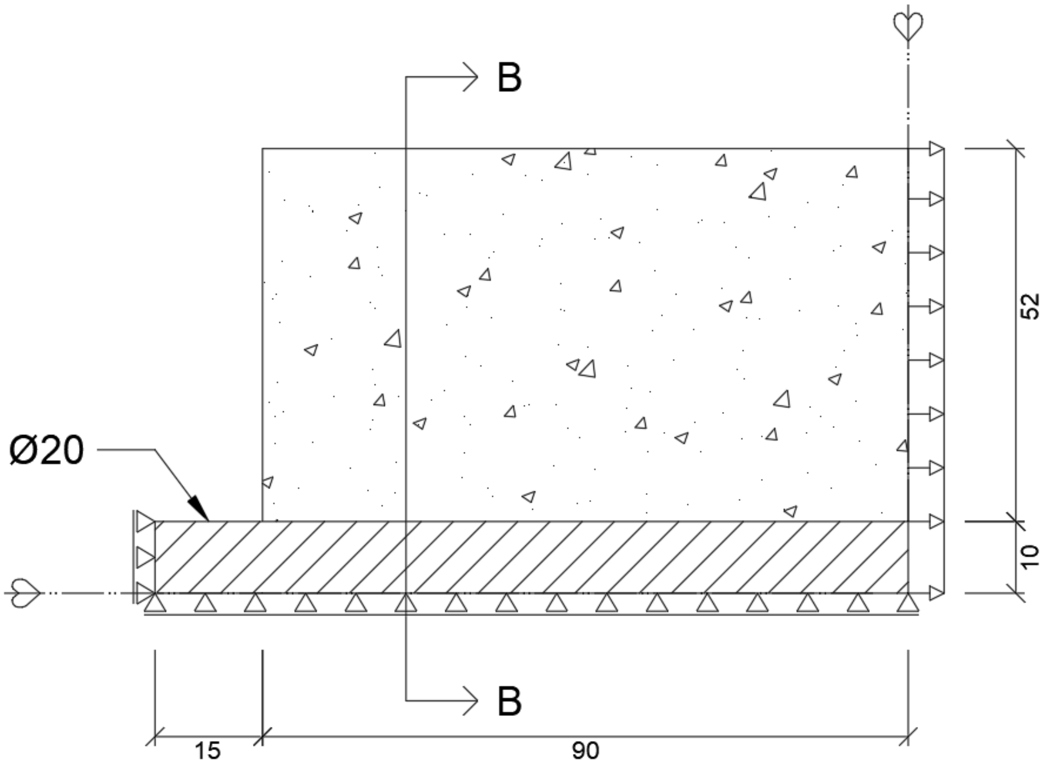
A.1 Singularly reinforced cylinder

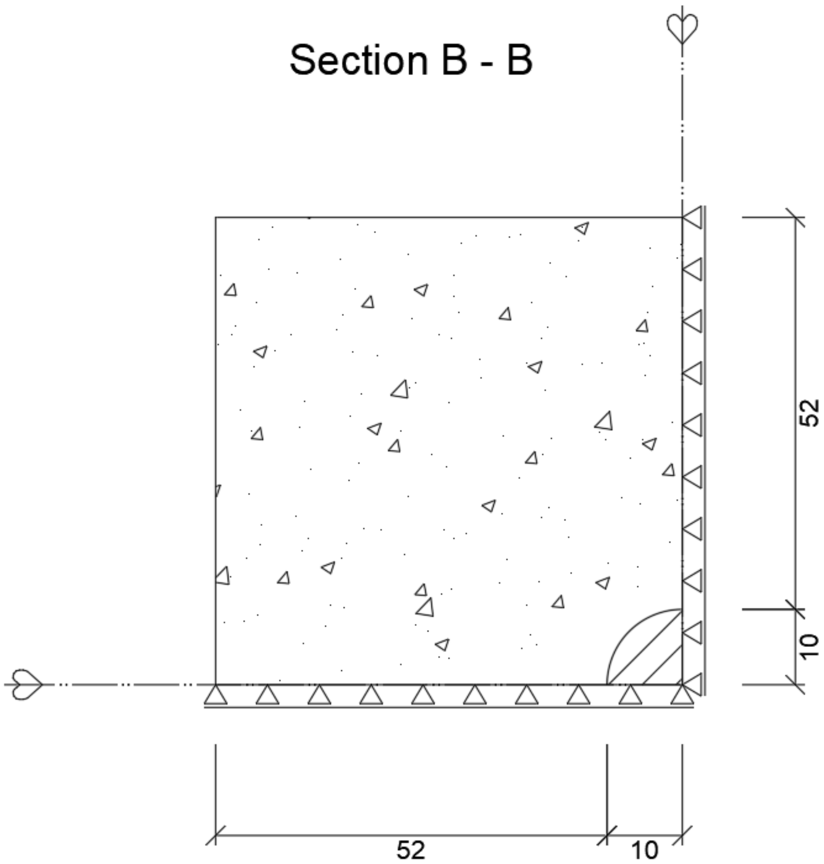


Section A - A

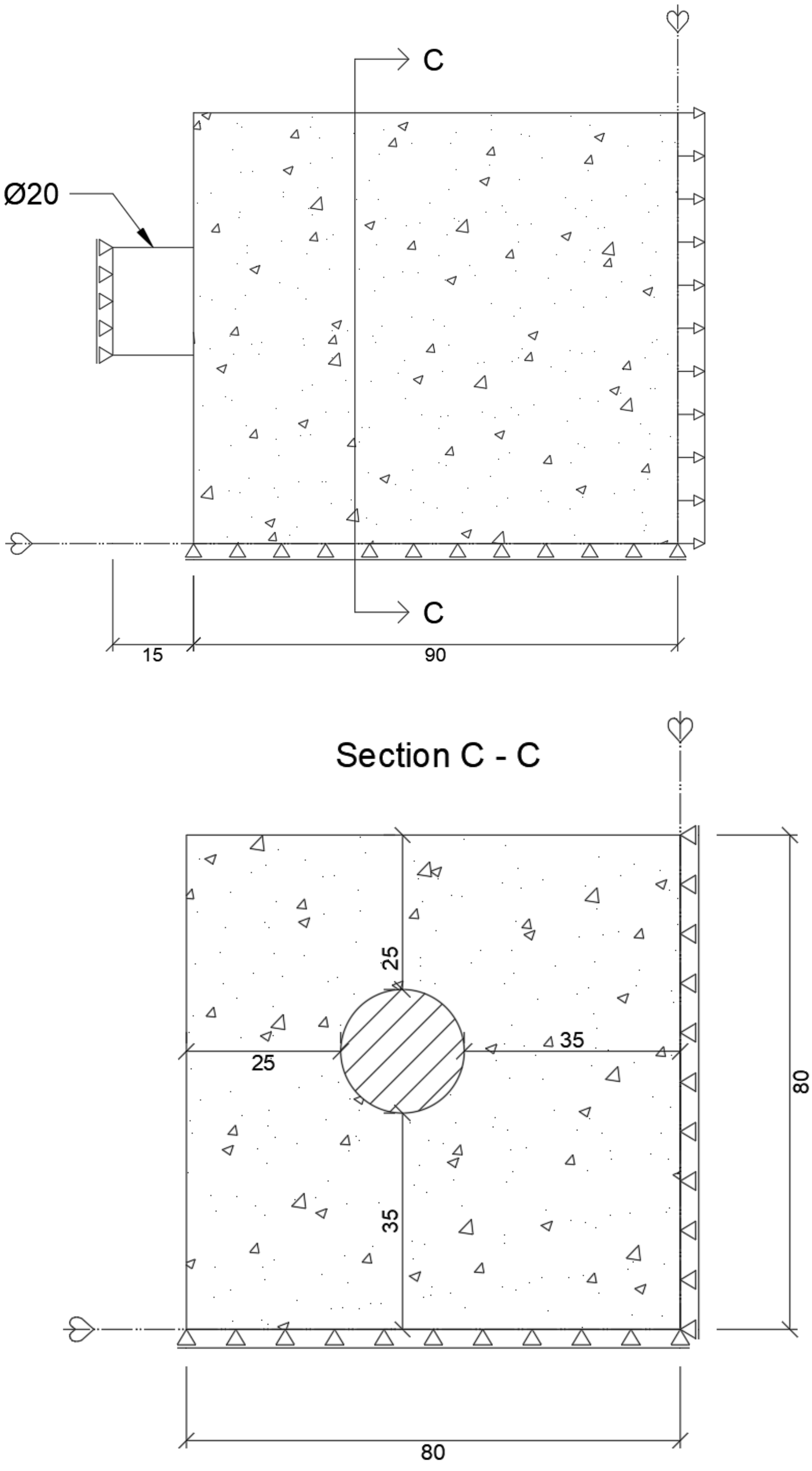


A.2 Singularly reinforced square prism

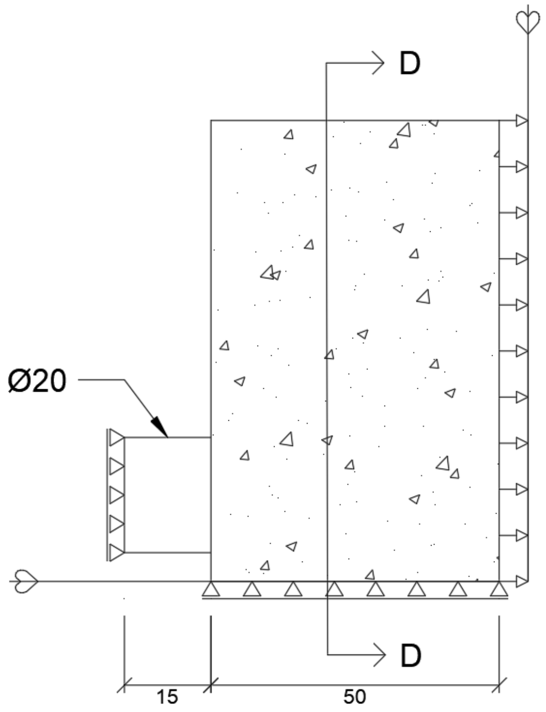




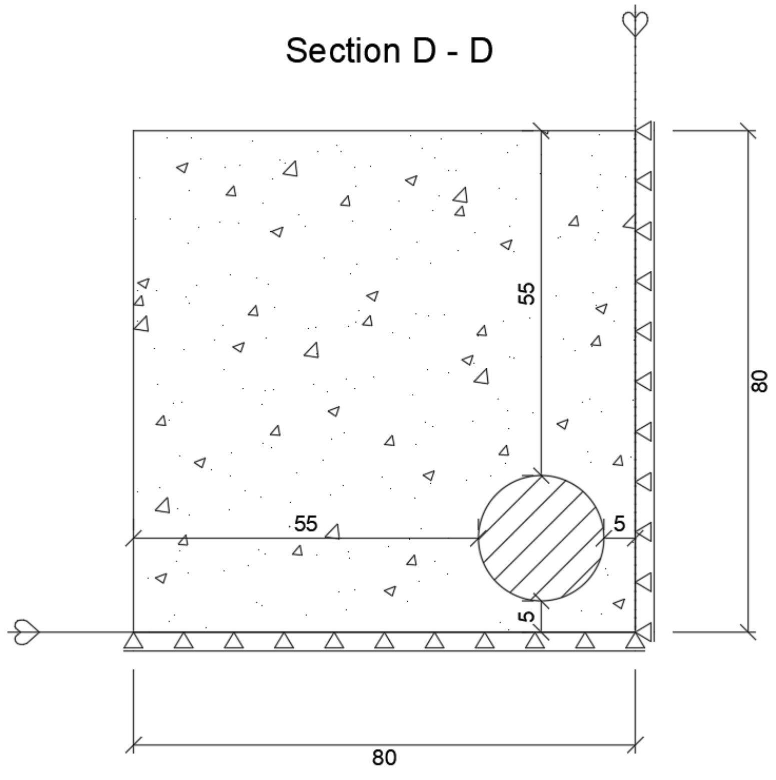
A.3 Square prism with four spaced rebars



A.4 Square prism with four concentrated rebars



Section D - D



B.1 Singularly reinforced square cylinder

```
newProject( "../../", 1 ) #Have to add the correct file path here
setModelAnalysisAspects( [ "STRUCT" ] )
setModelDimension( "3D" )
setDefaultMeshOrder( "QUADRATIC" )
setDefaultMesherType( "HEXQUAD" )
setDefaultMidSideNodeLocation( "LINEAR" )
setUnit( "LENGTH", "MM" )
setUnit( "MASS", "T" )
createCylinder( "Cover", [ 0, 0, 0 ], [ 1, 0, 0 ], 70, 85 )
createCylinder( "Reinforcement", [ 0, 0, 0 ], [ 1, 0, 0 ], 10, 100 )
createSheet( "Sheet 1", [[ -10, -100, 0 ],[ 490, -100, 0 ],[ 490, 400, 0 ],[ -10, 400, 0 ]]
)
createSheet( "Sheet 2", [[ -10, 0, -10 ],[ 490, 0, -10 ],[ 490, 0, 490 ],[ -10, 0, 490 ]] )
subtract( "Cover", [ "Sheet 1" ], True, True )
subtract( "Reinforcement", [ "Sheet 1" ], False, True )
subtract( "Cover", [ "Sheet 2" ], True, True )
subtract( "Reinforcement", [ "Sheet 2" ], False, True )
subtract( "Cover", [ "Reinforcement" ], True, True )
removeShape( [ "Cover_2" ] )
removeShape( [ "Cover_1" ] )
removeShape( [ "Reinforcement_1" ] )
removeShape( [ "Reinforcement_2" ] )
addMaterial( "Concrete", "CONCR", "TSCR", [] )
setParameter( "MATERIAL", "Concrete", "LINEAR/ELASTI/YOUNG", 34962 )
setParameter( "MATERIAL", "Concrete", "LINEAR/ELASTI/POISON", 0.15 )
setParameter( "MATERIAL", "Concrete", "MODTYP/TOTCRK", "ROTATE" )
setParameter( "MATERIAL", "Concrete", "TENSIL/TENCRV", "HORDYK" )
setParameter( "MATERIAL", "Concrete", "TENSIL/TENSTR", 3.2 )
setParameter( "MATERIAL", "Concrete", "TENSIL/GF1", 0.1437 )
setParameter( "MATERIAL", "Concrete", "TENSIL/CBSPEC", "GOVIND" )
setParameter( "MATERIAL", "Concrete", "TENSIL/POISRE/POIRED", "DAMAGE" )
setParameter( "MATERIAL", "Concrete", "COMPRS/COMCRV", "PARABO" )
setParameter( "MATERIAL", "Concrete", "COMPRS/COMSTR", 35 )
setParameter( "MATERIAL", "Concrete", "COMPRS/GC", 35.916 )
addGeometry( "Element geometry 1", "SOLID", "STRSOL", [] )
rename( "GEOMET", "Element geometry 1", "Cyllinder coordinates" )
setParameter( "GEOMET", "Cyllinder coordinates", "AXIAL", True )
setParameter( "GEOMET", "Cyllinder coordinates", "AXIAL/CYLIN", [ 0, -0.001, 0, 1, 0, 0
] )
setParameter( "GEOMET", "Cyllinder coordinates", "AXIAL/CYLIN", [ 0, -0.001, -0.001, 1,
0, 0 ] )
addElementData( "Newton Cotes" )
setElementClassType( "SHAPE", [ "Cover" ], "STRSOL" )
assignMaterial( "Concrete", "SHAPE", [ "Cover" ] )
assignGeometry( "Cyllinder coordinates", "SHAPE", [ "Cover" ] )
assignElementData( "Newton Cotes", "SHAPE", [ "Cover" ] )
addMaterial( "Steel", "MCSTEL", "ISOTRO", [] )
setParameter( "MATERIAL", "Steel", "LINEAR/ELASTI/YOUNG", 200000 )
setParameter( "MATERIAL", "Steel", "LINEAR/ELASTI/YOUNG", 200000 )
setParameter( "MATERIAL", "Steel", "LINEAR/ELASTI/POISON", 0.15 )
setElementClassType( "SHAPE", [ "Reinforcement" ], "STRSOL" )
assignMaterial( "Steel", "SHAPE", [ "Reinforcement" ] )
assignGeometry( "Cyllinder coordinates", "SHAPE", [ "Reinforcement" ] )
assignElementData( "Newton Cotes", "SHAPE", [ "Reinforcement" ] )
```

```
rename( "DATA", "Newton Cotes", "Newton Cotes" )
setParameter( "DATA", "Newton Cotes", "./ALPHAZ", [] )
removeParameter( "DATA", "Newton Cotes", "ALPHAZ" )
setParameter( "DATA", "Newton Cotes", "./ALPHAZ", [] )
removeParameter( "DATA", "Newton Cotes", "ALPHAZ" )
setParameter( "DATA", "Newton Cotes", "./INTEGR", [] )
setParameter( "DATA", "Newton Cotes", "./ALPHAZ", [] )
removeParameter( "DATA", "Newton Cotes", "ALPHAZ" )
setParameter( "DATA", "Newton Cotes", "./NUMINT", [] )
setParameter( "DATA", "Newton Cotes", "NUMINT", [ "NEWCOT" ] )
setParameter( "DATA", "Newton Cotes", "INTEGR", "" )
setParameter( "DATA", "Newton Cotes", "INTEGR", "REGULA" )
setParameter( "DATA", "Newton Cotes", "NUMINT", [ "NEWCOT" ] )
removeParameter( "DATA", "Newton Cotes", "INTEGR" )
setParameter( "DATA", "Newton Cotes", "./NUMINT", [] )
setParameter( "DATA", "Newton Cotes", "./NUMINT", [] )
removeParameter( "DATA", "Newton Cotes", "NUMINT" )
setParameter( "DATA", "Newton Cotes", "./INTEGR", [] )
setParameter( "DATA", "Newton Cotes", "./NUMINT", [] )
hide( "SHAPE", [ "Reinforcement" ] )
addMaterial( "Interface", "INTERF", "NONLIF", [] )
setParameter( "MATERIAL", "Interface", "LINEAR/ELAS6/DSNZ", 349619 )
setParameter( "MATERIAL", "Interface", "LINEAR/ELAS6/DSSX", 152008 )
setParameter( "MATERIAL", "Interface", "LINEAR/ELAS6/DSSY", 152008 )
setParameter( "MATERIAL", "Interface", "NONLIN/IFNOTE", "NOTENS" )
setParameter( "MATERIAL", "Interface", "NONLIN/NLEL7/NOTENS", [ 7.32e-06, 1e-05 ] )
createConnection( "Interface", "INTER", "SHAPEFACE", "SHAPEFACE" )
setParameter( "GEOMETRYCONNECTION", "Interface", "MODE", "CLOSED" )
setElementClassType( "GEOMETRYCONNECTION", "Interface", "STPLIF" )
assignMaterial( "Interface", "GEOMETRYCONNECTION", "Interface" )
assignElementData( "Newton Cotes", "GEOMETRYCONNECTION", "Interface" )
setParameter( "GEOMETRYCONNECTION", "Interface", "FLIP", False )
attachTo( "GEOMETRYCONNECTION", "Interface", "SOURCE", "Reinforcement", [[ 57.3573, -7.8392724, 6.208527 ]] )
attachTo( "GEOMETRYCONNECTION", "Interface", "TARGET", "Cover", [[ 48.753705, -7.8392724, 6.208527 ]] )
show( "SHAPE", [ "Reinforcement" ] )
addSet( "GEOMETRYSUPPORTSET", "BC" )
createSurfaceSupport( "Ux", "BC" )
setParameter( "GEOMETRYSUPPORT", "Ux", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Ux", "TRANSL", [ 1, 0, 0 ] )
setParameter( "GEOMETRYSUPPORT", "Ux", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Ux", "Reinforcement", [[ 100, -5.1880939, 4.26427 ]] )
createSurfaceSupport( "Uy", "BC" )
setParameter( "GEOMETRYSUPPORT", "Uy", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Uy", "TRANSL", [ 0, 1, 0 ] )
setParameter( "GEOMETRYSUPPORT", "Uy", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Uy", "Cover", [[ 48.753705, 0, 35.58562 ]] )
attach( "GEOMETRYSUPPORT", "Uy", "Reinforcement", [[ 57.3573, 0, 4.26427 ]] )
createSurfaceSupport( "Uz", "BC" )
setParameter( "GEOMETRYSUPPORT", "Uz", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Uz", "TRANSL", [ 0, 0, 1 ] )
setParameter( "GEOMETRYSUPPORT", "Uz", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Uz", "Cover", [[ 48.753705, -35.58562, 0 ]] )
attach( "GEOMETRYSUPPORT", "Uz", "Reinforcement", [[ 57.3573, -4.26427, 0 ]] )
addSet( "GEOMETRYSUPPORTSET", "Deformation" )
createSurfaceSupport( "Deformation", "Deformation" )
```

```
setParameter( "GEOMETRYSUPPORT", "Deformation", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Deformation", "TRANSL", [ 1, 0, 0 ] )
setParameter( "GEOMETRYSUPPORT", "Deformation", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Deformation", "Cover", [[ 0, -31.03323, 44.41438 ]] )
attach( "GEOMETRYSUPPORT", "Deformation", "Reinforcement", [[ 0, -4.6984488,
5.73573 ]] )
addSet( "GEOMETRYLOADSET", "Deformation" )
createSurfaceLoad( "Deformation", "Deformation" )
setParameter( "GEOMETRYLOAD", "Deformation", "LODTYP", "DEFORM" )
setParameter( "GEOMETRYLOAD", "Deformation", "DEFORM/SUPP", "Deformation" )
setParameter( "GEOMETRYLOAD", "Deformation", "DEFORM/TR/VALUE", -0.2 )
attach( "GEOMETRYLOAD", "Deformation", "Cover", [[ 0, -31.03323, 44.41438 ]] )
attach( "GEOMETRYLOAD", "Deformation", "Reinforcement", [[ 0, -4.6984488, 5.73573
]] )
setElementSize( [ "Cover" ], 6, -1, True )
clearMesherType( [ "Cover" ] )
clearMidSideNodeLocation( [ "Cover" ] )
setElementSize( [ "Reinforcement" ], 2, -1, True )
clearMesherType( [ "Reinforcement" ] )
clearMidSideNodeLocation( [ "Reinforcement" ] )
generateMesh( [] )
hideView( "GEOM" )
showView( "MESH" )
addAnalysis( "Analysis1" )
renameAnalysis( "Analysis1", "Non linear analysis" )
addAnalysisCommand( "Non linear analysis", "NONLIN", "Structural nonlinear" )
renameAnalysisCommand( "Non linear analysis", "Structural nonlinear", "Structural
nonlinear" )
renameAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"EXECUT(1)", "Deformation" )
setAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"EXECUT(1)/LOAD/STEPS/EXPLIC/SIZES", "0.01(100)" )
addAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"EXECUT(1)/LOAD/LOADNR" )
setAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"EXECUT(1)/LOAD/LOADNR", 1 )
setAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"EXECUT(1)/ITERAT/MAXITE", 30 )
setAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"EXECUT(1)/ITERAT/METHOD/NEWTON/TYPNAM", "MODIFI" )
addAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"EXECUT(1)/ITERAT/LINESE" )
setAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"EXECUT(1)/ITERAT/LINESE", True )
setAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"EXECUT(1)/ITERAT/CONVER/DISPLA", False )
addAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"EXECUT(1)/ITERAT/CONVER/ENERGY" )
setAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"EXECUT(1)/ITERAT/CONVER/ENERGY", True )
setAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"EXECUT(1)/ITERAT/CONVER/ENERGY/TOLCON", 0.001 )
setAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"EXECUT(1)/ITERAT/CONVER/ENERGY/NOCONV", "CONTIN" )
setAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"EXECUT(1)/ITERAT/CONVER/FORCE/NOCONV", "CONTIN" )
```

```
setAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"OUTPUT(1)/SELTYP", "USER" )
addAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"OUTPUT(1)/USER" )
addAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"OUTPUT(1)/USER/DISPLA(1)/TOTAL/TRANSL/LOCAL" )
addAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"OUTPUT(1)/USER/DISPLA(2)/TOTAL/TRANSL/AXIAL" )
addAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"OUTPUT(1)/USER/STRAIN(1)/TOTAL/GREEN/LOCAL" )
addAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"OUTPUT(1)/USER/STRAIN(2)/TOTAL/GREEN/PRINCI" )
addAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"OUTPUT(1)/USER/STRAIN(3)/TOTAL/GREEN/AXIAL" )
addAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"OUTPUT(1)/USER/STRAIN(4)/CRACK/GREEN" )
addAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"OUTPUT(1)/USER/STRAIN(5)/CRKWDT/GREEN/LOCAL" )
setAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"OUTPUT(1)/USER/STRAIN(1)/TOTAL/GREEN/LOCAL/LOCATI", "INTPNT" )
setAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"OUTPUT(1)/USER/STRAIN(2)/TOTAL/GREEN/PRINCI/LOCATI", "INTPNT" )
setAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"OUTPUT(1)/USER/STRAIN(3)/TOTAL/GREEN/AXIAL/LOCATI", "INTPNT" )
addAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"OUTPUT(1)/USER/STRESS(1)/TOTAL/CAUCHY/LOCAL" )
addAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"OUTPUT(1)/USER/STRESS(2)/TOTAL/CAUCHY/PRINCI" )
addAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"OUTPUT(1)/USER/STRESS(3)/TOTAL/CAUCHY/AXIAL" )
setAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"OUTPUT(1)/USER/STRESS(1)/TOTAL/CAUCHY/LOCAL/LOCATI", "INTPNT" )
setAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"OUTPUT(1)/USER/STRESS(2)/TOTAL/CAUCHY/PRINCI/LOCATI", "INTPNT" )
setAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"OUTPUT(1)/USER/STRESS(3)/TOTAL/CAUCHY/AXIAL/LOCATI", "INTPNT" )
addAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"OUTPUT(1)/USER/STRESS(4)/CRACK/CAUCHY/LOCAL" )
addAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"OUTPUT(1)/USER/FORCE(1)/REACTI/TRANSL/LOCAL" )
```

B.2 Singularly reinforced square prism

```
newProject( "../../", 1 ) #Have to add the correct file path here
setModelAnalysisAspects( [ "STRUCT" ] )
setModelDimension( "3D" )
setDefaultMeshOrder( "QUADRATIC" )
setDefaultMesherType( "HEXQUAD" )
setDefaultMidSideNodeLocation( "LINEAR" )
setUnit( "LENGTH", "MM" )
setUnit( "MASS", "T" )
fitAll(  )
createBlock( "Cover_1", [ 0, 0, 0 ], [ 90, 62, 62 ] )
createCylinder( "Cover_2", [ 0, 62, 0 ], [ 1, 0, 0 ], 16, 90 )
createCylinder( "Reinforcement", [ 0, 62, 0 ], [ 1, 0, 0 ], 10, 105 )
subtract( "Cover_1", [ "Cover_2" ], True, True )
subtract( "Cover_2", [ "Reinforcement" ], True, True )
createSheet( "Sheet 1", [[ -10, 0, 0 ],[ 190, 0, 0 ],[ 190, 200, 0 ],[ -10, 200, 0 ]] )
createSheet( "Sheet 2", [[ -10, 62, -20 ],[ 190, 62, -20 ],[ 190, 62, 180 ],[ -10, 62, 180
]] )
subtract( "Cover_2", [ "Sheet 2" ], True, True )
subtract( "Reinforcement", [ "Sheet 2" ], False, True )
subtract( "Cover_2", [ "Sheet 1" ], True, True )
subtract( "Reinforcement", [ "Sheet 1" ], False, True )
removeShape( [ "Cover_2_1" ] )
removeShape( [ "Reinforcement_1" ] )
removeShape( [ "Cover_2_2" ] )
removeShape( [ "Reinforcement_2" ] )
addMaterial( "Concrete", "CONCR", "TSCR", [] )
setParameter( "MATERIAL", "Concrete", "LINEAR/ELASTI/YOUNG", 34962 )
setParameter( "MATERIAL", "Concrete", "LINEAR/ELASTI/POISON", 0.15 )
setParameter( "MATERIAL", "Concrete", "MODTYP/TOTCRK", "ROTATE" )
setParameter( "MATERIAL", "Concrete", "TENSIL/TENSTR", 3.2 )
setParameter( "MATERIAL", "Concrete", "TENSIL/TENCRV", "HORDYK" )
setParameter( "MATERIAL", "Concrete", "TENSIL/GF1", 0.1437 )
setParameter( "MATERIAL", "Concrete", "TENSIL/CBSPEC", "GOVIND" )
setParameter( "MATERIAL", "Concrete", "TENSIL/POISRE/POIRED", "DAMAGE" )
setParameter( "MATERIAL", "Concrete", "COMPRS/COMCRV", "SATURA" )
setParameter( "MATERIAL", "Concrete", "COMPRS/COMCRV", "PARABO" )
setParameter( "MATERIAL", "Concrete", "COMPRS/COMSTR", 35 )
setParameter( "MATERIAL", "Concrete", "COMPRS/GC", 35.916 )
setParameter( "MATERIAL", "Concrete", "COMPRS/REDUCT/REDCRV", "NONE" )
addGeometry( "Element geometry 1", "SOLID", "STRSOL", [] )
rename( "GEOMET", "Element geometry 1", "Cyllinder coordinates" )
setParameter( "GEOMET", "Cyllinder coordinates", "AXIAL", True )
setParameter( "GEOMET", "Cyllinder coordinates", "AXIAL/CYLIN", [ 0, 62.001, -0.0001,
1, 0, 0 ] )
addElementData( "Newton Cotes" )
setElementClassType( "SHAPE", [ "Cover_2", "Cover_1" ], "STRSOL" )
assignMaterial( "Concrete", "SHAPE", [ "Cover_2", "Cover_1" ] )
assignGeometry( "Cyllinder coordinates", "SHAPE", [ "Cover_2", "Cover_1" ] )
assignElementData( "Newton Cotes", "SHAPE", [ "Cover_2", "Cover_1" ] )
addMaterial( "Steel", "MCSTEL", "ISOTRO", [] )
setParameter( "MATERIAL", "Steel", "LINEAR/ELASTI/YOUNG", 200000 )
setParameter( "MATERIAL", "Steel", "LINEAR/ELASTI/YOUNG", 200000 )
setParameter( "MATERIAL", "Steel", "LINEAR/ELASTI/POISON", 0.3 )
addGeometry( "Element geometry 1", "SOLID", "STRSOL", [] )
```

```
setElementClassType( "SHAPE", [ "Reinforcement" ], "STRSOL" )
assignMaterial( "Steel", "SHAPE", [ "Reinforcement" ] )
assignGeometry( "Cyllinder coordinates", "SHAPE", [ "Reinforcement" ] )
assignElementData( "Newton Cotes", "SHAPE", [ "Reinforcement" ] )
hide( "SHAPE", [ "Reinforcement" ] )
addMaterial( "Interface", "INTERF", "NONLIF", [] )
setParameter( "MATERIAL", "Interface", "LINEAR/ELAS6/DSNZ", 349618 )
setParameter( "MATERIAL", "Interface", "LINEAR/ELAS6/DSSX", 152008 )
setParameter( "MATERIAL", "Interface", "LINEAR/ELAS6/DSSY", 152008 )
setParameter( "MATERIAL", "Interface", "NONLIN/IFNOTE", "NOTENS" )
setParameter( "MATERIAL", "Interface", "NONLIN/NLEL7/NOTENS", [ 7.32e-06, 1e-05 ] )
createConnection( "Interface", "INTER", "SHAPEFACE", "SHAPEFACE" )
setParameter( "GEOMETRYCONNECTION", "Interface", "MODE", "CLOSED" )
setElementClassType( "GEOMETRYCONNECTION", "Interface", "STPLIF" )
assignMaterial( "Interface", "GEOMETRYCONNECTION", "Interface" )
assignElementData( "Newton Cotes", "GEOMETRYCONNECTION", "Interface" )
setParameter( "GEOMETRYCONNECTION", "Interface", "FLIP", False )
attachTo( "GEOMETRYCONNECTION", "Interface", "SOURCE", "Reinforcement", [[
60.225165, 54.160728, 6.208527 ]] )
attachTo( "GEOMETRYCONNECTION", "Interface", "TARGET", "Cover_2", [[ 51.62157,
54.160728, 6.208527 ]] )
show( "SHAPE", [ "Reinforcement" ] )
addSet( "GEOMETRYSUPPORTSET", "BC" )
createSurfaceSupport( "Ux", "BC" )
setParameter( "GEOMETRYSUPPORT", "Ux", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Ux", "TRANSL", [ 1, 0, 0 ] )
setParameter( "GEOMETRYSUPPORT", "Ux", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Ux", "Reinforcement", [[ 105, 56.26427, 4.6984488 ]] )
createSurfaceSupport( "Uy", "BC" )
setParameter( "GEOMETRYSUPPORT", "Uy", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Uy", "TRANSL", [ 0, 1, 0 ] )
setParameter( "GEOMETRYSUPPORT", "Uy", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Uy", "Cover_1", [[ 51.62157, 62, 42.384358 ]] )
attach( "GEOMETRYSUPPORT", "Uy", "Cover_2", [[ 51.62157, 62, 12.558562 ]] )
attach( "GEOMETRYSUPPORT", "Uy", "Reinforcement", [[ 60.225165, 62, 4.26427 ]] )
createSurfaceSupport( "Uz", "BC" )
setParameter( "GEOMETRYSUPPORT", "Uz", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Uz", "TRANSL", [ 0, 0, 1 ] )
setParameter( "GEOMETRYSUPPORT", "Uz", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Uz", "Cover_1", [[ 38.37843, 26.384358, 0 ]] )
attach( "GEOMETRYSUPPORT", "Uz", "Cover_2", [[ 51.62157, 49.441438, 0 ]] )
attach( "GEOMETRYSUPPORT", "Uz", "Reinforcement", [[ 60.225165, 57.73573, 0 ]] )
addSet( "GEOMETRYSUPPORTSET", "Deformation" )
createSurfaceSupport( "Deformation", "Deformation" )
setParameter( "GEOMETRYSUPPORT", "Deformation", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Deformation", "TRANSL", [ 1, 0, 0 ] )
setParameter( "GEOMETRYSUPPORT", "Deformation", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Deformation", "Cover_1", [[ 0, 26.438474, 42.384358 ]]
)
attach( "GEOMETRYSUPPORT", "Deformation", "Cover_2", [[ 0, 49.441438, 5.6862617 ]]
)
attach( "GEOMETRYSUPPORT", "Deformation", "Reinforcement", [[ 0, 57.73573,
5.1880939 ]] )
addSet( "GEOMETRYLOADSET", "Deformation" )
createSurfaceLoad( "Deformation", "Deformation" )
setParameter( "GEOMETRYLOAD", "Deformation", "LODTYP", "DEFORM" )
setParameter( "GEOMETRYLOAD", "Deformation", "DEFORM/SUPP", "Deformation" )
```

```
setParameter( "GEOMETRYLOAD", "Deformation", "DEFORM/TR/VALUE", -0.2 )
attach( "GEOMETRYLOAD", "Deformation", "Cover_1", [[ 0, 26.438474, 42.384358 ]] )
attach( "GEOMETRYLOAD", "Deformation", "Cover_2", [[ 0, 49.441438, 5.6862617 ]] )
attach( "GEOMETRYLOAD", "Deformation", "Reinforcement", [[ 0, 57.73573, 5.1880939
]] )
setParameter( "DATA", "Newton Cotes", "./INTEGR", [] )
setParameter( "DATA", "Newton Cotes", "INTEGR", "REGULA" )
setParameter( "DATA", "Newton Cotes", "./INTEGR", [] )
setParameter( "DATA", "Newton Cotes", "./NUMINT", [] )
setParameter( "DATA", "Newton Cotes", "NUMINT", [ "NEWCOT" ] )
setElementSize( [ "Cover_1" ], 8, -1, True )
clearMesherType( [ "Cover_1" ] )
clearMidSideNodeLocation( [ "Cover_1" ] )
setElementSize( [ "Cover_2", "Reinforcement" ], 2, -1, True )
clearMesherType( [ "Cover_2", "Reinforcement" ] )
clearMidSideNodeLocation( [ "Cover_2", "Reinforcement" ] )
generateMesh( [] )
hideView( "GEOM" )
showView( "MESH" )
addAnalysis( "Analysis1" )
renameAnalysis( "Analysis1", "Non linear analysis" )
renameAnalysis( "Non linear analysis", "Non linear analysis" )
addAnalysisCommand( "Non linear analysis", "NONLIN", "Structural nonlinear" )
renameAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"EXECUT(1)", "Deformation" )
setAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"EXECUT(1)/LOAD/STEPS/EXPLIC/SIZES", "0.01(100)" )
addAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"EXECUT(1)/LOAD/LOADNR" )
setAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"EXECUT(1)/LOAD/LOADNR", 1 )
setAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"EXECUT(1)/ITERAT/MAXITE", 30 )
setAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"EXECUT(1)/ITERAT/METHOD/NEWTON/TYPNAM", "MODIFI" )
addAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"EXECUT(1)/ITERAT/LINESE" )
setAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"EXECUT(1)/ITERAT/LINESE", True )
setAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"EXECUT(1)/ITERAT/CONVER/DISPLA", False )
addAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"EXECUT(1)/ITERAT/CONVER/ENERGY" )
setAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"EXECUT(1)/ITERAT/CONVER/ENERGY", True )
setAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"EXECUT(1)/ITERAT/CONVER/ENERGY/TOLCON", 0.001 )
setAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"EXECUT(1)/ITERAT/CONVER/ENERGY/NOCONV", "CONTIN" )
setAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"EXECUT(1)/ITERAT/CONVER/FORCE/NOCONV", "CONTIN" )
renameAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"OUTPUT(1)", "Output" )
setAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"OUTPUT(1)/SELTYP", "USER" )
addAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"OUTPUT(1)/USER" )
```

```
addAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"OUTPUT(1)/USER/DISPLA(1)/TOTAL/TRANSL/LOCAL" )
addAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"OUTPUT(1)/USER/DISPLA(2)/TOTAL/TRANSL/AXIAL" )
addAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"OUTPUT(1)/USER/STRAIN(1)/TOTAL/GREEN/LOCAL" )
addAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"OUTPUT(1)/USER/STRAIN(2)/TOTAL/GREEN/PRINCI" )
addAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"OUTPUT(1)/USER/STRAIN(3)/TOTAL/GREEN/AXIAL" )
setAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"OUTPUT(1)/USER/STRAIN(1)/TOTAL/GREEN/LOCAL/LOCATI", "INTPNT" )
setAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"OUTPUT(1)/USER/STRAIN(2)/TOTAL/GREEN/PRINCI/LOCATI", "INTPNT" )
setAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"OUTPUT(1)/USER/STRAIN(3)/TOTAL/GREEN/AXIAL/LOCATI", "INTPNT" )
addAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"OUTPUT(1)/USER/STRAIN(4)/CRACK/GREEN" )
addAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"OUTPUT(1)/USER/STRAIN(5)/CRKWDT/GREEN/LOCAL" )
addAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"OUTPUT(1)/USER/STRESS(1)/TOTAL/CAUCHY/LOCAL" )
addAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"OUTPUT(1)/USER/STRESS(2)/TOTAL/CAUCHY/PRINCI" )
addAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"OUTPUT(1)/USER/STRESS(3)/TOTAL/CAUCHY/AXIAL" )
setAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"OUTPUT(1)/USER/STRESS(1)/TOTAL/CAUCHY/LOCAL/LOCATI", "INTPNT" )
setAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"OUTPUT(1)/USER/STRESS(2)/TOTAL/CAUCHY/PRINCI/LOCATI", "INTPNT" )
setAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"OUTPUT(1)/USER/STRESS(3)/TOTAL/CAUCHY/AXIAL/LOCATI", "INTPNT" )
addAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"OUTPUT(1)/USER/STRESS(4)/CRACK/CAUCHY/LOCAL" )
addAnalysisCommandDetail( "Non linear analysis", "Structural nonlinear",
"OUTPUT(1)/USER/FORCE(1)/REACTI/TRANSL/LOCAL" )
```

B.3 Square prism with four spaced rebars

```
newProject( "../../", 10 ) # Add file path
setModelAnalysisAspects( [ "STRUCT" ] )
setModelDimension( "3D" )
setDefaultMeshOrder( "QUADRATIC" )
setDefaultMesherType( "HEXQUAD" )
setDefaultMidSideNodeLocation( "LINEAR" )
setUnit( "LENGTH", "MM" )
setUnit( "MASS", "T" )
createBlock( "Cover", [ 0, 0, 0 ], [ 50, 80, 80 ] )
createCylinder( "Reinforcement", [ 0, 35, 45 ], [ 1, 0, 0 ], 10, 65 )
createCylinder( "cover2", [ 0, 35, 45 ], [ 1, 0, 0 ], 25, 50 )
subtract( "Cover", [ "cover2" ], True, True )
subtract( "cover2", [ "Reinforcement" ], True, True )
hide( "SHAPE", [ "Reinforcement" ] )
show( "SHAPE", [ "Reinforcement" ] )
addMaterial( "Interface", "INTERF", "NONLIF", [] )
setParameter( "MATERIAL", "Interface", "LINEAR/ELAS6/DSNZ", 349618.67 )
setParameter( "MATERIAL", "Interface", "LINEAR/ELAS6/DSNZ", 349619 )
setParameter( "MATERIAL", "Interface", "LINEAR/ELAS6/DSSX", 152008.12 )
setParameter( "MATERIAL", "Interface", "LINEAR/ELAS6/DSSY", 152008.12 )
setParameter( "MATERIAL", "Interface", "NONLIN/IFNOTE", "NOTENS" )
setParameter( "MATERIAL", "Interface", "NONLIN/NLEL7/NOTENS", [ 0, 0 ] )
setParameter( "MATERIAL", "Interface", "NONLIN/NLEL7/NOTENS", [ 7.3222635e-06, 0 ]
)
setParameter( "MATERIAL", "Interface", "NONLIN/NLEL7/NOTENS", [ 7.32226e-06, 1e-
05 ] )
addGeometry( "Interface", "SHEET", "STPLIF", [] )
createConnection( "Connection 1", "INTER", "SHAPEFACE", "SHAPEFACE" )
setParameter( "GEOMETRYCONNECTION", "Connection 1", "MODE", "CLOSED" )
setElementClassType( "GEOMETRYCONNECTION", "Connection 1", "STPLIF" )
assignMaterial( "Interface", "GEOMETRYCONNECTION", "Connection 1" )
assignGeometry( "Interface", "GEOMETRYCONNECTION", "Connection 1" )
setParameter( "GEOMETRYCONNECTION", "Connection 1", "FLIP", False )
attachTo( "GEOMETRYCONNECTION", "Connection 1", "SOURCE", "cover2", [[ 28.67865,
26.049588, 40.540165 ]] )
attachTo( "GEOMETRYCONNECTION", "Connection 1", "TARGET", "Reinforcement", [[
37.282245, 26.049588, 40.540165 ]] )
addMaterial( "Concrete", "CONCR", "TSCR", [] )
setParameter( "MATERIAL", "Concrete", "LINEAR/ELASTI/YOUNG", 34961.867 )
setParameter( "MATERIAL", "Concrete", "LINEAR/ELASTI/YOUNG", 34961.9 )
setParameter( "MATERIAL", "Concrete", "LINEAR/ELASTI/POISON", 0.15 )
setParameter( "MATERIAL", "Concrete", "MODTYP/TOTCRK", "ROTATE" )
setParameter( "MATERIAL", "Concrete", "TENSIL/TENCRV", "LINEPS" )
setParameter( "MATERIAL", "Concrete", "TENSIL/TENCRV", "HORDYK" )
setParameter( "MATERIAL", "Concrete", "TENSIL/TENSTR", 3.2 )
setParameter( "MATERIAL", "Concrete", "TENSIL/TENSTR", 3.2 )
setParameter( "MATERIAL", "Concrete", "TENSIL/GF1", 0.14366374 )
setParameter( "MATERIAL", "Concrete", "TENSIL/CBSPEC", "GOVIND" )
setParameter( "MATERIAL", "Concrete", "TENSIL/POISRE/POIRED", "DAMAGE" )
setParameter( "MATERIAL", "Concrete", "COMPRS/COMCRV", "PARABO" )
setParameter( "MATERIAL", "Concrete", "COMPRS/GC", 35.915936 )
setParameter( "MATERIAL", "Concrete", "COMPRS/COMSTR", 35 )
addGeometry( "Element geometry 2", "SOLID", "STRSOL", [] )
rename( "GEOMET", "Element geometry 2", "Cover" )
```

```
setParameter( "GEOMET", "Cover", "AXIAL", True )
setParameter( "GEOMET", "Cover", "AXIAL/CYLIN", [ 0, 79.999, 0.001, 1, 0, 0 ] )
setParameter( "GEOMET", "Cover", "AXIAL/CYLIN", [ 0, 79.999, 0.001, 1, 0, 0 ] )
setElementClassType( "SHAPE", [ "Cover", "cover2" ], "STRSOL" )
# Remark start: For sampling shear stresses in the cover, this element geometry must
be used for the shapes Cover and cover2
#addGeometry( "Concrete shear stress", "SOLID", "STRSOL", [] )
#setParameter( "GEOMET", "Concrete shear stress", "AXIAL", True )
#setParameter( "GEOMET", "Concrete shear stress", "AXIAL/CYLIN", [ 0, 35, 45, 1, 0, 0 ]
)
# Remark end
assignMaterial( "Concrete", "SHAPE", [ "Cover", "cover2" ] )
assignGeometry( "Cover", "SHAPE", [ "Cover", "cover2" ] )
addMaterial( "Reinforcement", "MCSTEL", "ISOTRO", [] )
setParameter( "MATERIAL", "Reinforcement", "LINEAR/ELASTI/YOUNG", 200000 )
setParameter( "MATERIAL", "Reinforcement", "LINEAR/ELASTI/YOUNG", 200000 )
setParameter( "MATERIAL", "Reinforcement", "LINEAR/ELASTI/POISON", 0.3 )
addGeometry( "Element geometry 3", "SOLID", "STRSOL", [] )
rename( "GEOMET", "Element geometry 3", "Reinforcement" )
setParameter( "GEOMET", "Reinforcement", "AXIAL", True )
setParameter( "GEOMET", "Reinforcement", "AXIAL/CYLIN", [ 0, 35, 45, 1, 0, 0 ] )
setElementClassType( "SHAPE", [ "Reinforcement" ], "STRSOL" )
assignMaterial( "Reinforcement", "SHAPE", [ "Reinforcement" ] )
assignGeometry( "Reinforcement", "SHAPE", [ "Reinforcement" ] )
addSet( "GEOMETRYSUPPORTSET", "Boundary conditions" )
createSurfaceSupport( "Ux", "Boundary conditions" )
setParameter( "GEOMETRYSUPPORT", "Ux", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Ux", "TRANSL", [ 1, 0, 0 ] )
setParameter( "GEOMETRYSUPPORT", "Ux", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Ux", "Reinforcement", [[ 65, 36.317017, 44.343753 ]] )
createSurfaceSupport( "Uy", "Boundary conditions" )
setParameter( "GEOMETRYSUPPORT", "Uy", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Uy", "TRANSL", [ 0, 1, 0 ] )
setParameter( "GEOMETRYSUPPORT", "Uy", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Uy", "Cover", [[ 28.67865, 80, 45.88584 ]] )
createSurfaceSupport( "Uz", "Boundary conditions" )
setParameter( "GEOMETRYSUPPORT", "Uz", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Uz", "TRANSL", [ 0, 0, 1 ] )
setParameter( "GEOMETRYSUPPORT", "Uz", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Uz", "Cover", [[ 21.32135, 45.88584, 0 ]] )
addSet( "GEOMETRYSUPPORTSET", "Load" )
createSurfaceSupport( "Deformation", "Load" )
setParameter( "GEOMETRYSUPPORT", "Deformation", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Deformation", "TRANSL", [ 1, 0, 0 ] )
setParameter( "GEOMETRYSUPPORT", "Deformation", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Deformation", "Cover", [[ 0, 7.2407686, 31.168056 ]] )
attach( "GEOMETRYSUPPORT", "Deformation", "Reinforcement", [[ 0, 36.317017,
45.656247 ]] )
attach( "GEOMETRYSUPPORT", "Deformation", "cover2", [[ 0, 20.324542, 37.687474 ]] )
addSet( "GEOMETRYLOADSET", "Tensile load" )
createSurfaceLoad( "Deformation 0.1mm", "Tensile load" )
setParameter( "GEOMETRYLOAD", "Deformation 0.1mm", "LODTYP", "DEFORM" )
setParameter( "GEOMETRYLOAD", "Deformation 0.1mm", "DEFORM/SUPP",
"Deformation" )
setParameter( "GEOMETRYLOAD", "Deformation 0.1mm", "DEFORM/TR/VALUE", -0.1 )
attach( "GEOMETRYLOAD", "Deformation 0.1mm", "Cover", [[ 0, 7.2407686, 31.168056
]] )
```

```
attach( "GEOMETRYLOAD", "Deformation 0.1mm", "Reinforcement", [[ 0, 36.317017,
45.656247 ]] )
attach( "GEOMETRYLOAD", "Deformation 0.1mm", "cover2", [[ 0, 20.324542, 37.687474
]] )
setElementSize( [ "cover2", "Cover", "Reinforcement" ], 2.5, -1, True )
clearMesherType( [ "cover2", "Cover", "Reinforcement" ] )
clearMidSideNodeLocation( [ "cover2", "Cover", "Reinforcement" ] )
generateMesh( [] )
addAnalysis( "Analysis1" )
addAnalysisCommand( "Analysis1", "NONLIN", "Structural nonlinear" )
addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/LOADNR" )
setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/LOADNR", 1 )
setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/STEPS/EXPLIC/SIZES", "0.03(50)" )
setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/STEPS/EXPLIC/SIZES", "0.0300000(50)" )
setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/ITERAT/MAXITE", 10 )
setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/ITERAT/MAXITE", 30 )
setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/ITERAT/METHOD/NEWTON/TYPNAM", "MODIFI" )
addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/ITERAT/LINESE" )
setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/ITERAT/LINESE", True )
setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/ITERAT/CONTIN", True )
addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/ITERAT/CONVER/ENERGY" )
setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/ITERAT/CONVER/ENERGY", True )
setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/ITERAT/CONVER/DISPLA", False )
setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/ITERAT/CONVER/ENERGY/TOLCON", 0.001 )
setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/ITERAT/CONVER/ENERGY/TOLCON", 0.001 )
setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/ITERAT/CONVER/ENERGY/NOCONV", "CONTIN" )
setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/ITERAT/CONVER/FORCE/NOCONV", "CONTIN" )
setAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "OUTPUT(1)/SELTYP",
"USER" )
addAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "OUTPUT(1)/USER" )
addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"OUTPUT(1)/USER/DISPLA(1)/TOTAL/TRANSL/GLOBAL" )
addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"OUTPUT(1)/USER/DISPLA(2)/TOTAL/TRANSL/AXIAL" )
addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"OUTPUT(1)/USER/STRAIN(1)/TOTAL/GREEN/PRINCI" )
addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"OUTPUT(1)/USER/STRAIN(2)/TOTAL/GREEN/AXIAL" )
addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"OUTPUT(1)/USER/STRAIN(3)/TOTAL/GREEN/GLOBAL" )
```

```
addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"OUTPUT(1)/USER/STRAIN(4)/CRACK/GREEN" )
addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"OUTPUT(1)/USER/STRAIN(5)/CRKWDT/GREEN/PRINCI" )
addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"OUTPUT(1)/USER/STRESS(1)/TOTAL/CAUCHY/GLOBAL" )
addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"OUTPUT(1)/USER/STRESS(2)/TOTAL/CAUCHY/PRINCI" )
addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"OUTPUT(1)/USER/STRESS(3)/TOTAL/CAUCHY/AXIAL" )
addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"OUTPUT(1)/USER/STRESS(4)/TOTAL/TRACTI/LOCAL" )
addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"OUTPUT(1)/USER/STRESS(5)/CRACK/CAUCHY/LOCAL" )
addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"OUTPUT(1)/USER/FORCE(1)/REACTI/TRANSL/GLOBAL" )
addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"OUTPUT(1)/USER/ELMFOR(1)/TOTAL/TRANSL/GLOBAL" )
setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"OUTPUT(1)/USER/STRAIN(1)/TOTAL/GREEN/PRINCI/LOCATI", "INTPNT" )
setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"OUTPUT(1)/USER/STRAIN(2)/TOTAL/GREEN/AXIAL/LOCATI", "INTPNT" )
setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"OUTPUT(1)/USER/STRAIN(3)/TOTAL/GREEN/GLOBAL/LOCATI", "INTPNT" )
setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"OUTPUT(1)/USER/STRAIN(5)/CRKWDT/GREEN/PRINCI/LOCATI", "INTPNT" )
setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"OUTPUT(1)/USER/STRESS(1)/TOTAL/CAUCHY/GLOBAL/LOCATI", "INTPNT" )
setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"OUTPUT(1)/USER/STRESS(3)/TOTAL/CAUCHY/AXIAL/LOCATI", "INTPNT" )
setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"OUTPUT(1)/USER/STRESS(4)/TOTAL/TRACTI/LOCAL/LOCATI", "INTPNT" )
```

B.4 Square prism with four concentrated rebars

```
newProject( "../../", 10 ) # Add file path
setModelAnalysisAspects( [ "STRUCT" ] )
setModelDimension( "3D" )
setDefaultMeshOrder( "QUADRATIC" )
setDefaultMesherType( "HEXQUAD" )
setDefaultMidSideNodeLocation( "LINEAR" )
setUnit( "LENGTH", "MM" )
setUnit( "MASS", "T" )
createBlock( "cover", [ 0, 0, 0 ], [ 90, 80, 80 ] )
createCylinder( "reinforcement", [ 0, 65, 15 ], [ 1, 0, 0 ], 10, 105 )
createCylinder( "cover1", [ 0, 65, 15 ], [ 1, 0, 0 ], 25, 90 )
createSheet( "Sheet 1", [[ -40, -40, 0 ],[ 260, -40, 0 ],[ 260, 260, 0 ],[ -40, 260, 0 ]] )
createSheet( "Sheet 2", [[ -40, 80, -40 ],[ 260, 80, -40 ],[ 260, 80, 260 ],[ -40, 80, 260
]] )
subtract( "cover1", [ "Sheet 1", "Sheet 2" ], True, True )
removeShape( [ "cover1_1", "Sheet 1", "cover1_3", "cover1_2", "Sheet 2" ] )
subtract( "cover", [ "cover1" ], True, True )
subtract( "cover1", [ "reinforcement" ], True, True )
hide( "SHAPE", [ "reinforcement" ] )
addMaterial( "Interface", "INTERF", "NONLIF", [] )
setParameter( "MATERIAL", "Interface", "LINEAR/ELAS6/DSNZ", 349618.67 )
setParameter( "MATERIAL", "Interface", "LINEAR/ELAS6/DSNZ", 349619 )
setParameter( "MATERIAL", "Interface", "LINEAR/ELAS6/DSSX", 152008.12 )
setParameter( "MATERIAL", "Interface", "LINEAR/ELAS6/DSSY", 152008.12 )
setParameter( "MATERIAL", "Interface", "NONLIN/IFNOTE", "NOTSHR" )
setParameter( "MATERIAL", "Interface", "NONLIN/IFNOTE", "NOTENS" )
setParameter( "MATERIAL", "Interface", "NONLIN/NLEL7/NOTENS", [ 7.3222635e-06,
1e-05 ] )
show( "SHAPE", [ "reinforcement" ] )
addGeometry( "interface", "SHEET", "STPLIF", [] )
createConnection( "Interface", "INTER", "SHAPEFACE", "SHAPEFACE" )
setParameter( "GEOMETRYCONNECTION", "Interface", "MODE", "CLOSED" )
setElementClassType( "GEOMETRYCONNECTION", "Interface", "STPLIF" )
assignMaterial( "Interface", "GEOMETRYCONNECTION", "Interface" )
assignGeometry( "interface", "GEOMETRYCONNECTION", "Interface" )
setParameter( "GEOMETRYCONNECTION", "Interface", "FLIP", False )
attachTo( "GEOMETRYCONNECTION", "Interface", "SOURCE", "reinforcement", [[
60.225165, 56.049588, 10.540165 ]] )
attachTo( "GEOMETRYCONNECTION", "Interface", "TARGET", "cover1", [[ 51.62157,
56.049588, 10.540165 ]] )
addMaterial( "Concrete", "CONCR", "TSCR", [] )
setParameter( "MATERIAL", "Concrete", "LINEAR/ELASTI/YOUNG", 34961.867 )
setParameter( "MATERIAL", "Concrete", "LINEAR/ELASTI/POISON", 0.15 )
setParameter( "MATERIAL", "Concrete", "MODTYP/TOTCRK", "ROTATE" )
setParameter( "MATERIAL", "Concrete", "TENSIL/TENCRV", "HORDYK" )
setParameter( "MATERIAL", "Concrete", "TENSIL/TENSTR", 3.2 )
setParameter( "MATERIAL", "Concrete", "TENSIL/TENSTR", 3.2 )
setParameter( "MATERIAL", "Concrete", "TENSIL/TENSTR", 3.2 )
setParameter( "MATERIAL", "Concrete", "TENSIL/GF1", 0.14366374 )
setParameter( "MATERIAL", "Concrete", "TENSIL/CBSPEC", "GOVIND" )
setParameter( "MATERIAL", "Concrete", "TENSIL/POISRE/POIRED", "DAMAGE" )
setParameter( "MATERIAL", "Concrete", "COMPRS/COMCRV", "PARABO" )
setParameter( "MATERIAL", "Concrete", "COMPRS/COMSTR", 35 )
setParameter( "MATERIAL", "Concrete", "COMPRS/COMSTR", 3535.9159 )
```

```
setParameter( "MATERIAL", "Concrete", "COMPRS/GC", 35.915936 )
setParameter( "MATERIAL", "Concrete", "COMPRS/COMSTR", 35 )
addGeometry( "Element geometry 2", "SOLID", "STRSOL", [] )
rename( "GEOMET", "Element geometry 2", "Cover" )
setParameter( "GEOMET", "Cover", "AXIAL", True )
setParameter( "GEOMET", "Cover", "AXIAL/CYLIN", [ 0, 79.999, 0.001, 1, 0, 0 ] )
setParameter( "GEOMET", "Cover", "AXIAL/CYLIN", [ 0, 79.999, 0.001, 1, 0, 0 ] )
setElementClassType( "SHAPE", [ "cover", "cover1" ], "STRSOL" )
# Remark start: For sampling shear stresses in the cover, this element geometry must
be used for the shapes cover and cover1
#addGeometry( "cover shear stress", "SOLID", "STRSOL", [] )
#setParameter( "GEOMET", "cover shear stress", "AXIAL", True )
#setParameter( "GEOMET", "cover shear stress", "AXIAL/CYLIN", [ 0, 65, 15, 1, 0, 0 ] )
# Remark end
assignMaterial( "Concrete", "SHAPE", [ "cover", "cover1" ] )
assignGeometry( "Cover", "SHAPE", [ "cover", "cover1" ] )
addMaterial( "reinforcement", "MCSTEL", "ISOTRO", [] )
setParameter( "MATERIAL", "reinforcement", "LINEAR/ELASTI/YOUNG", 200000 )
setParameter( "MATERIAL", "reinforcement", "LINEAR/ELASTI/YOUNG", 200000 )
setParameter( "MATERIAL", "reinforcement", "LINEAR/ELASTI/POISON", 0.3 )
addGeometry( "Element geometry 3", "SOLID", "STRSOL", [] )
rename( "GEOMET", "Element geometry 3", "reinforcement" )
setParameter( "GEOMET", "reinforcement", "AXIAL", True )
setParameter( "GEOMET", "reinforcement", "AXIAL/CYLIN", [ 0, 65, 15, 1, 0, 0 ] )
setElementClassType( "SHAPE", [ "reinforcement" ], "STRSOL" )
assignMaterial( "reinforcement", "SHAPE", [ "reinforcement" ] )
assignGeometry( "reinforcement", "SHAPE", [ "reinforcement" ] )
setElementSize( [ "cover" ], 4, -1, True )
clearMesherType( [ "cover" ] )
clearMidSideNodeLocation( [ "cover" ] )
setElementSize( [ "reinforcement", "cover1" ], 3, -1, True )
clearMesherType( [ "reinforcement", "cover1" ] )
clearMidSideNodeLocation( [ "reinforcement", "cover1" ] )
addSet( "GEOMETRYSUPPORTSET", "Boundary conditions" )
createSurfaceSupport( "Ux", "Boundary conditions" )
setParameter( "GEOMETRYSUPPORT", "Ux", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Ux", "TRANSL", [ 1, 0, 0 ] )
setParameter( "GEOMETRYSUPPORT", "Ux", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Ux", "reinforcement", [[ 105, 66.317017, 14.343753 ]] )
createSurfaceSupport( "Uy", "Boundary conditions" )
setParameter( "GEOMETRYSUPPORT", "Uy", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Uy", "TRANSL", [ 0, 1, 0 ] )
setParameter( "GEOMETRYSUPPORT", "Uy", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Uy", "cover", [[ 51.62157, 80, 60.810785 ]] )
attach( "GEOMETRYSUPPORT", "Uy", "cover1", [[ 51.62157, 80, 14.924945 ]] )
createSurfaceSupport( "Uz", "Boundary conditions" )
setParameter( "GEOMETRYSUPPORT", "Uz", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Uz", "TRANSL", [ 0, 0, 1 ] )
setParameter( "GEOMETRYSUPPORT", "Uz", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Uz", "cover", [[ 38.37843, 25.810785, 0 ]] )
attach( "GEOMETRYSUPPORT", "Uz", "cover1", [[ 51.62157, 65.075055, 0 ]] )
addSet( "GEOMETRYSUPPORTSET", "Deformation" )
createSurfaceSupport( "Deformation", "Deformation" )
setParameter( "GEOMETRYSUPPORT", "Deformation", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Deformation", "TRANSL", [ 1, 0, 0 ] )
setParameter( "GEOMETRYSUPPORT", "Deformation", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Deformation", "cover", [[ 0, 34.11416, 60.810785 ]] )
```

```
attach( "GEOMETRYSUPPORT", "Deformation", "reinforcement", [[ 0, 66.317017,
15.656247 ]] )
attach( "GEOMETRYSUPPORT", "Deformation", "cover1", [[ 0, 50.324542, 7.6874739 ]] )
addSet( "GEOMETRYLOADSET", "Tensile load" )
createSurfaceLoad( "Load", "Tensile load" )
setParameter( "GEOMETRYLOAD", "Load", "LODTYP", "DEFORM" )
setParameter( "GEOMETRYLOAD", "Load", "DEFORM/SUPP", "Deformation" )
setParameter( "GEOMETRYLOAD", "Load", "DEFORM/TR/VALUE", -0.1 )
attach( "GEOMETRYLOAD", "Load", "cover", [[ 0, 34.11416, 60.810785 ]] )
attach( "GEOMETRYLOAD", "Load", "reinforcement", [[ 0, 66.317017, 15.656247 ]] )
attach( "GEOMETRYLOAD", "Load", "cover1", [[ 0, 50.324542, 7.6874739 ]] )
addAnalysis( "Analysis1" )
addAnalysisCommand( "Analysis1", "NONLIN", "Structural nonlinear" )
remove( "GEOMETRYLOAD", "Load" )
createSurfaceLoad( "Load", "Tensile load" )
setParameter( "GEOMETRYLOAD", "Load", "LODTYP", "DEFORM" )
setParameter( "GEOMETRYLOAD", "Load", "DEFORM/SUPP", "Deformation" )
setParameter( "GEOMETRYLOAD", "Load", "DEFORM/TR/VALUE", -0.1 )
attach( "GEOMETRYLOAD", "Load", "cover", [[ 0, 34.11416, 60.810785 ]] )
attach( "GEOMETRYLOAD", "Load", "reinforcement", [[ 0, 66.317017, 15.656247 ]] )
attach( "GEOMETRYLOAD", "Load", "cover1", [[ 0, 50.324542, 7.6874739 ]] )
setAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "EXECUT(1)/LOAD",
False )
generateMesh( [] )
hideView( "GEOM" )
showView( "MESH" )
setAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "EXECUT(1)/LOAD", True
)
addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/LOADNR" )
setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/LOADNR", 1 )
setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/STEPS/EXPLIC/SIZES", "0.02(100)" )
setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/STEPS/EXPLIC/SIZES", "0.0200000(100)" )
setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/ITERAT/MAXITE", 30 )
setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/ITERAT/METHOD/NEWTON/TYPNAM", "MODIFI" )
addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/ITERAT/LINESE" )
setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/ITERAT/LINESE", True )
setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/ITERAT/CONTIN", True )
setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/ITERAT/CONVER/SIMULT", True )
setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/ITERAT/CONVER/SIMULT", False )
setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/ITERAT/CONVER/DISPLA", False )
addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/ITERAT/CONVER/ENERGY" )
setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/ITERAT/CONVER/ENERGY", True )
```

```
setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/ITERAT/CONVER/ENERGY/TOLCON", 0.001 )
setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/ITERAT/CONVER/ENERGY/NOCONV", "CONTIN" )
setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/ITERAT/CONVER/FORCE/NOCONV", "CONTIN" )
setAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "OUTPUT(1)/SELTYP",
"USER" )
addAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "OUTPUT(1)/USER" )
addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"OUTPUT(1)/USER/DISPLA(1)/TOTAL/TRANSL/LOCAL" )
addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"OUTPUT(1)/USER/DISPLA(2)/TOTAL/TRANSL/GLOBAL" )
addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"OUTPUT(1)/USER/DISPLA(3)/TOTAL/TRANSL/AXIAL" )
addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"OUTPUT(1)/USER/STRAIN(1)/TOTAL/GREEN/GLOBAL" )
addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"OUTPUT(1)/USER/STRAIN(2)/TOTAL/GREEN/PRINCI" )
addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"OUTPUT(1)/USER/STRAIN(3)/TOTAL/GREEN/AXIAL" )
addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"OUTPUT(1)/USER/STRAIN(4)/CRACK/GREEN" )
addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"OUTPUT(1)/USER/STRAIN(5)/CRKWDT/GREEN/PRINCI" )
addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"OUTPUT(1)/USER/STRESS(1)/TOTAL/CAUCHY/GLOBAL" )
addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"OUTPUT(1)/USER/STRESS(2)/TOTAL/CAUCHY/PRINCI" )
addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"OUTPUT(1)/USER/STRESS(3)/TOTAL/CAUCHY/AXIAL" )
addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"OUTPUT(1)/USER/STRESS(4)/TOTAL/TRACTI/LOCAL" )
addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"OUTPUT(1)/USER/STRESS(5)/CRACK/CAUCHY/LOCAL" )
addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"OUTPUT(1)/USER/FORCE(1)/REACTI/TRANSL/GLOBAL" )
addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"OUTPUT(1)/USER/ELMFOR(1)/TOTAL/TRANSL/GLOBAL" )
addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"OUTPUT(1)/USER/PARAME(1)/BANDWI" )
setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"OUTPUT(1)/USER/STRAIN(1)/TOTAL/GREEN/GLOBAL/LOCATI", "INTPNT" )
setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"OUTPUT(1)/USER/STRAIN(2)/TOTAL/GREEN/PRINCI/LOCATI", "INTPNT" )
setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"OUTPUT(1)/USER/STRAIN(3)/TOTAL/GREEN/AXIAL/LOCATI", "INTPNT" )
setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"OUTPUT(1)/USER/STRESS(1)/TOTAL/CAUCHY/GLOBAL/LOCATI", "INTPNT" )
setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"OUTPUT(1)/USER/STRESS(2)/TOTAL/CAUCHY/PRINCI/LOCATI", "INTPNT" )
setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"OUTPUT(1)/USER/STRESS(3)/TOTAL/CAUCHY/AXIAL/LOCATI", "INTPNT" )
setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"OUTPUT(1)/USER/STRESS(4)/TOTAL/TRACTI/LOCAL/LOCATI", "INTPNT" )
```

C.1 Zeta plot

```
def read(filename,h):
	innhold = []
	f = open(filename,"r") #Takes inn a textfile which read a single loadstep. Remove
all non-essensiell data first
	for line in f:
		line = line.strip()
		line = line.split(" ")  #Splits the list at whitespaces
		while("" in line) : # While-loop removes whitespaces
			line.remove("")
		innhold.append(line)
	f.close
	for i in range(len(innhold)):
		for j in range(len(innhold[i])):
			innhold[i][j]= float(innhold[i][j])
	for i in range(len(innhold)):
		if len(innhold[i])>7: #7 represents what number to add from columns
			innhold[i].pop(0) #Removes elementnumber from list

	dictionary = {}
	x = []
	y = []
	for i in range(len(innhold)): #iterates through length of list
		for j in range(len(innhold[i])):
			if innhold[i][4] not in dictionary: #checks if item is not an existing key
in dictionary
				dictionary[innhold[i][4]] = [] #adds the item as a key with an
attached empty list
	for i in range(len(innhold)): #Iterer through all elements
		dictionary[innhold[i][4]].append(innhold[i][h]) #Iterates through each
element row after row and add STSY to the corresponding integration pont

	for key in dictionary:
	    x.append(key)
	    tau_mean = sum(dictionary[key])/len(dictionary[key])
	    tau_max = max(dictionary[key],key=abs)
	    y.append(abs(tau_mean/tau_max))
	x,y = (list(t) for t in zip(*sorted(zip(x, y))))
	return x,y

x1,y1 = read("S1_LS43.txt",1)
x2,y2 = read("S2_LS46.txt",1)
x3,y3 = read("S3_LS14.txt",2)
x4,y4 = read("S4_LS31.txt",2)


import matplotlib.pyplot as plt

plt.plot(x1,y1,label = "φ20c60",linewidth = 1)
plt.plot(x2,y2,label = "φ20c52Q",linewidth = 1)
plt.plot(x3,y3,label = "4φ20c25Q",linewidth = 1)
plt.plot(x4,y4,label = "4φ20c55Q",linewidth = 1)
plt.plot([0],[0],label = " End of transfer")

size = 35
```

```
plt.scatter(x1[-1],y1[-1],s=size,marker="o")
plt.scatter(x2[-1],y2[-1],s=size,marker="o")
plt.scatter(x3[-1],y3[-1],s=size,marker="o")
plt.scatter(x4[-1],y4[-1],s=size,marker="o")
plt.scatter(20,0.1 ,s=size,c="k")

plt.xlabel("X = 0 mm: Symmetry section")
plt.ylabel("Bond stress distribution factor ζ")
#plt.title("Bond stress distribution factor ζ along perimeter of reinforcement bars")

plt.legend()
plt.legend(loc="lower center")

plt.show()
```

C.2 Polar bond stress plot

```
    #polar coordinates plotting

def read(filename):
        innhold = []
        f = open(filename,"r") #Takes inn textfile, reads one load step. Have to remove all non-essential text first
        for line in f:
                line = line.strip()
                line = line.split(" ") #Splits up list at whitespaces
                while("" in line) :  # While-loop removes whitespaces
                        line.remove("")
                innhold.append(line)
        f.close
        for i in range(len(innhold)):
                for j in range(len(innhold[i])):
                        innhold[i][j]= float(innhold[i][j])
        for i in range(len(innhold)):
                if len(innhold[i])>7: #7 if coordinates are on, 4 if not
                        innhold[i].pop(0) #Removes elementnumbers from list
        bond_stress_all = []
        for i in range(len(innhold)-1):
            if innhold[i][0]==3:
                bond_stress_all.append(abs(innhold[i][2]))
            if innhold[i][0]==6:
                bond_stress_all.append(abs(innhold[i][2]))
        return bond_stress_all

import matplotlib.pyplot as plt
import math

#bond stresses obtained from the Φ20c60,mirroring the results to the four quadrants:
bond_stress_quarter_1 =
[1.024E+01,8.282E+00,1.030E+01,8.276E+00,1.023E+01,8.302E+00,1.020E+01,8.300E+00,1.021E+01,8.302E+00,1.017E+01,8.295E+00,1.021E+01,8.266E+00,1.025E+01,8.266E+00,1.020E+01]
bond_stress_quarter_2 = bond_stress_quarter_1.copy()
bond_stress_quarter_2.reverse()
bond_stress_quarter_2.pop(0)
bond_stress_half_1 = bond_stress_quarter_1+bond_stress_quarter_2
bond_stress_half_2 = bond_stress_half_1.copy()
bond_stress_half_2.reverse()
bond_stress_half_2.pop(0)

bond_stress_circular = bond_stress_half_1 + bond_stress_half_2

for i in range(len(bond_stress_circular)-1):
        bond_stress_circular[i] = (bond_stress_circular[i]+bond_stress_circular[i+1])/2
bond_stress_circular[-1] = bond_stress_circular[0]

#bond stresses obtained from the Φ20c52Q, mirroring the results to the four quadrants:
bond_stress_q_1 =
[9.907E+00,7.918E+00,9.888E+00,7.888E+00,9.818E+00,7.797E+00,9.655E+00,7.723E+00,9.794E+00,7.648E+00,9.687E+00,7.618E+00,9.610E+00,7.573E+00,9.618E+00,7.538E+00,9.602E+00]
```

```
bond_stress_q_2 = bond_stress_q_1.copy()
bond_stress_q_2.reverse()
bond_stress_q_2.pop(0)
bond_stress_h_1 = bond_stress_q_1+bond_stress_q_2
bond_stress_h_2 = bond_stress_h_1.copy()
bond_stress_h_2.reverse()
bond_stress_h_2.pop(0)

bond_stress_square = bond_stress_h_1 + bond_stress_h_2

for i in range(len(bond_stress_square)-1):
       bond_stress_square[i] = (bond_stress_square[i]+bond_stress_square[i+1])/2
bond_stress_square[-1] = bond_stress_square[0]

#bond stresses for the 4Φ20c25Q
bond_stress_square_4phi_c25 = read("elements at L_half_spec3.txt")
bond_stress_square_4phi_c25.append(bond_stress_square_4phi_c25[0])

for i in range(len(bond_stress_square_4phi_c25)-1):
      bond_stress_square_4phi_c25[i] =
(bond_stress_square_4phi_c25[i]+bond_stress_square_4phi_c25[i+1])/2
bond_stress_square_4phi_c25[-1] = bond_stress_square_4phi_c25[0]

#bond stresses for the 4Φ20c55Q
bond_stress_square_4phi_c55 = read("elements at L_half_spec4.txt")
bond_stress_square_4phi_c55.append(bond_stress_square_4phi_c55[0])

for i in range(len(bond_stress_square_4phi_c55)-1):
      bond_stress_square_4phi_c55[i] =
(bond_stress_square_4phi_c55[i]+bond_stress_square_4phi_c55[i+1])/2
bond_stress_square_4phi_c55[-1] = bond_stress_square_4phi_c55[0]

#defining the reinforcement r=10mm
r = 10
pi = math.pi
x = []
for i in range(51):
    x.append((i/(51-1)*2*pi))
y = [r]*len(x)

#creating the circumferencial step to be graphed against for circular and square
step = math.pi*2/(len(bond_stress_circular)-1)
circumference = []
for i in range(len(bond_stress_square)):
    circumference.append(i*step)

#creating the circumferencial step to be graphed against for  the 4Φ25c25Q
step = math.pi*2/(len(bond_stress_square_4phi_c25)-1)
circumference_4phi_c25Q = []
for i in range(len(bond_stress_square_4phi_c25)):
    circumference_4phi_c25Q.append(i*step)

#creating the circumferencial step to be graphed against for  the 4Φ25c55Q
step = math.pi*2/(len(bond_stress_square_4phi_c55)-1)
circumference_4phi_c55Q = []
for i in range(len(bond_stress_square_4phi_c55)):
    circumference_4phi_c55Q.append(i*step)
```

```
#adjusting the bond stresses so it mathces with the reinforcement
adjust = r
for i in range(len(bond_stress_circular)):
    bond_stress_circular[i] += adjust
for i in range(len(bond_stress_square)):
    bond_stress_square[i] += adjust
for i in range(len(bond_stress_square_4phi_c25)):
    bond_stress_square_4phi_c25[i] += adjust
for i in range(len(bond_stress_square_4phi_c55)):
    bond_stress_square_4phi_c55[i] += adjust

fig1 = plt.subplot(111,projection = "polar")

#plotting the reinforcement surface
fig1.plot(x,y,lw = 1.5,linestyle="solid",color="k")

#plotting the bond stresses
fig1.plot(circumference,bond_stress_circular)
fig1.plot(circumference,bond_stress_square)
fig1.plot(circumference_4phi_c25Q,bond_stress_square_4phi_c25)
fig1.plot(circumference_4phi_c55Q,bond_stress_square_4phi_c55)

#adjustments to the plot
fig1.set_yticklabels(["","","5","10","15"])
fig1.set_xticklabels(["E","", "N","","W","","S"])
fig1.set_ylim([0,30])
plt.grid(linestyle = "dotted",lw=0.7)
fig1.set_rlabel_position(0)
fig1.text(-0.2,22,"[MPa]")
fig1.set_title("Bond stress distribution at L / 2", va='bottom')
#fig1.legend(["Reinforcement bar
surface","Φ20c60","Φ20c52Q","4Φ20c25Q","4Φ20c55Q"],loc = "best")
fig1.plot([0,0],[10,25],lw=1.0,linestyle="dotted",color="k")
fig1.plot([math.pi/2,math.pi/2],[10,25],lw=1.0,linestyle="dotted",color="k")
fig1.plot([math.pi,math.pi],[10,25],lw=1.0,linestyle="dotted",color="k")
fig1.plot([math.pi*3/2,math.pi*3/2],[10,25],lw=1.0,linestyle="dotted",color="k")

size = 35
lw = 0.1
plt.scatter(0,25,s=size,c="k",marker="|")
plt.scatter(0,20,s=size,c="k",marker="|")
plt.scatter(0,15,s=size,c="k",marker="|")
plt.scatter(math.pi/2,25,s=size,c="k",marker="_",linewidth = lw)
plt.scatter(math.pi/2,20,s=size,c="k",marker="_",linewidth = lw)
plt.scatter(math.pi/2,15,s=size,c="k",marker="_",linewidth = lw)
plt.scatter(math.pi,25,s=size,c="k",marker="|")
plt.scatter(math.pi,20,s=size,c="k",marker="|")
plt.scatter(math.pi,15,s=size,c="k",marker="|")
plt.scatter(math.pi*3/2,25,s=size,c="k",marker="_",linewidth = lw)
plt.scatter(math.pi*3/2,20,s=size,c="k",marker="_",linewidth = lw)
plt.scatter(math.pi*3/2,15,s=size,c="k",marker="_",linewidth = lw)
plt.grid(False)
plt.show()
```

C.3 Bond stress integration

```
def read(filename):
    innhold = []
    f = open(filename,"r") #Takes inn a textfile and reads one load step, remove all
non essesial data first
    for line in f:
        line = line.strip()
        line = line.split(" ") #Solits at whitespace
        while("" in line) :  # While-loop removes whitespaces
            line.remove("")
        innhold.append(line)
    f.close
    for i in range(len(innhold)):
        for j in range(len(innhold[i])):
            innhold[i][j]= float(innhold[i][j])
    for i in range(len(innhold)):
        if len(innhold[i])>4:
            innhold[i].pop(0) #Removes elementnumber from list
    return innhold

resultat = read("step_20_traction_forces_tabulated.txt")
dictionary = {}
for i in range(len(resultat)):  #iterates through length of list
    for j in range(len(resultat[i])):
        if resultat[i][0] not in dictionary:#checks if item is not an existing key in
dictionary
            dictionary[resultat[i][0]] = [] #adds the item as a key with an attached
empty list

for i in range(len(resultat)):     #Iterates thorugh all elements
    dictionary[resultat[i][0]].append(resultat[i][2])  #Iterates thorugh elements row by
row, and adds STST to corresponding integration points.

newton_cotes_weights = [0.028,0.111,0.028,0.111,0.444,0.111,0.028,0.111,0.028]
#defined weights for newton cotes integration scheme

Fc = 0
element_area = 6.03449*5.17633 #interface element area

for key in dictionary:
    for i in range(len(dictionary[key])):
        n_c_w_index = int(key)-1
        Fc += element_area*newton_cotes_weights[n_c_w_index]*dictionary[key][i]

fctm = 3.2
import math
areal = (70**2-10**2)*math.pi*0.25
Fcr = fctm*areal #cracking force of specimen phi20c60
```

Norwegian University of
Science and Technology