The pull-out and shear resistance of a lattice girder from a precast concrete floor plate during a 4-point bending test

A numerical study about the pull-out and shear resistance of lattice girders from precast concrete floor plates

Bouwsema, Willem H.

Award date:
2021

The pull-out and shear resistance of a lattice girder from a precast concrete floor plate during a 4-point bending test

A numerical study about the pull-out and shear resistance of lattice girders from precast concrete floor plates

Bouwsema, Willem H.

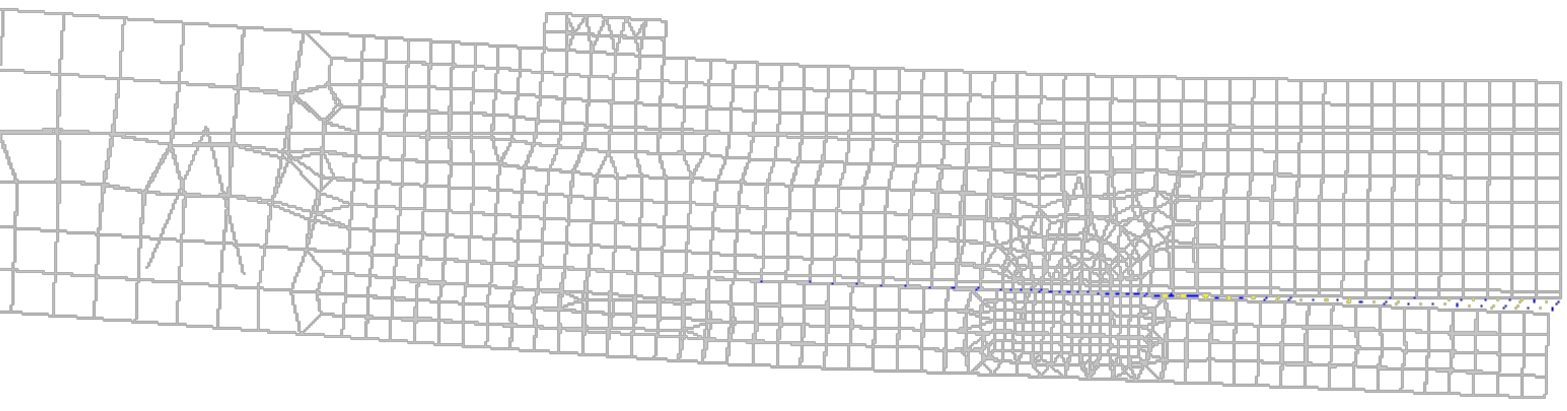
Award date:
2021

The pull-out and shear resistance of a lattice girder from a precast concrete floor plate during a 4-point bending test

A numerical study about the pull-out and shear resistance of lattice girders from precast concrete floor plates

Graduation research report

Report number:	Final Version
Name:	W.H. (Willem) Bouwsema
Student-ID no.:	1028762
Date:	15 January 2021
Supervisors:	Prof. ir. S.N.M. Wijte TU/e
	Dr. ir. R.J.M. Wolfs TU/e
	ir. C.M. Frissen DIANA FEA



ABSTRACT

The aim of this research is to get a better insight in the behavior of the lattice girders within a composite concrete floor during failure with the use of the Finite Element Software DIANA FEA. The initial cause of this research is the partial collapse of the parking garage at Eindhoven Airport in 2017. The so called critical detail, on which this research focuses, is located in the composite concrete floor at the longitudinal joint between two precast floor plates. Various studies have been conducted after the structural properties of this detail. The full size experiments, on which this research is based, was executed in the Structures Laboratory at Eindhoven University of Technology, see Wijte [1].

First of all, relevant codes and previous research has been studied. This was needed to get a better insight in the failure mechanism and in the DIANA modeling. This research resulted in a wide variety of influential parameters. Some of the needed parameters, like the concrete tensile strength, are clearly described. Other parameters, for instance the normal stiffness of the interface, have a large spread of values which yield good results. These values vary from 1.000 N/mm^3 to 60.000 N/mm^3.

The next step would be the modeling in DIANA. This started with smaller 2D models to validate certain behavior in DIANA. These models include the tensile behavior of concrete, the tensile and shear behavior of the concrete to concrete interface, the tensile behavior of the reinforcement and the bond-slip of the reinforcement. With these validated models, an attempt is made to combine them to create a full 2D model of the floor specimens described in Wijte [1].

Creating a functional and accurate model of these experiments, turned out to be harder than just combining the validated partial models. Many versions of these models have been calculated with the change of 1 parameter at a time. This eventually resulted in a model which was capable of describing the failure mechanism and loads by approximation. With these results, the choice was made to extend this 2D model to a 3D model.

To get a better insight in the behavior of this 3D model, the same partial models were created in 3D. With these models an attempt is made to create a representative 3D model of the anchorage of the lattice girder. The initial results of this model were promising. Another model was created to validate the lattice girder model. This model is of a single headed anchor, which was supposed to result in concrete cone failure. However, this model did not behave as expected. This model did not result in concrete cone failure and resulted in a substantial lower pull-out resistance than it theoretically should have. With these results the lattice girder model could not be validated. However, the lattice girder itself was still implemented in the final 3D model, since the lattice girder results were promising.

The final 3D model of the floor specimen was supposed to approximate the experimental results best of all. In the initial stages of the calculation, mainly before cracks are occurring, this model succeeded. Qualitatively, the model is capable of describing the actual failure mechanism found in the experiments very well. However quantitatively, the final model is not capable of describing the actual failure loads and deformations. The final model failed just after a load was reached of 40 kN, compared to the experiments which failed after reaching a load of 70 kN.

It could be concluded that the final 3D model is capable of describing the failure mechanism globally. However, more improvements are needed to also describe the actual quantitative results more accurately, especially for the pull-out and shear resistance of the lattice girder in the composite concrete floor.

TABLE OF CONTENTS

1 INTRODUCTION

1.1 Background and relevance

Since the partial collapse of the parking garage at Eindhoven Airport, in 2017 and the study after the cause, various studies have been conducted after the structural properties of the so called critical detail. The critical detail is located in the composite concrete floor at the longitudinal joint between two precast floor plates as is visualized in Figure 1.1. These studies vary from experimental tests to numerical analyses of composite concrete floors or elements of those floors. Many full scale experiments have been conducted to test the failure behavior of the composite concrete floors at the location of the critical detail. These experiments consist out of versions with weight reducing elements, which are the identical to the floors of the parking garage, and solid floor elements.

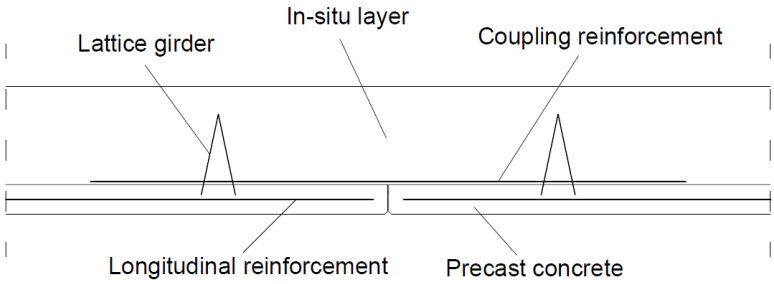


Figure 1.1: Visualization of the critical detail

These full-size experiments were conducted by a 4-point bending test. Many of the experiments show a similar kind of failure mechanism, mainly the delamination of the precast concrete floor plate from the in-situ concrete. However, there is a significant variation in the maximum allowable load over the different experiments. These variations will be further discussed in paragraph 3.6. Due to the large number of variables which could influence the critical load of the floor elements, it is preferred to have a numerical model in which the influence of these variables could be tested. With the large set of available experimental data, in which a limited number of variables is varied, the numerical models can be verified. These numerical models could then be used to describe the influence of said variables within the floor elements. These descriptions could then provide useful information for setting up new experiments.

The main concern is the influence of the several variables on the anchorage of the lattice girder in the precast concrete floor plate. However, to accurately describe the influence of the lattice girders, it is also necessary to research the influence of certain other variables, for example the tensile and shear behavior of the interface between the two concrete layers.

This research will make use of the 21 experiments of solid floor elements conducted by order of Betonhuis [1], at Eindhoven University of Technology. Initially new full-scale experiments were planned. These experiments were postponed due to the rise of COVID-19. Therefore, only existing experimental data from Betonhuis [1] is used to guide this graduation project.

1.2 Research goal

This graduation project aims to give a better insight in the behavior of the lattice girders within a composite concrete floor during failure. The final goal of this graduation project will be to describe the stresses, strains and crack formation surrounding the lattice girders within the concrete. This information will be obtained by creating a functional numerical modal which can accurately predict the influence of the pull out and shear resistance of the lattice girders within composite concrete floor. The research goal was formulated accordingly:

How can the influence of the pull-out and shear resistance of a lattice girder in a composite concrete floor, during failure, be determined with the use of a Finite Element Model?

The following sub goals are formulated to help achieve this main goal:

- How does the tensile behavior of concrete behave within a numerical model?
- How does the interface in between two concrete layers behave within a numerical model?
    - Interface in tension behavior.
    - Interface in shear behavior.
- How does the tensile strength of the reinforcement behave within a numerical model?
- How does the bond-slip of the reinforcement within concrete behave within a numerical model?

With the validation of all these partial models, it should be possible to combine them all within one large numerical model to recreate the failure behavior of a composite concrete floor.

1.3 Contents

First in this report, relevant literature and previous research regarding numerical models of composite concrete floors or parts of these floors is summarized. Secondly, the variables of the partial models will be described, with the validations of their corresponding numerical models. Lastly, a full scale 2D and 3D numerical model will be described and analyzed in comparison with the available experimental data. These results will also be discussed and recommendations for further research will be given.

2 CODES AND GUIDELINES

Many studies have been conducted researching all kind of aspects of concrete. These studies have resulted in the many different national and international concrete codes. For this study, the most useful codes are the Eurocode 2 [2], the Fédération Intarnationale du Béton Model Code (fib Model Code) [4] and the Guidelines for Nonlinear Finite Element Analysis of Concrete Structures [3]. An overview will be given about the relevant parts of these codes and guidelines in the following chapter.

2.1 Eurocode 2, EC2, 2016

The Dutch and European code on concrete, the NEN-EN 1992-1-1:2005+A1:2015+NB:2016 Beton – Algemeen [2], state a few equations that can be used to estimate the various values of concrete. These values can then be used in a Finite Element Model to simulate a realistic failure mechanism of a composite concrete floor. The following values and functions are according to the Eurocode for concrete classes lower than C50/60:

$$f_{cm} = f_{ck} + 8 \qquad \textit{Eq. 1}$$

$$f_{ctm} = 0{,}30\ {f_{ck}}^{(2/3)} \qquad \textit{Eq. 2}$$

$$E_{cm} = 22\ (f_{cm}/10)^{0{,}3} \qquad \textit{Eq. 3}$$

$$\varepsilon_{c1} = 0{,}7\ {f_{cm}}^{0{,}31} \leq 2{,}8 \qquad \textit{Eq. 4}$$

Where:

f_{ck} is the characteristic cylindrical compressive strength of concrete after 28 days in MPa

f_{cm} is the mean compressive strength of concrete in MPa

f_{ctm} is the mean tensile strength of concrete in MPa

E_{cm} is the secant-modulus of elasticity of concrete in GPa

ε_{c1} is the acting strain at maximum stress in ‰

2.2 Guidelines for Nonlinear Finite Element Analysis of Concrete Structures

The Guidelines for Nonlinear Finite Element Analysis of Concrete Structures [3] has been developed to be used for the nonlinear finite element analysis of basic reinforced and unreinforced concrete structures. This document streamlines the choices which have to be taken in order to create realistic results with a finite element model.

2.2.1 Concrete properties

In addition to the properties stated in paragraph 2.1, the fracture energy, Poisson ratio and the density of concrete will be stated.

$$G_F = 73\, f_{cm}^{0,18} \qquad \textit{Eq. 5}$$

ρ = 2500

v = 0,20

Where:

G_F is the fracture energy in Nmm/mm^2

ρ is the density for reinforced concrete in kg/m^3

v is the Poisson ratio

2.2.2 Constitutive models

The constitutive models, or material models, specify the stress-strain relationship that is assumed for the materials within the finite element model. These models often try to emulate the reality in a more simplified manner.

2.2.2.1 Tensile behavior of concrete

For the tensile behavior of concrete a total strain-based rotating crack model should be applied. For this model an exponential softening diagram, like the Hordijk softening, is preferred. To be able to calculate this curve, various parameters are needed. The parameters are the tensile strength of the concrete f_t (f_{ctm}), the fracture energy G_F, the equivalent length h_{eq}, as described in paragraph 2.2.2.3, and the ultimate strain parameter ϵ_u in case of tension combined with Hordijk softening, given by the following equation:

$$\varepsilon_u = 5{,}136\, \frac{G_F}{h_{eq}\, f_t} \qquad \textit{Eq. 6}$$

With these parameters and Equation 7, the Hordijk softening curve can be calculated as presented in Eq. 7.

$$\sigma = f_t \left(1 + \left(c_1 \frac{\varepsilon^{cr}}{\varepsilon_u}\right)^3 \exp\left(-c_2 \frac{\varepsilon^{cr}}{\varepsilon_u}\right) - \frac{\varepsilon^{cr}}{\varepsilon_u}(1 + c_1^3)\exp(-c_2)\right) \quad 0 \le \varepsilon^{cr} \le \varepsilon_u$$
$$\sigma = 0 \quad \varepsilon^{cr} \ge \varepsilon_u \qquad \textit{Eq. 7}$$

Where:

c_1 is usually 3,0

c_2 is usually 6,93

ϵ^{cr} is the actual occurring strain

It should be noted that Hordijk softening is described for a certain crack width. To be able to use this Hordijk softening in a FEM model with smeared cracking behavior, the crack width *w* is divided by the equivalent length h_{eq}, which results in a stress-strain relation as presented in Figure 2.1.

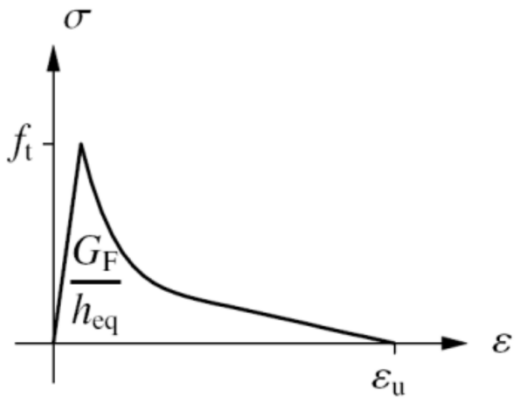


Figure 2.1: Hordijk softening [4]

2.2.2.2 Compression behavior of concrete

For the compressive behavior of concrete, it is recommended to use a parabolic stress-strain diagram with softening, this diagram is presented in Figure 2.2.

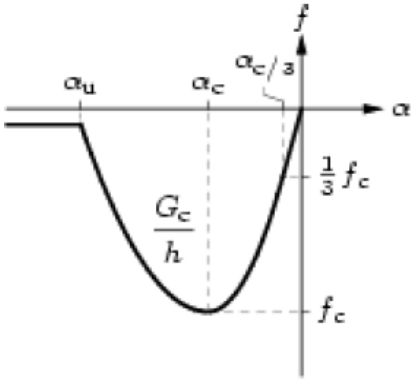


Figure 2.2: Parabolic compression diagram [4]

Where:

G_c is the compressive fracture energy

α is the compressive strain for progressive compression

f is the compressive strength

h is the equivalent length, in this case the crushing bandwidth

For a more extensive explanation of the parameters, see Guidelines for Nonlinear Finite Element Analysis of Concrete Structures [3] paragraph 2.4.1.4. This behavior might be simplified in the final models due to the complexity of the models. However, a simple elasto-plastic compressive stress-strain diagram should be avoided.

2.2.2.3 Equivalent length

The equivalent length (h_{eq}), also known as the crack-band width, is related to the dimensions of the finite element. This equivalent length is necessary to describe a softening stress-strain relationship (see Figure 2.1). The guidelines recommend an automatic procedure to determine the equivalent length, with a preferred method based on the initial direction of the crack and the element dimensions. In Figure 2.3, some examples are given to get a better understanding in what the equivalent length resembles.

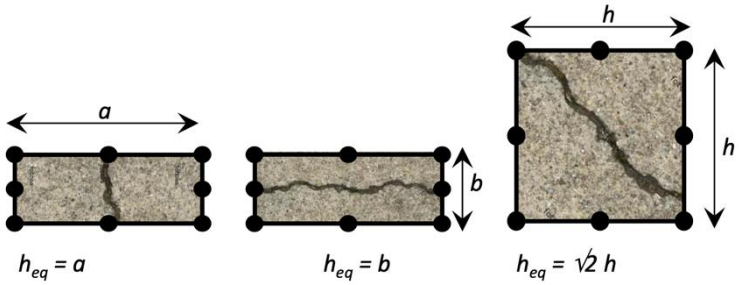


Figure 2.3: Examples of equivalent length based on element dimensions and crack direction [3]

2.2.2.4 Reinforcement Steel

For the reinforcement in the finite element models, an elasto-plastic material model with hardening should be used. For realistic results, all values should be used as mean values. This relationship is shown in Figure 2.4.

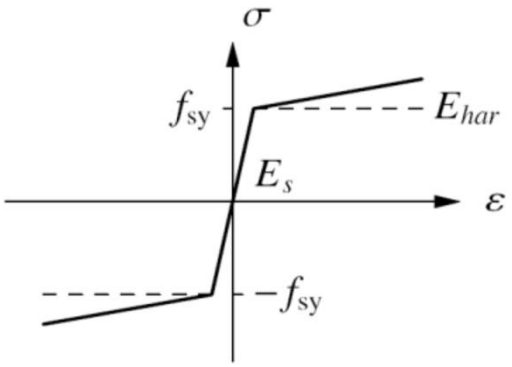


Figure 2.4: Stress-strain diagram for steel [3]

2.2.2.5 Finite Elements for Concrete

There are different preferred element types based on a 2D or a 3D model. In case of a 2D model, 8-node quadratic quadrilateral continuum elements should be used in order to get an accurate result. For 3D models, 20-node quadratic hexahedron continuum elements should be used. These elements are shown in Figure 2.5.

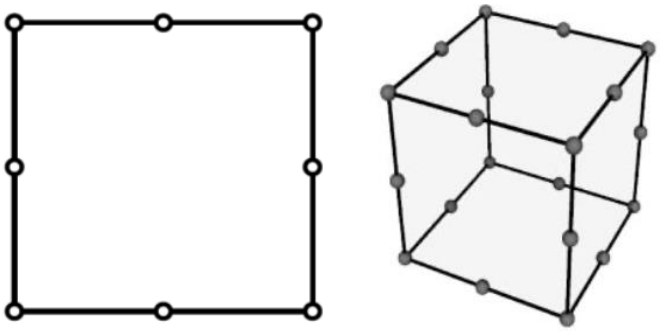


Figure 2.5: Quadratic quadrilateral continuum element (left) and quadratic Hexahedron continuum element (right) [3]

Linear elements are also possible, but this might result in unwanted behavior in some cases. The quadratic elements are capable of describing more deformation modes and more complex failure modes such as shear failure.

For the numerical integration, full integration schemes should be used. A reduced integration scheme for quadratic elements could result in various undesirable results, especially in case of extensive cracking behavior within the model. Therefore, the continuum elements should be integrated according to Figure 2.6.

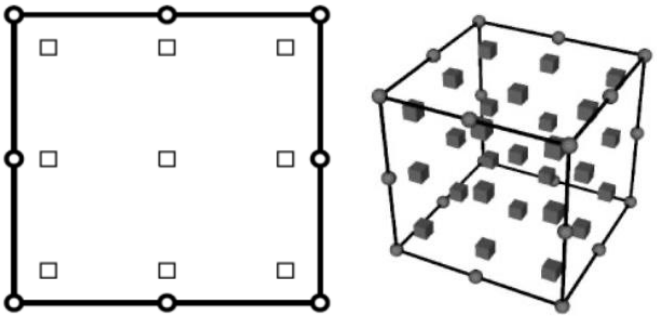


Figure 2.6: 3x3-point Gauss integration (left) and 3x3x3-point Gauss integration (right) [3]

2.2.2.6 Finite Elements for Reinforcement

For the reinforcement, it is preferred to use embedded reinforcement. However, these embedded elements only have an axial stiffness and only contribute to the "mother" concrete elements in which it is embedded. Furthermore, the slip between the concrete and reinforcement is usually ignored. Where bending and shear stiffness and/or bond-slip behavior is relevant, beam or truss elements in 2D or 3D, with the same integration schemes as the concrete, can be applied.

In the case of the reinforcement however, reduced integration could be applied. This reduced integration should not result in any spurious modes since the reinforcement is surrounded by concrete elements which will prevent these spurious modes.

Meshing

The mesh size should be chosen in a manner in which a relatively smooth stress field can be calculated. In case of a minimum mesh size, this will not result in any problems besides the increase of calculation time. However, for a maximum mesh size there are some guidelines. The maximum mesh size should be the minimum of the values presented in Table 2.1.

Structure	**Maximum element size**
2D	$\min(\frac{l}{50}, \frac{h}{6})$
3D	$\min\left(\frac{l}{50}, \frac{h}{6}, \frac{b}{6}\right)$

Table 2.1: Maximum mesh sizes [3]

2.2.2.7 Symmetry

For large models, it is advised to use symmetry axis if possible. By using symmetry, large models could be accurately simulated with a large reduction in model size therefore, a large reduction in calculation time.

2.3 Fédération Internationale du Béton, fib Model Code, 2010

The fib Model Code 2010 [4] serves as a basis for future codes for concrete structures. This code has specifically been used in this research to specify the various values for the modeled concrete and reinforcement including their respective interfaces. This model code has also been the main source for the Guidelines for Nonlinear Finite Element Analysis of Concrete Structures, which has been covered previously.

Most of the variables previously covered are also covered in this model code, however in most cases more extensively.

2.3.1 Concrete to Reinforcement Interface

The interface between concrete and reinforcement should be modeled in order to simulate a realistic shear behavior when the reinforcement is pulled out. This interface is usually modelled as a bond-slip relationship. This bond-slip relationship should be modelled differently for ribbed reinforcement bars and for plain (non-ribbed) surface reinforcement bars. This difference between these different surfaces will be depending on the parameters used to calculate the bond stress (τ_b). The bond stress is calculated through the following functions:

$$\tau_b = \tau_{bmax} (s/s_1)^{\alpha} \qquad for\ 0 \leq s \leq s_1 \qquad \textit{Eq. 8}$$

$$\tau_b = \tau_{bmax} \qquad for\ s_1 \leq s \leq s_2 \qquad \textit{Eq. 9}$$

$$\tau_b = \tau_{bmax} - (\tau_{bmax} - \tau_{bf})(s - s_2)/(s_3 - s_2) \qquad for\ s_2 \leq s \leq s_3 \qquad \textit{Eq. 10}$$

$$\tau_b = \tau_{bf} \qquad for\ s_3 \leq s \qquad \textit{Eq. 11}$$

Where:

τ_{bmax} is the maximum bond stress while ribs are still embedded in N/mm^2

s is the actual bond stress in N/mm^2

s_1 is the point where maximum bond stress is reached

s_2 is the point where bond stress starts declining

s_3 is the point where the ribs are fully pulled-out and the residual bond stress starts

α is the coefficient determining the bond characteristic, from constant stress (α = 0) to linear increasing bond stress (α = 1)

τ_{bf} is the residual bond stress (pure friction) in N/mm^2

Equation 8 through 11 are graphically presented in Figure 2.7 (Pull-Out).

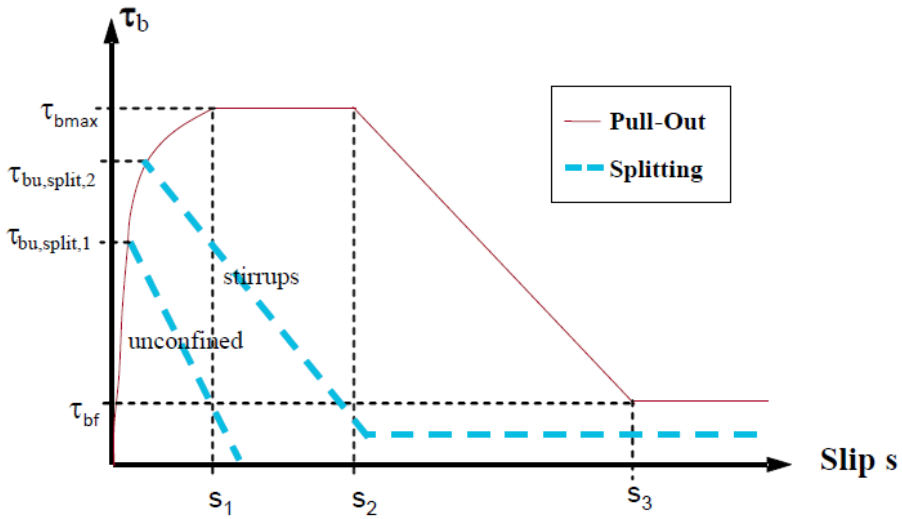


Figure 2.7: Analytical bond stress-slip relationship (only pull-out has been described) [4]

To calculate this bond-slip relationship, most of the parameters are predetermined. These parameters differ between the different bond conditions. In Table 2.2 the parameters are given for a good bond condition and all other bond conditions for ribbed reinforcement bars.

	Good bond conditions	All "other" bond conditions
τ_{bmax}	$2{,}5\sqrt{f_{cm}}$	$1{,}25\sqrt{f_{cm}}$
S_1	1,0 mm	1,8 mm
S_2	2,0 mm	3,6 mm
S_3	c_{clear}	c_{clear}
α	0,4	0,4
τ_{bf}	0,40 τ_{bmax}	τ_{bmax}

Table 2.2: Parameters for the mean bond stress-slip relationship of ribbed bars [4]

Where:

c_{clear} is the clear distance between individual ribs

To calculate the bond-slip of plain surface bars, the slip parameters are all set to the same value, since there are no ribs to create more resistance besides the friction between the plain surface bar and the concrete. For the same reason, there is no difference between the maximum bond stress and the residual bond stress. These parameters are presented in Table 2.3.

	Good bond conditions	All "other" bond conditions
$\tau_{bmax} = \tau_{bf}$	$0{,}1\sqrt{f_{cm}}$	$0{,}05\sqrt{f_{cm}}$
$S_1 = S_2 = S_3$	0,01 mm	0,01 mm
α	0,5	0,5

Table 2.3: Parameters for the mean bond stress-slip relationship of plain surface bars [4]

3 PREVIOUS RESEARCH

Besides these codes, there have been many more specific studies which can be related to this research. These studies vary from numerical to experimental research. In the upcoming chapter, an overview will be given about the relevant parts of these studies.

3.1 K. Lundgren, Chalmers University of Technology

3.1.1 Analyses of a lap splice in a lattice girder system, 2003, Numerical

The research described in K. Lundgren (2003), Analysis of a lap splice in a lattice girder system [5] focusses on a 2D finite element model, describing the reinforced joint, when subjected to bending. The geometry of the created 2D model is presented in Figure 3.1. This model was created with use of DIANA Finite Element Analysis 8.1.

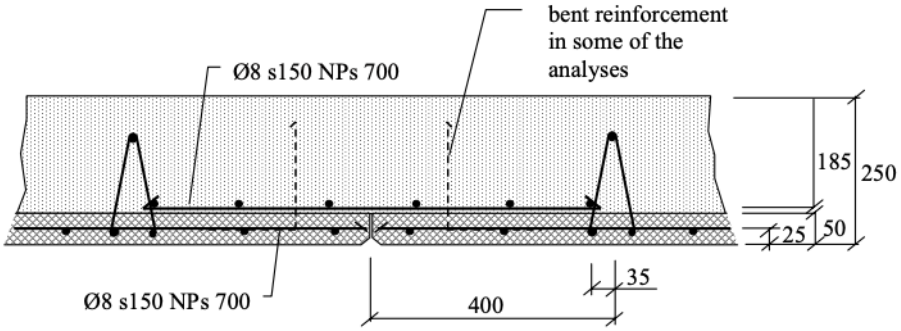


Figure 3.1: Modelled joint between two lattice girder elements [5]

To reduce the calculation time of this model, a symmetry line was applied in the middle to mirror the behavior of the in-situ concrete. Furthermore, the stresses in the main direction of the lattice girder are not included, since a 2D model was chosen and long-term effects such as creep and shrinkage are also not included. Creep and shrinkage could however be of influence for the eventual strength of the interface between the precast concrete and the in-situ concrete.

The concrete in this model was based on a total strain based rotating crack model which follows a smeared approach for the fracture energy. For the tension softening, the Horkdijk softening was chosen. For further specifications on the concrete used see K. Lundgren (2003). The steel reinforcement was modeled by the Von Mises yield criterion with an elastic modulus of 200.000 N/mm^2 and a yield strength of 700 N/mm^2. Slip between the reinforcement and the concrete is also present in the model, assuming unconfined (without any stirrups) concrete and “other” bond conditions, see paragraph 2.3.1.

Possibly most important for the current research is the interaction between the precast concrete and the in-situ concrete. This interface was modelled using a friction model, assuming a certain cohesion, see Figure 3.2. This figure shows the bond stress, which is the same as the shear stress.

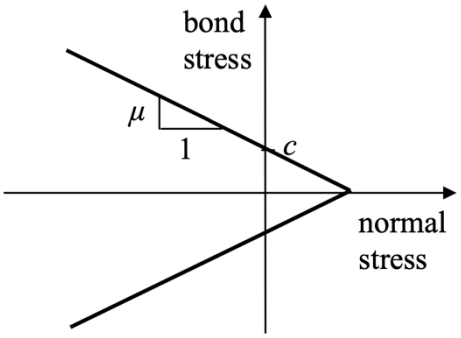


Figure 3.2: Coulomb friction criterion used for the interface elements between precast concrete and the in-situ concrete.

This friction model was used to allow the transfer of a certain shear stress, without the presence of any normal stress due to cohesion (*c*). However, the transferable shear stress will increase with the increase of the normal compressions stress. The parameters used to correctly model the interface are the cohesion (*c*), the friction coefficient (μ), the dilatation parameter (η), the normal stiffness modulus (*Kn*) and the shear stiffness modulus (*Kt*). The two most important parameters are the cohesion and the coefficient of friction. The following design equation was given according to fib (1998).

$$\tau = c + \mu\,(\sigma + \rho f_{yd}) \qquad \textit{Eq. 12}$$

Where:

τ is the shear strength in N/mm^2

c is the cohesion in N/mm^2

μ is the coefficient of friction

σ is the normal stress acting on the interface (in this equation positive when in compression) in N/mm^2

ρ is the amount of reinforcement crossing the interface

f_{yd} is the design yield strength of the reinforcement steel in N/mm^2

This equation was intended with reinforcement crossing the interface equally distributed. However, this is not the case in the research by Lundgren or this research, therefore the reinforcement will not be included in the equation. The cohesion and friction values used by Lundgren are presented in Table 3.1.

	Cohesion (*c*) [N/mm^2]	Friction (μ)
Fib (1998), design category 1	0,29	0,6
Fib (1998), design category 2	0,58	0,9
Nissen *et al.* (1986), experimental	1,69	1,54

Table 3.1: Values of the cohesion and the coefficient of friction for the interface [5]

No further information was found on the parameters of the dilatation, the normal stiffness modulus and the shear stiffness modulus. These parameters were chosen after modelling some of the tests of Nissen *et al.* (1986), resulting in a dilatation parameter η of 0,1, a normal stiffness modulus *Kn* of 1000 N/mm^3 and a shear stiffness modulus *Kt* of 100 N/mm^3.

3.1.2 Joints in lattice girder structures, 2005, Experimental and Numerical

The next research by Lundgren, K. Lundgren, J. Helgesson, R. Sylvén (2005), Joints in lattice girder structures [6], has mainly been used to verify and calibrate the parameters for the interface used in the previous research. Two detail experiments were carried out, one in case of a shear load on the concrete to concrete interface and one for tension on the interface. The study by Lundgren [6] focused only on the bond of the concrete interface without any reinforcement crossing this interface. Furthermore, the same principles were applied as in the previous study by Lundgren [5].

3.1.2.1 Shear tests

The specimens for the shear tests were modelled as presented in Figure 3.3. The surfaces of the concrete at the place of the interface were treated differently depending on the suppliers. One set of specimens had been brushed to roughen the area of the interface and the other set was treated with single grooves. For further details on the specific tests see K. Lundgren (2005).

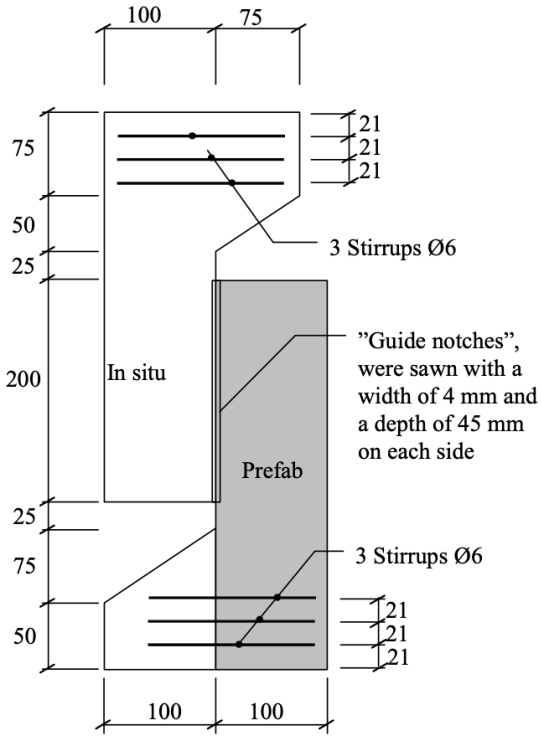


Figure 3.3: Geometry of shear test specimen [6]

The results of these experiments showed a large spread. The resulting shear strength of the brushed interfaces varied between 3,95 and 4,56 N/mm^2 and the results of the grooved interface varied between 0,51 and 1,60 N/mm^2.

3.1.2.2 Wedge split tests

The wedge split test was originally developed in purpose of measuring the fracture energy for homogenous concrete. In this experiment, the test was used to get a better understanding of the tensile behavior and the fracture energy of the interface. The specimens for the wedge split tests were modelled as presented in Figure 3.4 with an interface area of 15.000 mm^2. The interfaces of the specimens were treated the same as the shear test specimens. For further details on the specific tests see K. Lundgren (2005).

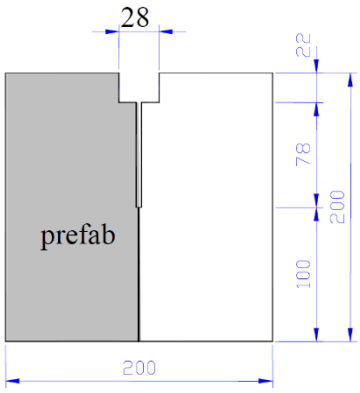


Figure 3.4: Geometry of shear test specimen [6]

After cracking of the interface, the load decreased very rapidly. Both kind of specimens showed the same kind of behavior, but the specimens with the brushed interfaces failed at a load twice as high as the specimens with the grooved interface.

$$F_{sp} = \frac{F_v}{2 \times \tan(\alpha)} \qquad \textit{Eq. 13}$$

Where:

F_{sp} is the splitting load over the interface in kN

F_v is the vertical load applied at the wedge in kN

α is the angle of the wedge (15°)

By dividing this splitting load by the area of the interface, the tensile strength of the interface can be calculated. For the brushed interfaces, this resulted in an average of 0,25 N/mm^2 and for the grooved interfaces in 0,12 N/mm^2.

3.1.2.3 Finite Element Models

All models were created in 2D, assuming a plane stress model. The concrete was modelled with 4-node quadrilateral isoparametric plane stress elements and the interface was modelled with interface elements, with separate nodes for the precast and the in-situ concrete. These models were then calibrated with the parameters according to the suppliers of the specimens and the experimental results. The most accurate results were obtained with a constant dilatation factor (η) of 0,5, Normal stiffness (Kn) of 3000 N/mm^3, Shear stiffness (Kt) of 350 and 400 N/mm^3 and a varying coefficient of friction (μ).

3.2 J.K. Weglarzy, University of Siegen

Weglarzy also researched prefabricated concrete slabs, specifically the permitted bar diameter, cross-sectional area of the reinforcement and the distance between the lattice girders, as described in the dissertation of J.K. Weglarzy [9]. The research by Weglarzy [9] consisted out of experiments and a numerical study, considering these experimental analyzed specimens.

The numerical research was conducted with the use of Abaqus FEA (2011). For the material model, a Concrete Damaged Plasticity Model (CDPM) was used. The behavior of concrete under pressure was described according to Eurocode 2 and the modulus of elasticity according to the fib Model Code (2010). However, an accurate interface was not included in these models. The interface in the created models was assumed as rigid. To be able to still evaluate the influence of the interface on the total load-bearing capacity of the model, a nearly frictionless interface without adhesion was assumed in the first row of elements. This assumption represents a pre-damage model. Without this pre-damage, the model would behave too stiff to accurately describe the experiments. For further details on this specific research, see Weglarzy [9].

3.3 M. Verbaten & K. Riemens, ABT

Verbaten and Riemens have created a non-linear finite element model based on the floor specimens tested at the University of Technology Eindhoven. These tests were part of the research conducted by Advies Bureau Hageman [11]. This model is described in *Niet-lineaire analyse breedplaatvloeren* [12]. The tested specimens were based on the floors of the parking garage at Eindhoven Airport.

The specimens had a total height of 450 mm, a length of 3,8 m and a width of 0,8 m. In these specimens, spherical weight reducing elements were applied. Over the width of the specimens, one whole sphere and two halves were present. These specimens were tested in a 4-point bending test as presented in Figure 3.5.

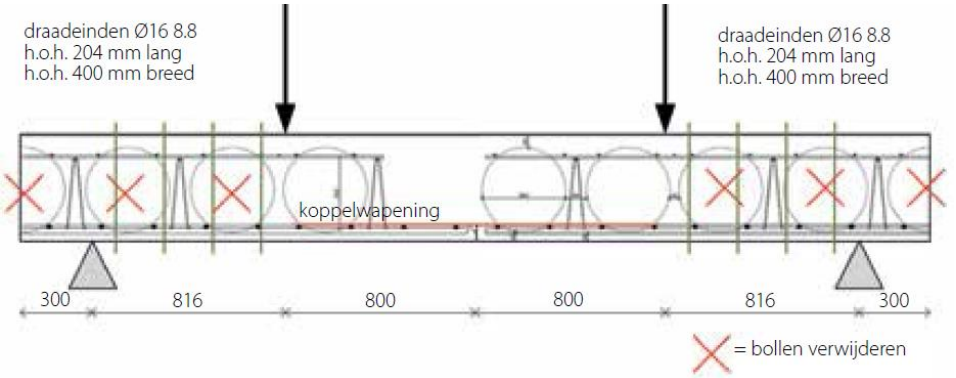


Figure 3.5: 4-point bending test set-up used for the specimens [11]

The finite element model of these specimens was created with the use of DIANA Finite Element Analysis (further on DIANA). To reduce the calculation time of this model, only a fourth of the width of the specimen has been modelled. This results in a model with only three halves of weight reducing elements. The measurements of this model are presented in Figure 3.6 and Figure 3.7.

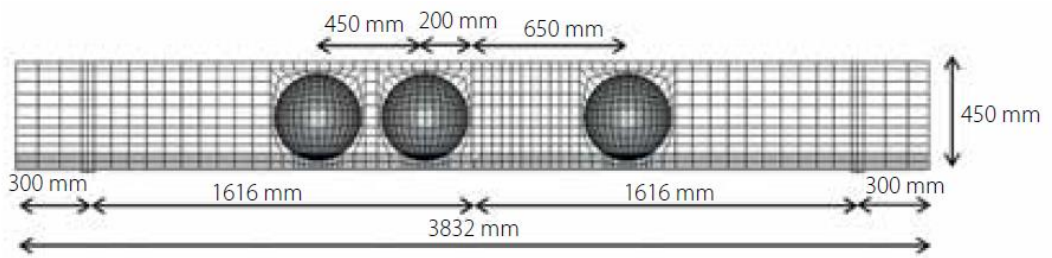


Figure 3.6: FEM model geometries longitudinal cross-section [11]

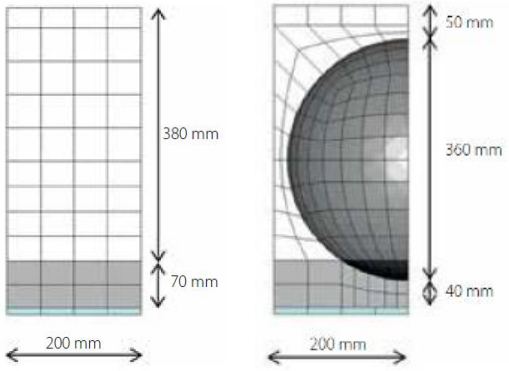


Figure 3.7: FEM model geometries cross-section [11]

3.3.1 Concrete Properties

To create a model with a concrete stress-stain relation in the most realistic way, the material properties need to match the reality as well. Usually calculations are executed with reduced material properties to be on the safe side. For these FEM calculations however, mean values for material properties are used. The mean values for the concrete are based on the compression tests and the concrete quality C45/55. This results in a f_{cm} = 31,6 N/mm^2 for the in-situ layer and f_{cm} = 53 N/mm^2 for the prefabricated concrete.

For the model, a total strain based rotating crack model was applied with smeared cracking and a Hordijk softening curve for the post cracking behavior in tension and a parabolic stress-strain relationship in compression. These principles are all according to RTD:1016-1:2017 [3].

3.3.2 Interface Properties

The interface in between the in-situ layer and the precast concrete was modelled in two different ways. At the ends of the model, a linear-elastic interface was applied with a high stiffness. In the middle of the model, at the place of the critical detail, a non-linear interface was applied.

For the non-linear interface, a Coulomb friction criterion was applied, in which an increasing normal stress will result in an increasing allowable shear stress. The applied parameters for the interface are presented in Table 3.2.

Cohesion (c)	0,5 N/mm^2
Friction coefficient (μ)	0,6
Friction angle (ϕ)	31°
Max f_t	0,5 N/mm^2
Kn	60.000 N/mm^3
Kt	6.000 N/mm^3

Table 3.2: Interface parameters [11]

3.3.3 Reinforcement

Two kinds of reinforcement models are implemented in the model. The first kind is the ‘embedded reinforcement’, these reinforcement bars only contribute to the stiffness of the surrounding concrete elements and will not come loose at any stress or deformation. This model is applied at the longitudinal reinforcement, since these bars will not reach their maximum allowable stress anyway.

The second kind of reinforcement model is ‘bond-slip reinforcement’. This model is applied on the lattice girders. This bond-slip model sets a nonlinear relation between the shear traction and the shear slip between reinforcement and concrete. The relations between the normal tractions and normal relative displacements are kept linear. In addition to this model, bending stiffness properties are also applied to the diagonals of the lattice girders to allow for dowel action, because the reinforcement bars will be modelled with beam elements. Furthermore, the reinforcement properties are the same as listed in chapter 4. For further details on the results of these models see *Niet-lineaire analyse breedplaatvloeren* [12].

3.4 L. Croes, University of Technology Eindhoven

Croes researched the behavior of unreinforced concrete to concrete interfaces under shear loading [13]. This research was conducted at the Structures Laboratory of the University of Technology in Eindhoven. The experiments consisted out of 48 concrete specimens which were loaded in varying levels of compression perpendicular to the interface, before loading in shear until failure.

The specimens were fabricated using different types of concrete and different surface treatments for the concrete to concrete interface. Some of these experiments can be related to some of the tested specimens described in Advies Bureau Hageman [11].The found average shear strength in the absence of compression perpendicular to the interface ranged from 0,57 N/mm^2 to 1,87 N/mm^2. The experiments were set up according to Figure 3.8.

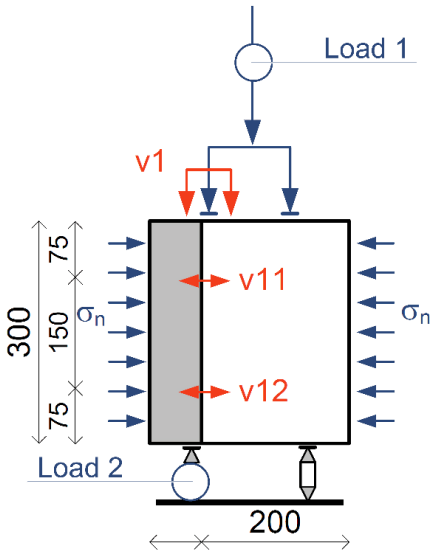


Figure 3.8: Direct shear test under compression [13]

Following the experiments, a numerical model was created to determine the stress distribution in the test-setup. This model was created as a linear elastic finite element model in Abaqus FEA. In this model, a tri-axial state of compression occurred near the load introduction. Peak shear stresses occurred which were slightly higher than the assumed interface average. However, the actual shear stress capacity may be larger. Furthermore, the difference in the modulus of elasticity between the two concretes further increases peak stresses, which may reduce the interface shear resistance. Concrete properties like creep and shrinkage may further influence the stress distribution, but this was not studied in this research.

3.5 M.J.M. Bliek, University of Technology Eindhoven

Bliek researched the pull out resistance of a lattice girder from a precast floor plate [14]. This was done by means of experimental research and numerical research.

3.5.1 Experimental research

The experiments conducted in this research were designed to test the pull out resistance of a lattice girder from a precast concrete floor plate. The specimens used for these experiments consisted out of a precast slab with a narrower in-situ layer on top. The lattice girder was located in the middle of the precast slab and the in-situ layer, running over the full length of the specimen. To get a clear measurement of just the anchorage of the lattice girder, a thin foam layer was applied in between the precast layer and the in-situ layer to prevent any adhesion between the two layers. The complete measurements are presented in Figure 3.9.

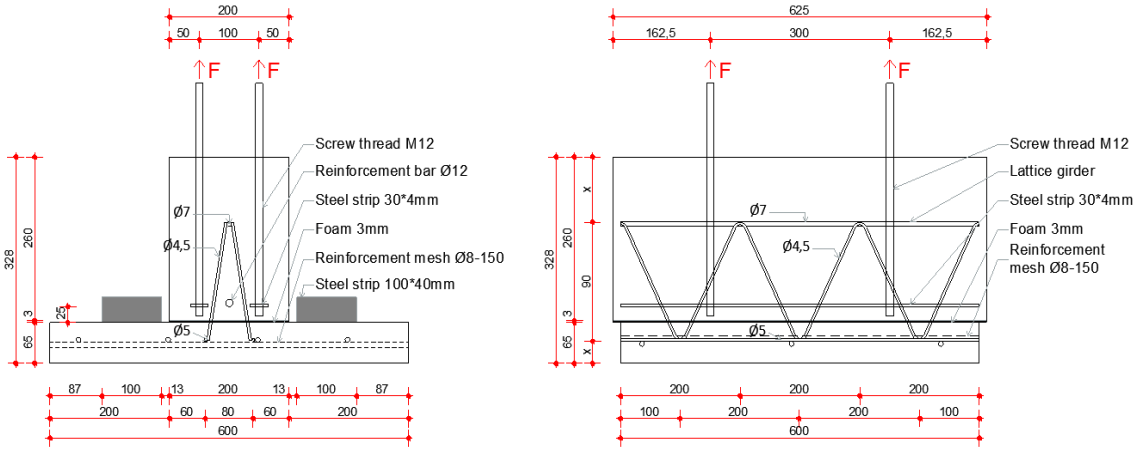


Figure 3.9: Cross-section of the specimen (left) and longitudinal section of the specimen (right) [14]

In these experiments, a deviation from the expected results was found. This was due to the bending of the precast plate while load was applied. This resulted in an early crack forming in the top layer of the precast plate, which in turn may have resulted in a lower maximum load of the anchorage of the lattice girder. For the complete results of these experiments, see the pull out resistance of a lattice girder from a precast floor plate [14].

3.5.2 Numerical research

To create the finite element model for this research, Abaqus FEA was used. This model was created with concrete smeared cracking. It turned out that this smeared cracking in Abaqus was more complicated than expected, which resulted in numerical instability problems.

To simplify the models, they were reduced to just 2D models. Starting with just a concrete square loaded in tension in one direction. Followed by a three-point bending test based on an experimental three-point bending test and finally a model in which a headed anchor is pulled out of a concrete floor, implementing the smeared cracking for the concrete. Due to reoccurring instability problems it was decided not to extent the model to 3D and not to remodel the anchors to a lattice girder system.

3.6 University of Technology Eindhoven

Following the partial collapse of the parking garage at Eindhoven airport, many experiments have been conducted at the Structures Laboratory of the University of Technology Eindhoven. Some of these experiments are used to verify the models created in this research, specifically the experiments described in Wijte [1]. The specimens for these experiments were all made from traditional concrete without any weight reducing elements. All of the specimens have the same total length of 3800 mm, width of 1000 mm and a precast floor plate with a thickness of 70 mm. The variation in the specimens has been made in the thickness of the in-situ layer, the coupling reinforcement, the concrete quality, the anchorage length of the reinforcement and the roughness of the interface. All the variations between the specimens are presented in Table 3.3 and a visualization of a specimen in set T01T03 is presented in Figure 3.10.

Specimen set	Coupling reinforcement	In-situ layer height	Concrete quality	Other
T01T03	Ø12-150	300 mm	C30/37	
T04T06	Ø16-150	300 mm	C30/37	
T10T12	Ø16-150	300 mm	C30/37	Reduced anchorage length
T13T15	Ø12-150	250 mm	C30/37	
T19T21	Ø12-150	250 mm	C20/25	
T22T24	Ø16-150	300 mm	C30/37	Lattice girders closer to the critical detail
R1R3	Ø16-150	300 mm	C30/37	Precast plate roughened

Table 3.3: Specifications floor specimen [1]

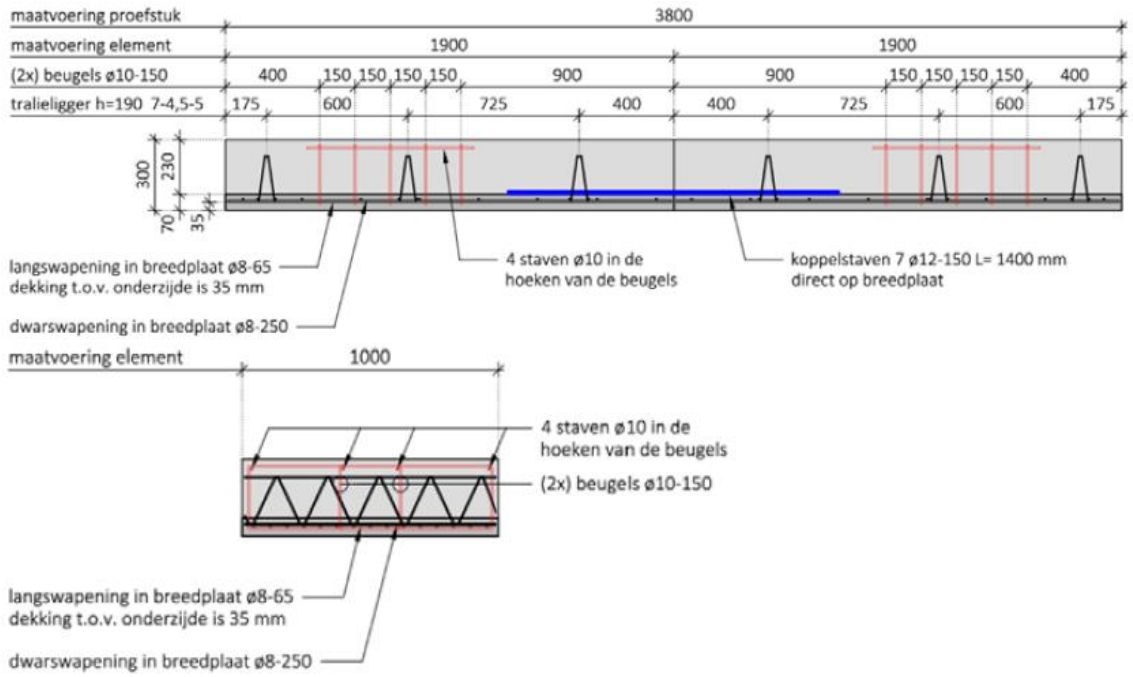


Figure 3.10: Cross-sections of specimen T01T03 [1]

All specimens were tested in the same manner, by a 4-point bending test. The set-up of this experiment can be seen in Figure 3.11, for specific information about all the measuring devices, see Wijte [1].

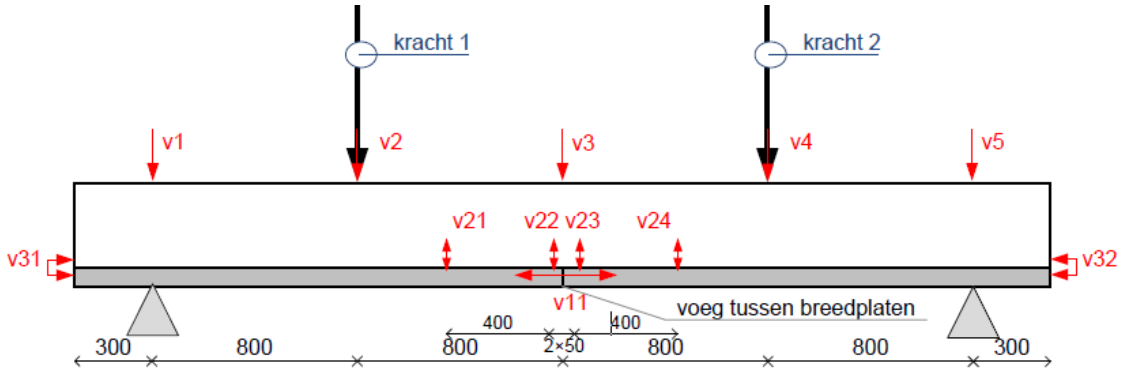


Figure 3.11: Test set-up of 4-point bending test [1]

For every type of specimen, three specimens have been tested. For example, for set T01T03 these specimens were T01, T02 and T03. All specimens were monitored for load-deformation, increase of horizontal width of the joint between the precast plates, measured at v11 in Figure 3.11 and the increase of the crack-width between the precast plate and the in-situ layer measured at v22 and v23 in Figure 3.11. This load-deformation, in vertical direction, was measured with the average load of the two applied loads (see v2 and v4 in Figure 3.11) in combination with the maximum deflection in the middle of the specimen (v3 –(v1+v5)/2 in Figure 3.11) . The load-deformation graph of T01T03 is presented in Figure 3.12, the graph of the horizontal joint is presented in Figure 3.13 and the graph of the vertical joint is presented in Figure 3.14.

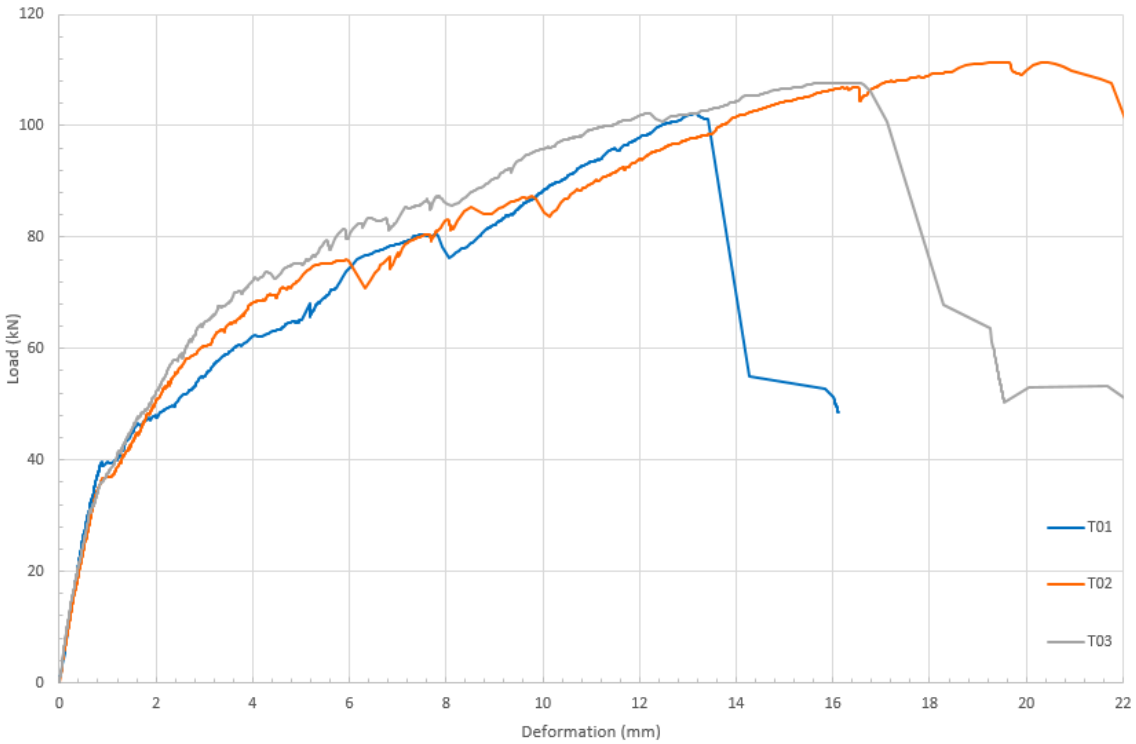


Figure 3.12: Load - Deformation graph T01T03 [1]

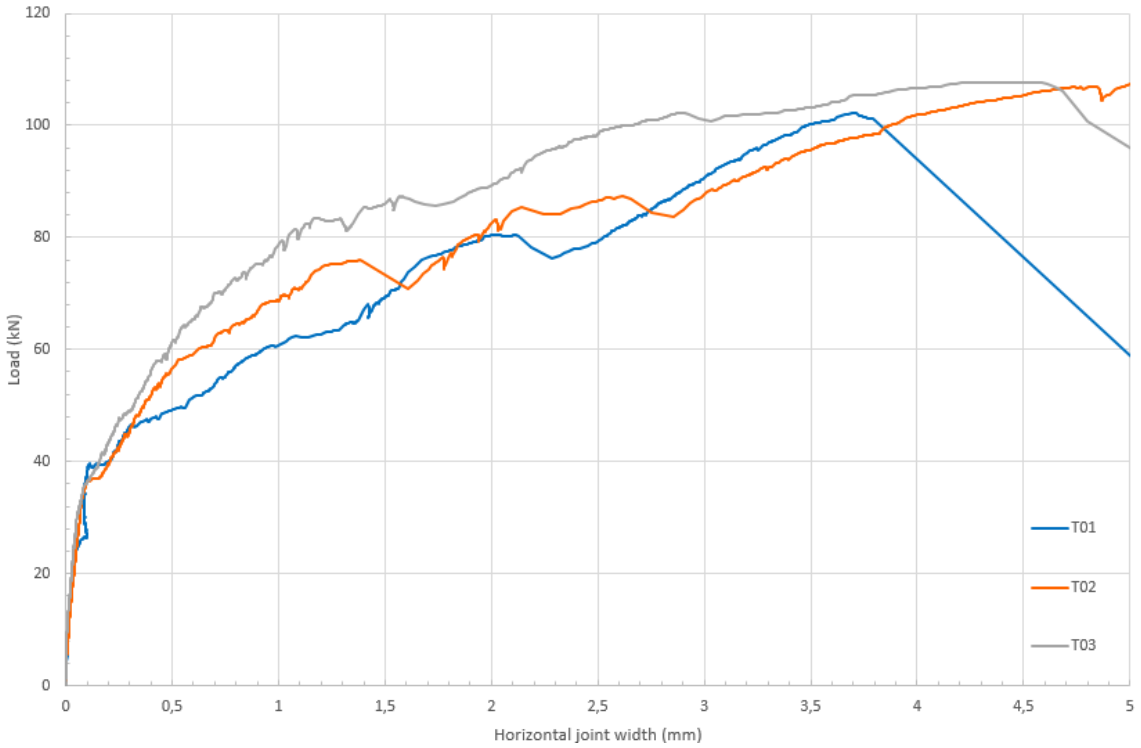


Figure 3.13: Load - Horizontal joint graph T01T03

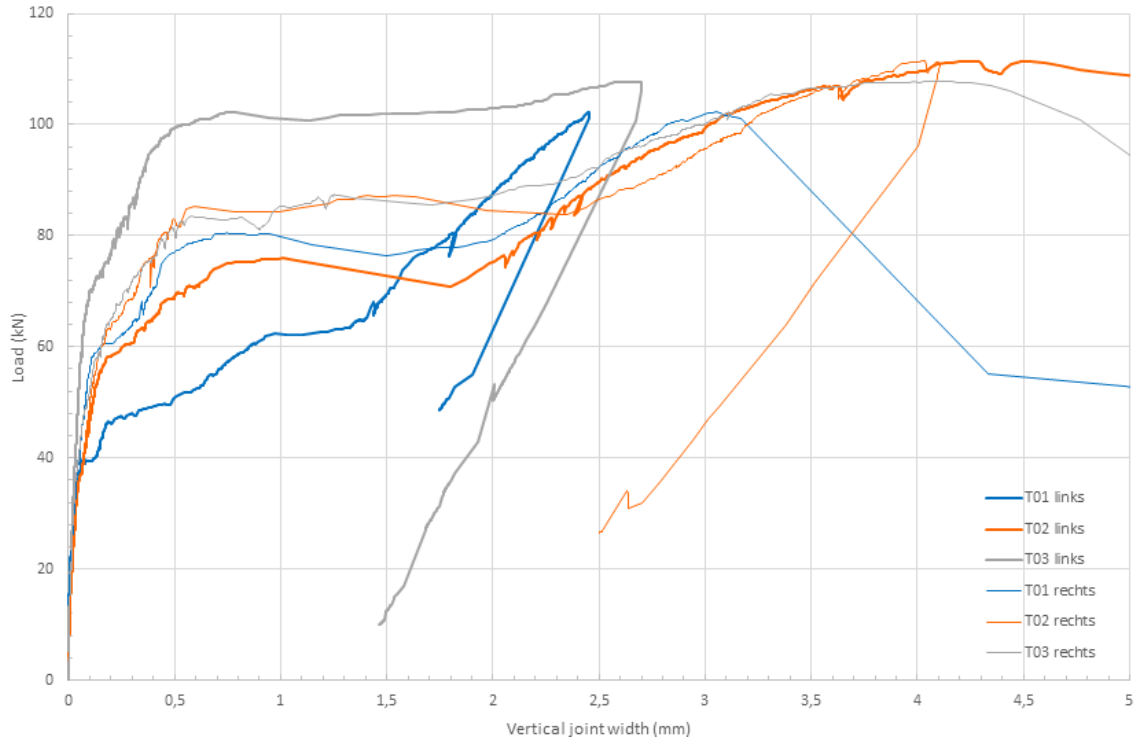


Figure 3.14: Load - Vertical joint graph T01T03

All three specimens showed a similar failure mechanism with a maximum difference of only 9 kN at the maximum load and a maximum difference in deflection of 4 mm.

Besides these specimens an additional material research has been conducted. At the date of the full experiment, also three concrete cubes have been tested. These cubes were casted on the same date as the full specimens. Three were casted per precast layer and three were casted per in-situ layer. The mean results of these cubes are presented in Table 3.4.

Cube specimen	$f_{cm,cube}$
Precast T01	62,2
Precast T02	61,1
Precast T03	61,0
In-situ T01	38,3
In-situ T02	36,7
In-situ T03	36,9

Table 3.4: Mean cube compression strength T01T03 [1]

To be able to model these full specimens later in DIANA, the $f_{cm,cube}$ has to be converted to f_{cm}. Therefore, the following functions have been described by Wijte [10]:

$$f_{cm} = 3 + 0{,}851 \times f_{cm,cube} \qquad \textit{Eq. 14}$$

$$f_{ck} = f_{cm} - 8 \qquad \textit{Eq. 15}$$

For a more elaborate explanations on the experiments, see Wijte (2018) [1].

4 NUMERICAL RESEARCH

To get a better understanding in how the different aspects of the concrete and the reinforcement will behave in a finite element model, all the parts will be modeled individually. These individual models will all be created with the use of the finite element software DIANA Finite Element Analysis 10.4. All finite element models will be scripted in python to make sure that every simulation, with the same script, will always result in the exact same model, despite the individual settings of the user. All the models will be based on the same kind of concrete and reinforcement. The parameters for these materials are given in Table 4.1 and Table 4.2. Furthermore, all models within this chapter will be 2D models and deformation controlled. The scripts to reproduce all the models will be included in the Appendixes.

Parameter		**Value**	**Unit**
Characteristic compression strength	f_{ck}	30	N/mm^2
Mean compression strength	f_{cm}	38	N/mm^2
Mean tensile strength	f_{ctm}	2,9	N/mm^2
Modulus of elasticity	E_{cm}	32000	N/mm^2
Ultimate strain	ϵ_u	2,3	‰
Fracture energy	G_F	0,1405	
Poisson ratio	v	0,2	
Density	ρ	2500	kg/m^3

Table 4.1: Material parameters for concrete

Parameter		**Value**	**Unit**
Mean yielding strength	f_{ym}	550	N/mm^2
Mean ultimate strength	f_{um}	594	N/mm^2
Modulus of Elasticity	E_s	200000	N/mm^2
Yielding strain	ϵ_{ym}	2,75	‰
Ultimate strain	ϵ_{um}	50	‰
Poisson ratio	v	0,3	
Density	ρ	7800	kg/m^3

Table 4.2: Material parameters for reinforcement

4.1 Tensile behavior of Concrete

The tensile behavior of concrete is one of the most important behaviors for the final model. The final model will be a full model of a 4-point bending test of a composite concrete floor. This applied bending will result in tensile stresses at the bottom of the floor. Since concrete will crack with relatively low tensile stresses, it is very important to be able to describe this behavior accurately. The tensile behavior of concrete will be described by the Hordijk softening curve as described in paragraph 2.2.2.1. DIANA allows for the option to implement this behavior automatically, however the parameters used for the Hordijk softening need to be filled in manually.

To be able to verify how the cracks will be spread through the concrete, two models have been tested. The first model consists of a single element with a size of 100 x 100 mm. This will guarantee that the full stain will be applied on this singe element. The second model consists of the exact same elements, but in this case in a grid of 10 x 10 elements. These models are presented in Figure 4.1: Concrete tension models, 1 element of 100x100mm2 (left), 100 elements of 100x100mm2 (right).

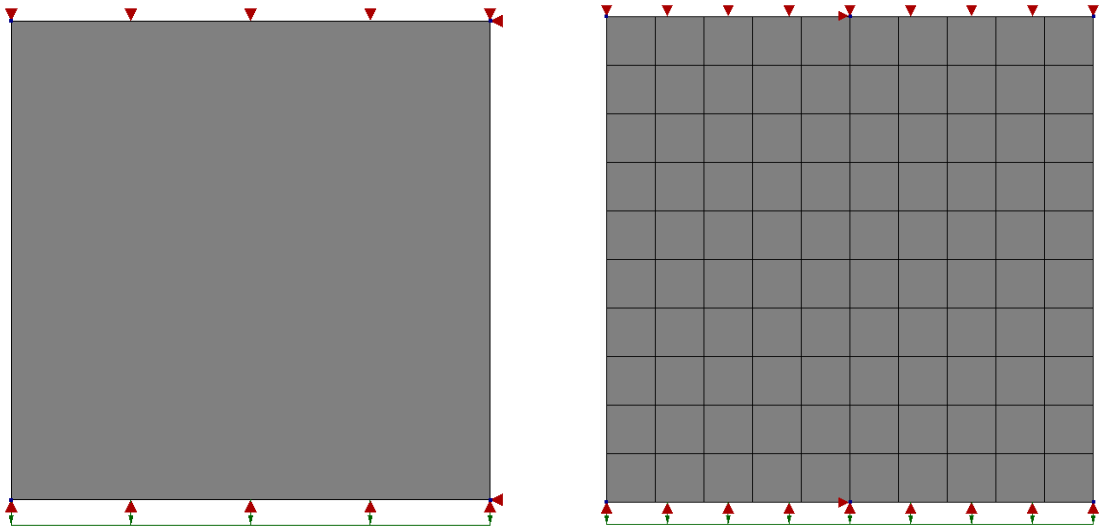


Figure 4.1: Concrete tension models, 1 element of 100x100mm^2 (left), 100 elements of 100x100mm^2 (right)

By using the exact same elements, there will be no difference in variables between the models. The only variable which had to be considered, was the ultimate strain, which is dependent on the equivalent length, see paragraph 2.2.2.1. Since the elements have the exact same size, the equivalent length will also be the same, thus also the ultimate strain.

To be able to reach that ultimate strain, a deformation of -0,24 mm will be applied on the bottom of the single element model and a deformation of -2,4 mm will be applied on the 100-element model. These deformations would force the models to reach the ultimate tensile stress, at which point a crack occurs and the concrete starts to soften (decrease stress) until the ultimate strain is reached. At the ultimate strain, the stress is equal to zero. Since these models consist out of a homogeneous material, without any imperfections, there should be no specific location where a crack could initiate. This results in the same stress-strain diagram over all the integration points present in the models. The results of these models and the Hordijk equation as described in 2.2.2.1 are presented in Figure 4.2.

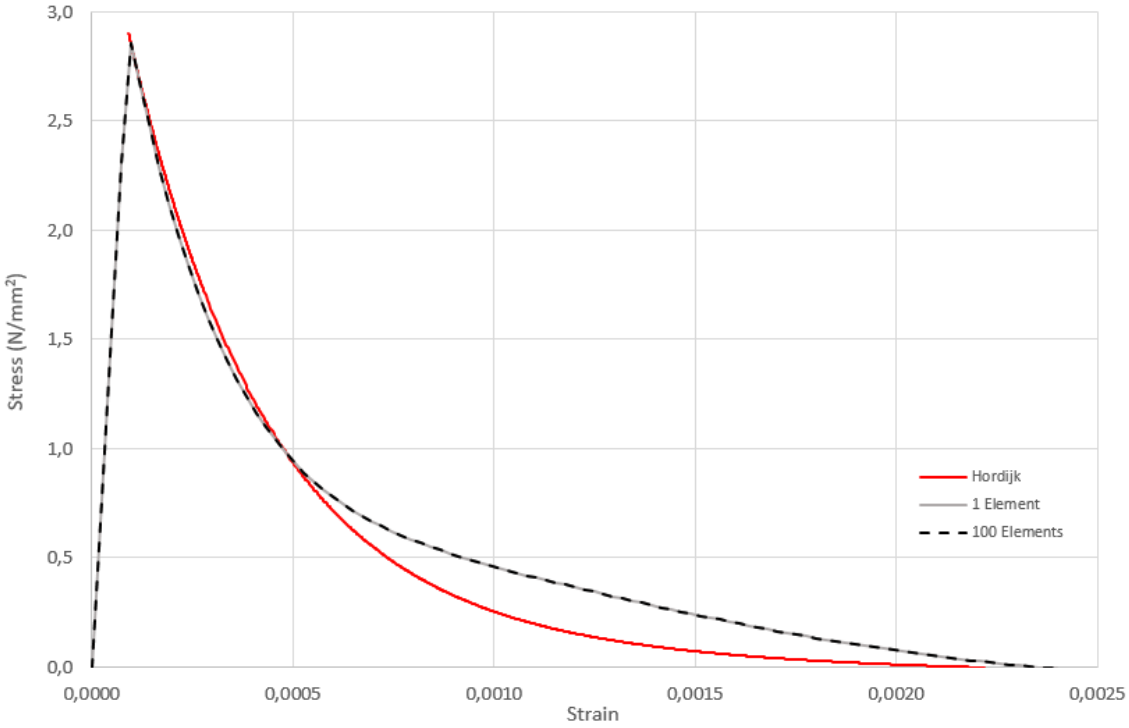


Figure 4.2: Hordijk stress-strain relation, analytic and based on DIANA models

As can be concluded from Figure 4.2, the models function almost exactly as they should according to the Hordijk equation. The maximum reached tensile stress in the models does not exceed the 2,9 N/mm^2 and the ultimate strain is indeed reached at 2,3‰. The scripts for these models are included in Appendix A1.1 and A1.2.

4.2 Concrete to Concrete Interface

The concrete to concrete interface will be tested in two different models. The first model will test the tensile behavior of the interface while the second model will focus on the shear capacity of the interface. Both models use the same geometry, only the supports and deformations are applied on different places in the models.

The models consist of a precast floor plate with a height of 70 mm and an in-situ layer with a height of 230 mm. The models have a total width of 400 mm and consist of 2D plane stress elements with a thickness of 100mm. This thickness will be used to distribute the stresses through the material. Without this thickness, no material properties could be assigned since there would be no volume to assign these properties to. However, both models are still only 2D.

4.2.1 Concrete Interface in Tension

The interface is based on the Coulomb friction criterion. For this test model, the used parameters are based on previous research as described in Chapter 3. The parameters of influence for the interface are presented in Table 4.3. For this specific tension test, only the maximum allowable tension stress is relatable to the results. Since DIANA will not be able to make these calculations with any incomplete properties, all the properties still have to be entered.

Cohesion (c)	0,5 N/mm^2
Dilatancy angle	31°
Friction angle (ϕ)	31°
Max $f_{t,I}$	0,5 N/mm^2
Kn	60.000 N/mm^3
Kt	6.000 N/mm^3
Interface opening model	Gapping model
Gap appearance	Brittle

Table 4.3: Interface parameters used in DIANA

The model created for this specific test is presented in Figure 4.3. This model is vertically restrained at the top and at the bottom edge. Only the top-middle and bottom-middle nodes are horizontally supported. The deformation is applied at the bottom edge of the model. The critical stress of the interface should be reached at a critical strain of 1,56 x 10^{-5}. This critical strain is calculated according to Equation 16. To reach this critical strain, a total deformation 4,69 x 10^{-3} mm should be applied, this deformation is calculated according to Equation 17.

$$\epsilon_{crit,I} = f_{t,I} / E_{cm} \qquad \textit{Eq. 16}$$

$$def_{crit,I} = h \times \epsilon_{crit,I} \qquad \textit{Eq. 17}$$

Where:

h is the total height of the model in mm

To reach this critical deformation, a total deformation of 5 x 10^{-3} mm is applied in 200 steps of 2,5 10^{-5} mm. The results of this test are presented in Figure 4.4. This figure shows the total deformations in vertical direction at the measuring points. The measuring points are taken in the middle of the interface (Interface Bot is the precast concrete and Interface Top is the in-situ layer) and at the bottom-middle node (Check Bot). As the figure shows, the concrete relaxes after the critical stress is reached, this is shown by the dotted line returning to 0 and the grey line to the critical deformation of 4,69 x 10^{-3} mm.

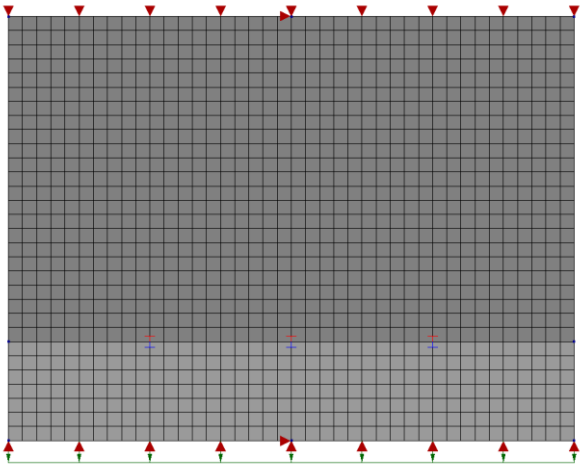


Figure 4.3: Concrete to concrete tension interface model

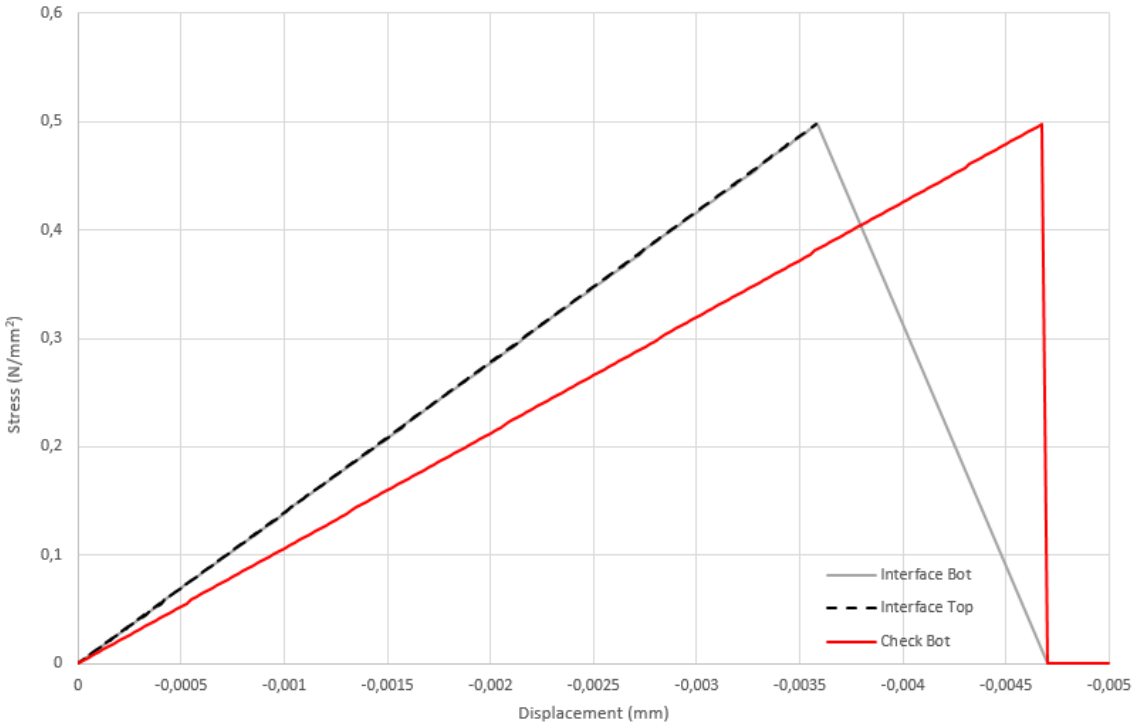


Figure 4.4: Stress-displacement relation interface in tension, based on DIANA model

It can be concluded that the interface model behaves accordingly. The maximum normal stress of the interface of 0,5 N/mm^2 is not exceeded and the displacement of the Interface Top does not exceed 3,59 mm, which is the deformation at which the critical strain would be reached (1,56 x 10^{-5} x 230 = 3,59). The script for this model is included in Appendix A2.1.

4.2.2 Concrete Interface in Shear

For the shear test of the interface, almost the same model as presented in Figure 4.3 is used. Only the supports and loads are different in this case. To be able to isolate the shear tension to the interface, the supports are changed to bodysupports. Before the horizontal load is applied at the precast plate, a vertical prestress of 10 N/mm^2 is first applied. Equation 12 has been used to calculate the maximum allowable shear stress. This results in a τ_{max} of 6,51 N/mm^2. Furthermore, the shear deformation needed to reach τ_{max} can be calculated according to Equation 18. This results in a deformation of 1,085 x 10^{-3} mm, which is presented in Figure 4.5.

$$def_{crit,\tau} = \frac{\tau_{max}}{Kt} \qquad \textit{Eq. 18}$$

Where:

$def_{crit,\tau}$ is the shear deformation needed to reach τ_{max} in mm

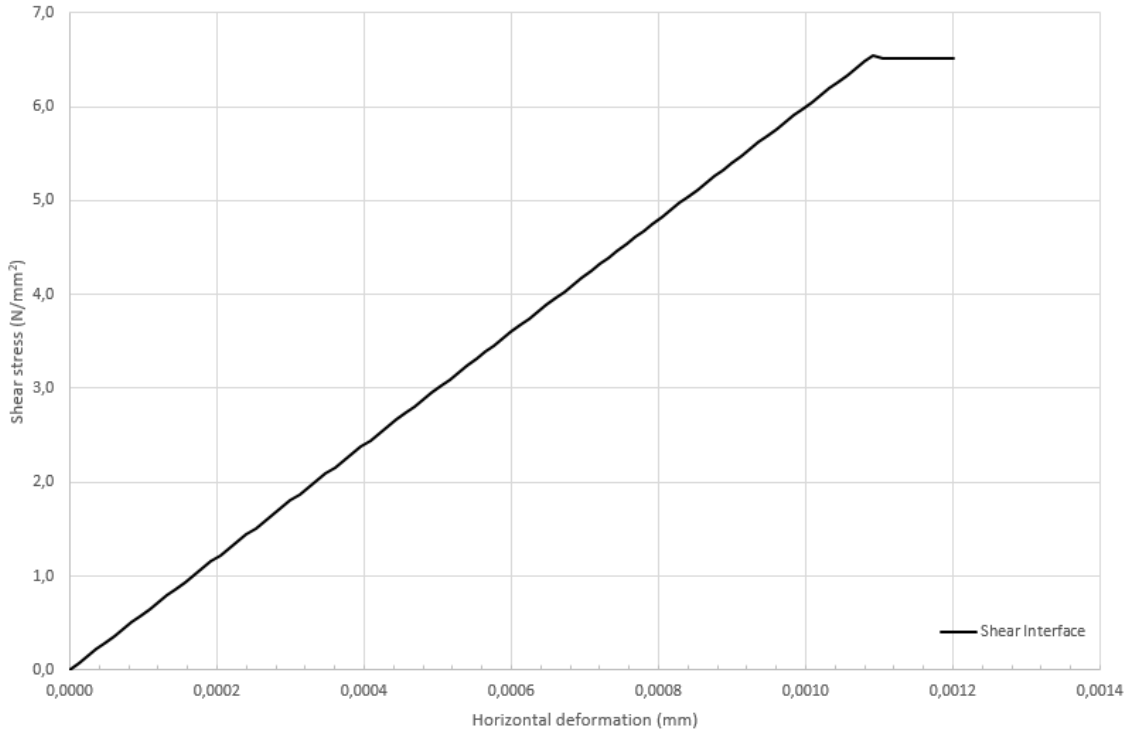


Figure 4.5: Stress-displacement relation interface in shear, based on DIANA model

This result is reached in a total of 110 steps of which the first 10 are used to apply the prestress on the model and the next 100 are used to reach the $def_{crit,\tau}$.

It can be concluded that the interface in shear behaves accordingly. The maximum shear stress is reached without exceeding it, after which a constant shear traction is maintained, which is represented by the horizontal part of the line in Figure 4.5. The script for this model is included in Appendix A2.2.

4.3 Tensile behavior of the Reinforcement

For the tensile behavior of the reinforcement, as described in paragraph 2.2.2.4, an elasto-plastic model with strain hardening is used. This model could be validated with any model in which the reinforcement would be deformed enough to reach the ultimate strain.

With the use of the parameters in Table 4.2, the reinforcement should show a linear elastic behavior until a stress of 550 N/mm^2 is reached. This stress should be reached at a strain of 2,75‰. Beyond this point the reinforcement should start its plastic deformation, this should reach an ultimate stress of 594 N/mm^2 at a strain of 5% after which it should fail. This analytical behavior is presented next to the DIANA results in Figure 4.6. Both lines follow a an identical track as should be expected.

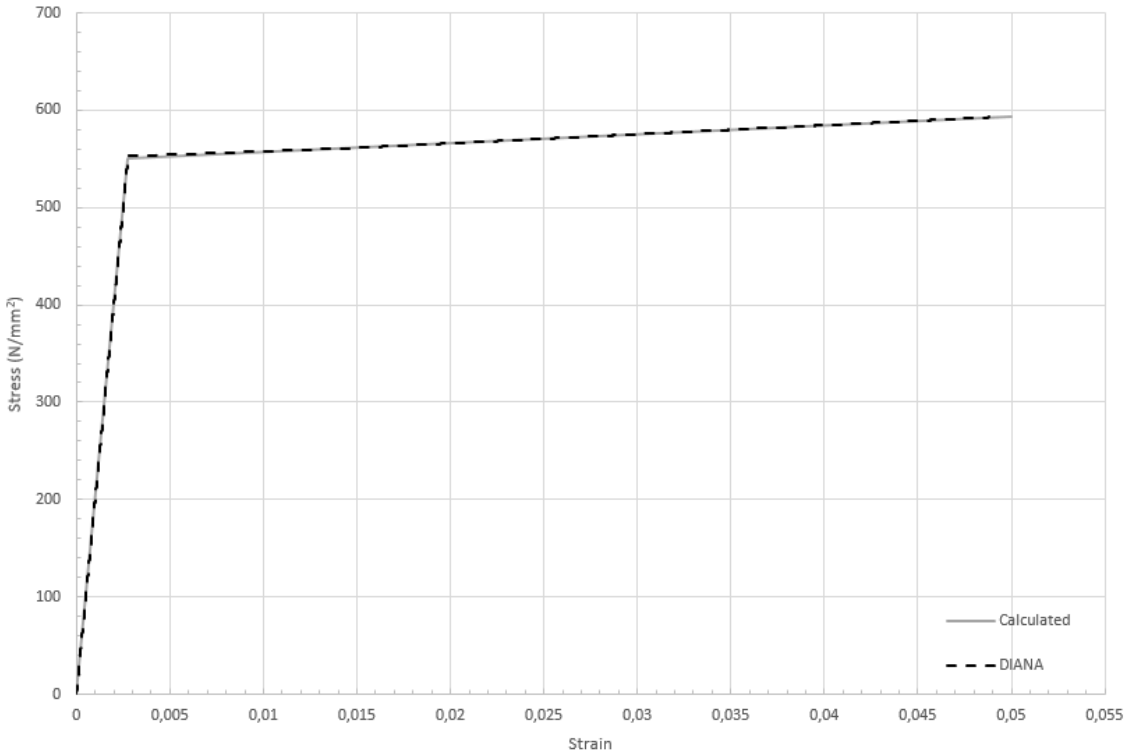


Figure 4.6: Stress-strain relation reinforcement steel, analytic and based on DIANA model

It can be concluded that the reinforcement model behaves accordingly. The script for this model is included in Appendix A3.

4.4 Bond-slip of the Reinforcement

The bond-slip of the reinforcement will be modeled according to the equations in paragraph 2.3.1. As described in this paragraph, there are a few different bond behaviors to take into account. These bond behaviors are different for the longitudinal reinforcements in the eventual model, which are supposed to be ribbed reinforcement bars and the diagonal bars of the lattice girders, which consist of plain surface bars. To verify the model, in both cases, a good bond is assumed.

Based on Table 2.2, this should result in a maximum shear stress of 15,41 N/mm^2 in case of a good bond condition for ribbed bars. This maximum shear stress should be reached after a slip of 1 mm, after which the shear stress should stay constant until a slip of 2 mm has been reached. Beyond a slip of 2 mm, the shear stress should gradually reduce until it reaches the residual shear stress, which is 6,16 N/mm^2 at a slip of 10 mm in this case. It should be noted that this value is dependent on the mean compressive strength of the concrete, which may be different for the precast plate and the in-situ layer.

In case of the plain surface bars, for the lattice girder diagonals, with a good bond condition, there is no difference in the maximum shear stress and the residual shear stress. Based on Table 2.3, this maximum shear stress should be reached after a slip of only 0,01 mm at a shear stress of 0,62 N/mm^2.

Both described cases, and the case of all other bond conditions for ribbed bars, have been tested in DIANA. The model used for these verifications are based on the previous described models. In this case, the precast plate is removed and the reinforcement bar is also cut at the bottom of the in-situ layer. The supports are located at the bottom and the top of the concrete to prevent any movement of the concrete. The deformation is applied at the bottom of the reinforcement bar. A 10 mm deformation is applied, this should allow the shear stress to reach its residual stress, beyond which it is going to be constant. This model is presented in Figure 4.7. The results of this model are presented in Figure 4.8.

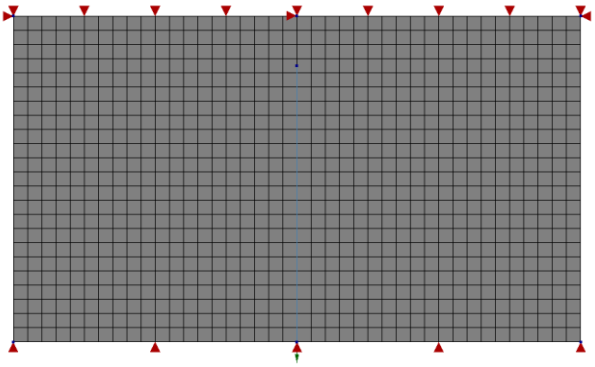


Figure 4.7: Bond-slip model

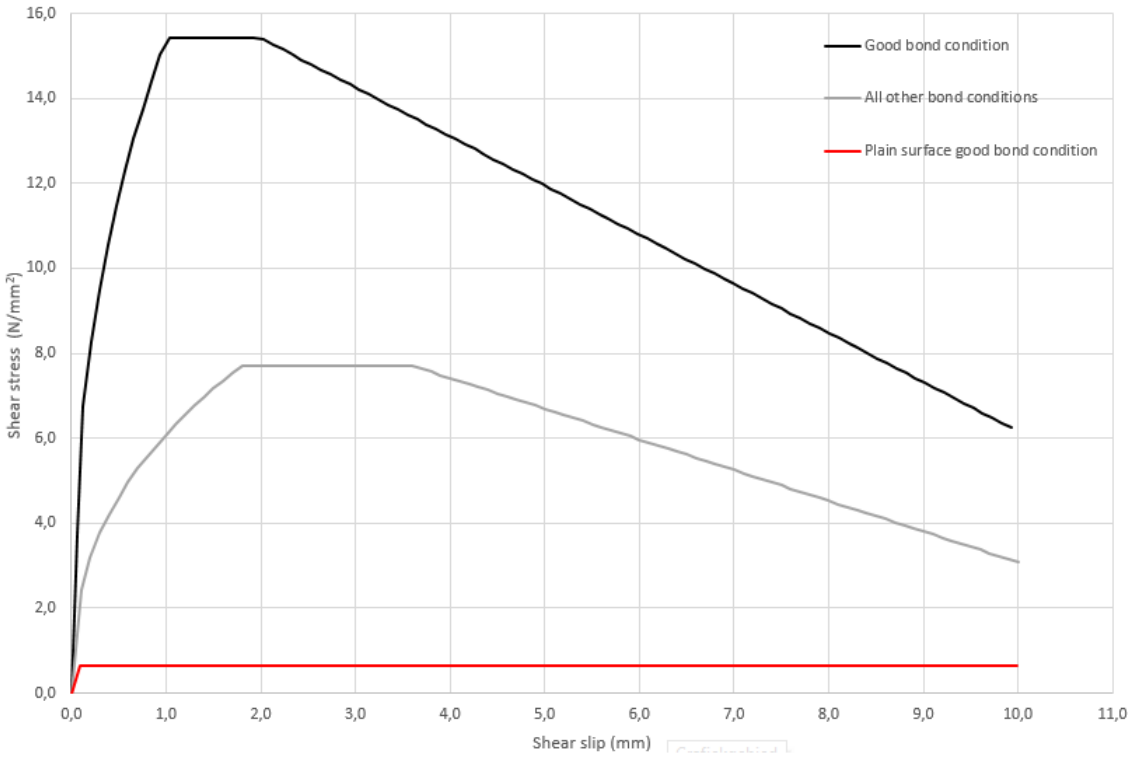


Figure 4.8: Bond-slip relation, based on DIANA models

It can be concluded that the bond-slip behavior of the reinforcement behaves according to the described relations in paragraph 2.3.1. The black and grey line take the same shape as is presented in Figure 2.7 and the red line remains constant after reaching a shear stress of 0,62 N/mm^2 as expected. The script for this model is included in Appendix A4.

5 ANALYSIS

By implementing all the verified material models into one single finite element model, an attempt is made to describe the actual failure behavior of the experiments described in paragraph 3.6. When this 2D model approaches the experimental results, a new version of the finite element model will be created to extend the model to 3D.

5.1 2D Analysis

All models, created to approach the real experimental results, are based on the first specimen of their respective set described in Wijte [1]. In Wijte [1], all the concrete properties are tested by cubic compression tests, which have to be converted to mean- and cylindrical values to be used in a realistic FEM model. These conversions are made according to Equation 14 and 15. The material properties used in the models are presented in Table 5.1. Subsequently, these values influence all the parameters specific for the concrete. The interface has been varied between different versions of the models and the steel properties are always the same, as presented in Table 4.2.

Specimen	f_{ck} Precast plate (N/mm^2)	f_{ck} In-situ layer (N/mm^2)
T01	48	27,6
T13	32,1	33,6
T19	28,3	9,6
T22	18,8	37,9
R1	41,7	29,6

Table 5.1: Converted concrete properties

All the different models of the specimens have been through a few versions, up to 11 versions. Almost all the simulations were initially too stiff with the material properties according to the experiments. By changing the value of one parameters at a time, for example reducing the elasticity modulus or the interface strength, the later versions were improved to describe the experimental results better. These results will be visualized in the next paragraph, along the description of the model.

5.1.1 FEM-model

The specific model described in this paragraph is the model based on specimen T13. The results of the other models will be covered in the next paragraph. Due to the problems encountered in creating this model, this is also the model in which the most iterations have been made. The final model described in this paragraph is version 10 (V10). There are however V11 variants of some specimen. An explanation of these versions will be given in the next paragraph.

The model and its mesh are presented in Figure 5.1 and Figure 5.2. The floor model has a total height of 250 mm, 70 mm for the precast plate and 180 for the in-situ layer, and a length of 1900 mm. To reduce the calculation time, only half of the total length (3800 mm) of the specimen has been. At the right side of the model, a support was applied which creates the symmetry condition in the in-situ layer. As visualized in Figure 5.2, the mesh on the left side of the model is 5 times larger than the mesh on the right side. This difference in element size has also been made to reduce the calculation time. This choice can safely be made because the influence of the failure mechanism should not reach any further than the load point of the model.

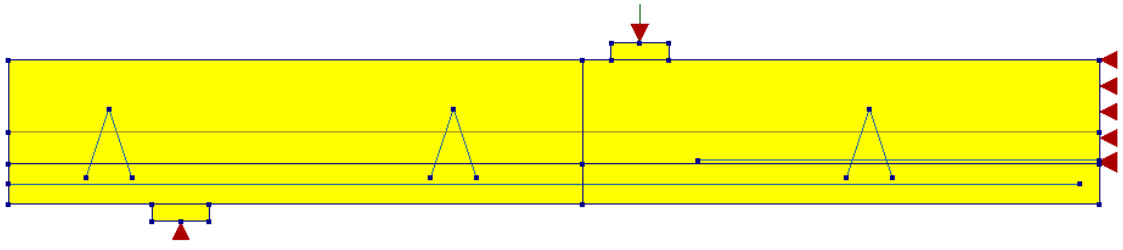


Figure 5.1: FEM-model of specimen T13 V10, DIANA

To describe the deformations accurately, the whole model consists of quadratic elements. This also benefits from the reduced amount of elements, since quadratic elements take much more calculation time than linear elements.

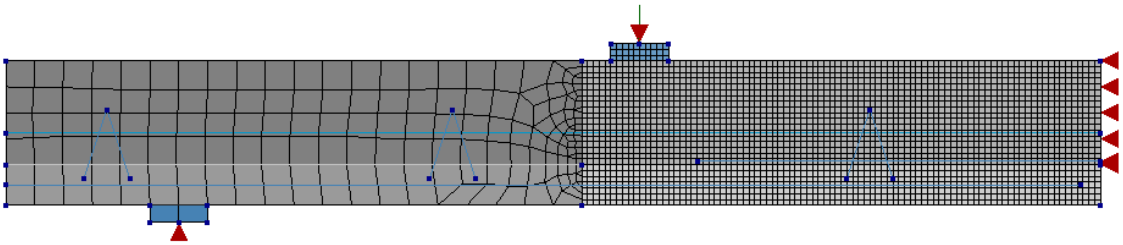


Figure 5.2: Mesh of specimen T13 V10, DIANA

The model has been created with 2D plane stress elements with a thickness equal to 1 m, this does not affect the calculation time but guarantees the exact same geometry as the experimental specimen. For this same reason, the exact amount of reinforcement has been modelled over the width of 1 m. Since this depth is only virtual, the reinforcement bars are modelled individually at the exact same coordinates, DIANA will smear this total amount of reinforcement out over the total virtual width of the model.

For the coupling reinforcement, this results in 7Ø12-150, although the center to center distance is in this case redundant. These bars have a modelled length of 700 mm until the symmetry line and an area of 113,1 mm^2. The longitudinal reinforcement in the precast plate has been modelled as 15Ø8-65, again the center to center distance is redundant for now. These bars have an area of 50,3 mm^2 and end 35 mm before the end of the precast plate at the right side of the model. These bars have all been modelled as bond-slip reinforcement according to a 'good bond' as described in paragraph 4.4.

The lattice girders have also been modelled as bond-slip reinforcement, only according to 'all other bonds'. These bars have been modelled diagonally, but only in XY-direction due to the limitations of this 2D model. Due to these same limitations, their usual anchorage cannot be modelled. Therefore, these lattice girders have been modelled with an anchor point with a specific value. This anchor point will stay in place up to a load of 10 kN. This value has been chosen to guarantee the failure of the concrete before the lattice girder itself would fail.

The interface for this model started out with the parameters as tested in paragraph 4.2 in version 0 (V0) of the calculation. However, these values did not result in an accurate description of the experimental results. Various parameters have been gradually changed to describe the results more fitting. The final interface parameters in calculation V10 are drastically different from the first version, but resulted in a much better fit over all compared to the previous versions of the model. The interface parameters used in T13 V10 are presented in Table 5.2. Especially the values for the normal stiffness and shear stiffness are much lower than tested before, even lower than used by Lundgren.

Cohesion (c)	0,2 N/mm^2
Friction angle (ϕ)	22°
Kn	10 N/mm^3
Kt	10 N/mm^3
Interface opening model	Gapping model
Gap appearance	Brittle

Table 5.2: Interface parameters used in T13 V10

5.1.2 2D results

The results of the initial models behaved, as previously mentioned, much stiffer than the experiments showed, especially in the elastic part of the test. Because this was the first problem to be solved, the first few versions were only loaded until a deformation of 5 mm was reached. After this initial stiffness of the finite element model approached the experiments, a larger deformation was applied.

The final T13 V10 model still behaved stiffer over the initial elastic part of the test, however, the results approached the experiments a lot more fitting after the first cracks occurred in the concrete. In Figure 5.3, the deformed and cracked model is visualized. However, this deformation is exaggerated, since the actual deformation would not be visible on this scale. Furthermore, the Interface at the right side of the model did actually break open, but even on this exaggerated scale, it is not visible in this version.

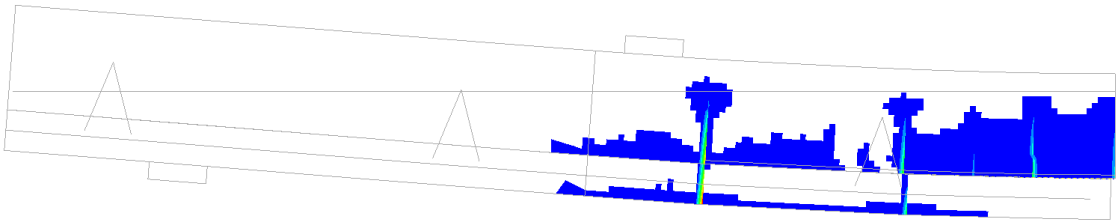


Figure 5.3: Principal crack-widths FEM model T13 V10 DIANA

The load-deformation graph of T13 is presented in Figure 5.4. This version has also been chosen as the final version, before moving to 3D, because of the stability of the calculation. As can be seen for T13 V0, there are a few peaks, which means the calculation could not converge. This was a real problem through all of the previous versions of the model. The same instability can be seen in Figure 5.5 and Figure 5.6. These figures show the increase of the width of the horizontal joint at the height of the precast plate and the increase of the width of the vertical joint between the precast plate and the in-situ layer respectively. Both these graphs show T13 V10 describing the actual experimental results even better.

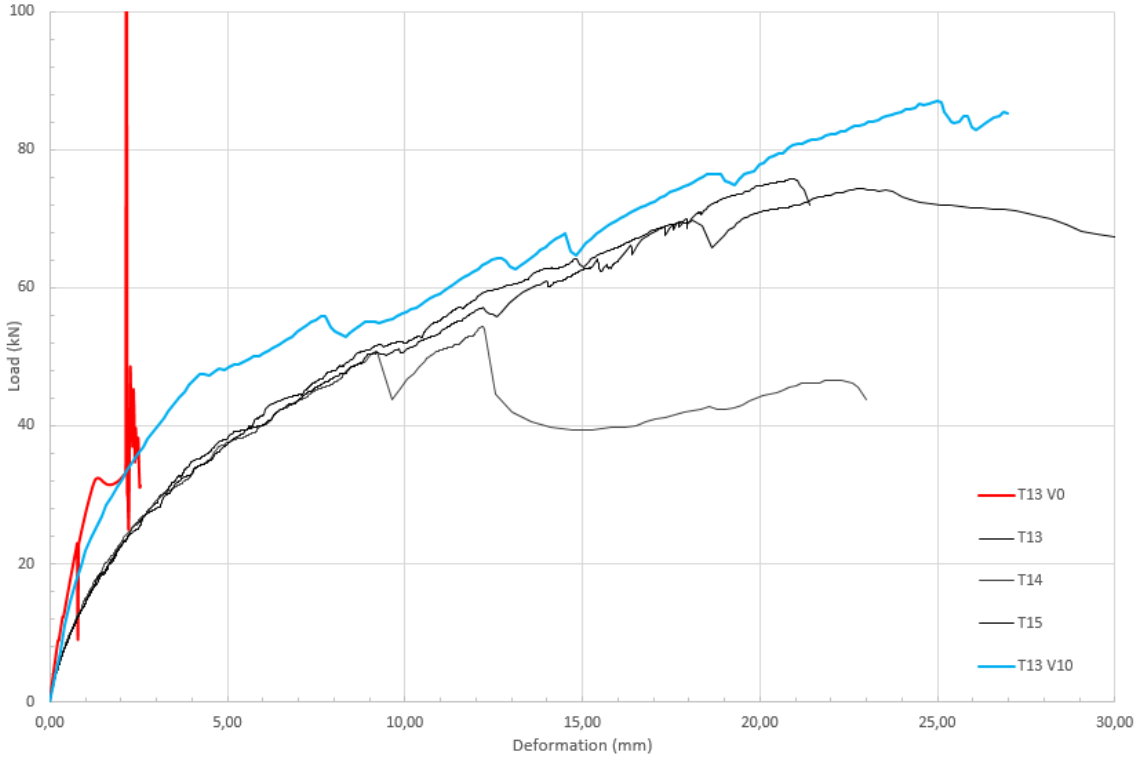


Figure 5.4: Load-deformation graph T13 DIANA model

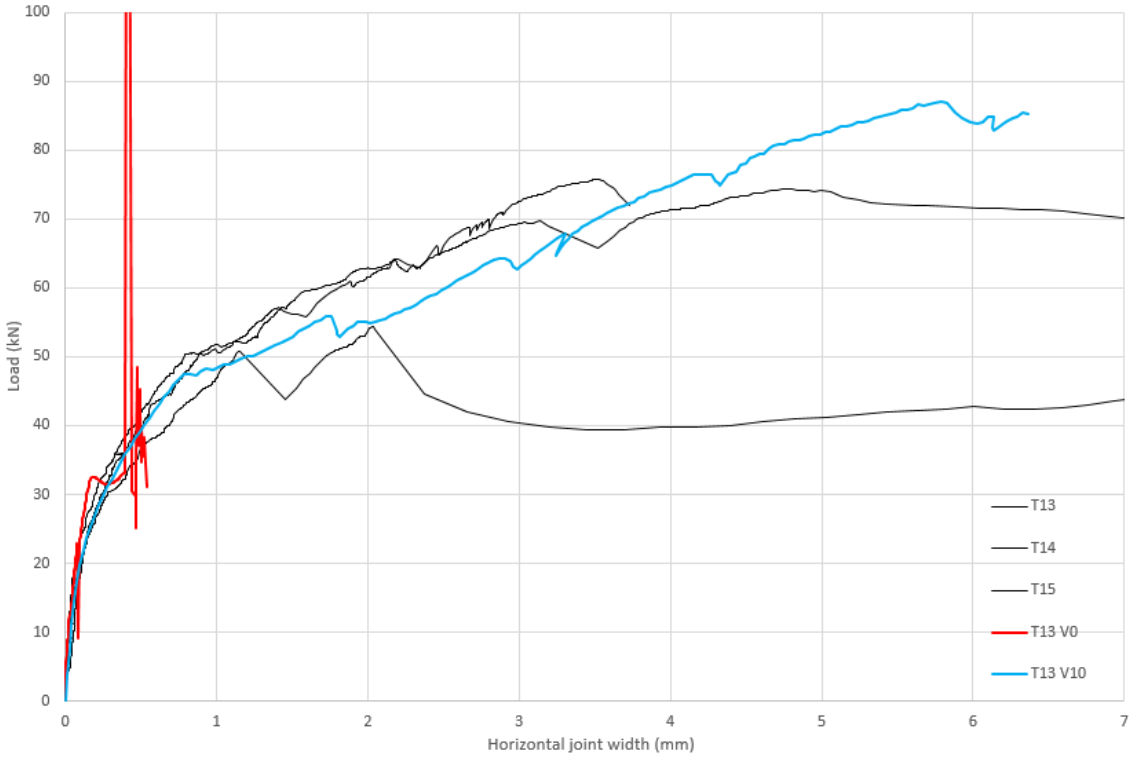


Figure 5.5: Load-horizontal joint deformation graph T13 DIANA model

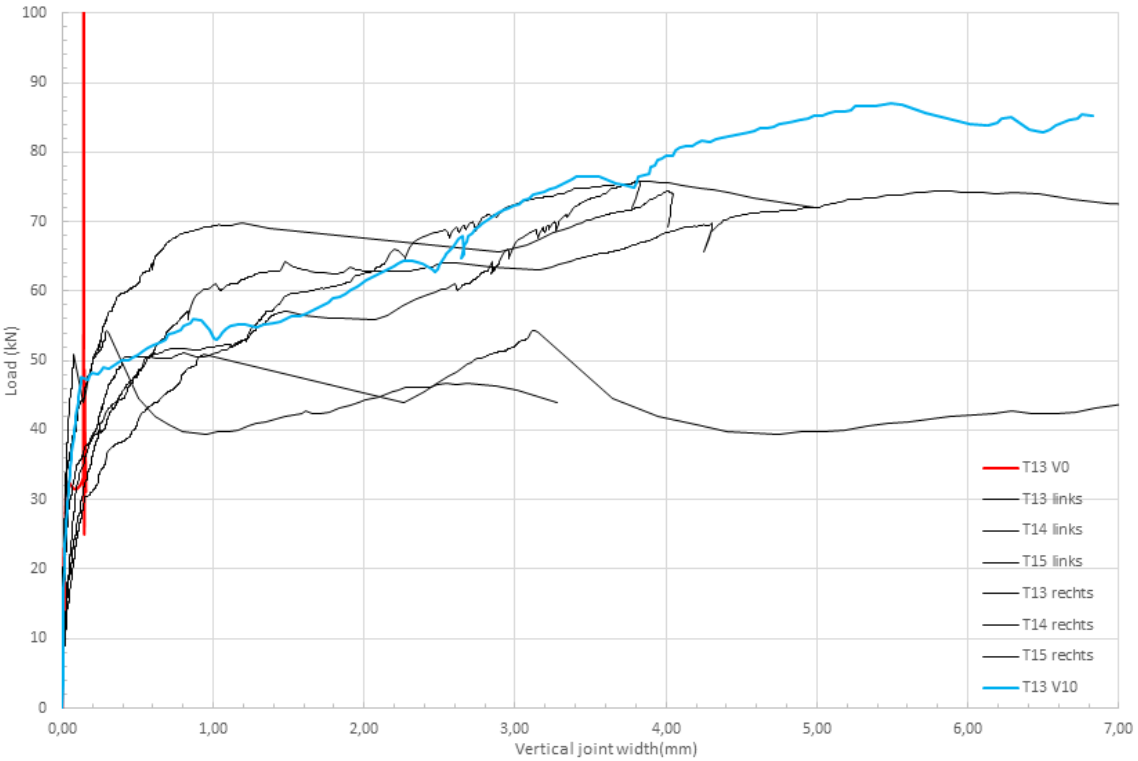


Figure 5.6: Load-vertical joint width graph T13 DIANA model

The parameters used in T13 V10 were also applied at the other specimen, but with a lesser fit. These calculations were also more unstable, some of these results will be elaborated in the next paragraph. With this in mind, the choice was made to extend the 2D model of T13 V10 to a 3D model. The scripts for these models are included in Appendix B.

5.1.3 Further results

As stated in the previous paragraph, this paragraph will focus on the results of the remaining specimen. Also stated previously, not all the specimen went through the same number of versions. However, the version number indicates which parameters have been changed. For the remaining specimens, version 11 (V11) will be used. This version has the same parameters as T13 V10, only with a more simplified analysis, for example a larger step size and no arc length control. This results in a stable calculation, however, with coarser results. This will also show that these parameters will not be able to describe the results of all the remaining experiments as good as T13 V10 did.

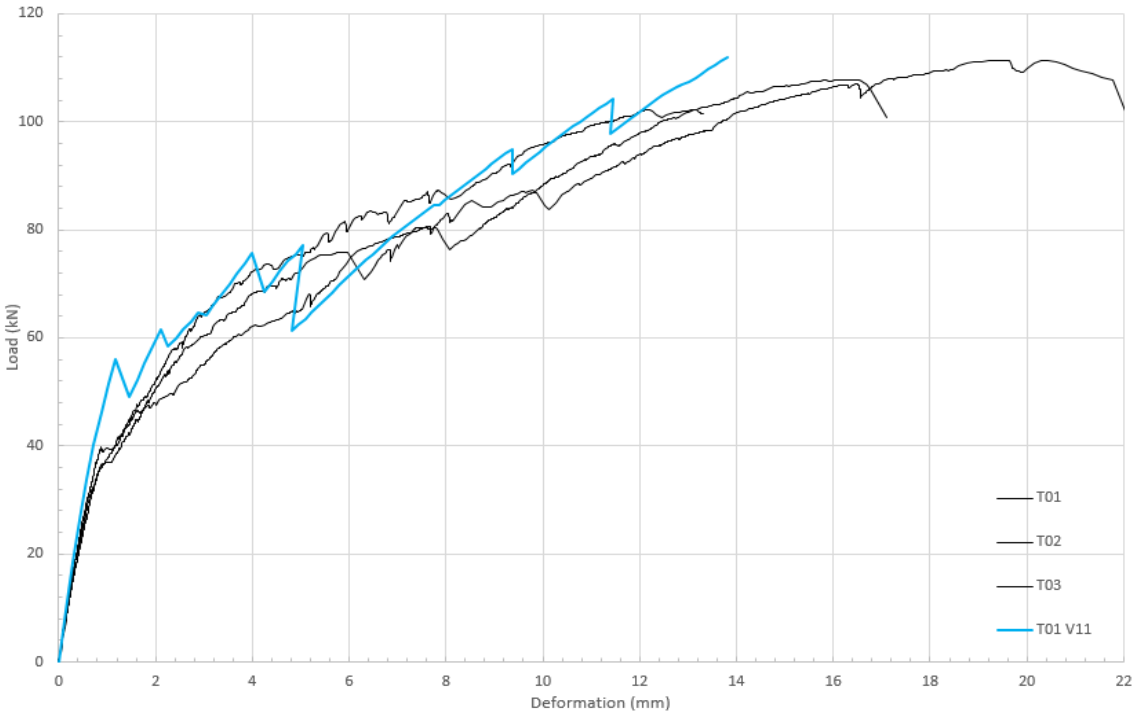


Figure 5.7: Load-deformation graph T01 DIANA model

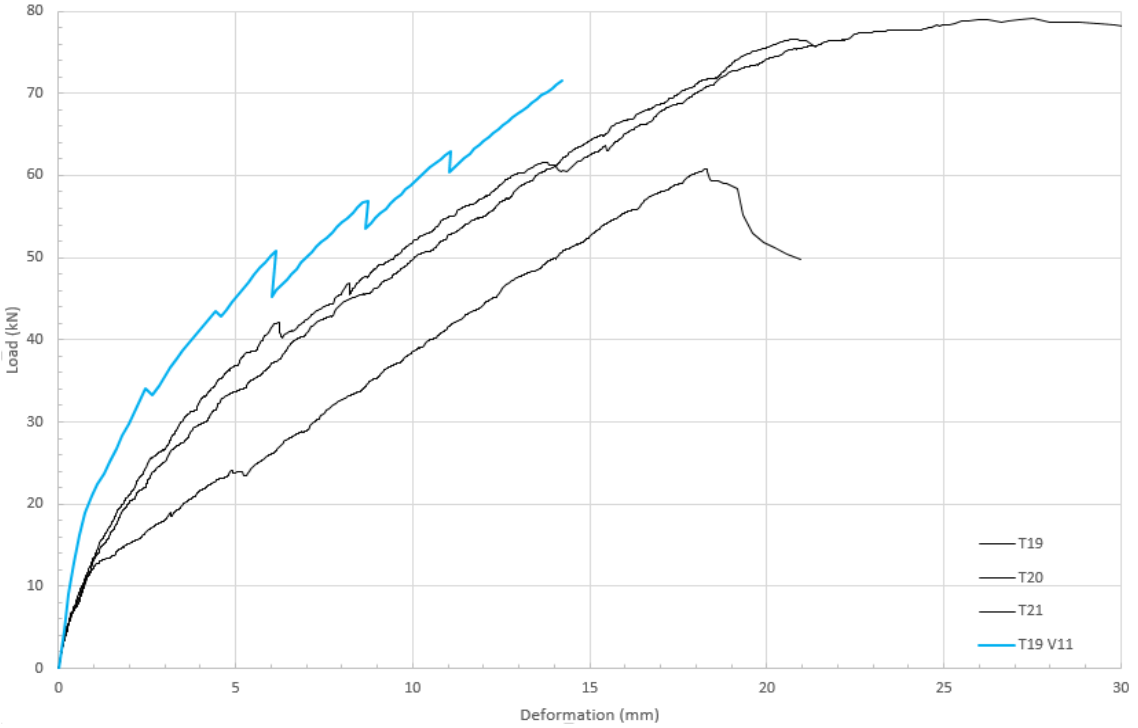


Figure 5.8: Load-deformation graph T19 DIANA model

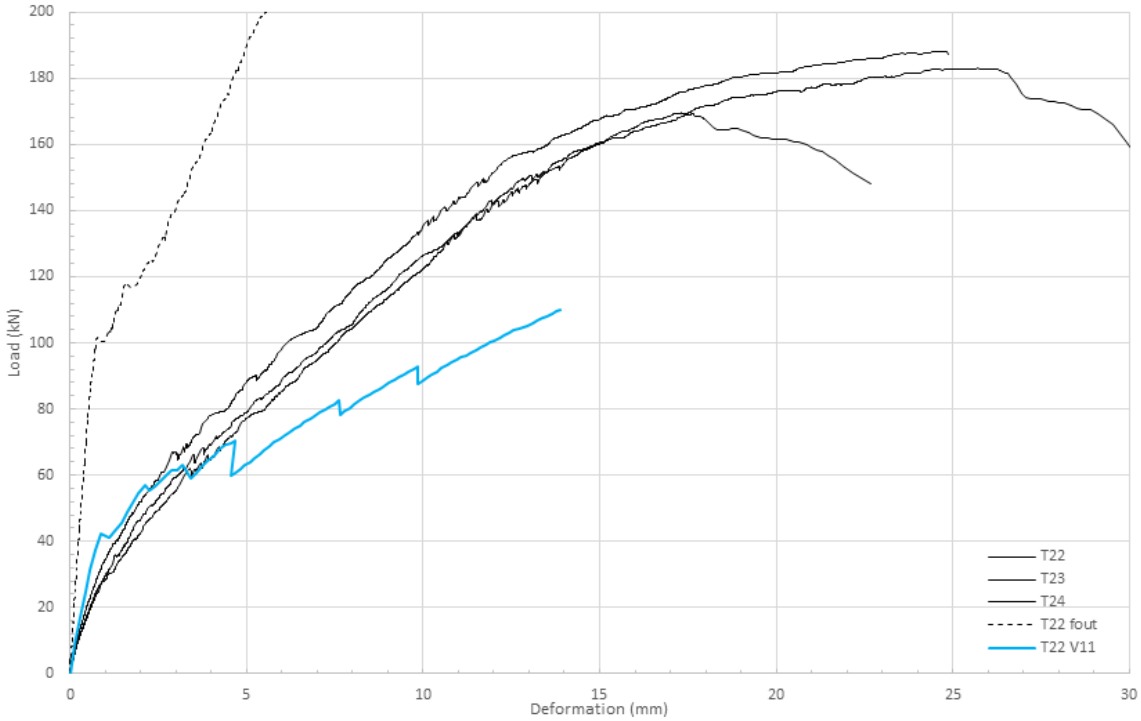


Figure 5.9: Load-deformation graph T22 DIANA model

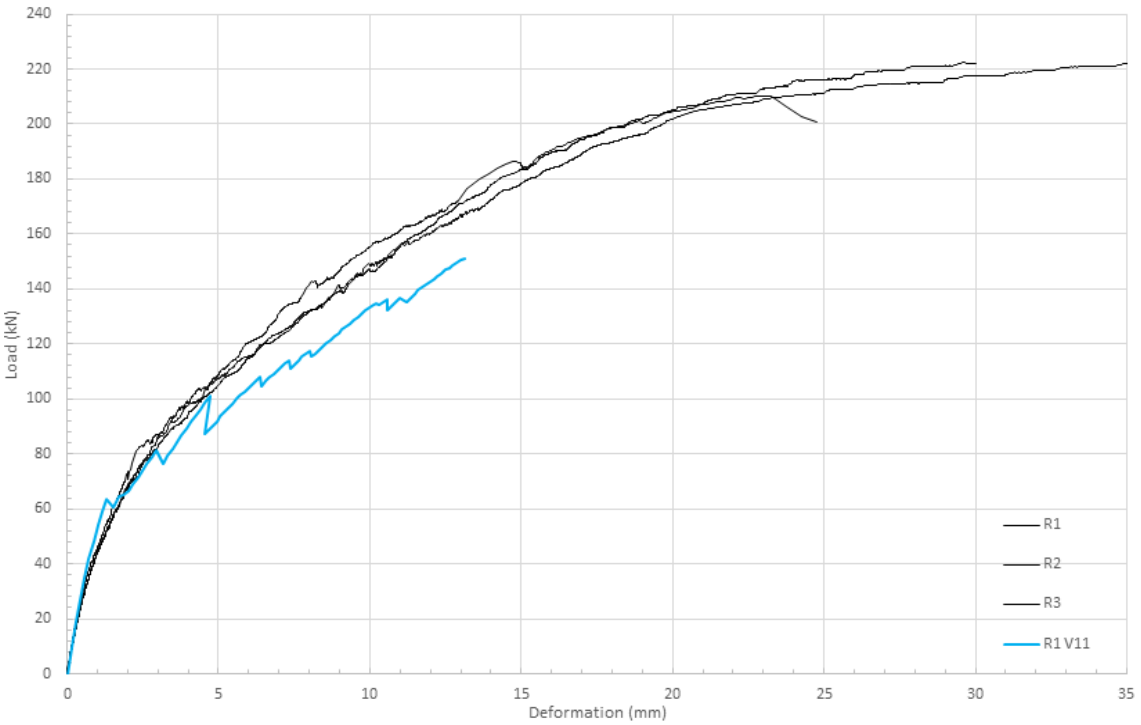


Figure 5.10: Load-deformation graph R1 DIANA model

The applied deformation for all these versions has been limited to 10 mm. This resulted in a similar deformation of 14 mm at the middle (the symmetry condition) for all the models. As can be seen for all the results, the initial stiffness of the models is still higher than the experimental results. This problem could not be solved without deviating from the material parameters as described according to Chapter 2. After cracking is initiated, a similar pattern can be seen for all the models, where a decrease in the load can be observed when a new crack occurs. The remaining graphs are included in Appendix C, these graphs show an overall similar pattern for the horizontal joint width, but almost no vertical joint width, this is the result of the simplified analysis. To improve these results a recommendation will be made in paragraph 6.2.

5.2 3D Anchorage of the lattice girder

To be able to extend the 2D model to a 3D version, all the individual models described in Chapter 4 have also been extended to 3D. This was necessary to get familiar with the way of scripting these models and the influence of the Z-axis in the model. The scripts of these models are also enclosed in Appendix D.

The most important difference between the 2D and 3D models, is the lattice girder. To be able to understand how the lattice girder is behaving, especially during failure, smaller models have been created. The first model is just a single vertical reinforcement bar in a concrete block to simulate a pull-out test. With these results, the lattice girder can be compared to the single vertical bar to verify if these loads are in the same order of magnitude. These models will first be elaborated on in the upcoming paragraph, before describing the analysis of the full 3D model.

To calculate the theoretical value of the anchor strength, the calculation has been simplified to a vertically loaded single headed anchor. The failure mechanism which is attempted to describe is concrete cone failure. This is the failure mechanism in which a concrete cone will be pulled out before the reinforcement itself will fail. The strength of a single headed anchor can be calculated according to Equation 19 from NEN-EN 1992-4:2018+NB:2019 [15].

$$N^0_{Rk,c} = k_1 \times \sqrt{f_{ck}} \times h_{ef}^{1,5} \qquad \textit{Eq. 19}$$

With:

$N^0_{Rk,c}$ is the characteristic resistance of a single headed anchor

K_1 is 12,7 for uncracked concrete and 8,9 for cracked concrete

f_{ck} is the characteristic compressive cylinder strength in MPa

h_{ef} is the embedment depth in mm

Since the calculation is supposed to simulate the full pull-out mechanism of the lattice girder, the value for cracked concrete is used. The compressive cylinder strength had to be changed to the mean compressive strength to be able to calculate a realistic value, which is in this case 38 N/mm^2. The embedment depth is based on the experiments, with an embedment depth of 24,5 mm. This should result in a theoretical anchor strength of 6.653 N. However, a few remarks should be taken into account while comparing this value to the DIANA results. In the first place, this function is based on a single headed anchor, without any connections to any other reinforcement. Secondly, while comparing this value to a full lattice girder, the diagonals need to be taken into account, logically, this should result in a higher pull-out resistance. Finally, this function has originally be designed for deeper embedment depths. The embedment depth of the experiments near the absolute limit of the viability of Equation 19.

Besides the actual lattice girder, three models have been created to compare the theoretical loads. The first two models are based on the geometry of the actual experiments and the third model has the option to thicken the precast concrete plate to extend the embedment depth. All models however, are based on a concrete block of 200 x 200 x h mm^3. Furthermore, the same material parameters have been used as described in Chapter 4 for the precast concrete and the in-situ concrete, the scripts for these models are included in Appendix E.

The first two models presented in Figure 5.11 are based on the geometry of the experiments. Both models consist of a precast concrete plate with a thickness of 70 mm (of which the top 24,5 mm is not visible in the figure) and an in-situ layer of 180 mm (not visible in the figure). The model on the left has just one horizontal reinforcement bar embedded in the precast concrete, connected to the vertical bar, while the second model has two horizontal reinforcement bars perpendicular to each other. The second bar was added to distribute the anchor load more evenly in the concrete, which should result in a more symmetrical cone failure of the concrete. The in-situ part of the models is in both cases modeled in the same way, since this should be of lesser influence to the result.

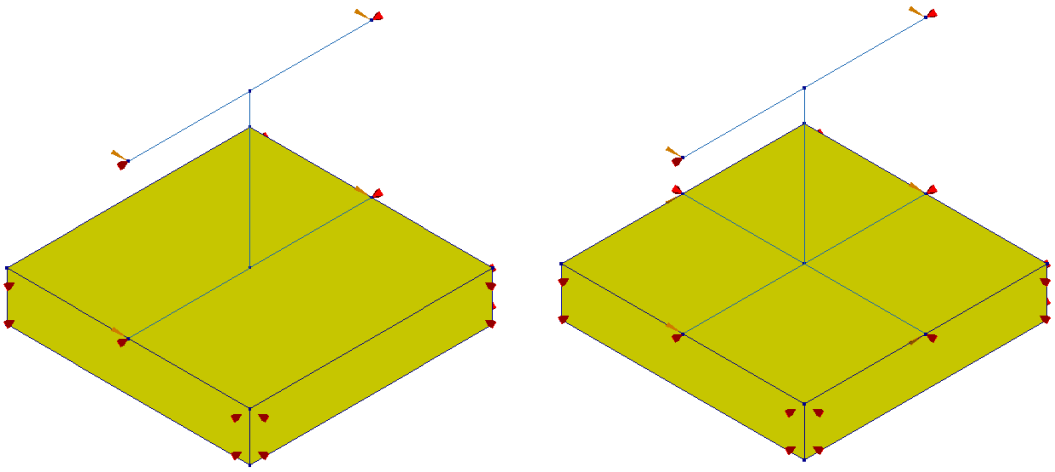


Figure 5.11: Anchor model, single horizontal bar (left), cross-bar (right), DIANA

The results of these models are presented in Figure 5.12. As could be expected, the single bar resulted in a lower resistance than the cross-bar, respectively 4,4 kN and 6,7 kN. These loads are also the maximum loads present in the vertical reinforcement, since the interface has already fully opened between the precast concrete and the in-situ layer, the reinforcement is the only thing tying the model together.

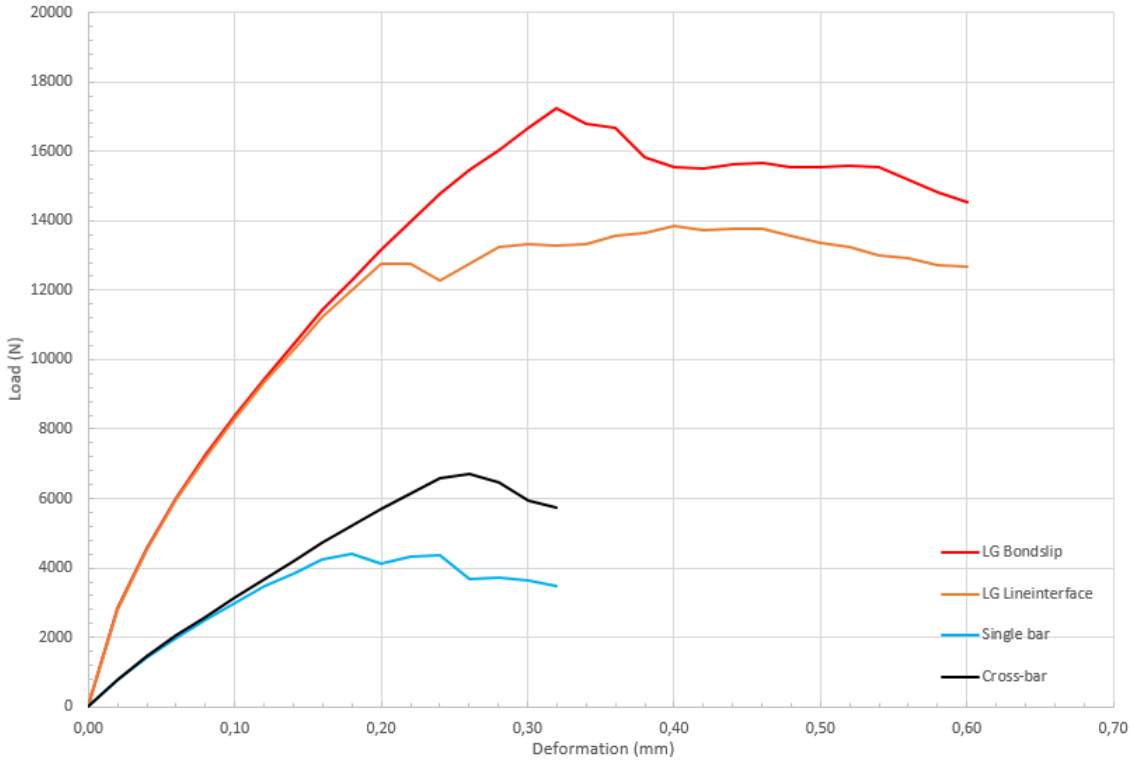


Figure 5.12: Load-deformation graph pull-out resistance of a single bar and lattice girder system, DIANA

The other two lines present in Figure 5.12 represent the pull-out resistance of the full lattice girder. The difference between these two results are the way in which the horizontal reinforcement bars in the precast concrete have been modelled. The bond-slip version is modeled in the same way as all the other reinforcement before. However, the bond-slip option in DIANA is only useful in case of a deformation in the normal direction of the reinforcement bar. In this case, the load acting on these reinforcement bars is perpendicular to their own direction. Therefore, another model has been made with a line interface between the reinforcement and the concrete. In this case, the bar should not stay connected to the concrete when the tension threshold of 0,5 N/mm^2 is reached. The model used for these calculations is presented in Figure 5.13.

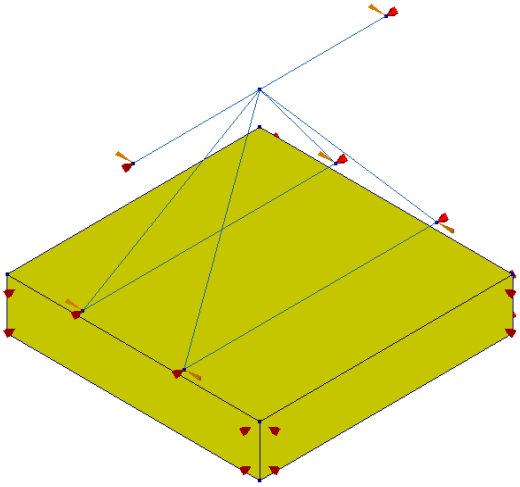


Figure 5.13: Lattice girder model, DIANA

The bond-slip model resulted in a maximum resistance of 17,2 kN and the line interface model in a resistance of 13,8 kN, which is a substantial difference. However, to be able to compare these values with a single headed anchor, the results have to be divided by 2. This is a result of the fact that the lattice girder consists of 4 diagonal bars, crossing the interface and 4 anchor points, which are only modelled half. Therefore, this model represents only 2 fully modelled anchors, with a resistance of 8,6 kN and 6,9 kN respectively.

The difference in the resistance between these two modeling types is substantial, but can be explained. The model with the line interface starts deforming after the maximum tension of 0,5 N/mm^2 has been reached in the line interface after which this interface opens and there is no further contact between the reinforcement and the concrete underneath. The bond-slip model however, starts only with deforming after the maximum allowable tension of 2,9 N/mm^2 in the concrete has been reached. This also applies to the elements underneath the reinforcement bar. This result can be seen in Figure 5.14 and Figure 5.15 , just beneath the horizontal reinforcement bars. Therefore, a larger load is needed to reach the same deformation. However, the line interface model is less stable than the bond-slip model. When implementing this in the full scale model, this could result in a unstable calculation overall or a very long calculation time.

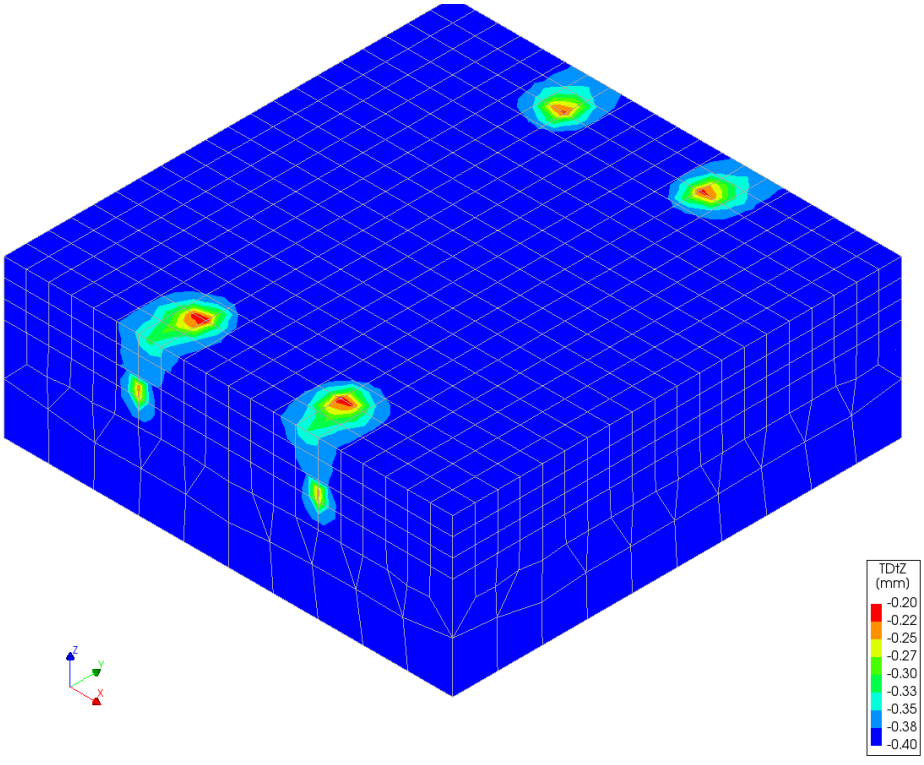


Figure 5.14: Vertical displacement, lattice girder bond-slip model, DIANA

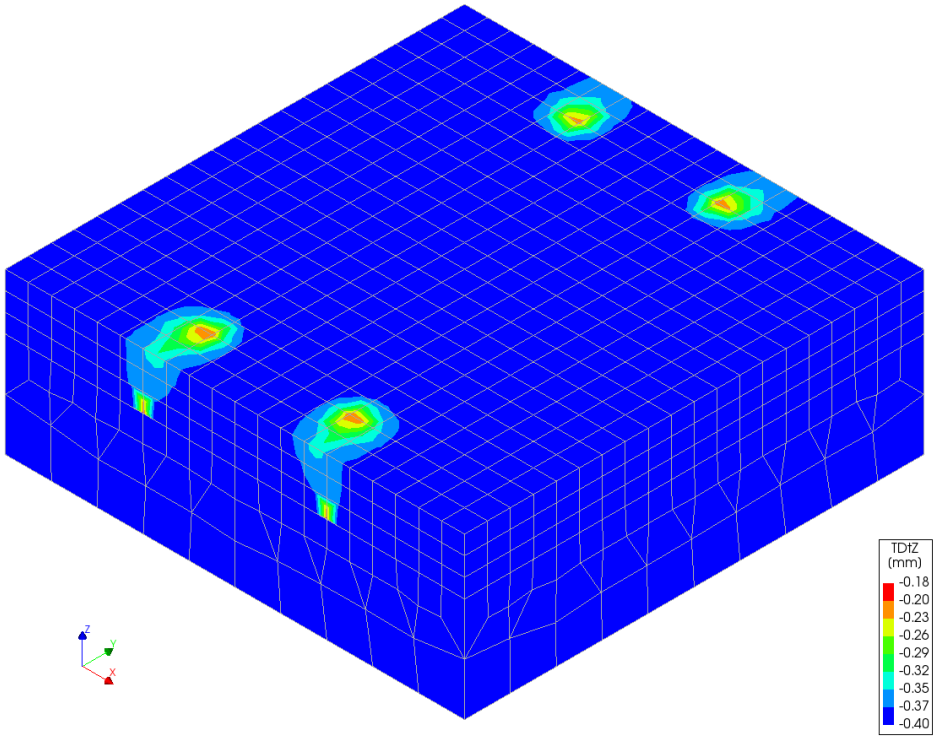


Figure 5.15: Vertical displacement, lattice girder line interface model, DIANA

The resistance of the cross-bar model and the lattice girder with the line interface approach the theoretical results very nicely, with only a difference of 74 N (1,1% more) and 269 N (4% more) respectively. To verify these values, another model has been created in which an attempt is made to model a real single headed anchor, to verify if the model still holds up if the embedment depth is varied. This model is presented in Figure 5.16. This model is very similar to the first anchor models, only in this version the bottom horizontal reinforcement has been replaced by a square (or circular) steel block, which acts like the headed anchor. The expected behavior from this model should be the pull-out of a cone of concrete starting at the steel anchor head or if the maximum steel load would be exceeded, a failure of the vertical reinforcement bar. This last failure mechanism can however be avoided by increasing the diameter of the vertical bar.

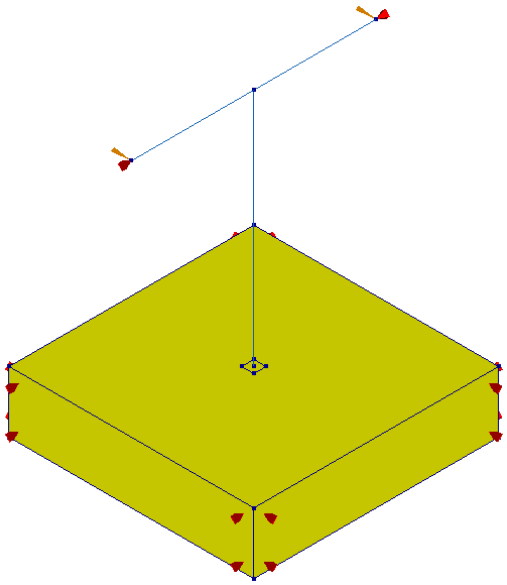


Figure 5.16: Headed anchor model, DIANA

For this model, 4 different anchorage depths have been tested. Starting with the standard depth of 24,5 mm up to 100 mm. The supposed maximum loads according to Equation 19 and the actual results from DIANA are presented in Table 5.3.

Anchor depth	Max load Eq. 19	Max load DIANA
24,5	6.653 N	5.710 N
50	19.397 N	7.039 N
70	32.131 N	7.483 N
100	54.863 N	7.677 N

Table 5.3: Headed anchor resistance according to Eq. 19 and DIANA

The lower anchor depth results in a maximum load somewhat similar to the load according to Equation 19. However, in the case of the deeper anchors, the difference becomes increasingly larger. The same DIANA results are also presented in Figure 5.17.

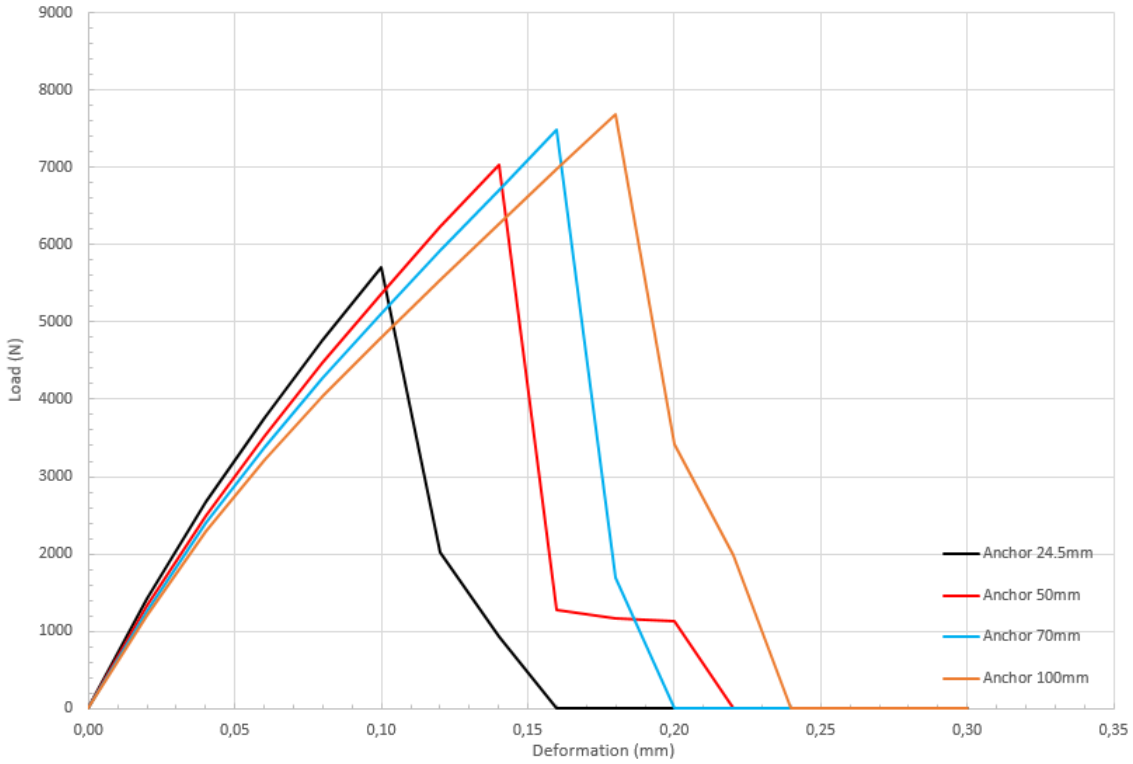


Figure 5.17: Load-deformation graph pull-out resistance of a single headed anchor, DIANA

This difference could be explained when taking a look at the visualization of the crack formation in the DIANA model, one load-step after the maximum load has been reached. As previously stated, at this load-step, the expectation would be to see a pulled-out cone of concrete, starting from the anchor head. However, this is not the case. As can be seen in Figure 5.18, the resulting cracked concrete at this step is relatively small and more cylindrical shaped. Furthermore, this cylinder does not extend all the way to the top of the precast plate, which would be expected.

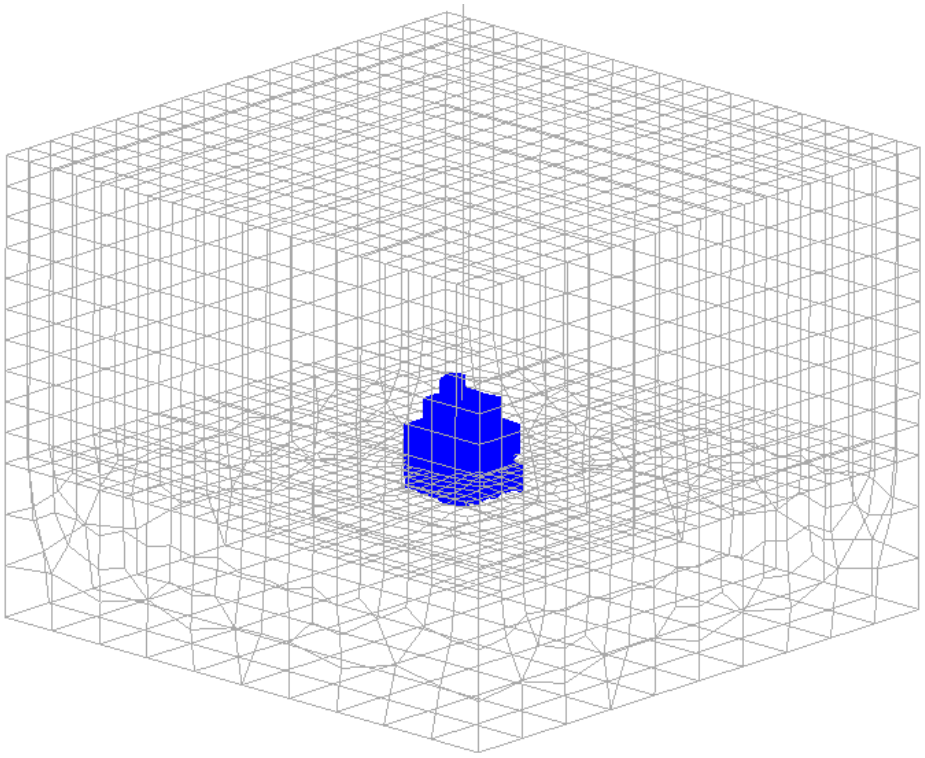


Figure 5.18: Crack-width 1 step after maximum load reached, DIANA

This cylindrical crack formation could be the result of several factors. In the first place, the mesh size. The mesh size at the anchor head is smaller than the mesh size of the concrete. DIANA solves this by gradually increasing the mesh size, starting at the first elements connected to the anchor head. This in itself is a good thing, but for a better result, an overall smaller mesh is needed, which in turn results in a significantly increasing calculation time.

Secondly, a different compression model for the concrete might be needed. The moment the concrete above the headed anchor reaches its maximum compression strength, the whole anchor is pulled through the first few concrete elements above the anchor. This should not happen because of the remaining concrete elements above the anchor.

It can be concluded that the headed anchor models, at least with their current bond-slip reinforcement, are not the right way to model this in DIANA. It should be noted that concrete cone failure can be modeled in DIANA. However, these concrete cone models are not a good representation of the actual attempted validation of a headed anchor. These models reached their anchorage resistance purely from an increased bond stress between the reinforcement and the concrete, which would not be a good representation of a plane surface bar with an anchor.

Furthermore, a point could be made that the resulting anchor resistance could be more reliable in case of a lower embedment depth, only in the case where the crack influence reaches all the way to the top of the precast plate. In this case, there would be no remaining concrete elements left above the affected area, which might have led to an increase of anchor resistance.

With this knowledge, an attempt is made to create a full 3D model of the floor specimen T13.

5.3 3D Floor model

With the addition of the third dimension for this model, the number of elements increase even more than before. To limit this number of elements, the choice has been made to model only a small width of the specimen. The remaining part of the floor will then be simulated by symmetry planes. A total width of 100 mm is modelled. Within this 100 mm, half a phase of a lattice girder can be modelled. The lattice girder will then also be mirrored by symmetry. To be able to calculate the total load for the full specimen, it should be a simple multiplication by 10 of the resulting loads from the DIANA model.

Even with this reduced width for the model, the number of elements would exceed the maximum allowable number of elements for the available computing power. Therefore, the size of the mesh needed also to be increased, which in turn results in an even less accurate calculation. This resulted in a mesh of 20 mm at the right side of the model and a mesh of 50 mm at the left size of the model.

As for the material properties for the full scale 3D model of specimen T13, the model started out with the same properties as with the 2D model. This resulted once again in a very unstable calculation as can be seen from the results in Figure 5.19.

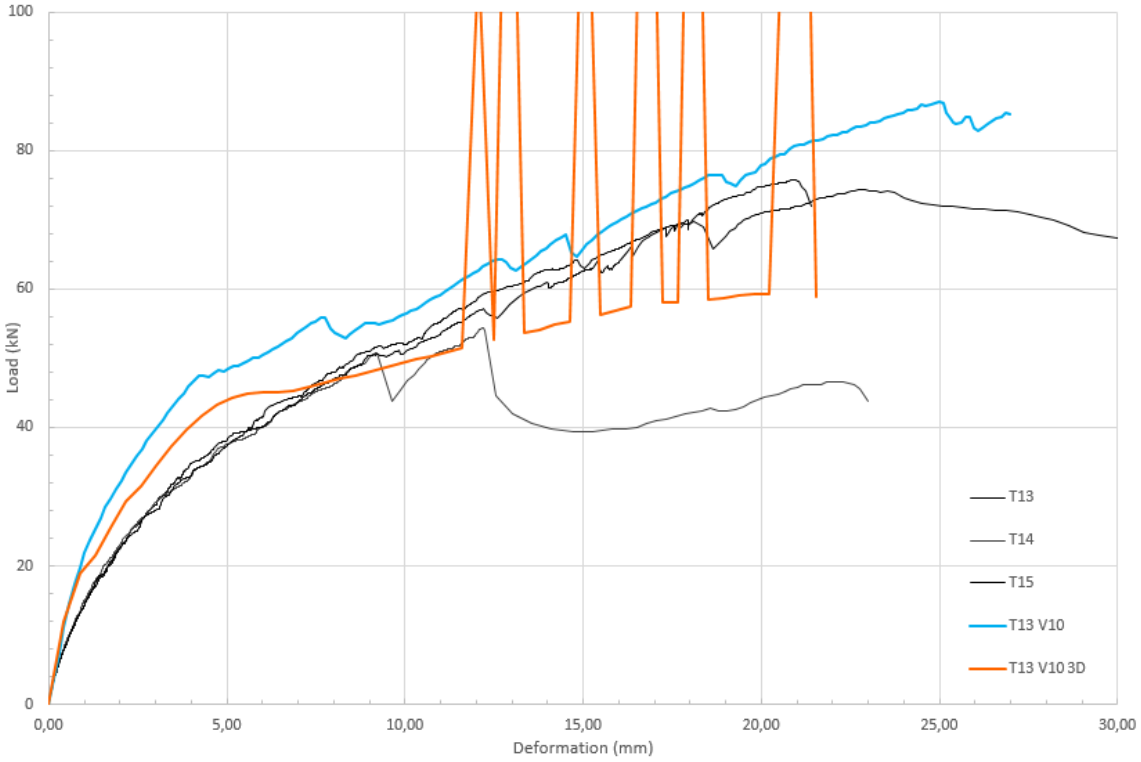


Figure 5.19: Load-deformation graph T13 3D DIANA model

Besides this instability, the interface between the precast concrete and the in-situ layer did not behave as expected. This turned out to be due to the low normal stiffness and shear stiffness of the interface. These values were increased to values significantly higher than used in the 2D model. In this case, the same values were applied as in the model by K. Riemens, see Table 4.3.

To be able to extract some useful information about the behavior surrounding the lattice girder, a finer mesh is needed. To prevent the use of a finer mesh through the whole model, the choice has been made to only apply a smaller mesh at the precast concrete surrounding the right most lattice girder, since just like in the previous 2D models, only the right most lattice girder should be of any influence to the results. The final 3D model with its mesh is presented in Figure 5.20 and Figure 5.21.

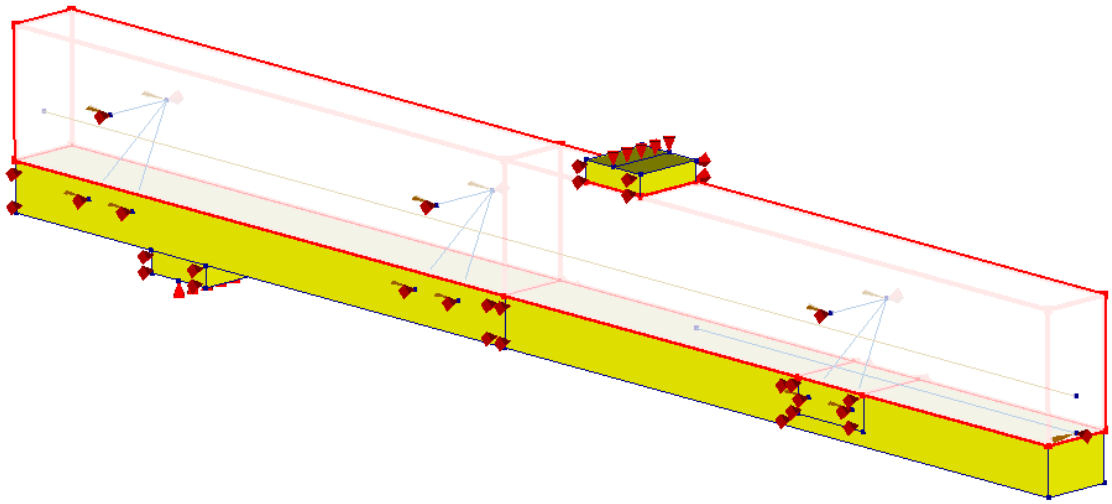


Figure 5.20: Full 3D geometry, DIANA model T13

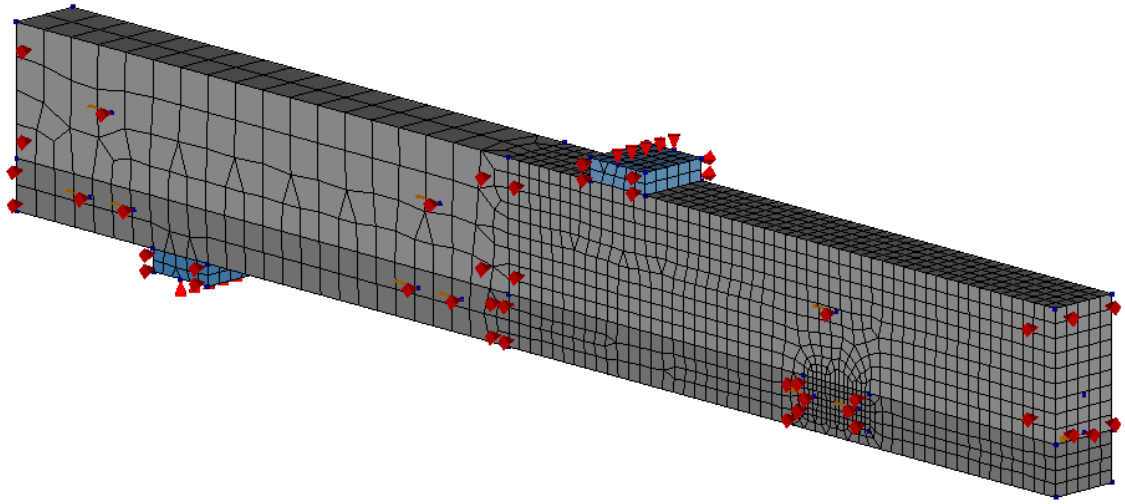


Figure 5.21: Full 3D mesh, DIANA model T13

This final model turned out the behave very similar to the real experiments, at least qualitatively. The way the precast concrete plate delaminates from the in-situ layer and subsequently the initiation of the various cracks in the in-situ layer and at the anchor points of the lattice girders are almost the exact same as the experiments showed. Even the final failure mechanism of the floor is the same, which is the final cracking through the in-situ layer just at the end of the coupling reinforcement.

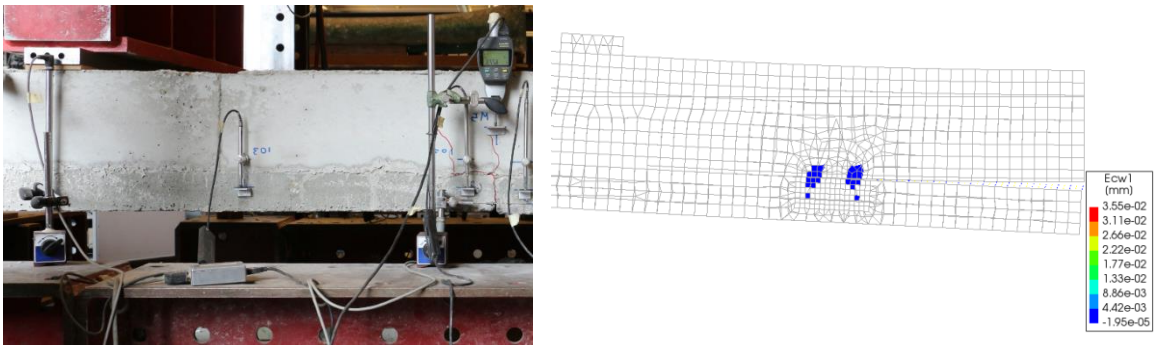


Figure 5.22: Initial formation of cracks in the in-situ layer, T13 experiment (left), DIANA model (right) [1]

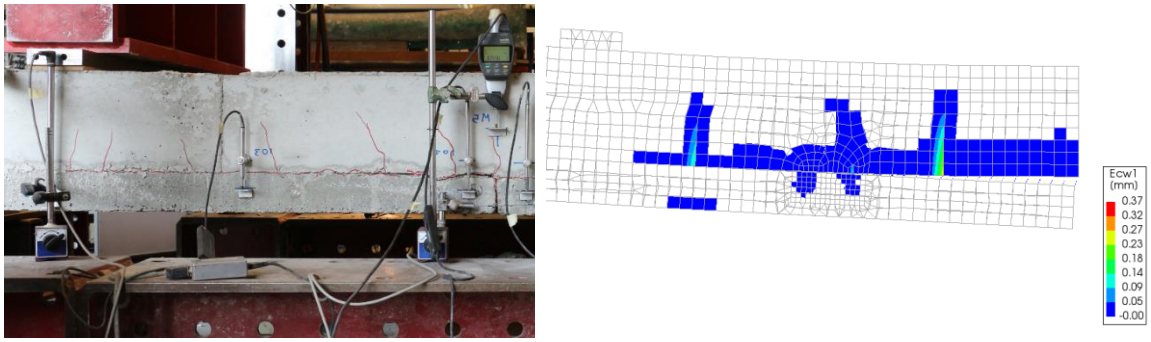


Figure 5.23:Advanced formation of cracks in the in-situ layer, T13 experiment (left), DIANA model (right) [1]

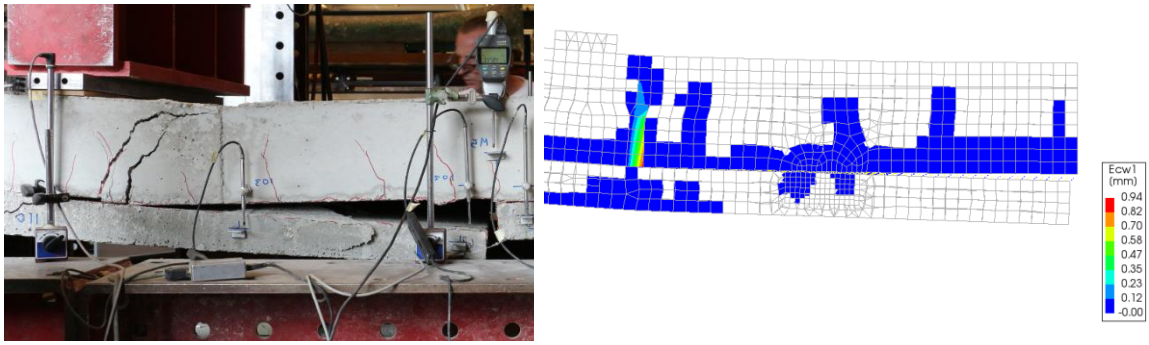


Figure 5.24: Final failure of specimen T13, T13 experiment (left), DIANA model (right) [1]

Qualitatively, this model might be the best result yet, this is visualized in Figure 5.22, Figure 5.23 and Figure 5.24. However, quantitatively are the results contradicting. As can be seen in Figure 5.25, the maximum load reached in the model is only just under 40 kN (peak of the blue line), while two of the experiments reached just over 70 kN. However, the initial stiffness of this model is much closer to the experimental results. Furthermore, the increase of the horizontal joint between the precast plates, as presented in Figure 5.26, looks promising in the initial stage of the calculation, however, the parameters of the interface still need to be adjusted. The same is true for the vertical joint width between the precast concrete and the in-situ layer, as presented in Figure 5.27.

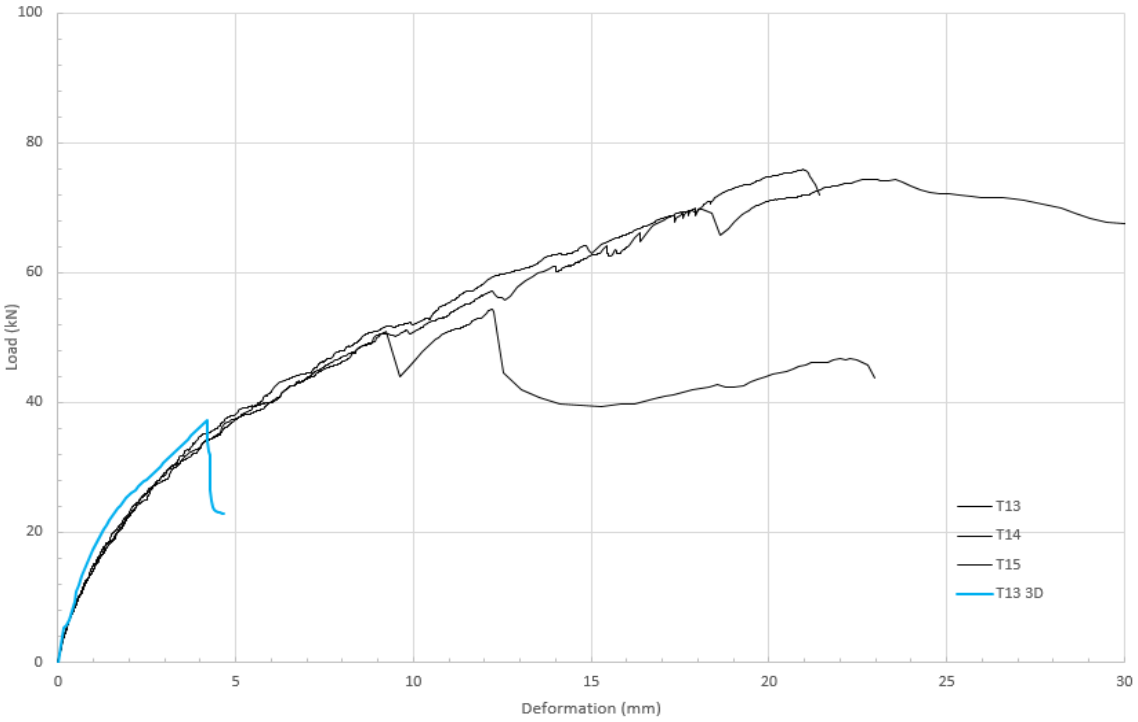


Figure 5.25: Load-deformation graph T13 3D final DIANA model

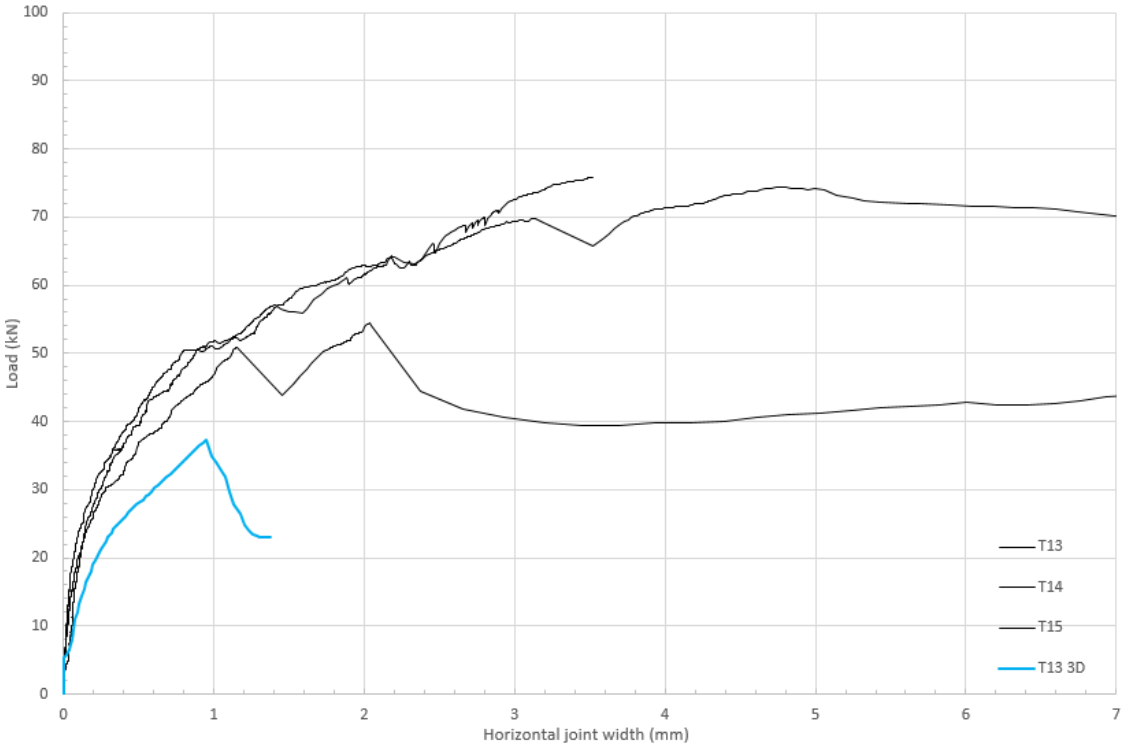


Figure 5.26: Load-horizontal joint deformation graph T13 3D final DIANA model

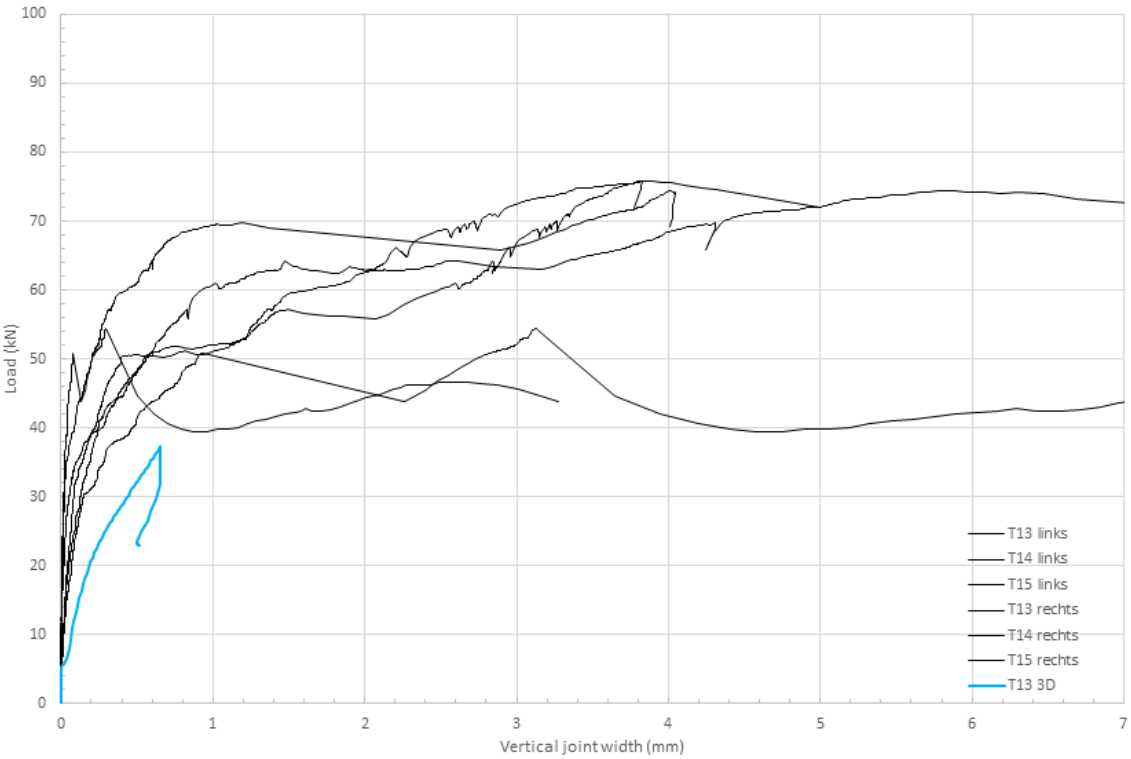


Figure 5.27: Load-vertical joint width graph T13 3D final DIANA model

Zoomed in at the lattice girder detail, it can be concluded that the interface will indeed open up. This is visualized in Figure 5.28. The red color indicates an opening of 0,057 mm opening and the blue color indicates an opening of only 0,0087 mm. Although this opening is still very small, the color indications are as expected. The blue color is localized at the intersection of the lattice girder diagonal bar with the interface. This indicates that the concrete surrounding the lattice girder diagonal is closer together than the concrete over the rest of the cross-section, which would be the red part.

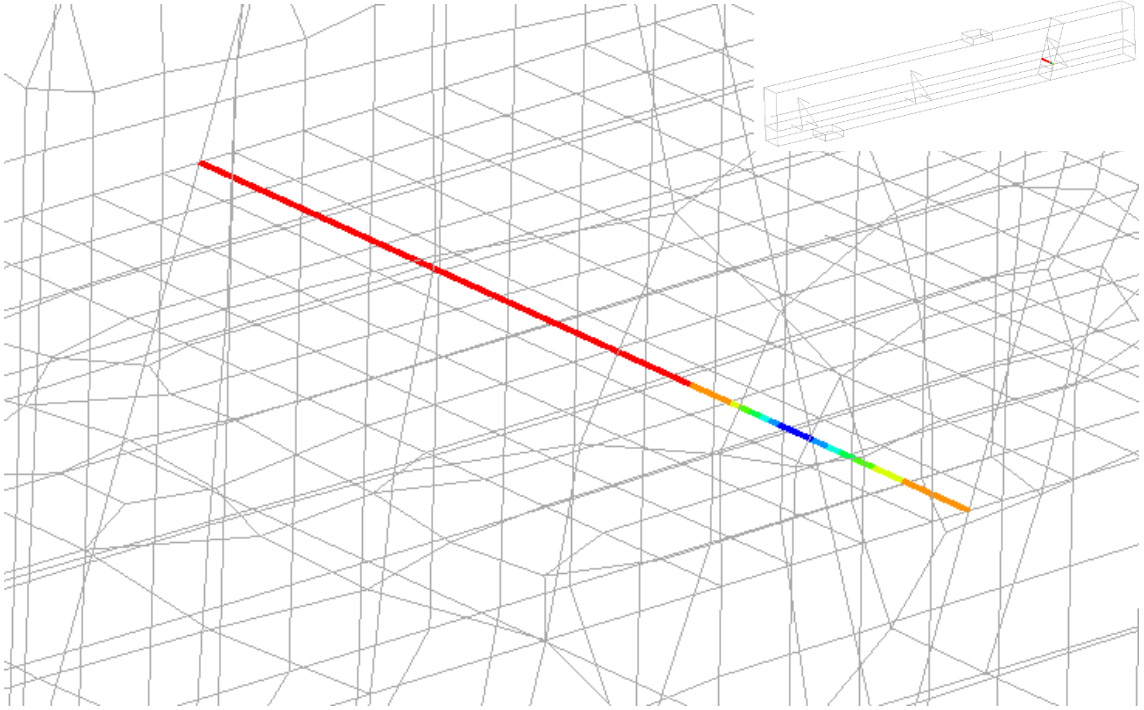


Figure 5.28: Interface relative displacement in Z-direction, T13 3D final DIANA model

This result can be further strengthened by a visualization of the crack-widths at the point where the lattice girder diagonals intersect with the interface. This is visualized in Figure 5.29 and Figure 5.30. Both these figures are cross-section planes taken over the normal axis of the lattice girder diagonal(s) showing the crack-widths. All the colored parts in the figures indicate cracked concrete. However, the blue color just indicates that the concrete is cracked, without any significant crack-width in contrast to the red color, which indicates a crack-width up to 0,61 mm.

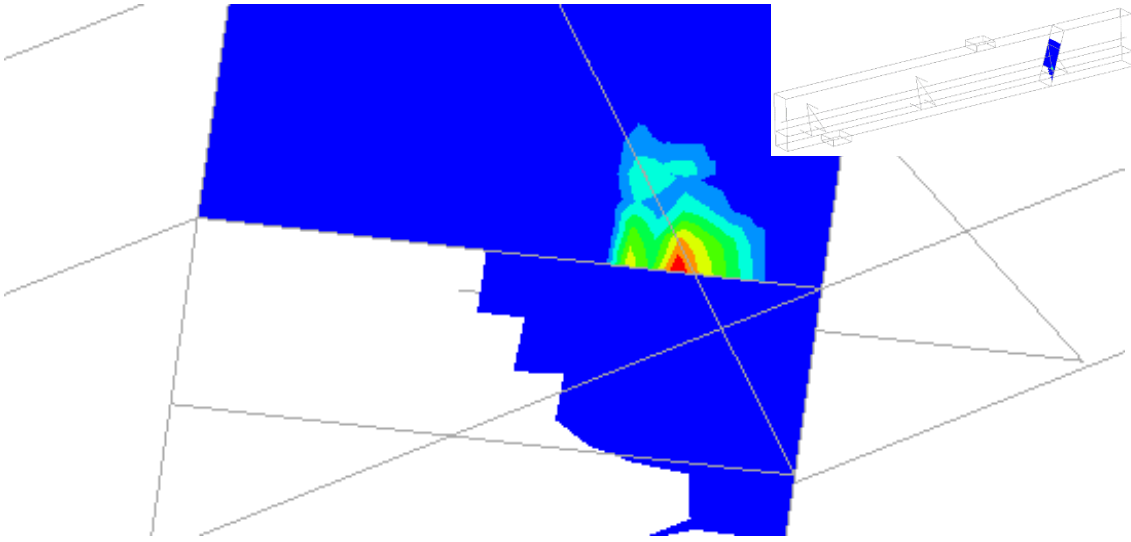


Figure 5.29: Crack-width in X-plane of the lattice girder diagonal, T13 3D final DIANA model

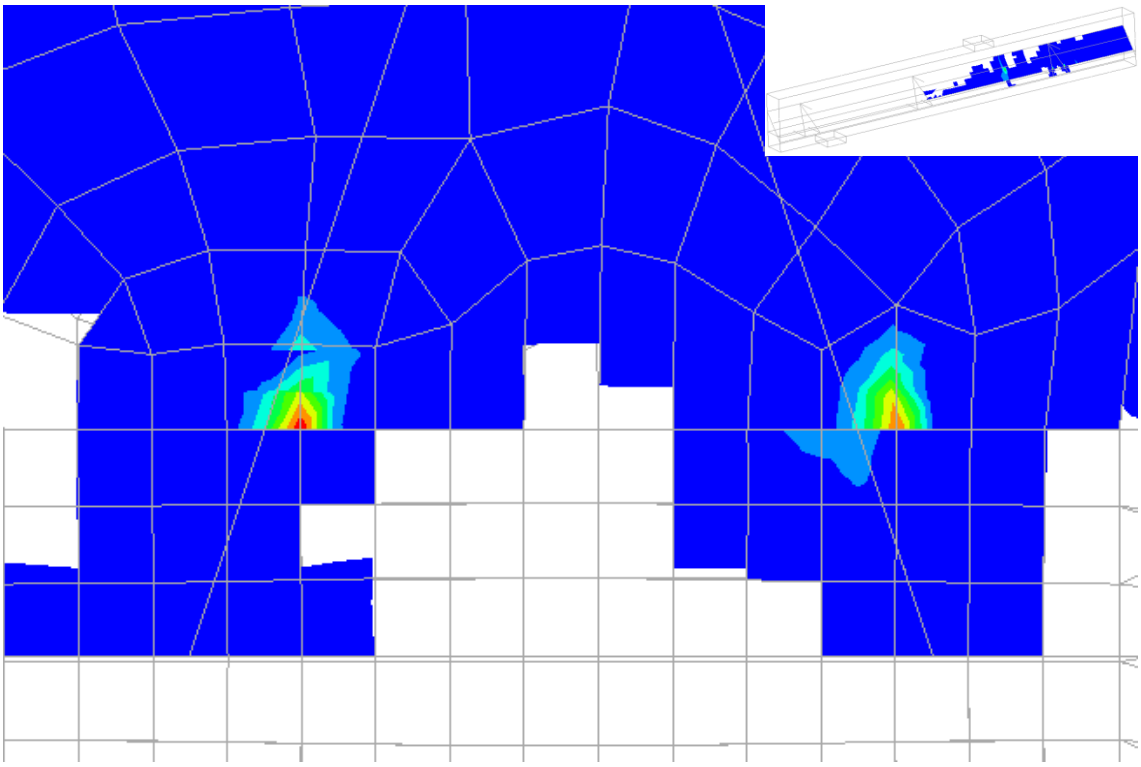


Figure 5.30: Crack-width in Y-plane of the lattice girder diagonals, T13 3D final DIANA model

As can be seen in Figure 5.29, the maximum crack-width corresponds with the minimum interface displacement in Figure 5.28. This behavior is also as expected. Figure 5.30 shows the maximum crack-widths at both the diagonals from the lattice girder. However, these maximum cracks do not appear at the exact location where the diagonals intersect with the interface. This can be explained again by the mesh size. By reducing the mesh size, this local problem could be solved.

It can be concluded that the intended failure mechanism indeed occurs. However, the degree to which this corresponds with the loads found in the experiments are not yet satisfactory. Due to the time limitations, the choice has been made to finalize this research with this model. However, in paragraph 6.2, some recommendations will be made for further research on this model. The script for this model is included in Appendix F.

6 CONCLUSIONS AND RECOMMENDATIONS

To be able to answer the question: “How can the influence of the pull-out and shear resistance of a lattice girder in a composite concrete floor, during failure, be determined with the use of a Finite Element Model?” the finite element software DIANA FEA 10.4 has been used. This software has been chosen as an alternative to the finite element software Abaqus, since this resulted in some problems during previous research. An attempt was made to create a realistic simulation of a 4-point bending test of a composite concrete floor element, as described in Wijte [1]. In this Chapter the conclusions of the results and recommendations for further research are provided.

6.1 Conclusions

6.1.1 Partial validation models

The individual validations of the different aspects of the concrete and reinforcement behavior show the capability of the used DIANA models. The similarity between the theoretical results and the DIANA results are very convincing.

Furthermore, when validating these models in 3D, the consequences of using small elements becomes a problem. To be able to make an accurate description of cracks in concrete, very small elements are desired, which reaches a limit very fast due to the limitations of the used computing power. This problem could easily be overcome, to a certain extent, by using a more powerful device. These consequences become even more prominent when moving to a full model.

However, it can be concluded that the partial DIANA models created for this research are, to a certain extent, capable of describing the theoretical behavior accurately.

6.1.2 Full 2D DIANA model

Creating a fully functional 2D model to simulate an experiment turned out to be a lot harder than just combining the validated partial models. When combined, there are so many influential parameters, of which some have unknown values. For instance the normal stiffness Kn used for the interface. Previous research leaves a margin from 1.000 N/mm^3 to 60.000 N/mm^3 for the normal stiffness of the interface in their models, while both models return promising results. Furthermore, the best fitted 2D results in this research are obtained with a normal stiffness of only 10 N/mm^3.

It can be concluded that the full 2D model could be capable of describing the load-deformation behavior of the critical detail in a composite concrete floor on a global scale. However, when more detailed information about the failure mechanism is required, more should be known about the behavior of the lattice girder and its surroundings. In this case a 2D model will not be sufficient.

6.1.3 3D anchorage models

The different anchorage models described in this research were not all successful. Two different lattice girder models were created. The first model used embedded reinforcement bars as the horizontal bars of the lattice girder and the second model used a line interface between the bar and the concrete. The lattice girder models initially showed promising results. However, the created headed anchor models, to validate the lattice girder models, were not successful. These models do not describe the expected failure mechanism, which is a kind of concrete cone failure, accurately. DIANA is capable of describing this concrete cone failure, but this was only achieved with a model which was not representative for the actual lattice girder properties. These concrete cone failure models reached their anchorage resistance purely from an increased bond stress between the reinforcement and the concrete.

It can be concluded that the created lattice girder models show promising results, but could not be validated. This leaves the question which of the models, the bond-slip model or the line interface model, results in the most accurate description of the actual pull-out of a lattice girder, or if a completely different approach would yield more accurate results.

6.1.4 Full 3D DIANA model

The 3D model describes the failure mechanism at the critical detail qualitatively accurately. However, quantitatively, the final calculation shows a significant underperformance in the resulting loads compared to the experimental results.

While the best performing lattice girder model, the bond-slip model, was implemented in the full model, as stated before, it is not yet known if this way of modeling the lattice girder in 3D yields the most accurate results. This might be an explanation for the underperformance of the overall model. However, this underperformance is probably a combination of various factors, of which the modelling of the lattice girder is just one factor.

Additionally, the experimental results show a full pull-out of the lattice girder, while the final model does not. The failure of the final model coincides with the cracks occurring on the bottom of the precast plate, long before the maximum experimental loads are reached. However, this could again be the result of the incorrect modelling of the lattice girder, or the modelling and the values of the parameters of the interface could be behaving too strong.

Besides the actual pull-out resistance of the lattice girder, the shear resistance of the embedded lattice girder is also of influence on the final results. However, this influence could not accurately be analyzed, since the lattice girder model could not be verified.

Lastly, the mesh size, especially surrounding the lattice girder, is too course to accurately describe the stresses and strains in detail. Furthermore, a smaller mesh may have led to more accurate crack formations and localizations, instead of the large blue areas, which indicate the cracked areas with small cracks strains, as presented in the figures in Chapter 5.3.

It can be concluded that qualitatively this final model is capable of describing the global failure mechanism accurately, especially with some more research. However, at its current full scale, quantitatively it is not capable of accurately describing the actual pull-out and shear resistance yet.

6.1.5 Overall conclusion

While this research intended to answer the question: “How can the influence of the pull out and shear resistance of a lattice girder in a composite concrete floor, during failure, be determined with the use of a Finite Element Model?”, it can be concluded that the final model within this research is not completely capable of answering this question alone.

The created finite element model is capable of simulating the failure mechanism of the experiments qualitatively accurately. Only the failure load found in the final calculation is a factor 2 smaller than in the experiments. More research is required to analyze if the chosen material parameters cause the difference in the results or if the chosen modeling approach, using bond-slip reinforcement, is limited to match the experimental results.

6.2 Recommendations

Based on the described results, a few recommendations can be made. Some of these recommendations might have been solved when more time was available for this research, others might be extensive enough for complete new graduation projects.

6.2.1 Improvements on the 2D model

Improving on the 2D models might have a lower priority, since the 3D model is capable of a much more accurate description of the actual experiments. However, there still might be a use for the 2D models concerning the global failure mechanism, since the 2D calculations take much less calculation time compared to their 3D equivalents.

The main improvements on the full 2D model could be made by improving on the actual pull-out and shear resistance of the lattice girders. This could be done by experimental research, in which a combination of tension and shear forces are exerted on the specimens or by first improving on the 3D models on a micro scale, after which the results of the 3D models could be included in the 2D model on a macro scale.

In addition, a parameter study should be conducted to the interface parameters, since realistic results could be achieved with very different interface parameters for 2D and 3D. However, with more time for this research, the 2D models could have already be improved upon with the found results of the 3D models.

6.2.2 Improvements on anchor models

To eventually be able to improve the full 3D model, first the anchorage models should be further researched. In the first place, the models of the single headed anchor should be improved upon. These models do not yet represent the real failure mechanism, which should occur by pulling out a headed anchor. Furthermore, the lattice girder models should be validated, or changed in case these models are not sufficient.

To start improving on the lattice girder models, the experiments executed by Bliek [14] could first be modelled. If this turns out to be successful, new experiments could be executed to also include the shear resistance of the lattice girder during failure. It would be advisable to limit a single research to just 1 or 2 different parameters, like the embedment depth of the lattice girder or the concrete quality. This would help to make the FEM models more manageable.

6.2.3 Improvements on the 3D model

Firstly, the full 3D model would greatly benefit from the proposed improvements, and hopefully, validations of the lattice girder models.

For the full 3D model a parameter study could also be executed. Such a research could for example just focus on the interface parameters. In this case the study would not directly focus on experimental data, but more on the specific influence of a certain parameter. For example: “how much does the tensile strength of the interface influence the overall strength of the model”. This research based the tensile strength of the interface on previous research to simulate a realistic result. A parameter study on the tensile strength of the interface, to get the influence of this parameter on the results, has not been performed.

Another improvement on the final model could be the size reduction of the mesh. Especially the mesh surrounding the lattice girder and the part of the interface which is influenced by the failure mechanism. By reducing this mesh size, a more realistic crack formation could be observed and the stress and strain gradients in the model would present more accurate information, which in turn could be used as a starting point for new experimental research as was originally intended for this research.

REFERENCES

[1] Wijte, S.N.M (2018), *Meetrapport 4-puntsbuigproeven voor het Betonhuis – Test breedplaatproefstukken*

[2] NEN-EN 1992-1-1:2005+A1:2015+NB:2016 Beton – Algemeen

[3] M.A.N. Hendriks, A. de Boer, B. Belletti (2017), *Guidelines for Nonlinear Finite Element Analysis of Concrete Structures, Rijkswaterstaat Centre for Infrastructure, Report RTD:1016-1:2017.*

[4] fédération internationale du béton (2010), *fib Model Code*

[5] K. Lundgren (2003), *Analysis of a lap splice in a lattice girder system*

[6] K. Lundgren, J. Helgesson, R. Sylvén (2005), *Joints in lattice girder structures*

[7] K. Lundgren (2007), *Lap splice over a grouted joint in a lattice girder system*

[8] TNO (2020): *DIANA Finite Element Analysis, User's Manual release 10.4*

[9] J.K. Weglarzy (2014), *Untersuchungen zum Tragstoß von nachträglich mit Ortbeton ergäntzen zweiachsig gespannten Stahlbeton-Fertigteilplatten*

[10] S.N.M. Wijte (2019), *Dossier 9780 Onderzoek constructieve veiligheid breedplaatvloeren in bestaande utiliteitsgebouwen*

[11] S.N.M. Wijte (2017), *Dossier 9663 Bezwijken parkeergarage Eindhoven airport*

[12] M. Verbaten, K. Riemens (2019), *Niet-lineaire analyse breedplaatvloeren, Cement 2019-02-04*

[13] L. Croes (2019), *Behavior of unreinforced concrete-to-concrete interfaces under shear loading*

[14] M.J.M. Bliek (2020), *Pull out resistance of a lattice girder from a precast floor plate*

[15] NEN-EN 1992-4:2018+NB:2019 – *7.2.1.4. Concrete cone failure*

APPENDICES

Appendix A – Partial 2D model scripts

- *A1 - Tensile behavior concrete*
  - *A1.1 – 1 element model*
  - *A1.2 – 100 element model*
- *A2 - Concrete to concrete interface*
  - *A2.1 - Interface in tension*
  - *A2.2 - Interface in shear*
- *A3 - Tensile behavior of the reinforcement*
- *A4 - Bond-slip of the reinforcement*

Appendix B – Full 2D model scripts

- *B1 – T13 V10*
- *B2 – T01 V11*
- *B3 – T19 V11*
- *B4 – T22 V11*
- *B5 – R1 V8*

Appendix C – V11 graphs

Appendix D – Partial 3D model scripts

- *D1 - Tensile behavior concrete*
  - *D1.1 – 1 element model*
  - *D1.2 – 1000 element model*
- *D2 - Concrete to concrete interface*
  - *D2.1 - Interface in tension*
  - *D2.2 - Interface in shear*
- *D3 - Bond-slip of the reinforcement*

Appendix E – Anchor model scripts

- *E1 - Single headed anchor model*
- *E2 - Vertical reinforcement model*
- *E3 - Lattice girder model bond-slip*
- *E4 - Lattice girder model line-interface*

Appendix F – Full 3D model script

A1.1 - 1 element model

```
newProject( "D:/Afstuderen/Python/Concrete/DIANA Output/Concrete 100x100", 10 )
setModelAnalysisAspects( [ "STRUCT" ] )
setModelDimension( "2D" )
setDefaultMeshOrder( "QUADRATIC" )
setDefaultMesherType( "HEXQUAD" )
setDefaultMidSideNodeLocation( "ONSHAP" )
setUnit( "LENGTH", "MM" )
setUnit( "FORCE", "N" )
setUnit( "ANGLE", "DEGREE" )

#Create Shapes
# createSheet( "Concrete", [[ 0,0,0 ],[ 400, 0, 0 ],[ 400, 300, 0 ],[200,300,0],[ 0, 300, 0 ]] )
# createSheet( "Concrete", [[ 0,0, 0 ],[ 400, 0, 0 ],[400,145,0],[390,145,0],[390,155,0],[400,155,0],[ 400, 300, 0 ],[200,300,0],[ 0, 300, 0 ],[0,155,0],[10,155,0],[10,145,0],[0,145,0]] )
# createSheet( "Concrete", [[ 0,0,0 ],[ 10, 0, 0 ],[10,4,0],[9,4,0],[9,6,0],[10,6,0],[ 10, 10, 0 ],[5,10,0],[0,10,0],[0,6,0],[1,6,0],[1,4,0],[0,4,0]] )
# createSheet( "Concrete", [[ 0,0,0 ],[ 100, 0, 0 ],[ 100, 100, 0 ],[50,100,0],[0,100,0]] )
createSheet( "Concrete", [[ 0,0,0 ],[ 100, 0, 0 ],[ 100, 100, 0 ],[0,100,0]] )

#Material Concrete
addMaterial( "Concrete", "CONCR", "TSCR", [] )
setParameter( "MATERIAL", "Concrete", "LINEAR/ELASTI/YOUNG", 32000 )
setParameter( "MATERIAL", "Concrete", "LINEAR/ELASTI/POISON", 0.2 )
setParameter( "MATERIAL", "Concrete", "LINEAR/MASS/DENSIT", 2.4e-09 )
setParameter( "MATERIAL", "Concrete", "MODTYP/TOTCRK", "ROTATE" )
setParameter( "MATERIAL", "Concrete", "TENSIL/TENCRV", "HORDYK" )
setParameter( "MATERIAL", "Concrete", "TENSIL/TENSTR", 2.9 )
setParameter( "MATERIAL", "Concrete", "TENSIL/GF1", 0.1405 )
setParameter( "MATERIAL", "Concrete", "TENSIL/RESTST", 0 )
setParameter( "MATERIAL", "Concrete", "TENSIL/CBSPEC", "ROTS" )
setParameter( "MATERIAL", "Concrete", "TENSIL/POISRE/POIRED", "DAMAGE" )
setParameter( "MATERIAL", "Concrete", "COMPRS/COMCRV", "MC2010" )
setParameter( "MATERIAL", "Concrete", "COMPRS/COMSTR", 38 )
setParameter( "MATERIAL", "Concrete", "COMPRS/EPSC1", 0.0023 )
setParameter( "MATERIAL", "Concrete", "COMPRS/EPSCU", 0.0035 )
setParameter( "MATERIAL", "Concrete", "COMPRS/CONFIN/CNFCRV", "NONE" )

addGeometry( "Block", "SHEET", "MEMBRA", [] )
setParameter( "GEOMET", "Block", "THICK", 100 )
setParameter( "GEOMET", "Block", "LOCAXS", True )
setParameter( "GEOMET", "Block", "LOCAXS/XAXIS", [ 1, 0, 0 ] )
addElementData( "Block" )

setElementClassType( "SHAPE", [ "Concrete" ], "MEMBRA" )
assignMaterial( "Concrete", "SHAPE", [ "Concrete" ] )
assignGeometry( "Block", "SHAPE", [ "Concrete" ] )
assignElementData( "Block", "SHAPE", [ "Concrete" ] )
# setParameter( "DATA", "Block", "./INTERF", [] )
# setParameter( "DATA", "Block", "INTERF", "BEAM" )

#Assign Supports
addSet( "GEOMETRYSUPPORTSET", "Support" )

createLineSupport( "Top Support", "Support" )
setParameter( "GEOMETRYSUPPORT", "Top Support", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Top Support", "TRANSL", [ 0, 1, 0 ] )
setParameter( "GEOMETRYSUPPORT", "Top Support", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Top Support", "Concrete", [[ 50,100,0 ]] )
# attach( "GEOMETRYSUPPORT", "Top Support", "Concrete", [[ 25, 100, 0 ],[75,100,0]] )

# createPointSupport( "Topp Support", "Support" )
# setParameter( "GEOMETRYSUPPORT", "Topp Support", "AXES", [ 1, 2 ] )
# setParameter( "GEOMETRYSUPPORT", "Topp Support", "TRANSL", [ 1, 0, 0 ] )
# setParameter( "GEOMETRYSUPPORT", "Topp Support", "ROTATI", [ 0, 0, 0 ] )
# attach( "GEOMETRYSUPPORT", "Topp Support", "Concrete", [[ 50, 100, 0 ]] )

createLineSupport( "Bot Support", "Support" )
```

```
setParameter( "GEOMETRYSUPPORT", "Bot Support", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Bot Support", "TRANSL", [ 0, 1, 0 ] )
setParameter( "GEOMETRYSUPPORT", "Bot Support", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Bot Support", "Concrete", [[ 50, 0, 0 ]] )

createPointSupport( "Topp Support", "Support" )
setParameter( "GEOMETRYSUPPORT", "Topp Support", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Topp Support", "TRANSL", [ 1, 0, 0 ] )
setParameter( "GEOMETRYSUPPORT", "Topp Support", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Topp Support", "Concrete", [[ 100, 100, 0 ]] )

createPointSupport( "Bott Support", "Support" )
setParameter( "GEOMETRYSUPPORT", "Bott Support", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Bott Support", "TRANSL", [ 1, 0, 0 ] )
setParameter( "GEOMETRYSUPPORT", "Bott Support", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Bott Support", "Concrete", [[ 100, 0, 0 ]] )

#Assign Loads
addSet( "GEOMETRYLOADSET", "Load" )

createLineLoad( "Load", "Load" )
setParameter( "GEOMETRYLOAD", "Load", "LODTYP", "DEFORM" )
setParameter( "GEOMETRYLOAD", "Load", "DEFORM/SUPP", "Bot Support" )
setParameter( "GEOMETRYLOAD", "Load", "DEFORM/TR/VALUE", -0.24 )
setParameter( "GEOMETRYLOAD", "Load", "DEFORM/TR/DIRECT", 2 )
attach( "GEOMETRYLOAD", "Load", "Concrete", [[ 50, 0, 0 ]] )

# createLineLoad( "Load", "Load" )
# setParameter( "GEOMETRYLOAD", "Load", "FORCE/VALUE", -2 )
# setParameter( "GEOMETRYLOAD", "Load", "FORCE/DIRECT", 2 )
# attach( "GEOMETRYLOAD", "Load", "Concrete", [[ 200, 0, 0 ]] )


#Set mesh properties
setElementSize( [ "Concrete" ], 100, -1, False )
setMesherType( [ "Concrete" ], "HEXQUAD" )
clearMidSideNodeLocation( [  "Concrete" ] )
generateMesh( [] )
hideView( "GEOM" )
showView( "MESH" )

#Analysis
addAnalysis( "Analysis1" )
addAnalysisCommand( "Analysis1", "NONLIN", "Structural nonlinear" )
renameAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "EXECUT(1)",
"Load" )
# setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/STEPS/STEPTY", "AUTOMA" )
# setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/STEPS/AUTOMA/SIZES", 1 )
# setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/STEPS/AUTOMA/MAXSIZ", 0.05 )
# setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/STEPS/AUTOMA/MINSIZ", 0.005 )
# setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/STEPS/AUTOMA/CUTBCK", 0.25 )
setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/STEPS/EXPLIC/SIZES", "0.01(100)" )
setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN", False )
setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN/REGULA/SET(1)/NODES(1)/RNGNRS", "ALL" )
setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN/REGULA/SET(1)/DIRECT", 2 )

setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/ITERAT/MAXITE", 40 )
addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/ITERAT/LINESE" )
setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/ITERAT/LINESE", True )
```

```
124 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/LINESE/ETAMIN", 0.1 )
125 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/LINESE/ETAMAX", 1 )
126 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/ENERGY" )
127 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/ENERGY", True )
128 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/DISPLA", False )
129 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/ENERGY/TOLCON", 0.01 )
130 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/ENERGY/NOCONV", "CONTIN" )
131 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/FORCE/NOCONV", "CONTIN" )
132 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/METHOD/METNAM", "SECANT" )
133
134 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "OUTPUT(1)/SELTYP",
    "USER" )
135
136 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/DISPLA(1)/TOTAL/TRANSL/GLOBAL" )
137 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(1)/TOTAL/GREEN/LOCAL/LOCATI", "INTPNT" )
138 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(1)/TOTAL/CAUCHY/LOCAL/LOCATI", "INTPNT" )
139
140 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(1)/TOTAL/GREEN/GLOBAL" )
141 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(2)/TOTAL/GREEN/PRINCI" )
142 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(4)/CRACK/GREEN" )
143 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(7)/CRKWDT/GREEN/GLOBAL" )
144 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(8)/CRKWDT/GREEN/PRINCI" )
145 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(3)/CRACK/CAUCHY/LOCAL" )
146 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(2)/TOTAL/CAUCHY/PRINCI" )
147
148
149 #Run Analysis
150 runSolver( [] )
151 showView( "RESULT" )
152 fitAll(  )
```

A1.2 - 100 element model

```
newProject( "D:/Afstuderen/Python/Concrete/DIANA Output/Concrete 1000x1000", 10 )
setModelAnalysisAspects( [ "STRUCT" ] )
setModelDimension( "2D" )
setDefaultMeshOrder( "QUADRATIC" )
setDefaultMesherType( "HEXQUAD" )
setDefaultMidSideNodeLocation( "ONSHAP" )
setUnit( "LENGTH", "MM" )
setUnit( "FORCE", "N" )
setUnit( "ANGLE", "DEGREE" )

#Create Shapes
createSheet( "Concrete", [[ 0,0,0 ],[500,0,0],[ 1000, 0, 0 ],[ 1000, 1000, 0 ],[500,1000,0],[0,1000,0]] )

#Material Concrete

addMaterial( "Concrete", "CONCR", "TSCR", [] )
setParameter( "MATERIAL", "Concrete", "LINEAR/ELASTI/YOUNG", 32000 )
setParameter( "MATERIAL", "Concrete", "LINEAR/ELASTI/POISON", 0.2 )
setParameter( "MATERIAL", "Concrete", "LINEAR/MASS/DENSIT", 2.4e-09 )
setParameter( "MATERIAL", "Concrete", "MODTYP/TOTCRK", "ROTATE" )
setParameter( "MATERIAL", "Concrete", "TENSIL/TENCRV", "HORDYK" )
setParameter( "MATERIAL", "Concrete", "TENSIL/TENSTR", 2.9 )
setParameter( "MATERIAL", "Concrete", "TENSIL/GF1", 0.1405 )
setParameter( "MATERIAL", "Concrete", "TENSIL/RESTST", 0 )
setParameter( "MATERIAL", "Concrete", "TENSIL/CBSPEC", "ROTS" )
setParameter( "MATERIAL", "Concrete", "TENSIL/POISRE/POIRED", "DAMAGE" )
setParameter( "MATERIAL", "Concrete", "COMPRS/COMCRV", "MC2010" )
setParameter( "MATERIAL", "Concrete", "COMPRS/COMSTR", 38 )
setParameter( "MATERIAL", "Concrete", "COMPRS/EPSC1", 0.0023 )
setParameter( "MATERIAL", "Concrete", "COMPRS/EPSCU", 0.0035 )
setParameter( "MATERIAL", "Concrete", "COMPRS/CONFIN/CNFCRV", "NONE" )

addGeometry( "Block", "SHEET", "MEMBRA", [] )
setParameter( "GEOMET", "Block", "THICK", 1000 )
setParameter( "GEOMET", "Block", "LOCAXS", True )
setParameter( "GEOMET", "Block", "LOCAXS/XAXIS", [ 1, 0, 0 ] )
addElementData( "Block" )

setElementClassType( "SHAPE", [ "Concrete" ], "MEMBRA" )
assignMaterial( "Concrete", "SHAPE", [ "Concrete" ] )
assignGeometry( "Block", "SHAPE", [ "Concrete" ] )
assignElementData( "Block", "SHAPE", [ "Concrete" ] )
setParameter( "DATA", "Block", "./INTERF", [] )
setParameter( "DATA", "Block", "INTERF", "BEAM" )

#Assign Supports
addSet( "GEOMETRYSUPPORTSET", "Support" )

#Line supports
createLineSupport( "Top Support", "Support" )
setParameter( "GEOMETRYSUPPORT", "Top Support", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Top Support", "TRANSL", [ 0, 1, 0 ] )
setParameter( "GEOMETRYSUPPORT", "Top Support", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Top Support", "Concrete", [[ 250, 1000, 0 ],[750,1000,0]] )

createLineSupport( "Bot Support", "Support" )
setParameter( "GEOMETRYSUPPORT", "Bot Support", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Bot Support", "TRANSL", [ 0, 1, 0 ] )
setParameter( "GEOMETRYSUPPORT", "Bot Support", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Bot Support", "Concrete", [[ 250, 0, 0 ],[750,0,0]] )

#Point supports
createPointSupport( "Topp Support", "Support" )
setParameter( "GEOMETRYSUPPORT", "Topp Support", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Topp Support", "TRANSL", [ 1, 0, 0 ] )
setParameter( "GEOMETRYSUPPORT", "Topp Support", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Topp Support", "Concrete", [[ 500, 1000, 0 ]] )

createPointSupport( "Bott Support", "Support" )
```

```
70  setParameter( "GEOMETRYSUPPORT", "Bott Support", "AXES", [ 1, 2 ] )
71  setParameter( "GEOMETRYSUPPORT", "Bott Support", "TRANSL", [ 1, 0, 0 ] )
72  setParameter( "GEOMETRYSUPPORT", "Bott Support", "ROTATI", [ 0, 0, 0 ] )
73  attach( "GEOMETRYSUPPORT", "Bott Support", "Concrete", [[ 500, 0, 0 ]] )
74
75  #Assign Loads
76  addSet( "GEOMETRYLOADSET", "Load" )
77
78  createLineLoad( "Load", "Load" )
79  setParameter( "GEOMETRYLOAD", "Load", "LODTYP", "DEFORM" )
80  setParameter( "GEOMETRYLOAD", "Load", "DEFORM/SUPP", "Bot Support" )
81  setParameter( "GEOMETRYLOAD", "Load", "DEFORM/TR/VALUE", -2.4 )
82  setParameter( "GEOMETRYLOAD", "Load", "DEFORM/TR/DIRECT", 2 )
83  attach( "GEOMETRYLOAD", "Load", "Concrete", [[ 250, 0, 0 ],[750,0,0]] )
84
85  # createLineLoad( "Load", "Load" )
86  # setParameter( "GEOMETRYLOAD", "Load", "FORCE/VALUE", -2 )
87  # setParameter( "GEOMETRYLOAD", "Load", "FORCE/DIRECT", 2 )
88  # attach( "GEOMETRYLOAD", "Load", "Concrete", [[ 200, 0, 0 ]] )
89
90  #Set mesh properties
91  setElementSize( [ "Concrete" ], 100, -1, False )
92  setMesherType( [ "Concrete" ], "HEXQUAD" )
93  clearMidSideNodeLocation( [  "Concrete" ] )
94  generateMesh( [] )
95  hideView( "GEOM" )
96  showView( "MESH" )
97
98  #Analysis
99  addAnalysis( "Analysis1" )
100 addAnalysisCommand( "Analysis1", "NONLIN", "Structural nonlinear" )
101 renameAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "EXECUT(1)",
    "Load" )
102 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/STEPTY", "AUTOMA" )
103 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/SIZES", 1 )
104 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/MAXSIZ", 0.05 )
105 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/MINSIZ", 0.005 )
106 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/CUTBCK", 0.25 )
107 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/SIZES", "0.01(100)" )
108 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN", False )
109 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN/REGULA/SET(1)/NODES(1)/RNGNRS", "ALL" )
110 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN/REGULA/SET(1)/DIRECT", 2 )
111
112 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/MAXITE", 30 )
113 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/LINESE" )
114 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/LINESE", True )
115 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/LINESE/ETAMIN", 0.1 )
116 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/LINESE/ETAMAX", 1 )
117 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/ENERGY" )
118 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/ENERGY", True )
119 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/DISPLA", False )
120 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/ENERGY/TOLCON", 0.01 )
121 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
```

```
    "EXECUT(1)/ITERAT/CONVER/ENERGY/NOCONV", "CONTIN" )
122 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/FORCE/NOCONV", "CONTIN" )
123 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/METHOD/METNAM", "SECANT" )
124
125 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "OUTPUT(1)/SELTYP",
    "USER" )
126
127 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/DISPLA(1)/TOTAL/TRANSL/GLOBAL" )
128 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(1)/TOTAL/GREEN/LOCAL/LOCATI", "INTPNT" )
129 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(1)/TOTAL/CAUCHY/LOCAL/LOCATI", "INTPNT" )
130
131 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(1)/TOTAL/GREEN/GLOBAL" )
132 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(2)/TOTAL/GREEN/PRINCI" )
133 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(4)/CRACK/GREEN" )
134 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(7)/CRKWDT/GREEN/GLOBAL" )
135 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(8)/CRKWDT/GREEN/PRINCI" )
136 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(3)/CRACK/CAUCHY/LOCAL" )
137 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(2)/TOTAL/CAUCHY/PRINCI" )
138
139 #Run Analysis
140 runSolver( [] )
141 showView( "RESULT" )
142 fitAll(  )
143
```

A2.1 - Interface in tension

```
newProject( "D:/Afstuderen/Python/Interface/DIANA Output/Interface vertical load Coulomb friction", 10 )
setModelAnalysisAspects( [ "STRUCT" ] )
setModelDimension( "2D" )
setDefaultMeshOrder( "QUADRATIC" )
setDefaultMesherType( "HEXQUAD" )
setDefaultMidSideNodeLocation( "ONSHAP" )
setUnit( "LENGTH", "MM" )
setUnit( "FORCE", "N" )
setUnit( "ANGLE", "DEGREE" )

#Create Shapes
createSheet( "Pressure Layer", [[ 0, 70, 0 ],[ 400, 70, 0 ],[ 400, 300, 0 ],[200,300,0],[ 0, 300, 0 ]] )
createSheet( "Plate", [[ 0, 0, 0 ],[ 200, 0, 0 ],[ 400, 0, 0 ],[ 400, 70, 0 ],[ 0, 70, 0 ]] )

#Material Concrete

addMaterial( "Concrete", "CONCR", "TSCR", [] )
setParameter( "MATERIAL", "Concrete", "LINEAR/ELASTI/YOUNG", 32000 )
setParameter( "MATERIAL", "Concrete", "LINEAR/ELASTI/POISON", 0.2 )
setParameter( "MATERIAL", "Concrete", "LINEAR/MASS/DENSIT", 2.4e-09 )
setParameter( "MATERIAL", "Concrete", "MODTYP/TOTCRK", "ROTATE" )
setParameter( "MATERIAL", "Concrete", "TENSIL/TENCRV", "HORDYK" )
setParameter( "MATERIAL", "Concrete", "TENSIL/TENSTR", 2.9 )
setParameter( "MATERIAL", "Concrete", "TENSIL/GF1", 0.1405 )
setParameter( "MATERIAL", "Concrete", "TENSIL/CBSPEC", "ROTS" )
# setParameter( "MATERIAL", "Concrete", "TENSIL/CRACKB", 10 )
setParameter( "MATERIAL", "Concrete", "TENSIL/POISRE/POIRED", "DAMAGE" )
setParameter( "MATERIAL", "Concrete", "COMPRS/COMCRV", "MC2010" )
setParameter( "MATERIAL", "Concrete", "COMPRS/COMSTR", 38 )
setParameter( "MATERIAL", "Concrete", "COMPRS/EPSC1", 0.0023 )
setParameter( "MATERIAL", "Concrete", "COMPRS/EPSCU", 0.0035 )
setParameter( "MATERIAL", "Concrete", "COMPRS/CONFIN/CNFCRV", "NONE" )

addGeometry( "Plate", "SHEET", "MEMBRA", [] )
setParameter( "GEOMET", "Plate", "THICK", 200 )
setParameter( "GEOMET", "Plate", "LOCAXS", True )
setParameter( "GEOMET", "Plate", "LOCAXS/XAXIS", [ 1, 0, 0 ] )
addElementData( "Plate" )

setElementClassType( "SHAPE", [ "Pressure Layer","Plate" ], "MEMBRA" )
assignMaterial( "Concrete", "SHAPE", [ "Pressure Layer","Plate" ] )
assignGeometry( "Plate", "SHAPE", [ "Pressure Layer","Plate" ] )
assignElementData( "Plate", "SHAPE", [ "Pressure Layer","Plate" ] )
setParameter( "DATA", "Plate", "./INTERF", [] )
setParameter( "DATA", "Plate", "INTERF", "BEAM" )

#Material Interface Concrete-Concrete
addMaterial( "Interface Concrete-Concrete", "INTERF", "FRICTI", [] )
setParameter( "MATERIAL", "Interface Concrete-Concrete", "LINEAR/IFTYP", "LIN2D" )
setParameter( "MATERIAL", "Interface Concrete-Concrete", "COULOM/OPNTYP", "GAP" )
setParameter( "MATERIAL", "Interface Concrete-Concrete", "COULOM/GAPVAL", 0.5 )
setParameter( "MATERIAL", "Interface Concrete-Concrete", "COULOM/COHESI", 0.5 )
setParameter( "MATERIAL", "Interface Concrete-Concrete", "COULOM/PHI", 31 )
setParameter( "MATERIAL", "Interface Concrete-Concrete", "COULOM/PSI", 31 )
setParameter( "MATERIAL", "Interface Concrete-Concrete", "LINEAR/ELAS2/DSNY", 60000 )
setParameter( "MATERIAL", "Interface Concrete-Concrete", "LINEAR/ELAS2/DSSX", 6000 )

addGeometry( "Interface", "LINE", "STLIIF", [] )
setParameter( "GEOMET", "Interface", "LIFMEM/THICK", 200 )
addElementData( "Interface Concrete-Concrete" )
createConnection( "Interface Concrete-Concrete", "INTER", "SHAPEEDGE", "SHAPEEDGE" )
setParameter( "GEOMETRYCONNECTION", "Interface Concrete-Concrete", "MODE", "CLOSED" )
setElementClassType( "GEOMETRYCONNECTION", "Interface Concrete-Concrete", "STLIIF" )
assignMaterial( "Interface Concrete-Concrete", "GEOMETRYCONNECTION", "Interface Concrete-Concrete" )
assignGeometry( "Interface", "GEOMETRYCONNECTION", "Interface Concrete-Concrete" )
assignElementData( "Interface Concrete-Concrete", "GEOMETRYCONNECTION", "Interface Concrete-Concrete" )
```

```
67  setParameter( "GEOMETRYCONNECTION", "Interface Concrete-Concrete", "FLIP", False )
68  attachTo( "GEOMETRYCONNECTION", "Interface Concrete-Concrete", "SOURCE", "Pressure
    Layer", [[ 200, 70, 0 ]] )
69  attachTo( "GEOMETRYCONNECTION", "Interface Concrete-Concrete", "TARGET", "Plate", [[
    200, 70, 0 ]] )
70
71  setParameter( "DATA", "Interface Concrete-Concrete", "./INTEGR", [] )
72  setParameter( "DATA", "Interface Concrete-Concrete", "INTEGR", "HIGH" )
73
74  #Assign Supports
75  addSet( "GEOMETRYSUPPORTSET", "Support" )
76
77  createLineSupport( "Top Support", "Support" )
78  setParameter( "GEOMETRYSUPPORT", "Top Support", "AXES", [ 1, 2 ] )
79  setParameter( "GEOMETRYSUPPORT", "Top Support", "TRANSL", [ 0, 1, 0 ] )
80  setParameter( "GEOMETRYSUPPORT", "Top Support", "ROTATI", [ 0, 0, 0 ] )
81  attach( "GEOMETRYSUPPORT", "Top Support", "Pressure Layer", [[ 100, 300, 0
    ],[300,300,0]] )
82
83  createPointSupport( "Topp Support", "Support" )
84  setParameter( "GEOMETRYSUPPORT", "Topp Support", "AXES", [ 1, 2 ] )
85  setParameter( "GEOMETRYSUPPORT", "Topp Support", "TRANSL", [ 1, 0, 0 ] )
86  setParameter( "GEOMETRYSUPPORT", "Topp Support", "ROTATI", [ 0, 0, 0 ] )
87  attach( "GEOMETRYSUPPORT", "Topp Support", "Pressure Layer", [[ 200, 300, 0 ]] )
88
89  createLineSupport( "Bot Support", "Support" )
90  setParameter( "GEOMETRYSUPPORT", "Bot Support", "AXES", [ 1, 2 ] )
91  setParameter( "GEOMETRYSUPPORT", "Bot Support", "TRANSL", [ 0, 1, 0 ] )
92  setParameter( "GEOMETRYSUPPORT", "Bot Support", "ROTATI", [ 0, 0, 0 ] )
93  attach( "GEOMETRYSUPPORT", "Bot Support", "Plate", [[ 100, 0, 0 ],[300,0,0]] )
94
95  createPointSupport( "Botp Support", "Support" )
96  setParameter( "GEOMETRYSUPPORT", "Botp Support", "AXES", [ 1, 2 ] )
97  setParameter( "GEOMETRYSUPPORT", "Botp Support", "TRANSL", [ 1, 0, 0 ] )
98  setParameter( "GEOMETRYSUPPORT", "Botp Support", "ROTATI", [ 0, 0, 0 ] )
99  attach( "GEOMETRYSUPPORT", "Botp Support", "Plate", [[ 200, 0, 0 ]] )
100
101 #Assign Loads
102 addSet( "GEOMETRYLOADSET", "Load" )
103
104 createLineLoad( "Load", "Load" )
105 setParameter( "GEOMETRYLOAD", "Load", "LODTYP", "DEFORM" )
106 setParameter( "GEOMETRYLOAD", "Load", "DEFORM/SUPP", "Bot Support" )
107 setParameter( "GEOMETRYLOAD", "Load", "DEFORM/TR/VALUE", -0.005 )
108 setParameter( "GEOMETRYLOAD", "Load", "DEFORM/TR/DIRECT", 2 )
109 attach( "GEOMETRYLOAD", "Load", "Plate", [[ 100, 0, 0 ],[300,0,0]] )
110
111 #Set mesh properties
112 setElementSize( [ "Pressure Layer","Plate" ], 10, -1, True )
113 setMesherType( [ "Pressure Layer","Plate" ], "HEXQUAD" )
114 clearMidSideNodeLocation( [  "Pressure Layer","Plate" ] )
115 generateMesh( [] )
116 hideView( "GEOM" )
117 showView( "MESH" )
118
119 #Analysis
120 addAnalysis( "Analysis1" )
121 addAnalysisCommand( "Analysis1", "NONLIN", "Structural nonlinear" )
122 renameAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "EXECUT(1)",
    "Load" )
123 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/STEPTY", "AUTOMA" )
124 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/SIZES", 1 )
125 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/MAXSIZ", 0.005 )
126 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/MINSIZ", 0.0005 )
127 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/CUTBCK", 0.25 )
128 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
```

```
    "EXECUT(1)/LOAD/STEPS/EXPLIC/SIZES", "1.0000(46),0.1000(20)" )
129 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN", False )
130 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN/REGULA/SET(1)/NODES(1)/RNGNRS", "ALL" )
131 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN/REGULA/SET(1)/DIRECT", 2 )
132
133 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/MAXITE", 25 )
134 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/LINESE" )
135 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/LINESE", True )
136 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/LINESE/ETAMIN", 0.1 )
137 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/LINESE/ETAMAX", 1 )
138 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/ENERGY" )
139 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/ENERGY", True )
140 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/DISPLA", False )
141 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/ENERGY/TOLCON", 0.001 )
142 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/ENERGY/NOCONV", "CONTIN" )
143 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/FORCE/NOCONV", "CONTIN" )
144 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/METHOD/METNAM", "SECANT" )
145 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "OUTPUT(1)/SELTYP",
    "USER" )
146
147 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/DISPLA(1)/TOTAL/TRANSL/GLOBAL" )
148 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(1)/TOTAL/GREEN/GLOBAL" )
149 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(2)/TOTAL/GREEN/PRINCI" )
150 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(2)/TOTAL/GREEN/PRINCI" )
151 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(4)/CRACK/GREEN" )
152 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(7)/CRKWDT/GREEN/GLOBAL" )
153 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(8)/CRKWDT/GREEN/PRINCI" )
154 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(3)/CRACK/CAUCHY/LOCAL" )
155 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(2)/TOTAL/CAUCHY/PRINCI" )
156 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(4)/TOTAL/CAUCHY/GLOBAL" )
157
158 #Run Analysis
159 runSolver( [] )
160 showView( "RESULT" )
161 fitAll(  )
```

A2.2 - Interface in shear

```
newProject( "D:/Afstuderen/Python/Interface/DIANA Output/Interface horizontal load Plate Deformation", 10 )
setModelAnalysisAspects( [ "STRUCT" ] )
setModelDimension( "2D" )
setDefaultMeshOrder( "QUADRATIC" )
setDefaultMesherType( "HEXQUAD" )
setDefaultMidSideNodeLocation( "ONSHAP" )
setUnit( "LENGTH", "MM" )
setUnit( "FORCE", "N" )
setUnit( "ANGLE", "DEGREE" )

#Create Shapes
createSheet( "Pressure Layer", [[ 0, 70, 0 ],[200, 70, 0],[ 400, 70, 0 ],[ 400, 300, 0 ],[200,300,0],[ 0, 300, 0 ]] )
createSheet( "Plate", [[ 0, 0, 0 ],[ 200, 0, 0 ],[ 400, 0, 0 ],[ 400, 70, 0 ],[200, 70, 0],[ 0, 70, 0 ]] )

#Create Composed Line
createLine ( "Composed Line Horizontal", [0,150,0],[400,150,0] )
createLine ( "Composed Line Vertical", [200,0,0],[200,300,0] )

#Material Concrete Linear
addMaterial( "Concrete", "CONCR", "LEI", [] )
setParameter( "MATERIAL", "Concrete", "LINEAR/ELASTI/YOUNG", 32000 )
setParameter( "MATERIAL", "Concrete", "LINEAR/ELASTI/POISON", 0.2 )
setParameter( "MATERIAL", "Concrete", "LINEAR/MASS/DENSIT", 2.4e-09 )

#Material Concrete Non Linear
# addMaterial( "Concrete", "CONCR", "TSCR", [] )
# setParameter( "MATERIAL", "Concrete", "LINEAR/ELASTI/YOUNG", 32000 )
# setParameter( "MATERIAL", "Concrete", "LINEAR/ELASTI/POISON", 0.2 )
# setParameter( "MATERIAL", "Concrete", "LINEAR/MASS/DENSIT", 2.4e-09 )
# setParameter( "MATERIAL", "Concrete", "MODTYP/TOTCRK", "ROTATE" )
# setParameter( "MATERIAL", "Concrete", "TENSIL/TENCRV", "HORDYK" )
# setParameter( "MATERIAL", "Concrete", "TENSIL/TENSTR", 2.9 )
# setParameter( "MATERIAL", "Concrete", "TENSIL/GF1", 0.1405 )
# setParameter( "MATERIAL", "Concrete", "TENSIL/CBSPEC", "ROTS" )
# setParameter( "MATERIAL", "Concrete", "TENSIL/POISRE/POIRED", "DAMAGE" )
# setParameter( "MATERIAL", "Concrete", "COMPRS/COMCRV", "MC2010" )
# setParameter( "MATERIAL", "Concrete", "COMPRS/COMSTR", 38 )
# setParameter( "MATERIAL", "Concrete", "COMPRS/EPSC1", 0.0023 )
# setParameter( "MATERIAL", "Concrete", "COMPRS/EPSCU", 0.0035 )
# setParameter( "MATERIAL", "Concrete", "COMPRS/CONFIN/CNFCRV", "NONE" )

addGeometry( "Plate", "SHEET", "MEMBRA", [] )
setParameter( "GEOMET", "Plate", "THICK", 100 )
setParameter( "GEOMET", "Plate", "LOCAXS", True )
setParameter( "GEOMET", "Plate", "LOCAXS/XAXIS", [ 1, 0, 0 ] )
addElementData( "Plate" )

setElementClassType( "SHAPE", [ "Pressure Layer","Plate" ], "MEMBRA" )
assignMaterial( "Concrete", "SHAPE", [ "Pressure Layer","Plate" ] )
assignGeometry( "Plate", "SHAPE", [ "Pressure Layer","Plate" ] )
assignElementData( "Plate", "SHAPE", [ "Pressure Layer","Plate" ] )
setParameter( "DATA", "Plate", "./INTERF", [] )
setParameter( "DATA", "Plate", "INTERF", "BEAM" )

#Material Interface Concrete-Concrete
addMaterial( "Interface Concrete-Concrete", "INTERF", "FRICTI", [] )
setParameter( "MATERIAL", "Interface Concrete-Concrete", "LINEAR/IFTYP", "LIN2D" )
setParameter( "MATERIAL", "Interface Concrete-Concrete", "COULOM/OPNTYP", "GAP" )
setParameter( "MATERIAL", "Interface Concrete-Concrete", "COULOM/GAPVAL", 0.5 )
setParameter( "MATERIAL", "Interface Concrete-Concrete", "COULOM/COHESI", 0.5 )
setParameter( "MATERIAL", "Interface Concrete-Concrete", "COULOM/PHI", 31 )
setParameter( "MATERIAL", "Interface Concrete-Concrete", "COULOM/PSI", 31 )
setParameter( "MATERIAL", "Interface Concrete-Concrete", "LINEAR/ELAS2/DSNY", 60000 )
setParameter( "MATERIAL", "Interface Concrete-Concrete", "LINEAR/ELAS2/DSSX", 6000 )
setParameter( "MATERIAL", "Interface Concrete-Concrete", "COULOM/FRCHSR/MODE2", 0 )

addGeometry( "Interface", "LINE", "STLIIF", [] )
setParameter( "GEOMET", "Interface", "LIFMEM/THICK", 100 )
```

```
addElementData( "Interface Concrete-Concrete" )
createConnection( "Interface Concrete-Concrete", "INTER", "SHAPEEDGE", "SHAPEEDGE" )
setParameter( "GEOMETRYCONNECTION", "Interface Concrete-Concrete", "MODE", "CLOSED" )
setElementClassType( "GEOMETRYCONNECTION", "Interface Concrete-Concrete", "STLIIF" )
assignMaterial( "Interface Concrete-Concrete", "GEOMETRYCONNECTION", "Interface Concrete-Concrete" )
assignGeometry( "Interface", "GEOMETRYCONNECTION", "Interface Concrete-Concrete" )
assignElementData( "Interface Concrete-Concrete", "GEOMETRYCONNECTION", "Interface Concrete-Concrete" )
setParameter( "GEOMETRYCONNECTION", "Interface Concrete-Concrete", "FLIP", False )
attachTo( "GEOMETRYCONNECTION", "Interface Concrete-Concrete", "SOURCE", "Pressure Layer", [[ 100, 70, 0 ]] )
attachTo( "GEOMETRYCONNECTION", "Interface Concrete-Concrete", "TARGET", "Plate", [[ 100, 70, 0 ]] )
attachTo( "GEOMETRYCONNECTION", "Interface Concrete-Concrete", "SOURCE", "Pressure Layer", [[ 300, 70, 0 ]] )
attachTo( "GEOMETRYCONNECTION", "Interface Concrete-Concrete", "TARGET", "Plate", [[ 300, 70, 0 ]] )

setParameter( "DATA", "Interface Concrete-Concrete", "./INTEGR", [] )
setParameter( "DATA", "Interface Concrete-Concrete", "INTEGR", "HIGH" )

#Composed Line
addGeometry( "Composed Line", "LINE", "COMLIN", [] )
setParameter( "GEOMET", "Composed Line", "DISTAN/THICK", 100 )
setParameter( "GEOMET", "Composed Line", "LOCAXS", True )
setElementClassType( "SHAPE", [ "Composed Line Horizontal","Composed Line Vertical" ], "COMLIN" )
assignGeometry( "Composed Line", "SHAPE", [ "Composed Line Horizontal","Composed Line Vertical" ] )

#Assign Supports
addSet( "GEOMETRYSUPPORTSET", "Support" )

# createLineSupport( "Top Support", "Support" )
# setParameter( "GEOMETRYSUPPORT", "Top Support", "AXES", [ 1, 2 ] )
# setParameter( "GEOMETRYSUPPORT", "Top Support", "TRANSL", [ 0, 1, 0 ] )
# setParameter( "GEOMETRYSUPPORT", "Top Support", "ROTATI", [ 0, 0, 0 ] )
# attach( "GEOMETRYSUPPORT", "Top Support", "Pressure Layer", [[ 100, 300, 0 ],[300,300,0]] )

# createLineSupport( "Left Support", "Support" )
# setParameter( "GEOMETRYSUPPORT", "Left Support", "AXES", [ 1, 2 ] )
# setParameter( "GEOMETRYSUPPORT", "Left Support", "TRANSL", [ 1, 0, 0 ] )
# setParameter( "GEOMETRYSUPPORT", "Left Support", "ROTATI", [ 0, 0, 0 ] )
# attach( "GEOMETRYSUPPORT", "Left Support", "Pressure Layer", [[ 0, 185, 0 ]] )

# createPointSupport( "Topp Support", "Support" )
# setParameter( "GEOMETRYSUPPORT", "Topp Support", "AXES", [ 1, 2 ] )
# setParameter( "GEOMETRYSUPPORT", "Topp Support", "TRANSL", [ 1, 0, 0 ] )
# setParameter( "GEOMETRYSUPPORT", "Topp Support", "ROTATI", [ 0, 0, 0 ] )
# attach( "GEOMETRYSUPPORT", "Topp Support", "Pressure Layer", [[ 200, 300, 0 ]] )

# createLineSupport( "Right Support", "Support" )
# setParameter( "GEOMETRYSUPPORT", "Right Support", "AXES", [ 1, 2 ] )
# setParameter( "GEOMETRYSUPPORT", "Right Support", "TRANSL", [ 1, 0, 0 ] )
# setParameter( "GEOMETRYSUPPORT", "Right Support", "ROTATI", [ 0, 0, 0 ] )
# attach( "GEOMETRYSUPPORT", "Right Support", "Pressure Layer", [[ 400, 185, 0 ]] )
# attach( "GEOMETRYSUPPORT", "Right Support", "Plate", [[ 400, 35, 0 ]] )

createBodySupport( "Top Support", "Support" )
setParameter( "GEOMETRYSUPPORT", "Top Support", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Top Support", "TRANSL", [ 0, 1, 0 ] )
setParameter( "GEOMETRYSUPPORT", "Top Support", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Top Support", [ "Pressure Layer" ] )

createBodySupport( "Horizontal Support", "Support" )
setParameter( "GEOMETRYSUPPORT", "Horizontal Support", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Horizontal Support", "TRANSL", [ 1, 0, 0 ] )
setParameter( "GEOMETRYSUPPORT", "Horizontal Support", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Horizontal Support", [ "Plate","Pressure Layer" ] )
```

```
131
132  # createLineSupport( "Bot Support", "Support" )
133  # setParameter( "GEOMETRYSUPPORT", "Bot Support", "AXES", [ 1, 2 ] )
134  # setParameter( "GEOMETRYSUPPORT", "Bot Support", "TRANSL", [ 0, 1, 0 ] )
135  # setParameter( "GEOMETRYSUPPORT", "Bot Support", "ROTATI", [ 0, 0, 0 ] )
136  # attach( "GEOMETRYSUPPORT", "Bot Support", "Plate", [[ 100, 0, 0 ],[300,0,0]] )
137
138  #Assign Loads
139  addSet( "GEOMETRYLOADSET", "Prestress" )
140  addSet( "GEOMETRYLOADSET", "Deformation" )
141
142  #Deformations
143  # createLineLoad( "Deform", "Deformation" )
144  # setParameter( "GEOMETRYLOAD", "Deform", "LODTYP", "DEFORM" )
145  # setParameter( "GEOMETRYLOAD", "Deform", "DEFORM/SUPP", "Interface Support" )
146  # setParameter( "GEOMETRYLOAD", "Deform", "DEFORM/TR/VALUE", 0.1 )
147  # setParameter( "GEOMETRYLOAD", "Deform", "DEFORM/TR/DIRECT", 1 )
148  # # attach( "GEOMETRYLOAD", "Deform", "Plate", [[ 0,35,0],[400,35,0]] )
149  # # attach( "GEOMETRYLOAD", "Deform", "Plate", [[400,35,0]] )
150  # attach( "GEOMETRYLOAD", "Deform", "Plate", [[ 0,35,0]] )
151
152  createBodyLoad( "Deform", "Deformation" )
153  setParameter( "GEOMETRYLOAD", "Deform", "LODTYP", "DEFORM" )
154  setParameter( "GEOMETRYLOAD", "Deform", "DEFORM/TR/VALUE", 0.0012 )
155  attach( "GEOMETRYLOAD", "Deform", [ "Plate" ] )
156
157  # createLineLoad( "Prestress", "Load" )
158  # setParameter( "GEOMETRYLOAD", "Prestress", "LODTYP", "DEFORM" )
159  # setParameter( "GEOMETRYLOAD", "Prestress", "DEFORM/SUPP", "Interface Support" )
160  # setParameter( "GEOMETRYLOAD", "Prestress", "DEFORM/TR/VALUE", 0.01 )
161  # setParameter( "GEOMETRYLOAD", "Prestress", "DEFORM/TR/DIRECT", 2 )
162  # attach( "GEOMETRYLOAD", "Prestress", "Plate", [[ 100,0,0],[300,0,0]] )
163
164  #Forces
165  createLineLoad( "Up", "Prestress" )
166  setParameter( "GEOMETRYLOAD", "Up", "FORCE/VALUE", 1000 )
167  setParameter( "GEOMETRYLOAD", "Up", "FORCE/DIRECT", 2 )
168  attach( "GEOMETRYLOAD", "Up", "Plate", [[ 100, 0, 0 ],[ 300, 0, 0 ]] )
169
170  # createLineLoad( "Horizontal", "Prestress" )
171  # setParameter( "GEOMETRYLOAD", "Horizontal", "FORCE/VALUE", 400 )
172  # setParameter( "GEOMETRYLOAD", "Horizontal", "FORCE/DIRECT", 1 )
173  # attach( "GEOMETRYLOAD", "Horizontal", "Plate", [[ 0, 35, 0 ]] )
174
175  #Set mesh properties
176  setElementSize( [ "Pressure Layer","Plate" ], 10, -1, True )
177  setMesherType( [ "Pressure Layer","Plate" ], "HEXQUAD" )
178  clearMidSideNodeLocation( [  "Pressure Layer","Plate" ] )
179  generateMesh( [] )
180  hideView( "GEOM" )
181  showView( "MESH" )
182
183  #Analysis
184  addAnalysis( "Analysis1" )
185  addAnalysisCommand( "Analysis1", "NONLIN", "Structural nonlinear" )
186  renameAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "EXECUT(1)",
     "Prestress" )
187
188  setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
     "EXECUT(1)/LOAD/LOADNR", 1 )
189  setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
     "EXECUT(1)/LOAD/STEPS/STEPTY", "AUTOMA" )
190  setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
     "EXECUT(1)/LOAD/STEPS/AUTOMA/SIZES", 1 )
191  setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
     "EXECUT(1)/LOAD/STEPS/AUTOMA/MAXSIZ", 0.1 )
192  setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
     "EXECUT(1)/LOAD/STEPS/AUTOMA/MINSIZ", 0.0001 )
193  setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
     "EXECUT(1)/LOAD/STEPS/AUTOMA/CUTBCK", 0.25 )
194  # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
```

```
    "EXECUT(1)/LOAD/STEPS/AUTOMA/MAXSTP", 20 )
195
196 renameAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "EXECUT(2)",
    "Deformation" )
197
198 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(2)/LOAD/LOADNR", 2 )
199 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(2)/LOAD/STEPS/STEPTY", "AUTOMA" )
200 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(2)/LOAD/STEPS/AUTOMA/SIZES", 1 )
201 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(2)/LOAD/STEPS/AUTOMA/MAXSIZ", 0.01 )
202 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(2)/LOAD/STEPS/AUTOMA/MINSIZ", 0.001 )
203 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(2)/LOAD/STEPS/AUTOMA/CUTBCK", 0.25 )
204 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(2)/LOAD/STEPS/AUTOMA/MAXSTP", 500 )
205
206 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "OUTPUT(1)/SELTYP",
    "USER" )
207
208 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/DISPLA(2)/TOTAL/TRANSL/GLOBAL" )
209 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(1)/TOTAL/GREEN/GLOBAL" )
210 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(2)/EFFECT/CAUCHY/GLOBAL" )
211 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(3)/EFFECT/TRACTI/FRICTI" )
212 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(1)/TOTAL/CAUCHY/GLOBAL" )
213 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(4)/EFFECT/TRACTI/LOCAL" )
214 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(5)/TOTAL/SHEAR/LOCAL" )
215 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/METHOD/METNAM", "SECANT" )
216
217 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/SIZES", "0.0001(3000)" )
218 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN", False )
219 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN/REGULA/SET(1)/NODES(1)/RNGNRS", "ALL" )
220 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN/REGULA/SET(1)/DIRECT", 2 )
221
222 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/STEPTY", "AUTOMA" )
223 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/SIZES", 100 )
224 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/MAXSIZ", 0.1 )
225 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/MINSIZ", 0.00001 )
226 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/CUTBCK", 0.25 )
227 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/MAXITE", 25 )
228
229 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/MAXITE", 25 )
230 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/LINESE" )
231 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/LINESE", True )
232 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/LINESE/ETAMIN", 0.1 )
233 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
```

```
    "EXECUT(1)/ITERAT/LINESE/ETAMAX", 1 )
234 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/ENERGY" )
235 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/ENERGY", True )
236 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/DISPLA", False )
237 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/ENERGY/TOLCON", 0.001 )
238 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/ENERGY/NOCONV", "CONTIN" )
239 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/FORCE/NOCONV", "CONTIN" )
240 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "OUTPUT(1)/SELTYP",
    "USER" )
241
242 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/DISPLA(1)/TOTAL/TRANSL/GLOBAL" )
243 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(1)/TOTAL/GREEN/GLOBAL" )
244 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(2)/TOTAL/GREEN/PRINCI" )
245 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(2)/TOTAL/GREEN/PRINCI" )
246 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(4)/CRACK/GREEN" )
247 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(7)/CRKWDT/GREEN/GLOBAL" )
248 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(8)/CRKWDT/GREEN/PRINCI" )
249 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(3)/CRACK/CAUCHY/LOCAL" )
250 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(2)/TOTAL/CAUCHY/PRINCI" )
251 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(4)/TOTAL/CAUCHY/GLOBAL" )
252
253 #Run Analysis
254 # runSolver( [] )
255 # showView( "RESULT" )
256 fitAll(  )
257 setViewSettingValue( "view setting", "RESULT/CONTOU/EQUIDI/NUMBER", 10 )
```

A3 - Tensile behavior of the reinforcement

```
newProject( "D:/Afstuderen/Python/Reinforcement/DIANA Output/Single Reinforcement", 10 )
setModelAnalysisAspects( [ "STRUCT" ] )
setModelDimension( "2D" )
setDefaultMeshOrder( "QUADRATIC" )
setDefaultMesherType( "HEXQUAD" )
setDefaultMidSideNodeLocation( "ONSHAP" )
setUnit( "LENGTH", "MM" )
setUnit( "FORCE", "N" )
setUnit( "ANGLE", "DEGREE" )

#Create Shapes
createSheet( "Pressure Layer", [[ 0, 70, 0 ],[ 400, 70, 0 ],[ 400, 300, 0 ],[200,300,0],[ 0, 300, 0 ]] )
createSheet( "Plate", [[ 0, 0, 0 ],[ 200, 0, 0 ],[ 400, 0, 0 ],[ 400, 70, 0 ],[ 0, 70, 0 ]] )

fitAll(  )

#Create Reinforcement
createLine( "Bar_M8", [ 200, 35, 0],[ 200, 265, 0] )
# createLine( "Bar_M8", [ 160, 35, 0],[ 240, 265, 0] )

#Regroup Material Parts
addSet( "SHAPESET", "Reinforcement" )
moveToShapeSet( [ "Bar_M8" ], "Reinforcement" )
rename( "SHAPESET", "Shapes", "Concrete" )

#Material Concrete
addMaterial( "Concrete", "CONCR", "TSCR", [] )
setParameter( "MATERIAL", "Concrete", "LINEAR/ELASTI/YOUNG", 32000 )
setParameter( "MATERIAL", "Concrete", "LINEAR/ELASTI/POISON", 0.2 )
setParameter( "MATERIAL", "Concrete", "LINEAR/MASS/DENSIT", 2.4e-09 )
setParameter( "MATERIAL", "Concrete", "MODTYP/TOTCRK", "ROTATE" )
setParameter( "MATERIAL", "Concrete", "TENSIL/TENCRV", "HORDYK" )
setParameter( "MATERIAL", "Concrete", "TENSIL/TENSTR", 2.9 )
setParameter( "MATERIAL", "Concrete", "TENSIL/GF1", 0.1405 )
setParameter( "MATERIAL", "Concrete", "TENSIL/CBSPEC", "ROTS" )
setParameter( "MATERIAL", "Concrete", "TENSIL/POISRE/POIRED", "DAMAGE" )
setParameter( "MATERIAL", "Concrete", "COMPRS/COMCRV", "MC2010" )
setParameter( "MATERIAL", "Concrete", "COMPRS/COMSTR", 38 )
setParameter( "MATERIAL", "Concrete", "COMPRS/EPSC1", 0.0023 )
setParameter( "MATERIAL", "Concrete", "COMPRS/EPSCU", 0.0035 )
setParameter( "MATERIAL", "Concrete", "COMPRS/CONFIN/CNFCRV", "NONE" )

addGeometry( "Plate", "SHEET", "MEMBRA", [] )
setParameter( "GEOMET", "Plate", "THICK", 200 )
setParameter( "GEOMET", "Plate", "LOCAXS", True )
setParameter( "GEOMET", "Plate", "LOCAXS/XAXIS", [ 1, 0, 0 ] )
addElementData( "Plate" )

setElementClassType( "SHAPE", [ "Pressure Layer","Plate" ], "MEMBRA" )
assignMaterial( "Concrete", "SHAPE", [ "Pressure Layer","Plate" ] )
assignGeometry( "Plate", "SHAPE", [ "Pressure Layer","Plate" ] )
assignElementData( "Plate", "SHAPE", [ "Pressure Layer","Plate" ] )
setParameter( "DATA", "Plate", "./INTERF", [] )
setParameter( "DATA", "Plate", "INTERF", "BEAM" )

#Material Reinfo_M8
addMaterial( "Reinfo_M8", "REINFO", "REBOND", [ "ANCHOR" ] )
setParameter( "MATERIAL", "Reinfo_M8", "REBARS/ELASTI/YOUNG", 200000 )
setParameter( "MATERIAL", "Reinfo_M8", "REBARS/POISON/POISON", 0.3 )
setParameter( "MATERIAL", "Reinfo_M8", "REBARS/PLATYP", "VMISES" )
setParameter( "MATERIAL", "Reinfo_M8", "REBARS/PLASTI/TRESSH", "KAPSIG" )
# setParameter( "MATERIAL", "Reinfo_M8", "REBARS/PLASTI/KAPSIG", [ 0.00275, 550, 0.05, 594 ] )
setParameter( "MATERIAL", "Reinfo_M8", "REBARS/PLASTI/KAPSIG", [ 0, 550, 0.04725, 594 ] )
setParameter( "MATERIAL", "Reinfo_M8", "RESLIP/SHFTYP", "BONDS6" )
setParameter( "MATERIAL", "Reinfo_M8", "RESLIP/DSNY", 1000 )
setParameter( "MATERIAL", "Reinfo_M8", "RESLIP/DSSX", 100 )
```

```
# setParameter( "MATERIAL", "Reinfo_M8", "RESLIP/BONDS6/SLPVAL", [ 7.706, 3.082, 0.1, 1.8, 3.6, 10, 0.4 ] )
setParameter( "MATERIAL", "Reinfo_M8", "RESLIP/BONDS6/SLPVAL", [ 0.63, 0.63, 0.01, 0.02, 0.03, 0.04, 0.5 ] )
# setParameter( "MATERIAL", "Reinfo_M8", "RESLIP/BONDS4/SLPVAL", 38 )
# setParameter( "MATERIAL", "Reinfo_M8", "RESLIP/BONDS4/DIAMET", 8 )
# setParameter( "MATERIAL", "Reinfo_M8", "RESLIP/BONDS4/TAUFAC", 1 )
setParameter( "MATERIAL", "Reinfo_M8", "REANCH/DTIPNX", 10000 )
setParameter( "MATERIAL", "Reinfo_M8", "REANCH/DTIPSY", 10000 )
setParameter( "MATERIAL", "Reinfo_M8", "REANCH/TIPLOC", "BOTH" )
addGeometry( "Reinfo_M8", "RELINE", "REBAR", [] )
setParameter( "GEOMET", "Reinfo_M8", "REITYP", "REITRU" )
setParameter( "GEOMET", "Reinfo_M8", "REITYP", "CIRBEA" )
setParameter( "GEOMET", "Reinfo_M8", "CIRBEA/CIRCLE", 8 )
setParameter( "GEOMET", "Reinfo_M8", "TIPELM/SURFAC", 50.3 )
assignMaterial( "Reinfo_M8", "SHAPE", [ "Bar_M8" ] )
assignGeometry( "Reinfo_M8", "SHAPE", [ "Bar_M8" ] )
convertToReinforcement( [ "Bar_M8" ] )
addElementData( "Bar" )
assignElementData( "Bar", "SHAPE", [ "Bar_M8" ] )
setParameter( "DATA", "Bar", "./INTERF", [] )
setParameter( "DATA", "Bar", "INTERF", "BEAM" )
setReinforcementDiscretization( [ "Bar_M8" ], "ELEMENT" )
setContinuousInInterfaces( [ "Bar_M8" ], True )

#Material Interface Concrete-Concrete
addMaterial( "Interface Concrete-Concrete", "INTERF", "FRICTI", [] )
setParameter( "MATERIAL", "Interface Concrete-Concrete", "LINEAR/IFTYP", "LIN2D" )
setParameter( "MATERIAL", "Interface Concrete-Concrete", "COULOM/OPNTYP", "GAP" )
setParameter( "MATERIAL", "Interface Concrete-Concrete", "COULOM/GAPVAL", 0.5 )
setParameter( "MATERIAL", "Interface Concrete-Concrete", "COULOM/COHESI", 0.5 )
setParameter( "MATERIAL", "Interface Concrete-Concrete", "COULOM/PHI", 31 )
setParameter( "MATERIAL", "Interface Concrete-Concrete", "COULOM/PSI", [] )
setMaterialFunction( "Interface Concrete-Concrete", "COULOM/PSI", "" )
setParameter( "MATERIAL", "Interface Concrete-Concrete", "COULOM/PSI", 31 )
setParameter( "MATERIAL", "Interface Concrete-Concrete", "LINEAR/ELAS2/DSNY", 60000 )
setParameter( "MATERIAL", "Interface Concrete-Concrete", "LINEAR/ELAS2/DSSX", 6000 )

addGeometry( "Interface", "LINE", "STLIIF", [] )
setParameter( "GEOMET", "Interface", "LIFMEM/THICK", 200 )
addElementData( "Interface Concrete-Concrete" )
createConnection( "Interface Concrete-Concrete", "INTER", "SHAPEEDGE", "SHAPEEDGE" )
setParameter( "GEOMETRYCONNECTION", "Interface Concrete-Concrete", "MODE", "CLOSED" )
setElementClassType( "GEOMETRYCONNECTION", "Interface Concrete-Concrete", "STLIIF" )
assignMaterial( "Interface Concrete-Concrete", "GEOMETRYCONNECTION", "Interface Concrete-Concrete" )
assignGeometry( "Interface", "GEOMETRYCONNECTION", "Interface Concrete-Concrete" )
assignElementData( "Interface Concrete-Concrete", "GEOMETRYCONNECTION", "Interface Concrete-Concrete" )
setParameter( "GEOMETRYCONNECTION", "Interface Concrete-Concrete", "FLIP", False )
attachTo( "GEOMETRYCONNECTION", "Interface Concrete-Concrete", "SOURCE", "Pressure Layer", [[ 200, 70, 0 ]] )
attachTo( "GEOMETRYCONNECTION", "Interface Concrete-Concrete", "TARGET", "Plate", [[ 200, 70, 0 ]] )

setParameter( "DATA", "Interface Concrete-Concrete", "./INTEGR", [] )
setParameter( "DATA", "Interface Concrete-Concrete", "INTEGR", "HIGH" )

#Assign Supports
addSet( "GEOMETRYSUPPORTSET", "Support" )

createLineSupport( "Top Support", "Support" )
setParameter( "GEOMETRYSUPPORT", "Top Support", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Top Support", "TRANSL", [ 0, 1, 0 ] )
setParameter( "GEOMETRYSUPPORT", "Top Support", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Top Support", "Pressure Layer", [[ 100, 300, 0 ],[300,300,0]] )

createPointSupport( "Topp Support", "Support" )
setParameter( "GEOMETRYSUPPORT", "Topp Support", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Topp Support", "TRANSL", [ 1, 0, 0 ] )
```

```
131 setParameter( "GEOMETRYSUPPORT", "Topp Support", "ROTATI", [ 0, 0, 0 ] )
132 attach( "GEOMETRYSUPPORT", "Topp Support", "Pressure Layer", [[ 200, 300, 0 ]] )
133
134 createLineSupport( "Bot Support", "Support" )
135 setParameter( "GEOMETRYSUPPORT", "Bot Support", "AXES", [ 1, 2 ] )
136 setParameter( "GEOMETRYSUPPORT", "Bot Support", "TRANSL", [ 0, 1, 0 ] )
137 setParameter( "GEOMETRYSUPPORT", "Bot Support", "ROTATI", [ 0, 0, 0 ] )
138 attach( "GEOMETRYSUPPORT", "Bot Support", "Plate", [[ 100, 0, 0 ],[300,0,0]] )
139
140 createPointSupport( "Botp Support", "Support" )
141 setParameter( "GEOMETRYSUPPORT", "Botp Support", "AXES", [ 1, 2 ] )
142 setParameter( "GEOMETRYSUPPORT", "Botp Support", "TRANSL", [ 1, 0, 0 ] )
143 setParameter( "GEOMETRYSUPPORT", "Botp Support", "ROTATI", [ 0, 0, 0 ] )
144 attach( "GEOMETRYSUPPORT", "Botp Support", "Plate", [[ 200, 0, 0 ]] )
145
146 #Assign Loads
147 addSet( "GEOMETRYLOADSET", "Load" )
148
149 createLineLoad( "Load", "Load" )
150 setParameter( "GEOMETRYLOAD", "Load", "LODTYP", "DEFORM" )
151 setParameter( "GEOMETRYLOAD", "Load", "DEFORM/SUPP", "Bot Support" )
152 setParameter( "GEOMETRYLOAD", "Load", "DEFORM/TR/VALUE", -0.1 )
153 setParameter( "GEOMETRYLOAD", "Load", "DEFORM/TR/DIRECT", 2 )
154 attach( "GEOMETRYLOAD", "Load", "Plate", [[ 100, 0, 0 ],[300,0,0]] )
155
156 #Set mesh properties
157 setElementSize( [ "Pressure Layer","Plate","Bar_M8" ], 10, -1, True )
158 setMesherType( [ "Pressure Layer","Plate","Bar_M8" ], "HEXQUAD" )
159 clearMidSideNodeLocation( [  "Pressure Layer","Plate","Bar_M8" ] )
160 generateMesh( [] )
161 hideView( "GEOM" )
162 showView( "MESH" )
163
164 #Analysis
165 addAnalysis( "Analysis1" )
166 addAnalysisCommand( "Analysis1", "NONLIN", "Structural nonlinear" )
167 renameAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "EXECUT(1)",
    "Load" )
168 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/SIZES", "0.10000(150)" )
169 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN", False )
170 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN/REGULA/SET(1)/NODES(1)/RNGNRS", "ALL" )
171 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN/REGULA/SET(1)/DIRECT", 2 )
172
173 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/MAXITE", 25 )
174 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/LINESE" )
175 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/LINESE", True )
176 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/LINESE/ETAMIN", 0.1 )
177 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/LINESE/ETAMAX", 1 )
178 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/ENERGY" )
179 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/ENERGY", True )
180 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/DISPLA", False )
181 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/ENERGY/TOLCON", 0.001 )
182 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/ENERGY/NOCONV", "CONTIN" )
183 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/FORCE/NOCONV", "CONTIN" )
184 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/METHOD/METNAM", "SECANT" )
```

```
185
186 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "OUTPUT(1)/SELTYP",
    "USER" )
187
188 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/DISPLA(1)/TOTAL/TRANSL/GLOBAL" )
189 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(1)/TOTAL/GREEN/LOCAL" )
190 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(1)/TOTAL/CAUCHY/LOCAL" )
191
192 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(1)/TOTAL/GREEN/GLOBAL" )
193 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(2)/TOTAL/GREEN/PRINCI" )
194 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(4)/CRACK/GREEN" )
195 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(7)/CRKWDT/GREEN/GLOBAL" )
196 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(8)/CRKWDT/GREEN/PRINCI" )
197 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(3)/CRACK/CAUCHY/LOCAL" )
198 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(2)/TOTAL/CAUCHY/PRINCI" )
199
200 #Run Analysis
201 runSolver( [] )
202 showView( "RESULT" )
203
```

A4 - Bond-slip of the reinforcement

```
newProject( "D:/Afstuderen/Python/Reinforcement/DIANA Output/Pull out Reinforcement", 10 )
setModelAnalysisAspects( [ "STRUCT" ] )
setModelDimension( "2D" )
setDefaultMeshOrder( "QUADRATIC" )
setDefaultMesherType( "HEXQUAD" )
setDefaultMidSideNodeLocation( "ONSHAP" )
setUnit( "LENGTH", "MM" )
setUnit( "FORCE", "N" )
setUnit( "ANGLE", "DEGREE" )

#Create Shapes
createSheet( "Pressure Layer", [[ 0, 70, 0 ],[ 400, 70, 0 ],[ 400, 300, 0 ],[200,300,0],[ 0, 300, 0 ]] )

#Create Reinforcement
createLine("Bar_M8", [ 200, 70, 0],[ 200, 265, 0] )

#Regroup Material Parts
addSet( "SHAPESET", "Reinforcement" )
moveToShapeSet( [ "Bar_M8" ], "Reinforcement" )
rename( "SHAPESET", "Shapes", "Concrete" )

#Material Concrete
addMaterial( "Concrete", "CONCR", "TSCR", [] )
setParameter( "MATERIAL", "Concrete", "LINEAR/ELASTI/YOUNG", 32000 )
setParameter( "MATERIAL", "Concrete", "LINEAR/ELASTI/POISON", 0.2 )
setParameter( "MATERIAL", "Concrete", "LINEAR/MASS/DENSIT", 2.4e-09 )
setParameter( "MATERIAL", "Concrete", "MODTYP/TOTCRK", "ROTATE" )
setParameter( "MATERIAL", "Concrete", "TENSIL/TENCRV", "HORDYK" )
setParameter( "MATERIAL", "Concrete", "TENSIL/TENSTR", 2.9 )
setParameter( "MATERIAL", "Concrete", "TENSIL/GF1", 0.1405 )
setParameter( "MATERIAL", "Concrete", "TENSIL/CBSPEC", "ROTS" )
setParameter( "MATERIAL", "Concrete", "TENSIL/POISRE/POIRED", "DAMAGE" )
setParameter( "MATERIAL", "Concrete", "COMPRS/COMCRV", "MC2010" )
setParameter( "MATERIAL", "Concrete", "COMPRS/COMSTR", 38 )
setParameter( "MATERIAL", "Concrete", "COMPRS/EPSC1", 0.0023 )
setParameter( "MATERIAL", "Concrete", "COMPRS/EPSCU", 0.0035 )
setParameter( "MATERIAL", "Concrete", "COMPRS/CONFIN/CNFCRV", "NONE" )

addGeometry( "Plate", "SHEET", "MEMBRA", [] )
setParameter( "GEOMET", "Plate", "THICK", 200 )
setParameter( "GEOMET", "Plate", "LOCAXS", True )
setParameter( "GEOMET", "Plate", "LOCAXS/XAXIS", [ 1, 0, 0 ] )
addElementData( "Plate" )

setElementClassType( "SHAPE", [ "Pressure Layer" ], "MEMBRA" )
assignMaterial( "Concrete", "SHAPE", [ "Pressure Layer"] )
assignGeometry( "Plate", "SHAPE", [ "Pressure Layer" ] )
assignElementData( "Plate", "SHAPE", [ "Pressure Layer" ] )
setParameter( "DATA", "Plate", "./INTERF", [] )
setParameter( "DATA", "Plate", "INTERF", "BEAM" )

#Material Reinfo_M8
# addMaterial( "Reinfo_M8", "REINFO", "REBOND", [ "ANCHOR" ] )
addMaterial( "Reinfo_M8", "REINFO", "REBOND", [] )
setParameter( "MATERIAL", "Reinfo_M8", "REBARS/ELASTI/YOUNG", 200000 )
setParameter( "MATERIAL", "Reinfo_M8", "REBARS/POISON/POISON", 0.3 )
setParameter( "MATERIAL", "Reinfo_M8", "REBARS/PLATYP", "VMISES" )
setParameter( "MATERIAL", "Reinfo_M8", "REBARS/PLASTI/TRESSH", "KAPSIG" )
setParameter( "MATERIAL", "Reinfo_M8", "REBARS/PLASTI/KAPSIG", [ 0, 550, 0.04725, 594 ] )
setParameter( "MATERIAL", "Reinfo_M8", "RESLIP/SHFTYP", "BONDS6" )
setParameter( "MATERIAL", "Reinfo_M8", "RESLIP/DSNY", 1000 )
setParameter( "MATERIAL", "Reinfo_M8", "RESLIP/DSSX", 100 )
# Plain surface good bond
setParameter( "MATERIAL", "Reinfo_M8", "RESLIP/BONDS6/SLPVAL", [ 0.63, 0.63, 0.01, 0.02, 0.03, 0.04, 0.5 ] )
# Good bond condition
# setParameter( "MATERIAL", "Reinfo_M8", "RESLIP/BONDS6/SLPVAL", [ 15.411, 6.164, 0.1, 1, 2, 10, 0.4 ] )
```

```
67  # Other bond conditions
68  # setParameter( "MATERIAL", "Reinfo_M8", "RESLIP/BONDS6/SLPVAL", [ 7.706, 3.082,
    0.1, 1.8, 3.6, 10, 0.4 ] )
69  # setParameter( "MATERIAL", "Reinfo_M8", "REANCH/DTIPNX", 10000 )
70  # setParameter( "MATERIAL", "Reinfo_M8", "REANCH/DTIPSY", 10000 )
71  # setParameter( "MATERIAL", "Reinfo_M8", "REANCH/TIPLOC", "END" )
72
73  addGeometry( "Reinfo_M8", "RELINE", "REBAR", [] )
74  setParameter( "GEOMET", "Reinfo_M8", "REITYP", "REITRU" )
75  setParameter( "GEOMET", "Reinfo_M8", "REITYP", "CIRBEA" )
76  setParameter( "GEOMET", "Reinfo_M8", "CIRBEA/CIRCLE", 30 )
77  setParameter( "GEOMET", "Reinfo_M8", "TIPELM/SURFAC", 706.9 )
78  assignMaterial( "Reinfo_M8", "SHAPE", [ "Bar_M8" ] )
79  assignGeometry( "Reinfo_M8", "SHAPE", [ "Bar_M8" ] )
80  convertToReinforcement( [ "Bar_M8" ] )
81  addElementData( "Bar" )
82  assignElementData( "Bar", "SHAPE", [ "Bar_M8" ] )
83  setParameter( "DATA", "Bar", "./INTERF", [] )
84  setParameter( "DATA", "Bar", "INTERF", "BEAM" )
85  setReinforcementDiscretization( [ "Bar_M8" ], "ELEMENT" )
86  setContinuousInInterfaces( [ "Bar_M8" ], True )
87
88  #Assign Supports
89  addSet( "GEOMETRYSUPPORTSET", "Support" )
90
91  createLineSupport( "Top Support", "Support" )
92  setParameter( "GEOMETRYSUPPORT", "Top Support", "AXES", [ 1, 2 ] )
93  setParameter( "GEOMETRYSUPPORT", "Top Support", "TRANSL", [ 0, 1, 0 ] )
94  setParameter( "GEOMETRYSUPPORT", "Top Support", "ROTATI", [ 0, 0, 0 ] )
95  attach( "GEOMETRYSUPPORT", "Top Support", "Pressure Layer", [[ 100, 300, 0
    ],[300,300,0]] )
96
97  createPointSupport( "Topl Support", "Support" )
98  setParameter( "GEOMETRYSUPPORT", "Topl Support", "AXES", [ 1, 2 ] )
99  setParameter( "GEOMETRYSUPPORT", "Topl Support", "TRANSL", [ 1, 0, 0 ] )
100 setParameter( "GEOMETRYSUPPORT", "Topl Support", "ROTATI", [ 0, 0, 0 ] )
101 attach( "GEOMETRYSUPPORT", "Topl Support", "Pressure Layer", [[ 0, 300, 0 ]] )
102
103 createPointSupport( "Topm Support", "Support" )
104 setParameter( "GEOMETRYSUPPORT", "Topm Support", "AXES", [ 1, 2 ] )
105 setParameter( "GEOMETRYSUPPORT", "Topm Support", "TRANSL", [ 1, 0, 0 ] )
106 setParameter( "GEOMETRYSUPPORT", "Topm Support", "ROTATI", [ 0, 0, 0 ] )
107 attach( "GEOMETRYSUPPORT", "Topm Support", "Pressure Layer", [[ 200, 300, 0 ]] )
108
109 createPointSupport( "Topr Support", "Support" )
110 setParameter( "GEOMETRYSUPPORT", "Topr Support", "AXES", [ 1, 2 ] )
111 setParameter( "GEOMETRYSUPPORT", "Topr Support", "TRANSL", [ 1, 0, 0 ] )
112 setParameter( "GEOMETRYSUPPORT", "Topr Support", "ROTATI", [ 0, 0, 0 ] )
113 attach( "GEOMETRYSUPPORT", "Topr Support", "Pressure Layer", [[ 400, 300, 0 ]] )
114
115 createLineSupport( "Bot Support", "Support" )
116 setParameter( "GEOMETRYSUPPORT", "Bot Support", "AXES", [ 1, 2 ] )
117 setParameter( "GEOMETRYSUPPORT", "Bot Support", "TRANSL", [ 0, 1, 0 ] )
118 setParameter( "GEOMETRYSUPPORT", "Bot Support", "ROTATI", [ 0, 0, 0 ] )
119 attach( "GEOMETRYSUPPORT", "Bot Support", "Pressure Layer", [[ 100, 70, 0
    ],[300,70,0]] )
120
121 createPointSupport( "Botp Support", "Support" )
122 setParameter( "GEOMETRYSUPPORT", "Botp Support", "AXES", [ 1, 2 ] )
123 setParameter( "GEOMETRYSUPPORT", "Botp Support", "TRANSL", [ 0, 1, 0 ] )
124 setParameter( "GEOMETRYSUPPORT", "Botp Support", "ROTATI", [ 0, 0, 0 ] )
125 attach( "GEOMETRYSUPPORT", "Botp Support", "Bar_M8", [[ 200, 70, 0 ]] )
126
127 #Assign Loads
128 addSet( "GEOMETRYLOADSET", "Load" )
129
130 createPointLoad( "Load", "Load" )
131 setParameter( "GEOMETRYLOAD", "Load", "LODTYP", "DEFORM" )
132 setParameter( "GEOMETRYLOAD", "Load", "DEFORM/SUPP", "Botp Support" )
133 setParameter( "GEOMETRYLOAD", "Load", "DEFORM/TR/VALUE", -10 )
134 setParameter( "GEOMETRYLOAD", "Load", "DEFORM/TR/DIRECT", 2 )
```

```
135 attach( "GEOMETRYLOAD", "Load", "Bar_M8", [[ 200, 70, 0 ]] )
136
137 #Set mesh properties
138 setElementSize( [ "Pressure Layer","Bar_M8" ], 10, -1, True )
139 setMesherType( [ "Pressure Layer","Bar_M8" ], "HEXQUAD" )
140 clearMidSideNodeLocation( [  "Pressure Layer","Bar_M8" ] )
141 generateMesh( [] )
142 hideView( "GEOM" )
143 showView( "MESH" )
144
145 #Analysis
146 addAnalysis( "Analysis1" )
147 addAnalysisCommand( "Analysis1", "NONLIN", "Structural nonlinear" )
148 renameAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "EXECUT(1)",
    "Load" )
149 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/STEPTY", "AUTOMA" )
150 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/SIZES", 1 )
151 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/MAXSIZ", 0.05 )
152 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/MINSIZ", 0.005 )
153 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/CUTBCK", 0.25 )
154 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/SIZES", "0.010000(100)" )
155 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN", False )
156 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN/REGULA/SET(1)/NODES(1)/RNGNRS", "ALL" )
157 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN/REGULA/SET(1)/DIRECT", 2 )
158
159 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/MAXITE", 25 )
160 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/LINESE" )
161 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/LINESE", True )
162 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/LINESE/ETAMIN", 0.1 )
163 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/LINESE/ETAMAX", 1 )
164 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/ENERGY" )
165 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/ENERGY", True )
166 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/DISPLA", False )
167 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/ENERGY/TOLCON", 0.001 )
168 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/ENERGY/NOCONV", "CONTIN" )
169 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/FORCE/NOCONV", "CONTIN" )
170 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/METHOD/METNAM", "SECANT" )
171
172 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "OUTPUT(1)/SELTYP",
    "USER" )
173
174 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/DISPLA(1)/TOTAL/TRANSL/GLOBAL" )
175 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(1)/TOTAL/GREEN/GLOBAL" )
176 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(1)/TOTAL/CAUCHY/GLOBAL" )
177
178 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(1)/TOTAL/GREEN/GLOBAL" )
```

```
179  # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
     "OUTPUT(1)/USER/STRAIN(2)/TOTAL/GREEN/PRINCI" )
180  # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
     "OUTPUT(1)/USER/STRAIN(4)/CRACK/GREEN" )
181  # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
     "OUTPUT(1)/USER/STRAIN(7)/CRKWDT/GREEN/GLOBAL" )
182  # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
     "OUTPUT(1)/USER/STRAIN(8)/CRKWDT/GREEN/PRINCI" )
183  # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
     "OUTPUT(1)/USER/STRESS(3)/CRACK/CAUCHY/GLOBAL" )
184  # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
     "OUTPUT(1)/USER/STRESS(2)/TOTAL/CAUCHY/PRINCI" )
185
186  #Run Analysis
187  runSolver( [] )
188  showView( "RESULT" )
189  fitAll(  )
```

B1 - T13 V10

```
newProject( "D:/Afstuderen/Python/Proefstukken/DIANA Output/T13 V10 Reduced Elements new", 10 )


###Project Settings###
setModelAnalysisAspects( [ "STRUCT" ] )
setModelDimension( "2D" )
setDefaultMeshOrder( "QUADRATIC" )
setDefaultMesherType( "HEXQUAD" )
setDefaultMidSideNodeLocation( "ONSHAP" )
setUnit( "LENGTH", "MM" )
setUnit( "FORCE", "N" )
setUnit( "ANGLE", "DEGREE" )


#Create Shapes
createSheet( "Pressure Layer", [[ 0, 70, 0 ],[ 1900, 70, 0 ],[ 1900, 250, 0 ],[ 1150, 250, 0 ],[1050, 250, 0 ],[ 0, 250, 0 ]] )
createSheet( "Slab", [[ 0, 0, 0 ],[ 250, 0, 0 ],[ 350, 0, 0 ],[ 1900, 0, 0 ],[ 1900, 70, 0],[ 0, 70, 0 ]] )

createSheet( "Support", [[ 250, 0, 0 ],[ 350, 0, 0 ],[ 350, -30, 0 ],[ 300, -30, 0 ],[ 250, -30, 0 ]] )
createSheet( "Loadplate", [[1050, 250, 0 ],[ 1150, 250, 0 ],[ 1150, 280, 0 ],[ 1100, 280, 0 ],[ 1050, 280, 0 ]] )

#Create Reinforcement
createLine( "Koppel_M12a", [ 1200, 76, 0 ], [ 1900, 76, 0 ] )
createLine( "Koppel_M12b", [ 1200, 76, 0 ], [ 1900, 76, 0 ] )
createLine( "Koppel_M12c", [ 1200, 76, 0 ], [ 1900, 76, 0 ] )
createLine( "Koppel_M12d", [ 1200, 76, 0 ], [ 1900, 76, 0 ] )
createLine( "Koppel_M12e", [ 1200, 76, 0 ], [ 1900, 76, 0 ] )
createLine( "Koppel_M12f", [ 1200, 76, 0 ], [ 1900, 76, 0 ] )
createLine( "Koppel_M12g", [ 1200, 76, 0 ], [ 1900, 76, 0 ] )

createLine( "Reinfo_M8a", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8b", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8c", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8d", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8e", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8f", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8g", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8h", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8i", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8j", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8k", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8l", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8m", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8n", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8o", [ 0, 35, 0 ], [ 1865, 35, 0 ] )

createLine( "Bar_M41al", [ 135, 45.5, 0],[ 175, 165.5, 0] )
createLine( "Bar_M41ar", [ 215, 45.5, 0],[ 175, 165.5, 0] )
createLine( "Bar_M41bl", [ 135, 45.5, 0],[ 175, 165.5, 0] )
createLine( "Bar_M41br", [ 215, 45.5, 0],[ 175, 165.5, 0] )
createLine( "Bar_M41cl", [ 135, 45.5, 0],[ 175, 165.5, 0] )
createLine( "Bar_M41cr", [ 215, 45.5, 0],[ 175, 165.5, 0] )
createLine( "Bar_M41dl", [ 135, 45.5, 0],[ 175, 165.5, 0] )
createLine( "Bar_M41dr", [ 215, 45.5, 0],[ 175, 165.5, 0] )
createLine( "Bar_M41el", [ 135, 45.5, 0],[ 175, 165.5, 0] )
createLine( "Bar_M41er", [ 215, 45.5, 0],[ 175, 165.5, 0] )
createLine( "Bar_M41fl", [ 135, 45.5, 0],[ 175, 165.5, 0] )
createLine( "Bar_M41fr", [ 215, 45.5, 0],[ 175, 165.5, 0] )
createLine( "Bar_M41gl", [ 135, 45.5, 0],[ 175, 165.5, 0] )
createLine( "Bar_M41gr", [ 215, 45.5, 0],[ 175, 165.5, 0] )
createLine( "Bar_M41hl", [ 135, 45.5, 0],[ 175, 165.5, 0] )
createLine( "Bar_M41hr", [ 215, 45.5, 0],[ 175, 165.5, 0] )
createLine( "Bar_M41il", [ 135, 45.5, 0],[ 175, 165.5, 0] )
createLine( "Bar_M41ir", [ 215, 45.5, 0],[ 175, 165.5, 0] )
createLine( "Bar_M41jl", [ 135, 45.5, 0],[ 175, 165.5, 0] )
createLine( "Bar_M41jr", [ 215, 45.5, 0],[ 175, 165.5, 0] )
```

```
67
68  createLine( "Bar_M42al", [ 735, 45.5, 0],[ 775, 165.5, 0] )
69  createLine( "Bar_M42ar", [ 815, 45.5, 0],[ 775, 165.5, 0] )
70  createLine( "Bar_M42bl", [ 735, 45.5, 0],[ 775, 165.5, 0] )
71  createLine( "Bar_M42br", [ 815, 45.5, 0],[ 775, 165.5, 0] )
72  createLine( "Bar_M42cl", [ 735, 45.5, 0],[ 775, 165.5, 0] )
73  createLine( "Bar_M42cr", [ 815, 45.5, 0],[ 775, 165.5, 0] )
74  createLine( "Bar_M42dl", [ 735, 45.5, 0],[ 775, 165.5, 0] )
75  createLine( "Bar_M42dr", [ 815, 45.5, 0],[ 775, 165.5, 0] )
76  createLine( "Bar_M42el", [ 735, 45.5, 0],[ 775, 165.5, 0] )
77  createLine( "Bar_M42er", [ 815, 45.5, 0],[ 775, 165.5, 0] )
78  createLine( "Bar_M42fl", [ 735, 45.5, 0],[ 775, 165.5, 0] )
79  createLine( "Bar_M42fr", [ 815, 45.5, 0],[ 775, 165.5, 0] )
80  createLine( "Bar_M42gl", [ 735, 45.5, 0],[ 775, 165.5, 0] )
81  createLine( "Bar_M42gr", [ 815, 45.5, 0],[ 775, 165.5, 0] )
82  createLine( "Bar_M42hl", [ 735, 45.5, 0],[ 775, 165.5, 0] )
83  createLine( "Bar_M42hr", [ 815, 45.5, 0],[ 775, 165.5, 0] )
84  createLine( "Bar_M42il", [ 735, 45.5, 0],[ 775, 165.5, 0] )
85  createLine( "Bar_M42ir", [ 815, 45.5, 0],[ 775, 165.5, 0] )
86  createLine( "Bar_M42jl", [ 735, 45.5, 0],[ 775, 165.5, 0] )
87  createLine( "Bar_M42jr", [ 815, 45.5, 0],[ 775, 165.5, 0] )
88
89  createLine( "Bar_M43al", [ 1460, 45.5, 0],[ 1500, 165.5, 0] )
90  createLine( "Bar_M43ar", [ 1540, 45.5, 0],[ 1500, 165.5, 0] )
91  createLine( "Bar_M43bl", [ 1460, 45.5, 0],[ 1500, 165.5, 0] )
92  createLine( "Bar_M43br", [ 1540, 45.5, 0],[ 1500, 165.5, 0] )
93  createLine( "Bar_M43cl", [ 1460, 45.5, 0],[ 1500, 165.5, 0] )
94  createLine( "Bar_M43cr", [ 1540, 45.5, 0],[ 1500, 165.5, 0] )
95  createLine( "Bar_M43dl", [ 1460, 45.5, 0],[ 1500, 165.5, 0] )
96  createLine( "Bar_M43dr", [ 1540, 45.5, 0],[ 1500, 165.5, 0] )
97  createLine( "Bar_M43el", [ 1460, 45.5, 0],[ 1500, 165.5, 0] )
98  createLine( "Bar_M43er", [ 1540, 45.5, 0],[ 1500, 165.5, 0] )
99  createLine( "Bar_M43fl", [ 1460, 45.5, 0],[ 1500, 165.5, 0] )
100 createLine( "Bar_M43fr", [ 1540, 45.5, 0],[ 1500, 165.5, 0] )
101 createLine( "Bar_M43gl", [ 1460, 45.5, 0],[ 1500, 165.5, 0] )
102 createLine( "Bar_M43gr", [ 1540, 45.5, 0],[ 1500, 165.5, 0] )
103 createLine( "Bar_M43hl", [ 1460, 45.5, 0],[ 1500, 165.5, 0] )
104 createLine( "Bar_M43hr", [ 1540, 45.5, 0],[ 1500, 165.5, 0] )
105 createLine( "Bar_M43il", [ 1460, 45.5, 0],[ 1500, 165.5, 0] )
106 createLine( "Bar_M43ir", [ 1540, 45.5, 0],[ 1500, 165.5, 0] )
107 createLine( "Bar_M43jl", [ 1460, 45.5, 0],[ 1500, 165.5, 0] )
108 createLine( "Bar_M43jr", [ 1540, 45.5, 0],[ 1500, 165.5, 0] )
109
110 createLine( "Composed Line", [ 0, 125, 0 ], [ 1900, 125, 0 ] )
111
112 createSheet( "Mesh Sheet", [[ 1000, 0, 0 ],[ 1000, 300, 0 ],[ 1000, 300, 1000 ],[
    1000, 0, 1000 ]] )
113
114 #Splitting Mesh Parts
115 subtract( "Pressure Layer", [ "Mesh Sheet" ], True, True )
116 subtract( "Slab", [ "Mesh Sheet" ], False, True )
117
118 #Regroup Material Parts
119 addSet( "SHAPESET", "Coupling Reinforcement" )
120 moveToShapeSet( [
    "Koppel_M12a","Koppel_M12b","Koppel_M12c","Koppel_M12d","Koppel_M12e","Koppel_M12f","K
    oppel_M12g" ], "Coupling Reinforcement" )
121 addSet( "SHAPESET", "Longitudinal Reinforcement" )
122 moveToShapeSet( [
    "Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d","Reinfo_M8e","Reinfo_M8f","Reinfo_
    M8g","Reinfo_M8h","Reinfo_M8i","Reinfo_M8j","Reinfo_M8k","Reinfo_M8l","Reinfo_M8m","Re
    info_M8n","Reinfo_M8o" ], "Longitudinal Reinforcement" )
123 addSet( "SHAPESET", "Lattice girders" )
124 moveToShapeSet( [
    "Bar_M41al","Bar_M41ar","Bar_M41bl","Bar_M41br","Bar_M41cl","Bar_M41cr","Bar_M41dl","B
    ar_M41dr","Bar_M41el","Bar_M41er","Bar_M41fl","Bar_M41fr","Bar_M41gl","Bar_M41gr","Bar
    _M41hl","Bar_M41hr","Bar_M41il","Bar_M41ir","Bar_M41jl","Bar_M41jr","Bar_M42al","Bar_M
    42ar","Bar_M42bl","Bar_M42br","Bar_M42cl","Bar_M42cr","Bar_M42dl","Bar_M42dr","Bar_M42
    el","Bar_M42er","Bar_M42fl","Bar_M42fr","Bar_M42gl","Bar_M42gr","Bar_M42hl","Bar_M42hr
    ","Bar_M42il","Bar_M42ir","Bar_M42jl","Bar_M42jr","Bar_M43al","Bar_M43ar","Bar_M43bl",
    "Bar_M43br","Bar_M43cl","Bar_M43cr","Bar_M43dl","Bar_M43dr","Bar_M43el","Bar_M43er","B
```

```
    ar_M43fl","Bar_M43fr","Bar_M43gl","Bar_M43gr","Bar_M43hl","Bar_M43hr","Bar_M43il","Bar
    _M43ir","Bar_M43jl","Bar_M43jr" ], "Lattice girders" )
125 addSet( "SHAPESET", "Steel plates" )
126 moveToShapeSet( [ "Loadplate", "Support" ], "Steel plates" )
127 rename( "SHAPESET", "Shapes", "Concrete" )
128 addSet( "SHAPESET", "Composed Line" )
129 moveToShapeSet( [ "Composed Line" ], "Composed Line" )
130
131 #Material Precast concrete
132 addMaterial( "Precast concrete", "CONCR", "TSCR", [] )
133 setParameter( "MATERIAL", "Precast concrete", "LINEAR/ELASTI/YOUNG", 33372 )
134 setParameter( "MATERIAL", "Precast concrete", "LINEAR/ELASTI/POISON", 0.2 )
135 setParameter( "MATERIAL", "Precast concrete", "LINEAR/MASS/DENSIT", 2.4e-09 )
136 setParameter( "MATERIAL", "Precast concrete", "MODTYP/TOTCRK", "ROTATE" )
137 setParameter( "MATERIAL", "Precast concrete", "TENSIL/TENCRV", "HORDYK" )
138 setParameter( "MATERIAL", "Precast concrete", "TENSIL/TENSTR", 3.03 )
139 setParameter( "MATERIAL", "Precast concrete", "TENSIL/GF1", 0.1419 )
140 setParameter( "MATERIAL", "Precast concrete", "TENSIL/CBSPEC", "ROTS" )
141 setParameter( "MATERIAL", "Precast concrete", "TENSIL/POISRE/POIRED", "DAMAGE" )
142 # setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMCRV", "MC2010" )
143 # setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMSTR", 55.9 )
144 # setParameter( "MATERIAL", "Precast concrete", "COMPRS/EPSC1", 0.0024 )
145 # setParameter( "MATERIAL", "Precast concrete", "COMPRS/EPSCU", 0.0035 )
146 # setParameter( "MATERIAL", "Precast concrete", "COMPRS/CONFIN/CNFCRV", "NONE" )
147 setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMCRV", "CONSTA" )
148 setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMSTR", 40.1 )
149
150 addGeometry( "Precast concrete", "SHEET", "MEMBRA", [] )
151 setParameter( "GEOMET", "Precast concrete", "THICK", 1000 )
152 setParameter( "GEOMET", "Precast concrete", "LOCAXS", True )
153 setParameter( "GEOMET", "Precast concrete", "LOCAXS/XAXIS", [ 1, 0, 0 ] )
154
155 addElementData( "Precast concrete" )
156 setElementClassType( "SHAPE", [ "Slab", "Slab_1" ], "MEMBRA" )
157 assignMaterial( "Precast concrete", "SHAPE", [ "Slab", "Slab_1" ] )
158 assignGeometry( "Precast concrete", "SHAPE", [ "Slab", "Slab_1" ] )
159 assignElementData( "Precast concrete", "SHAPE", [ "Slab", "Slab_1" ] )
160
161 # setParameter( "DATA", "Precast concrete", "INTEGR", "REDUCE" )
162
163 #Material Pressure layer
164 addMaterial( "Pressure layer", "CONCR", "TSCR", [] )
165 setParameter( "MATERIAL", "Pressure layer", "LINEAR/ELASTI/YOUNG", 26045 )
166 setParameter( "MATERIAL", "Pressure layer", "LINEAR/ELASTI/POISON", 0.2 )
167 setParameter( "MATERIAL", "Pressure layer", "LINEAR/MASS/DENSIT", 2.4e-09 )
168 setParameter( "MATERIAL", "Pressure layer", "MODTYP/TOTCRK", "ROTATE" )
169 setParameter( "MATERIAL", "Pressure layer", "TENSIL/TENCRV", "HORDYK" )
170 setParameter( "MATERIAL", "Pressure layer", "TENSIL/TENSTR", 1.35 )
171 setParameter( "MATERIAL", "Pressure layer", "TENSIL/GF1", 0.1223 )
172 setParameter( "MATERIAL", "Pressure layer", "TENSIL/CBSPEC", "ROTS" )
173 setParameter( "MATERIAL", "Pressure layer", "TENSIL/POISRE/POIRED", "DAMAGE" )
174 # setParameter( "MATERIAL", "Pressure layer", "COMPRS/COMCRV", "MC2010" )
175 # setParameter( "MATERIAL", "Pressure layer", "COMPRS/COMSTR", 35.6 )
176 # setParameter( "MATERIAL", "Pressure layer", "COMPRS/EPSC1", 0.0021 )
177 # setParameter( "MATERIAL", "Pressure layer", "COMPRS/EPSCU", 0.0035 )
178 # setParameter( "MATERIAL", "Pressure layer", "COMPRS/CONFIN/CNFCRV", "NONE" )
179 setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMCRV", "CONSTA" )
180 setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMSTR", 17.6 )
181
182 addGeometry( "Pressure layer", "SHEET", "MEMBRA", [] )
183 setParameter( "GEOMET", "Pressure layer", "THICK", 1000 )
184 setParameter( "GEOMET", "Pressure layer", "LOCAXS", True )
185 setParameter( "GEOMET", "Pressure layer", "LOCAXS/XAXIS", [ 1, 0, 0 ] )
186
187 addElementData( "Pressure layer" )
188 setElementClassType( "SHAPE", [ "Pressure Layer", "Pressure Layer_1" ], "MEMBRA" )
189 assignMaterial( "Pressure layer", "SHAPE", [ "Pressure Layer", "Pressure Layer_1" ] )
190 assignGeometry( "Pressure layer", "SHAPE", [ "Pressure Layer", "Pressure Layer_1" ] )
191 assignElementData( "Pressure layer", "SHAPE", [ "Pressure Layer", "Pressure Layer_1"
    ] )
192
```

```
193  # setParameter( "DATA", "Pressure layer", "INTEGR", "REDUCE" )
194
195  #Material Steel
196  addMaterial( "Steel", "MCSTEL", "ISOTRO", [] )
197  setParameter( "MATERIAL", "Steel", "LINEAR/ELASTI/YOUNG", 210000 )
198  setParameter( "MATERIAL", "Steel", "LINEAR/ELASTI/POISON", 0.3 )
199  setParameter( "MATERIAL", "Steel", "LINEAR/MASS/DENSIT", 0 )
200  addGeometry( "Plate", "SHEET", "MEMBRA", [] )
201  setParameter( "GEOMET", "Plate", "THICK", 1000 )
202  addElementData( "Plate" )
203
204  setElementClassType( "SHAPE", [ "Support", "Loadplate" ], "MEMBRA" )
205  assignMaterial( "Steel", "SHAPE", [ "Support", "Loadplate" ] )
206  assignGeometry( "Plate", "SHAPE", [ "Support", "Loadplate" ] )
207  assignElementData( "Plate", "SHAPE", [ "Support", "Loadplate" ] )
208  # setParameter( "DATA", "Plate", "INTEGR", "REDUCE" )
209
210  #Concrete Interface
211  # addMaterial( "Interface Concrete", "INTERF", "FRICTI", [] )
212  # setParameter( "MATERIAL", "Interface Concrete", "LINEAR/IFTYP", "LIN2D" )
213  # setParameter( "MATERIAL", "Interface Concrete", "COULOM/OPNTYP", "GAP" )
214  # setParameter( "MATERIAL", "Interface Concrete", "COULOM/GAPVAL", 0.5 )
215  # setParameter( "MATERIAL", "Interface Concrete", "COULOM/COHESI", 0.2 )
216  # setParameter( "MATERIAL", "Interface Concrete", "COULOM/PHI", 22 )
217  # setParameter( "MATERIAL", "Interface Concrete", "COULOM/PSI", 5.71 )
218  # setParameter( "MATERIAL", "Interface Concrete", "LINEAR/ELAS2/DSNY", 10 )
219  # setParameter( "MATERIAL", "Interface Concrete", "LINEAR/ELAS2/DSSX", 10 )
220  # setParameter( "MATERIAL", "Interface Concrete", "COULOM/FRCHSR/MODE2", 0 )
221
222  addMaterial( "Interface Concrete", "INTERF", "FRICEL", [] )
223  setParameter( "MATERIAL", "Interface Concrete", "LINEAR/IFTYP", "LIN2D" )
224  setParameter( "MATERIAL", "Interface Concrete", "NEFRIC/SPCTYP", "USER" )
225  setParameter( "MATERIAL", "Interface Concrete", "NEFRIC/COHESI", 0.5)
226  setParameter( "MATERIAL", "Interface Concrete", "NEFRIC/PHI", 31 )
227  setParameter( "MATERIAL", "Interface Concrete", "LINEAR/ELAS2/DSNY", 60000 )
228  setParameter( "MATERIAL", "Interface Concrete", "LINEAR/ELAS2/DSSX", 6000 )
229
230  addGeometry( "Interface Concrete", "LINE", "STLIIF", [] )
231  setParameter( "GEOMET", "Interface Concrete", "LIFMEM/THICK", 1000 )
232  addElementData( "Interface Concrete" )
233  createConnection( "Interface Concrete", "INTER", "SHAPEEDGE", "SHAPEEDGE" )
234  setParameter( "GEOMETRYCONNECTION", "Interface Concrete", "MODE", "CLOSED" )
235  setElementClassType( "GEOMETRYCONNECTION", "Interface Concrete", "STLIIF" )
236  assignMaterial( "Interface Concrete", "GEOMETRYCONNECTION", "Interface Concrete" )
237  assignGeometry( "Interface Concrete", "GEOMETRYCONNECTION", "Interface Concrete" )
238  assignElementData( "Interface Concrete", "GEOMETRYCONNECTION", "Interface Concrete" )
239  setParameter( "GEOMETRYCONNECTION", "Interface Concrete", "FLIP", False )
240  attachTo( "GEOMETRYCONNECTION", "Interface Concrete", "SOURCE", "Pressure Layer", [[
     500, 70, 0 ]] )
241  attachTo( "GEOMETRYCONNECTION", "Interface Concrete", "TARGET", "Slab", [[ 500, 70,
     0 ]] )
242  attachTo( "GEOMETRYCONNECTION", "Interface Concrete", "SOURCE", "Pressure Layer_1",
     [[ 1450, 70, 0 ]] )
243  attachTo( "GEOMETRYCONNECTION", "Interface Concrete", "TARGET", "Slab_1", [[ 1450,
     70, 0 ]] )
244
245  # setParameter( "DATA", "Interface Concrete", "INTEGR", "REDUCE" )
246
247  #Steel Interface
248  addMaterial( "Interface Steel", "INTERF", "NONLIF", [] )
249  setParameter( "MATERIAL", "Interface Steel", "LINEAR/IFTYP", "LIN2D" )
250  setParameter( "MATERIAL", "Interface Steel", "LINEAR/ELAS2/DSNY", 100000 )
251  setParameter( "MATERIAL", "Interface Steel", "LINEAR/ELAS2/DSSX", 0.001 )
252  setParameter( "MATERIAL", "Interface Steel", "NONLIN/IFNOTE", "NOTSHR" )
253  setParameter( "MATERIAL", "Interface Steel", "NONLIN/NLEL8/NOTENS", [ 0.001, 0 ] )
254  setParameter( "MATERIAL", "Interface Steel", "NONLIN/NLEL8/NOSHTE", [ 0.001, 0 ] )
255  addGeometry( "Interface Steel", "LINE", "STLIIF", [] )
256  setParameter( "GEOMET", "Interface Steel", "LIFMEM/THICK", 1000 )
257  addElementData( "Interface Steel" )
258  createConnection( "Interface Steel", "INTER", "SHAPEEDGE", "SHAPEEDGE" )
259
```

```
setParameter( "GEOMETRYCONNECTION", "Interface Steel", "CONTYP", "INTER" )
setParameter( "GEOMETRYCONNECTION", "Interface Steel", "MODE", "AUTO" )
setElementClassType( "GEOMETRYCONNECTION", "Interface Steel", "STLIIF" )
assignMaterial( "Interface Steel", "GEOMETRYCONNECTION", "Interface Steel" )
assignGeometry( "Interface Steel", "GEOMETRYCONNECTION", "Interface Steel" )
assignElementData( "Interface Steel", "GEOMETRYCONNECTION", "Interface Steel" )
setParameter( "GEOMETRYCONNECTION", "Interface Steel", "FLIP", False )
attachTo( "GEOMETRYCONNECTION", "Interface Steel", "SOURCE", "Pressure Layer_1", [[ 1100, 250, 0 ]] )
attachTo( "GEOMETRYCONNECTION", "Interface Steel", "TARGET", "Loadplate", [[ 1100, 250, 0 ]] )
attachTo( "GEOMETRYCONNECTION", "Interface Steel", "SOURCE", "Slab", [[ 300, 0, 0 ]] )
attachTo( "GEOMETRYCONNECTION", "Interface Steel", "TARGET", "Support", [[ 300, 0, 0 ]] )

# setParameter( "DATA", "Interface Steel", "INTEGR", "REDUCE" )
# setParameter( "DATA", "Interface Steel", "NUMINT", [ "LOBATT" ] )

#Material Reinforcement Koppel_M12
addMaterial( "Koppel_M12", "REINFO", "REBOND", [ ] )
setParameter( "MATERIAL", "Koppel_M12", "REBARS/ELASTI/YOUNG", 200000 )
setParameter( "MATERIAL", "Koppel_M12", "REBARS/POISON/POISON", 0.3 )
setParameter( "MATERIAL", "Koppel_M12", "REBARS/PLATYP", "VMISES" )
setParameter( "MATERIAL", "Koppel_M12", "REBARS/PLASTI/TRESSH", "KAPSIG" )
setParameter( "MATERIAL", "Koppel_M12", "REBARS/PLASTI/KAPSIG", [ 0, 550, 0.04725, 594 ] )
setParameter( "MATERIAL", "Koppel_M12", "RESLIP/SHFTYP", "BONDS6" )
setParameter( "MATERIAL", "Koppel_M12", "RESLIP/DSNY", 100000 )
setParameter( "MATERIAL", "Koppel_M12", "RESLIP/DSSX", 10000 )
setParameter( "MATERIAL", "Koppel_M12", "RESLIP/BONDS6/SLPVAL", [ 15.411, 6.164, 0.1, 1, 2, 10, 0.4 ] )
addGeometry( "Koppel_M12", "RELINE", "REBAR", [] )
setParameter( "GEOMET", "Koppel_M12", "REITYP", "REITRU" )
setParameter( "GEOMET", "Koppel_M12", "REITYP", "CIRBEA" )
setParameter( "GEOMET", "Koppel_M12", "CIRBEA/CIRCLE", 12)
setParameter( "GEOMET", "Koppel_M12", "TIPELM/SURFAC", 113.1 )
assignMaterial( "Koppel_M12", "SHAPE", [ "Koppel_M12a","Koppel_M12b","Koppel_M12c","Koppel_M12d","Koppel_M12e","Koppel_M12f","Koppel_M12g" ] )
assignGeometry( "Koppel_M12", "SHAPE", [ "Koppel_M12a","Koppel_M12b","Koppel_M12c","Koppel_M12d","Koppel_M12e","Koppel_M12f","Koppel_M12g" ] )
convertToReinforcement( [ "Koppel_M12a","Koppel_M12b","Koppel_M12c","Koppel_M12d","Koppel_M12e","Koppel_M12f","Koppel_M12g" ] )
addElementData( "Koppel_M12" )
assignElementData( "Koppel_M12", "SHAPE", [ "Koppel_M12a","Koppel_M12b","Koppel_M12c","Koppel_M12d","Koppel_M12e","Koppel_M12f","Koppel_M12g" ] )
setParameter( "DATA", "Koppel_M12", "INTERF", "BEAM" )
setReinforcementDiscretization( [ "Koppel_M12a","Koppel_M12b","Koppel_M12c","Koppel_M12d","Koppel_M12e","Koppel_M12f","Koppel_M12g" ], "ELEMENT" )
setContinuousInInterfaces( [ "Koppel_M12a","Koppel_M12b","Koppel_M12c","Koppel_M12d","Koppel_M12e","Koppel_M12f","Koppel_M12g" ], False )

#Material Reinforcement M8
addMaterial( "Reinfo_M8", "REINFO", "REBOND", [] )
setParameter( "MATERIAL", "Reinfo_M8", "REBARS/ELASTI/YOUNG", 200000 )
setParameter( "MATERIAL", "Reinfo_M8", "REBARS/POISON/POISON", 0.3 )
setParameter( "MATERIAL", "Reinfo_M8", "REBARS/PLATYP", "VMISES" )
setParameter( "MATERIAL", "Reinfo_M8", "REBARS/PLASTI/TRESSH", "KAPSIG" )
setParameter( "MATERIAL", "Reinfo_M8", "REBARS/PLASTI/KAPSIG", [ 0, 550, 0.04725, 594 ] )
setParameter( "MATERIAL", "Reinfo_M8", "RESLIP/SHFTYP", "BONDS6" )
setParameter( "MATERIAL", "Reinfo_M8", "RESLIP/DSNY", 100000 )
setParameter( "MATERIAL", "Reinfo_M8", "RESLIP/DSSX", 10000 )
setParameter( "MATERIAL", "Reinfo_M8", "RESLIP/BONDS6/SLPVAL", [ 15.411, 6.164, 0.1, 1, 2, 10, 0.4 ] )
addGeometry( "Reinfo_M8", "RELINE", "REBAR", [] )
```

```
setParameter( "GEOMET", "Reinfo_M8", "REITYP", "REITRU" )
setParameter( "GEOMET", "Reinfo_M8", "REITYP", "CIRBEA" )
setParameter( "GEOMET", "Reinfo_M8", "CIRBEA/CIRCLE", 8)
setParameter( "GEOMET", "Reinfo_M8", "TIPELM/SURFAC", 50.3 )
assignMaterial( "Reinfo_M8", "SHAPE", [
"Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d","Reinfo_M8e","Reinfo_M8f","Reinfo_
M8g","Reinfo_M8h","Reinfo_M8i","Reinfo_M8j","Reinfo_M8k","Reinfo_M8l","Reinfo_M8m","Re
info_M8n","Reinfo_M8o" ] )
assignGeometry( "Reinfo_M8", "SHAPE", [
"Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d","Reinfo_M8e","Reinfo_M8f","Reinfo_
M8g","Reinfo_M8h","Reinfo_M8i","Reinfo_M8j","Reinfo_M8k","Reinfo_M8l","Reinfo_M8m","Re
info_M8n","Reinfo_M8o" ] )
convertToReinforcement( [
"Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d","Reinfo_M8e","Reinfo_M8f","Reinfo_
M8g","Reinfo_M8h","Reinfo_M8i","Reinfo_M8j","Reinfo_M8k","Reinfo_M8l","Reinfo_M8m","Re
info_M8n","Reinfo_M8o" ] )
addElementData( "Reinfo_M8" )
assignElementData( "Reinfo_M8", "SHAPE", [
"Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d","Reinfo_M8e","Reinfo_M8f","Reinfo_
M8g","Reinfo_M8h","Reinfo_M8i","Reinfo_M8j","Reinfo_M8k","Reinfo_M8l","Reinfo_M8m","Re
info_M8n","Reinfo_M8o" ] )
setParameter( "DATA", "Reinfo_M8", "INTERF", "BEAM" )
setReinforcementDiscretization( [
"Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d","Reinfo_M8e","Reinfo_M8f","Reinfo_
M8g","Reinfo_M8h","Reinfo_M8i","Reinfo_M8j","Reinfo_M8k","Reinfo_M8l","Reinfo_M8m","Re
info_M8n","Reinfo_M8o" ], "ELEMENT" )
setContinuousInInterfaces( [
"Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d","Reinfo_M8e","Reinfo_M8f","Reinfo_
M8g","Reinfo_M8h","Reinfo_M8i","Reinfo_M8j","Reinfo_M8k","Reinfo_M8l","Reinfo_M8m","Re
info_M8n","Reinfo_M8o" ], False )

#Material Reinforcement M5
addMaterial( "Reinfo_M5", "REINFO", "REBOND", [ "ANCHOR" ] )
setParameter( "MATERIAL", "Reinfo_M5", "REBARS/ELASTI/YOUNG", 200000 )
setParameter( "MATERIAL", "Reinfo_M5", "REBARS/POISON/POISON", 0.3 )
setParameter( "MATERIAL", "Reinfo_M5", "REBARS/PLATYP", "VMISES" )
setParameter( "MATERIAL", "Reinfo_M5", "REBARS/PLASTI/TRESSH", "KAPSIG" )
setParameter( "MATERIAL", "Reinfo_M5", "REBARS/PLASTI/KAPSIG", [ 0, 550, 0.04725,
594 ] )
setParameter( "MATERIAL", "Reinfo_M5", "RESLIP/SHFTYP", "BONDS6" )
setParameter( "MATERIAL", "Reinfo_M5", "RESLIP/DSNY", 100000 )
setParameter( "MATERIAL", "Reinfo_M5", "RESLIP/DSSX", 10000 )

setParameter( "MATERIAL", "Reinfo_M5", "RESLIP/BONDS6/SLPVAL", [ 0.63, 0.63, 0.01,
0.02, 0.03, 0.04, 0.5 ] )

setParameter( "MATERIAL", "Reinfo_M5", "REANCH/DTIPNX", 10000 )
setParameter( "MATERIAL", "Reinfo_M5", "REANCH/DTIPSY", 10000 )
setParameter( "MATERIAL", "Reinfo_M5", "REANCH/TIPLOC", "BOTH" )

addGeometry( "Reinfo_M5", "RELINE", "REBAR", [] )
setParameter( "GEOMET", "Reinfo_M5", "REITYP", "REITRU" )
setParameter( "GEOMET", "Reinfo_M5", "REITYP", "CIRBEA" )
setParameter( "GEOMET", "Reinfo_M5", "CIRBEA/CIRCLE", 5 )
setParameter( "GEOMET", "Reinfo_M5", "TIPELM/SURFAC", 19.6 )
assignMaterial( "Reinfo_M5", "SHAPE", [
"Bar_M41al","Bar_M41ar","Bar_M41bl","Bar_M41br","Bar_M41cl","Bar_M41cr","Bar_M41dl","B
ar_M41dr","Bar_M41el","Bar_M41er","Bar_M41fl","Bar_M41fr","Bar_M41gl","Bar_M41gr","Bar
_M41hl","Bar_M41hr","Bar_M41il","Bar_M41ir","Bar_M41jl","Bar_M41jr","Bar_M42al","Bar_M
42ar","Bar_M42bl","Bar_M42br","Bar_M42cl","Bar_M42cr","Bar_M42dl","Bar_M42dr","Bar_M42
el","Bar_M42er","Bar_M42fl","Bar_M42fr","Bar_M42gl","Bar_M42gr","Bar_M42hl","Bar_M42hr
","Bar_M42il","Bar_M42ir","Bar_M42jl","Bar_M42jr","Bar_M43al","Bar_M43ar","Bar_M43bl",
"Bar_M43br","Bar_M43cl","Bar_M43cr","Bar_M43dl","Bar_M43dr","Bar_M43el","Bar_M43er","B
ar_M43fl","Bar_M43fr","Bar_M43gl","Bar_M43gr","Bar_M43hl","Bar_M43hr","Bar_M43il","Bar
_M43ir","Bar_M43jl","Bar_M43jr" ] )
assignGeometry( "Reinfo_M5", "SHAPE", [
"Bar_M41al","Bar_M41ar","Bar_M41bl","Bar_M41br","Bar_M41cl","Bar_M41cr","Bar_M41dl","B
ar_M41dr","Bar_M41el","Bar_M41er","Bar_M41fl","Bar_M41fr","Bar_M41gl","Bar_M41gr","Bar
_M41hl","Bar_M41hr","Bar_M41il","Bar_M41ir","Bar_M41jl","Bar_M41jr","Bar_M42al","Bar_M
42ar","Bar_M42bl","Bar_M42br","Bar_M42cl","Bar_M42cr","Bar_M42dl","Bar_M42dr","Bar_M42
el","Bar_M42er","Bar_M42fl","Bar_M42fr","Bar_M42gl","Bar_M42gr","Bar_M42hl","Bar_M42hr
```

```
","Bar_M42il","Bar_M42ir","Bar_M42jl","Bar_M42jr","Bar_M43al","Bar_M43ar","Bar_M43bl",
"Bar_M43br","Bar_M43cl","Bar_M43cr","Bar_M43dl","Bar_M43dr","Bar_M43el","Bar_M43er","B
ar_M43fl","Bar_M43fr","Bar_M43gl","Bar_M43gr","Bar_M43hl","Bar_M43hr","Bar_M43il","Bar
_M43ir","Bar_M43jl","Bar_M43jr" ] )
convertToReinforcement( [
"Bar_M41al","Bar_M41ar","Bar_M41bl","Bar_M41br","Bar_M41cl","Bar_M41cr","Bar_M41dl","B
ar_M41dr","Bar_M41el","Bar_M41er","Bar_M41fl","Bar_M41fr","Bar_M41gl","Bar_M41gr","Bar
_M41hl","Bar_M41hr","Bar_M41il","Bar_M41ir","Bar_M41jl","Bar_M41jr","Bar_M42al","Bar_M
42ar","Bar_M42bl","Bar_M42br","Bar_M42cl","Bar_M42cr","Bar_M42dl","Bar_M42dr","Bar_M42
el","Bar_M42er","Bar_M42fl","Bar_M42fr","Bar_M42gl","Bar_M42gr","Bar_M42hl","Bar_M42hr
","Bar_M42il","Bar_M42ir","Bar_M42jl","Bar_M42jr","Bar_M43al","Bar_M43ar","Bar_M43bl",
"Bar_M43br","Bar_M43cl","Bar_M43cr","Bar_M43dl","Bar_M43dr","Bar_M43el","Bar_M43er","B
ar_M43fl","Bar_M43fr","Bar_M43gl","Bar_M43gr","Bar_M43hl","Bar_M43hr","Bar_M43il","Bar
_M43ir","Bar_M43jl","Bar_M43jr" ] )
addElementData( "Bar" )
assignElementData( "Bar", "SHAPE", [
"Bar_M41al","Bar_M41ar","Bar_M41bl","Bar_M41br","Bar_M41cl","Bar_M41cr","Bar_M41dl","B
ar_M41dr","Bar_M41el","Bar_M41er","Bar_M41fl","Bar_M41fr","Bar_M41gl","Bar_M41gr","Bar
_M41hl","Bar_M41hr","Bar_M41il","Bar_M41ir","Bar_M41jl","Bar_M41jr","Bar_M42al","Bar_M
42ar","Bar_M42bl","Bar_M42br","Bar_M42cl","Bar_M42cr","Bar_M42dl","Bar_M42dr","Bar_M42
el","Bar_M42er","Bar_M42fl","Bar_M42fr","Bar_M42gl","Bar_M42gr","Bar_M42hl","Bar_M42hr
","Bar_M42il","Bar_M42ir","Bar_M42jl","Bar_M42jr","Bar_M43al","Bar_M43ar","Bar_M43bl",
"Bar_M43br","Bar_M43cl","Bar_M43cr","Bar_M43dl","Bar_M43dr","Bar_M43el","Bar_M43er","B
ar_M43fl","Bar_M43fr","Bar_M43gl","Bar_M43gr","Bar_M43hl","Bar_M43hr","Bar_M43il","Bar
_M43ir","Bar_M43jl","Bar_M43jr" ] )
setParameter( "DATA", "Bar", "./INTERF", [] )
setParameter( "DATA", "Bar", "INTERF", "BEAM" )
setReinforcementDiscretization( [
"Bar_M41al","Bar_M41ar","Bar_M41bl","Bar_M41br","Bar_M41cl","Bar_M41cr","Bar_M41dl","B
ar_M41dr","Bar_M41el","Bar_M41er","Bar_M41fl","Bar_M41fr","Bar_M41gl","Bar_M41gr","Bar
_M41hl","Bar_M41hr","Bar_M41il","Bar_M41ir","Bar_M41jl","Bar_M41jr","Bar_M42al","Bar_M
42ar","Bar_M42bl","Bar_M42br","Bar_M42cl","Bar_M42cr","Bar_M42dl","Bar_M42dr","Bar_M42
el","Bar_M42er","Bar_M42fl","Bar_M42fr","Bar_M42gl","Bar_M42gr","Bar_M42hl","Bar_M42hr
","Bar_M42il","Bar_M42ir","Bar_M42jl","Bar_M42jr","Bar_M43al","Bar_M43ar","Bar_M43bl",
"Bar_M43br","Bar_M43cl","Bar_M43cr","Bar_M43dl","Bar_M43dr","Bar_M43el","Bar_M43er","B
ar_M43fl","Bar_M43fr","Bar_M43gl","Bar_M43gr","Bar_M43hl","Bar_M43hr","Bar_M43il","Bar
_M43ir","Bar_M43jl","Bar_M43jr" ], "SECTION" )

#Composed Line
addGeometry( "Composed Line", "LINE", "COMLIN", [] )
setParameter( "GEOMET", "Composed Line", "DISTAN/THICK", 250 )
setElementClassType( "SHAPE", [ "Composed Line" ], "COMLIN" )
assignGeometry( "Composed Line", "SHAPE", [ "Composed Line" ] )
resetElementData( "SHAPE", [ "Composed Line" ] )

#Assign Supports
addSet( "GEOMETRYSUPPORTSET", "Support" )

createPointSupport( "Support", "Support" )
setParameter( "GEOMETRYSUPPORT", "Support", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Support", "TRANSL", [ 0, 1, 0 ] )
setParameter( "GEOMETRYSUPPORT", "Support", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Support", "Support", [[ 300, -30, 0 ]] )

addSet( "GEOMETRYSUPPORTSET", "Sym_X" )

createLineSupport( "Sym_X", "Sym_X" )
setParameter( "GEOMETRYSUPPORT", "Sym_X", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Sym_X", "TRANSL", [ 1, 0, 0 ] )
setParameter( "GEOMETRYSUPPORT", "Sym_X", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Sym_X", "Pressure Layer_1", [[ 1900, 160, 0 ]] )

createPointSupport( "Symp_X", "Symp_X" )
setParameter( "GEOMETRYSUPPORT", "Symp_X", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Symp_X", "TRANSL", [ 1, 0, 0 ] )
setParameter( "GEOMETRYSUPPORpT", "Symp_X", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Symp_X", "Koppel_M12a", [[ 1900, 76, 0 ]] )
attach( "GEOMETRYSUPPORT", "Symp_X", "Koppel_M12b", [[ 1900, 76, 0 ]] )
attach( "GEOMETRYSUPPORT", "Symp_X", "Koppel_M12c", [[ 1900, 76, 0 ]] )
attach( "GEOMETRYSUPPORT", "Symp_X", "Koppel_M12d", [[ 1900, 76, 0 ]] )
attach( "GEOMETRYSUPPORT", "Symp_X", "Koppel_M12e", [[ 1900, 76, 0 ]] )
```

```
attach( "GEOMETRYSUPPORT", "Symp_X", "Koppel_M12f", [[ 1900, 76, 0 ]] )
attach( "GEOMETRYSUPPORT", "Symp_X", "Koppel_M12g", [[ 1900, 76, 0 ]] )

addSet( "GEOMETRYSUPPORTSET", "Load point" )

createPointSupport( "Load point", "Load point" )
setParameter( "GEOMETRYSUPPORT", "Load point", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Load point", "TRANSL", [ 0, 1, 0 ] )
setParameter( "GEOMETRYSUPPORT", "Load point", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Load point", "Loadplate", [[ 1100, 280, 0 ]] )

#Assign Loads
addSet( "GEOMETRYLOADSET", "Load" )

createPointLoad( "Load", "Load" )
setParameter( "GEOMETRYLOAD", "Load", "LODTYP", "DEFORM" )
setParameter( "GEOMETRYLOAD", "Load", "DEFORM/TR/VALUE", -0.1 )
setParameter( "GEOMETRYLOAD", "Load", "DEFORM/TR/DIRECT", 2 )
attach( "GEOMETRYLOAD", "Load", "Loadplate", [[ 1100, 280, 0 ]] )

#Set mesh properties
setElementSize( [  "Loadplate", "Pressure Layer_1", "Slab_1",
"Koppel_M12a","Koppel_M12b","Koppel_M12c","Koppel_M12d","Koppel_M12e","Koppel_M12f","K
oppel_M12g","Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d","Reinfo_M8e","Reinfo_M
8f","Reinfo_M8g","Reinfo_M8h","Reinfo_M8i","Reinfo_M8j","Reinfo_M8k","Reinfo_M8l","Rei
nfo_M8m","Reinfo_M8n","Reinfo_M8o","Bar_M41al","Bar_M41ar","Bar_M41bl","Bar_M41br","Ba
r_M41cl","Bar_M41cr","Bar_M41dl","Bar_M41dr","Bar_M41el","Bar_M41er","Bar_M41fl","Bar_
M41fr","Bar_M41gl","Bar_M41gr","Bar_M41hl","Bar_M41hr","Bar_M41il","Bar_M41ir","Bar_M4
1jl","Bar_M41jr","Bar_M42al","Bar_M42ar","Bar_M42bl","Bar_M42br","Bar_M42cl","Bar_M42c
r","Bar_M42dl","Bar_M42dr","Bar_M42el","Bar_M42er","Bar_M42fl","Bar_M42fr","Bar_M42gl"
,"Bar_M42gr","Bar_M42hl","Bar_M42hr","Bar_M42il","Bar_M42ir","Bar_M42jl","Bar_M42jr","
Bar_M43al","Bar_M43ar","Bar_M43bl","Bar_M43br","Bar_M43cl","Bar_M43cr","Bar_M43dl","Ba
r_M43dr","Bar_M43el","Bar_M43er","Bar_M43fl","Bar_M43fr","Bar_M43gl","Bar_M43gr","Bar_
M43hl","Bar_M43hr","Bar_M43il","Bar_M43ir","Bar_M43jl","Bar_M43jr" ], 10, -1, True )
setMesherType( [  "Loadplate", "Pressure Layer_1", "Slab_1",
"Koppel_M12a","Koppel_M12b","Koppel_M12c","Koppel_M12d","Koppel_M12e","Koppel_M12f","K
oppel_M12g","Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d","Reinfo_M8e","Reinfo_M
8f","Reinfo_M8g","Reinfo_M8h","Reinfo_M8i","Reinfo_M8j","Reinfo_M8k","Reinfo_M8l","Rei
nfo_M8m","Reinfo_M8n","Reinfo_M8o","Bar_M41al","Bar_M41ar","Bar_M41bl","Bar_M41br","Ba
r_M41cl","Bar_M41cr","Bar_M41dl","Bar_M41dr","Bar_M41el","Bar_M41er","Bar_M41fl","Bar_
M41fr","Bar_M41gl","Bar_M41gr","Bar_M41hl","Bar_M41hr","Bar_M41il","Bar_M41ir","Bar_M4
1jl","Bar_M41jr","Bar_M42al","Bar_M42ar","Bar_M42bl","Bar_M42br","Bar_M42cl","Bar_M42c
r","Bar_M42dl","Bar_M42dr","Bar_M42el","Bar_M42er","Bar_M42fl","Bar_M42fr","Bar_M42gl"
,"Bar_M42gr","Bar_M42hl","Bar_M42hr","Bar_M42il","Bar_M42ir","Bar_M42jl","Bar_M42jr","
Bar_M43al","Bar_M43ar","Bar_M43bl","Bar_M43br","Bar_M43cl","Bar_M43cr","Bar_M43dl","Ba
r_M43dr","Bar_M43el","Bar_M43er","Bar_M43fl","Bar_M43fr","Bar_M43gl","Bar_M43gr","Bar_
M43hl","Bar_M43hr","Bar_M43il","Bar_M43ir","Bar_M43jl","Bar_M43jr" ], "HEXQUAD" )
clearMidSideNodeLocation( [  "Loadplate", "Pressure Layer_1", "Slab_1",
"Koppel_M12a","Koppel_M12b","Koppel_M12c","Koppel_M12d","Koppel_M12e","Koppel_M12f","K
oppel_M12g","Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d","Reinfo_M8e","Reinfo_M
8f","Reinfo_M8g","Reinfo_M8h","Reinfo_M8i","Reinfo_M8j","Reinfo_M8k","Reinfo_M8l","Rei
nfo_M8m","Reinfo_M8n","Reinfo_M8o","Bar_M41al","Bar_M41ar","Bar_M41bl","Bar_M41br","Ba
r_M41cl","Bar_M41cr","Bar_M41dl","Bar_M41dr","Bar_M41el","Bar_M41er","Bar_M41fl","Bar_
M41fr","Bar_M41gl","Bar_M41gr","Bar_M41hl","Bar_M41hr","Bar_M41il","Bar_M41ir","Bar_M4
1jl","Bar_M41jr","Bar_M42al","Bar_M42ar","Bar_M42bl","Bar_M42br","Bar_M42cl","Bar_M42c
r","Bar_M42dl","Bar_M42dr","Bar_M42el","Bar_M42er","Bar_M42fl","Bar_M42fr","Bar_M42gl"
,"Bar_M42gr","Bar_M42hl","Bar_M42hr","Bar_M42il","Bar_M42ir","Bar_M42jl","Bar_M42jr","
Bar_M43al","Bar_M43ar","Bar_M43bl","Bar_M43br","Bar_M43cl","Bar_M43cr","Bar_M43dl","Ba
r_M43dr","Bar_M43el","Bar_M43er","Bar_M43fl","Bar_M43fr","Bar_M43gl","Bar_M43gr","Bar_
M43hl","Bar_M43hr","Bar_M43il","Bar_M43ir","Bar_M43jl","Bar_M43jr" ] )
setElementSize( [ "Support","Pressure Layer", "Slab" ], 50, -1, True )
setMesherType( [ "Support","Pressure Layer", "Slab" ], "HEXQUAD" )
setEdgeGrading( "Pressure Layer", [[ 500, 70, 0 ]], [], 50, 50, 10, 0.25, False,
True )
setEdgeGrading( "Slab", [], [[ 500, 70, 0 ]], 50, 50, 10, 0.25, False, True )
generateMesh( [] )
hideView( "GEOM" )
showView( "MESH" )

#Analysis
addAnalysis( "Analysis1" )
```

```
423 addAnalysisCommand( "Analysis1", "NONLIN", "Structural nonlinear" )
424 renameAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "EXECUT(1)",
    "Load" )
425 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/LOADNR" )
426 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/LOADNR", 1 )
427 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/STEPTY", "AUTOMA" )
428 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/SIZES", 250 )
429 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/MAXSIZ", 0.01 )
430 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/MINSIZ", 0.00001 )
431 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/CUTBCK", 0.25 )
432 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/MAXSTP", 100 )
433 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/ARCLEN", True )
434 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/ARCLEN/REGULA/SET" )
435 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/ARCLEN/REGULA/SET(1)/NODES(1)/RNGNRS", "1-9361" )
436 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/ARCLEN/REGULA/SET(1)/DIRECT", 2 )
437 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/SIZES", "0.1(1000)" )
438 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN", True )
439 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN/REGULA/SET" )
440 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN/REGULA/SET(1)/NODES(1)/RNGNRS", "734 755 756 761
    835-851 940 941-2365(89) 2548-2567 2652 2653 2831 2832 3010 3011 3189 3190 3368 3369
    3547 3548 3726 3727 3905 3906 4084 4085 4263 4264 4442 4443 4621 4622 4800 4801 4979
    4980 5158 5159 5337 5338 5516 5517 5695 5784 5786 5875-5881 5970 5971-6505(89)
    6684-6693 6871 6872 7050 7051 7229 7230 7408 7409 7587 7588 7766 7993-7995 8133 8134
    8272 8273 8411 8412 8550 8551 8689 8690 8828 8829" )
441 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN/REGULA/SET(1)/DIRECT", 2 )
442 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/MAXITE", 250 )
443 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/LINESE", True )
444 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/ENERGY", True )
445 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/DISPLA", False )
446 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/FORCE", True )
447 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/ENERGY/TOLCON", 0.001 )
448 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/ENERGY/NOCONV", "CONTIN" )
449 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/FORCE/TOLCON", 0.01 )
450 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/FORCE/NOCONV", "CONTIN" )
451 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/METHOD/METNAM", "SECANT" )
452
453 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "OUTPUT(1)/SELTYP",
    "USER" )
454 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "OUTPUT(1)/USER" )
455 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/DISPLA(1)/TOTAL" )
456 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/FORCE(1)/REACTI" )
457 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
```

```
    "OUTPUT(1)/USER/STRAIN(1)/TOTAL/GREEN" )
458 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(2)/TOTAL/GREEN/PRINCI" )
459 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(3)/CRKWDT" )
460 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(4)/CRACK/GREEN" )
461 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(5)/CRKWDT/GREEN/PRINCI" )
462 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(1)/TOTAL/CAUCHY" )
463 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(2)/TOTAL/CAUCHY/PRINCI" )
464 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(3)/TOTAL/FORCE/LOCAL" )
465 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(4)/TOTAL/MOMENT/LOCAL" )
466
467 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(1)/TOTAL/GREEN/GLOBAL/LOCATI", "INTPNT" )
468 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(1)/TOTAL/CAUCHY/GLOBAL/LOCATI", "INTPNT" )
469
470 #Run Analysis
471 # runSolver( [] )
472 # showView( "RESULT" )
473 fitAll( )
474
475
476
```

B2 - T01 V11

```
newProject( "D:/Afstuderen/Python/Proefstukken/DIANA Output/T01 V11 Reduced Elements", 10 )


###Project Settings###
setModelAnalysisAspects( [ "STRUCT" ] )
setModelDimension( "2D" )
setDefaultMeshOrder( "QUADRATIC" )
setDefaultMesherType( "HEXQUAD" )
setDefaultMidSideNodeLocation( "ONSHAP" )
setUnit( "LENGTH", "MM" )
setUnit( "FORCE", "N" )
setUnit( "ANGLE", "DEGREE" )


#Create Shapes
createSheet( "Pressure Layer", [[ 0, 70, 0 ],[ 1900, 70, 0 ],[ 1900, 300, 0 ],[ 1150, 300, 0 ],[1050, 300, 0 ],[ 0, 300, 0 ]] )
createSheet( "Slab", [[ 0, 0, 0 ],[ 250, 0, 0 ],[ 350, 0, 0 ],[ 1900, 0, 0 ],[ 1900, 70, 0],[ 0, 70, 0 ]] )

createSheet( "Support", [[ 250, 0, 0 ],[ 350, 0, 0 ],[ 350, -30, 0 ],[ 300, -30, 0 ],[ 250, -30, 0 ]] )
createSheet( "Loadplate", [[1050, 300, 0 ],[ 1150, 300, 0 ],[ 1150, 330, 0 ],[ 1100, 330, 0 ],[ 1050, 330, 0 ]] )

#Create Reinforcement
createLine( "Koppel_M12a", [ 1200, 76, 0 ], [ 1900, 76, 0 ] )
createLine( "Koppel_M12b", [ 1200, 76, 0 ], [ 1900, 76, 0 ] )
createLine( "Koppel_M12c", [ 1200, 76, 0 ], [ 1900, 76, 0 ] )
createLine( "Koppel_M12d", [ 1200, 76, 0 ], [ 1900, 76, 0 ] )
createLine( "Koppel_M12e", [ 1200, 76, 0 ], [ 1900, 76, 0 ] )
createLine( "Koppel_M12f", [ 1200, 76, 0 ], [ 1900, 76, 0 ] )
createLine( "Koppel_M12g", [ 1200, 76, 0 ], [ 1900, 76, 0 ] )

createLine( "Reinfo_M8a", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8b", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8c", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8d", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8e", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8f", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8g", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8h", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8i", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8j", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8k", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8l", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8m", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8n", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8o", [ 0, 35, 0 ], [ 1865, 35, 0 ] )

createLine( "Bar_M41al", [ 135, 45.5, 0],[ 175, 235.5, 0] )
createLine( "Bar_M41ar", [ 215, 45.5, 0],[ 175, 235.5, 0] )
createLine( "Bar_M41bl", [ 135, 45.5, 0],[ 175, 235.5, 0] )
createLine( "Bar_M41br", [ 215, 45.5, 0],[ 175, 235.5, 0] )
createLine( "Bar_M41cl", [ 135, 45.5, 0],[ 175, 235.5, 0] )
createLine( "Bar_M41cr", [ 215, 45.5, 0],[ 175, 235.5, 0] )
createLine( "Bar_M41dl", [ 135, 45.5, 0],[ 175, 235.5, 0] )
createLine( "Bar_M41dr", [ 215, 45.5, 0],[ 175, 235.5, 0] )
createLine( "Bar_M41el", [ 135, 45.5, 0],[ 175, 235.5, 0] )
createLine( "Bar_M41er", [ 215, 45.5, 0],[ 175, 235.5, 0] )
createLine( "Bar_M41fl", [ 135, 45.5, 0],[ 175, 235.5, 0] )
createLine( "Bar_M41fr", [ 215, 45.5, 0],[ 175, 235.5, 0] )
createLine( "Bar_M41gl", [ 135, 45.5, 0],[ 175, 235.5, 0] )
createLine( "Bar_M41gr", [ 215, 45.5, 0],[ 175, 235.5, 0] )
createLine( "Bar_M41hl", [ 135, 45.5, 0],[ 175, 235.5, 0] )
createLine( "Bar_M41hr", [ 215, 45.5, 0],[ 175, 235.5, 0] )
createLine( "Bar_M41il", [ 135, 45.5, 0],[ 175, 235.5, 0] )
createLine( "Bar_M41ir", [ 215, 45.5, 0],[ 175, 235.5, 0] )
createLine( "Bar_M41jl", [ 135, 45.5, 0],[ 175, 235.5, 0] )
createLine( "Bar_M41jr", [ 215, 45.5, 0],[ 175, 235.5, 0] )
```

```
67
68  createLine( "Bar_M42al", [ 735, 45.5, 0],[ 775, 235.5, 0] )
69  createLine( "Bar_M42ar", [ 815, 45.5, 0],[ 775, 235.5, 0] )
70  createLine( "Bar_M42bl", [ 735, 45.5, 0],[ 775, 235.5, 0] )
71  createLine( "Bar_M42br", [ 815, 45.5, 0],[ 775, 235.5, 0] )
72  createLine( "Bar_M42cl", [ 735, 45.5, 0],[ 775, 235.5, 0] )
73  createLine( "Bar_M42cr", [ 815, 45.5, 0],[ 775, 235.5, 0] )
74  createLine( "Bar_M42dl", [ 735, 45.5, 0],[ 775, 235.5, 0] )
75  createLine( "Bar_M42dr", [ 815, 45.5, 0],[ 775, 235.5, 0] )
76  createLine( "Bar_M42el", [ 735, 45.5, 0],[ 775, 235.5, 0] )
77  createLine( "Bar_M42er", [ 815, 45.5, 0],[ 775, 235.5, 0] )
78  createLine( "Bar_M42fl", [ 735, 45.5, 0],[ 775, 235.5, 0] )
79  createLine( "Bar_M42fr", [ 815, 45.5, 0],[ 775, 235.5, 0] )
80  createLine( "Bar_M42gl", [ 735, 45.5, 0],[ 775, 235.5, 0] )
81  createLine( "Bar_M42gr", [ 815, 45.5, 0],[ 775, 235.5, 0] )
82  createLine( "Bar_M42hl", [ 735, 45.5, 0],[ 775, 235.5, 0] )
83  createLine( "Bar_M42hr", [ 815, 45.5, 0],[ 775, 235.5, 0] )
84  createLine( "Bar_M42il", [ 735, 45.5, 0],[ 775, 235.5, 0] )
85  createLine( "Bar_M42ir", [ 815, 45.5, 0],[ 775, 235.5, 0] )
86  createLine( "Bar_M42jl", [ 735, 45.5, 0],[ 775, 235.5, 0] )
87  createLine( "Bar_M42jr", [ 815, 45.5, 0],[ 775, 235.5, 0] )
88
89  createLine( "Bar_M43al", [ 1460, 45.5, 0],[ 1500, 235.5, 0] )
90  createLine( "Bar_M43ar", [ 1540, 45.5, 0],[ 1500, 235.5, 0] )
91  createLine( "Bar_M43bl", [ 1460, 45.5, 0],[ 1500, 235.5, 0] )
92  createLine( "Bar_M43br", [ 1540, 45.5, 0],[ 1500, 235.5, 0] )
93  createLine( "Bar_M43cl", [ 1460, 45.5, 0],[ 1500, 235.5, 0] )
94  createLine( "Bar_M43cr", [ 1540, 45.5, 0],[ 1500, 235.5, 0] )
95  createLine( "Bar_M43dl", [ 1460, 45.5, 0],[ 1500, 235.5, 0] )
96  createLine( "Bar_M43dr", [ 1540, 45.5, 0],[ 1500, 235.5, 0] )
97  createLine( "Bar_M43el", [ 1460, 45.5, 0],[ 1500, 235.5, 0] )
98  createLine( "Bar_M43er", [ 1540, 45.5, 0],[ 1500, 235.5, 0] )
99  createLine( "Bar_M43fl", [ 1460, 45.5, 0],[ 1500, 235.5, 0] )
100 createLine( "Bar_M43fr", [ 1540, 45.5, 0],[ 1500, 235.5, 0] )
101 createLine( "Bar_M43gl", [ 1460, 45.5, 0],[ 1500, 235.5, 0] )
102 createLine( "Bar_M43gr", [ 1540, 45.5, 0],[ 1500, 235.5, 0] )
103 createLine( "Bar_M43hl", [ 1460, 45.5, 0],[ 1500, 235.5, 0] )
104 createLine( "Bar_M43hr", [ 1540, 45.5, 0],[ 1500, 235.5, 0] )
105 createLine( "Bar_M43il", [ 1460, 45.5, 0],[ 1500, 235.5, 0] )
106 createLine( "Bar_M43ir", [ 1540, 45.5, 0],[ 1500, 235.5, 0] )
107 createLine( "Bar_M43jl", [ 1460, 45.5, 0],[ 1500, 235.5, 0] )
108 createLine( "Bar_M43jr", [ 1540, 45.5, 0],[ 1500, 235.5, 0] )
109
110 createLine( "Composed Line", [ 0, 150, 0 ], [ 1900, 150, 0 ] )
111
112 createSheet( "Mesh Sheet", [[ 1000, 0, 0 ],[ 1000, 300, 0 ],[ 1000, 300, 1000 ],[
    1000, 0, 1000 ]] )
113
114 #Splitting Mesh Parts
115 subtract( "Pressure Layer", [ "Mesh Sheet" ], True, True )
116 subtract( "Slab", [ "Mesh Sheet" ], False, True )
117
118 #Regroup Material Parts
119 addSet( "SHAPESET", "Coupling Reinforcement" )
120 moveToShapeSet( [
    "Koppel_M12a","Koppel_M12b","Koppel_M12c","Koppel_M12d","Koppel_M12e","Koppel_M12f","K
    oppel_M12g" ], "Coupling Reinforcement" )
121 addSet( "SHAPESET", "Longitudinal Reinforcement" )
122 moveToShapeSet( [
    "Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d","Reinfo_M8e","Reinfo_M8f","Reinfo_
    M8g","Reinfo_M8h","Reinfo_M8i","Reinfo_M8j","Reinfo_M8k","Reinfo_M8l","Reinfo_M8m","Re
    info_M8n","Reinfo_M8o" ], "Longitudinal Reinforcement" )
123 addSet( "SHAPESET", "Lattice girders" )
124 moveToShapeSet( [
    "Bar_M41al","Bar_M41ar","Bar_M41bl","Bar_M41br","Bar_M41cl","Bar_M41cr","Bar_M41dl","B
    ar_M41dr","Bar_M41el","Bar_M41er","Bar_M41fl","Bar_M41fr","Bar_M41gl","Bar_M41gr","Bar
    _M41hl","Bar_M41hr","Bar_M41il","Bar_M41ir","Bar_M41jl","Bar_M41jr","Bar_M42al","Bar_M
    42ar","Bar_M42bl","Bar_M42br","Bar_M42cl","Bar_M42cr","Bar_M42dl","Bar_M42dr","Bar_M42
    el","Bar_M42er","Bar_M42fl","Bar_M42fr","Bar_M42gl","Bar_M42gr","Bar_M42hl","Bar_M42hr
    ","Bar_M42il","Bar_M42ir","Bar_M42jl","Bar_M42jr","Bar_M43al","Bar_M43ar","Bar_M43bl",
    "Bar_M43br","Bar_M43cl","Bar_M43cr","Bar_M43dl","Bar_M43dr","Bar_M43el","Bar_M43er","B
```

```
    ar_M43fl","Bar_M43fr","Bar_M43gl","Bar_M43gr","Bar_M43hl","Bar_M43hr","Bar_M43il","Bar
    _M43ir","Bar_M43jl","Bar_M43jr" ], "Lattice girders" )
125 addSet( "SHAPESET", "Steel plates" )
126 moveToShapeSet( [ "Loadplate", "Support" ], "Steel plates" )
127 rename( "SHAPESET", "Shapes", "Concrete" )
128 addSet( "SHAPESET", "Composed Line" )
129 moveToShapeSet( [ "Composed Line" ], "Composed Line" )
130
131 #Material Precast concrete
132 addMaterial( "Precast concrete", "CONCR", "TSCR", [] )
133 setParameter( "MATERIAL", "Precast concrete", "LINEAR/ELASTI/YOUNG", 36874 )
134 setParameter( "MATERIAL", "Precast concrete", "LINEAR/ELASTI/POISON", 0.2 )
135 setParameter( "MATERIAL", "Precast concrete", "LINEAR/MASS/DENSIT", 2.4e-09 )
136 setParameter( "MATERIAL", "Precast concrete", "MODTYP/TOTCRK", "ROTATE" )
137 setParameter( "MATERIAL", "Precast concrete", "TENSIL/TENCRV", "HORDYK" )
138 setParameter( "MATERIAL", "Precast concrete", "TENSIL/TENSTR", 3.96 )
139 setParameter( "MATERIAL", "Precast concrete", "TENSIL/GF1", 0.1506 )
140 setParameter( "MATERIAL", "Precast concrete", "TENSIL/CBSPEC", "ROTS" )
141 setParameter( "MATERIAL", "Precast concrete", "TENSIL/POISRE/POIRED", "DAMAGE" )
142 # setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMCRV", "MC2010" )
143 # setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMSTR", 55.9 )
144 # setParameter( "MATERIAL", "Precast concrete", "COMPRS/EPSC1", 0.0024 )
145 # setParameter( "MATERIAL", "Precast concrete", "COMPRS/EPSCU", 0.0035 )
146 # setParameter( "MATERIAL", "Precast concrete", "COMPRS/CONFIN/CNFCRV", "NONE" )
147 setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMCRV", "CONSTA" )
148 setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMSTR", 55.9 )
149
150 addGeometry( "Precast concrete", "SHEET", "MEMBRA", [] )
151 setParameter( "GEOMET", "Precast concrete", "THICK", 1000 )
152 setParameter( "GEOMET", "Precast concrete", "LOCAXS", True )
153 setParameter( "GEOMET", "Precast concrete", "LOCAXS/XAXIS", [ 1, 0, 0 ] )
154
155 addElementData( "Precast concrete" )
156 setElementClassType( "SHAPE", [ "Slab", "Slab_1" ], "MEMBRA" )
157 assignMaterial( "Precast concrete", "SHAPE", [ "Slab", "Slab_1" ] )
158 assignGeometry( "Precast concrete", "SHAPE", [ "Slab", "Slab_1" ] )
159 assignElementData( "Precast concrete", "SHAPE", [ "Slab", "Slab_1" ] )
160
161 # setParameter( "DATA", "Precast concrete", "INTEGR", "REDUCE" )
162
163 #Material Pressure layer
164 addMaterial( "Pressure layer", "CONCR", "TSCR", [] )
165 setParameter( "MATERIAL", "Pressure layer", "LINEAR/ELASTI/YOUNG", 32198 )
166 setParameter( "MATERIAL", "Pressure layer", "LINEAR/ELASTI/POISON", 0.2 )
167 setParameter( "MATERIAL", "Pressure layer", "LINEAR/MASS/DENSIT", 2.4e-09 )
168 setParameter( "MATERIAL", "Pressure layer", "MODTYP/TOTCRK", "ROTATE" )
169 setParameter( "MATERIAL", "Pressure layer", "TENSIL/TENCRV", "HORDYK" )
170 setParameter( "MATERIAL", "Pressure layer", "TENSIL/TENSTR", 2.74 )
171 setParameter( "MATERIAL", "Pressure layer", "TENSIL/GF1", 0.1389 )
172 setParameter( "MATERIAL", "Pressure layer", "TENSIL/CBSPEC", "ROTS" )
173 setParameter( "MATERIAL", "Pressure layer", "TENSIL/POISRE/POIRED", "DAMAGE" )
174 # setParameter( "MATERIAL", "Pressure layer", "COMPRS/COMCRV", "MC2010" )
175 # setParameter( "MATERIAL", "Pressure layer", "COMPRS/COMSTR", 35.6 )
176 # setParameter( "MATERIAL", "Pressure layer", "COMPRS/EPSC1", 0.0021 )
177 # setParameter( "MATERIAL", "Pressure layer", "COMPRS/EPSCU", 0.0035 )
178 # setParameter( "MATERIAL", "Pressure layer", "COMPRS/CONFIN/CNFCRV", "NONE" )
179 setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMCRV", "CONSTA" )
180 setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMSTR", 35.6 )
181
182 addGeometry( "Pressure layer", "SHEET", "MEMBRA", [] )
183 setParameter( "GEOMET", "Pressure layer", "THICK", 1000 )
184 setParameter( "GEOMET", "Pressure layer", "LOCAXS", True )
185 setParameter( "GEOMET", "Pressure layer", "LOCAXS/XAXIS", [ 1, 0, 0 ] )
186
187 addElementData( "Pressure layer" )
188 setElementClassType( "SHAPE", [ "Pressure Layer", "Pressure Layer_1" ], "MEMBRA" )
189 assignMaterial( "Pressure layer", "SHAPE", [ "Pressure Layer", "Pressure Layer_1" ] )
190 assignGeometry( "Pressure layer", "SHAPE", [ "Pressure Layer", "Pressure Layer_1" ] )
191 assignElementData( "Pressure layer", "SHAPE", [ "Pressure Layer", "Pressure Layer_1"
    ] )
192
```

```
193  # setParameter( "DATA", "Pressure layer", "INTEGR", "REDUCE" )
194
195  #Material Steel
196  addMaterial( "Steel", "MCSTEL", "ISOTRO", [] )
197  setParameter( "MATERIAL", "Steel", "LINEAR/ELASTI/YOUNG", 210000 )
198  setParameter( "MATERIAL", "Steel", "LINEAR/ELASTI/POISON", 0.3 )
199  setParameter( "MATERIAL", "Steel", "LINEAR/MASS/DENSIT", 0 )
200  addGeometry( "Plate", "SHEET", "MEMBRA", [] )
201  setParameter( "GEOMET", "Plate", "THICK", 1000 )
202  addElementData( "Plate" )
203
204  setElementClassType( "SHAPE", [ "Support", "Loadplate" ], "MEMBRA" )
205  assignMaterial( "Steel", "SHAPE", [ "Support", "Loadplate" ] )
206  assignGeometry( "Plate", "SHAPE", [ "Support", "Loadplate" ] )
207  assignElementData( "Plate", "SHAPE", [ "Support", "Loadplate" ] )
208  # setParameter( "DATA", "Plate", "INTEGR", "REDUCE" )
209
210  #Concrete Interface
211  # addMaterial( "Interface Concrete", "INTERF", "FRICTI", [] )
212  # setParameter( "MATERIAL", "Interface Concrete", "LINEAR/IFTYP", "LIN2D" )
213  # setParameter( "MATERIAL", "Interface Concrete", "COULOM/OPNTYP", "GAP" )
214  # setParameter( "MATERIAL", "Interface Concrete", "COULOM/GAPVAL", 0.5 )
215  # setParameter( "MATERIAL", "Interface Concrete", "COULOM/COHESI", 0.29 )
216  # setParameter( "MATERIAL", "Interface Concrete", "COULOM/PHI", 30 )
217  # setParameter( "MATERIAL", "Interface Concrete", "COULOM/PSI", 5.71 )
218  # setParameter( "MATERIAL", "Interface Concrete", "LINEAR/ELAS2/DSNY", 1000 )
219  # setParameter( "MATERIAL", "Interface Concrete", "LINEAR/ELAS2/DSSX", 100 )
220  # setParameter( "MATERIAL", "Interface Concrete", "COULOM/FRCHSR/MODE2", 0 )
221
222  addMaterial( "Interface Concrete", "INTERF", "FRICEL", [] )
223  setParameter( "MATERIAL", "Interface Concrete", "LINEAR/IFTYP", "LIN2D" )
224  setParameter( "MATERIAL", "Interface Concrete", "NEFRIC/SPCTYP", "USER" )
225  setParameter( "MATERIAL", "Interface Concrete", "NEFRIC/COHESI", 0.2 )
226  setParameter( "MATERIAL", "Interface Concrete", "NEFRIC/PHI", 22 )
227  setParameter( "MATERIAL", "Interface Concrete", "LINEAR/ELAS2/DSNY", 100 )
228  setParameter( "MATERIAL", "Interface Concrete", "LINEAR/ELAS2/DSSX", 100 )
229
230  addGeometry( "Interface Concrete", "LINE", "STLIIF", [] )
231  setParameter( "GEOMET", "Interface Concrete", "LIFMEM/THICK", 1000 )
232  addElementData( "Interface Concrete" )
233  createConnection( "Interface Concrete", "INTER", "SHAPEEDGE", "SHAPEEDGE" )
234  setParameter( "GEOMETRYCONNECTION", "Interface Concrete", "MODE", "CLOSED" )
235  setElementClassType( "GEOMETRYCONNECTION", "Interface Concrete", "STLIIF" )
236  assignMaterial( "Interface Concrete", "GEOMETRYCONNECTION", "Interface Concrete" )
237  assignGeometry( "Interface Concrete", "GEOMETRYCONNECTION", "Interface Concrete" )
238  assignElementData( "Interface Concrete", "GEOMETRYCONNECTION", "Interface Concrete" )
239  setParameter( "GEOMETRYCONNECTION", "Interface Concrete", "FLIP", False )
240  attachTo( "GEOMETRYCONNECTION", "Interface Concrete", "SOURCE", "Pressure Layer", [[
     500, 70, 0 ]] )
241  attachTo( "GEOMETRYCONNECTION", "Interface Concrete", "TARGET", "Slab", [[ 500, 70,
     0 ]] )
242  attachTo( "GEOMETRYCONNECTION", "Interface Concrete", "SOURCE", "Pressure Layer_1",
     [[ 1450, 70, 0 ]] )
243  attachTo( "GEOMETRYCONNECTION", "Interface Concrete", "TARGET", "Slab_1", [[ 1450,
     70, 0 ]] )
244
245  # setParameter( "DATA", "Interface Concrete", "INTEGR", "REDUCE" )
246
247  #Steel Interface
248  addMaterial( "Interface Steel", "INTERF", "NONLIF", [] )
249  setParameter( "MATERIAL", "Interface Steel", "LINEAR/IFTYP", "LIN2D" )
250  setParameter( "MATERIAL", "Interface Steel", "LINEAR/ELAS2/DSNY", 32000 )
251  setParameter( "MATERIAL", "Interface Steel", "LINEAR/ELAS2/DSSX", 0.00032 )
252  setParameter( "MATERIAL", "Interface Steel", "NONLIN/IFNOTE", "NOTSHR" )
253  setParameter( "MATERIAL", "Interface Steel", "NONLIN/NLEL8/NOTENS", [ 0.001, 0 ] )
254  setParameter( "MATERIAL", "Interface Steel", "NONLIN/NLEL8/NOSHTE", [ 0.001, 0 ] )
255  addGeometry( "Interface Steel", "LINE", "STLIIF", [] )
256  setParameter( "GEOMET", "Interface Steel", "LIFMEM/THICK", 1000 )
257  addElementData( "Interface Steel" )
258  createConnection( "Interface Steel", "INTER", "SHAPEEDGE", "SHAPEEDGE" )
259
```

```
setParameter( "GEOMETRYCONNECTION", "Interface Steel", "CONTYP", "INTER" )
setParameter( "GEOMETRYCONNECTION", "Interface Steel", "MODE", "AUTO" )
setElementClassType( "GEOMETRYCONNECTION", "Interface Steel", "STLIIF" )
assignMaterial( "Interface Steel", "GEOMETRYCONNECTION", "Interface Steel" )
assignGeometry( "Interface Steel", "GEOMETRYCONNECTION", "Interface Steel" )
assignElementData( "Interface Steel", "GEOMETRYCONNECTION", "Interface Steel" )
setParameter( "GEOMETRYCONNECTION", "Interface Steel", "FLIP", False )
attachTo( "GEOMETRYCONNECTION", "Interface Steel", "SOURCE", "Pressure Layer_1", [[ 1100, 300, 0 ]] )
attachTo( "GEOMETRYCONNECTION", "Interface Steel", "TARGET", "Loadplate", [[ 1100, 300, 0 ]] )
attachTo( "GEOMETRYCONNECTION", "Interface Steel", "SOURCE", "Slab", [[ 300, 0, 0 ]] )
attachTo( "GEOMETRYCONNECTION", "Interface Steel", "TARGET", "Support", [[ 300, 0, 0 ]] )

# setParameter( "DATA", "Interface Steel", "INTEGR", "REDUCE" )
# setParameter( "DATA", "Interface Steel", "NUMINT", [ "LOBATT" ] )

#Material Reinforcement Koppel_M12
addMaterial( "Koppel_M12", "REINFO", "REBOND", [ ] )
setParameter( "MATERIAL", "Koppel_M12", "REBARS/ELASTI/YOUNG", 200000 )
setParameter( "MATERIAL", "Koppel_M12", "REBARS/POISON/POISON", 0.3 )
setParameter( "MATERIAL", "Koppel_M12", "REBARS/PLATYP", "VMISES" )
setParameter( "MATERIAL", "Koppel_M12", "REBARS/PLASTI/TRESSH", "KAPSIG" )
setParameter( "MATERIAL", "Koppel_M12", "REBARS/PLASTI/KAPSIG", [ 0, 550, 0.04725, 594 ] )
setParameter( "MATERIAL", "Koppel_M12", "RESLIP/SHFTYP", "BONDS6" )
setParameter( "MATERIAL", "Koppel_M12", "RESLIP/DSNY", 1000 )
setParameter( "MATERIAL", "Koppel_M12", "RESLIP/DSSX", 100 )
setParameter( "MATERIAL", "Koppel_M12", "RESLIP/BONDS6/SLPVAL", [ 15.411, 6.164, 0.1, 1, 2, 10, 0.4 ] )
addGeometry( "Koppel_M12", "RELINE", "REBAR", [] )
setParameter( "GEOMET", "Koppel_M12", "REITYP", "REITRU" )
setParameter( "GEOMET", "Koppel_M12", "REITYP", "CIRBEA" )
setParameter( "GEOMET", "Koppel_M12", "CIRBEA/CIRCLE", 12)
setParameter( "GEOMET", "Koppel_M12", "TIPELM/SURFAC", 113.1 )
assignMaterial( "Koppel_M12", "SHAPE", [ "Koppel_M12a","Koppel_M12b","Koppel_M12c","Koppel_M12d","Koppel_M12e","Koppel_M12f","Koppel_M12g" ] )
assignGeometry( "Koppel_M12", "SHAPE", [ "Koppel_M12a","Koppel_M12b","Koppel_M12c","Koppel_M12d","Koppel_M12e","Koppel_M12f","Koppel_M12g" ] )
convertToReinforcement( [ "Koppel_M12a","Koppel_M12b","Koppel_M12c","Koppel_M12d","Koppel_M12e","Koppel_M12f","Koppel_M12g" ] )
addElementData( "Koppel_M12" )
assignElementData( "Koppel_M12", "SHAPE", [ "Koppel_M12a","Koppel_M12b","Koppel_M12c","Koppel_M12d","Koppel_M12e","Koppel_M12f","Koppel_M12g" ] )
setParameter( "DATA", "Koppel_M12", "INTERF", "BEAM" )
setReinforcementDiscretization( [ "Koppel_M12a","Koppel_M12b","Koppel_M12c","Koppel_M12d","Koppel_M12e","Koppel_M12f","Koppel_M12g" ], "ELEMENT" )
setContinuousInInterfaces( [ "Koppel_M12a","Koppel_M12b","Koppel_M12c","Koppel_M12d","Koppel_M12e","Koppel_M12f","Koppel_M12g" ], False )

#Material Reinforcement M8
addMaterial( "Reinfo_M8", "REINFO", "REBOND", [] )
setParameter( "MATERIAL", "Reinfo_M8", "REBARS/ELASTI/YOUNG", 200000 )
setParameter( "MATERIAL", "Reinfo_M8", "REBARS/POISON/POISON", 0.3 )
setParameter( "MATERIAL", "Reinfo_M8", "REBARS/PLATYP", "VMISES" )
setParameter( "MATERIAL", "Reinfo_M8", "REBARS/PLASTI/TRESSH", "KAPSIG" )
setParameter( "MATERIAL", "Reinfo_M8", "REBARS/PLASTI/KAPSIG", [ 0, 550, 0.04725, 594 ] )
setParameter( "MATERIAL", "Reinfo_M8", "RESLIP/SHFTYP", "BONDS6" )
setParameter( "MATERIAL", "Reinfo_M8", "RESLIP/DSNY", 1000 )
setParameter( "MATERIAL", "Reinfo_M8", "RESLIP/DSSX", 100 )
setParameter( "MATERIAL", "Reinfo_M8", "RESLIP/BONDS6/SLPVAL", [ 15.411, 6.164, 0.1, 1, 2, 10, 0.4 ] )
addGeometry( "Reinfo_M8", "RELINE", "REBAR", [] )
```

```
312 setParameter( "GEOMET", "Reinfo_M8", "REITYP", "REITRU" )
313 setParameter( "GEOMET", "Reinfo_M8", "REITYP", "CIRBEA" )
314 setParameter( "GEOMET", "Reinfo_M8", "CIRBEA/CIRCLE", 8)
315 setParameter( "GEOMET", "Reinfo_M8", "TIPELM/SURFAC", 50.3 )
316 assignMaterial( "Reinfo_M8", "SHAPE", [
    "Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d","Reinfo_M8e","Reinfo_M8f","Reinfo_
    M8g","Reinfo_M8h","Reinfo_M8i","Reinfo_M8j","Reinfo_M8k","Reinfo_M8l","Reinfo_M8m","Re
    info_M8n","Reinfo_M8o" ] )
317 assignGeometry( "Reinfo_M8", "SHAPE", [
    "Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d","Reinfo_M8e","Reinfo_M8f","Reinfo_
    M8g","Reinfo_M8h","Reinfo_M8i","Reinfo_M8j","Reinfo_M8k","Reinfo_M8l","Reinfo_M8m","Re
    info_M8n","Reinfo_M8o" ] )
318 convertToReinforcement( [
    "Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d","Reinfo_M8e","Reinfo_M8f","Reinfo_
    M8g","Reinfo_M8h","Reinfo_M8i","Reinfo_M8j","Reinfo_M8k","Reinfo_M8l","Reinfo_M8m","Re
    info_M8n","Reinfo_M8o" ] )
319 addElementData( "Reinfo_M8" )
320 assignElementData( "Reinfo_M8", "SHAPE", [
    "Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d","Reinfo_M8e","Reinfo_M8f","Reinfo_
    M8g","Reinfo_M8h","Reinfo_M8i","Reinfo_M8j","Reinfo_M8k","Reinfo_M8l","Reinfo_M8m","Re
    info_M8n","Reinfo_M8o" ] )
321 setParameter( "DATA", "Reinfo_M8", "INTERF", "BEAM" )
322 setReinforcementDiscretization( [
    "Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d","Reinfo_M8e","Reinfo_M8f","Reinfo_
    M8g","Reinfo_M8h","Reinfo_M8i","Reinfo_M8j","Reinfo_M8k","Reinfo_M8l","Reinfo_M8m","Re
    info_M8n","Reinfo_M8o" ], "ELEMENT" )
323 setContinuousInInterfaces( [
    "Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d","Reinfo_M8e","Reinfo_M8f","Reinfo_
    M8g","Reinfo_M8h","Reinfo_M8i","Reinfo_M8j","Reinfo_M8k","Reinfo_M8l","Reinfo_M8m","Re
    info_M8n","Reinfo_M8o" ], False )
324
325 #Material Reinforcement M4.5
326 addMaterial( "Reinfo_M4.5", "REINFO", "REBOND", [ "ANCHOR" ] )
327 setParameter( "MATERIAL", "Reinfo_M4.5", "REBARS/ELASTI/YOUNG", 200000 )
328 setParameter( "MATERIAL", "Reinfo_M4.5", "REBARS/POISON/POISON", 0.3 )
329 setParameter( "MATERIAL", "Reinfo_M4.5", "REBARS/PLATYP", "VMISES" )
330 setParameter( "MATERIAL", "Reinfo_M4.5", "REBARS/PLASTI/TRESSH", "KAPSIG" )
331 setParameter( "MATERIAL", "Reinfo_M4.5", "REBARS/PLASTI/KAPSIG", [ 0, 550, 0.04725,
    594 ] )
332 setParameter( "MATERIAL", "Reinfo_M4.5", "RESLIP/SHFTYP", "BONDS6" )
333 setParameter( "MATERIAL", "Reinfo_M4.5", "RESLIP/DSNY", 1000 )
334 setParameter( "MATERIAL", "Reinfo_M4.5", "RESLIP/DSSX", 100 )
335
336 setParameter( "MATERIAL", "Reinfo_M4.5", "RESLIP/BONDS6/SLPVAL", [ 7.706, 3.082,
    0.1, 1.8, 3.2, 10, 0.4 ] )
337
338 setParameter( "MATERIAL", "Reinfo_M4.5", "REANCH/DTIPNX", 10000 )
339 setParameter( "MATERIAL", "Reinfo_M4.5", "REANCH/DTIPSY", 10000 )
340 setParameter( "MATERIAL", "Reinfo_M4.5", "REANCH/TIPLOC", "BOTH" )
341
342 addGeometry( "Reinfo_M4.5", "RELINE", "REBAR", [] )
343 setParameter( "GEOMET", "Reinfo_M4.5", "REITYP", "REITRU" )
344 setParameter( "GEOMET", "Reinfo_M4.5", "REITYP", "CIRBEA" )
345 setParameter( "GEOMET", "Reinfo_M4.5", "CIRBEA/CIRCLE", 4.5 )
346 setParameter( "GEOMET", "Reinfo_M4.5", "TIPELM/SURFAC", 15.9 )
347 assignMaterial( "Reinfo_M4.5", "SHAPE", [
    "Bar_M41al","Bar_M41ar","Bar_M41bl","Bar_M41br","Bar_M41cl","Bar_M41cr","Bar_M41dl","B
    ar_M41dr","Bar_M41el","Bar_M41er","Bar_M41fl","Bar_M41fr","Bar_M41gl","Bar_M41gr","Bar
    _M41hl","Bar_M41hr","Bar_M41il","Bar_M41ir","Bar_M41jl","Bar_M41jr","Bar_M42al","Bar_M
    42ar","Bar_M42bl","Bar_M42br","Bar_M42cl","Bar_M42cr","Bar_M42dl","Bar_M42dr","Bar_M42
    el","Bar_M42er","Bar_M42fl","Bar_M42fr","Bar_M42gl","Bar_M42gr","Bar_M42hl","Bar_M42hr
    ","Bar_M42il","Bar_M42ir","Bar_M42jl","Bar_M42jr","Bar_M43al","Bar_M43ar","Bar_M43bl",
    "Bar_M43br","Bar_M43cl","Bar_M43cr","Bar_M43dl","Bar_M43dr","Bar_M43el","Bar_M43er","B
    ar_M43fl","Bar_M43fr","Bar_M43gl","Bar_M43gr","Bar_M43hl","Bar_M43hr","Bar_M43il","Bar
    _M43ir","Bar_M43jl","Bar_M43jr" ] )
348 assignGeometry( "Reinfo_M4.5", "SHAPE", [
    "Bar_M41al","Bar_M41ar","Bar_M41bl","Bar_M41br","Bar_M41cl","Bar_M41cr","Bar_M41dl","B
    ar_M41dr","Bar_M41el","Bar_M41er","Bar_M41fl","Bar_M41fr","Bar_M41gl","Bar_M41gr","Bar
    _M41hl","Bar_M41hr","Bar_M41il","Bar_M41ir","Bar_M41jl","Bar_M41jr","Bar_M42al","Bar_M
    42ar","Bar_M42bl","Bar_M42br","Bar_M42cl","Bar_M42cr","Bar_M42dl","Bar_M42dr","Bar_M42
    el","Bar_M42er","Bar_M42fl","Bar_M42fr","Bar_M42gl","Bar_M42gr","Bar_M42hl","Bar_M42hr
```

```
","Bar_M42il","Bar_M42ir","Bar_M42jl","Bar_M42jr","Bar_M43al","Bar_M43ar","Bar_M43bl",
"Bar_M43br","Bar_M43cl","Bar_M43cr","Bar_M43dl","Bar_M43dr","Bar_M43el","Bar_M43er","B
ar_M43fl","Bar_M43fr","Bar_M43gl","Bar_M43gr","Bar_M43hl","Bar_M43hr","Bar_M43il","Bar
_M43ir","Bar_M43jl","Bar_M43jr" ] )
convertToReinforcement( [
"Bar_M41al","Bar_M41ar","Bar_M41bl","Bar_M41br","Bar_M41cl","Bar_M41cr","Bar_M41dl","B
ar_M41dr","Bar_M41el","Bar_M41er","Bar_M41fl","Bar_M41fr","Bar_M41gl","Bar_M41gr","Bar
_M41hl","Bar_M41hr","Bar_M41il","Bar_M41ir","Bar_M41jl","Bar_M41jr","Bar_M42al","Bar_M
42ar","Bar_M42bl","Bar_M42br","Bar_M42cl","Bar_M42cr","Bar_M42dl","Bar_M42dr","Bar_M42
el","Bar_M42er","Bar_M42fl","Bar_M42fr","Bar_M42gl","Bar_M42gr","Bar_M42hl","Bar_M42hr
","Bar_M42il","Bar_M42ir","Bar_M42jl","Bar_M42jr","Bar_M43al","Bar_M43ar","Bar_M43bl",
"Bar_M43br","Bar_M43cl","Bar_M43cr","Bar_M43dl","Bar_M43dr","Bar_M43el","Bar_M43er","B
ar_M43fl","Bar_M43fr","Bar_M43gl","Bar_M43gr","Bar_M43hl","Bar_M43hr","Bar_M43il","Bar
_M43ir","Bar_M43jl","Bar_M43jr" ] )
addElementData( "Bar" )
assignElementData( "Bar", "SHAPE", [
"Bar_M41al","Bar_M41ar","Bar_M41bl","Bar_M41br","Bar_M41cl","Bar_M41cr","Bar_M41dl","B
ar_M41dr","Bar_M41el","Bar_M41er","Bar_M41fl","Bar_M41fr","Bar_M41gl","Bar_M41gr","Bar
_M41hl","Bar_M41hr","Bar_M41il","Bar_M41ir","Bar_M41jl","Bar_M41jr","Bar_M42al","Bar_M
42ar","Bar_M42bl","Bar_M42br","Bar_M42cl","Bar_M42cr","Bar_M42dl","Bar_M42dr","Bar_M42
el","Bar_M42er","Bar_M42fl","Bar_M42fr","Bar_M42gl","Bar_M42gr","Bar_M42hl","Bar_M42hr
","Bar_M42il","Bar_M42ir","Bar_M42jl","Bar_M42jr","Bar_M43al","Bar_M43ar","Bar_M43bl",
"Bar_M43br","Bar_M43cl","Bar_M43cr","Bar_M43dl","Bar_M43dr","Bar_M43el","Bar_M43er","B
ar_M43fl","Bar_M43fr","Bar_M43gl","Bar_M43gr","Bar_M43hl","Bar_M43hr","Bar_M43il","Bar
_M43ir","Bar_M43jl","Bar_M43jr" ] )
setParameter( "DATA", "Bar", "./INTERF", [] )
setParameter( "DATA", "Bar", "INTERF", "BEAM" )
setReinforcementDiscretization( [
"Bar_M41al","Bar_M41ar","Bar_M41bl","Bar_M41br","Bar_M41cl","Bar_M41cr","Bar_M41dl","B
ar_M41dr","Bar_M41el","Bar_M41er","Bar_M41fl","Bar_M41fr","Bar_M41gl","Bar_M41gr","Bar
_M41hl","Bar_M41hr","Bar_M41il","Bar_M41ir","Bar_M41jl","Bar_M41jr","Bar_M42al","Bar_M
42ar","Bar_M42bl","Bar_M42br","Bar_M42cl","Bar_M42cr","Bar_M42dl","Bar_M42dr","Bar_M42
el","Bar_M42er","Bar_M42fl","Bar_M42fr","Bar_M42gl","Bar_M42gr","Bar_M42hl","Bar_M42hr
","Bar_M42il","Bar_M42ir","Bar_M42jl","Bar_M42jr","Bar_M43al","Bar_M43ar","Bar_M43bl",
"Bar_M43br","Bar_M43cl","Bar_M43cr","Bar_M43dl","Bar_M43dr","Bar_M43el","Bar_M43er","B
ar_M43fl","Bar_M43fr","Bar_M43gl","Bar_M43gr","Bar_M43hl","Bar_M43hr","Bar_M43il","Bar
_M43ir","Bar_M43jl","Bar_M43jr" ], "SECTION" )

#Composed Line
addGeometry( "Composed Line", "LINE", "COMLIN", [] )
setParameter( "GEOMET", "Composed Line", "DISTAN/THICK", 300 )
setElementClassType( "SHAPE", [ "Composed Line" ], "COMLIN" )
assignGeometry( "Composed Line", "SHAPE", [ "Composed Line" ] )
resetElementData( "SHAPE", [ "Composed Line" ] )

#Assign Supports
addSet( "GEOMETRYSUPPORTSET", "Support" )

createPointSupport( "Support", "Support" )
setParameter( "GEOMETRYSUPPORT", "Support", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Support", "TRANSL", [ 0, 1, 0 ] )
setParameter( "GEOMETRYSUPPORT", "Support", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Support", "Support", [[ 300, -30, 0 ]] )

addSet( "GEOMETRYSUPPORTSET", "Sym_X" )

createLineSupport( "Sym_X", "Sym_X" )
setParameter( "GEOMETRYSUPPORT", "Sym_X", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Sym_X", "TRANSL", [ 1, 0, 0 ] )
setParameter( "GEOMETRYSUPPORT", "Sym_X", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Sym_X", "Pressure Layer_1", [[ 1900, 185, 0 ]] )

createPointSupport( "Symp_X", "Symp_X" )
setParameter( "GEOMETRYSUPPORT", "Symp_X", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Symp_X", "TRANSL", [ 1, 0, 0 ] )
setParameter( "GEOMETRYSUPPORpT", "Symp_X", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Symp_X", "Koppel_M12a", [[ 1900, 76, 0 ]] )
attach( "GEOMETRYSUPPORT", "Symp_X", "Koppel_M12b", [[ 1900, 76, 0 ]] )
attach( "GEOMETRYSUPPORT", "Symp_X", "Koppel_M12c", [[ 1900, 76, 0 ]] )
attach( "GEOMETRYSUPPORT", "Symp_X", "Koppel_M12d", [[ 1900, 76, 0 ]] )
attach( "GEOMETRYSUPPORT", "Symp_X", "Koppel_M12e", [[ 1900, 76, 0 ]] )
```

```
attach( "GEOMETRYSUPPORT", "Symp_X", "Koppel_M12f", [[ 1900, 76, 0 ]] )
attach( "GEOMETRYSUPPORT", "Symp_X", "Koppel_M12g", [[ 1900, 76, 0 ]] )

addSet( "GEOMETRYSUPPORTSET", "Load point" )

createPointSupport( "Load point", "Load point" )
setParameter( "GEOMETRYSUPPORT", "Load point", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Load point", "TRANSL", [ 0, 1, 0 ] )
setParameter( "GEOMETRYSUPPORT", "Load point", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Load point", "Loadplate", [[ 1100, 330, 0 ]] )

#Assign Loads
addSet( "GEOMETRYLOADSET", "Load" )

createPointLoad( "Load", "Load" )
setParameter( "GEOMETRYLOAD", "Load", "LODTYP", "DEFORM" )
setParameter( "GEOMETRYLOAD", "Load", "DEFORM/TR/VALUE", -0.1 )
setParameter( "GEOMETRYLOAD", "Load", "DEFORM/TR/DIRECT", 2 )
attach( "GEOMETRYLOAD", "Load", "Loadplate", [[ 1100, 330, 0 ]] )

#Set mesh properties
setElementSize( [  "Loadplate", "Pressure Layer_1", "Slab_1",
"Koppel_M12a","Koppel_M12b","Koppel_M12c","Koppel_M12d","Koppel_M12e","Koppel_M12f","K
oppel_M12g","Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d","Reinfo_M8e","Reinfo_M
8f","Reinfo_M8g","Reinfo_M8h","Reinfo_M8i","Reinfo_M8j","Reinfo_M8k","Reinfo_M8l","Rei
nfo_M8m","Reinfo_M8n","Reinfo_M8o","Bar_M41al","Bar_M41ar","Bar_M41bl","Bar_M41br","Ba
r_M41cl","Bar_M41cr","Bar_M41dl","Bar_M41dr","Bar_M41el","Bar_M41er","Bar_M41fl","Bar_
M41fr","Bar_M41gl","Bar_M41gr","Bar_M41hl","Bar_M41hr","Bar_M41il","Bar_M41ir","Bar_M4
1jl","Bar_M41jr","Bar_M42al","Bar_M42ar","Bar_M42bl","Bar_M42br","Bar_M42cl","Bar_M42c
r","Bar_M42dl","Bar_M42dr","Bar_M42el","Bar_M42er","Bar_M42fl","Bar_M42fr","Bar_M42gl"
,"Bar_M42gr","Bar_M42hl","Bar_M42hr","Bar_M42il","Bar_M42ir","Bar_M42jl","Bar_M42jr","
Bar_M43al","Bar_M43ar","Bar_M43bl","Bar_M43br","Bar_M43cl","Bar_M43cr","Bar_M43dl","Ba
r_M43dr","Bar_M43el","Bar_M43er","Bar_M43fl","Bar_M43fr","Bar_M43gl","Bar_M43gr","Bar_
M43hl","Bar_M43hr","Bar_M43il","Bar_M43ir","Bar_M43jl","Bar_M43jr" ], 10, -1, True )
setMesherType( [  "Loadplate", "Pressure Layer_1", "Slab_1",
"Koppel_M12a","Koppel_M12b","Koppel_M12c","Koppel_M12d","Koppel_M12e","Koppel_M12f","K
oppel_M12g","Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d","Reinfo_M8e","Reinfo_M
8f","Reinfo_M8g","Reinfo_M8h","Reinfo_M8i","Reinfo_M8j","Reinfo_M8k","Reinfo_M8l","Rei
nfo_M8m","Reinfo_M8n","Reinfo_M8o","Bar_M41al","Bar_M41ar","Bar_M41bl","Bar_M41br","Ba
r_M41cl","Bar_M41cr","Bar_M41dl","Bar_M41dr","Bar_M41el","Bar_M41er","Bar_M41fl","Bar_
M41fr","Bar_M41gl","Bar_M41gr","Bar_M41hl","Bar_M41hr","Bar_M41il","Bar_M41ir","Bar_M4
1jl","Bar_M41jr","Bar_M42al","Bar_M42ar","Bar_M42bl","Bar_M42br","Bar_M42cl","Bar_M42c
r","Bar_M42dl","Bar_M42dr","Bar_M42el","Bar_M42er","Bar_M42fl","Bar_M42fr","Bar_M42gl"
,"Bar_M42gr","Bar_M42hl","Bar_M42hr","Bar_M42il","Bar_M42ir","Bar_M42jl","Bar_M42jr","
Bar_M43al","Bar_M43ar","Bar_M43bl","Bar_M43br","Bar_M43cl","Bar_M43cr","Bar_M43dl","Ba
r_M43dr","Bar_M43el","Bar_M43er","Bar_M43fl","Bar_M43fr","Bar_M43gl","Bar_M43gr","Bar_
M43hl","Bar_M43hr","Bar_M43il","Bar_M43ir","Bar_M43jl","Bar_M43jr" ], "HEXQUAD" )
clearMidSideNodeLocation( [  "Loadplate", "Pressure Layer_1", "Slab_1",
"Koppel_M12a","Koppel_M12b","Koppel_M12c","Koppel_M12d","Koppel_M12e","Koppel_M12f","K
oppel_M12g","Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d","Reinfo_M8e","Reinfo_M
8f","Reinfo_M8g","Reinfo_M8h","Reinfo_M8i","Reinfo_M8j","Reinfo_M8k","Reinfo_M8l","Rei
nfo_M8m","Reinfo_M8n","Reinfo_M8o","Bar_M41al","Bar_M41ar","Bar_M41bl","Bar_M41br","Ba
r_M41cl","Bar_M41cr","Bar_M41dl","Bar_M41dr","Bar_M41el","Bar_M41er","Bar_M41fl","Bar_
M41fr","Bar_M41gl","Bar_M41gr","Bar_M41hl","Bar_M41hr","Bar_M41il","Bar_M41ir","Bar_M4
1jl","Bar_M41jr","Bar_M42al","Bar_M42ar","Bar_M42bl","Bar_M42br","Bar_M42cl","Bar_M42c
r","Bar_M42dl","Bar_M42dr","Bar_M42el","Bar_M42er","Bar_M42fl","Bar_M42fr","Bar_M42gl"
,"Bar_M42gr","Bar_M42hl","Bar_M42hr","Bar_M42il","Bar_M42ir","Bar_M42jl","Bar_M42jr","
Bar_M43al","Bar_M43ar","Bar_M43bl","Bar_M43br","Bar_M43cl","Bar_M43cr","Bar_M43dl","Ba
r_M43dr","Bar_M43el","Bar_M43er","Bar_M43fl","Bar_M43fr","Bar_M43gl","Bar_M43gr","Bar_
M43hl","Bar_M43hr","Bar_M43il","Bar_M43ir","Bar_M43jl","Bar_M43jr" ] )
setElementSize( [ "Support","Pressure Layer", "Slab" ], 50, -1, True )
setMesherType( [ "Support","Pressure Layer", "Slab" ], "HEXQUAD" )
setEdgeGrading( "Pressure Layer", [[ 500, 70, 0 ]], [], 50, 50, 10, 0.25, False,
True )
setEdgeGrading( "Slab", [], [[ 500, 70, 0 ]], 50, 50, 10, 0.25, False, True )
generateMesh( [] )
hideView( "GEOM" )
showView( "MESH" )

#Analysis
addAnalysis( "Analysis1" )
```

```
423 addAnalysisCommand( "Analysis1", "NONLIN", "Structural nonlinear" )
424 renameAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "EXECUT(1)",
    "Load" )
425 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/LOADNR" )
426 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/LOADNR", 1 )
427 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/STEPTY", "AUTOMA" )
428 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/SIZES", 250 )
429 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/MAXSIZ", 0.01 )
430 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/MINSIZ", 0.00001 )
431 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/CUTBCK", 0.25 )
432 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/MAXSTP", 100 )
433 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/ARCLEN", True )
434 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/ARCLEN/REGULA/SET" )
435 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/ARCLEN/REGULA/SET(1)/NODES(1)/RNGNRS", "1-9361" )
436 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/ARCLEN/REGULA/SET(1)/DIRECT", 2 )
437 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/SIZES", "1(100)" )
438 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN", True )
439 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN/REGULA/SET" )
440 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN/REGULA/SET(1)/NODES(1)/RNGNRS", "844 865 866 871
    945-966 1055 1056-2925(89) 3108-3132 3217 3218 3396 3397 3575 3576 3754 3755 3933
    3934 4112 4113 4291 4292 4470 4471 4649 4650 4828 4829 5007 5008 5186 5187 5365 5366
    5544 5545 5723 5724 5902 5903 6081 6082 6260 6261 6439 6440 6618 6619 6797 6798 6976
    6977 7155 7244 7246 7335-7341 7430 7431-7965(89) 8144-8153 8331 8332 8510 8511 8689
    8690 8868 8869 9047 9048 9226 9453-9455 9593 9594 9732 9733 9871 9872 10010 10011
    10149 10150 10288 10289" )
441 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN/REGULA/SET(1)/DIRECT", 2 )
442 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/MAXITE", 250 )
443 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/LINESE", False )
444 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/ENERGY", True )
445 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/DISPLA", False )
446 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/FORCE", True )
447 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/ENERGY/TOLCON", 0.001 )
448 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/ENERGY/NOCONV", "CONTIN" )
449 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/FORCE/TOLCON", 0.01 )
450 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/FORCE/NOCONV", "CONTIN" )
451 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/METHOD/METNAM", "SECANT" )
452
453 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "OUTPUT(1)/SELTYP",
    "USER" )
454 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "OUTPUT(1)/USER" )
455 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/DISPLA(1)/TOTAL" )
456 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/FORCE(1)/REACTI" )
```

```
457 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(1)/TOTAL/GREEN" )
458 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(2)/TOTAL/GREEN/PRINCI" )
459 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(3)/CRKWDT" )
460 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(4)/CRACK/GREEN" )
461 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(5)/CRKWDT/GREEN/PRINCI" )
462 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(1)/TOTAL/CAUCHY" )
463 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(2)/TOTAL/CAUCHY/PRINCI" )
464 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(3)/TOTAL/FORCE/LOCAL" )
465 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(4)/TOTAL/MOMENT/LOCAL" )
466
467 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(1)/TOTAL/GREEN/GLOBAL/LOCATI", "INTPNT" )
468 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(1)/TOTAL/CAUCHY/GLOBAL/LOCATI", "INTPNT" )
469
470 #Run Analysis
471 # runSolver( [] )
472 # showView( "RESULT" )
473 fitAll( )
474
475
476
```

B3 - T19 V11

```
newProject( "D:/Afstuderen/Python/Proefstukken/DIANA Output/T19 V11 Reduced Elements FRICEL", 10 )


###Project Settings###
setModelAnalysisAspects( [ "STRUCT" ] )
setModelDimension( "2D" )
setDefaultMeshOrder( "QUADRATIC" )
setDefaultMesherType( "HEXQUAD" )
setDefaultMidSideNodeLocation( "ONSHAP" )
setUnit( "LENGTH", "MM" )
setUnit( "FORCE", "N" )
setUnit( "ANGLE", "DEGREE" )


#Create Shapes
createSheet( "Pressure Layer", [[ 0, 70, 0 ],[ 1900, 70, 0 ],[ 1900, 250, 0 ],[ 1150, 250, 0 ],[1050, 250, 0 ],[ 0, 250, 0 ]] )
createSheet( "Slab", [[ 0, 0, 0 ],[ 250, 0, 0 ],[ 350, 0, 0 ],[ 1900, 0, 0 ],[ 1900, 70, 0],[ 0, 70, 0 ]] )

createSheet( "Support", [[ 250, 0, 0 ],[ 350, 0, 0 ],[ 350, -30, 0 ],[ 300, -30, 0 ],[ 250, -30, 0 ]] )
createSheet( "Loadplate", [[1050, 250, 0 ],[ 1150, 250, 0 ],[ 1150, 280, 0 ],[ 1100, 280, 0 ],[ 1050, 280, 0 ]] )

#Create Reinforcement
createLine( "Koppel_M12a", [ 1200, 76, 0 ], [ 1900, 76, 0 ] )
createLine( "Koppel_M12b", [ 1200, 76, 0 ], [ 1900, 76, 0 ] )
createLine( "Koppel_M12c", [ 1200, 76, 0 ], [ 1900, 76, 0 ] )
createLine( "Koppel_M12d", [ 1200, 76, 0 ], [ 1900, 76, 0 ] )
createLine( "Koppel_M12e", [ 1200, 76, 0 ], [ 1900, 76, 0 ] )
createLine( "Koppel_M12f", [ 1200, 76, 0 ], [ 1900, 76, 0 ] )
createLine( "Koppel_M12g", [ 1200, 76, 0 ], [ 1900, 76, 0 ] )

createLine( "Reinfo_M8a", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8b", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8c", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8d", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8e", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8f", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8g", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8h", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8i", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8j", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8k", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8l", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8m", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8n", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8o", [ 0, 35, 0 ], [ 1865, 35, 0 ] )

createLine( "Bar_M41al", [ 135, 45.5, 0],[ 175, 165.5, 0] )
createLine( "Bar_M41ar", [ 215, 45.5, 0],[ 175, 165.5, 0] )
createLine( "Bar_M41bl", [ 135, 45.5, 0],[ 175, 165.5, 0] )
createLine( "Bar_M41br", [ 215, 45.5, 0],[ 175, 165.5, 0] )
createLine( "Bar_M41cl", [ 135, 45.5, 0],[ 175, 165.5, 0] )
createLine( "Bar_M41cr", [ 215, 45.5, 0],[ 175, 165.5, 0] )
createLine( "Bar_M41dl", [ 135, 45.5, 0],[ 175, 165.5, 0] )
createLine( "Bar_M41dr", [ 215, 45.5, 0],[ 175, 165.5, 0] )
createLine( "Bar_M41el", [ 135, 45.5, 0],[ 175, 165.5, 0] )
createLine( "Bar_M41er", [ 215, 45.5, 0],[ 175, 165.5, 0] )
createLine( "Bar_M41fl", [ 135, 45.5, 0],[ 175, 165.5, 0] )
createLine( "Bar_M41fr", [ 215, 45.5, 0],[ 175, 165.5, 0] )
createLine( "Bar_M41gl", [ 135, 45.5, 0],[ 175, 165.5, 0] )
createLine( "Bar_M41gr", [ 215, 45.5, 0],[ 175, 165.5, 0] )
createLine( "Bar_M41hl", [ 135, 45.5, 0],[ 175, 165.5, 0] )
createLine( "Bar_M41hr", [ 215, 45.5, 0],[ 175, 165.5, 0] )
createLine( "Bar_M41il", [ 135, 45.5, 0],[ 175, 165.5, 0] )
createLine( "Bar_M41ir", [ 215, 45.5, 0],[ 175, 165.5, 0] )
createLine( "Bar_M41jl", [ 135, 45.5, 0],[ 175, 165.5, 0] )
createLine( "Bar_M41jr", [ 215, 45.5, 0],[ 175, 165.5, 0] )
```

```
67
68  createLine( "Bar_M42al", [ 735, 45.5, 0],[ 775, 165.5, 0] )
69  createLine( "Bar_M42ar", [ 815, 45.5, 0],[ 775, 165.5, 0] )
70  createLine( "Bar_M42bl", [ 735, 45.5, 0],[ 775, 165.5, 0] )
71  createLine( "Bar_M42br", [ 815, 45.5, 0],[ 775, 165.5, 0] )
72  createLine( "Bar_M42cl", [ 735, 45.5, 0],[ 775, 165.5, 0] )
73  createLine( "Bar_M42cr", [ 815, 45.5, 0],[ 775, 165.5, 0] )
74  createLine( "Bar_M42dl", [ 735, 45.5, 0],[ 775, 165.5, 0] )
75  createLine( "Bar_M42dr", [ 815, 45.5, 0],[ 775, 165.5, 0] )
76  createLine( "Bar_M42el", [ 735, 45.5, 0],[ 775, 165.5, 0] )
77  createLine( "Bar_M42er", [ 815, 45.5, 0],[ 775, 165.5, 0] )
78  createLine( "Bar_M42fl", [ 735, 45.5, 0],[ 775, 165.5, 0] )
79  createLine( "Bar_M42fr", [ 815, 45.5, 0],[ 775, 165.5, 0] )
80  createLine( "Bar_M42gl", [ 735, 45.5, 0],[ 775, 165.5, 0] )
81  createLine( "Bar_M42gr", [ 815, 45.5, 0],[ 775, 165.5, 0] )
82  createLine( "Bar_M42hl", [ 735, 45.5, 0],[ 775, 165.5, 0] )
83  createLine( "Bar_M42hr", [ 815, 45.5, 0],[ 775, 165.5, 0] )
84  createLine( "Bar_M42il", [ 735, 45.5, 0],[ 775, 165.5, 0] )
85  createLine( "Bar_M42ir", [ 815, 45.5, 0],[ 775, 165.5, 0] )
86  createLine( "Bar_M42jl", [ 735, 45.5, 0],[ 775, 165.5, 0] )
87  createLine( "Bar_M42jr", [ 815, 45.5, 0],[ 775, 165.5, 0] )
88
89  createLine( "Bar_M43al", [ 1460, 45.5, 0],[ 1500, 165.5, 0] )
90  createLine( "Bar_M43ar", [ 1540, 45.5, 0],[ 1500, 165.5, 0] )
91  createLine( "Bar_M43bl", [ 1460, 45.5, 0],[ 1500, 165.5, 0] )
92  createLine( "Bar_M43br", [ 1540, 45.5, 0],[ 1500, 165.5, 0] )
93  createLine( "Bar_M43cl", [ 1460, 45.5, 0],[ 1500, 165.5, 0] )
94  createLine( "Bar_M43cr", [ 1540, 45.5, 0],[ 1500, 165.5, 0] )
95  createLine( "Bar_M43dl", [ 1460, 45.5, 0],[ 1500, 165.5, 0] )
96  createLine( "Bar_M43dr", [ 1540, 45.5, 0],[ 1500, 165.5, 0] )
97  createLine( "Bar_M43el", [ 1460, 45.5, 0],[ 1500, 165.5, 0] )
98  createLine( "Bar_M43er", [ 1540, 45.5, 0],[ 1500, 165.5, 0] )
99  createLine( "Bar_M43fl", [ 1460, 45.5, 0],[ 1500, 165.5, 0] )
100 createLine( "Bar_M43fr", [ 1540, 45.5, 0],[ 1500, 165.5, 0] )
101 createLine( "Bar_M43gl", [ 1460, 45.5, 0],[ 1500, 165.5, 0] )
102 createLine( "Bar_M43gr", [ 1540, 45.5, 0],[ 1500, 165.5, 0] )
103 createLine( "Bar_M43hl", [ 1460, 45.5, 0],[ 1500, 165.5, 0] )
104 createLine( "Bar_M43hr", [ 1540, 45.5, 0],[ 1500, 165.5, 0] )
105 createLine( "Bar_M43il", [ 1460, 45.5, 0],[ 1500, 165.5, 0] )
106 createLine( "Bar_M43ir", [ 1540, 45.5, 0],[ 1500, 165.5, 0] )
107 createLine( "Bar_M43jl", [ 1460, 45.5, 0],[ 1500, 165.5, 0] )
108 createLine( "Bar_M43jr", [ 1540, 45.5, 0],[ 1500, 165.5, 0] )
109
110 createLine( "Composed Line", [ 0, 125, 0 ], [ 1900, 125, 0 ] )
111
112 createSheet( "Mesh Sheet", [[ 1000, 0, 0 ],[ 1000, 300, 0 ],[ 1000, 300, 1000 ],[
    1000, 0, 1000 ]] )
113
114 #Splitting Mesh Parts
115 subtract( "Pressure Layer", [ "Mesh Sheet" ], True, True )
116 subtract( "Slab", [ "Mesh Sheet" ], False, True )
117
118 #Regroup Material Parts
119 addSet( "SHAPESET", "Coupling Reinforcement" )
120 moveToShapeSet( [
    "Koppel_M12a","Koppel_M12b","Koppel_M12c","Koppel_M12d","Koppel_M12e","Koppel_M12f","K
    oppel_M12g" ], "Coupling Reinforcement" )
121 addSet( "SHAPESET", "Longitudinal Reinforcement" )
122 moveToShapeSet( [
    "Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d","Reinfo_M8e","Reinfo_M8f","Reinfo_
    M8g","Reinfo_M8h","Reinfo_M8i","Reinfo_M8j","Reinfo_M8k","Reinfo_M8l","Reinfo_M8m","Re
    info_M8n","Reinfo_M8o" ], "Longitudinal Reinforcement" )
123 addSet( "SHAPESET", "Lattice girders" )
124 moveToShapeSet( [
    "Bar_M41al","Bar_M41ar","Bar_M41bl","Bar_M41br","Bar_M41cl","Bar_M41cr","Bar_M41dl","B
    ar_M41dr","Bar_M41el","Bar_M41er","Bar_M41fl","Bar_M41fr","Bar_M41gl","Bar_M41gr","Bar
    _M41hl","Bar_M41hr","Bar_M41il","Bar_M41ir","Bar_M41jl","Bar_M41jr","Bar_M42al","Bar_M
    42ar","Bar_M42bl","Bar_M42br","Bar_M42cl","Bar_M42cr","Bar_M42dl","Bar_M42dr","Bar_M42
    el","Bar_M42er","Bar_M42fl","Bar_M42fr","Bar_M42gl","Bar_M42gr","Bar_M42hl","Bar_M42hr
    ","Bar_M42il","Bar_M42ir","Bar_M42jl","Bar_M42jr","Bar_M43al","Bar_M43ar","Bar_M43bl",
    "Bar_M43br","Bar_M43cl","Bar_M43cr","Bar_M43dl","Bar_M43dr","Bar_M43el","Bar_M43er","B
```

```
    ar_M43fl","Bar_M43fr","Bar_M43gl","Bar_M43gr","Bar_M43hl","Bar_M43hr","Bar_M43il","Bar
    _M43ir","Bar_M43jl","Bar_M43jr" ], "Lattice girders" )
125 addSet( "SHAPESET", "Steel plates" )
126 moveToShapeSet( [ "Loadplate", "Support" ], "Steel plates" )
127 rename( "SHAPESET", "Shapes", "Concrete" )
128 addSet( "SHAPESET", "Composed Line" )
129 moveToShapeSet( [ "Composed Line" ], "Composed Line" )
130
131 #Material Precast concrete
132 addMaterial( "Precast concrete", "CONCR", "TSCR", [] )
133 setParameter( "MATERIAL", "Precast concrete", "LINEAR/ELASTI/YOUNG", 32382 )
134 setParameter( "MATERIAL", "Precast concrete", "LINEAR/ELASTI/POISON", 0.2 )
135 setParameter( "MATERIAL", "Precast concrete", "LINEAR/MASS/DENSIT", 2.4e-09 )
136 setParameter( "MATERIAL", "Precast concrete", "MODTYP/TOTCRK", "ROTATE" )
137 setParameter( "MATERIAL", "Precast concrete", "TENSIL/TENCRV", "HORDYK" )
138 setParameter( "MATERIAL", "Precast concrete", "TENSIL/TENSTR", 2.78 )
139 setParameter( "MATERIAL", "Precast concrete", "TENSIL/GF1", 0.1393 )
140 setParameter( "MATERIAL", "Precast concrete", "TENSIL/CBSPEC", "ROTS" )
141 setParameter( "MATERIAL", "Precast concrete", "TENSIL/POISRE/POIRED", "DAMAGE" )
142 # setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMCRV", "MC2010" )
143 # setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMSTR", 55.9 )
144 # setParameter( "MATERIAL", "Precast concrete", "COMPRS/EPSC1", 0.0024 )
145 # setParameter( "MATERIAL", "Precast concrete", "COMPRS/EPSCU", 0.0035 )
146 # setParameter( "MATERIAL", "Precast concrete", "COMPRS/CONFIN/CNFCRV", "NONE" )
147 setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMCRV", "CONSTA" )
148 setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMSTR", 36.3 )
149
150 addGeometry( "Precast concrete", "SHEET", "MEMBRA", [] )
151 setParameter( "GEOMET", "Precast concrete", "THICK", 1000 )
152 setParameter( "GEOMET", "Precast concrete", "LOCAXS", True )
153 setParameter( "GEOMET", "Precast concrete", "LOCAXS/XAXIS", [ 1, 0, 0 ] )
154
155 addElementData( "Precast concrete" )
156 setElementClassType( "SHAPE", [ "Slab", "Slab_1" ], "MEMBRA" )
157 assignMaterial( "Precast concrete", "SHAPE", [ "Slab", "Slab_1" ] )
158 assignGeometry( "Precast concrete", "SHAPE", [ "Slab", "Slab_1" ] )
159 assignElementData( "Precast concrete", "SHAPE", [ "Slab", "Slab_1" ] )
160
161 # setParameter( "DATA", "Precast concrete", "INTEGR", "REDUCE" )
162
163 #Material Pressure layer
164 addMaterial( "Pressure layer", "CONCR", "TSCR", [] )
165 setParameter( "MATERIAL", "Pressure layer", "LINEAR/ELASTI/YOUNG", 26045 )
166 setParameter( "MATERIAL", "Pressure layer", "LINEAR/ELASTI/POISON", 0.2 )
167 setParameter( "MATERIAL", "Pressure layer", "LINEAR/MASS/DENSIT", 2.4e-09 )
168 setParameter( "MATERIAL", "Pressure layer", "MODTYP/TOTCRK", "ROTATE" )
169 setParameter( "MATERIAL", "Pressure layer", "TENSIL/TENCRV", "HORDYK" )
170 setParameter( "MATERIAL", "Pressure layer", "TENSIL/TENSTR", 1.35 )
171 setParameter( "MATERIAL", "Pressure layer", "TENSIL/GF1", 0.1223 )
172 setParameter( "MATERIAL", "Pressure layer", "TENSIL/CBSPEC", "ROTS" )
173 setParameter( "MATERIAL", "Pressure layer", "TENSIL/POISRE/POIRED", "DAMAGE" )
174 # setParameter( "MATERIAL", "Pressure layer", "COMPRS/COMCRV", "MC2010" )
175 # setParameter( "MATERIAL", "Pressure layer", "COMPRS/COMSTR", 35.6 )
176 # setParameter( "MATERIAL", "Pressure layer", "COMPRS/EPSC1", 0.0021 )
177 # setParameter( "MATERIAL", "Pressure layer", "COMPRS/EPSCU", 0.0035 )
178 # setParameter( "MATERIAL", "Pressure layer", "COMPRS/CONFIN/CNFCRV", "NONE" )
179 setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMCRV", "CONSTA" )
180 setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMSTR", 17.6 )
181
182 addGeometry( "Pressure layer", "SHEET", "MEMBRA", [] )
183 setParameter( "GEOMET", "Pressure layer", "THICK", 1000 )
184 setParameter( "GEOMET", "Pressure layer", "LOCAXS", True )
185 setParameter( "GEOMET", "Pressure layer", "LOCAXS/XAXIS", [ 1, 0, 0 ] )
186
187 addElementData( "Pressure layer" )
188 setElementClassType( "SHAPE", [ "Pressure Layer", "Pressure Layer_1" ], "MEMBRA" )
189 assignMaterial( "Pressure layer", "SHAPE", [ "Pressure Layer", "Pressure Layer_1" ] )
190 assignGeometry( "Pressure layer", "SHAPE", [ "Pressure Layer", "Pressure Layer_1" ] )
191 assignElementData( "Pressure layer", "SHAPE", [ "Pressure Layer", "Pressure Layer_1"
    ] )
192
```

```
193  # setParameter( "DATA", "Pressure layer", "INTEGR", "REDUCE" )
194
195  #Material Steel
196  addMaterial( "Steel", "MCSTEL", "ISOTRO", [] )
197  setParameter( "MATERIAL", "Steel", "LINEAR/ELASTI/YOUNG", 210000 )
198  setParameter( "MATERIAL", "Steel", "LINEAR/ELASTI/POISON", 0.3 )
199  setParameter( "MATERIAL", "Steel", "LINEAR/MASS/DENSIT", 7.8e-09 )
200  setParameter( "MATERIAL", "Steel", "LINEAR/MASS/DENSIT", 0 )
201  addGeometry( "Plate", "SHEET", "MEMBRA", [] )
202  setParameter( "GEOMET", "Plate", "THICK", 1000 )
203  addElementData( "Plate" )
204
205  setElementClassType( "SHAPE", [ "Support", "Loadplate" ], "MEMBRA" )
206  assignMaterial( "Steel", "SHAPE", [ "Support", "Loadplate" ] )
207  assignGeometry( "Plate", "SHAPE", [ "Support", "Loadplate" ] )
208  assignElementData( "Plate", "SHAPE", [ "Support", "Loadplate" ] )
209  # setParameter( "DATA", "Plate", "INTEGR", "REDUCE" )
210
211  #Concrete Interface
212  # addMaterial( "Interface Concrete", "INTERF", "FRICTI", [] )
213  # setParameter( "MATERIAL", "Interface Concrete", "LINEAR/IFTYP", "LIN2D" )
214  # setParameter( "MATERIAL", "Interface Concrete", "COULOM/OPNTYP", "GAP" )
215  # setParameter( "MATERIAL", "Interface Concrete", "COULOM/GAPVAL", 0.5 )
216  # setParameter( "MATERIAL", "Interface Concrete", "COULOM/COHESI", 0.2 )
217  # setParameter( "MATERIAL", "Interface Concrete", "COULOM/PHI", 22 )
218  # setParameter( "MATERIAL", "Interface Concrete", "COULOM/PSI", 22 )
219  # setParameter( "MATERIAL", "Interface Concrete", "LINEAR/ELAS2/DSNY", 10 )
220  # setParameter( "MATERIAL", "Interface Concrete", "LINEAR/ELAS2/DSSX", 10 )
221  # setParameter( "MATERIAL", "Interface Concrete", "COULOM/FRCHSR/MODE2", 0 )
222
223  addMaterial( "Interface Concrete", "INTERF", "FRICEL", [] )
224  setParameter( "MATERIAL", "Interface Concrete", "LINEAR/IFTYP", "LIN2D" )
225  setParameter( "MATERIAL", "Interface Concrete", "NEFRIC/SPCTYP", "USER" )
226  setParameter( "MATERIAL", "Interface Concrete", "NEFRIC/COHESI", 0.2 )
227  setParameter( "MATERIAL", "Interface Concrete", "NEFRIC/PHI", 22 )
228  setParameter( "MATERIAL", "Interface Concrete", "LINEAR/ELAS2/DSNY", 100 )
229  setParameter( "MATERIAL", "Interface Concrete", "LINEAR/ELAS2/DSSX", 100 )
230
231  addGeometry( "Interface Concrete", "LINE", "STLIIF", [] )
232  setParameter( "GEOMET", "Interface Concrete", "LIFMEM/THICK", 1000 )
233  addElementData( "Interface Concrete" )
234  createConnection( "Interface Concrete", "INTER", "SHAPEEDGE", "SHAPEEDGE" )
235  setParameter( "GEOMETRYCONNECTION", "Interface Concrete", "MODE", "CLOSED" )
236  setElementClassType( "GEOMETRYCONNECTION", "Interface Concrete", "STLIIF" )
237  assignMaterial( "Interface Concrete", "GEOMETRYCONNECTION", "Interface Concrete" )
238  assignGeometry( "Interface Concrete", "GEOMETRYCONNECTION", "Interface Concrete" )
239  assignElementData( "Interface Concrete", "GEOMETRYCONNECTION", "Interface Concrete" )
240  setParameter( "GEOMETRYCONNECTION", "Interface Concrete", "FLIP", False )
241  attachTo( "GEOMETRYCONNECTION", "Interface Concrete", "SOURCE", "Pressure Layer", [[
     500, 70, 0 ]] )
242  attachTo( "GEOMETRYCONNECTION", "Interface Concrete", "TARGET", "Slab", [[ 500, 70,
     0 ]] )
243  attachTo( "GEOMETRYCONNECTION", "Interface Concrete", "SOURCE", "Pressure Layer_1",
     [[ 1450, 70, 0 ]] )
244  attachTo( "GEOMETRYCONNECTION", "Interface Concrete", "TARGET", "Slab_1", [[ 1450,
     70, 0 ]] )
245
246  # setParameter( "DATA", "Interface Concrete", "INTEGR", "REDUCE" )
247
248  #Steel Interface
249  addMaterial( "Interface Steel", "INTERF", "NONLIF", [] )
250  setParameter( "MATERIAL", "Interface Steel", "LINEAR/IFTYP", "LIN2D" )
251  setParameter( "MATERIAL", "Interface Steel", "LINEAR/ELAS2/DSNY", 32000 )
252  setParameter( "MATERIAL", "Interface Steel", "LINEAR/ELAS2/DSSX", 0.00032 )
253  setParameter( "MATERIAL", "Interface Steel", "NONLIN/IFNOTE", "NOTSHR" )
254  setParameter( "MATERIAL", "Interface Steel", "NONLIN/NLEL8/NOTENS", [ 0.001, 0 ] )
255  setParameter( "MATERIAL", "Interface Steel", "NONLIN/NLEL8/NOSHTE", [ 0.001, 0 ] )
256  addGeometry( "Interface Steel", "LINE", "STLIIF", [] )
257  setParameter( "GEOMET", "Interface Steel", "LIFMEM/THICK", 1000 )
258  addElementData( "Interface Steel" )
259  createConnection( "Interface Steel", "INTER", "SHAPEEDGE", "SHAPEEDGE" )
```

```
260
261 setParameter( "GEOMETRYCONNECTION", "Interface Steel", "CONTYP", "INTER" )
262 setParameter( "GEOMETRYCONNECTION", "Interface Steel", "MODE", "AUTO" )
263 setElementClassType( "GEOMETRYCONNECTION", "Interface Steel", "STLIIF" )
264 assignMaterial( "Interface Steel", "GEOMETRYCONNECTION", "Interface Steel" )
265 assignGeometry( "Interface Steel", "GEOMETRYCONNECTION", "Interface Steel" )
266 assignElementData( "Interface Steel", "GEOMETRYCONNECTION", "Interface Steel" )
267 setParameter( "GEOMETRYCONNECTION", "Interface Steel", "FLIP", False )
268 attachTo( "GEOMETRYCONNECTION", "Interface Steel", "SOURCE", "Pressure Layer_1", [[
    1100, 250, 0 ]] )
269 attachTo( "GEOMETRYCONNECTION", "Interface Steel", "TARGET", "Loadplate", [[ 1100,
    250, 0 ]] )
270 attachTo( "GEOMETRYCONNECTION", "Interface Steel", "SOURCE", "Slab", [[ 300, 0, 0 ]] )
271 attachTo( "GEOMETRYCONNECTION", "Interface Steel", "TARGET", "Support", [[ 300, 0, 0
    ]] )
272
273 # setParameter( "DATA", "Interface Steel", "INTEGR", "REDUCE" )
274 # setParameter( "DATA", "Interface Steel", "NUMINT", [ "LOBATT" ] )
275
276 #Material Reinforcement Koppel_M12
277 addMaterial( "Koppel_M12", "REINFO", "REBOND", [ ] )
278 setParameter( "MATERIAL", "Koppel_M12", "REBARS/ELASTI/YOUNG", 200000 )
279 setParameter( "MATERIAL", "Koppel_M12", "REBARS/POISON/POISON", 0.3 )
280 setParameter( "MATERIAL", "Koppel_M12", "REBARS/MASS/DENSIT", 7.8e-09 )
281 setParameter( "MATERIAL", "Koppel_M12", "REBARS/PLATYP", "VMISES" )
282 setParameter( "MATERIAL", "Koppel_M12", "REBARS/PLASTI/TRESSH", "KAPSIG" )
283 setParameter( "MATERIAL", "Koppel_M12", "REBARS/PLASTI/KAPSIG", [ 0, 550, 0.04725,
    594 ] )
284 setParameter( "MATERIAL", "Koppel_M12", "RESLIP/SHFTYP", "BONDS6" )
285 setParameter( "MATERIAL", "Koppel_M12", "RESLIP/DSNY", 1000 )
286 setParameter( "MATERIAL", "Koppel_M12", "RESLIP/DSSX", 100 )
287 setParameter( "MATERIAL", "Koppel_M12", "RESLIP/BONDS6/SLPVAL", [ 10.47, 4.19, 0.1,
    1, 2, 10, 0.4 ] )
288 addGeometry( "Koppel_M12", "RELINE", "REBAR", [] )
289 setParameter( "GEOMET", "Koppel_M12", "REITYP", "REITRU" )
290 setParameter( "GEOMET", "Koppel_M12", "REITYP", "CIRBEA" )
291 setParameter( "GEOMET", "Koppel_M12", "CIRBEA/CIRCLE", 12)
292 setParameter( "GEOMET", "Koppel_M12", "TIPELM/SURFAC", 113.1 )
293 assignMaterial( "Koppel_M12", "SHAPE", [
    "Koppel_M12a","Koppel_M12b","Koppel_M12c","Koppel_M12d","Koppel_M12e","Koppel_M12f","K
    oppel_M12g" ] )
294 assignGeometry( "Koppel_M12", "SHAPE", [
    "Koppel_M12a","Koppel_M12b","Koppel_M12c","Koppel_M12d","Koppel_M12e","Koppel_M12f","K
    oppel_M12g" ] )
295 convertToReinforcement( [
    "Koppel_M12a","Koppel_M12b","Koppel_M12c","Koppel_M12d","Koppel_M12e","Koppel_M12f","K
    oppel_M12g" ] )
296 addElementData( "Koppel_M12" )
297 assignElementData( "Koppel_M12", "SHAPE", [
    "Koppel_M12a","Koppel_M12b","Koppel_M12c","Koppel_M12d","Koppel_M12e","Koppel_M12f","K
    oppel_M12g" ] )
298 setParameter( "DATA", "Koppel_M12", "INTERF", "BEAM" )
299 setReinforcementDiscretization( [
    "Koppel_M12a","Koppel_M12b","Koppel_M12c","Koppel_M12d","Koppel_M12e","Koppel_M12f","K
    oppel_M12g" ], "ELEMENT" )
300 setContinuousInInterfaces( [
    "Koppel_M12a","Koppel_M12b","Koppel_M12c","Koppel_M12d","Koppel_M12e","Koppel_M12f","K
    oppel_M12g" ], False )
301
302 #Material Reinforcement M8
303 addMaterial( "Reinfo_M8", "REINFO", "REBOND", [] )
304 setParameter( "MATERIAL", "Reinfo_M8", "REBARS/ELASTI/YOUNG", 200000 )
305 setParameter( "MATERIAL", "Reinfo_M8", "REBARS/POISON/POISON", 0.3 )
306 setParameter( "MATERIAL", "Reinfo_M8", "REBARS/MASS/DENSIT", 7.8e-09 )
307 setParameter( "MATERIAL", "Reinfo_M8", "REBARS/PLATYP", "VMISES" )
308 setParameter( "MATERIAL", "Reinfo_M8", "REBARS/PLASTI/TRESSH", "KAPSIG" )
309 setParameter( "MATERIAL", "Reinfo_M8", "REBARS/PLASTI/KAPSIG", [ 0, 550, 0.04725,
    594 ] )
310 setParameter( "MATERIAL", "Reinfo_M8", "RESLIP/SHFTYP", "BONDS6" )
311 setParameter( "MATERIAL", "Reinfo_M8", "RESLIP/DSNY", 1000 )
312 setParameter( "MATERIAL", "Reinfo_M8", "RESLIP/DSSX", 100 )
```

```
313 setParameter( "MATERIAL", "Reinfo_M8", "RESLIP/BONDS6/SLPVAL", [ 15.06, 6.02, 0.1,
    1, 2, 10, 0.4 ] )
314 addGeometry( "Reinfo_M8", "RELINE", "REBAR", [] )
315 setParameter( "GEOMET", "Reinfo_M8", "REITYP", "REITRU" )
316 setParameter( "GEOMET", "Reinfo_M8", "REITYP", "CIRBEA" )
317 setParameter( "GEOMET", "Reinfo_M8", "CIRBEA/CIRCLE", 8)
318 setParameter( "GEOMET", "Reinfo_M8", "TIPELM/SURFAC", 50.3 )
319 assignMaterial( "Reinfo_M8", "SHAPE", [
    "Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d","Reinfo_M8e","Reinfo_M8f","Reinfo_
    M8g","Reinfo_M8h","Reinfo_M8i","Reinfo_M8j","Reinfo_M8k","Reinfo_M8l","Reinfo_M8m","Re
    info_M8n","Reinfo_M8o" ] )
320 assignGeometry( "Reinfo_M8", "SHAPE", [
    "Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d","Reinfo_M8e","Reinfo_M8f","Reinfo_
    M8g","Reinfo_M8h","Reinfo_M8i","Reinfo_M8j","Reinfo_M8k","Reinfo_M8l","Reinfo_M8m","Re
    info_M8n","Reinfo_M8o" ] )
321 convertToReinforcement( [
    "Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d","Reinfo_M8e","Reinfo_M8f","Reinfo_
    M8g","Reinfo_M8h","Reinfo_M8i","Reinfo_M8j","Reinfo_M8k","Reinfo_M8l","Reinfo_M8m","Re
    info_M8n","Reinfo_M8o" ] )
322 addElementData( "Reinfo_M8" )
323 assignElementData( "Reinfo_M8", "SHAPE", [
    "Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d","Reinfo_M8e","Reinfo_M8f","Reinfo_
    M8g","Reinfo_M8h","Reinfo_M8i","Reinfo_M8j","Reinfo_M8k","Reinfo_M8l","Reinfo_M8m","Re
    info_M8n","Reinfo_M8o" ] )
324 setParameter( "DATA", "Reinfo_M8", "INTERF", "BEAM" )
325 setReinforcementDiscretization( [
    "Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d","Reinfo_M8e","Reinfo_M8f","Reinfo_
    M8g","Reinfo_M8h","Reinfo_M8i","Reinfo_M8j","Reinfo_M8k","Reinfo_M8l","Reinfo_M8m","Re
    info_M8n","Reinfo_M8o" ], "ELEMENT" )
326 setContinuousInInterfaces( [
    "Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d","Reinfo_M8e","Reinfo_M8f","Reinfo_
    M8g","Reinfo_M8h","Reinfo_M8i","Reinfo_M8j","Reinfo_M8k","Reinfo_M8l","Reinfo_M8m","Re
    info_M8n","Reinfo_M8o" ], False )
327
328 #Material Reinforcement M5
329 addMaterial( "Reinfo_M5", "REINFO", "REBOND", [ "ANCHOR" ] )
330 setParameter( "MATERIAL", "Reinfo_M5", "REBARS/ELASTI/YOUNG", 200000 )
331 setParameter( "MATERIAL", "Reinfo_M5", "REBARS/POISON/POISON", 0.3 )
332 setParameter( "MATERIAL", "Reinfo_M5", "REBARS/MASS/DENSIT", 7.8e-09 )
333 setParameter( "MATERIAL", "Reinfo_M5", "REBARS/PLATYP", "VMISES" )
334 setParameter( "MATERIAL", "Reinfo_M5", "REBARS/PLASTI/TRESSH", "KAPSIG" )
335 setParameter( "MATERIAL", "Reinfo_M5", "REBARS/PLASTI/KAPSIG", [ 0, 550, 0.04725,
    594 ] )
336 setParameter( "MATERIAL", "Reinfo_M5", "RESLIP/SHFTYP", "BONDS6" )
337 setParameter( "MATERIAL", "Reinfo_M5", "RESLIP/DSNY", 1000 )
338 setParameter( "MATERIAL", "Reinfo_M5", "RESLIP/DSSX", 100 )
339
340 setParameter( "MATERIAL", "Reinfo_M5", "RESLIP/BONDS6/SLPVAL", [ 0.42, 0.42, 0.01,
    0.02, 0.03, 0.04, 0.5 ] )
341
342 setParameter( "MATERIAL", "Reinfo_M5", "REANCH/DTIPNX", 10000 )
343 setParameter( "MATERIAL", "Reinfo_M5", "REANCH/DTIPSY", 10000 )
344 setParameter( "MATERIAL", "Reinfo_M5", "REANCH/TIPLOC", "BOTH" )
345
346 addGeometry( "Reinfo_M5", "RELINE", "REBAR", [] )
347 setParameter( "GEOMET", "Reinfo_M5", "REITYP", "REITRU" )
348 setParameter( "GEOMET", "Reinfo_M5", "REITYP", "CIRBEA" )
349 setParameter( "GEOMET", "Reinfo_M5", "CIRBEA/CIRCLE", 5 )
350 setParameter( "GEOMET", "Reinfo_M5", "TIPELM/SURFAC", 19.6 )
351 assignMaterial( "Reinfo_M5", "SHAPE", [
    "Bar_M41al","Bar_M41ar","Bar_M41bl","Bar_M41br","Bar_M41cl","Bar_M41cr","Bar_M41dl","B
    ar_M41dr","Bar_M41el","Bar_M41er","Bar_M41fl","Bar_M41fr","Bar_M41gl","Bar_M41gr","Bar
    _M41hl","Bar_M41hr","Bar_M41il","Bar_M41ir","Bar_M41jl","Bar_M41jr","Bar_M42al","Bar_M
    42ar","Bar_M42bl","Bar_M42br","Bar_M42cl","Bar_M42cr","Bar_M42dl","Bar_M42dr","Bar_M42
    el","Bar_M42er","Bar_M42fl","Bar_M42fr","Bar_M42gl","Bar_M42gr","Bar_M42hl","Bar_M42hr
    ","Bar_M42il","Bar_M42ir","Bar_M42jl","Bar_M42jr","Bar_M43al","Bar_M43ar","Bar_M43bl",
    "Bar_M43br","Bar_M43cl","Bar_M43cr","Bar_M43dl","Bar_M43dr","Bar_M43el","Bar_M43er","B
    ar_M43fl","Bar_M43fr","Bar_M43gl","Bar_M43gr","Bar_M43hl","Bar_M43hr","Bar_M43il","Bar
    _M43ir","Bar_M43jl","Bar_M43jr" ] )
352 assignGeometry( "Reinfo_M5", "SHAPE", [
    "Bar_M41al","Bar_M41ar","Bar_M41bl","Bar_M41br","Bar_M41cl","Bar_M41cr","Bar_M41dl","B
```

```
ar_M41dr","Bar_M41el","Bar_M41er","Bar_M41fl","Bar_M41fr","Bar_M41gl","Bar_M41gr","Bar
_M41hl","Bar_M41hr","Bar_M41il","Bar_M41ir","Bar_M41jl","Bar_M41jr","Bar_M42al","Bar_M
42ar","Bar_M42bl","Bar_M42br","Bar_M42cl","Bar_M42cr","Bar_M42dl","Bar_M42dr","Bar_M42
el","Bar_M42er","Bar_M42fl","Bar_M42fr","Bar_M42gl","Bar_M42gr","Bar_M42hl","Bar_M42hr
","Bar_M42il","Bar_M42ir","Bar_M42jl","Bar_M42jr","Bar_M43al","Bar_M43ar","Bar_M43bl",
"Bar_M43br","Bar_M43cl","Bar_M43cr","Bar_M43dl","Bar_M43dr","Bar_M43el","Bar_M43er","B
ar_M43fl","Bar_M43fr","Bar_M43gl","Bar_M43gr","Bar_M43hl","Bar_M43hr","Bar_M43il","Bar
_M43ir","Bar_M43jl","Bar_M43jr" ] )
353 convertToReinforcement( [
"Bar_M41al","Bar_M41ar","Bar_M41bl","Bar_M41br","Bar_M41cl","Bar_M41cr","Bar_M41dl","B
ar_M41dr","Bar_M41el","Bar_M41er","Bar_M41fl","Bar_M41fr","Bar_M41gl","Bar_M41gr","Bar
_M41hl","Bar_M41hr","Bar_M41il","Bar_M41ir","Bar_M41jl","Bar_M41jr","Bar_M42al","Bar_M
42ar","Bar_M42bl","Bar_M42br","Bar_M42cl","Bar_M42cr","Bar_M42dl","Bar_M42dr","Bar_M42
el","Bar_M42er","Bar_M42fl","Bar_M42fr","Bar_M42gl","Bar_M42gr","Bar_M42hl","Bar_M42hr
","Bar_M42il","Bar_M42ir","Bar_M42jl","Bar_M42jr","Bar_M43al","Bar_M43ar","Bar_M43bl",
"Bar_M43br","Bar_M43cl","Bar_M43cr","Bar_M43dl","Bar_M43dr","Bar_M43el","Bar_M43er","B
ar_M43fl","Bar_M43fr","Bar_M43gl","Bar_M43gr","Bar_M43hl","Bar_M43hr","Bar_M43il","Bar
_M43ir","Bar_M43jl","Bar_M43jr" ] )
354 addElementData( "Bar" )
355 assignElementData( "Bar", "SHAPE", [
"Bar_M41al","Bar_M41ar","Bar_M41bl","Bar_M41br","Bar_M41cl","Bar_M41cr","Bar_M41dl","B
ar_M41dr","Bar_M41el","Bar_M41er","Bar_M41fl","Bar_M41fr","Bar_M41gl","Bar_M41gr","Bar
_M41hl","Bar_M41hr","Bar_M41il","Bar_M41ir","Bar_M41jl","Bar_M41jr","Bar_M42al","Bar_M
42ar","Bar_M42bl","Bar_M42br","Bar_M42cl","Bar_M42cr","Bar_M42dl","Bar_M42dr","Bar_M42
el","Bar_M42er","Bar_M42fl","Bar_M42fr","Bar_M42gl","Bar_M42gr","Bar_M42hl","Bar_M42hr
","Bar_M42il","Bar_M42ir","Bar_M42jl","Bar_M42jr","Bar_M43al","Bar_M43ar","Bar_M43bl",
"Bar_M43br","Bar_M43cl","Bar_M43cr","Bar_M43dl","Bar_M43dr","Bar_M43el","Bar_M43er","B
ar_M43fl","Bar_M43fr","Bar_M43gl","Bar_M43gr","Bar_M43hl","Bar_M43hr","Bar_M43il","Bar
_M43ir","Bar_M43jl","Bar_M43jr" ] )
356 setParameter( "DATA", "Bar", "./INTERF", [] )
357 setParameter( "DATA", "Bar", "INTERF", "BEAM" )
358 setReinforcementDiscretization( [
"Bar_M41al","Bar_M41ar","Bar_M41bl","Bar_M41br","Bar_M41cl","Bar_M41cr","Bar_M41dl","B
ar_M41dr","Bar_M41el","Bar_M41er","Bar_M41fl","Bar_M41fr","Bar_M41gl","Bar_M41gr","Bar
_M41hl","Bar_M41hr","Bar_M41il","Bar_M41ir","Bar_M41jl","Bar_M41jr","Bar_M42al","Bar_M
42ar","Bar_M42bl","Bar_M42br","Bar_M42cl","Bar_M42cr","Bar_M42dl","Bar_M42dr","Bar_M42
el","Bar_M42er","Bar_M42fl","Bar_M42fr","Bar_M42gl","Bar_M42gr","Bar_M42hl","Bar_M42hr
","Bar_M42il","Bar_M42ir","Bar_M42jl","Bar_M42jr","Bar_M43al","Bar_M43ar","Bar_M43bl",
"Bar_M43br","Bar_M43cl","Bar_M43cr","Bar_M43dl","Bar_M43dr","Bar_M43el","Bar_M43er","B
ar_M43fl","Bar_M43fr","Bar_M43gl","Bar_M43gr","Bar_M43hl","Bar_M43hr","Bar_M43il","Bar
_M43ir","Bar_M43jl","Bar_M43jr" ], "SECTION" )
359
360 #Composed Line
361 addGeometry( "Composed Line", "LINE", "COMLIN", [] )
362 setParameter( "GEOMET", "Composed Line", "DISTAN/THICK", 250 )
363 setElementClassType( "SHAPE", [ "Composed Line" ], "COMLIN" )
364 assignGeometry( "Composed Line", "SHAPE", [ "Composed Line" ] )
365 resetElementData( "SHAPE", [ "Composed Line" ] )
366
367 #Assign Supports
368 addSet( "GEOMETRYSUPPORTSET", "Support" )
369
370 createPointSupport( "Support", "Support" )
371 setParameter( "GEOMETRYSUPPORT", "Support", "AXES", [ 1, 2 ] )
372 setParameter( "GEOMETRYSUPPORT", "Support", "TRANSL", [ 0, 1, 0 ] )
373 setParameter( "GEOMETRYSUPPORT", "Support", "ROTATI", [ 0, 0, 0 ] )
374 attach( "GEOMETRYSUPPORT", "Support", "Support", [[ 300, -30, 0 ]] )
375
376 addSet( "GEOMETRYSUPPORTSET", "Sym_X" )
377
378 createLineSupport( "Sym_X", "Sym_X" )
379 setParameter( "GEOMETRYSUPPORT", "Sym_X", "AXES", [ 1, 2 ] )
380 setParameter( "GEOMETRYSUPPORT", "Sym_X", "TRANSL", [ 1, 0, 0 ] )
381 setParameter( "GEOMETRYSUPPORT", "Sym_X", "ROTATI", [ 0, 0, 0 ] )
382 attach( "GEOMETRYSUPPORT", "Sym_X", "Pressure Layer_1", [[ 1900, 160, 0 ]] )
383
384 createPointSupport( "Symp_X", "Symp_X" )
385 setParameter( "GEOMETRYSUPPORT", "Symp_X", "AXES", [ 1, 2 ] )
386 setParameter( "GEOMETRYSUPPORT", "Symp_X", "TRANSL", [ 1, 0, 0 ] )
387 setParameter( "GEOMETRYSUPPORpT", "Symp_X", "ROTATI", [ 0, 0, 0 ] )
388 attach( "GEOMETRYSUPPORT", "Symp_X", "Koppel_M12a", [[ 1900, 76, 0 ]] )
```

```
attach( "GEOMETRYSUPPORT", "Symp_X", "Koppel_M12b", [[ 1900, 76, 0 ]] )
attach( "GEOMETRYSUPPORT", "Symp_X", "Koppel_M12c", [[ 1900, 76, 0 ]] )
attach( "GEOMETRYSUPPORT", "Symp_X", "Koppel_M12d", [[ 1900, 76, 0 ]] )
attach( "GEOMETRYSUPPORT", "Symp_X", "Koppel_M12e", [[ 1900, 76, 0 ]] )
attach( "GEOMETRYSUPPORT", "Symp_X", "Koppel_M12f", [[ 1900, 76, 0 ]] )
attach( "GEOMETRYSUPPORT", "Symp_X", "Koppel_M12g", [[ 1900, 76, 0 ]] )

addSet( "GEOMETRYSUPPORTSET", "Load point" )

createPointSupport( "Load point", "Load point" )
setParameter( "GEOMETRYSUPPORT", "Load point", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Load point", "TRANSL", [ 0, 1, 0 ] )
setParameter( "GEOMETRYSUPPORT", "Load point", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Load point", "Loadplate", [[ 1100, 280, 0 ]] )

#Assign Loads
addSet( "GEOMETRYLOADSET", "Load" )

createPointLoad( "Load", "Load" )
setParameter( "GEOMETRYLOAD", "Load", "LODTYP", "DEFORM" )
setParameter( "GEOMETRYLOAD", "Load", "DEFORM/TR/VALUE", -0.1 )
setParameter( "GEOMETRYLOAD", "Load", "DEFORM/TR/DIRECT", 2 )
attach( "GEOMETRYLOAD", "Load", "Loadplate", [[ 1100, 280, 0 ]] )

#Set mesh properties
setElementSize( [  "Loadplate", "Pressure Layer_1", "Slab_1",
"Koppel_M12a","Koppel_M12b","Koppel_M12c","Koppel_M12d","Koppel_M12e","Koppel_M12f","K
oppel_M12g","Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d","Reinfo_M8e","Reinfo_M
8f","Reinfo_M8g","Reinfo_M8h","Reinfo_M8i","Reinfo_M8j","Reinfo_M8k","Reinfo_M8l","Rei
nfo_M8m","Reinfo_M8n","Reinfo_M8o","Bar_M41al","Bar_M41ar","Bar_M41bl","Bar_M41br","Ba
r_M41cl","Bar_M41cr","Bar_M41dl","Bar_M41dr","Bar_M41el","Bar_M41er","Bar_M41fl","Bar_
M41fr","Bar_M41gl","Bar_M41gr","Bar_M41hl","Bar_M41hr","Bar_M41il","Bar_M41ir","Bar_M4
1jl","Bar_M41jr","Bar_M42al","Bar_M42ar","Bar_M42bl","Bar_M42br","Bar_M42cl","Bar_M42c
r","Bar_M42dl","Bar_M42dr","Bar_M42el","Bar_M42er","Bar_M42fl","Bar_M42fr","Bar_M42gl"
,"Bar_M42gr","Bar_M42hl","Bar_M42hr","Bar_M42il","Bar_M42ir","Bar_M42jl","Bar_M42jr","
Bar_M43al","Bar_M43ar","Bar_M43bl","Bar_M43br","Bar_M43cl","Bar_M43cr","Bar_M43dl","Ba
r_M43dr","Bar_M43el","Bar_M43er","Bar_M43fl","Bar_M43fr","Bar_M43gl","Bar_M43gr","Bar_
M43hl","Bar_M43hr","Bar_M43il","Bar_M43ir","Bar_M43jl","Bar_M43jr" ], 10, -1, True )
setMesherType( [  "Loadplate", "Pressure Layer_1", "Slab_1",
"Koppel_M12a","Koppel_M12b","Koppel_M12c","Koppel_M12d","Koppel_M12e","Koppel_M12f","K
oppel_M12g","Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d","Reinfo_M8e","Reinfo_M
8f","Reinfo_M8g","Reinfo_M8h","Reinfo_M8i","Reinfo_M8j","Reinfo_M8k","Reinfo_M8l","Rei
nfo_M8m","Reinfo_M8n","Reinfo_M8o","Bar_M41al","Bar_M41ar","Bar_M41bl","Bar_M41br","Ba
r_M41cl","Bar_M41cr","Bar_M41dl","Bar_M41dr","Bar_M41el","Bar_M41er","Bar_M41fl","Bar_
M41fr","Bar_M41gl","Bar_M41gr","Bar_M41hl","Bar_M41hr","Bar_M41il","Bar_M41ir","Bar_M4
1jl","Bar_M41jr","Bar_M42al","Bar_M42ar","Bar_M42bl","Bar_M42br","Bar_M42cl","Bar_M42c
r","Bar_M42dl","Bar_M42dr","Bar_M42el","Bar_M42er","Bar_M42fl","Bar_M42fr","Bar_M42gl"
,"Bar_M42gr","Bar_M42hl","Bar_M42hr","Bar_M42il","Bar_M42ir","Bar_M42jl","Bar_M42jr","
Bar_M43al","Bar_M43ar","Bar_M43bl","Bar_M43br","Bar_M43cl","Bar_M43cr","Bar_M43dl","Ba
r_M43dr","Bar_M43el","Bar_M43er","Bar_M43fl","Bar_M43fr","Bar_M43gl","Bar_M43gr","Bar_
M43hl","Bar_M43hr","Bar_M43il","Bar_M43ir","Bar_M43jl","Bar_M43jr" ], "HEXQUAD" )
clearMidSideNodeLocation( [  "Loadplate", "Pressure Layer_1", "Slab_1",
"Koppel_M12a","Koppel_M12b","Koppel_M12c","Koppel_M12d","Koppel_M12e","Koppel_M12f","K
oppel_M12g","Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d","Reinfo_M8e","Reinfo_M
8f","Reinfo_M8g","Reinfo_M8h","Reinfo_M8i","Reinfo_M8j","Reinfo_M8k","Reinfo_M8l","Rei
nfo_M8m","Reinfo_M8n","Reinfo_M8o","Bar_M41al","Bar_M41ar","Bar_M41bl","Bar_M41br","Ba
r_M41cl","Bar_M41cr","Bar_M41dl","Bar_M41dr","Bar_M41el","Bar_M41er","Bar_M41fl","Bar_
M41fr","Bar_M41gl","Bar_M41gr","Bar_M41hl","Bar_M41hr","Bar_M41il","Bar_M41ir","Bar_M4
1jl","Bar_M41jr","Bar_M42al","Bar_M42ar","Bar_M42bl","Bar_M42br","Bar_M42cl","Bar_M42c
r","Bar_M42dl","Bar_M42dr","Bar_M42el","Bar_M42er","Bar_M42fl","Bar_M42fr","Bar_M42gl"
,"Bar_M42gr","Bar_M42hl","Bar_M42hr","Bar_M42il","Bar_M42ir","Bar_M42jl","Bar_M42jr","
Bar_M43al","Bar_M43ar","Bar_M43bl","Bar_M43br","Bar_M43cl","Bar_M43cr","Bar_M43dl","Ba
r_M43dr","Bar_M43el","Bar_M43er","Bar_M43fl","Bar_M43fr","Bar_M43gl","Bar_M43gr","Bar_
M43hl","Bar_M43hr","Bar_M43il","Bar_M43ir","Bar_M43jl","Bar_M43jr" ] )
setElementSize( [ "Support","Pressure Layer", "Slab" ], 50, -1, True )
setMesherType( [ "Support","Pressure Layer", "Slab" ], "HEXQUAD" )
setEdgeGrading( "Pressure Layer", [[ 500, 70, 0 ]], [], 50, 50, 10, 0.25, False,
True )
setEdgeGrading( "Slab", [], [[ 500, 70, 0 ]], 50, 50, 10, 0.25, False, True )
generateMesh( [] )
hideView( "GEOM" )
```

```
423  showView( "MESH" )
424
425  #Analysis
426  addAnalysis( "Analysis1" )
427  addAnalysisCommand( "Analysis1", "NONLIN", "Structural nonlinear" )
428  renameAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "EXECUT(1)",
     "Load" )
429  addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
     "EXECUT(1)/LOAD/LOADNR" )
430  setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
     "EXECUT(1)/LOAD/LOADNR", 1 )
431  # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
     "EXECUT(1)/LOAD/STEPS/STEPTY", "AUTOMA" )
432  # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
     "EXECUT(1)/LOAD/STEPS/AUTOMA/SIZES", 250 )
433  # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
     "EXECUT(1)/LOAD/STEPS/AUTOMA/MAXSIZ", 0.01 )
434  # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
     "EXECUT(1)/LOAD/STEPS/AUTOMA/MINSIZ", 0.00001 )
435  # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
     "EXECUT(1)/LOAD/STEPS/AUTOMA/CUTBCK", 0.25 )
436  # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
     "EXECUT(1)/LOAD/STEPS/AUTOMA/MAXSTP", 100 )
437  # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
     "EXECUT(1)/LOAD/STEPS/AUTOMA/ARCLEN", True )
438  # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
     "EXECUT(1)/LOAD/STEPS/AUTOMA/ARCLEN/REGULA/SET" )
439  # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
     "EXECUT(1)/LOAD/STEPS/AUTOMA/ARCLEN/REGULA/SET(1)/NODES(1)/RNGNRS", "1-9361" )
440  # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
     "EXECUT(1)/LOAD/STEPS/AUTOMA/ARCLEN/REGULA/SET(1)/DIRECT", 2 )
441  setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
     "EXECUT(1)/LOAD/STEPS/EXPLIC/SIZES", "1(100)" )
442  # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
     "EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN", True )
443  # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
     "EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN/REGULA/SET" )
444  # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
     "EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN/REGULA/SET(1)/NODES(1)/RNGNRS", "734 755 756 761
     835-851 940 941-2365(89) 2548-2567 2652 2653 2831 2832 3010 3011 3189 3190 3368 3369
     3547 3548 3726 3727 3905 3906 4084 4085 4263 4264 4442 4443 4621 4622 4800 4801 4979
     4980 5158 5159 5337 5338 5516 5517 5695 5784 5786 5875-5881 5970 5971-6505(89)
     6684-6693 6871 6872 7050 7051 7229 7230 7408 7409 7587 7588 7766 7993-7995 8133 8134
     8272 8273 8411 8412 8550 8551 8689 8690 8828 8829" )
445  # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
     "EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN/REGULA/SET(1)/DIRECT", 2 )
446  setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
     "EXECUT(1)/ITERAT/MAXITE", 250 )
447  setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
     "EXECUT(1)/ITERAT/LINESE", False )
448  setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
     "EXECUT(1)/ITERAT/CONVER/ENERGY", True )
449  setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
     "EXECUT(1)/ITERAT/CONVER/DISPLA", False )
450  setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
     "EXECUT(1)/ITERAT/CONVER/FORCE", True )
451  # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
     "EXECUT(1)/ITERAT/CONVER/ENERGY/TOLCON", 0.001 )
452  setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
     "EXECUT(1)/ITERAT/CONVER/ENERGY/NOCONV", "CONTIN" )
453  # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
     "EXECUT(1)/ITERAT/CONVER/FORCE/TOLCON", 0.01 )
454  setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
     "EXECUT(1)/ITERAT/CONVER/FORCE/NOCONV", "CONTIN" )
455  # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
     "EXECUT(1)/ITERAT/METHOD/METNAM", "SECANT" )
456
457  setAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "OUTPUT(1)/SELTYP",
     "USER" )
458  addAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "OUTPUT(1)/USER" )
459  addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
```

```
    "OUTPUT(1)/USER/DISPLA(1)/TOTAL" )
460 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/FORCE(1)/REACTI" )
461 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(1)/TOTAL/GREEN" )
462 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(2)/TOTAL/GREEN/PRINCI" )
463 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(3)/CRKWDT" )
464 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(4)/CRACK/GREEN" )
465 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(5)/CRKWDT/GREEN/PRINCI" )
466 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(1)/TOTAL/CAUCHY" )
467 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(2)/TOTAL/CAUCHY/PRINCI" )
468 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(3)/TOTAL/FORCE/LOCAL" )
469 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(4)/TOTAL/MOMENT/LOCAL" )
470 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(5)/EFFECT/TRACTI/GLOBAL" )
471 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(6)/EFFECT/TRACTI/LOCAL" )
472 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(7)/EFFECT/TRACTI/FRICTI" )
473
474 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(1)/TOTAL/GREEN/GLOBAL/LOCATI", "INTPNT" )
475 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(1)/TOTAL/CAUCHY/GLOBAL/LOCATI", "INTPNT" )
476
477 #Run Analysis
478 # runSolver( [] )
479 # showView( "RESULT" )
480 fitAll( )
481
482
483
```

B4 - T22 V11

```
newProject( "D:/Afstuderen/Python/Proefstukken/DIANA Output/T22 V11 Reduced Elements FRICEL", 10 )


###Project Settings###
setModelAnalysisAspects( [ "STRUCT" ] )
setModelDimension( "2D" )
setDefaultMeshOrder( "QUADRATIC" )
setDefaultMesherType( "HEXQUAD" )
setDefaultMidSideNodeLocation( "ONSHAP" )
setUnit( "LENGTH", "MM" )
setUnit( "FORCE", "N" )
setUnit( "ANGLE", "DEGREE" )


#Create Shapes
createSheet( "Pressure Layer", [[ 0, 70, 0 ],[ 1900, 70, 0 ],[ 1900, 300, 0 ],[ 1150, 300, 0 ],[1050, 300, 0 ],[ 0, 300, 0 ]] )
createSheet( "Slab", [[ 0, 0, 0 ],[ 250, 0, 0 ],[ 350, 0, 0 ],[ 1900, 0, 0 ],[ 1900, 70, 0],[ 0, 70, 0 ]] )

createSheet( "Support", [[ 250, 0, 0 ],[ 350, 0, 0 ],[ 350, -30, 0 ],[ 300, -30, 0 ],[ 250, -30, 0 ]] )
createSheet( "Loadplate", [[1050, 300, 0 ],[ 1150, 300, 0 ],[ 1150, 330, 0 ],[ 1100, 330, 0 ],[ 1050, 330, 0 ]] )

#Create Reinforcement
createLine( "Koppel_M12a", [ 1200, 76, 0 ], [ 1900, 76, 0 ] )
createLine( "Koppel_M12b", [ 1200, 76, 0 ], [ 1900, 76, 0 ] )
createLine( "Koppel_M12c", [ 1200, 76, 0 ], [ 1900, 76, 0 ] )
createLine( "Koppel_M12d", [ 1200, 76, 0 ], [ 1900, 76, 0 ] )
createLine( "Koppel_M12e", [ 1200, 76, 0 ], [ 1900, 76, 0 ] )
createLine( "Koppel_M12f", [ 1200, 76, 0 ], [ 1900, 76, 0 ] )
createLine( "Koppel_M12g", [ 1200, 76, 0 ], [ 1900, 76, 0 ] )

createLine( "Reinfo_M8a", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8b", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8c", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8d", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8e", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8f", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8g", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8h", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8i", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8j", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8k", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8l", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8m", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8n", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8o", [ 0, 35, 0 ], [ 1865, 35, 0 ] )

createLine( "Bar_M41al", [ 135, 45.5, 0],[ 175, 235.5, 0] )
createLine( "Bar_M41ar", [ 215, 45.5, 0],[ 175, 235.5, 0] )
createLine( "Bar_M41bl", [ 135, 45.5, 0],[ 175, 235.5, 0] )
createLine( "Bar_M41br", [ 215, 45.5, 0],[ 175, 235.5, 0] )
createLine( "Bar_M41cl", [ 135, 45.5, 0],[ 175, 235.5, 0] )
createLine( "Bar_M41cr", [ 215, 45.5, 0],[ 175, 235.5, 0] )
createLine( "Bar_M41dl", [ 135, 45.5, 0],[ 175, 235.5, 0] )
createLine( "Bar_M41dr", [ 215, 45.5, 0],[ 175, 235.5, 0] )
createLine( "Bar_M41el", [ 135, 45.5, 0],[ 175, 235.5, 0] )
createLine( "Bar_M41er", [ 215, 45.5, 0],[ 175, 235.5, 0] )
createLine( "Bar_M41fl", [ 135, 45.5, 0],[ 175, 235.5, 0] )
createLine( "Bar_M41fr", [ 215, 45.5, 0],[ 175, 235.5, 0] )
createLine( "Bar_M41gl", [ 135, 45.5, 0],[ 175, 235.5, 0] )
createLine( "Bar_M41gr", [ 215, 45.5, 0],[ 175, 235.5, 0] )
createLine( "Bar_M41hl", [ 135, 45.5, 0],[ 175, 235.5, 0] )
createLine( "Bar_M41hr", [ 215, 45.5, 0],[ 175, 235.5, 0] )
createLine( "Bar_M41il", [ 135, 45.5, 0],[ 175, 235.5, 0] )
createLine( "Bar_M41ir", [ 215, 45.5, 0],[ 175, 235.5, 0] )
createLine( "Bar_M41jl", [ 135, 45.5, 0],[ 175, 235.5, 0] )
createLine( "Bar_M41jr", [ 215, 45.5, 0],[ 175, 235.5, 0] )
```

```
67
68  createLine( "Bar_M42al", [ 735, 45.5, 0],[ 775, 235.5, 0] )
69  createLine( "Bar_M42ar", [ 815, 45.5, 0],[ 775, 235.5, 0] )
70  createLine( "Bar_M42bl", [ 735, 45.5, 0],[ 775, 235.5, 0] )
71  createLine( "Bar_M42br", [ 815, 45.5, 0],[ 775, 235.5, 0] )
72  createLine( "Bar_M42cl", [ 735, 45.5, 0],[ 775, 235.5, 0] )
73  createLine( "Bar_M42cr", [ 815, 45.5, 0],[ 775, 235.5, 0] )
74  createLine( "Bar_M42dl", [ 735, 45.5, 0],[ 775, 235.5, 0] )
75  createLine( "Bar_M42dr", [ 815, 45.5, 0],[ 775, 235.5, 0] )
76  createLine( "Bar_M42el", [ 735, 45.5, 0],[ 775, 235.5, 0] )
77  createLine( "Bar_M42er", [ 815, 45.5, 0],[ 775, 235.5, 0] )
78  createLine( "Bar_M42fl", [ 735, 45.5, 0],[ 775, 235.5, 0] )
79  createLine( "Bar_M42fr", [ 815, 45.5, 0],[ 775, 235.5, 0] )
80  createLine( "Bar_M42gl", [ 735, 45.5, 0],[ 775, 235.5, 0] )
81  createLine( "Bar_M42gr", [ 815, 45.5, 0],[ 775, 235.5, 0] )
82  createLine( "Bar_M42hl", [ 735, 45.5, 0],[ 775, 235.5, 0] )
83  createLine( "Bar_M42hr", [ 815, 45.5, 0],[ 775, 235.5, 0] )
84  createLine( "Bar_M42il", [ 735, 45.5, 0],[ 775, 235.5, 0] )
85  createLine( "Bar_M42ir", [ 815, 45.5, 0],[ 775, 235.5, 0] )
86  createLine( "Bar_M42jl", [ 735, 45.5, 0],[ 775, 235.5, 0] )
87  createLine( "Bar_M42jr", [ 815, 45.5, 0],[ 775, 235.5, 0] )
88
89  createLine( "Bar_M43al", [ 1460, 45.5, 0],[ 1500, 235.5, 0] )
90  createLine( "Bar_M43ar", [ 1540, 45.5, 0],[ 1500, 235.5, 0] )
91  createLine( "Bar_M43bl", [ 1460, 45.5, 0],[ 1500, 235.5, 0] )
92  createLine( "Bar_M43br", [ 1540, 45.5, 0],[ 1500, 235.5, 0] )
93  createLine( "Bar_M43cl", [ 1460, 45.5, 0],[ 1500, 235.5, 0] )
94  createLine( "Bar_M43cr", [ 1540, 45.5, 0],[ 1500, 235.5, 0] )
95  createLine( "Bar_M43dl", [ 1460, 45.5, 0],[ 1500, 235.5, 0] )
96  createLine( "Bar_M43dr", [ 1540, 45.5, 0],[ 1500, 235.5, 0] )
97  createLine( "Bar_M43el", [ 1460, 45.5, 0],[ 1500, 235.5, 0] )
98  createLine( "Bar_M43er", [ 1540, 45.5, 0],[ 1500, 235.5, 0] )
99  createLine( "Bar_M43fl", [ 1460, 45.5, 0],[ 1500, 235.5, 0] )
100 createLine( "Bar_M43fr", [ 1540, 45.5, 0],[ 1500, 235.5, 0] )
101 createLine( "Bar_M43gl", [ 1460, 45.5, 0],[ 1500, 235.5, 0] )
102 createLine( "Bar_M43gr", [ 1540, 45.5, 0],[ 1500, 235.5, 0] )
103 createLine( "Bar_M43hl", [ 1460, 45.5, 0],[ 1500, 235.5, 0] )
104 createLine( "Bar_M43hr", [ 1540, 45.5, 0],[ 1500, 235.5, 0] )
105 createLine( "Bar_M43il", [ 1460, 45.5, 0],[ 1500, 235.5, 0] )
106 createLine( "Bar_M43ir", [ 1540, 45.5, 0],[ 1500, 235.5, 0] )
107 createLine( "Bar_M43jl", [ 1460, 45.5, 0],[ 1500, 235.5, 0] )
108 createLine( "Bar_M43jr", [ 1540, 45.5, 0],[ 1500, 235.5, 0] )
109
110 createLine( "Composed Line", [ 0, 150, 0 ], [ 1900, 150, 0 ] )
111
112 createSheet( "Mesh Sheet", [[ 1000, 0, 0 ],[ 1000, 300, 0 ],[ 1000, 300, 1000 ],[
    1000, 0, 1000 ]] )
113
114 #Splitting Mesh Parts
115 subtract( "Pressure Layer", [ "Mesh Sheet" ], True, True )
116 subtract( "Slab", [ "Mesh Sheet" ], False, True )
117
118 #Regroup Material Parts
119 addSet( "SHAPESET", "Coupling Reinforcement" )
120 moveToShapeSet( [
    "Koppel_M12a","Koppel_M12b","Koppel_M12c","Koppel_M12d","Koppel_M12e","Koppel_M12f","K
    oppel_M12g" ], "Coupling Reinforcement" )
121 addSet( "SHAPESET", "Longitudinal Reinforcement" )
122 moveToShapeSet( [
    "Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d","Reinfo_M8e","Reinfo_M8f","Reinfo_
    M8g","Reinfo_M8h","Reinfo_M8i","Reinfo_M8j","Reinfo_M8k","Reinfo_M8l","Reinfo_M8m","Re
    info_M8n","Reinfo_M8o" ], "Longitudinal Reinforcement" )
123 addSet( "SHAPESET", "Lattice girders" )
124 moveToShapeSet( [
    "Bar_M41al","Bar_M41ar","Bar_M41bl","Bar_M41br","Bar_M41cl","Bar_M41cr","Bar_M41dl","B
    ar_M41dr","Bar_M41el","Bar_M41er","Bar_M41fl","Bar_M41fr","Bar_M41gl","Bar_M41gr","Bar
    _M41hl","Bar_M41hr","Bar_M41il","Bar_M41ir","Bar_M41jl","Bar_M41jr","Bar_M42al","Bar_M
    42ar","Bar_M42bl","Bar_M42br","Bar_M42cl","Bar_M42cr","Bar_M42dl","Bar_M42dr","Bar_M42
    el","Bar_M42er","Bar_M42fl","Bar_M42fr","Bar_M42gl","Bar_M42gr","Bar_M42hl","Bar_M42hr
    ","Bar_M42il","Bar_M42ir","Bar_M42jl","Bar_M42jr","Bar_M43al","Bar_M43ar","Bar_M43bl",
    "Bar_M43br","Bar_M43cl","Bar_M43cr","Bar_M43dl","Bar_M43dr","Bar_M43el","Bar_M43er","B
```

```
    ar_M43fl","Bar_M43fr","Bar_M43gl","Bar_M43gr","Bar_M43hl","Bar_M43hr","Bar_M43il","Bar
    _M43ir","Bar_M43jl","Bar_M43jr" ], "Lattice girders" )
125 addSet( "SHAPESET", "Steel plates" )
126 moveToShapeSet( [ "Loadplate", "Support" ], "Steel plates" )
127 rename( "SHAPESET", "Shapes", "Concrete" )
128 addSet( "SHAPESET", "Composed Line" )
129 moveToShapeSet( [ "Composed Line" ], "Composed Line" )
130
131 #Material Precast concrete
132 addMaterial( "Precast concrete", "CONCR", "TSCR", [] )
133 setParameter( "MATERIAL", "Precast concrete", "LINEAR/ELASTI/YOUNG", 34749 )
134 setParameter( "MATERIAL", "Precast concrete", "LINEAR/ELASTI/POISON", 0.2 )
135 setParameter( "MATERIAL", "Precast concrete", "LINEAR/MASS/DENSIT", 2.4e-09 )
136 setParameter( "MATERIAL", "Precast concrete", "MODTYP/TOTCRK", "ROTATE" )
137 setParameter( "MATERIAL", "Precast concrete", "TENSIL/TENCRV", "HORDYK" )
138 setParameter( "MATERIAL", "Precast concrete", "TENSIL/TENSTR", 3.38 )
139 setParameter( "MATERIAL", "Precast concrete", "TENSIL/GF1", 0.1454 )
140 setParameter( "MATERIAL", "Precast concrete", "TENSIL/CBSPEC", "ROTS" )
141 setParameter( "MATERIAL", "Precast concrete", "TENSIL/POISRE/POIRED", "DAMAGE" )
142 # setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMCRV", "MC2010" )
143 # setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMSTR", 55.9 )
144 # setParameter( "MATERIAL", "Precast concrete", "COMPRS/EPSC1", 0.0024 )
145 # setParameter( "MATERIAL", "Precast concrete", "COMPRS/EPSCU", 0.0035 )
146 # setParameter( "MATERIAL", "Precast concrete", "COMPRS/CONFIN/CNFCRV", "NONE" )
147 setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMCRV", "CONSTA" )
148 setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMSTR", 45.9 )
149
150 addGeometry( "Precast concrete", "SHEET", "MEMBRA", [] )
151 setParameter( "GEOMET", "Precast concrete", "THICK", 1000 )
152 setParameter( "GEOMET", "Precast concrete", "LOCAXS", True )
153 setParameter( "GEOMET", "Precast concrete", "LOCAXS/XAXIS", [ 1, 0, 0 ] )
154
155 addElementData( "Precast concrete" )
156 setElementClassType( "SHAPE", [ "Slab", "Slab_1" ], "MEMBRA" )
157 assignMaterial( "Precast concrete", "SHAPE", [ "Slab", "Slab_1" ] )
158 assignGeometry( "Precast concrete", "SHAPE", [ "Slab", "Slab_1" ] )
159 assignElementData( "Precast concrete", "SHAPE", [ "Slab", "Slab_1" ] )
160
161 # setParameter( "DATA", "Precast concrete", "INTEGR", "REDUCE" )
162
163 #Material Pressure layer
164 addMaterial( "Pressure layer", "CONCR", "TSCR", [] )
165 setParameter( "MATERIAL", "Pressure layer", "LINEAR/ELASTI/YOUNG", 29580 )
166 setParameter( "MATERIAL", "Pressure layer", "LINEAR/ELASTI/POISON", 0.2 )
167 setParameter( "MATERIAL", "Pressure layer", "LINEAR/MASS/DENSIT", 2.4e-09 )
168 setParameter( "MATERIAL", "Pressure layer", "MODTYP/TOTCRK", "ROTATE" )
169 setParameter( "MATERIAL", "Pressure layer", "TENSIL/TENCRV", "HORDYK" )
170 setParameter( "MATERIAL", "Pressure layer", "TENSIL/TENSTR", 2.12 )
171 setParameter( "MATERIAL", "Pressure layer", "TENSIL/GF1", 0.1320 )
172 setParameter( "MATERIAL", "Pressure layer", "TENSIL/CBSPEC", "ROTS" )
173 setParameter( "MATERIAL", "Pressure layer", "TENSIL/POISRE/POIRED", "DAMAGE" )
174 # setParameter( "MATERIAL", "Pressure layer", "COMPRS/COMCRV", "MC2010" )
175 # setParameter( "MATERIAL", "Pressure layer", "COMPRS/COMSTR", 35.6 )
176 # setParameter( "MATERIAL", "Pressure layer", "COMPRS/EPSC1", 0.0021 )
177 # setParameter( "MATERIAL", "Pressure layer", "COMPRS/EPSCU", 0.0035 )
178 # setParameter( "MATERIAL", "Pressure layer", "COMPRS/CONFIN/CNFCRV", "NONE" )
179 setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMCRV", "CONSTA" )
180 setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMSTR", 26.8 )
181
182 addGeometry( "Pressure layer", "SHEET", "MEMBRA", [] )
183 setParameter( "GEOMET", "Pressure layer", "THICK", 1000 )
184 setParameter( "GEOMET", "Pressure layer", "LOCAXS", True )
185 setParameter( "GEOMET", "Pressure layer", "LOCAXS/XAXIS", [ 1, 0, 0 ] )
186
187 addElementData( "Pressure layer" )
188 setElementClassType( "SHAPE", [ "Pressure Layer", "Pressure Layer_1" ], "MEMBRA" )
189 assignMaterial( "Pressure layer", "SHAPE", [ "Pressure Layer", "Pressure Layer_1" ] )
190 assignGeometry( "Pressure layer", "SHAPE", [ "Pressure Layer", "Pressure Layer_1" ] )
191 assignElementData( "Pressure layer", "SHAPE", [ "Pressure Layer", "Pressure Layer_1"
    ] )
192
```

```
193  # setParameter( "DATA", "Pressure layer", "INTEGR", "REDUCE" )
194
195  #Material Steel
196  addMaterial( "Steel", "MCSTEL", "ISOTRO", [] )
197  setParameter( "MATERIAL", "Steel", "LINEAR/ELASTI/YOUNG", 210000 )
198  setParameter( "MATERIAL", "Steel", "LINEAR/ELASTI/POISON", 0.3 )
199  setParameter( "MATERIAL", "Steel", "LINEAR/MASS/DENSIT", 7.8e-09 )
200  setParameter( "MATERIAL", "Steel", "LINEAR/MASS/DENSIT", 0 )
201  addGeometry( "Plate", "SHEET", "MEMBRA", [] )
202  setParameter( "GEOMET", "Plate", "THICK", 1000 )
203  addElementData( "Plate" )
204
205  setElementClassType( "SHAPE", [ "Support", "Loadplate" ], "MEMBRA" )
206  assignMaterial( "Steel", "SHAPE", [ "Support", "Loadplate" ] )
207  assignGeometry( "Plate", "SHAPE", [ "Support", "Loadplate" ] )
208  assignElementData( "Plate", "SHAPE", [ "Support", "Loadplate" ] )
209  # setParameter( "DATA", "Plate", "INTEGR", "REDUCE" )
210
211  #Concrete Interface
212  # addMaterial( "Interface Concrete", "INTERF", "FRICTI", [] )
213  # setParameter( "MATERIAL", "Interface Concrete", "LINEAR/IFTYP", "LIN2D" )
214  # setParameter( "MATERIAL", "Interface Concrete", "COULOM/OPNTYP", "GAP" )
215  # setParameter( "MATERIAL", "Interface Concrete", "COULOM/GAPVAL", 0.5 )
216  # setParameter( "MATERIAL", "Interface Concrete", "COULOM/COHESI", 0.2 )
217  # setParameter( "MATERIAL", "Interface Concrete", "COULOM/PHI", 22 )
218  # setParameter( "MATERIAL", "Interface Concrete", "COULOM/PSI", 22 )
219  # setParameter( "MATERIAL", "Interface Concrete", "LINEAR/ELAS2/DSNY", 10 )
220  # setParameter( "MATERIAL", "Interface Concrete", "LINEAR/ELAS2/DSSX", 10 )
221  # setParameter( "MATERIAL", "Interface Concrete", "COULOM/FRCHSR/MODE2", 0 )
222
223  addMaterial( "Interface Concrete", "INTERF", "FRICEL", [] )
224  setParameter( "MATERIAL", "Interface Concrete", "LINEAR/IFTYP", "LIN2D" )
225  setParameter( "MATERIAL", "Interface Concrete", "NEFRIC/SPCTYP", "USER" )
226  setParameter( "MATERIAL", "Interface Concrete", "NEFRIC/COHESI", 0.2 )
227  setParameter( "MATERIAL", "Interface Concrete", "NEFRIC/PHI", 22 )
228  setParameter( "MATERIAL", "Interface Concrete", "LINEAR/ELAS2/DSNY", 100 )
229  setParameter( "MATERIAL", "Interface Concrete", "LINEAR/ELAS2/DSSX", 100 )
230
231  addGeometry( "Interface Concrete", "LINE", "STLIIF", [] )
232  setParameter( "GEOMET", "Interface Concrete", "LIFMEM/THICK", 1000 )
233  addElementData( "Interface Concrete" )
234  createConnection( "Interface Concrete", "INTER", "SHAPEEDGE", "SHAPEEDGE" )
235  setParameter( "GEOMETRYCONNECTION", "Interface Concrete", "MODE", "CLOSED" )
236  setElementClassType( "GEOMETRYCONNECTION", "Interface Concrete", "STLIIF" )
237  assignMaterial( "Interface Concrete", "GEOMETRYCONNECTION", "Interface Concrete" )
238  assignGeometry( "Interface Concrete", "GEOMETRYCONNECTION", "Interface Concrete" )
239  assignElementData( "Interface Concrete", "GEOMETRYCONNECTION", "Interface Concrete" )
240  setParameter( "GEOMETRYCONNECTION", "Interface Concrete", "FLIP", False )
241  attachTo( "GEOMETRYCONNECTION", "Interface Concrete", "SOURCE", "Pressure Layer", [[
     500, 70, 0 ]] )
242  attachTo( "GEOMETRYCONNECTION", "Interface Concrete", "TARGET", "Slab", [[ 500, 70,
     0 ]] )
243  attachTo( "GEOMETRYCONNECTION", "Interface Concrete", "SOURCE", "Pressure Layer_1",
     [[ 1450, 70, 0 ]] )
244  attachTo( "GEOMETRYCONNECTION", "Interface Concrete", "TARGET", "Slab_1", [[ 1450,
     70, 0 ]] )
245
246  # setParameter( "DATA", "Interface Concrete", "INTEGR", "REDUCE" )
247
248  #Steel Interface
249  addMaterial( "Interface Steel", "INTERF", "NONLIF", [] )
250  setParameter( "MATERIAL", "Interface Steel", "LINEAR/IFTYP", "LIN2D" )
251  setParameter( "MATERIAL", "Interface Steel", "LINEAR/ELAS2/DSNY", 32000 )
252  setParameter( "MATERIAL", "Interface Steel", "LINEAR/ELAS2/DSSX", 0.00032 )
253  setParameter( "MATERIAL", "Interface Steel", "NONLIN/IFNOTE", "NOTSHR" )
254  setParameter( "MATERIAL", "Interface Steel", "NONLIN/NLEL8/NOTENS", [ 0.001, 0 ] )
255  setParameter( "MATERIAL", "Interface Steel", "NONLIN/NLEL8/NOSHTE", [ 0.001, 0 ] )
256  addGeometry( "Interface Steel", "LINE", "STLIIF", [] )
257  setParameter( "GEOMET", "Interface Steel", "LIFMEM/THICK", 1000 )
258  addElementData( "Interface Steel" )
259  createConnection( "Interface Steel", "INTER", "SHAPEEDGE", "SHAPEEDGE" )
```

```
setParameter( "GEOMETRYCONNECTION", "Interface Steel", "CONTYP", "INTER" )
setParameter( "GEOMETRYCONNECTION", "Interface Steel", "MODE", "AUTO" )
setElementClassType( "GEOMETRYCONNECTION", "Interface Steel", "STLIIF" )
assignMaterial( "Interface Steel", "GEOMETRYCONNECTION", "Interface Steel" )
assignGeometry( "Interface Steel", "GEOMETRYCONNECTION", "Interface Steel" )
assignElementData( "Interface Steel", "GEOMETRYCONNECTION", "Interface Steel" )
setParameter( "GEOMETRYCONNECTION", "Interface Steel", "FLIP", False )
attachTo( "GEOMETRYCONNECTION", "Interface Steel", "SOURCE", "Pressure Layer_1", [[ 1100, 300, 0 ]] )
attachTo( "GEOMETRYCONNECTION", "Interface Steel", "TARGET", "Loadplate", [[ 1100, 300, 0 ]] )
attachTo( "GEOMETRYCONNECTION", "Interface Steel", "SOURCE", "Slab", [[ 300, 0, 0 ]] )
attachTo( "GEOMETRYCONNECTION", "Interface Steel", "TARGET", "Support", [[ 300, 0, 0 ]] )

# setParameter( "DATA", "Interface Steel", "INTEGR", "REDUCE" )
# setParameter( "DATA", "Interface Steel", "NUMINT", [ "LOBATT" ] )

#Material Reinforcement Koppel_M12
addMaterial( "Koppel_M12", "REINFO", "REBOND", [ ] )
setParameter( "MATERIAL", "Koppel_M12", "REBARS/ELASTI/YOUNG", 200000 )
setParameter( "MATERIAL", "Koppel_M12", "REBARS/POISON/POISON", 0.3 )
setParameter( "MATERIAL", "Koppel_M12", "REBARS/MASS/DENSIT", 7.8e-09 )
setParameter( "MATERIAL", "Koppel_M12", "REBARS/PLATYP", "VMISES" )
setParameter( "MATERIAL", "Koppel_M12", "REBARS/PLASTI/TRESSH", "KAPSIG" )
setParameter( "MATERIAL", "Koppel_M12", "REBARS/PLASTI/KAPSIG", [ 0, 550, 0.04725, 594 ] )
setParameter( "MATERIAL", "Koppel_M12", "RESLIP/SHFTYP", "BONDS6" )
setParameter( "MATERIAL", "Koppel_M12", "RESLIP/DSNY", 1000 )
setParameter( "MATERIAL", "Koppel_M12", "RESLIP/DSSX", 100 )
setParameter( "MATERIAL", "Koppel_M12", "RESLIP/BONDS6/SLPVAL", [ 12.95, 5.18, 0.1, 1, 2, 10, 0.4 ] )
addGeometry( "Koppel_M12", "RELINE", "REBAR", [] )
setParameter( "GEOMET", "Koppel_M12", "REITYP", "REITRU" )
setParameter( "GEOMET", "Koppel_M12", "REITYP", "CIRBEA" )
setParameter( "GEOMET", "Koppel_M12", "CIRBEA/CIRCLE", 12)
setParameter( "GEOMET", "Koppel_M12", "TIPELM/SURFAC", 113.1 )
assignMaterial( "Koppel_M12", "SHAPE", [ "Koppel_M12a","Koppel_M12b","Koppel_M12c","Koppel_M12d","Koppel_M12e","Koppel_M12f","Koppel_M12g" ] )
assignGeometry( "Koppel_M12", "SHAPE", [ "Koppel_M12a","Koppel_M12b","Koppel_M12c","Koppel_M12d","Koppel_M12e","Koppel_M12f","Koppel_M12g" ] )
convertToReinforcement( [ "Koppel_M12a","Koppel_M12b","Koppel_M12c","Koppel_M12d","Koppel_M12e","Koppel_M12f","Koppel_M12g" ] )
addElementData( "Koppel_M12" )
assignElementData( "Koppel_M12", "SHAPE", [ "Koppel_M12a","Koppel_M12b","Koppel_M12c","Koppel_M12d","Koppel_M12e","Koppel_M12f","Koppel_M12g" ] )
setParameter( "DATA", "Koppel_M12", "INTERF", "BEAM" )
setReinforcementDiscretization( [ "Koppel_M12a","Koppel_M12b","Koppel_M12c","Koppel_M12d","Koppel_M12e","Koppel_M12f","Koppel_M12g" ], "ELEMENT" )
setContinuousInInterfaces( [ "Koppel_M12a","Koppel_M12b","Koppel_M12c","Koppel_M12d","Koppel_M12e","Koppel_M12f","Koppel_M12g" ], False )

#Material Reinforcement M8
addMaterial( "Reinfo_M8", "REINFO", "REBOND", [] )
setParameter( "MATERIAL", "Reinfo_M8", "REBARS/ELASTI/YOUNG", 200000 )
setParameter( "MATERIAL", "Reinfo_M8", "REBARS/POISON/POISON", 0.3 )
setParameter( "MATERIAL", "Reinfo_M8", "REBARS/MASS/DENSIT", 7.8e-09 )
setParameter( "MATERIAL", "Reinfo_M8", "REBARS/PLATYP", "VMISES" )
setParameter( "MATERIAL", "Reinfo_M8", "REBARS/PLASTI/TRESSH", "KAPSIG" )
setParameter( "MATERIAL", "Reinfo_M8", "REBARS/PLASTI/KAPSIG", [ 0, 550, 0.04725, 594 ] )
setParameter( "MATERIAL", "Reinfo_M8", "RESLIP/SHFTYP", "BONDS6" )
setParameter( "MATERIAL", "Reinfo_M8", "RESLIP/DSNY", 1000 )
setParameter( "MATERIAL", "Reinfo_M8", "RESLIP/DSSX", 100 )
```

```
313 setParameter( "MATERIAL", "Reinfo_M8", "RESLIP/BONDS6/SLPVAL", [ 16.94, 6.77, 0.1,
    1, 2, 10, 0.4 ] )
314 addGeometry( "Reinfo_M8", "RELINE", "REBAR", [] )
315 setParameter( "GEOMET", "Reinfo_M8", "REITYP", "REITRU" )
316 setParameter( "GEOMET", "Reinfo_M8", "REITYP", "CIRBEA" )
317 setParameter( "GEOMET", "Reinfo_M8", "CIRBEA/CIRCLE", 8)
318 setParameter( "GEOMET", "Reinfo_M8", "TIPELM/SURFAC", 50.3 )
319 assignMaterial( "Reinfo_M8", "SHAPE", [
    "Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d","Reinfo_M8e","Reinfo_M8f","Reinfo_
    M8g","Reinfo_M8h","Reinfo_M8i","Reinfo_M8j","Reinfo_M8k","Reinfo_M8l","Reinfo_M8m","Re
    info_M8n","Reinfo_M8o" ] )
320 assignGeometry( "Reinfo_M8", "SHAPE", [
    "Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d","Reinfo_M8e","Reinfo_M8f","Reinfo_
    M8g","Reinfo_M8h","Reinfo_M8i","Reinfo_M8j","Reinfo_M8k","Reinfo_M8l","Reinfo_M8m","Re
    info_M8n","Reinfo_M8o" ] )
321 convertToReinforcement( [
    "Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d","Reinfo_M8e","Reinfo_M8f","Reinfo_
    M8g","Reinfo_M8h","Reinfo_M8i","Reinfo_M8j","Reinfo_M8k","Reinfo_M8l","Reinfo_M8m","Re
    info_M8n","Reinfo_M8o" ] )
322 addElementData( "Reinfo_M8" )
323 assignElementData( "Reinfo_M8", "SHAPE", [
    "Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d","Reinfo_M8e","Reinfo_M8f","Reinfo_
    M8g","Reinfo_M8h","Reinfo_M8i","Reinfo_M8j","Reinfo_M8k","Reinfo_M8l","Reinfo_M8m","Re
    info_M8n","Reinfo_M8o" ] )
324 setParameter( "DATA", "Reinfo_M8", "INTERF", "BEAM" )
325 setReinforcementDiscretization( [
    "Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d","Reinfo_M8e","Reinfo_M8f","Reinfo_
    M8g","Reinfo_M8h","Reinfo_M8i","Reinfo_M8j","Reinfo_M8k","Reinfo_M8l","Reinfo_M8m","Re
    info_M8n","Reinfo_M8o" ], "ELEMENT" )
326 setContinuousInInterfaces( [
    "Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d","Reinfo_M8e","Reinfo_M8f","Reinfo_
    M8g","Reinfo_M8h","Reinfo_M8i","Reinfo_M8j","Reinfo_M8k","Reinfo_M8l","Reinfo_M8m","Re
    info_M8n","Reinfo_M8o" ], False )
327
328 #Material Reinforcement M4.5
329 addMaterial( "Reinfo_M4.5", "REINFO", "REBOND", [ "ANCHOR" ] )
330 setParameter( "MATERIAL", "Reinfo_M4.5", "REBARS/ELASTI/YOUNG", 200000 )
331 setParameter( "MATERIAL", "Reinfo_M4.5", "REBARS/POISON/POISON", 0.3 )
332 setParameter( "MATERIAL", "Reinfo_M4.5", "REBARS/PLATYP", "VMISES" )
333 setParameter( "MATERIAL", "Reinfo_M4.5", "REBARS/PLASTI/TRESSH", "KAPSIG" )
334 setParameter( "MATERIAL", "Reinfo_M4.5", "REBARS/PLASTI/KAPSIG", [ 0, 550, 0.04725,
    594 ] )
335 setParameter( "MATERIAL", "Reinfo_M4.5", "RESLIP/SHFTYP", "BONDS6" )
336 setParameter( "MATERIAL", "Reinfo_M4.5", "RESLIP/DSNY", 1000 )
337 setParameter( "MATERIAL", "Reinfo_M4.5", "RESLIP/DSSX", 100 )
338
339 setParameter( "MATERIAL", "Reinfo_M4.5", "RESLIP/BONDS6/SLPVAL", [ 0.52, 0.52, 0.01,
    0.02, 0.03, 0.04, 0.5 ] )
340
341 setParameter( "MATERIAL", "Reinfo_M4.5", "REANCH/DTIPNX", 10000 )
342 setParameter( "MATERIAL", "Reinfo_M4.5", "REANCH/DTIPSY", 10000 )
343 setParameter( "MATERIAL", "Reinfo_M4.5", "REANCH/TIPLOC", "BOTH" )
344
345 addGeometry( "Reinfo_M4.5", "RELINE", "REBAR", [] )
346 setParameter( "GEOMET", "Reinfo_M4.5", "REITYP", "REITRU" )
347 setParameter( "GEOMET", "Reinfo_M4.5", "REITYP", "CIRBEA" )
348 setParameter( "GEOMET", "Reinfo_M4.5", "CIRBEA/CIRCLE", 4.5 )
349 setParameter( "GEOMET", "Reinfo_M4.5", "TIPELM/SURFAC", 15.9 )
350 assignMaterial( "Reinfo_M4.5", "SHAPE", [
    "Bar_M41al","Bar_M41ar","Bar_M41bl","Bar_M41br","Bar_M41cl","Bar_M41cr","Bar_M41dl","B
    ar_M41dr","Bar_M41el","Bar_M41er","Bar_M41fl","Bar_M41fr","Bar_M41gl","Bar_M41gr","Bar
    _M41hl","Bar_M41hr","Bar_M41il","Bar_M41ir","Bar_M41jl","Bar_M41jr","Bar_M42al","Bar_M
    42ar","Bar_M42bl","Bar_M42br","Bar_M42cl","Bar_M42cr","Bar_M42dl","Bar_M42dr","Bar_M42
    el","Bar_M42er","Bar_M42fl","Bar_M42fr","Bar_M42gl","Bar_M42gr","Bar_M42hl","Bar_M42hr
    ","Bar_M42il","Bar_M42ir","Bar_M42jl","Bar_M42jr","Bar_M43al","Bar_M43ar","Bar_M43bl",
    "Bar_M43br","Bar_M43cl","Bar_M43cr","Bar_M43dl","Bar_M43dr","Bar_M43el","Bar_M43er","B
    ar_M43fl","Bar_M43fr","Bar_M43gl","Bar_M43gr","Bar_M43hl","Bar_M43hr","Bar_M43il","Bar
    _M43ir","Bar_M43jl","Bar_M43jr" ] )
351 assignGeometry( "Reinfo_M4.5", "SHAPE", [
    "Bar_M41al","Bar_M41ar","Bar_M41bl","Bar_M41br","Bar_M41cl","Bar_M41cr","Bar_M41dl","B
    ar_M41dr","Bar_M41el","Bar_M41er","Bar_M41fl","Bar_M41fr","Bar_M41gl","Bar_M41gr","Bar
```

```
_M41hl","Bar_M41hr","Bar_M41il","Bar_M41ir","Bar_M41jl","Bar_M41jr","Bar_M42al","Bar_M
42ar","Bar_M42bl","Bar_M42br","Bar_M42cl","Bar_M42cr","Bar_M42dl","Bar_M42dr","Bar_M42
el","Bar_M42er","Bar_M42fl","Bar_M42fr","Bar_M42gl","Bar_M42gr","Bar_M42hl","Bar_M42hr
","Bar_M42il","Bar_M42ir","Bar_M42jl","Bar_M42jr","Bar_M43al","Bar_M43ar","Bar_M43bl",
"Bar_M43br","Bar_M43cl","Bar_M43cr","Bar_M43dl","Bar_M43dr","Bar_M43el","Bar_M43er","B
ar_M43fl","Bar_M43fr","Bar_M43gl","Bar_M43gr","Bar_M43hl","Bar_M43hr","Bar_M43il","Bar
_M43ir","Bar_M43jl","Bar_M43jr" ] )
convertToReinforcement( [
"Bar_M41al","Bar_M41ar","Bar_M41bl","Bar_M41br","Bar_M41cl","Bar_M41cr","Bar_M41dl","B
ar_M41dr","Bar_M41el","Bar_M41er","Bar_M41fl","Bar_M41fr","Bar_M41gl","Bar_M41gr","Bar
_M41hl","Bar_M41hr","Bar_M41il","Bar_M41ir","Bar_M41jl","Bar_M41jr","Bar_M42al","Bar_M
42ar","Bar_M42bl","Bar_M42br","Bar_M42cl","Bar_M42cr","Bar_M42dl","Bar_M42dr","Bar_M42
el","Bar_M42er","Bar_M42fl","Bar_M42fr","Bar_M42gl","Bar_M42gr","Bar_M42hl","Bar_M42hr
","Bar_M42il","Bar_M42ir","Bar_M42jl","Bar_M42jr","Bar_M43al","Bar_M43ar","Bar_M43bl",
"Bar_M43br","Bar_M43cl","Bar_M43cr","Bar_M43dl","Bar_M43dr","Bar_M43el","Bar_M43er","B
ar_M43fl","Bar_M43fr","Bar_M43gl","Bar_M43gr","Bar_M43hl","Bar_M43hr","Bar_M43il","Bar
_M43ir","Bar_M43jl","Bar_M43jr" ] )
addElementData( "Bar" )
assignElementData( "Bar", "SHAPE", [
"Bar_M41al","Bar_M41ar","Bar_M41bl","Bar_M41br","Bar_M41cl","Bar_M41cr","Bar_M41dl","B
ar_M41dr","Bar_M41el","Bar_M41er","Bar_M41fl","Bar_M41fr","Bar_M41gl","Bar_M41gr","Bar
_M41hl","Bar_M41hr","Bar_M41il","Bar_M41ir","Bar_M41jl","Bar_M41jr","Bar_M42al","Bar_M
42ar","Bar_M42bl","Bar_M42br","Bar_M42cl","Bar_M42cr","Bar_M42dl","Bar_M42dr","Bar_M42
el","Bar_M42er","Bar_M42fl","Bar_M42fr","Bar_M42gl","Bar_M42gr","Bar_M42hl","Bar_M42hr
","Bar_M42il","Bar_M42ir","Bar_M42jl","Bar_M42jr","Bar_M43al","Bar_M43ar","Bar_M43bl",
"Bar_M43br","Bar_M43cl","Bar_M43cr","Bar_M43dl","Bar_M43dr","Bar_M43el","Bar_M43er","B
ar_M43fl","Bar_M43fr","Bar_M43gl","Bar_M43gr","Bar_M43hl","Bar_M43hr","Bar_M43il","Bar
_M43ir","Bar_M43jl","Bar_M43jr" ] )
setParameter( "DATA", "Bar", "./INTERF", [] )
setParameter( "DATA", "Bar", "INTERF", "BEAM" )
setReinforcementDiscretization( [
"Bar_M41al","Bar_M41ar","Bar_M41bl","Bar_M41br","Bar_M41cl","Bar_M41cr","Bar_M41dl","B
ar_M41dr","Bar_M41el","Bar_M41er","Bar_M41fl","Bar_M41fr","Bar_M41gl","Bar_M41gr","Bar
_M41hl","Bar_M41hr","Bar_M41il","Bar_M41ir","Bar_M41jl","Bar_M41jr","Bar_M42al","Bar_M
42ar","Bar_M42bl","Bar_M42br","Bar_M42cl","Bar_M42cr","Bar_M42dl","Bar_M42dr","Bar_M42
el","Bar_M42er","Bar_M42fl","Bar_M42fr","Bar_M42gl","Bar_M42gr","Bar_M42hl","Bar_M42hr
","Bar_M42il","Bar_M42ir","Bar_M42jl","Bar_M42jr","Bar_M43al","Bar_M43ar","Bar_M43bl",
"Bar_M43br","Bar_M43cl","Bar_M43cr","Bar_M43dl","Bar_M43dr","Bar_M43el","Bar_M43er","B
ar_M43fl","Bar_M43fr","Bar_M43gl","Bar_M43gr","Bar_M43hl","Bar_M43hr","Bar_M43il","Bar
_M43ir","Bar_M43jl","Bar_M43jr" ], "SECTION" )

#Composed Line
addGeometry( "Composed Line", "LINE", "COMLIN", [] )
setParameter( "GEOMET", "Composed Line", "DISTAN/THICK", 300 )
setElementClassType( "SHAPE", [ "Composed Line" ], "COMLIN" )
assignGeometry( "Composed Line", "SHAPE", [ "Composed Line" ] )
resetElementData( "SHAPE", [ "Composed Line" ] )

#Assign Supports
addSet( "GEOMETRYSUPPORTSET", "Support" )

createPointSupport( "Support", "Support" )
setParameter( "GEOMETRYSUPPORT", "Support", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Support", "TRANSL", [ 0, 1, 0 ] )
setParameter( "GEOMETRYSUPPORT", "Support", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Support", "Support", [[ 300, -30, 0 ]] )

addSet( "GEOMETRYSUPPORTSET", "Sym_X" )

createLineSupport( "Sym_X", "Sym_X" )
setParameter( "GEOMETRYSUPPORT", "Sym_X", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Sym_X", "TRANSL", [ 1, 0, 0 ] )
setParameter( "GEOMETRYSUPPORT", "Sym_X", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Sym_X", "Pressure Layer_1", [[ 1900, 185, 0 ]] )

createPointSupport( "Symp_X", "Symp_X" )
setParameter( "GEOMETRYSUPPORT", "Symp_X", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Symp_X", "TRANSL", [ 1, 0, 0 ] )
setParameter( "GEOMETRYSUPPORpT", "Symp_X", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Symp_X", "Koppel_M12a", [[ 1900, 76, 0 ]] )
attach( "GEOMETRYSUPPORT", "Symp_X", "Koppel_M12b", [[ 1900, 76, 0 ]] )
```

```
attach( "GEOMETRYSUPPORT", "Symp_X", "Koppel_M12c", [[ 1900, 76, 0 ]] )
attach( "GEOMETRYSUPPORT", "Symp_X", "Koppel_M12d", [[ 1900, 76, 0 ]] )
attach( "GEOMETRYSUPPORT", "Symp_X", "Koppel_M12e", [[ 1900, 76, 0 ]] )
attach( "GEOMETRYSUPPORT", "Symp_X", "Koppel_M12f", [[ 1900, 76, 0 ]] )
attach( "GEOMETRYSUPPORT", "Symp_X", "Koppel_M12g", [[ 1900, 76, 0 ]] )

addSet( "GEOMETRYSUPPORTSET", "Load point" )

createPointSupport( "Load point", "Load point" )
setParameter( "GEOMETRYSUPPORT", "Load point", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Load point", "TRANSL", [ 0, 1, 0 ] )
setParameter( "GEOMETRYSUPPORT", "Load point", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Load point", "Loadplate", [[ 1100, 330, 0 ]] )

#Assign Loads
addSet( "GEOMETRYLOADSET", "Load" )

createPointLoad( "Load", "Load" )
setParameter( "GEOMETRYLOAD", "Load", "LODTYP", "DEFORM" )
setParameter( "GEOMETRYLOAD", "Load", "DEFORM/TR/VALUE", -0.1 )
setParameter( "GEOMETRYLOAD", "Load", "DEFORM/TR/DIRECT", 2 )
attach( "GEOMETRYLOAD", "Load", "Loadplate", [[ 1100, 330, 0 ]] )

#Set mesh properties
setElementSize( [  "Loadplate", "Pressure Layer_1", "Slab_1",
"Koppel_M12a","Koppel_M12b","Koppel_M12c","Koppel_M12d","Koppel_M12e","Koppel_M12f","K
oppel_M12g","Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d","Reinfo_M8e","Reinfo_M
8f","Reinfo_M8g","Reinfo_M8h","Reinfo_M8i","Reinfo_M8j","Reinfo_M8k","Reinfo_M8l","Rei
nfo_M8m","Reinfo_M8n","Reinfo_M8o","Bar_M41al","Bar_M41ar","Bar_M41bl","Bar_M41br","Ba
r_M41cl","Bar_M41cr","Bar_M41dl","Bar_M41dr","Bar_M41el","Bar_M41er","Bar_M41fl","Bar_
M41fr","Bar_M41gl","Bar_M41gr","Bar_M41hl","Bar_M41hr","Bar_M41il","Bar_M41ir","Bar_M4
1jl","Bar_M41jr","Bar_M42al","Bar_M42ar","Bar_M42bl","Bar_M42br","Bar_M42cl","Bar_M42c
r","Bar_M42dl","Bar_M42dr","Bar_M42el","Bar_M42er","Bar_M42fl","Bar_M42fr","Bar_M42gl"
,"Bar_M42gr","Bar_M42hl","Bar_M42hr","Bar_M42il","Bar_M42ir","Bar_M42jl","Bar_M42jr","
Bar_M43al","Bar_M43ar","Bar_M43bl","Bar_M43br","Bar_M43cl","Bar_M43cr","Bar_M43dl","Ba
r_M43dr","Bar_M43el","Bar_M43er","Bar_M43fl","Bar_M43fr","Bar_M43gl","Bar_M43gr","Bar_
M43hl","Bar_M43hr","Bar_M43il","Bar_M43ir","Bar_M43jl","Bar_M43jr" ], 10, -1, True )
setMesherType( [  "Loadplate", "Pressure Layer_1", "Slab_1",
"Koppel_M12a","Koppel_M12b","Koppel_M12c","Koppel_M12d","Koppel_M12e","Koppel_M12f","K
oppel_M12g","Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d","Reinfo_M8e","Reinfo_M
8f","Reinfo_M8g","Reinfo_M8h","Reinfo_M8i","Reinfo_M8j","Reinfo_M8k","Reinfo_M8l","Rei
nfo_M8m","Reinfo_M8n","Reinfo_M8o","Bar_M41al","Bar_M41ar","Bar_M41bl","Bar_M41br","Ba
r_M41cl","Bar_M41cr","Bar_M41dl","Bar_M41dr","Bar_M41el","Bar_M41er","Bar_M41fl","Bar_
M41fr","Bar_M41gl","Bar_M41gr","Bar_M41hl","Bar_M41hr","Bar_M41il","Bar_M41ir","Bar_M4
1jl","Bar_M41jr","Bar_M42al","Bar_M42ar","Bar_M42bl","Bar_M42br","Bar_M42cl","Bar_M42c
r","Bar_M42dl","Bar_M42dr","Bar_M42el","Bar_M42er","Bar_M42fl","Bar_M42fr","Bar_M42gl"
,"Bar_M42gr","Bar_M42hl","Bar_M42hr","Bar_M42il","Bar_M42ir","Bar_M42jl","Bar_M42jr","
Bar_M43al","Bar_M43ar","Bar_M43bl","Bar_M43br","Bar_M43cl","Bar_M43cr","Bar_M43dl","Ba
r_M43dr","Bar_M43el","Bar_M43er","Bar_M43fl","Bar_M43fr","Bar_M43gl","Bar_M43gr","Bar_
M43hl","Bar_M43hr","Bar_M43il","Bar_M43ir","Bar_M43jl","Bar_M43jr" ], "HEXQUAD" )
clearMidSideNodeLocation( [  "Loadplate", "Pressure Layer_1", "Slab_1",
"Koppel_M12a","Koppel_M12b","Koppel_M12c","Koppel_M12d","Koppel_M12e","Koppel_M12f","K
oppel_M12g","Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d","Reinfo_M8e","Reinfo_M
8f","Reinfo_M8g","Reinfo_M8h","Reinfo_M8i","Reinfo_M8j","Reinfo_M8k","Reinfo_M8l","Rei
nfo_M8m","Reinfo_M8n","Reinfo_M8o","Bar_M41al","Bar_M41ar","Bar_M41bl","Bar_M41br","Ba
r_M41cl","Bar_M41cr","Bar_M41dl","Bar_M41dr","Bar_M41el","Bar_M41er","Bar_M41fl","Bar_
M41fr","Bar_M41gl","Bar_M41gr","Bar_M41hl","Bar_M41hr","Bar_M41il","Bar_M41ir","Bar_M4
1jl","Bar_M41jr","Bar_M42al","Bar_M42ar","Bar_M42bl","Bar_M42br","Bar_M42cl","Bar_M42c
r","Bar_M42dl","Bar_M42dr","Bar_M42el","Bar_M42er","Bar_M42fl","Bar_M42fr","Bar_M42gl"
,"Bar_M42gr","Bar_M42hl","Bar_M42hr","Bar_M42il","Bar_M42ir","Bar_M42jl","Bar_M42jr","
Bar_M43al","Bar_M43ar","Bar_M43bl","Bar_M43br","Bar_M43cl","Bar_M43cr","Bar_M43dl","Ba
r_M43dr","Bar_M43el","Bar_M43er","Bar_M43fl","Bar_M43fr","Bar_M43gl","Bar_M43gr","Bar_
M43hl","Bar_M43hr","Bar_M43il","Bar_M43ir","Bar_M43jl","Bar_M43jr" ] )
setElementSize( [ "Support","Pressure Layer", "Slab" ], 50, -1, True )
setMesherType( [ "Support","Pressure Layer", "Slab" ], "HEXQUAD" )
setEdgeGrading( "Pressure Layer", [[ 500, 70, 0 ]], [], 50, 50, 10, 0.25, False,
True )
setEdgeGrading( "Slab", [], [[ 500, 70, 0 ]], 50, 50, 10, 0.25, False, True )
generateMesh( [] )
hideView( "GEOM" )
showView( "MESH" )
```

```
423
424 #Analysis
425 addAnalysis( "Analysis1" )
426 addAnalysisCommand( "Analysis1", "NONLIN", "Structural nonlinear" )
427 renameAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "EXECUT(1)",
    "Load" )
428 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/LOADNR" )
429 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/LOADNR", 1 )
430 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/STEPTY", "AUTOMA" )
431 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/SIZES", 250 )
432 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/MAXSIZ", 0.01 )
433 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/MINSIZ", 0.00001 )
434 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/CUTBCK", 0.25 )
435 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/MAXSTP", 100 )
436 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/ARCLEN", True )
437 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/ARCLEN/REGULA/SET" )
438 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/ARCLEN/REGULA/SET(1)/NODES(1)/RNGNRS", "1-9361" )
439 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/ARCLEN/REGULA/SET(1)/DIRECT", 2 )
440 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/SIZES", "1(100)" )
441 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN", True )
442 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN/REGULA/SET" )
443 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN/REGULA/SET(1)/NODES(1)/RNGNRS", "844 865 866 871
    945-966 1055 1056-2925(89) 3108-3132 3217 3218 3396 3397 3575 3576 3754 3755 3933
    3934 4112 4113 4291 4292 4470 4471 4649 4650 4828 4829 5007 5008 5186 5187 5365 5366
    5544 5545 5723 5724 5902 5903 6081 6082 6260 6261 6439 6440 6618 6619 6797 6798 6976
    6977 7155 7244 7246 7335-7341 7430 7431-7965(89) 8144-8153 8331 8332 8510 8511 8689
    8690 8868 8869 9047 9048 9226 9453-9455 9593 9594 9732 9733 9871 9872 10010 10011
    10149 10150 10288 10289" )
444 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN/REGULA/SET(1)/DIRECT", 2 )
445 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/MAXITE", 250 )
446 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/LINESE", False )
447 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/ENERGY", True )
448 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/DISPLA", False )
449 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/FORCE", True )
450 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/ENERGY/TOLCON", 0.001 )
451 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/ENERGY/NOCONV", "CONTIN" )
452 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/FORCE/TOLCON", 0.01 )
453 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/FORCE/NOCONV", "CONTIN" )
454 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/METHOD/METNAM", "SECANT" )
455
456 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "OUTPUT(1)/SELTYP",
    "USER" )
457 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "OUTPUT(1)/USER" )
458 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
```

```
    "OUTPUT(1)/USER/DISPLA(1)/TOTAL" )
459 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/FORCE(1)/REACTI" )
460 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(1)/TOTAL/GREEN" )
461 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(2)/TOTAL/GREEN/PRINCI" )
462 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(3)/CRKWDT" )
463 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(4)/CRACK/GREEN" )
464 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(5)/CRKWDT/GREEN/PRINCI" )
465 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(1)/TOTAL/CAUCHY" )
466 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(2)/TOTAL/CAUCHY/PRINCI" )
467 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(3)/TOTAL/FORCE/LOCAL" )
468 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(4)/TOTAL/MOMENT/LOCAL" )
469 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(5)/EFFECT/TRACTI/GLOBAL" )
470 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(6)/EFFECT/TRACTI/LOCAL" )
471 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(7)/EFFECT/TRACTI/FRICTI" )
472
473 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(1)/TOTAL/GREEN/GLOBAL/LOCATI", "INTPNT" )
474 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(1)/TOTAL/CAUCHY/GLOBAL/LOCATI", "INTPNT" )
475
476 #Run Analysis
477 # runSolver( [] )
478 # showView( "RESULT" )
479 fitAll( )
480
481
482
```

B5 - R1 V8

```
newProject( "D:/Afstuderen/Python/Proefstukken/DIANA Output/R1 V8 Reduced Elements FRICEL", 10 )

###Project Settings###
setModelAnalysisAspects( [ "STRUCT" ] )
setModelDimension( "2D" )
setDefaultMeshOrder( "QUADRATIC" )
setDefaultMesherType( "HEXQUAD" )
setDefaultMidSideNodeLocation( "ONSHAP" )
setUnit( "LENGTH", "MM" )
setUnit( "FORCE", "N" )
setUnit( "ANGLE", "DEGREE" )


#Create Shapes
createSheet( "Pressure Layer", [[ 0, 70, 0 ],[ 1900, 70, 0 ],[ 1900, 300, 0 ],[ 1150, 300, 0 ],[1050, 300, 0 ],[ 0, 300, 0 ]] )
createSheet( "Slab", [[ 0, 0, 0 ],[ 250, 0, 0 ],[ 350, 0, 0 ],[ 1900, 0, 0 ],[ 1900, 70, 0],[ 0, 70, 0 ]] )

createSheet( "Support", [[ 250, 0, 0 ],[ 350, 0, 0 ],[ 350, -30, 0 ],[ 300, -30, 0 ],[ 250, -30, 0 ]] )
createSheet( "Loadplate", [[1050, 300, 0 ],[ 1150, 300, 0 ],[ 1150, 330, 0 ],[ 1100, 330, 0 ],[ 1050, 330, 0 ]] )

#Create Reinforcement
createLine( "Koppel_M12a", [ 1050, 76, 0 ], [ 1900, 76, 0 ] )
createLine( "Koppel_M12b", [ 1050, 76, 0 ], [ 1900, 76, 0 ] )
createLine( "Koppel_M12c", [ 1050, 76, 0 ], [ 1900, 76, 0 ] )
createLine( "Koppel_M12d", [ 1050, 76, 0 ], [ 1900, 76, 0 ] )
createLine( "Koppel_M12e", [ 1050, 76, 0 ], [ 1900, 76, 0 ] )
createLine( "Koppel_M12f", [ 1050, 76, 0 ], [ 1900, 76, 0 ] )
createLine( "Koppel_M12g", [ 1050, 76, 0 ], [ 1900, 76, 0 ] )

createLine( "Reinfo_M8a", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8b", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8c", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8d", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8e", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8f", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8g", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8h", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8i", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8j", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8k", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8l", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8m", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8n", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8o", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8p", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8q", [ 0, 35, 0 ], [ 1865, 35, 0 ] )
createLine( "Reinfo_M8r", [ 0, 35, 0 ], [ 1865, 35, 0 ] )

createLine( "Bar_M41al", [ 135, 45, 0],[ 175, 245, 0] )
createLine( "Bar_M41ar", [ 215, 45, 0],[ 175, 245, 0] )
createLine( "Bar_M41bl", [ 135, 45, 0],[ 175, 245, 0] )
createLine( "Bar_M41br", [ 215, 45, 0],[ 175, 245, 0] )
createLine( "Bar_M41cl", [ 135, 45, 0],[ 175, 245, 0] )
createLine( "Bar_M41cr", [ 215, 45, 0],[ 175, 245, 0] )
createLine( "Bar_M41dl", [ 135, 45, 0],[ 175, 245, 0] )
createLine( "Bar_M41dr", [ 215, 45, 0],[ 175, 245, 0] )
createLine( "Bar_M41el", [ 135, 45, 0],[ 175, 245, 0] )
createLine( "Bar_M41er", [ 215, 45, 0],[ 175, 245, 0] )
createLine( "Bar_M41fl", [ 135, 45, 0],[ 175, 245, 0] )
createLine( "Bar_M41fr", [ 215, 45, 0],[ 175, 245, 0] )
createLine( "Bar_M41gl", [ 135, 45, 0],[ 175, 245, 0] )
createLine( "Bar_M41gr", [ 215, 45, 0],[ 175, 245, 0] )
createLine( "Bar_M41hl", [ 135, 45, 0],[ 175, 245, 0] )
createLine( "Bar_M41hr", [ 215, 45, 0],[ 175, 245, 0] )
createLine( "Bar_M41il", [ 135, 45, 0],[ 175, 245, 0] )
createLine( "Bar_M41ir", [ 215, 45, 0],[ 175, 245, 0] )
```

```
createLine( "Bar_M41jl", [ 135, 45, 0],[ 175, 245, 0] )
createLine( "Bar_M41jr", [ 215, 45, 0],[ 175, 245, 0] )

createLine( "Bar_M42al", [ 760, 45, 0],[ 800, 245, 0] )
createLine( "Bar_M42ar", [ 840, 45, 0],[ 800, 245, 0] )
createLine( "Bar_M42bl", [ 760, 45, 0],[ 800, 245, 0] )
createLine( "Bar_M42br", [ 840, 45, 0],[ 800, 245, 0] )
createLine( "Bar_M42cl", [ 760, 45, 0],[ 800, 245, 0] )
createLine( "Bar_M42cr", [ 840, 45, 0],[ 800, 245, 0] )
createLine( "Bar_M42dl", [ 760, 45, 0],[ 800, 245, 0] )
createLine( "Bar_M42dr", [ 840, 45, 0],[ 800, 245, 0] )
createLine( "Bar_M42el", [ 760, 45, 0],[ 800, 245, 0] )
createLine( "Bar_M42er", [ 840, 45, 0],[ 800, 245, 0] )
createLine( "Bar_M42fl", [ 760, 45, 0],[ 800, 245, 0] )
createLine( "Bar_M42fr", [ 840, 45, 0],[ 800, 245, 0] )
createLine( "Bar_M42gl", [ 760, 45, 0],[ 800, 245, 0] )
createLine( "Bar_M42gr", [ 840, 45, 0],[ 800, 245, 0] )
createLine( "Bar_M42hl", [ 760, 45, 0],[ 800, 245, 0] )
createLine( "Bar_M42hr", [ 840, 45, 0],[ 800, 245, 0] )
createLine( "Bar_M42il", [ 760, 45, 0],[ 800, 245, 0] )
createLine( "Bar_M42ir", [ 840, 45, 0],[ 800, 245, 0] )
createLine( "Bar_M42jl", [ 760, 45, 0],[ 800, 245, 0] )
createLine( "Bar_M42jr", [ 840, 45, 0],[ 800, 245, 0] )

createLine( "Bar_M43al", [ 1460, 45, 0],[ 1500, 245, 0] )
createLine( "Bar_M43ar", [ 1540, 45, 0],[ 1500, 245, 0] )
createLine( "Bar_M43bl", [ 1460, 45, 0],[ 1500, 245, 0] )
createLine( "Bar_M43br", [ 1540, 45, 0],[ 1500, 245, 0] )
createLine( "Bar_M43cl", [ 1460, 45, 0],[ 1500, 245, 0] )
createLine( "Bar_M43cr", [ 1540, 45, 0],[ 1500, 245, 0] )
createLine( "Bar_M43dl", [ 1460, 45, 0],[ 1500, 245, 0] )
createLine( "Bar_M43dr", [ 1540, 45, 0],[ 1500, 245, 0] )
createLine( "Bar_M43el", [ 1460, 45, 0],[ 1500, 245, 0] )
createLine( "Bar_M43er", [ 1540, 45, 0],[ 1500, 245, 0] )
createLine( "Bar_M43fl", [ 1460, 45, 0],[ 1500, 245, 0] )
createLine( "Bar_M43fr", [ 1540, 45, 0],[ 1500, 245, 0] )
createLine( "Bar_M43gl", [ 1460, 45, 0],[ 1500, 245, 0] )
createLine( "Bar_M43gr", [ 1540, 45, 0],[ 1500, 245, 0] )
createLine( "Bar_M43hl", [ 1460, 45, 0],[ 1500, 245, 0] )
createLine( "Bar_M43hr", [ 1540, 45, 0],[ 1500, 245, 0] )
createLine( "Bar_M43il", [ 1460, 45, 0],[ 1500, 245, 0] )
createLine( "Bar_M43ir", [ 1540, 45, 0],[ 1500, 245, 0] )
createLine( "Bar_M43jl", [ 1460, 45, 0],[ 1500, 245, 0] )
createLine( "Bar_M43jr", [ 1540, 45, 0],[ 1500, 245, 0] )

createLine( "Composed Line", [ 0, 150, 0 ], [ 1900, 150, 0 ] )

createSheet( "Mesh Sheet", [[ 1000, 0, 0 ],[ 1000, 300, 0 ],[ 1000, 300, 1000 ],[
1000, 0, 1000 ]] )

#Splitting Mesh Parts
subtract( "Pressure Layer", [ "Mesh Sheet" ], True, True )
subtract( "Slab", [ "Mesh Sheet" ], False, True )

#Regroup Material Parts
addSet( "SHAPESET", "Coupling Reinforcement" )
moveToShapeSet( [
"Koppel_M12a","Koppel_M12b","Koppel_M12c","Koppel_M12d","Koppel_M12e","Koppel_M12f","K
oppel_M12g" ], "Coupling Reinforcement" )
addSet( "SHAPESET", "Longitudinal Reinforcement" )
moveToShapeSet( [
"Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d","Reinfo_M8e","Reinfo_M8f","Reinfo_
M8g","Reinfo_M8h","Reinfo_M8i","Reinfo_M8j","Reinfo_M8k","Reinfo_M8l","Reinfo_M8m","Re
info_M8n","Reinfo_M8o","Reinfo_M8p","Reinfo_M8q","Reinfo_M8r" ], "Longitudinal
Reinforcement" )
addSet( "SHAPESET", "Lattice girders" )
moveToShapeSet( [
"Bar_M41al","Bar_M41ar","Bar_M41bl","Bar_M41br","Bar_M41cl","Bar_M41cr","Bar_M41dl","B
ar_M41dr","Bar_M41el","Bar_M41er","Bar_M41fl","Bar_M41fr","Bar_M41gl","Bar_M41gr","Bar
_M41hl","Bar_M41hr","Bar_M41il","Bar_M41ir","Bar_M41jl","Bar_M41jr","Bar_M42al","Bar_M
42ar","Bar_M42bl","Bar_M42br","Bar_M42cl","Bar_M42cr","Bar_M42dl","Bar_M42dr","Bar_M42
```

```
el","Bar_M42er","Bar_M42fl","Bar_M42fr","Bar_M42gl","Bar_M42gr","Bar_M42hl","Bar_M42hr
","Bar_M42il","Bar_M42ir","Bar_M42jl","Bar_M42jr","Bar_M43al","Bar_M43ar","Bar_M43bl",
"Bar_M43br","Bar_M43cl","Bar_M43cr","Bar_M43dl","Bar_M43dr","Bar_M43el","Bar_M43er","B
ar_M43fl","Bar_M43fr","Bar_M43gl","Bar_M43gr","Bar_M43hl","Bar_M43hr","Bar_M43il","Bar
_M43ir","Bar_M43jl","Bar_M43jr" ], "Lattice girders" )
addSet( "SHAPESET", "Steel plates" )
moveToShapeSet( [ "Loadplate", "Support" ], "Steel plates" )
rename( "SHAPESET", "Shapes", "Concrete" )
addSet( "SHAPESET", "Composed Line" )
moveToShapeSet( [ "Composed Line" ], "Composed Line" )

#Material Precast concrete
addMaterial( "Precast concrete", "CONCR", "TSCR", [] )
setParameter( "MATERIAL", "Precast concrete", "LINEAR/ELASTI/YOUNG", 35594 )
setParameter( "MATERIAL", "Precast concrete", "LINEAR/ELASTI/POISON", 0.2 )
setParameter( "MATERIAL", "Precast concrete", "LINEAR/MASS/DENSIT", 2.4e-09 )
setParameter( "MATERIAL", "Precast concrete", "MODTYP/TOTCRK", "ROTATE" )
setParameter( "MATERIAL", "Precast concrete", "TENSIL/TENCRV", "HORDYK" )
setParameter( "MATERIAL", "Precast concrete", "TENSIL/TENSTR", 3.61)
setParameter( "MATERIAL", "Precast concrete", "TENSIL/GF1", 0.1475 )
setParameter( "MATERIAL", "Precast concrete", "TENSIL/CBSPEC", "ROTS" )
setParameter( "MATERIAL", "Precast concrete", "TENSIL/POISRE/POIRED", "DAMAGE" )
# setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMCRV", "MC2010" )
# setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMSTR", 49.7 )
# setParameter( "MATERIAL", "Precast concrete", "COMPRS/EPSC1", 0.0023 )
# setParameter( "MATERIAL", "Precast concrete", "COMPRS/EPSCU", 0.0035 )
# setParameter( "MATERIAL", "Precast concrete", "COMPRS/CONFIN/CNFCRV", "NONE" )
setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMCRV", "CONSTA" )
setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMSTR", 49.7 )

addGeometry( "Precast concrete", "SHEET", "MEMBRA", [] )
setParameter( "GEOMET", "Precast concrete", "THICK", 1000 )
setParameter( "GEOMET", "Precast concrete", "LOCAXS", True )
setParameter( "GEOMET", "Precast concrete", "LOCAXS/XAXIS", [ 1, 0, 0 ] )

addElementData( "Precast concrete" )
setElementClassType( "SHAPE", [ "Slab", "Slab_1" ], "MEMBRA" )
assignMaterial( "Precast concrete", "SHAPE", [ "Slab", "Slab_1" ] )
assignGeometry( "Precast concrete", "SHAPE", [ "Slab", "Slab_1" ] )
assignElementData( "Precast concrete", "SHAPE", [ "Slab", "Slab_1" ] )

#Material Pressure layer
addMaterial( "Pressure layer", "CONCR", "TSCR", [] )
setParameter( "MATERIAL", "Pressure layer", "LINEAR/ELASTI/YOUNG", 32742 )
setParameter( "MATERIAL", "Pressure layer", "LINEAR/ELASTI/POISON", 0.2 )
setParameter( "MATERIAL", "Pressure layer", "LINEAR/MASS/DENSIT", 2.4e-09 )
setParameter( "MATERIAL", "Pressure layer", "MODTYP/TOTCRK", "ROTATE" )
setParameter( "MATERIAL", "Pressure layer", "TENSIL/TENCRV", "HORDYK" )
setParameter( "MATERIAL", "Pressure layer", "TENSIL/TENSTR", 2.87 )
setParameter( "MATERIAL", "Pressure layer", "TENSIL/GF1", 0.1403 )
setParameter( "MATERIAL", "Pressure layer", "TENSIL/CBSPEC", "ROTS" )
setParameter( "MATERIAL", "Pressure layer", "TENSIL/POISRE/POIRED", "DAMAGE" )
# setParameter( "MATERIAL", "Pressure layer", "COMPRS/COMCRV", "MC2010" )
# setParameter( "MATERIAL", "Pressure layer", "COMPRS/COMSTR", 37.6 )
# setParameter( "MATERIAL", "Pressure layer", "COMPRS/EPSC1", 0.0022 )
# setParameter( "MATERIAL", "Pressure layer", "COMPRS/EPSCU", 0.0035 )
# setParameter( "MATERIAL", "Pressure layer", "COMPRS/CONFIN/CNFCRV", "NONE" )
setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMCRV", "CONSTA" )
setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMSTR", 37.6 )

addGeometry( "Pressure layer", "SHEET", "MEMBRA", [] )
setParameter( "GEOMET", "Pressure layer", "THICK", 1000 )
setParameter( "GEOMET", "Pressure layer", "LOCAXS", True )
setParameter( "GEOMET", "Pressure layer", "LOCAXS/XAXIS", [ 1, 0, 0 ] )

addElementData( "Pressure layer" )
setElementClassType( "SHAPE", [ "Pressure Layer", "Pressure Layer_1" ], "MEMBRA" )
assignMaterial( "Pressure layer", "SHAPE", [ "Pressure Layer", "Pressure Layer_1" ] )
assignGeometry( "Pressure layer", "SHAPE", [ "Pressure Layer", "Pressure Layer_1"] )
assignElementData( "Pressure layer", "SHAPE", [ "Pressure Layer", "Pressure Layer_1"
] )
```

```
192
193 #Material Steel
194 addMaterial( "Steel", "MCSTEL", "ISOTRO", [] )
195 setParameter( "MATERIAL", "Steel", "LINEAR/ELASTI/YOUNG", 210000 )
196 setParameter( "MATERIAL", "Steel", "LINEAR/ELASTI/POISON", 0.3 )
197 setParameter( "MATERIAL", "Steel", "LINEAR/MASS/DENSIT", 0 )
198 addGeometry( "Plate", "SHEET", "MEMBRA", [] )
199 setParameter( "GEOMET", "Plate", "THICK", 1000 )
200 addElementData( "Plate" )
201
202 setElementClassType( "SHAPE", [ "Support", "Loadplate" ], "MEMBRA" )
203 assignMaterial( "Steel", "SHAPE", [ "Support", "Loadplate" ] )
204 assignGeometry( "Plate", "SHAPE", [ "Support", "Loadplate" ] )
205 assignElementData( "Plate", "SHAPE", [ "Support", "Loadplate" ] )
206
207 #Concrete Interface
208 # addMaterial( "Interface Concrete", "INTERF", "FRICTI", [] )
209 # setParameter( "MATERIAL", "Interface Concrete", "LINEAR/IFTYP", "LIN2D" )
210 # setParameter( "MATERIAL", "Interface Concrete", "COULOM/OPNTYP", "GAP" )
211 # setParameter( "MATERIAL", "Interface Concrete", "COULOM/GAPVAL", 0.5 )
212 # setParameter( "MATERIAL", "Interface Concrete", "COULOM/COHESI", 0.2 )
213 # setParameter( "MATERIAL", "Interface Concrete", "COULOM/PHI", 22 )
214 # setParameter( "MATERIAL", "Interface Concrete", "COULOM/PSI", 22 )
215 # setParameter( "MATERIAL", "Interface Concrete", "LINEAR/ELAS2/DSNY", 10 )
216 # setParameter( "MATERIAL", "Interface Concrete", "LINEAR/ELAS2/DSSX", 10 )
217 # setParameter( "MATERIAL", "Interface Concrete", "COULOM/FRCHSR/MODE2", 0 )
218
219 addMaterial( "Interface Concrete", "INTERF", "FRICEL", [] )
220 setParameter( "MATERIAL", "Interface Concrete", "LINEAR/IFTYP", "LIN2D" )
221 setParameter( "MATERIAL", "Interface Concrete", "NEFRIC/SPCTYP", "USER" )
222 setParameter( "MATERIAL", "Interface Concrete", "NEFRIC/COHESI", 0.2 )
223 setParameter( "MATERIAL", "Interface Concrete", "NEFRIC/PHI", 22 )
224 setParameter( "MATERIAL", "Interface Concrete", "LINEAR/ELAS2/DSNY", 100 )
225 setParameter( "MATERIAL", "Interface Concrete", "LINEAR/ELAS2/DSSX", 100 )
226
227 addGeometry( "Interface Concrete", "LINE", "STLIIF", [] )
228 setParameter( "GEOMET", "Interface Concrete", "LIFMEM/THICK", 1000 )
229 addElementData( "Interface Concrete" )
230 createConnection( "Interface Concrete", "INTER", "SHAPEEDGE", "SHAPEEDGE" )
231 setParameter( "GEOMETRYCONNECTION", "Interface Concrete", "MODE", "CLOSED" )
232 setElementClassType( "GEOMETRYCONNECTION", "Interface Concrete", "STLIIF" )
233 assignMaterial( "Interface Concrete", "GEOMETRYCONNECTION", "Interface Concrete" )
234 assignGeometry( "Interface Concrete", "GEOMETRYCONNECTION", "Interface Concrete" )
235 assignElementData( "Interface Concrete", "GEOMETRYCONNECTION", "Interface Concrete" )
236 setParameter( "GEOMETRYCONNECTION", "Interface Concrete", "FLIP", False )
237 attachTo( "GEOMETRYCONNECTION", "Interface Concrete", "SOURCE", "Pressure Layer", [[
    500, 70, 0 ]] )
238 attachTo( "GEOMETRYCONNECTION", "Interface Concrete", "TARGET", "Slab", [[ 500, 70,
    0 ]] )
239 attachTo( "GEOMETRYCONNECTION", "Interface Concrete", "SOURCE", "Pressure Layer_1",
    [[ 1450, 70, 0 ]] )
240 attachTo( "GEOMETRYCONNECTION", "Interface Concrete", "TARGET", "Slab_1", [[ 1450,
    70, 0 ]] )
241
242 #Steel Interface
243 addMaterial( "Interface Steel", "INTERF", "NONLIF", [] )
244 setParameter( "MATERIAL", "Interface Steel", "LINEAR/IFTYP", "LIN2D" )
245 setParameter( "MATERIAL", "Interface Steel", "LINEAR/ELAS2/DSNY", 32000 )
246 setParameter( "MATERIAL", "Interface Steel", "LINEAR/ELAS2/DSSX", 0.00032 )
247 setParameter( "MATERIAL", "Interface Steel", "NONLIN/IFNOTE", "NOTSHR" )
248 setParameter( "MATERIAL", "Interface Steel", "NONLIN/NLEL8/NOTENS", [ 0.001, 0 ] )
249 setParameter( "MATERIAL", "Interface Steel", "NONLIN/NLEL8/NOSHTE", [ 0.001, 0 ] )
250 addGeometry( "Interface Steel", "LINE", "STLIIF", [] )
251 setParameter( "GEOMET", "Interface Steel", "LIFMEM/THICK", 1000 )
252 addElementData( "Interface Steel" )
253 createConnection( "Interface Steel", "INTER", "SHAPEEDGE", "SHAPEEDGE" )
254
255 setParameter( "GEOMETRYCONNECTION", "Interface Steel", "CONTYP", "INTER" )
256 setParameter( "GEOMETRYCONNECTION", "Interface Steel", "MODE", "AUTO" )
257 setElementClassType( "GEOMETRYCONNECTION", "Interface Steel", "STLIIF" )
258 assignMaterial( "Interface Steel", "GEOMETRYCONNECTION", "Interface Steel" )
```

```
259 assignGeometry( "Interface Steel", "GEOMETRYCONNECTION", "Interface Steel" )
260 assignElementData( "Interface Steel", "GEOMETRYCONNECTION", "Interface Steel" )
261 setParameter( "GEOMETRYCONNECTION", "Interface Steel", "FLIP", False )
262 attachTo( "GEOMETRYCONNECTION", "Interface Steel", "SOURCE", "Pressure Layer_1", [[
    1100, 300, 0 ]] )
263 attachTo( "GEOMETRYCONNECTION", "Interface Steel", "TARGET", "Loadplate", [[ 1100,
    300, 0 ]] )
264 attachTo( "GEOMETRYCONNECTION", "Interface Steel", "SOURCE", "Slab", [[ 300, 0, 0 ]] )
265 attachTo( "GEOMETRYCONNECTION", "Interface Steel", "TARGET", "Support", [[ 300, 0, 0
    ]] )
266
267 #Material Reinforcement Koppel
268 addMaterial( "Koppel", "REINFO", "REBOND", [ ] )
269 setParameter( "MATERIAL", "Koppel", "REBARS/ELASTI/YOUNG", 200000 )
270 setParameter( "MATERIAL", "Koppel", "REBARS/POISON/POISON", 0.3 )
271 setParameter( "MATERIAL", "Koppel", "REBARS/MASS/DENSIT", 7.8e-09 )
272 setParameter( "MATERIAL", "Koppel", "REBARS/PLATYP", "VMISES" )
273 setParameter( "MATERIAL", "Koppel", "REBARS/PLASTI/TRESSH", "KAPSIG" )
274 setParameter( "MATERIAL", "Koppel", "REBARS/PLASTI/KAPSIG", [ 0, 550, 0.04725, 594 ] )
275 setParameter( "MATERIAL", "Koppel", "RESLIP/SHFTYP", "BONDS6" )
276 setParameter( "MATERIAL", "Koppel", "RESLIP/DSNY", 1000 )
277 setParameter( "MATERIAL", "Koppel", "RESLIP/DSSX", 100 )
278 setParameter( "MATERIAL", "Koppel", "RESLIP/BONDS6/SLPVAL", [ 15.34, 6.13, 0.1, 1,
    2, 10, 0.4 ] )
279 addGeometry( "Koppel", "RELINE", "REBAR", [] )
280 setParameter( "GEOMET", "Koppel", "REITYP", "REITRU" )
281 setParameter( "GEOMET", "Koppel", "REITYP", "CIRBEA" )
282 setParameter( "GEOMET", "Koppel", "CIRBEA/CIRCLE", 16)
283 setParameter( "GEOMET", "Koppel", "TIPELM/SURFAC", 201)
284 assignMaterial( "Koppel", "SHAPE", [
    "Koppel_M12a","Koppel_M12b","Koppel_M12c","Koppel_M12d","Koppel_M12e","Koppel_M12f","K
    oppel_M12g" ] )
285 assignGeometry( "Koppel", "SHAPE", [
    "Koppel_M12a","Koppel_M12b","Koppel_M12c","Koppel_M12d","Koppel_M12e","Koppel_M12f","K
    oppel_M12g" ] )
286 convertToReinforcement( [
    "Koppel_M12a","Koppel_M12b","Koppel_M12c","Koppel_M12d","Koppel_M12e","Koppel_M12f","K
    oppel_M12g" ] )
287 addElementData( "Koppel" )
288 assignElementData( "Koppel", "SHAPE", [
    "Koppel_M12a","Koppel_M12b","Koppel_M12c","Koppel_M12d","Koppel_M12e","Koppel_M12f","K
    oppel_M12g" ] )
289 setParameter( "DATA", "Koppel", "INTERF", "BEAM" )
290 setReinforcementDiscretization( [
    "Koppel_M12a","Koppel_M12b","Koppel_M12c","Koppel_M12d","Koppel_M12e","Koppel_M12f","K
    oppel_M12g" ], "ELEMENT" )
291 setContinuousInInterfaces( [
    "Koppel_M12a","Koppel_M12b","Koppel_M12c","Koppel_M12d","Koppel_M12e","Koppel_M12f","K
    oppel_M12g" ], False )
292
293 #Material Reinforcement Longitudinal
294 addMaterial( "Longitudinal", "REINFO", "REBOND", [] )
295 setParameter( "MATERIAL", "Longitudinal", "REBARS/ELASTI/YOUNG", 200000 )
296 setParameter( "MATERIAL", "Longitudinal", "REBARS/POISON/POISON", 0.3 )
297 setParameter( "MATERIAL", "Longitudinal", "REBARS/MASS/DENSIT", 7.8e-09 )
298 setParameter( "MATERIAL", "Longitudinal", "REBARS/PLATYP", "VMISES" )
299 setParameter( "MATERIAL", "Longitudinal", "REBARS/PLASTI/TRESSH", "KAPSIG" )
300 setParameter( "MATERIAL", "Longitudinal", "REBARS/PLASTI/KAPSIG", [ 0, 550, 0.04725,
    594 ] )
301 setParameter( "MATERIAL", "Longitudinal", "RESLIP/SHFTYP", "BONDS6" )
302 setParameter( "MATERIAL", "Longitudinal", "RESLIP/DSNY", 1000 )
303 setParameter( "MATERIAL", "Longitudinal", "RESLIP/DSSX", 100 )
304 setParameter( "MATERIAL", "Longitudinal", "RESLIP/BONDS6/SLPVAL", [ 17.63, 7.05,
    0.1, 1, 2, 10, 0.4 ] )
305 addGeometry( "Longitudinal", "RELINE", "REBAR", [] )
306 setParameter( "GEOMET", "Longitudinal", "REITYP", "REITRU" )
307 setParameter( "GEOMET", "Longitudinal", "REITYP", "CIRBEA" )
308 setParameter( "GEOMET", "Longitudinal", "CIRBEA/CIRCLE", 10)
309 setParameter( "GEOMET", "Longitudinal", "TIPELM/SURFAC", 78.5 )
310 assignMaterial( "Longitudinal", "SHAPE", [
    "Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d","Reinfo_M8e","Reinfo_M8f","Reinfo_
```

```
M8g","Reinfo_M8h","Reinfo_M8i","Reinfo_M8j","Reinfo_M8k","Reinfo_M8l","Reinfo_M8m","Re
info_M8n","Reinfo_M8o","Reinfo_M8p","Reinfo_M8q","Reinfo_M8r" ] )
311 assignGeometry( "Longitudinal", "SHAPE", [
"Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d","Reinfo_M8e","Reinfo_M8f","Reinfo_
M8g","Reinfo_M8h","Reinfo_M8i","Reinfo_M8j","Reinfo_M8k","Reinfo_M8l","Reinfo_M8m","Re
info_M8n","Reinfo_M8o","Reinfo_M8p","Reinfo_M8q","Reinfo_M8r" ] )
312 convertToReinforcement( [
"Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d","Reinfo_M8e","Reinfo_M8f","Reinfo_
M8g","Reinfo_M8h","Reinfo_M8i","Reinfo_M8j","Reinfo_M8k","Reinfo_M8l","Reinfo_M8m","Re
info_M8n","Reinfo_M8o","Reinfo_M8p","Reinfo_M8q","Reinfo_M8r" ] )
313 addElementData( "Longitudinal" )
314 assignElementData( "Longitudinal", "SHAPE", [
"Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d","Reinfo_M8e","Reinfo_M8f","Reinfo_
M8g","Reinfo_M8h","Reinfo_M8i","Reinfo_M8j","Reinfo_M8k","Reinfo_M8l","Reinfo_M8m","Re
info_M8n","Reinfo_M8o","Reinfo_M8p","Reinfo_M8q","Reinfo_M8r" ] )
315 setParameter( "DATA", "Longitudinal", "INTERF", "BEAM" )
316 setReinforcementDiscretization( [
"Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d","Reinfo_M8e","Reinfo_M8f","Reinfo_
M8g","Reinfo_M8h","Reinfo_M8i","Reinfo_M8j","Reinfo_M8k","Reinfo_M8l","Reinfo_M8m","Re
info_M8n","Reinfo_M8o","Reinfo_M8p","Reinfo_M8q","Reinfo_M8r" ], "ELEMENT" )
317 setContinuousInInterfaces( [
"Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d","Reinfo_M8e","Reinfo_M8f","Reinfo_
M8g","Reinfo_M8h","Reinfo_M8i","Reinfo_M8j","Reinfo_M8k","Reinfo_M8l","Reinfo_M8m","Re
info_M8n","Reinfo_M8o","Reinfo_M8p","Reinfo_M8q","Reinfo_M8r" ], False )
318
319 #Material Reinforcement Lattice girder
320 addMaterial( "Lattice girder", "REINFO", "REBOND", [ "ANCHOR" ] )
321 setParameter( "MATERIAL", "Lattice girder", "REBARS/ELASTI/YOUNG", 200000 )
322 setParameter( "MATERIAL", "Lattice girder", "REBARS/POISON/POISON", 0.3 )
323 setParameter( "MATERIAL", "Lattice girder", "REBARS/MASS/DENSIT", 7.8e-09 )
324 setParameter( "MATERIAL", "Lattice girder", "REBARS/PLATYP", "VMISES" )
325 setParameter( "MATERIAL", "Lattice girder", "REBARS/PLASTI/TRESSH", "KAPSIG" )
326 setParameter( "MATERIAL", "Lattice girder", "REBARS/PLASTI/KAPSIG", [ 0, 550,
0.04725, 594 ] )
327 setParameter( "MATERIAL", "Lattice girder", "RESLIP/SHFTYP", "BONDS6" )
328 setParameter( "MATERIAL", "Lattice girder", "RESLIP/DSNY", 1000 )
329 setParameter( "MATERIAL", "Lattice girder", "RESLIP/DSSX", 100 )
330
331 setParameter( "MATERIAL", "Lattice girder", "RESLIP/BONDS6/SLPVAL", [ 0.61, 0.61,
0.01, 0.02, 0.03, 0.04, 0.5 ] )
332
333 setParameter( "MATERIAL", "Lattice girder", "REANCH/DTIPNX", 10000 )
334 setParameter( "MATERIAL", "Lattice girder", "REANCH/DTIPSY", 10000 )
335 setParameter( "MATERIAL", "Lattice girder", "REANCH/TIPLOC", "BOTH" )
336
337 addGeometry( "Lattice girder", "RELINE", "REBAR", [] )
338 setParameter( "GEOMET", "Lattice girder", "REITYP", "REITRU" )
339 setParameter( "GEOMET", "Lattice girder", "REITYP", "CIRBEA" )
340 setParameter( "GEOMET", "Lattice girder", "CIRBEA/CIRCLE", 5 )
341 setParameter( "GEOMET", "Lattice girder", "TIPELM/SURFAC", 19.6 )
342 assignMaterial( "Lattice girder", "SHAPE", [
"Bar_M41al","Bar_M41ar","Bar_M41bl","Bar_M41br","Bar_M41cl","Bar_M41cr","Bar_M41dl","B
ar_M41dr","Bar_M41el","Bar_M41er","Bar_M41fl","Bar_M41fr","Bar_M41gl","Bar_M41gr","Bar
_M41hl","Bar_M41hr","Bar_M41il","Bar_M41ir","Bar_M41jl","Bar_M41jr","Bar_M42al","Bar_M
42ar","Bar_M42bl","Bar_M42br","Bar_M42cl","Bar_M42cr","Bar_M42dl","Bar_M42dr","Bar_M42
el","Bar_M42er","Bar_M42fl","Bar_M42fr","Bar_M42gl","Bar_M42gr","Bar_M42hl","Bar_M42hr
","Bar_M42il","Bar_M42ir","Bar_M42jl","Bar_M42jr","Bar_M43al","Bar_M43ar","Bar_M43bl",
"Bar_M43br","Bar_M43cl","Bar_M43cr","Bar_M43dl","Bar_M43dr","Bar_M43el","Bar_M43er","B
ar_M43fl","Bar_M43fr","Bar_M43gl","Bar_M43gr","Bar_M43hl","Bar_M43hr","Bar_M43il","Bar
_M43ir","Bar_M43jl","Bar_M43jr" ] )
343 assignGeometry( "Lattice girder", "SHAPE", [
"Bar_M41al","Bar_M41ar","Bar_M41bl","Bar_M41br","Bar_M41cl","Bar_M41cr","Bar_M41dl","B
ar_M41dr","Bar_M41el","Bar_M41er","Bar_M41fl","Bar_M41fr","Bar_M41gl","Bar_M41gr","Bar
_M41hl","Bar_M41hr","Bar_M41il","Bar_M41ir","Bar_M41jl","Bar_M41jr","Bar_M42al","Bar_M
42ar","Bar_M42bl","Bar_M42br","Bar_M42cl","Bar_M42cr","Bar_M42dl","Bar_M42dr","Bar_M42
el","Bar_M42er","Bar_M42fl","Bar_M42fr","Bar_M42gl","Bar_M42gr","Bar_M42hl","Bar_M42hr
","Bar_M42il","Bar_M42ir","Bar_M42jl","Bar_M42jr","Bar_M43al","Bar_M43ar","Bar_M43bl",
"Bar_M43br","Bar_M43cl","Bar_M43cr","Bar_M43dl","Bar_M43dr","Bar_M43el","Bar_M43er","B
ar_M43fl","Bar_M43fr","Bar_M43gl","Bar_M43gr","Bar_M43hl","Bar_M43hr","Bar_M43il","Bar
_M43ir","Bar_M43jl","Bar_M43jr"] )
344 convertToReinforcement( [
```

```
"Bar_M41al","Bar_M41ar","Bar_M41bl","Bar_M41br","Bar_M41cl","Bar_M41cr","Bar_M41dl","B
ar_M41dr","Bar_M41el","Bar_M41er","Bar_M41fl","Bar_M41fr","Bar_M41gl","Bar_M41gr","Bar
_M41hl","Bar_M41hr","Bar_M41il","Bar_M41ir","Bar_M41jl","Bar_M41jr","Bar_M42al","Bar_M
42ar","Bar_M42bl","Bar_M42br","Bar_M42cl","Bar_M42cr","Bar_M42dl","Bar_M42dr","Bar_M42
el","Bar_M42er","Bar_M42fl","Bar_M42fr","Bar_M42gl","Bar_M42gr","Bar_M42hl","Bar_M42hr
","Bar_M42il","Bar_M42ir","Bar_M42jl","Bar_M42jr","Bar_M43al","Bar_M43ar","Bar_M43bl",
"Bar_M43br","Bar_M43cl","Bar_M43cr","Bar_M43dl","Bar_M43dr","Bar_M43el","Bar_M43er","B
ar_M43fl","Bar_M43fr","Bar_M43gl","Bar_M43gr","Bar_M43hl","Bar_M43hr","Bar_M43il","Bar
_M43ir","Bar_M43jl","Bar_M43jr" ] )
addElementData( "Bar" )
assignElementData( "Bar", "SHAPE", [
"Bar_M41al","Bar_M41ar","Bar_M41bl","Bar_M41br","Bar_M41cl","Bar_M41cr","Bar_M41dl","B
ar_M41dr","Bar_M41el","Bar_M41er","Bar_M41fl","Bar_M41fr","Bar_M41gl","Bar_M41gr","Bar
_M41hl","Bar_M41hr","Bar_M41il","Bar_M41ir","Bar_M41jl","Bar_M41jr","Bar_M42al","Bar_M
42ar","Bar_M42bl","Bar_M42br","Bar_M42cl","Bar_M42cr","Bar_M42dl","Bar_M42dr","Bar_M42
el","Bar_M42er","Bar_M42fl","Bar_M42fr","Bar_M42gl","Bar_M42gr","Bar_M42hl","Bar_M42hr
","Bar_M42il","Bar_M42ir","Bar_M42jl","Bar_M42jr","Bar_M43al","Bar_M43ar","Bar_M43bl",
"Bar_M43br","Bar_M43cl","Bar_M43cr","Bar_M43dl","Bar_M43dr","Bar_M43el","Bar_M43er","B
ar_M43fl","Bar_M43fr","Bar_M43gl","Bar_M43gr","Bar_M43hl","Bar_M43hr","Bar_M43il","Bar
_M43ir","Bar_M43jl","Bar_M43jr"] )
setParameter( "DATA", "Bar", "./INTERF", [] )
setParameter( "DATA", "Bar", "INTERF", "BEAM" )
setReinforcementDiscretization( [
"Bar_M41al","Bar_M41ar","Bar_M41bl","Bar_M41br","Bar_M41cl","Bar_M41cr","Bar_M41dl","B
ar_M41dr","Bar_M41el","Bar_M41er","Bar_M41fl","Bar_M41fr","Bar_M41gl","Bar_M41gr","Bar
_M41hl","Bar_M41hr","Bar_M41il","Bar_M41ir","Bar_M41jl","Bar_M41jr","Bar_M42al","Bar_M
42ar","Bar_M42bl","Bar_M42br","Bar_M42cl","Bar_M42cr","Bar_M42dl","Bar_M42dr","Bar_M42
el","Bar_M42er","Bar_M42fl","Bar_M42fr","Bar_M42gl","Bar_M42gr","Bar_M42hl","Bar_M42hr
","Bar_M42il","Bar_M42ir","Bar_M42jl","Bar_M42jr","Bar_M43al","Bar_M43ar","Bar_M43bl",
"Bar_M43br","Bar_M43cl","Bar_M43cr","Bar_M43dl","Bar_M43dr","Bar_M43el","Bar_M43er","B
ar_M43fl","Bar_M43fr","Bar_M43gl","Bar_M43gr","Bar_M43hl","Bar_M43hr","Bar_M43il","Bar
_M43ir","Bar_M43jl","Bar_M43jr"], "SECTION" )

#Composed Line
addGeometry( "Composed Line", "LINE", "COMLIN", [] )
setParameter( "GEOMET", "Composed Line", "DISTAN/THICK", 300 )
setElementClassType( "SHAPE", [ "Composed Line" ], "COMLIN" )
assignGeometry( "Composed Line", "SHAPE", [ "Composed Line" ] )
resetElementData( "SHAPE", [ "Composed Line" ] )

#Assign Supports
addSet( "GEOMETRYSUPPORTSET", "Support" )

createPointSupport( "Support", "Support" )
setParameter( "GEOMETRYSUPPORT", "Support", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Support", "TRANSL", [ 0, 1, 0 ] )
setParameter( "GEOMETRYSUPPORT", "Support", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Support", "Support", [[ 300, -30, 0 ]] )

addSet( "GEOMETRYSUPPORTSET", "Sym_X" )

createLineSupport( "Sym_X", "Sym_X" )
setParameter( "GEOMETRYSUPPORT", "Sym_X", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Sym_X", "TRANSL", [ 1, 0, 0 ] )
setParameter( "GEOMETRYSUPPORT", "Sym_X", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Sym_X", "Pressure Layer_1", [[ 1900, 185, 0 ]] )

createPointSupport( "Symp_X", "Symp_X" )
setParameter( "GEOMETRYSUPPORT", "Symp_X", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Symp_X", "TRANSL", [ 1, 0, 0 ] )
setParameter( "GEOMETRYSUPPORpT", "Symp_X", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Symp_X", "Koppel_M12a", [[ 1900, 76, 0 ]] )
attach( "GEOMETRYSUPPORT", "Symp_X", "Koppel_M12b", [[ 1900, 76, 0 ]] )
attach( "GEOMETRYSUPPORT", "Symp_X", "Koppel_M12c", [[ 1900, 76, 0 ]] )
attach( "GEOMETRYSUPPORT", "Symp_X", "Koppel_M12d", [[ 1900, 76, 0 ]] )
attach( "GEOMETRYSUPPORT", "Symp_X", "Koppel_M12e", [[ 1900, 76, 0 ]] )
attach( "GEOMETRYSUPPORT", "Symp_X", "Koppel_M12f", [[ 1900, 76, 0 ]] )
attach( "GEOMETRYSUPPORT", "Symp_X", "Koppel_M12g", [[ 1900, 76, 0 ]] )

addSet( "GEOMETRYSUPPORTSET", "Load point" )

```

```
createPointSupport( "Load point", "Load point" )
setParameter( "GEOMETRYSUPPORT", "Load point", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Load point", "TRANSL", [ 0, 1, 0 ] )
setParameter( "GEOMETRYSUPPORT", "Load point", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Load point", "Loadplate", [[ 1100, 330, 0 ]] )

#Assign Loads
addSet( "GEOMETRYLOADSET", "Load" )

createPointLoad( "Load", "Load" )
setParameter( "GEOMETRYLOAD", "Load", "LODTYP", "DEFORM" )
setParameter( "GEOMETRYLOAD", "Load", "DEFORM/TR/VALUE", -0.1 )
setParameter( "GEOMETRYLOAD", "Load", "DEFORM/TR/DIRECT", 2 )
attach( "GEOMETRYLOAD", "Load", "Loadplate", [[ 1100, 330, 0 ]] )

#Set mesh properties
setElementSize( [ "Loadplate", "Pressure Layer_1", "Slab_1",
"Koppel_M12a","Koppel_M12b","Koppel_M12c","Koppel_M12d","Koppel_M12e","Koppel_M12f","K
oppel_M12g","Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d","Reinfo_M8e","Reinfo_M
8f","Reinfo_M8g","Reinfo_M8h","Reinfo_M8i","Reinfo_M8j","Reinfo_M8k","Reinfo_M8l","Rei
nfo_M8m","Reinfo_M8n","Reinfo_M8o","Bar_M41al","Bar_M41ar","Bar_M41bl","Bar_M41br","Ba
r_M41cl","Bar_M41cr","Bar_M41dl","Bar_M41dr","Bar_M41el","Bar_M41er","Bar_M41fl","Bar_
M41fr","Bar_M41gl","Bar_M41gr","Bar_M41hl","Bar_M41hr","Bar_M41il","Bar_M41ir","Bar_M4
1jl","Bar_M41jr","Bar_M42al","Bar_M42ar","Bar_M42bl","Bar_M42br","Bar_M42cl","Bar_M42c
r","Bar_M42dl","Bar_M42dr","Bar_M42el","Bar_M42er","Bar_M42fl","Bar_M42fr","Bar_M42gl"
,"Bar_M42gr","Bar_M42hl","Bar_M42hr","Bar_M42il","Bar_M42ir","Bar_M42jl","Bar_M42jr","
Bar_M43al","Bar_M43ar","Bar_M43bl","Bar_M43br","Bar_M43cl","Bar_M43cr","Bar_M43dl","Ba
r_M43dr","Bar_M43el","Bar_M43er","Bar_M43fl","Bar_M43fr","Bar_M43gl","Bar_M43gr","Bar_
M43hl","Bar_M43hr","Bar_M43il","Bar_M43ir","Bar_M43jl","Bar_M43jr" ], 10, -1, True )
setMesherType( [ "Loadplate", "Pressure Layer_1", "Slab_1",
"Koppel_M12a","Koppel_M12b","Koppel_M12c","Koppel_M12d","Koppel_M12e","Koppel_M12f","K
oppel_M12g","Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d","Reinfo_M8e","Reinfo_M
8f","Reinfo_M8g","Reinfo_M8h","Reinfo_M8i","Reinfo_M8j","Reinfo_M8k","Reinfo_M8l","Rei
nfo_M8m","Reinfo_M8n","Reinfo_M8o","Bar_M41al","Bar_M41ar","Bar_M41bl","Bar_M41br","Ba
r_M41cl","Bar_M41cr","Bar_M41dl","Bar_M41dr","Bar_M41el","Bar_M41er","Bar_M41fl","Bar_
M41fr","Bar_M41gl","Bar_M41gr","Bar_M41hl","Bar_M41hr","Bar_M41il","Bar_M41ir","Bar_M4
1jl","Bar_M41jr","Bar_M42al","Bar_M42ar","Bar_M42bl","Bar_M42br","Bar_M42cl","Bar_M42c
r","Bar_M42dl","Bar_M42dr","Bar_M42el","Bar_M42er","Bar_M42fl","Bar_M42fr","Bar_M42gl"
,"Bar_M42gr","Bar_M42hl","Bar_M42hr","Bar_M42il","Bar_M42ir","Bar_M42jl","Bar_M42jr","
Bar_M43al","Bar_M43ar","Bar_M43bl","Bar_M43br","Bar_M43cl","Bar_M43cr","Bar_M43dl","Ba
r_M43dr","Bar_M43el","Bar_M43er","Bar_M43fl","Bar_M43fr","Bar_M43gl","Bar_M43gr","Bar_
M43hl","Bar_M43hr","Bar_M43il","Bar_M43ir","Bar_M43jl","Bar_M43jr" ], "HEXQUAD" )
clearMidSideNodeLocation( [ "Loadplate", "Pressure Layer_1", "Slab_1",
"Koppel_M12a","Koppel_M12b","Koppel_M12c","Koppel_M12d","Koppel_M12e","Koppel_M12f","K
oppel_M12g","Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d","Reinfo_M8e","Reinfo_M
8f","Reinfo_M8g","Reinfo_M8h","Reinfo_M8i","Reinfo_M8j","Reinfo_M8k","Reinfo_M8l","Rei
nfo_M8m","Reinfo_M8n","Reinfo_M8o","Bar_M41al","Bar_M41ar","Bar_M41bl","Bar_M41br","Ba
r_M41cl","Bar_M41cr","Bar_M41dl","Bar_M41dr","Bar_M41el","Bar_M41er","Bar_M41fl","Bar_
M41fr","Bar_M41gl","Bar_M41gr","Bar_M41hl","Bar_M41hr","Bar_M41il","Bar_M41ir","Bar_M4
1jl","Bar_M41jr","Bar_M42al","Bar_M42ar","Bar_M42bl","Bar_M42br","Bar_M42cl","Bar_M42c
r","Bar_M42dl","Bar_M42dr","Bar_M42el","Bar_M42er","Bar_M42fl","Bar_M42fr","Bar_M42gl"
,"Bar_M42gr","Bar_M42hl","Bar_M42hr","Bar_M42il","Bar_M42ir","Bar_M42jl","Bar_M42jr","
Bar_M43al","Bar_M43ar","Bar_M43bl","Bar_M43br","Bar_M43cl","Bar_M43cr","Bar_M43dl","Ba
r_M43dr","Bar_M43el","Bar_M43er","Bar_M43fl","Bar_M43fr","Bar_M43gl","Bar_M43gr","Bar_
M43hl","Bar_M43hr","Bar_M43il","Bar_M43ir","Bar_M43jl","Bar_M43jr" ] )
setElementSize( [ "Support","Pressure Layer", "Slab" ], 50, -1, True )
setMesherType( [ "Support","Pressure Layer", "Slab" ], "HEXQUAD" )
setEdgeGrading( "Pressure Layer", [[ 500, 70, 0 ]], [], 50, 50, 10, 0.25, False,
True )
setEdgeGrading( "Slab", [], [[ 500, 70, 0 ]], 50, 50, 10, 0.25, False, True )
generateMesh( [] )
hideView( "GEOM" )
showView( "MESH" )

#Analysis
addAnalysis( "Analysis1" )
addAnalysisCommand( "Analysis1", "NONLIN", "Structural nonlinear" )
renameAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "EXECUT(1)",
"Load" )
addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/LOADNR" )
```

```
421 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/LOADNR", 1 )
422 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/STEPTY", "AUTOMA" )
423 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/SIZES", 250 )
424 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/MAXSIZ", 0.01 )
425 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/MINSIZ", 0.00001 )
426 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/CUTBCK", 0.25 )
427 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/MAXSTP", 100 )
428 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/ARCLEN", True )
429 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/ARCLEN/REGULA/SET" )
430 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/ARCLEN/REGULA/SET(1)/NODES(1)/RNGNRS", "1-9361" )
431 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/ARCLEN/REGULA/SET(1)/DIRECT", 2 )
432 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/SIZES", "1 (100)" )
433 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN", True )
434 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN/REGULA/SET" )
435 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN/REGULA/SET(1)/NODES(1)/RNGNRS", "844 865 866 871
    945-966 1055 1056-2925(89) 3108-3132 3217 3218 3396 3397 3575 3576 3754 3755 3933
    3934 4112 4113 4291 4292 4470 4471 4649 4650 4828 4829 5007 5008 5186 5187 5365 5366
    5544 5545 5723 5724 5902 5903 6081 6082 6260 6261 6439 6440 6618 6619 6797 6798 6976
    6977 7155 7244 7246 7335-7341 7430 7431-7965(89) 8144-8153 8331 8332 8510 8511 8689
    8690 8868 8869 9047 9048 9226 9453-9455 9593 9594 9732 9733 9871 9872 10010 10011
    10149 10150 10288 10289" )
436 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN/REGULA/SET(1)/DIRECT", 2 )
437 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/MAXITE", 250 )
438 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/LINESE", False )
439 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/ENERGY", True )
440 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/DISPLA", False )
441 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/FORCE", True )
442 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/ENERGY/TOLCON", 0.001 )
443 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/ENERGY/NOCONV", "CONTIN" )
444 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/FORCE/TOLCON", 0.01 )
445 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/FORCE/NOCONV", "CONTIN" )
446 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/METHOD/METNAM", "SECANT" )
447
448 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "OUTPUT(1)/SELTYP",
    "USER" )
449 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "OUTPUT(1)/USER" )
450 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/DISPLA(1)/TOTAL" )
451 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/FORCE(1)/REACTI" )
452 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(1)/TOTAL/GREEN" )
453 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(2)/TOTAL/GREEN/PRINCI" )
454 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
```

```
    "OUTPUT(1)/USER/STRAIN(3)/CRKWDT" )
455 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(4)/CRACK/GREEN" )
456 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(5)/CRKWDT/GREEN/PRINCI" )
457 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(1)/TOTAL/CAUCHY" )
458 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(2)/TOTAL/CAUCHY/PRINCI" )
459 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(3)/TOTAL/FORCE/LOCAL" )
460 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(4)/TOTAL/MOMENT/LOCAL" )
461 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(5)/EFFECT/TRACTI/GLOBAL" )
462 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(6)/EFFECT/TRACTI/LOCAL" )
463 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(7)/EFFECT/TRACTI/FRICTI" )
464
465 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(1)/TOTAL/GREEN/GLOBAL/LOCATI", "INTPNT" )
466 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(1)/TOTAL/CAUCHY/GLOBAL/LOCATI", "INTPNT" )
467
468 #Run Analysis
469 # runSolver( [] )
470 # showView( "RESULT" )
471 fitAll( )
472
473
474
```

C - V11 graphs

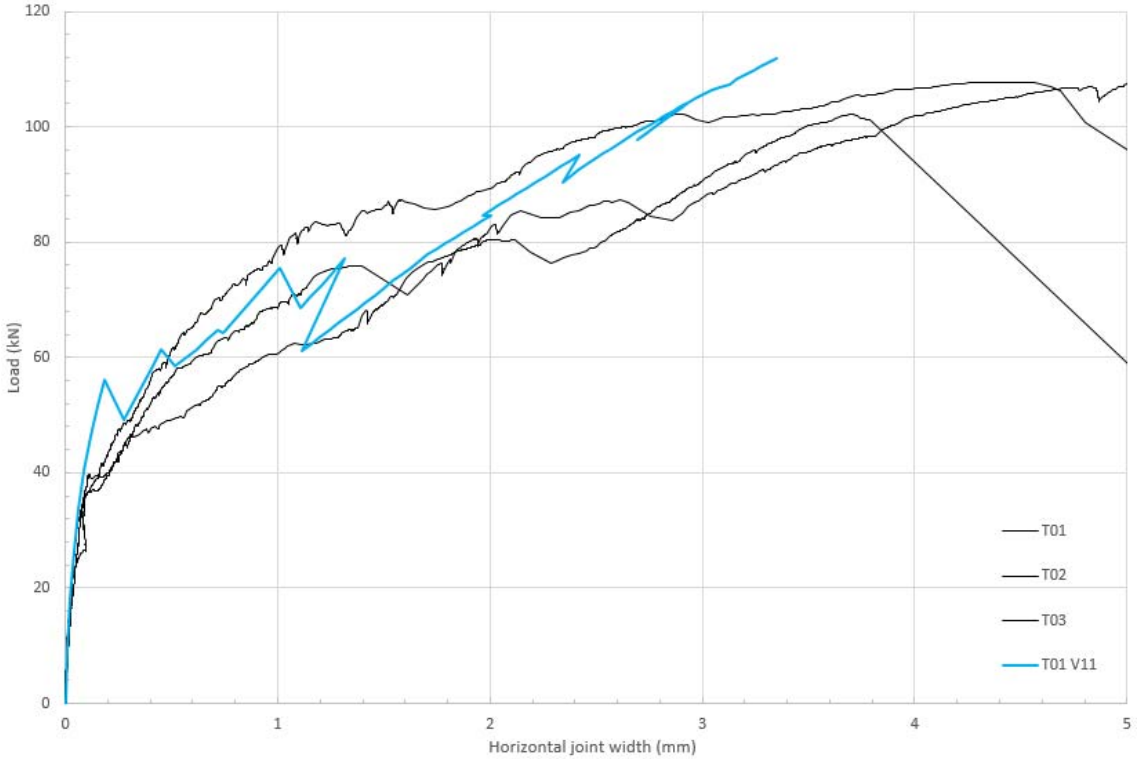


Figure 1: Load-horizontal joint deformation graph T01 DIANA model

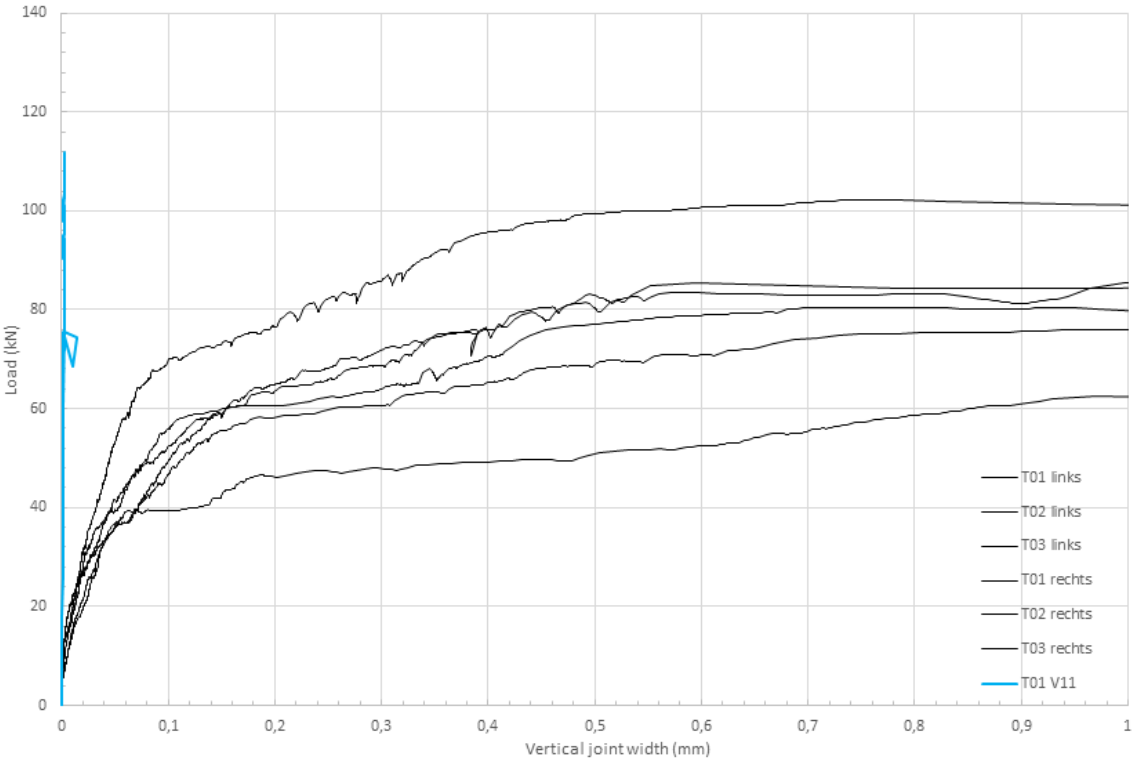


Figure 2: Load-vertical joint width graph T01 DIANA model

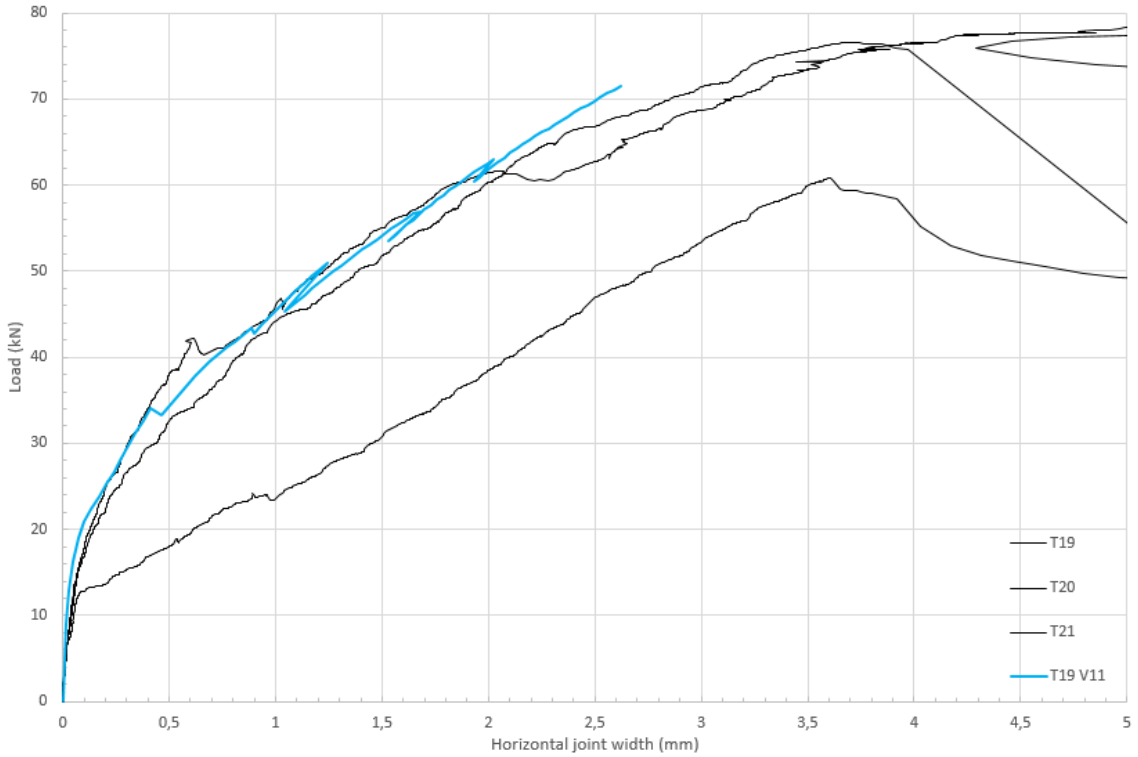


Figure 3: Load-horizontal joint deformation graph T19 DIANA model

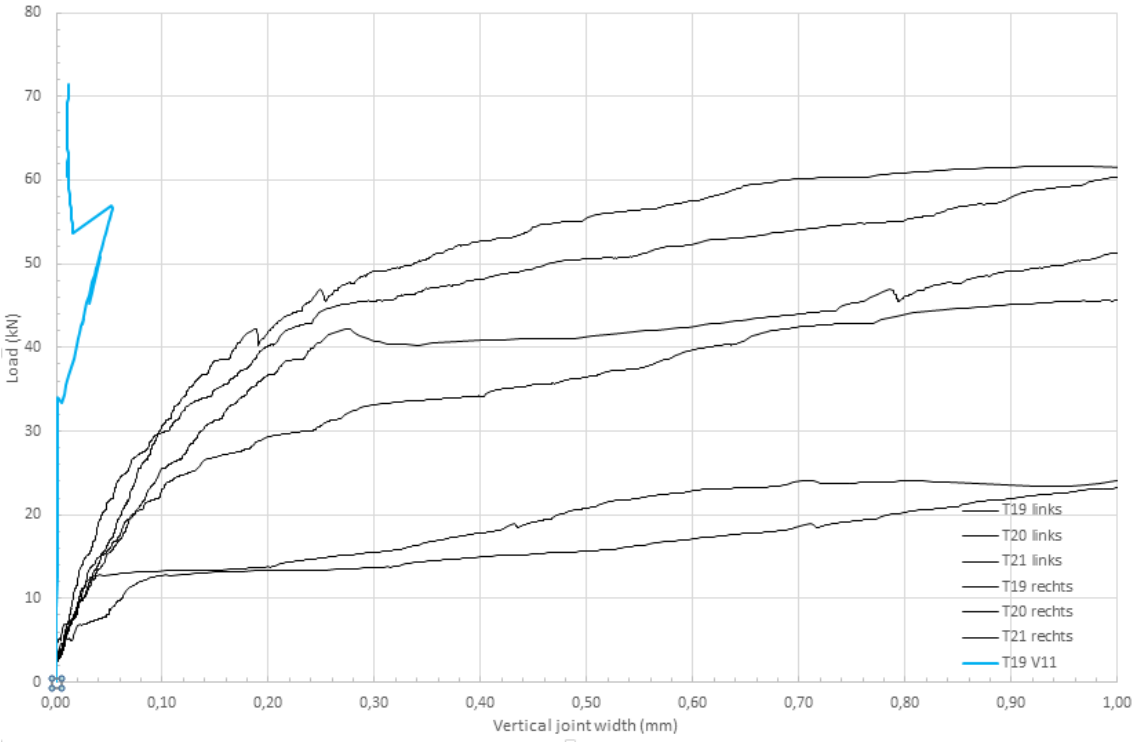


Figure 4: Load-vertical joint width graph T19 DIANA model

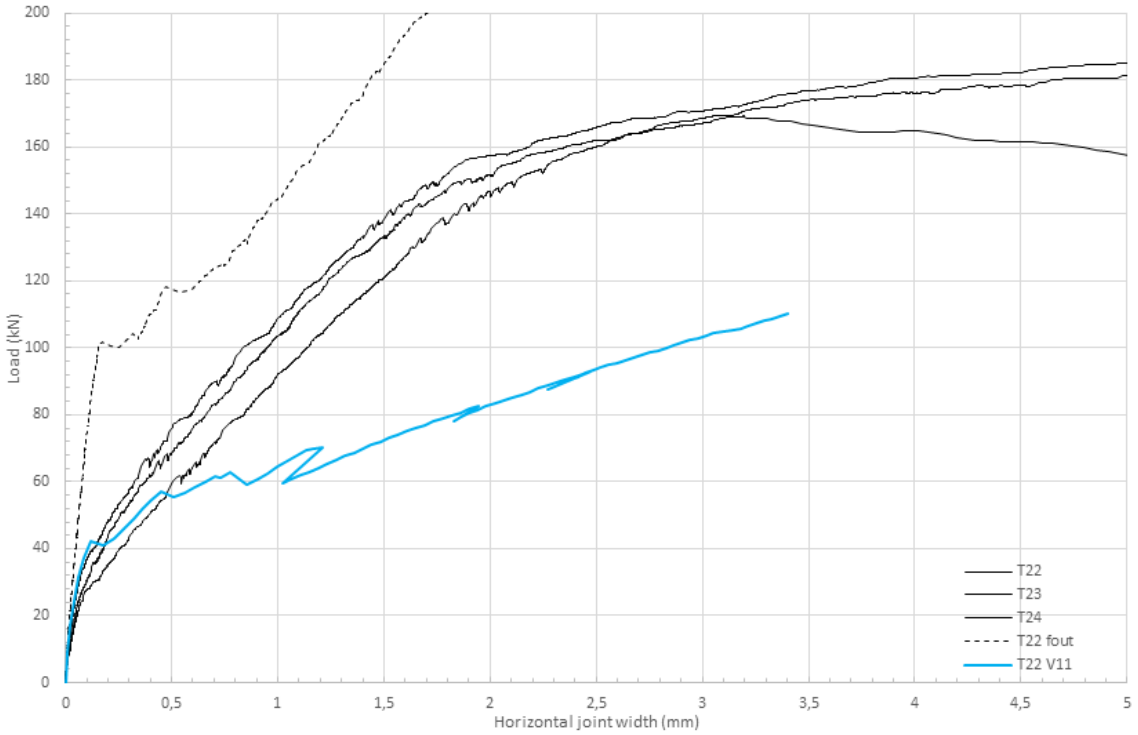


Figure 5: Load-horizontal joint deformation graph T22 DIANA model

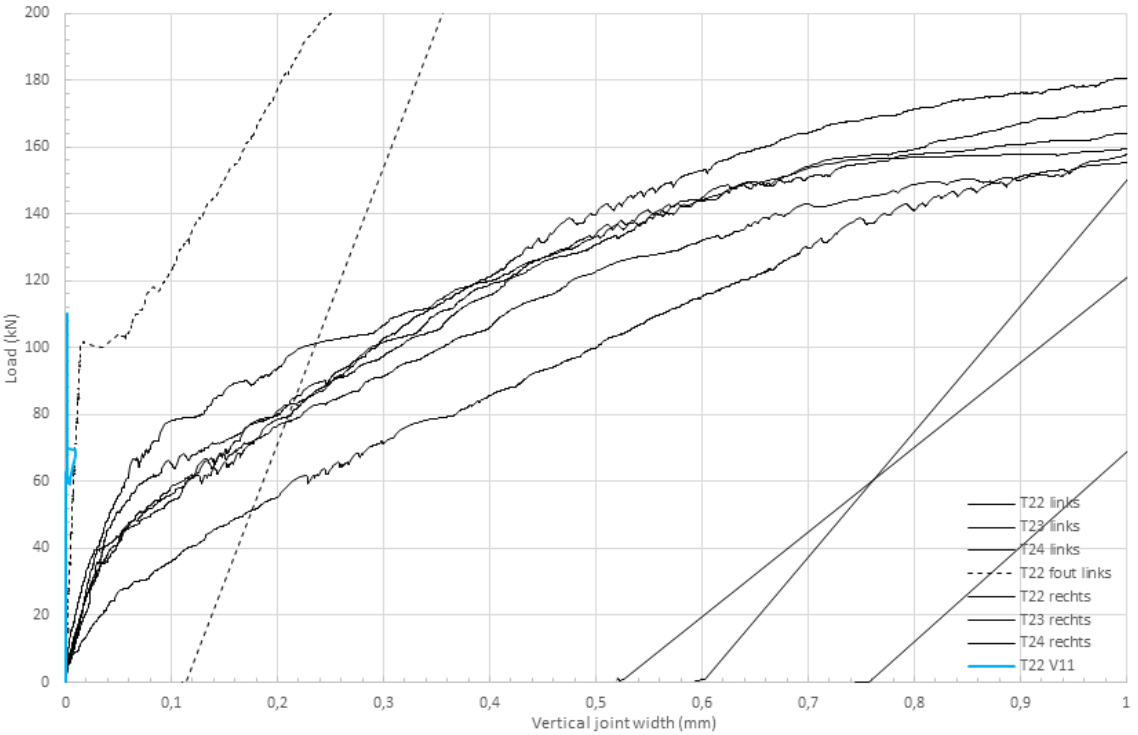


Figure 6: Load-vertical joint width graph T22 DIANA model

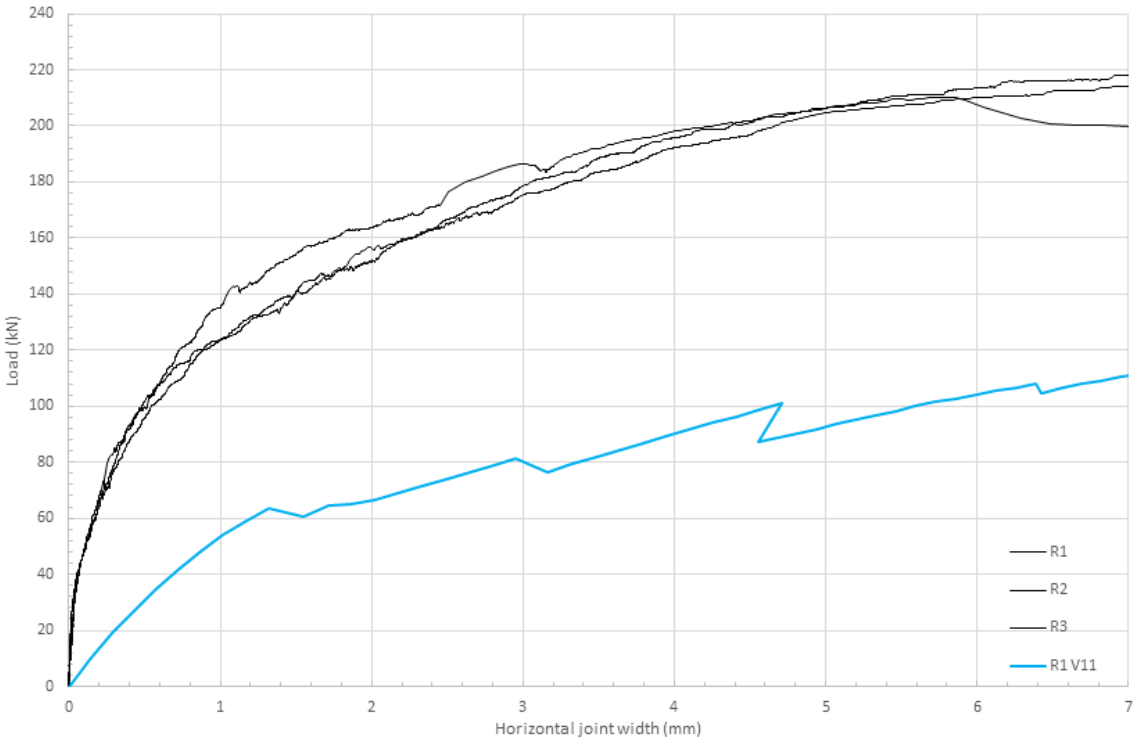


Figure 7: Load-horizontal joint deformation graph R1 DIANA model

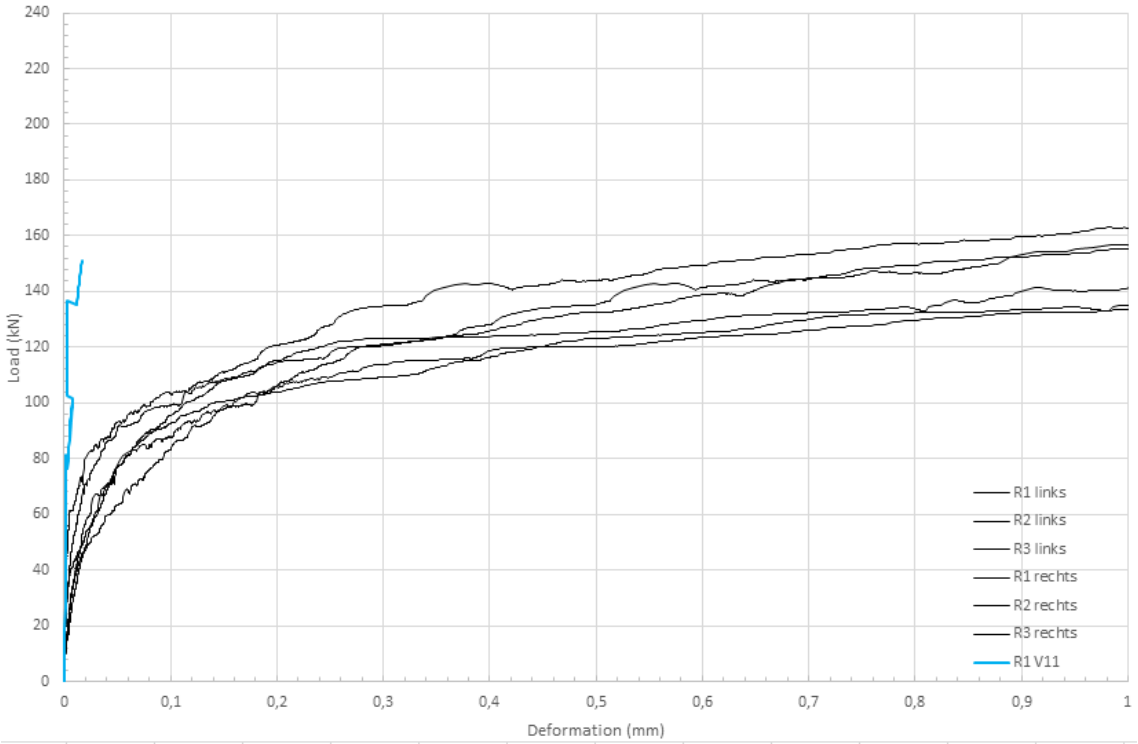


Figure 8: Load-vertical joint width graph R1 DIANA model

D1.1 - 1 element model

```
newProject( "D:/Afstuderen/Python/3D/DIANA Output/ 3D Concrete 100x100", 10 )
setModelAnalysisAspects( [ "STRUCT" ] )
setModelDimension( "3D" )
setDefaultMeshOrder( "QUADRATIC" )
setDefaultMesherType( "HEXQUAD" )
setDefaultMidSideNodeLocation( "ONSHAP" )
setUnit( "LENGTH", "MM" )
setUnit( "FORCE", "N" )
setUnit( "ANGLE", "DEGREE" )

#Create Shapes
createSheet( "Concrete", [[0, 0, 0],[100, 0, 0],[100, 100, 0],[0, 100, 0]] )

extrudeProfile( [ "Concrete" ], [[ 0, 0, 100 ]])

#Regroup Material Parts
rename( "SHAPESET", "Shapes", "Concrete" )

#Material Precast concrete
addMaterial( "Precast concrete", "CONCR", "TSCR", [] )
setParameter( "MATERIAL", "Precast concrete", "LINEAR/ELASTI/YOUNG", 32000 )
setParameter( "MATERIAL", "Precast concrete", "LINEAR/ELASTI/POISON", 0.2 )
setParameter( "MATERIAL", "Precast concrete", "LINEAR/MASS/DENSIT", 2.4e-09 )
setParameter( "MATERIAL", "Precast concrete", "MODTYP/TOTCRK", "ROTATE" )
setParameter( "MATERIAL", "Precast concrete", "TENSIL/TENCRV", "HORDYK" )
setParameter( "MATERIAL", "Precast concrete", "TENSIL/TENSTR", 2.9)
setParameter( "MATERIAL", "Precast concrete", "TENSIL/GF1", 0.1405 )
setParameter( "MATERIAL", "Precast concrete", "TENSIL/RESTST", 0 )
setParameter( "MATERIAL", "Precast concrete", "TENSIL/CBSPEC", "ROTS" )
setParameter( "MATERIAL", "Precast concrete", "TENSIL/POISRE/POIRED", "DAMAGE" )
setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMCRV", "MC2010" )
setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMSTR", 38 )
setParameter( "MATERIAL", "Precast concrete", "COMPRS/EPSC1", 0.0023 )
setParameter( "MATERIAL", "Precast concrete", "COMPRS/EPSCU", 0.0035 )
setParameter( "MATERIAL", "Precast concrete", "COMPRS/CONFIN/CNFCRV", "NONE" )
# setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMCRV", "CONSTA" )
# setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMSTR", 38 )

addGeometry( "Precast concrete", "SOLID", "STRSOL", [] )
setParameter( "GEOMET", "Precast concrete", "LOCAXS", True )
setParameter( "GEOMET", "Precast concrete", "LOCAXS/XAXIS", [ 1, 0, 0 ] )
setParameter( "GEOMET", "Precast concrete", "LOCAXS/YAXIS", [ 0, 1, 0 ] )

addElementData( "Precast concrete" )
setElementClassType( "SHAPE", [ "Concrete" ], "STRSOL" )
assignMaterial( "Precast concrete", "SHAPE", [ "Concrete" ] )
assignGeometry( "Precast concrete", "SHAPE", [ "Concrete" ] )
assignElementData( "Precast concrete", "SHAPE", [ "Concrete"] )

#Assign Supports
addSet( "GEOMETRYSUPPORTSET", "Support" )

createSurfaceSupport( "Top Support", "Support" )
setParameter( "GEOMETRYSUPPORT", "Top Support", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Top Support", "TRANSL", [ 0, 0, 1 ] )
setParameter( "GEOMETRYSUPPORT", "Top Support", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Top Support", "Concrete", [[ 50, 50, 100 ]] )

createSurfaceSupport( "Bot Support", "Support" )
setParameter( "GEOMETRYSUPPORT", "Bot Support", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Bot Support", "TRANSL", [ 0, 0, 1 ] )
setParameter( "GEOMETRYSUPPORT", "Bot Support", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Bot Support", "Concrete", [[ 50, 50, 0 ]] )

createPointSupport( "Topp Support", "Support" )
setParameter( "GEOMETRYSUPPORT", "Topp Support", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Topp Support", "TRANSL", [ 1, 1, 0 ] )
setParameter( "GEOMETRYSUPPORT", "Topp Support", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Topp Support", "Concrete", [[ 0, 0, 100 ]] )

createPointSupport( "Bott Support", "Support" )
```

```
72  setParameter( "GEOMETRYSUPPORT", "Bott Support", "AXES", [ 1, 2 ] )
73  setParameter( "GEOMETRYSUPPORT", "Bott Support", "TRANSL", [ 1, 1, 0 ] )
74  setParameter( "GEOMETRYSUPPORT", "Bott Support", "ROTATI", [ 0, 0, 0 ] )
75  attach( "GEOMETRYSUPPORT", "Bott Support", "Concrete", [[ 0, 0, 0 ]] )
76
77  #Assign Loads
78  addSet( "GEOMETRYLOADSET", "Load" )
79
80  # createLineLoad( "Load", "Load" )
81  # setParameter( "GEOMETRYLOAD", "Load", "LODTYP", "DEFORM" )
82  # setParameter( "GEOMETRYLOAD", "Load", "DEFORM/TR/VALUE", 0.1 )
83  # setParameter( "GEOMETRYLOAD", "Load", "DEFORM/TR/DIRECT", 1 )
84  # attach( "GEOMETRYLOAD", "Load", "Pressure Layer", [[ 200, 100, 250 ]] )
85
86  createSurfaceLoad( "Load", "Load" )
87  setParameter( "GEOMETRYLOAD", "Load", "LODTYP", "DEFORM" )
88  setParameter( "GEOMETRYLOAD", "Load", "DEFORM/SUPP", "Bot Support" )
89  setParameter( "GEOMETRYLOAD", "Load", "DEFORM/TR/VALUE", -0.24 )
90  setParameter( "GEOMETRYLOAD", "Load", "DEFORM/TR/DIRECT", 3 )
91  attach( "GEOMETRYLOAD", "Load", "Concrete", [[ 50, 50, 0 ]] )
92
93  #Set mesh properties
94  setElementSize( [ "Concrete" ], 100, -1, False )
95  setMesherType( [ "Concrete" ], "HEXQUAD" )
96  clearMidSideNodeLocation( [  "Concrete" ] )
97  generateMesh( [] )
98  hideView( "GEOM" )
99  showView( "MESH" )
100
101 #Analysis
102 addAnalysis( "Analysis1" )
103 addAnalysisCommand( "Analysis1", "NONLIN", "Structural nonlinear" )
104 renameAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "EXECUT(1)",
    "Load" )
105 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/LOADNR" )
106 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/LOADNR", 1 )
107 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/STEPTY", "AUTOMA" )
108 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/SIZES", 250 )
109 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/MAXSIZ", 0.01 )
110 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/MINSIZ", 0.00001 )
111 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/CUTBCK", 0.25 )
112 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/MAXSTP", 100 )
113 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/ARCLEN", True )
114 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/ARCLEN/REGULA/SET" )
115 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/ARCLEN/REGULA/SET(1)/NODES(1)/RNGNRS", "1-9361" )
116 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/ARCLEN/REGULA/SET(1)/DIRECT", 2 )
117 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/SIZES", "0.01(100)" )
118 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN", True )
119 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN/REGULA/SET" )
120 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN/REGULA/SET(1)/NODES(1)/RNGNRS", "3 4 9 10 41-96
    237-242 349-354 569-2000(53) 2001-2113 2133-2653(20) 3541-9848(53) 9992 10047-10109
    10163 10202 10223-10227 10281-10283 10323 10324 10364 10365 10405 10406 10446 10447
    10487 10488 10528 10529 10569 10570 10610 10611 10651 10652 10692 10693 10733 10734
    10774 10775 10815 10816 10856 10857 10897 10898 10938 10939 10979 10980 11020 11021
    11061 11062 11102 11103 11143 11144 11184 11185 11225 11226 11266 11267 11307 11308
```

```
    11348 11349 11389 11390 11430 11592-11847 12237 12239 12344 12346 12451 12453 12558
    12560 12665 12667 12772 12774 12879 12881 12986 12988 13093 13095 13200 13202 13307
    13309 13414 13416 13521 13523 13628 13630 13735 13737 13842 13844 13949 13951 14056
    14058 14163 14165 14270 14272 14377 14379 14484 14486 14591 14593 14698 14700 14805
    14807 14912 14914 15019 15021 15126 15128 15181-15183 15289 15290 15396 15397 15503
    15504 15610 15663 15664 15770 15771 15877 15878 15984 15985 16091 17825-17827
    17985-17987 18145-18147 18305-18307 18465 18518-18520 18678-18680 18838-18840
    18998-19000 19158 19211-19213 19371-19373 19531-19533 19691-19693 19851 19904-19906
    20064-20066 20224-20226 20384-20386 20544 20597-20599 20757-20759 20917-20919
    21077-21079 21237 21290-21292 21450-21452 21610-21612 21770-21772 21930 21983-21985
    22143-22145 22303-22305 22463-22465 22623 22676-22678 22836-22838 22996-22998
    23156-23158 23316 23369-23371 23529-23531 23689-23691 23849-23851 24009 24062-24064
    24222-24224 24382-24384 24542-24544 24702 24755-24757 24915-24917 25075-25077
    25235-25237 25395 25448-25450 25608-25610 25768-25770 25928-25930 26088 26141-26143
    26301-26303 26461-26463 26621-26623 26781 26834-26836 26994-26996 27154-27156
    27314-27316 27474 27527-27529 27687-27689 27847-27849 28007-28009 28167 28220-28222
    28380-28382 28540-28542 28700-28702 28860 28913-28915 29073-29075 29233-29235
    29393-29395 29553 29606-29608 29766-29768 29926-29928 30086-30088 30246 30299-30301
    30459-30461 30619-30621 30779-30781 30939 30992-30994 31152-31154 31312-31314
    31472-31474 31632 31685-31687 31845-31847 32005-32007 32165-32167 32325 32378-32380
    32538-32540 32698-32700 32858-32860 33018 33071-33073 33231-33233 33391-33393
    33551-33553 33711 33764-33766 33924-33926 34084-34086 34244-34246 34404 34457-34459
    34617-34619 34777-34779 34937-34939 35097 35150-35152 35310-35312 35470-35472
    35630-35632 35790 35843-35845 36003-36005 36163-36165 36323-36325 36483 36536-36538
    36696-36698 36856-36858 37016-37018 37176-37441(53) 37443 37445-37472 37501 37554
    37659-39090(53) 39091 39095 39183-39210 39262 39263 39315 39316 39611-40772(43)
    40773-40801 40854-42338(53) 42420-42449 42503 42504 42548 42551-42579 42650 42704
    42705 42749 42844 42846 42931 42933 43018 43020 43105 43107 43192 43194 43279 43281
    43366 43368 43453 43455 43540 43542 43627 43629 43714 43716 43801 43803 43888 43890
    43975 43977 44062 44064 44149 44151 44236 44238 44323 44325 44410 44412 44497 44499
    44584 44586 44671 44673 44758 44760 44845 44847 44932 44934 45019 45021 45106 45108
    45193 45195 45238-45323 45950 45951 46057 46110 46111 46217 46374 46376 46481 46483
    46588 46590 46695 46697 46802 46804 46909 46911 47016 47018 47123 47125 47230 47232
    47337 47339 47444 47446 47551 47553 47658 47660 47765 47767 47872 47874 47979 47981
    48086 48088 48193 48195 48300 48302 48407 48409 48514 48516 48621 48623 48728 48730
    48835 48837 48942 48944 49049 49051 49156 49158 49263 49265 49318-49321 49479
    49532-49534 49692 49745-49747 49905 49958-49960 50118 50171-50173 50331 50384-50386
    50544 50597-50599 50757 50810-50812 50970 51023-51025 51183 51236-51238 51396
    51449-51451 51609 51662-51664 51822 51875-51877 52035 52088-52090 52248 52301-52303
    52461 52514-52516 52674 52727-52729 52887 52940-52942 53100 53153-53155 53313
    53366-53368 53526 53579-53581 53739 53792-53794 53952 54005-54007 54165 54218-54220
    54378 54431-54433 54591 54644-54646 54804 54857-54859 55017 55070-55072 55230 55283
    57181 57228-57230 57269-57271 57310-57312 57351-57353 57392-57394 57433-57435
    57474-57476 57583-57585 57692-57694 57801-57803 57910-57912 58019-58021 58128-58130
    58237-58239 58346-58348 58455-58457 58564-58566 58673-58675 58782-58784 58891-58893
    59000-59002 59109-59111 59218-59220" )
121 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN/REGULA/SET(1)/DIRECT", 3 )
122 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/MAXITE", 70 )
123 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/LINESE", True )
124 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/ENERGY", True )
125 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/DISPLA", False )
126 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/FORCE", True )
127 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/ENERGY/TOLCON", 0.001 )
128 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/ENERGY/NOCONV", "CONTIN" )
129 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/FORCE/TOLCON", 0.01 )
130 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/FORCE/NOCONV", "CONTIN" )
131 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/METHOD/METNAM", "SECANT" )
132 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/METHOD/METNAM", "NEWTON" )
133
```

```
134 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "OUTPUT(1)/SELTYP",
    "USER" )
135 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "OUTPUT(1)/USER" )
136
137 ## OUTPUT ##
138 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/DISPLA(1)/TOTAL/TRANSL/GLOBAL" )
139 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(1)/TOTAL/GREEN/LOCAL/LOCATI", "INTPNT" )
140 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(1)/TOTAL/CAUCHY/LOCAL/LOCATI", "INTPNT" )
141
142
143 #Run Analysis
144 # runSolver( [] )
145 # showView( "RESULT" )
146 fitAll( )
147
148
```

D1.2 - 1000 element model

```
newProject( "D:/Afstuderen/Python/3D/DIANA Output/ 3D Concrete 1000x1000", 10 )
setModelAnalysisAspects( [ "STRUCT" ] )
setModelDimension( "3D" )
setDefaultMeshOrder( "QUADRATIC" )
setDefaultMesherType( "HEXQUAD" )
setDefaultMidSideNodeLocation( "ONSHAP" )
setUnit( "LENGTH", "MM" )
setUnit( "FORCE", "N" )
setUnit( "ANGLE", "DEGREE" )

#Create Shapes
# createSheet( "Concrete", [[0, 0, 0],[1000, 0, 0],[1000, 1000, 0],[0, 1000, 0]] )

# extrudeProfile( [ "Concrete" ], [[ 0, 0, 1000 ]])

createBlock( "Concrete", [ 0, 0, 0 ], [ 1000, 1000, 1000 ] )

#Regroup Material Parts
rename( "SHAPESET", "Shapes", "Concrete" )

#Material Precast concrete
addMaterial( "Precast concrete", "CONCR", "TSCR", [] )
setParameter( "MATERIAL", "Precast concrete", "LINEAR/ELASTI/YOUNG", 32000 )
setParameter( "MATERIAL", "Precast concrete", "LINEAR/ELASTI/POISON", 0.2 )
setParameter( "MATERIAL", "Precast concrete", "LINEAR/MASS/DENSIT", 2.4e-09 )
setParameter( "MATERIAL", "Precast concrete", "MODTYP/TOTCRK", "ROTATE" )
setParameter( "MATERIAL", "Precast concrete", "TENSIL/TENCRV", "HORDYK" )
setParameter( "MATERIAL", "Precast concrete", "TENSIL/TENSTR", 2.9)
setParameter( "MATERIAL", "Precast concrete", "TENSIL/GF1", 0.1405 )
setParameter( "MATERIAL", "Precast concrete", "TENSIL/RESTST", 0 )
setParameter( "MATERIAL", "Precast concrete", "TENSIL/CBSPEC", "ROTS" )
setParameter( "MATERIAL", "Precast concrete", "TENSIL/POISRE/POIRED", "DAMAGE" )
setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMCRV", "MC2010" )
setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMSTR", 38 )
setParameter( "MATERIAL", "Precast concrete", "COMPRS/EPSC1", 0.0023 )
setParameter( "MATERIAL", "Precast concrete", "COMPRS/EPSCU", 0.0035 )
setParameter( "MATERIAL", "Precast concrete", "COMPRS/CONFIN/CNFCRV", "NONE" )
# setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMCRV", "CONSTA" )
# setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMSTR", 38 )

addGeometry( "Precast concrete", "SOLID", "STRSOL", [] )
setParameter( "GEOMET", "Precast concrete", "LOCAXS", True )
setParameter( "GEOMET", "Precast concrete", "LOCAXS/XAXIS", [ 1, 0, 0 ] )
setParameter( "GEOMET", "Precast concrete", "LOCAXS/YAXIS", [ 0, 1, 0 ] )

addElementData( "Precast concrete" )
setElementClassType( "SHAPE", [ "Concrete" ], "STRSOL" )
assignMaterial( "Precast concrete", "SHAPE", [ "Concrete" ] )
assignGeometry( "Precast concrete", "SHAPE", [ "Concrete" ] )
assignElementData( "Precast concrete", "SHAPE", [ "Concrete"] )

#Assign Supports
addSet( "GEOMETRYSUPPORTSET", "Support" )

createSurfaceSupport( "Top Support", "Support" )
setParameter( "GEOMETRYSUPPORT", "Top Support", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Top Support", "TRANSL", [ 0, 0, 1 ] )
setParameter( "GEOMETRYSUPPORT", "Top Support", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Top Support", "Concrete", [[ 500, 500, 1000 ]] )

createSurfaceSupport( "Bot Support", "Support" )
setParameter( "GEOMETRYSUPPORT", "Bot Support", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Bot Support", "TRANSL", [ 0, 0, 1 ] )
setParameter( "GEOMETRYSUPPORT", "Bot Support", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Bot Support", "Concrete", [[ 500, 500, 0 ]] )

createPointSupport( "Topp Support", "Support" )
setParameter( "GEOMETRYSUPPORT", "Topp Support", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Topp Support", "TRANSL", [ 1, 1, 0 ] )
setParameter( "GEOMETRYSUPPORT", "Topp Support", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Topp Support", "Concrete", [[ 0, 0, 1000 ]] )
```

```
72
73  createPointSupport( "Bott Support", "Support" )
74  setParameter( "GEOMETRYSUPPORT", "Bott Support", "AXES", [ 1, 2 ] )
75  setParameter( "GEOMETRYSUPPORT", "Bott Support", "TRANSL", [ 1, 1, 0 ] )
76  setParameter( "GEOMETRYSUPPORT", "Bott Support", "ROTATI", [ 0, 0, 0 ] )
77  attach( "GEOMETRYSUPPORT", "Bott Support", "Concrete", [[ 0, 0, 0 ]] )
78
79  #Assign Loads
80  addSet( "GEOMETRYLOADSET", "Load" )
81
82  # createLineLoad( "Load", "Load" )
83  # setParameter( "GEOMETRYLOAD", "Load", "LODTYP", "DEFORM" )
84  # setParameter( "GEOMETRYLOAD", "Load", "DEFORM/TR/VALUE", 0.1 )
85  # setParameter( "GEOMETRYLOAD", "Load", "DEFORM/TR/DIRECT", 1 )
86  # attach( "GEOMETRYLOAD", "Load", "Pressure Layer", [[ 200, 100, 250 ]] )
87
88  createSurfaceLoad( "Load", "Load" )
89  setParameter( "GEOMETRYLOAD", "Load", "LODTYP", "DEFORM" )
90  setParameter( "GEOMETRYLOAD", "Load", "DEFORM/SUPP", "Bot Support" )
91  setParameter( "GEOMETRYLOAD", "Load", "DEFORM/TR/VALUE", -2.4 )
92  setParameter( "GEOMETRYLOAD", "Load", "DEFORM/TR/DIRECT", 3 )
93  attach( "GEOMETRYLOAD", "Load", "Concrete", [[ 500, 500, 0 ]] )
94
95  #Set mesh properties
96  setElementSize( [ "Concrete" ], 100, -1, False )
97  setMesherType( [ "Concrete" ], "HEXQUAD" )
98  clearMidSideNodeLocation( [  "Concrete" ] )
99  generateMesh( [] )
100 hideView( "GEOM" )
101 showView( "MESH" )
102
103 #Analysis
104 addAnalysis( "Analysis1" )
105 addAnalysisCommand( "Analysis1", "NONLIN", "Structural nonlinear" )
106 renameAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "EXECUT(1)",
    "Load" )
107 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/STEPTY", "AUTOMA" )
108 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/SIZES", 1 )
109 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/MAXSIZ", 0.05 )
110 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/MINSIZ", 0.005 )
111 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/CUTBCK", 0.25 )
112 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/SIZES", "0.01(100)" )
113 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN", False )
114 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN/REGULA/SET(1)/NODES(1)/RNGNRS", "ALL" )
115 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN/REGULA/SET(1)/DIRECT", 2 )
116
117 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/MAXITE", 30 )
118 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/LINESE" )
119 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/LINESE", True )
120 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/LINESE/ETAMIN", 0.1 )
121 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/LINESE/ETAMAX", 1 )
122 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/ENERGY" )
123 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/ENERGY", True )
124 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/DISPLA", False )
```

```
125 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/ENERGY/TOLCON", 0.01 )
126 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/ENERGY/NOCONV", "CONTIN" )
127 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/FORCE/NOCONV", "CONTIN" )
128 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/METHOD/METNAM", "SECANT" )
129
130 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "OUTPUT(1)/SELTYP",
    "USER" )
131
132 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/DISPLA(1)/TOTAL/TRANSL/GLOBAL" )
133 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(1)/TOTAL/GREEN/LOCAL/LOCATI", "INTPNT" )
134 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(1)/TOTAL/CAUCHY/LOCAL/LOCATI", "INTPNT" )
135
136 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(1)/TOTAL/GREEN/GLOBAL" )
137 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(2)/TOTAL/GREEN/PRINCI" )
138 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(4)/CRACK/GREEN" )
139 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(7)/CRKWDT/GREEN/GLOBAL" )
140 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(8)/CRKWDT/GREEN/PRINCI" )
141 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(3)/CRACK/CAUCHY/LOCAL" )
142 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(2)/TOTAL/CAUCHY/PRINCI" )
143
144 #Run Analysis
145 # runSolver( [] )
146 # showView( "RESULT" )
147 fitAll( )
148
149
```

D2.1 - Interface in tension

```
newProject( "D:/Afstuderen/Python/3D/DIANA Output/ 3D Interface in tension coulomb friction", 10 )
setModelAnalysisAspects( [ "STRUCT" ] )
setModelDimension( "3D" )
setDefaultMeshOrder( "QUADRATIC" )
setDefaultMesherType( "HEXQUAD" )
setDefaultMidSideNodeLocation( "ONSHAP" )
setUnit( "LENGTH", "MM" )
setUnit( "FORCE", "N" )
setUnit( "ANGLE", "DEGREE" )

#Create Shapes
createBlock( "Precast Concrete", [ 0, 0, 0 ], [ 400, 400, 70 ] )
createBlock( "Pressure Layer", [ 0, 0, 70 ], [ 400, 400, 230 ] )
createPointBody( "point 1", [ 200, 200, 300 ] )
createPointBody( "point 2", [ 200, 200, 0 ] )

imprintIntersection( "Pressure Layer", "point 1", True )
imprintIntersection( "Precast Concrete", "point 2", True )

removeShape( [ "point 1" ] )
removeShape( [ "point 2" ] )

#Material Precast concrete
addMaterial( "Precast concrete", "CONCR", "TSCR", [] )
setParameter( "MATERIAL", "Precast concrete", "LINEAR/ELASTI/YOUNG", 32000 )
setParameter( "MATERIAL", "Precast concrete", "LINEAR/ELASTI/POISON", 0.2 )
setParameter( "MATERIAL", "Precast concrete", "LINEAR/MASS/DENSIT", 2.4e-09 )
setParameter( "MATERIAL", "Precast concrete", "MODTYP/TOTCRK", "ROTATE" )
setParameter( "MATERIAL", "Precast concrete", "TENSIL/TENCRV", "HORDYK" )
setParameter( "MATERIAL", "Precast concrete", "TENSIL/TENSTR", 2.9)
setParameter( "MATERIAL", "Precast concrete", "TENSIL/GF1", 0.1405 )
setParameter( "MATERIAL", "Precast concrete", "TENSIL/RESTST", 0 )
setParameter( "MATERIAL", "Precast concrete", "TENSIL/CBSPEC", "ROTS" )
setParameter( "MATERIAL", "Precast concrete", "TENSIL/POISRE/POIRED", "DAMAGE" )
setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMCRV", "MC2010" )
setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMSTR", 38 )
setParameter( "MATERIAL", "Precast concrete", "COMPRS/EPSC1", 0.0023 )
setParameter( "MATERIAL", "Precast concrete", "COMPRS/EPSCU", 0.0035 )
setParameter( "MATERIAL", "Precast concrete", "COMPRS/CONFIN/CNFCRV", "NONE" )
# setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMCRV", "CONSTA" )
# setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMSTR", 38 )

addGeometry( "Precast concrete", "SOLID", "STRSOL", [] )
setParameter( "GEOMET", "Precast concrete", "LOCAXS", True )
setParameter( "GEOMET", "Precast concrete", "LOCAXS/XAXIS", [ 1, 0, 0 ] )
setParameter( "GEOMET", "Precast concrete", "LOCAXS/YAXIS", [ 0, 1, 0 ] )

addElementData( "Precast concrete" )
setElementClassType( "SHAPE", [ "Precast Concrete","Pressure Layer" ], "STRSOL" )
assignMaterial( "Precast concrete", "SHAPE", [ "Precast Concrete","Pressure Layer" ] )
assignGeometry( "Precast concrete", "SHAPE", [ "Precast Concrete","Pressure Layer" ] )
assignElementData( "Precast concrete", "SHAPE", [ "Precast Concrete","Pressure Layer"] )

#Material Interface Concrete-Concrete
addMaterial( "Interface Concrete", "INTERF", "FRICTI", [] )
setParameter( "MATERIAL", "Interface Concrete", "LINEAR/ELAS6/DSNZ", 60000 )
setParameter( "MATERIAL", "Interface Concrete", "LINEAR/ELAS6/DSSX", 6000 )
setParameter( "MATERIAL", "Interface Concrete", "LINEAR/ELAS6/DSSY", 6000 )
setParameter( "MATERIAL", "Interface Concrete", "COULOM/OPNTYP", "GAP" )
setParameter( "MATERIAL", "Interface Concrete", "COULOM/GAPVAL", 0.5 )
setParameter( "MATERIAL", "Interface Concrete", "COULOM/COHESI", 0.5 )
setParameter( "MATERIAL", "Interface Concrete", "COULOM/PHI", 31 )
setParameter( "MATERIAL", "Interface Concrete", "COULOM/PSI", 31 )
setParameter( "MATERIAL", "Interface Concrete", "COULOM/FRCHSR/MODE2", 0 )

addGeometry( "Interface Concrete", "SHEET", "STPLIF", [] )
setParameter( "GEOMET", "Interface Concrete", "LOCAXS", True )
setParameter( "GEOMET", "Interface Concrete", "LOCAXS/XAXIS", [ 1, 0, 0 ] )

```

```
addElementData( "Interface Concrete" )
createConnection( "Interface Concrete", "INTER", "SHAPEFACE", "SHAPEFACE" )
setParameter( "GEOMETRYCONNECTION", "Interface Concrete", "MODE", "CLOSED" )
setElementClassType( "GEOMETRYCONNECTION", "Interface Concrete", "STPLIF" )
assignMaterial( "Interface Concrete", "GEOMETRYCONNECTION", "Interface Concrete" )
assignGeometry( "Interface Concrete", "GEOMETRYCONNECTION", "Interface Concrete" )
assignElementData( "Interface Concrete", "GEOMETRYCONNECTION", "Interface Concrete" )
setParameter( "GEOMETRYCONNECTION", "Interface Concrete", "FLIP", False )
attachTo( "GEOMETRYCONNECTION", "Interface Concrete", "SOURCE", "Precast Concrete", [[ 200, 200, 70 ]] )
attachTo( "GEOMETRYCONNECTION", "Interface Concrete", "TARGET", "Pressure Layer", [[ 200, 200, 70 ]] )

#Assign Supports
addSet( "GEOMETRYSUPPORTSET", "Support" )

createSurfaceSupport( "Top Support", "Support" )
setParameter( "GEOMETRYSUPPORT", "Top Support", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Top Support", "TRANSL", [ 0, 0, 1 ] )
setParameter( "GEOMETRYSUPPORT", "Top Support", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Top Support", "Pressure Layer", [[ 200, 200, 300 ]] )

createPointSupport( "Topp Support", "Support" )
setParameter( "GEOMETRYSUPPORT", "Topp Support", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Topp Support", "TRANSL", [ 1, 1, 0 ] )
setParameter( "GEOMETRYSUPPORT", "Topp Support", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Topp Support", "Pressure Layer", [[ 200, 200, 300 ]] )

createSurfaceSupport( "Bot Support", "Support" )
setParameter( "GEOMETRYSUPPORT", "Bot Support", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Bot Support", "TRANSL", [ 0, 0, 1 ] )
setParameter( "GEOMETRYSUPPORT", "Bot Support", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Bot Support", "Precast Concrete", [[200,200,0]] )

createPointSupport( "Botp Support", "Support" )
setParameter( "GEOMETRYSUPPORT", "Botp Support", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Botp Support", "TRANSL", [ 1, 1, 0 ] )
setParameter( "GEOMETRYSUPPORT", "Botp Support", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Botp Support", "Precast Concrete", [[ 200, 200, 0 ]] )

#Assign Loads
addSet( "GEOMETRYLOADSET", "Load" )

# createLineLoad( "Load", "Load" )
# setParameter( "GEOMETRYLOAD", "Load", "LODTYP", "DEFORM" )
# setParameter( "GEOMETRYLOAD", "Load", "DEFORM/TR/VALUE", 0.1 )
# setParameter( "GEOMETRYLOAD", "Load", "DEFORM/TR/DIRECT", 1 )
# attach( "GEOMETRYLOAD", "Load", "Pressure Layer", [[ 200, 100, 250 ]] )

createSurfaceLoad( "Load", "Load" )
setParameter( "GEOMETRYLOAD", "Load", "LODTYP", "DEFORM" )
setParameter( "GEOMETRYLOAD", "Load", "DEFORM/SUPP", "Botp Support" )
setParameter( "GEOMETRYLOAD", "Load", "DEFORM/TR/VALUE", -0.005 )
setParameter( "GEOMETRYLOAD", "Load", "DEFORM/TR/DIRECT", 3 )
attach( "GEOMETRYLOAD", "Load", "Precast Concrete", [[ 200, 200, 0 ]] )

#Set mesh properties
setElementSize( [ "Pressure Layer","Precast Concrete" ], 33.75, -1, True )
setMesherType( [ "Pressure Layer","Precast Concrete" ], "HEXQUAD" )
clearMidSideNodeLocation( [  "Pressure Layer","Precast Concrete" ] )
generateMesh( [] )
hideView( "GEOM" )
showView( "MESH" )

#Analysis
addAnalysis( "Analysis1" )
addAnalysisCommand( "Analysis1", "NONLIN", "Structural nonlinear" )
renameAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "EXECUT(1)", "Load" )
setAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "EXECUT(1)/LOAD/STEPS/STEPTY", "AUTOMA" )
```

```
137 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/SIZES", 1 )
138 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/MAXSIZ", 0.005 )
139 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/MINSIZ", 0.0005 )
140 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/CUTBCK", 0.25 )
141 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/SIZES", "1.0000(46),0.1000(20)" )
142 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN", False )
143 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN/REGULA/SET(1)/NODES(1)/RNGNRS", "ALL" )
144 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN/REGULA/SET(1)/DIRECT", 2 )
145
146 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/MAXITE", 25 )
147 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/LINESE" )
148 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/LINESE", True )
149 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/LINESE/ETAMIN", 0.1 )
150 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/LINESE/ETAMAX", 1 )
151 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/ENERGY" )
152 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/ENERGY", True )
153 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/DISPLA", False )
154 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/ENERGY/TOLCON", 0.001 )
155 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/ENERGY/NOCONV", "CONTIN" )
156 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/FORCE/NOCONV", "CONTIN" )
157 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/METHOD/METNAM", "SECANT" )
158 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "OUTPUT(1)/SELTYP",
    "USER" )
159
160 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/DISPLA(1)/TOTAL/TRANSL/GLOBAL" )
161 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(1)/TOTAL/GREEN/GLOBAL" )
162 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(2)/TOTAL/GREEN/PRINCI" )
163 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(2)/TOTAL/GREEN/PRINCI" )
164 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(4)/CRACK/GREEN" )
165 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(7)/CRKWDT/GREEN/GLOBAL" )
166 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(8)/CRKWDT/GREEN/PRINCI" )
167 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(3)/CRACK/CAUCHY/LOCAL" )
168 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(2)/TOTAL/CAUCHY/PRINCI" )
169 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(4)/TOTAL/CAUCHY/GLOBAL" )
170
171 #Run Analysis
172 # runSolver( [] )
173 # showView( "RESULT" )
174 fitAll( )
175
176
```

D2.2 - Interface in shear

```
newProject( "D:/Afstuderen/Python/3D/DIANA Output/ 3D Interface in shear", 10 )
setModelAnalysisAspects( [ "STRUCT" ] )
setModelDimension( "3D" )
setDefaultMeshOrder( "QUADRATIC" )
setDefaultMesherType( "HEXQUAD" )
setDefaultMidSideNodeLocation( "ONSHAP" )
setUnit( "LENGTH", "MM" )
setUnit( "FORCE", "N" )
setUnit( "ANGLE", "DEGREE" )

#Create Shapes
createBlock( "Precast Concrete", [ 0, 0, 0 ], [ 400, 400, 70 ] )
createBlock( "Pressure Layer", [ 0, 0, 70 ], [ 400, 400, 230 ] )
# createLine( "line 1", [ 200, 0, 300 ],[ 200, 400, 300 ] )

# imprintIntersection( "Pressure Layer", "line 1", True )

# removeShape( [ "line 1" ] )


#Material Precast concrete
addMaterial( "Precast concrete", "CONCR", "LEI", [] )
setParameter( "MATERIAL", "Precast concrete", "LINEAR/ELASTI/YOUNG", 32000 )
setParameter( "MATERIAL", "Precast concrete", "LINEAR/ELASTI/POISON", 0.2 )
setParameter( "MATERIAL", "Precast concrete", "LINEAR/MASS/DENSIT", 2.4e-09 )

# addMaterial( "Precast concrete", "CONCR", "TSCR", [] )
# setParameter( "MATERIAL", "Precast concrete", "LINEAR/ELASTI/YOUNG", 32000 )
# setParameter( "MATERIAL", "Precast concrete", "LINEAR/ELASTI/POISON", 0.2 )
# setParameter( "MATERIAL", "Precast concrete", "LINEAR/MASS/DENSIT", 2.4e-09 )
# setParameter( "MATERIAL", "Precast concrete", "MODTYP/TOTCRK", "ROTATE" )
# setParameter( "MATERIAL", "Precast concrete", "TENSIL/TENCRV", "HORDYK" )
# setParameter( "MATERIAL", "Precast concrete", "TENSIL/TENSTR", 2.9)
# setParameter( "MATERIAL", "Precast concrete", "TENSIL/GF1", 0.1405 )
# setParameter( "MATERIAL", "Precast concrete", "TENSIL/RESTST", 0 )
# setParameter( "MATERIAL", "Precast concrete", "TENSIL/CBSPEC", "ROTS" )
# setParameter( "MATERIAL", "Precast concrete", "TENSIL/POISRE/POIRED", "DAMAGE" )
# setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMCRV", "MC2010" )
# setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMSTR", 38 )
# setParameter( "MATERIAL", "Precast concrete", "COMPRS/EPSC1", 0.0023 )
# setParameter( "MATERIAL", "Precast concrete", "COMPRS/EPSCU", 0.0035 )
# setParameter( "MATERIAL", "Precast concrete", "COMPRS/CONFIN/CNFCRV", "NONE" )
# # setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMCRV", "CONSTA" )
# # setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMSTR", 38 )

addGeometry( "Precast concrete", "SOLID", "STRSOL", [] )
setParameter( "GEOMET", "Precast concrete", "LOCAXS", True )
setParameter( "GEOMET", "Precast concrete", "LOCAXS/XAXIS", [ 1, 0, 0 ] )
setParameter( "GEOMET", "Precast concrete", "LOCAXS/YAXIS", [ 0, 1, 0 ] )

addElementData( "Precast concrete" )
setElementClassType( "SHAPE", [ "Precast Concrete","Pressure Layer" ], "STRSOL" )
assignMaterial( "Precast concrete", "SHAPE", [ "Precast Concrete","Pressure Layer" ] )
assignGeometry( "Precast concrete", "SHAPE", [ "Precast Concrete","Pressure Layer" ] )
assignElementData( "Precast concrete", "SHAPE", [ "Precast Concrete","Pressure
Layer"] )

#Material Interface Concrete-Concrete
addMaterial( "Interface Concrete", "INTERF", "FRICTI", [] )
setParameter( "MATERIAL", "Interface Concrete", "LINEAR/ELAS6/DSNZ", 60000 )
setParameter( "MATERIAL", "Interface Concrete", "LINEAR/ELAS6/DSSX", 6000 )
setParameter( "MATERIAL", "Interface Concrete", "LINEAR/ELAS6/DSSY", 6000 )
setParameter( "MATERIAL", "Interface Concrete", "COULOM/OPNTYP", "GAP" )
setParameter( "MATERIAL", "Interface Concrete", "COULOM/GAPVAL", 0.5 )
setParameter( "MATERIAL", "Interface Concrete", "COULOM/COHESI", 0.5 )
setParameter( "MATERIAL", "Interface Concrete", "COULOM/PHI", 31 )
setParameter( "MATERIAL", "Interface Concrete", "COULOM/PSI", 31 )
setParameter( "MATERIAL", "Interface Concrete", "COULOM/FRCHSR/MODE2", 0 )

addGeometry( "Interface Concrete", "SHEET", "STPLIF", [] )
setParameter( "GEOMET", "Interface Concrete", "LOCAXS", True )
```

```
setParameter( "GEOMET", "Interface Concrete", "LOCAXS/XAXIS", [ 1, 0, 0 ] )

addElementData( "Interface Concrete" )
createConnection( "Interface Concrete", "INTER", "SHAPEFACE", "SHAPEFACE" )
setParameter( "GEOMETRYCONNECTION", "Interface Concrete", "MODE", "CLOSED" )
setElementClassType( "GEOMETRYCONNECTION", "Interface Concrete", "STPLIF" )
assignMaterial( "Interface Concrete", "GEOMETRYCONNECTION", "Interface Concrete" )
assignGeometry( "Interface Concrete", "GEOMETRYCONNECTION", "Interface Concrete" )
assignElementData( "Interface Concrete", "GEOMETRYCONNECTION", "Interface Concrete" )
setParameter( "GEOMETRYCONNECTION", "Interface Concrete", "FLIP", False )
attachTo( "GEOMETRYCONNECTION", "Interface Concrete", "SOURCE", "Precast Concrete",
[[ 200, 200, 70 ]] )
attachTo( "GEOMETRYCONNECTION", "Interface Concrete", "TARGET", "Pressure Layer", [[
200, 200, 70 ]] )

#Assign Supports
addSet( "GEOMETRYSUPPORTSET", "Support" )

createSurfaceSupport( "Top Support", "Support" )
setParameter( "GEOMETRYSUPPORT", "Top Support", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Top Support", "TRANSL", [ 0, 0, 1 ] )
setParameter( "GEOMETRYSUPPORT", "Top Support", "ROTATI", [ 0, 0, 0 ] )
# attach( "GEOMETRYSUPPORT", "Top Support", "Pressure Layer", [[ 100, 200, 300 ],[
300, 200, 300 ]] )
attach( "GEOMETRYSUPPORT", "Top Support", "Pressure Layer", [[ 200, 200, 300 ]] )

# createLineSupport( "Topp Support", "Support" )
# setParameter( "GEOMETRYSUPPORT", "Topp Support", "AXES", [ 1, 2 ] )
# setParameter( "GEOMETRYSUPPORT", "Topp Support", "TRANSL", [ 1, 0, 0 ] )
# setParameter( "GEOMETRYSUPPORT", "Topp Support", "ROTATI", [ 0, 0, 0 ] )
# attach( "GEOMETRYSUPPORT", "Topp Support", "Pressure Layer", [[ 200, 200, 300 ]] )

# createSurfaceSupport( "Bot Support", "Support" )
# setParameter( "GEOMETRYSUPPORT", "Bot Support", "AXES", [ 1, 2 ] )
# setParameter( "GEOMETRYSUPPORT", "Bot Support", "TRANSL", [ 0, 0, 1 ] )
# setParameter( "GEOMETRYSUPPORT", "Bot Support", "ROTATI", [ 0, 0, 0 ] )
# attach( "GEOMETRYSUPPORT", "Bot Support", "Precast Concrete", [[200,200,0]] )

# createPointSupport( "Botp Support", "Support" )
# setParameter( "GEOMETRYSUPPORT", "Botp Support", "AXES", [ 1, 2 ] )
# setParameter( "GEOMETRYSUPPORT", "Botp Support", "TRANSL", [ 1, 1, 0 ] )
# setParameter( "GEOMETRYSUPPORT", "Botp Support", "ROTATI", [ 0, 0, 0 ] )
# attach( "GEOMETRYSUPPORT", "Botp Support", "Precast Concrete", [[ 200, 200, 0 ]] )

createBodySupport( "Horizontal Support", "Support" )
setParameter( "GEOMETRYSUPPORT", "Horizontal Support", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Horizontal Support", "TRANSL", [ 1, 1, 0 ] )
setParameter( "GEOMETRYSUPPORT", "Horizontal Support", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Horizontal Support", ["Precast Concrete", "Pressure
Layer"] )

#Assign Loads
addSet( "GEOMETRYLOADSET", "Prestress" )
addSet( "GEOMETRYLOADSET", "Deformation" )

# createLineLoad( "Load", "Load" )
# setParameter( "GEOMETRYLOAD", "Load", "LODTYP", "DEFORM" )
# setParameter( "GEOMETRYLOAD", "Load", "DEFORM/TR/VALUE", 0.1 )
# setParameter( "GEOMETRYLOAD", "Load", "DEFORM/TR/DIRECT", 1 )
# attach( "GEOMETRYLOAD", "Load", "Pressure Layer", [[ 200, 100, 250 ]] )

createBodyLoad( "Deform", "Deformation" )
setParameter( "GEOMETRYLOAD", "Deform", "LODTYP", "DEFORM" )
setParameter( "GEOMETRYLOAD", "Deform", "DEFORM/TR/VALUE", 0.0012 )
attach( "GEOMETRYLOAD", "Deform", [ "Precast Concrete" ] )

#Forces
createSurfaceLoad( "Up", "Prestress" )
setParameter( "GEOMETRYLOAD", "Up", "FORCE/VALUE", 10 )
setParameter( "GEOMETRYLOAD", "Up", "FORCE/DIRECT", 3 )
attach( "GEOMETRYLOAD", "Up", "Precast Concrete", [[ 200, 200, 0 ]] )
```

```
138
139 #Set mesh properties
140 setElementSize( [ "Pressure Layer","Precast Concrete" ], 33.75, -1, True )
141 setMesherType( [ "Pressure Layer","Precast Concrete" ], "HEXQUAD" )
142 clearMidSideNodeLocation( [  "Pressure Layer","Precast Concrete" ] )
143 generateMesh( [] )
144 hideView( "GEOM" )
145 showView( "MESH" )
146
147 #Analysis
148 addAnalysis( "Analysis1" )
149 addAnalysisCommand( "Analysis1", "NONLIN", "Structural nonlinear" )
150 renameAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "EXECUT(1)",
    "Prestress" )
151
152 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/LOADNR", 1 )
153 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/STEPTY", "AUTOMA" )
154 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/SIZES", 1 )
155 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/MAXSIZ", 0.1 )
156 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/MINSIZ", 0.0001 )
157 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/CUTBCK", 0.25 )
158 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/MAXSTP", 20 )
159
160 renameAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "EXECUT(2)",
    "Deformation" )
161
162 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(2)/LOAD/LOADNR", 2 )
163 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(2)/LOAD/STEPS/STEPTY", "AUTOMA" )
164 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(2)/LOAD/STEPS/AUTOMA/SIZES", 1 )
165 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(2)/LOAD/STEPS/AUTOMA/MAXSIZ", 0.01 )
166 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(2)/LOAD/STEPS/AUTOMA/MINSIZ", 0.001 )
167 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(2)/LOAD/STEPS/AUTOMA/CUTBCK", 0.25 )
168 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(2)/LOAD/STEPS/AUTOMA/MAXSTP", 500 )
169
170 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "OUTPUT(1)/SELTYP",
    "USER" )
171
172 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/DISPLA(2)/TOTAL/TRANSL/GLOBAL" )
173 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(1)/TOTAL/GREEN/GLOBAL" )
174 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(2)/EFFECT/CAUCHY/GLOBAL" )
175 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(3)/EFFECT/TRACTI/FRICTI" )
176 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(1)/TOTAL/CAUCHY/GLOBAL" )
177 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(4)/EFFECT/TRACTI/LOCAL" )
178 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(5)/TOTAL/SHEAR/LOCAL" )
179 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/METHOD/METNAM", "SECANT" )
180
181 #Run Analysis
182 # runSolver( [] )
183 # showView( "RESULT" )
```

```
184   fitAll( )
185
186
```

D3 - Bond-slip of the reinforcement

```
newProject( "D:/Afstuderen/Python/3D/DIANA Output/ 3D Pull out reinforcement", 10 )
setModelAnalysisAspects( [ "STRUCT" ] )
setModelDimension( "3D" )
setDefaultMeshOrder( "QUADRATIC" )
setDefaultMesherType( "HEXQUAD" )
setDefaultMidSideNodeLocation( "ONSHAP" )
setUnit( "LENGTH", "MM" )
setUnit( "FORCE", "N" )
setUnit( "ANGLE", "DEGREE" )

#Create Shapes
# createSheet( "Concrete", [[0, 0, 0],[1000, 0, 0],[1000, 1000, 0],[0, 1000, 0]] )

# extrudeProfile( [ "Concrete" ], [[ 0, 0, 1000 ]])

createBlock( "Pressure Layer", [ 0, 0, 70 ], [ 400, 400, 230 ] )
createPointBody( "point 1", [ 200, 200, 300 ] )

imprintIntersection( "Pressure Layer", "point 1", True )

removeShape( [ "point 1" ] )

#Create Reinforcement
createLine("Bar_M8", [ 200, 200, 70],[ 200, 200, 265] )

#Regroup Material Parts
addSet( "SHAPESET", "Reinforcement" )
moveToShapeSet( [ "Bar_M8" ], "Reinforcement" )
rename( "SHAPESET", "Shapes", "Concrete" )

#Material Precast concrete
addMaterial( "Precast concrete", "CONCR", "TSCR", [] )
setParameter( "MATERIAL", "Precast concrete", "LINEAR/ELASTI/YOUNG", 32000 )
setParameter( "MATERIAL", "Precast concrete", "LINEAR/ELASTI/POISON", 0.2 )
setParameter( "MATERIAL", "Precast concrete", "LINEAR/MASS/DENSIT", 2.4e-09 )
setParameter( "MATERIAL", "Precast concrete", "MODTYP/TOTCRK", "ROTATE" )
setParameter( "MATERIAL", "Precast concrete", "TENSIL/TENCRV", "HORDYK" )
setParameter( "MATERIAL", "Precast concrete", "TENSIL/TENSTR", 2.9)
setParameter( "MATERIAL", "Precast concrete", "TENSIL/GF1", 0.1405 )
setParameter( "MATERIAL", "Precast concrete", "TENSIL/RESTST", 0 )
setParameter( "MATERIAL", "Precast concrete", "TENSIL/CBSPEC", "ROTS" )
setParameter( "MATERIAL", "Precast concrete", "TENSIL/POISRE/POIRED", "DAMAGE" )
setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMCRV", "MC2010" )
setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMSTR", 38 )
setParameter( "MATERIAL", "Precast concrete", "COMPRS/EPSC1", 0.0023 )
setParameter( "MATERIAL", "Precast concrete", "COMPRS/EPSCU", 0.0035 )
setParameter( "MATERIAL", "Precast concrete", "COMPRS/CONFIN/CNFCRV", "NONE" )
# setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMCRV", "CONSTA" )
# setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMSTR", 38 )

addGeometry( "Precast concrete", "SOLID", "STRSOL", [] )
setParameter( "GEOMET", "Precast concrete", "LOCAXS", True )
setParameter( "GEOMET", "Precast concrete", "LOCAXS/XAXIS", [ 1, 0, 0 ] )
setParameter( "GEOMET", "Precast concrete", "LOCAXS/YAXIS", [ 0, 1, 0 ] )

addElementData( "Precast concrete" )
setElementClassType( "SHAPE", [ "Pressure Layer" ], "STRSOL" )
assignMaterial( "Precast concrete", "SHAPE", [ "Pressure Layer" ] )
assignGeometry( "Precast concrete", "SHAPE", [ "Pressure Layer" ] )
assignElementData( "Precast concrete", "SHAPE", [ "Pressure Layer"] )

#Material Reinfo_M8
# addMaterial( "Reinfo_M8", "REINFO", "REBOND", [ "ANCHOR" ] )
addMaterial( "Reinfo_M8", "REINFO", "REBOND", [] )
setParameter( "MATERIAL", "Reinfo_M8", "REBARS/ELASTI/YOUNG", 200000 )
setParameter( "MATERIAL", "Reinfo_M8", "REBARS/POISON/POISON", 0.3 )
setParameter( "MATERIAL", "Reinfo_M8", "REBARS/PLATYP", "VMISES" )
setParameter( "MATERIAL", "Reinfo_M8", "REBARS/PLASTI/TRESSH", "KAPSIG" )
setParameter( "MATERIAL", "Reinfo_M8", "REBARS/PLASTI/KAPSIG", [ 0, 550, 0.04725,
594 ] )
setParameter( "MATERIAL", "Reinfo_M8", "RESLIP/SHFTYP", "BONDS6" )
```

```
71  setParameter( "MATERIAL", "Reinfo_M8", "RESLIP/DSNY", 1000 )
72  setParameter( "MATERIAL", "Reinfo_M8", "RESLIP/DSSX", 100 )
73  # Plain surface good bond
74  setParameter( "MATERIAL", "Reinfo_M8", "RESLIP/BONDS6/SLPVAL", [ 0.63, 0.63, 0.01,
    0.02, 0.03, 0.04, 0.5 ] )
75  # Good bond condition
76  # setParameter( "MATERIAL", "Reinfo_M8", "RESLIP/BONDS6/SLPVAL", [ 15.411, 6.164,
    0.1, 1, 2, 10, 0.4 ] )
77  # Other bond conditions
78  # setParameter( "MATERIAL", "Reinfo_M8", "RESLIP/BONDS6/SLPVAL", [ 7.706, 3.082,
    0.1, 1.8, 3.6, 10, 0.4 ] )
79  # setParameter( "MATERIAL", "Reinfo_M8", "REANCH/DTIPNX", 10000 )
80  # setParameter( "MATERIAL", "Reinfo_M8", "REANCH/DTIPSY", 10000 )
81  # setParameter( "MATERIAL", "Reinfo_M8", "REANCH/TIPLOC", "END" )
82
83  addGeometry( "Reinfo_M8", "RELINE", "REBAR", [] )
84  setParameter( "GEOMET", "Reinfo_M8", "REITYP", "REITRU" )
85  setParameter( "GEOMET", "Reinfo_M8", "REITYP", "CIRBEA" )
86  setParameter( "GEOMET", "Reinfo_M8", "CIRBEA/CIRCLE", 30 )
87  setParameter( "GEOMET", "Reinfo_M8", "TIPELM/SURFAC", 706.9 )
88  assignMaterial( "Reinfo_M8", "SHAPE", [ "Bar_M8" ] )
89  assignGeometry( "Reinfo_M8", "SHAPE", [ "Bar_M8" ] )
90  convertToReinforcement( [ "Bar_M8" ] )
91  addElementData( "Bar" )
92  assignElementData( "Bar", "SHAPE", [ "Bar_M8" ] )
93  setParameter( "DATA", "Bar", "./INTERF", [] )
94  setParameter( "DATA", "Bar", "INTERF", "BEAM" )
95  setReinforcementDiscretization( [ "Bar_M8" ], "ELEMENT" )
96  setContinuousInInterfaces( [ "Bar_M8" ], True )
97
98  #Assign Supports
99  addSet( "GEOMETRYSUPPORTSET", "Support" )
100
101 createSurfaceSupport( "Top Support", "Support" )
102 setParameter( "GEOMETRYSUPPORT", "Top Support", "AXES", [ 1, 2 ] )
103 setParameter( "GEOMETRYSUPPORT", "Top Support", "TRANSL", [ 0, 0, 1 ] )
104 setParameter( "GEOMETRYSUPPORT", "Top Support", "ROTATI", [ 0, 0, 0 ] )
105 attach( "GEOMETRYSUPPORT", "Top Support", "Pressure Layer", [[ 200, 200, 3000 ]] )
106
107 createPointSupport( "Topp Support", "Support" )
108 setParameter( "GEOMETRYSUPPORT", "Topp Support", "AXES", [ 1, 2 ] )
109 setParameter( "GEOMETRYSUPPORT", "Topp Support", "TRANSL", [ 1, 1, 0 ] )
110 setParameter( "GEOMETRYSUPPORT", "Topp Support", "ROTATI", [ 0, 0, 0 ] )
111 attach( "GEOMETRYSUPPORT", "Topp Support", "Pressure Layer", [[ 200, 200, 300 ]] )
112
113 createSurfaceSupport( "Bot Support", "Support" )
114 setParameter( "GEOMETRYSUPPORT", "Bot Support", "AXES", [ 1, 2 ] )
115 setParameter( "GEOMETRYSUPPORT", "Bot Support", "TRANSL", [ 0, 0, 1 ] )
116 setParameter( "GEOMETRYSUPPORT", "Bot Support", "ROTATI", [ 0, 0, 0 ] )
117 attach( "GEOMETRYSUPPORT", "Bot Support", "Pressure Layer", [[200,200,70]] )
118
119 createPointSupport( "Botp Support", "Support" )
120 setParameter( "GEOMETRYSUPPORT", "Botp Support", "AXES", [ 1, 2 ] )
121 setParameter( "GEOMETRYSUPPORT", "Botp Support", "TRANSL", [ 0, 0, 1 ] )
122 setParameter( "GEOMETRYSUPPORT", "Botp Support", "ROTATI", [ 0, 0, 0 ] )
123 attach( "GEOMETRYSUPPORT", "Botp Support", "Bar_M8", [[ 200, 200, 70 ]] )
124
125 #Assign Loads
126 addSet( "GEOMETRYLOADSET", "Load" )
127
128 # createLineLoad( "Load", "Load" )
129 # setParameter( "GEOMETRYLOAD", "Load", "LODTYP", "DEFORM" )
130 # setParameter( "GEOMETRYLOAD", "Load", "DEFORM/TR/VALUE", 0.1 )
131 # setParameter( "GEOMETRYLOAD", "Load", "DEFORM/TR/DIRECT", 1 )
132 # attach( "GEOMETRYLOAD", "Load", "Pressure Layer", [[ 200, 100, 250 ]] )
133
134 createPointLoad( "Load", "Load" )
135 setParameter( "GEOMETRYLOAD", "Load", "LODTYP", "DEFORM" )
136 setParameter( "GEOMETRYLOAD", "Load", "DEFORM/SUPP", "Botp Support" )
137 setParameter( "GEOMETRYLOAD", "Load", "DEFORM/TR/VALUE", -10 )
138 setParameter( "GEOMETRYLOAD", "Load", "DEFORM/TR/DIRECT", 3 )
```

```
attach( "GEOMETRYLOAD", "Load", "Bar_M8", [[ 200, 200, 70 ]] )

#Set mesh properties
setElementSize( [ "Pressure Layer","Bar_M8" ], 33.75, -1, True )
setMesherType( [ "Pressure Layer","Bar_M8" ], "HEXQUAD" )
clearMidSideNodeLocation( [  "Pressure Layer","Bar_M8" ] )
generateMesh( [] )
hideView( "GEOM" )
showView( "MESH" )

#Analysis
addAnalysis( "Analysis1" )
addAnalysisCommand( "Analysis1", "NONLIN", "Structural nonlinear" )
renameAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "EXECUT(1)",
"Load" )
# setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/STEPS/STEPTY", "AUTOMA" )
# setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/STEPS/AUTOMA/SIZES", 1 )
# setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/STEPS/AUTOMA/MAXSIZ", 0.05 )
# setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/STEPS/AUTOMA/MINSIZ", 0.005 )
# setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/STEPS/AUTOMA/CUTBCK", 0.25 )
setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/STEPS/EXPLIC/SIZES", "0.01(100)" )
# setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN", False )
# setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN/REGULA/SET(1)/NODES(1)/RNGNRS", "ALL" )
# setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN/REGULA/SET(1)/DIRECT", 2 )

# setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/ITERAT/MAXITE", 30 )
# addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/ITERAT/LINESE" )
# setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/ITERAT/LINESE", True )
# setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/ITERAT/LINESE/ETAMIN", 0.1 )
# setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/ITERAT/LINESE/ETAMAX", 1 )
# 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", 0.01 )
# 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",
"EXECUT(1)/ITERAT/METHOD/METNAM", "SECANT" )

# setAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "OUTPUT(1)/SELTYP",
"USER" )

# addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"OUTPUT(1)/USER/DISPLA(1)/TOTAL/TRANSL/GLOBAL" )
# setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"OUTPUT(1)/USER/STRAIN(1)/TOTAL/GREEN/LOCAL/LOCATI", "INTPNT" )
# setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"OUTPUT(1)/USER/STRESS(1)/TOTAL/CAUCHY/LOCAL/LOCATI", "INTPNT" )

# # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"OUTPUT(1)/USER/STRAIN(1)/TOTAL/GREEN/GLOBAL" )
```

```
183  # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
     "OUTPUT(1)/USER/STRAIN(2)/TOTAL/GREEN/PRINCI" )
184  # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
     "OUTPUT(1)/USER/STRAIN(4)/CRACK/GREEN" )
185  # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
     "OUTPUT(1)/USER/STRAIN(7)/CRKWDT/GREEN/GLOBAL" )
186  # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
     "OUTPUT(1)/USER/STRAIN(8)/CRKWDT/GREEN/PRINCI" )
187  # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
     "OUTPUT(1)/USER/STRESS(3)/CRACK/CAUCHY/LOCAL" )
188  # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
     "OUTPUT(1)/USER/STRESS(2)/TOTAL/CAUCHY/PRINCI" )
189
190  #Run Analysis
191  # runSolver( [] )
192  # showView( "RESULT" )
193  fitAll( )
194
195
```

E1 - Single headed anchor model

```
newProject( "D:/Afstuderen/Python/3D/DIANA Output/ Anchor depth 100mm", 10 )
setModelAnalysisAspects( [ "STRUCT" ] )
setModelDimension( "3D" )
setDefaultMeshOrder( "QUADRATIC" )
setDefaultMesherType( "HEXQUAD" )
setDefaultMidSideNodeLocation( "ONSHAP" )
setUnit( "LENGTH", "MM" )
setUnit( "FORCE", "N" )
setUnit( "ANGLE", "DEGREE" )

#Parameters
X1 = 200
X2 = 400
LX = X2-X1
Y1 = 0
Y2 = 200
LY = Y2-Y1
XM = X1+(X2-X1)/2
YM = Y1+(Y2-Y1)/2

#Thickness layers
T2 = 100
T1 = 150-T2
T3 = 180

T1M = T1/2
T2M = (T2/2)+T1
T3M = (T3/2)+T1+T2

#Anchor size
W1 = 10
H1 = 6
WX = XM-(W1/2)
WY = YM-(W1/2)
WZ = T1-H1
R1 = W1/2

#Create Shapes
createBlock( "SlabBottom", [ X1, Y1, 0 ], [ LX, LY, T1 ] )
createBlock( "SlabTop", [ X1, Y1, T1 ], [ LX, LY, T2 ] )
createBlock( "Pressure layer", [ X1, Y1, T1+T2 ], [ LX, LY, T3 ] )

#Create Reinforcement
createLine( "Koppel_M12a", [ X1, 30, T1+T2+6 ], [ X2, 30, T1+T2+6 ] )
createLine( "Koppel_M12b", [ X1, 180, T1+T2+6 ], [ X2, 180, T1+T2+6 ] )

createLine( "Reinfo_M8a", [ X1, 2.5, 35 ], [ X2, 2.5, 35 ] )
createLine( "Reinfo_M8b", [ X1, 67.5, 35 ], [ X2, 67.5, 35 ] )
createLine( "Reinfo_M8c", [ X1, 132.5, 35 ], [ X2, 132.5, 35 ] )
createLine( "Reinfo_M8d", [ X1, 197.5, 35 ], [ X2, 197.5, 35 ] )

createPolyline( "LG1-mb", [[ XM, 0, T1+T2+T3-84.5 ],[ XM, YM, T1+T2+T3-84.5 ],[ XM, Y2, T1+T2+T3-84.5 ]], False )
createLine( "LG1", [ XM, YM, T1],[ XM, YM, T1+T2+T3-84.5 ])
createBlock( "Anchor", [ WX, WY, WZ], [W1, W1, H1])
# createCylinder( "Anchor", [ XM, YM, WZ], [ 0, 0, 1 ], R1, H1 )
createPointBody( "point 1", [ XM, YM, T1 ] )

imprintIntersection( "Anchor", "point 1", True )
removeShape( [ "point 1" ] )

imprintIntersection( "SlabBottom", "Anchor", True )
imprintIntersection( "SlabTop", "Anchor", True )

#Regroup Material Parts
addSet( "GEOMETRYREINFOSET", "Coupling Reinforcement" )
moveToShapeSet( [ "Koppel_M12a","Koppel_M12b" ], "Coupling Reinforcement" )
addSet( "GEOMETRYREINFOSET", "Longitudinal Reinforcement" )
moveToShapeSet( [ "Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d"], "Longitudinal Reinforcement" )
addSet( "GEOMETRYREINFOSET", "Lattice girders" )
```

```
moveToShapeSet( [ "LG1-mb", "LG1"], "Lattice girders" )
addSet( "SHAPESET", "Steel" )
moveToShapeSet( [ "Anchor"], "Steel" )
rename( "SHAPESET", "Shapes", "Concrete" )

#Material Precast concrete
addMaterial( "Precast concrete", "CONCR", "TSCR", [] )
setParameter( "MATERIAL", "Precast concrete", "LINEAR/ELASTI/YOUNG", 32000 )
setParameter( "MATERIAL", "Precast concrete", "LINEAR/ELASTI/POISON", 0.2 )
setParameter( "MATERIAL", "Precast concrete", "LINEAR/MASS/DENSIT", 2.4e-09 )
setParameter( "MATERIAL", "Precast concrete", "MODTYP/TOTCRK", "ROTATE" )
setParameter( "MATERIAL", "Precast concrete", "TENSIL/TENCRV", "HORDYK" )
setParameter( "MATERIAL", "Precast concrete", "TENSIL/TENSTR", 2.9 )
setParameter( "MATERIAL", "Precast concrete", "TENSIL/GF1", 0.1405 )
setParameter( "MATERIAL", "Precast concrete", "TENSIL/CBSPEC", "ROTS" )
setParameter( "MATERIAL", "Precast concrete", "TENSIL/POISRE/POIRED", "DAMAGE" )
setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMCRV", "MC2010" )
setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMSTR", 38 )
setParameter( "MATERIAL", "Precast concrete", "COMPRS/EPSC1", 0.0023 )
setParameter( "MATERIAL", "Precast concrete", "COMPRS/EPSCU", 0.0035 )
setParameter( "MATERIAL", "Precast concrete", "COMPRS/CONFIN/CNFCRV", "NONE" )
# setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMCRV", "CONSTA" )
# setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMSTR", 38 )

addGeometry( "Precast concrete", "SOLID", "STRSOL", [] )
setParameter( "GEOMET", "Precast concrete", "LOCAXS", True )
setParameter( "GEOMET", "Precast concrete", "LOCAXS/XAXIS", [ 1, 0, 0 ] )
setParameter( "GEOMET", "Precast concrete", "LOCAXS/YAXIS", [ 0, 1, 0 ] )

setElementClassType( "SHAPE", [ "SlabBottom", "SlabTop" ], "STRSOL" )
assignMaterial( "Precast concrete", "SHAPE", [ "SlabBottom", "SlabTop" ] )
assignGeometry( "Precast concrete", "SHAPE", [ "SlabBottom", "SlabTop" ] )

#Material Pressure layer
addMaterial( "Pressure layer", "CONCR", "TSCR", [] )
setParameter( "MATERIAL", "Pressure layer", "LINEAR/ELASTI/YOUNG", 32000 )
setParameter( "MATERIAL", "Pressure layer", "LINEAR/ELASTI/POISON", 0.2 )
setParameter( "MATERIAL", "Pressure layer", "LINEAR/MASS/DENSIT", 2.4e-09 )
setParameter( "MATERIAL", "Pressure layer", "MODTYP/TOTCRK", "ROTATE" )
setParameter( "MATERIAL", "Pressure layer", "TENSIL/TENCRV", "HORDYK" )
setParameter( "MATERIAL", "Pressure layer", "TENSIL/TENSTR", 2.9 )
setParameter( "MATERIAL", "Pressure layer", "TENSIL/GF1", 0.1405 )
setParameter( "MATERIAL", "Pressure layer", "TENSIL/CBSPEC", "ROTS" )
setParameter( "MATERIAL", "Pressure layer", "TENSIL/POISRE/POIRED", "DAMAGE" )
setParameter( "MATERIAL", "Pressure layer", "COMPRS/COMCRV", "MC2010" )
setParameter( "MATERIAL", "Pressure layer", "COMPRS/COMSTR", 38 )
setParameter( "MATERIAL", "Pressure layer", "COMPRS/EPSC1", 0.0023 )
setParameter( "MATERIAL", "Pressure layer", "COMPRS/EPSCU", 0.0035 )
setParameter( "MATERIAL", "Pressure layer", "COMPRS/CONFIN/CNFCRV", "NONE" )
#setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMCRV", "CONSTA" )
#setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMSTR", 38 )

addGeometry( "Pressure layer", "SOLID", "STRSOL", [] )
setParameter( "GEOMET", "Pressure layer", "LOCAXS", True )
setParameter( "GEOMET", "Pressure layer", "LOCAXS/XAXIS", [ 1, 0, 0 ] )
setParameter( "GEOMET", "Pressure layer", "LOCAXS/YAXIS", [ 0, 1, 0 ] )

setElementClassType( "SHAPE", [ "Pressure layer" ], "STRSOL" )
assignMaterial( "Precast concrete", "SHAPE", [ "Pressure layer"] )
assignGeometry( "Precast concrete", "SHAPE", [ "Pressure layer"] )

#Material Steel
addMaterial( "Steel", "MCSTEL", "ISOTRO", [] )
setParameter( "MATERIAL", "Steel", "LINEAR/ELASTI/YOUNG", 200000 )
setParameter( "MATERIAL", "Steel", "LINEAR/ELASTI/POISON", 0.3 )
setParameter( "MATERIAL", "Steel", "LINEAR/MASS/DENSIT", 0 )
addGeometry( "Anchor", "SOLID", "STRSOL", [] )
addElementData( "Anchor" )

setElementClassType( "SHAPE", [ "Anchor" ], "STRSOL" )
assignMaterial( "Steel", "SHAPE", [ "Anchor" ] )
```

```
assignGeometry( "Anchor", "SHAPE", [ "Anchor" ] )
assignElementData( "Anchor", "SHAPE", [ "Anchor" ] )

#Concrete Interface
addMaterial( "Interface Concrete", "INTERF", "FRICTI", [] )
setParameter( "MATERIAL", "Interface Concrete", "LINEAR/ELAS6/DSNZ", 60000 )
setParameter( "MATERIAL", "Interface Concrete", "LINEAR/ELAS6/DSSX", 6000 )
setParameter( "MATERIAL", "Interface Concrete", "LINEAR/ELAS6/DSSY", 6000 )
setParameter( "MATERIAL", "Interface Concrete", "COULOM/OPNTYP", "GAP" )
setParameter( "MATERIAL", "Interface Concrete", "COULOM/GAPVAL", 0.5 )
setParameter( "MATERIAL", "Interface Concrete", "COULOM/COHESI", 0.5 )
setParameter( "MATERIAL", "Interface Concrete", "COULOM/PHI", 30 )
setParameter( "MATERIAL", "Interface Concrete", "COULOM/PSI", 30 )
setParameter( "MATERIAL", "Interface Concrete", "COULOM/FRCHSR/MODE2", 0 )


#addMaterial( "Interface no tension", "INTERF", "NONLIF", [] )
#setParameter( "MATERIAL", "Interface no tension", "LINEAR/ELAS6/DSSY", 10000 )
#setParameter( "MATERIAL", "Interface no tension", "LINEAR/ELAS6/DSSX", 10000 )
#setParameter( "MATERIAL", "Interface no tension", "LINEAR/ELAS6/DSNZ", 100000 )

addGeometry( "Interface Concrete", "SHEET", "STPLIF", [] )
setParameter( "GEOMET", "Interface Concrete", "LOCAXS", True )
setParameter( "GEOMET", "Interface Concrete", "LOCAXS/XAXIS", [ 1, 0, 0 ] )

addElementData( "Interface Concrete" )
createConnection( "Interface Concrete", "INTER", "SHAPEFACE", "SHAPEFACE" )
setParameter( "GEOMETRYCONNECTION", "Interface Concrete", "MODE", "CLOSED" )
setElementClassType( "GEOMETRYCONNECTION", "Interface Concrete", "STPLIF" )
assignMaterial( "Interface Concrete", "GEOMETRYCONNECTION", "Interface Concrete" )
#assignMaterial( "Interface no tension", "GEOMETRYCONNECTION", "Interface Concrete" )
assignGeometry( "Interface Concrete", "GEOMETRYCONNECTION", "Interface Concrete" )
assignElementData( "Interface Concrete", "GEOMETRYCONNECTION", "Interface Concrete" )
setParameter( "GEOMETRYCONNECTION", "Interface Concrete", "FLIP", False )
attachTo( "GEOMETRYCONNECTION", "Interface Concrete", "SOURCE", "SlabTop", [ [ XM, YM, T1+T2 ] ] )
attachTo( "GEOMETRYCONNECTION", "Interface Concrete", "TARGET", "Pressure layer", [ [ XM, YM, T1+T2 ] ] )

#Material Reinforcement Koppel_M12
addMaterial( "Koppel_M12", "REINFO", "REBOND", [ ] )
setParameter( "MATERIAL", "Koppel_M12", "REBARS/ELASTI/YOUNG", 200000 )
setParameter( "MATERIAL", "Koppel_M12", "REBARS/POISON/POISON", 0.3 )
setParameter( "MATERIAL", "Koppel_M12", "REBARS/PLATYP", "VMISES" )
setParameter( "MATERIAL", "Koppel_M12", "REBARS/PLASTI/TRESSH", "KAPSIG" )
setParameter( "MATERIAL", "Koppel_M12", "REBARS/PLASTI/KAPSIG", [ 0, 550, 0.04725, 594 ] )
setParameter( "MATERIAL", "Koppel_M12", "RESLIP/SHFTYP", "BONDS6" )
setParameter( "MATERIAL", "Koppel_M12", "RESLIP/DSNY", 100000 )
setParameter( "MATERIAL", "Koppel_M12", "RESLIP/DSSX", 10000 )
setParameter( "MATERIAL", "Koppel_M12", "RESLIP/BONDS6/SLPVAL", [ 15.41, 6.16, 0.1, 1, 2, 10, 0.4 ] )
addGeometry( "Koppel_M12", "RELINE", "REBAR", [] )
setParameter( "GEOMET", "Koppel_M12", "REITYP", "REITRU" )
setParameter( "GEOMET", "Koppel_M12", "REITYP", "CIRBEA" )
setParameter( "GEOMET", "Koppel_M12", "CIRBEA/CIRCLE", 12)
setParameter( "GEOMET", "Koppel_M12", "ORIENT/ZAXIS", [ 0, 0, 1 ] )
setParameter( "GEOMET", "Koppel_M12", "TIPELM/SURFAC", 113.1 )
#assignMaterial( "Koppel_M12", "SHAPE", [ "Koppel_M12a","Koppel_M12b" ] )
#assignGeometry( "Koppel_M12", "SHAPE", [ "Koppel_M12a","Koppel_M12b" ] )
#convertToReinforcement( [ "Koppel_M12a","Koppel_M12b" ] )
#addElementData( "Koppel_M12" )
#assignElementData( "Koppel_M12", "SHAPE", [ "Koppel_M12a","Koppel_M12b" ] )
#setParameter( "DATA", "Koppel_M12", "INTERF", "BEAM" )
#setReinforcementDiscretization( [ "Koppel_M12a","Koppel_M12b" ], "ELEMENT" )
#setContinuousInInterfaces( [ "Koppel_M12a","Koppel_M12b" ], False )
setReinforcementType( "GEOMETRYREINFOSET", "Coupling Reinforcement", "BEAM_NUMERICAL_BOND_SLIP" )
setContinuousInInterfaces( "GEOMETRYREINFOSET", "Coupling Reinforcement", False )
assignMaterial( "Koppel_M12", "GEOMETRYREINFOSET", [ "Coupling Reinforcement"])
assignGeometry( "Koppel_M12", "GEOMETRYREINFOSET", [ "Coupling Reinforcement"])
```

```
207
208 #Material Reinforcement M8
209 addMaterial( "Reinfo_M8", "REINFO", "REBOND", [] )
210 setParameter( "MATERIAL", "Reinfo_M8", "REBARS/ELASTI/YOUNG", 200000 )
211 setParameter( "MATERIAL", "Reinfo_M8", "REBARS/POISON/POISON", 0.3 )
212 setParameter( "MATERIAL", "Reinfo_M8", "REBARS/PLATYP", "VMISES" )
213 setParameter( "MATERIAL", "Reinfo_M8", "REBARS/PLASTI/TRESSH", "KAPSIG" )
214 setParameter( "MATERIAL", "Reinfo_M8", "REBARS/PLASTI/KAPSIG", [ 0, 550, 0.04725,
    594 ] )
215 setParameter( "MATERIAL", "Reinfo_M8", "RESLIP/SHFTYP", "BONDS6" )
216 setParameter( "MATERIAL", "Reinfo_M8", "RESLIP/DSNY", 100000 )
217 setParameter( "MATERIAL", "Reinfo_M8", "RESLIP/DSSX", 10000 )
218 setParameter( "MATERIAL", "Reinfo_M8", "RESLIP/BONDS6/SLPVAL", [ 15.41, 6.16, 0.1,
    1, 2, 10, 0.4 ] )
219 addGeometry( "Reinfo_M8", "RELINE", "REBAR", [] )
220 setParameter( "GEOMET", "Reinfo_M8", "REITYP", "REITRU" )
221 setParameter( "GEOMET", "Reinfo_M8", "REITYP", "CIRBEA" )
222 setParameter( "GEOMET", "Reinfo_M8", "CIRBEA/CIRCLE", 8)
223 setParameter( "GEOMET", "Reinfo_M8", "ORIENT/ZAXIS", [ 0, 0, 1 ] )
224 setParameter( "GEOMET", "Reinfo_M8", "TIPELM/SURFAC", 50.3 )
225 #assignMaterial( "Reinfo_M8", "SHAPE", [
    "Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d" ] )
226 #assignGeometry( "Reinfo_M8", "SHAPE", [
    "Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d" ] )
227 #convertToReinforcement( [ "Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d"] )
228 #addElementData( "Reinfo_M8" )
229 #assignElementData( "Reinfo_M8", "SHAPE", [
    "Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d"] )
230 #setParameter( "DATA", "Reinfo_M8", "INTERF", "BEAM" )
231 #setReinforcementDiscretization( [
    "Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d" ], "ELEMENT" )
232 #setContinuousInInterfaces( [ "Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d" ],
    False )
233 setReinforcementType( "GEOMETRYREINFOSET", "Longitudinal Reinforcement",
    "BEAM_NUMERICAL_BOND_SLIP" )
234 setContinuousInInterfaces( "GEOMETRYREINFOSET", "Longitudinal Reinforcement", False )
235 assignMaterial( "Reinfo_M8", "GEOMETRYREINFOSET", [ "Longitudinal Reinforcement"])
236 assignGeometry( "Reinfo_M8", "GEOMETRYREINFOSET", [ "Longitudinal Reinforcement"])
237
238 #Material Reinforcement M5
239 addMaterial( "Reinfo_M5", "REINFO", "REBOND", [] )
240 setParameter( "MATERIAL", "Reinfo_M5", "REBARS/ELASTI/YOUNG", 200000 )
241 setParameter( "MATERIAL", "Reinfo_M5", "REBARS/POISON/POISON", 0.3 )
242 setParameter( "MATERIAL", "Reinfo_M5", "REBARS/PLATYP", "VMISES" )
243 setParameter( "MATERIAL", "Reinfo_M5", "REBARS/PLASTI/TRESSH", "KAPSIG" )
244 setParameter( "MATERIAL", "Reinfo_M5", "REBARS/PLASTI/KAPSIG", [ 0, 550, 0.04725,
    594 ] )
245 setParameter( "MATERIAL", "Reinfo_M5", "RESLIP/SHFTYP", "BONDS6" )
246 setParameter( "MATERIAL", "Reinfo_M5", "RESLIP/DSNY", 100000 )
247 setParameter( "MATERIAL", "Reinfo_M5", "RESLIP/DSSX", 10000 )
248 setParameter( "MATERIAL", "Reinfo_M5", "RESLIP/BONDS6/SLPVAL", [ 0.63, 0.63, 0.01,
    0.02, 0.03, 0.04, 0.5 ] )
249 addGeometry( "Reinfo_M5", "RELINE", "REBAR", [] )
250 setParameter( "GEOMET", "Reinfo_M5", "REITYP", "REITRU" )
251 setParameter( "GEOMET", "Reinfo_M5", "REITYP", "CIRBEA" )
252 setParameter( "GEOMET", "Reinfo_M5", "CIRBEA/CIRCLE", 8 )
253 # setParameter( "GEOMET", "Reinfo_M5", "ORIENT/ZAXIS", [ 0, 0, 1 ] )
254 setParameter( "GEOMET", "Reinfo_M5", "TIPELM/SURFAC", 50.3 )
255
256 setReinforcementType( "GEOMETRYREINFOSET", "Lattice girders",
    "BEAM_NUMERICAL_BOND_SLIP" )
257 setContinuousInInterfaces( "GEOMETRYREINFOSET", "Lattice girders", True )
258 assignMaterial( "Reinfo_M5", "GEOMETRYREINFOSET", [ "Lattice girders" ] )
259 assignGeometry( "Reinfo_M5", "GEOMETRYREINFOSET", [ "Lattice girders" ] )
260
261 # setParameter( "DATA", "Bar", "NUMINT", [ "GAUSS" ] )
262 # setParameter( "DATA", "Bar", "INTEGR", "HIGH" )
263
264 #Composed Line
265 #addGeometry( "Composed Line", "LINE", "COMLIN", [] )
266 #setParameter( "GEOMET", "Composed Line", "DISTAN/THICK", 250 )
```

```
#setElementClassType( "SHAPE", [ "Composed Line" ], "COMLIN" )
#assignGeometry( "Composed Line", "SHAPE", [ "Composed Line" ] )
#resetElementData( "SHAPE", [ "Composed Line" ] )

#Tyings
createPointTying( "Tying Top", "Tying" )
setParameter( "GEOMETRYTYING", "Tying Top", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYTYING", "Tying Top", "TRANSL", [ 1, 1, 1 ] )
setParameter( "GEOMETRYTYING", "Tying Top", "ROTATI", [ 1, 1, 1 ] )
attachTo( "GEOMETRYTYING", "Tying Top", "SLAVE", "LG1", [ [ XM, YM, T1+T2+T3-84.5 ] ] )
attachTo( "GEOMETRYTYING", "Tying Top", "MASTER", "LG1-mb", [ [ XM, YM, T1+T2+T3-84.5 ] ] )

createPointTying( "Tying Bot", "Tying" )
setParameter( "GEOMETRYTYING", "Tying Bot", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYTYING", "Tying Bot", "TRANSL", [ 1, 1, 1 ] )
setParameter( "GEOMETRYTYING", "Tying Bot", "ROTATI", [ 1, 1, 1 ] )
attachTo( "GEOMETRYTYING", "Tying Bot", "SLAVE", "Anchor", [ [ XM, YM, T1 ] ] )
attachTo( "GEOMETRYTYING", "Tying Bot", "MASTER", "LG1", [ [ XM, YM, T1 ] ] )

#Assign Supports
addSet( "GEOMETRYSUPPORTSET", "Supports" )

createSurfaceSupport( "Support X", "Supports" )
setParameter( "GEOMETRYSUPPORT", "Support X", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Support X", "TRANSL", [ 1, 0, 0 ] )
setParameter( "GEOMETRYSUPPORT", "Support X", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Support X", "SlabBottom", [ [ X1, YM, T1M ], [ X2, YM, T1M ] ] )
attach( "GEOMETRYSUPPORT", "Support X", "SlabTop", [ [ X1, YM, T2M ], [ X2, YM, T2M ] ] )
attach( "GEOMETRYSUPPORT", "Support X", "Pressure layer", [ [ X1, YM, T3M ], [ X2, YM, T3M ] ] )

createSurfaceSupport( "Support Y", "Supports" )
setParameter( "GEOMETRYSUPPORT", "Support Y", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Support Y", "TRANSL", [ 0, 1, 0 ] )
setParameter( "GEOMETRYSUPPORT", "Support Y", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Support Y", "SlabBottom", [ [ XM, Y1, T1M ], [ XM, Y2, T1M ] ] )
attach( "GEOMETRYSUPPORT", "Support Y", "SlabTop", [ [ XM, Y1, T2M ], [ XM, Y2, T2M ] ] )
attach( "GEOMETRYSUPPORT", "Support Y", "Pressure layer", [ [ XM, Y1, T3M ], [ XM, Y2, T3M ] ] )

createSurfaceSupport( "Support Z", "Supports" )
setParameter( "GEOMETRYSUPPORT", "Support Z", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Support Z", "TRANSL", [ 0, 0, 1 ] )
setParameter( "GEOMETRYSUPPORT", "Support Z", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Support Z", "Pressure layer", [ [ XM, YM, T1+T2+T3 ] ] )

createSurfaceSupport( "Deform", "Supports" )
setParameter( "GEOMETRYSUPPORT", "Deform", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Deform", "TRANSL", [ 0, 0, 1 ] )
setParameter( "GEOMETRYSUPPORT", "Deform", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Deform", "SlabBottom", [ [ XM, YM, 0 ] ] )

createPointSupport( "Support Y beam", "Supports" )
setParameter( "GEOMETRYSUPPORT", "Support Y beam", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Support Y beam", "TRANSL", [ 0, 1, 0 ] )
setParameter( "GEOMETRYSUPPORT", "Support Y beam", "ROTATI", [ 1, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Support Y beam", "LG1-mb", [ [ XM, Y1, T1+T2+T3-84.5 ], [ XM, Y2, T1+T2+T3-84.5 ] ] )

#Assign Loads
addSet( "GEOMETRYLOADSET", "Load" )

# createLineLoad( "Load", "Load" )
# setParameter( "GEOMETRYLOAD", "Load", "LODTYP", "DEFORM" )
# setParameter( "GEOMETRYLOAD", "Load", "DEFORM/TR/VALUE", 0.1 )
```

```
# setParameter( "GEOMETRYLOAD", "Load", "DEFORM/TR/DIRECT", 1 )
# attach( "GEOMETRYLOAD", "Load", "Pressure Layer", [[ 200, 100, 250 ]] )

createSurfaceLoad( "Load", "Load" )
setParameter( "GEOMETRYLOAD", "Load", "LODTYP", "DEFORM" )
setParameter( "GEOMETRYLOAD", "Load", "DEFORM/TR/VALUE", -0.1 )
setParameter( "GEOMETRYLOAD", "Load", "DEFORM/TR/DIRECT", 3 )
attach( "GEOMETRYLOAD", "Load", "SlabBottom", [ [ XM, YM, 0 ] ] )

#Set mesh properties
setElementSize( [ "Anchor" ], 4, -1, True )
clearMesherType( [ "Anchor" ] )
setElementSize( [ "SlabTop" ], 10, -1, True )
clearMesherType( [ "SlabTop" ] )
setElementSize( [ "Pressure layer", "SlabBottom" ], 20, -1, True )
clearMesherType( [ "Pressure layer", "SlabBottom" ] )
generateMesh( [] )
hideView( "GEOM" )
showView( "MESH" )

#Analysis
addAnalysis( "Analysis1" )
addAnalysisCommand( "Analysis1", "NONLIN", "Structural nonlinear" )
renameAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "EXECUT(1)",
"Load" )
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/STEPTY", "AUTOMA" )
# setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/STEPS/AUTOMA/SIZES", 250 )
# setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/STEPS/AUTOMA/MAXSIZ", 0.01 )
# setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/STEPS/AUTOMA/MINSIZ", 0.00001 )
# setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/STEPS/AUTOMA/CUTBCK", 0.25 )
# setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/STEPS/AUTOMA/MAXSTP", 100 )
# setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/STEPS/AUTOMA/ARCLEN", True )
# addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/STEPS/AUTOMA/ARCLEN/REGULA/SET" )
# setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/STEPS/AUTOMA/ARCLEN/REGULA/SET(1)/NODES(1)/RNGNRS", "1-9361" )
# setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/STEPS/AUTOMA/ARCLEN/REGULA/SET(1)/DIRECT", 2 )
setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/STEPS/EXPLIC/SIZES", "0.2(15)" )
# setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN", True )
# addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN/REGULA/SET" )
# setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN/REGULA/SET(1)/NODES(1)/RNGNRS", "3 4 9 10 41-96
237-242 349-354 569-2000(53) 2001-2113 2133-2653(20) 3541-9848(53) 9992 10047-10109
10163 10202 10223-10227 10281-10283 10323 10324 10364 10365 10405 10406 10446 10447
10487 10488 10528 10529 10569 10570 10610 10611 10651 10652 10692 10693 10733 10734
10774 10775 10815 10816 10856 10857 10897 10898 10938 10939 10979 10980 11020 11021
11061 11062 11102 11103 11143 11144 11184 11185 11225 11226 11266 11267 11307 11308
11348 11349 11389 11390 11430 11592-11847 12237 12239 12344 12346 12451 12453 12558
12560 12665 12667 12772 12774 12879 12881 12986 12988 13093 13095 13200 13202 13307
13309 13414 13416 13521 13523 13628 13630 13735 13737 13842 13844 13949 13951 14056
14058 14163 14165 14270 14272 14377 14379 14484 14486 14591 14593 14698 14700 14805
14807 14912 14914 15019 15021 15126 15128 15181-15183 15289 15290 15396 15397 15503
15504 15610 15663 15664 15770 15771 15877 15878 15984 15985 16091 17825-17827
17985-17987 18145-18147 18305-18307 18465 18518-18520 18678-18680 18838-18840
18998-19000 19158 19211-19213 19371-19373 19531-19533 19691-19693 19851 19904-19906
20064-20066 20224-20226 20384-20386 20544 20597-20599 20757-20759 20917-20919
```

```
    21077-21079 21237 21290-21292 21450-21452 21610-21612 21770-21772 21930 21983-21985
    22143-22145 22303-22305 22463-22465 22623 22676-22678 22836-22838 22996-22998
    23156-23158 23316 23369-23371 23529-23531 23689-23691 23849-23851 24009 24062-24064
    24222-24224 24382-24384 24542-24544 24702 24755-24757 24915-24917 25075-25077
    25235-25237 25395 25448-25450 25608-25610 25768-25770 25928-25930 26088 26141-26143
    26301-26303 26461-26463 26621-26623 26781 26834-26836 26994-26996 27154-27156
    27314-27316 27474 27527-27529 27687-27689 27847-27849 28007-28009 28167 28220-28222
    28380-28382 28540-28542 28700-28702 28860 28913-28915 29073-29075 29233-29235
    29393-29395 29553 29606-29608 29766-29768 29926-29928 30086-30088 30246 30299-30301
    30459-30461 30619-30621 30779-30781 30939 30992-30994 31152-31154 31312-31314
    31472-31474 31632 31685-31687 31845-31847 32005-32007 32165-32167 32325 32378-32380
    32538-32540 32698-32700 32858-32860 33018 33071-33073 33231-33233 33391-33393
    33551-33553 33711 33764-33766 33924-33926 34084-34086 34244-34246 34404 34457-34459
    34617-34619 34777-34779 34937-34939 35097 35150-35152 35310-35312 35470-35472
    35630-35632 35790 35843-35845 36003-36005 36163-36165 36323-36325 36483 36536-36538
    36696-36698 36856-36858 37016-37018 37176-37441(53) 37443 37445-37472 37501 37554
    37659-39090(53) 39091 39095 39183-39210 39262 39263 39315 39316 39611-40772(43)
    40773-40801 40854-42338(53) 42420-42449 42503 42504 42548 42551-42579 42650 42704
    42705 42749 42844 42846 42931 42933 43018 43020 43105 43107 43192 43194 43279 43281
    43366 43368 43453 43455 43540 43542 43627 43629 43714 43716 43801 43803 43888 43890
    43975 43977 44062 44064 44149 44151 44236 44238 44323 44325 44410 44412 44497 44499
    44584 44586 44671 44673 44758 44760 44845 44847 44932 44934 45019 45021 45106 45108
    45193 45195 45238-45323 45950 45951 46057 46110 46111 46217 46374 46376 46481 46483
    46588 46590 46695 46697 46802 46804 46909 46911 47016 47018 47123 47125 47230 47232
    47337 47339 47444 47446 47551 47553 47658 47660 47765 47767 47872 47874 47979 47981
    48086 48088 48193 48195 48300 48302 48407 48409 48514 48516 48621 48623 48728 48730
    48835 48837 48942 48944 49049 49051 49156 49158 49263 49265 49318-49321 49479
    49532-49534 49692 49745-49747 49905 49958-49960 50118 50171-50173 50331 50384-50386
    50544 50597-50599 50757 50810-50812 50970 51023-51025 51183 51236-51238 51396
    51449-51451 51609 51662-51664 51822 51875-51877 52035 52088-52090 52248 52301-52303
    52461 52514-52516 52674 52727-52729 52887 52940-52942 53100 53153-53155 53313
    53366-53368 53526 53579-53581 53739 53792-53794 53952 54005-54007 54165 54218-54220
    54378 54431-54433 54591 54644-54646 54804 54857-54859 55017 55070-55072 55230 55283
    57181 57228-57230 57269-57271 57310-57312 57351-57353 57392-57394 57433-57435
    57474-57476 57583-57585 57692-57694 57801-57803 57910-57912 58019-58021 58128-58130
    58237-58239 58346-58348 58455-58457 58564-58566 58673-58675 58782-58784 58891-58893
    59000-59002 59109-59111 59218-59220" )
369 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN/REGULA/SET(1)/DIRECT", 3 )
370 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/MAXITE", 25 )
371 #setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/LINESE", True )
372 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/ENERGY", True )
373 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/DISPLA", False )
374 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/FORCE", True )
375 #setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/ENERGY/TOLCON", 0.001 )
376 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/ENERGY/NOCONV", "CONTIN" )
377 #setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/FORCE/TOLCON", 0.01 )
378 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/FORCE/NOCONV", "CONTIN" )
379 #setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/METHOD/METNAM", "SECANT" )
380
381 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "OUTPUT(1)/SELTYP",
    "USER" )
382 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "OUTPUT(1)/USER" )
383
384 ## OUTPUT ##
385 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/DISPLA(1)/TOTAL" )
386
387 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/FORCE(1)/REACTI" )
388
```

```
389 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(1)/TOTAL/GREEN" )
390 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(2)/TOTAL/GREEN/PRINCI" )
391 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(3)/CRKWDT" )
392 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(4)/CRACK/GREEN" )
393 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(5)/CRKWDT/GREEN/PRINCI" )
394 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(6)/TOTAL/TRACTI/LOCAL" )
395 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(7)/TOTAL/GREEN/LOCAL" )
396
397 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(1)/TOTAL/CAUCHY" )
398 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(2)/TOTAL/CAUCHY/PRINCI" )
399 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(3)/TOTAL/FORCE/LOCAL" )
400 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(4)/TOTAL/MOMENT/LOCAL" )
401 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(5)/TOTAL/TRACTI/LOCAL" )
402 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(6)/TOTAL/CAUCHY/LOCAL" )
403
404 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(1)/TOTAL/GREEN/GLOBAL/LOCATI", "INTPNT" )
405 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(1)/TOTAL/CAUCHY/GLOBAL/LOCATI", "INTPNT" )
406 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(6)/TOTAL/CAUCHY/LOCAL/LOCATI", "INTPNT" )
407 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(7)/TOTAL/GREEN/LOCAL/LOCATI", "INTPNT" )
408
409 #Run Analysis
410 # runSolver( [] )
411 # showView( "RESULT" )
412 fitAll( )
413
414 # renameViewSettingsDirectory( "view setting", "RESULT/CONTOU/CLIPS/PLANES(1)",
    "X=260" )
415 # setViewSettingValue( "view setting", "RESULT/CONTOU/CLIPS/PLANES(1)/LOCATI", "260
    0.000000 0.000000" )
416 # renameViewSettingsDirectory( "view setting", "RESULT/CONTOU/CLIPS/PLANES(2)",
    "X=340" )
417 # setViewSettingValue( "view setting", "RESULT/CONTOU/CLIPS/PLANES(2)/LOCATI", "340
    0.000000 0.000000" )
418 # renameViewSettingsDirectory( "view setting", "RESULT/CONTOU/CLIPS/PLANES(4)",
    "Y=50" )
419 # setViewSettingValue( "view setting", "RESULT/CONTOU/CLIPS/PLANES(4)/NORMAL",
    "0.00000 1.000000 0.000000" )
420 # setViewSettingValue( "view setting", "RESULT/CONTOU/CLIPS/PLANES(4)/LOCATI", "0
    100.000000 0.000000" )
421
422 #setViewSettingValue( "view setting", "RESULT/CONTOU/AUTRNG", "LIMITS" )
423 #setViewSettingValue( "view setting", "RESULT/CONTOU/LIMITS/MINVAL", -600 )
424 #setViewSettingValue( "view setting", "RESULT/CONTOU/LIMITS/MAXVAL", 600 )
425 #setViewSettingValue( "view setting", "RESULT/CONTOU/EQUIDI/NUMBER", 11 )
426
```

E2 - Vertical reinforcement model

```
newProject( "D:/Afstuderen/Python/3D/DIANA Output/ Vertical Anchor 30 mm", 10 )
setModelAnalysisAspects( [ "STRUCT" ] )
setModelDimension( "3D" )
setDefaultMeshOrder( "QUADRATIC" )
setDefaultMesherType( "HEXQUAD" )
setDefaultMidSideNodeLocation( "ONSHAP" )
setUnit( "LENGTH", "MM" )
setUnit( "FORCE", "N" )
setUnit( "ANGLE", "DEGREE" )

#Parameters
X1 = 200
X2 = 400
LX = X2-X1
Y1 = 0
Y2 = 200
LY = Y2-Y1
XM = X1+(X2-X1)/2
YM = Y1+(Y2-Y1)/2

#Thickness layers
T2 = 30
T1 = 70-T2
T3 = 180

#Anchor size
W1 = 10
H1 = 6
WX = XM-(W1/2)
WY = YM-(W1/2)
WZ = T1-H1
R1 = W1/2

#Create Shapes
createBlock( "SlabBottom", [ X1, Y1, 0 ], [ LX, LY, T1 ] )
createBlock( "SlabTop", [ X1, Y1, T1 ], [ LX, LY, T2 ] )
createBlock( "Pressure layer", [ X1, Y1, T1+T2 ], [ LX, LY, T3 ] )

#Create Reinforcement
createLine( "Koppel_M12a", [ X1, 30, 76 ], [ X2, 30, 76 ] )
createLine( "Koppel_M12b", [ X1, 180, 76 ], [ X2, 180, 76 ] )

createLine( "Reinfo_M8a", [ X1, 2.5, 35 ], [ X2, 2.5, 35 ] )
createLine( "Reinfo_M8b", [ X1, 67.5, 35 ], [ X2, 67.5, 35 ] )
createLine( "Reinfo_M8c", [ X1, 132.5, 35 ], [ X2, 132.5, 35 ] )
createLine( "Reinfo_M8d", [ X1, 197.5, 35 ], [ X2, 197.5, 35 ] )

createPolyline( "LG1-mb", [[ XM, 0, 165.5],[ XM, YM, 165.5],[ XM, Y2, 165.5]], False )
createLine( "LG1", [ XM, YM, T1],[ XM, YM, 165.5])
createBlock( "Anchor", [ WX, WY, WZ], [W1, W1, H1])
# createCylinder( "Anchor", [ XM, YM, WZ], [ 0, 0, 1 ], R1, H1 )
createPointBody( "point 1", [ XM, YM, T1 ] )

imprintIntersection( "Anchor", "point 1", True )
removeShape( [ "point 1" ] )

imprintIntersection( "SlabBottom", "Anchor", True )
imprintIntersection( "SlabTop", "Anchor", True )

#Regroup Material Parts
addSet( "GEOMETRYREINFOSET", "Coupling Reinforcement" )
moveToShapeSet( [ "Koppel_M12a","Koppel_M12b" ], "Coupling Reinforcement" )
addSet( "GEOMETRYREINFOSET", "Longitudinal Reinforcement" )
moveToShapeSet( [ "Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d"],
"Longitudinal Reinforcement" )
addSet( "GEOMETRYREINFOSET", "Lattice girders" )
moveToShapeSet( [ "LG1-mb", "LG1"], "Lattice girders" )
addSet( "SHAPESET", "Steel" )
moveToShapeSet( [ "Anchor"], "Steel" )
rename( "SHAPESET", "Shapes", "Concrete" )

```

```
71  #Material Precast concrete
72  addMaterial( "Precast concrete", "CONCR", "TSCR", [] )
73  setParameter( "MATERIAL", "Precast concrete", "LINEAR/ELASTI/YOUNG", 32000 )
74  setParameter( "MATERIAL", "Precast concrete", "LINEAR/ELASTI/POISON", 0.2 )
75  setParameter( "MATERIAL", "Precast concrete", "LINEAR/MASS/DENSIT", 2.4e-09 )
76  setParameter( "MATERIAL", "Precast concrete", "MODTYP/TOTCRK", "ROTATE" )
77  setParameter( "MATERIAL", "Precast concrete", "TENSIL/TENCRV", "HORDYK" )
78  setParameter( "MATERIAL", "Precast concrete", "TENSIL/TENSTR", 2.9 )
79  setParameter( "MATERIAL", "Precast concrete", "TENSIL/GF1", 0.1405 )
80  setParameter( "MATERIAL", "Precast concrete", "TENSIL/CBSPEC", "ROTS" )
81  setParameter( "MATERIAL", "Precast concrete", "TENSIL/POISRE/POIRED", "DAMAGE" )
82  setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMCRV", "MC2010" )
83  setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMSTR", 38 )
84  setParameter( "MATERIAL", "Precast concrete", "COMPRS/EPSC1", 0.0023 )
85  setParameter( "MATERIAL", "Precast concrete", "COMPRS/EPSCU", 0.0035 )
86  setParameter( "MATERIAL", "Precast concrete", "COMPRS/CONFIN/CNFCRV", "NONE" )
87  # setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMCRV", "CONSTA" )
88  # setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMSTR", 38 )
89
90  addGeometry( "Precast concrete", "SOLID", "STRSOL", [] )
91  setParameter( "GEOMET", "Precast concrete", "LOCAXS", True )
92  setParameter( "GEOMET", "Precast concrete", "LOCAXS/XAXIS", [ 1, 0, 0 ] )
93  setParameter( "GEOMET", "Precast concrete", "LOCAXS/YAXIS", [ 0, 1, 0 ] )
94
95  setElementClassType( "SHAPE", [ "SlabBottom", "SlabTop" ], "STRSOL" )
96  assignMaterial( "Precast concrete", "SHAPE", [ "SlabBottom", "SlabTop" ] )
97  assignGeometry( "Precast concrete", "SHAPE", [ "SlabBottom", "SlabTop" ] )
98
99  #Material Pressure layer
100 addMaterial( "Pressure layer", "CONCR", "TSCR", [] )
101 setParameter( "MATERIAL", "Pressure layer", "LINEAR/ELASTI/YOUNG", 32000 )
102 setParameter( "MATERIAL", "Pressure layer", "LINEAR/ELASTI/POISON", 0.2 )
103 setParameter( "MATERIAL", "Pressure layer", "LINEAR/MASS/DENSIT", 2.4e-09 )
104 setParameter( "MATERIAL", "Pressure layer", "MODTYP/TOTCRK", "ROTATE" )
105 setParameter( "MATERIAL", "Pressure layer", "TENSIL/TENCRV", "HORDYK" )
106 setParameter( "MATERIAL", "Pressure layer", "TENSIL/TENSTR", 2.9 )
107 setParameter( "MATERIAL", "Pressure layer", "TENSIL/GF1", 0.1405 )
108 setParameter( "MATERIAL", "Pressure layer", "TENSIL/CBSPEC", "ROTS" )
109 setParameter( "MATERIAL", "Pressure layer", "TENSIL/POISRE/POIRED", "DAMAGE" )
110 setParameter( "MATERIAL", "Pressure layer", "COMPRS/COMCRV", "MC2010" )
111 setParameter( "MATERIAL", "Pressure layer", "COMPRS/COMSTR", 38 )
112 setParameter( "MATERIAL", "Pressure layer", "COMPRS/EPSC1", 0.0023 )
113 setParameter( "MATERIAL", "Pressure layer", "COMPRS/EPSCU", 0.0035 )
114 setParameter( "MATERIAL", "Pressure layer", "COMPRS/CONFIN/CNFCRV", "NONE" )
115 #setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMCRV", "CONSTA" )
116 #setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMSTR", 38 )
117
118 addGeometry( "Pressure layer", "SOLID", "STRSOL", [] )
119 setParameter( "GEOMET", "Pressure layer", "LOCAXS", True )
120 setParameter( "GEOMET", "Pressure layer", "LOCAXS/XAXIS", [ 1, 0, 0 ] )
121 setParameter( "GEOMET", "Pressure layer", "LOCAXS/YAXIS", [ 0, 1, 0 ] )
122
123 setElementClassType( "SHAPE", [ "Pressure layer" ], "STRSOL" )
124 assignMaterial( "Precast concrete", "SHAPE", [ "Pressure layer"] )
125 assignGeometry( "Precast concrete", "SHAPE", [ "Pressure layer"] )
126
127 #Material Steel
128 addMaterial( "Steel", "MCSTEL", "ISOTRO", [] )
129 setParameter( "MATERIAL", "Steel", "LINEAR/ELASTI/YOUNG", 200000 )
130 setParameter( "MATERIAL", "Steel", "LINEAR/ELASTI/POISON", 0.3 )
131 setParameter( "MATERIAL", "Steel", "LINEAR/MASS/DENSIT", 0 )
132 addGeometry( "Anchor", "SOLID", "STRSOL", [] )
133 addElementData( "Anchor" )
134
135 setElementClassType( "SHAPE", [ "Anchor" ], "STRSOL" )
136 assignMaterial( "Steel", "SHAPE", [ "Anchor" ] )
137 assignGeometry( "Anchor", "SHAPE", [ "Anchor" ] )
138 assignElementData( "Anchor", "SHAPE", [ "Anchor" ] )
139
140 #Concrete Interface
141 addMaterial( "Interface Concrete", "INTERF", "FRICTI", [] )
```

```
142 setParameter( "MATERIAL", "Interface Concrete", "LINEAR/ELAS6/DSNZ", 60000 )
143 setParameter( "MATERIAL", "Interface Concrete", "LINEAR/ELAS6/DSSX", 6000 )
144 setParameter( "MATERIAL", "Interface Concrete", "LINEAR/ELAS6/DSSY", 6000 )
145 setParameter( "MATERIAL", "Interface Concrete", "COULOM/OPNTYP", "GAP" )
146 setParameter( "MATERIAL", "Interface Concrete", "COULOM/GAPVAL", 0.5 )
147 setParameter( "MATERIAL", "Interface Concrete", "COULOM/COHESI", 0.5 )
148 setParameter( "MATERIAL", "Interface Concrete", "COULOM/PHI", 30 )
149 setParameter( "MATERIAL", "Interface Concrete", "COULOM/PSI", 30 )
150 setParameter( "MATERIAL", "Interface Concrete", "COULOM/FRCHSR/MODE2", 0 )
151
152
153 #addMaterial( "Interface no tension", "INTERF", "NONLIF", [] )
154 #setParameter( "MATERIAL", "Interface no tension", "LINEAR/ELAS6/DSSY", 10000 )
155 #setParameter( "MATERIAL", "Interface no tension", "LINEAR/ELAS6/DSSX", 10000 )
156 #setParameter( "MATERIAL", "Interface no tension", "LINEAR/ELAS6/DSNZ", 100000 )
157
158 addGeometry( "Interface Concrete", "SHEET", "STPLIF", [] )
159 setParameter( "GEOMET", "Interface Concrete", "LOCAXS", True )
160 setParameter( "GEOMET", "Interface Concrete", "LOCAXS/XAXIS", [ 1, 0, 0 ] )
161
162 addElementData( "Interface Concrete" )
163 createConnection( "Interface Concrete", "INTER", "SHAPEFACE", "SHAPEFACE" )
164 setParameter( "GEOMETRYCONNECTION", "Interface Concrete", "MODE", "CLOSED" )
165 setElementClassType( "GEOMETRYCONNECTION", "Interface Concrete", "STPLIF" )
166 assignMaterial( "Interface Concrete", "GEOMETRYCONNECTION", "Interface Concrete" )
167 #assignMaterial( "Interface no tension", "GEOMETRYCONNECTION", "Interface Concrete" )
168 assignGeometry( "Interface Concrete", "GEOMETRYCONNECTION", "Interface Concrete" )
169 assignElementData( "Interface Concrete", "GEOMETRYCONNECTION", "Interface Concrete" )
170 setParameter( "GEOMETRYCONNECTION", "Interface Concrete", "FLIP", False )
171 attachTo( "GEOMETRYCONNECTION", "Interface Concrete", "SOURCE", "SlabTop", [ [ XM,
    YM, 70 ] ] )
172 attachTo( "GEOMETRYCONNECTION", "Interface Concrete", "TARGET", "Pressure layer", [
    [ XM, YM, 70 ] ] )
173
174 #Material Reinforcement Koppel_M12
175 addMaterial( "Koppel_M12", "REINFO", "REBOND", [ ] )
176 setParameter( "MATERIAL", "Koppel_M12", "REBARS/ELASTI/YOUNG", 200000 )
177 setParameter( "MATERIAL", "Koppel_M12", "REBARS/POISON/POISON", 0.3 )
178 setParameter( "MATERIAL", "Koppel_M12", "REBARS/PLATYP", "VMISES" )
179 setParameter( "MATERIAL", "Koppel_M12", "REBARS/PLASTI/TRESSH", "KAPSIG" )
180 setParameter( "MATERIAL", "Koppel_M12", "REBARS/PLASTI/KAPSIG", [ 0, 550, 0.04725,
    594 ] )
181 setParameter( "MATERIAL", "Koppel_M12", "RESLIP/SHFTYP", "BONDS6" )
182 setParameter( "MATERIAL", "Koppel_M12", "RESLIP/DSNY", 100000 )
183 setParameter( "MATERIAL", "Koppel_M12", "RESLIP/DSSX", 10000 )
184 setParameter( "MATERIAL", "Koppel_M12", "RESLIP/BONDS6/SLPVAL", [ 15.41, 6.16, 0.1,
    1, 2, 10, 0.4 ] )
185 addGeometry( "Koppel_M12", "RELINE", "REBAR", [] )
186 setParameter( "GEOMET", "Koppel_M12", "REITYP", "REITRU" )
187 setParameter( "GEOMET", "Koppel_M12", "REITYP", "CIRBEA" )
188 setParameter( "GEOMET", "Koppel_M12", "CIRBEA/CIRCLE", 12)
189 setParameter( "GEOMET", "Koppel_M12", "ORIENT/ZAXIS", [ 0, 0, 1 ] )
190 setParameter( "GEOMET", "Koppel_M12", "TIPELM/SURFAC", 113.1 )
191 #assignMaterial( "Koppel_M12", "SHAPE", [ "Koppel_M12a","Koppel_M12b" ] )
192 #assignGeometry( "Koppel_M12", "SHAPE", [ "Koppel_M12a","Koppel_M12b" ] )
193 #convertToReinforcement( [ "Koppel_M12a","Koppel_M12b" ] )
194 #addElementData( "Koppel_M12" )
195 #assignElementData( "Koppel_M12", "SHAPE", [ "Koppel_M12a","Koppel_M12b" ] )
196 #setParameter( "DATA", "Koppel_M12", "INTERF", "BEAM" )
197 #setReinforcementDiscretization( [ "Koppel_M12a","Koppel_M12b" ], "ELEMENT" )
198 #setContinuousInInterfaces( [ "Koppel_M12a","Koppel_M12b" ], False )
199 setReinforcementType( "GEOMETRYREINFOSET", "Coupling Reinforcement",
    "BEAM_NUMERICAL_BOND_SLIP" )
200 setContinuousInInterfaces( "GEOMETRYREINFOSET", "Coupling Reinforcement", False )
201 assignMaterial( "Koppel_M12", "GEOMETRYREINFOSET", [ "Coupling Reinforcement"])
202 assignGeometry( "Koppel_M12", "GEOMETRYREINFOSET", [ "Coupling Reinforcement"])
203
204 #Material Reinforcement M8
205 addMaterial( "Reinfo_M8", "REINFO", "REBOND", [] )
206 setParameter( "MATERIAL", "Reinfo_M8", "REBARS/ELASTI/YOUNG", 200000 )
207 setParameter( "MATERIAL", "Reinfo_M8", "REBARS/POISON/POISON", 0.3 )
```

```
208  setParameter( "MATERIAL", "Reinfo_M8", "REBARS/PLATYP", "VMISES" )
209  setParameter( "MATERIAL", "Reinfo_M8", "REBARS/PLASTI/TRESSH", "KAPSIG" )
210  setParameter( "MATERIAL", "Reinfo_M8", "REBARS/PLASTI/KAPSIG", [ 0, 550, 0.04725,
     594 ] )
211  setParameter( "MATERIAL", "Reinfo_M8", "RESLIP/SHFTYP", "BONDS6" )
212  setParameter( "MATERIAL", "Reinfo_M8", "RESLIP/DSNY", 100000 )
213  setParameter( "MATERIAL", "Reinfo_M8", "RESLIP/DSSX", 10000 )
214  setParameter( "MATERIAL", "Reinfo_M8", "RESLIP/BONDS6/SLPVAL", [ 15.41, 6.16, 0.1,
     1, 2, 10, 0.4 ] )
215  addGeometry( "Reinfo_M8", "RELINE", "REBAR", [] )
216  setParameter( "GEOMET", "Reinfo_M8", "REITYP", "REITRU" )
217  setParameter( "GEOMET", "Reinfo_M8", "REITYP", "CIRBEA" )
218  setParameter( "GEOMET", "Reinfo_M8", "CIRBEA/CIRCLE", 8)
219  setParameter( "GEOMET", "Reinfo_M8", "ORIENT/ZAXIS", [ 0, 0, 1 ] )
220  setParameter( "GEOMET", "Reinfo_M8", "TIPELM/SURFAC", 50.3 )
221  #assignMaterial( "Reinfo_M8", "SHAPE", [
     "Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d" ] )
222  #assignGeometry( "Reinfo_M8", "SHAPE", [
     "Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d" ] )
223  #convertToReinforcement( [ "Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d"] )
224  #addElementData( "Reinfo_M8" )
225  #assignElementData( "Reinfo_M8", "SHAPE", [
     "Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d"] )
226  #setParameter( "DATA", "Reinfo_M8", "INTERF", "BEAM" )
227  #setReinforcementDiscretization( [
     "Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d" ], "ELEMENT" )
228  #setContinuousInInterfaces( [ "Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d" ],
     False )
229  setReinforcementType( "GEOMETRYREINFOSET", "Longitudinal Reinforcement",
     "BEAM_NUMERICAL_BOND_SLIP" )
230  setContinuousInInterfaces( "GEOMETRYREINFOSET", "Longitudinal Reinforcement", False )
231  assignMaterial( "Reinfo_M8", "GEOMETRYREINFOSET", [ "Longitudinal Reinforcement"])
232  assignGeometry( "Reinfo_M8", "GEOMETRYREINFOSET", [ "Longitudinal Reinforcement"])
233
234  #Material Reinforcement M5
235  addMaterial( "Reinfo_M5", "REINFO", "REBOND", [] )
236  setParameter( "MATERIAL", "Reinfo_M5", "REBARS/ELASTI/YOUNG", 200000 )
237  setParameter( "MATERIAL", "Reinfo_M5", "REBARS/POISON/POISON", 0.3 )
238  setParameter( "MATERIAL", "Reinfo_M5", "REBARS/PLATYP", "VMISES" )
239  setParameter( "MATERIAL", "Reinfo_M5", "REBARS/PLASTI/TRESSH", "KAPSIG" )
240  setParameter( "MATERIAL", "Reinfo_M5", "REBARS/PLASTI/KAPSIG", [ 0, 550, 0.04725,
     594 ] )
241  setParameter( "MATERIAL", "Reinfo_M5", "RESLIP/SHFTYP", "BONDS6" )
242  setParameter( "MATERIAL", "Reinfo_M5", "RESLIP/DSNY", 100000 )
243  setParameter( "MATERIAL", "Reinfo_M5", "RESLIP/DSSX", 10000 )
244  setParameter( "MATERIAL", "Reinfo_M5", "RESLIP/BONDS6/SLPVAL", [ 0.63, 0.63, 0.01,
     0.02, 0.03, 0.04, 0.5 ] )
245  addGeometry( "Reinfo_M5", "RELINE", "REBAR", [] )
246  setParameter( "GEOMET", "Reinfo_M5", "REITYP", "REITRU" )
247  setParameter( "GEOMET", "Reinfo_M5", "REITYP", "CIRBEA" )
248  setParameter( "GEOMET", "Reinfo_M5", "CIRBEA/CIRCLE", 5 )
249  # setParameter( "GEOMET", "Reinfo_M5", "ORIENT/ZAXIS", [ 0, 0, 1 ] )
250  setParameter( "GEOMET", "Reinfo_M5", "TIPELM/SURFAC", 19.6 )
251
252  setReinforcementType( "GEOMETRYREINFOSET", "Lattice girders",
     "BEAM_NUMERICAL_BOND_SLIP" )
253  setContinuousInInterfaces( "GEOMETRYREINFOSET", "Lattice girders", True )
254  assignMaterial( "Reinfo_M5", "GEOMETRYREINFOSET", [ "Lattice girders" ] )
255  assignGeometry( "Reinfo_M5", "GEOMETRYREINFOSET", [ "Lattice girders" ] )
256
257  # setParameter( "DATA", "Bar", "NUMINT", [ "GAUSS" ] )
258  # setParameter( "DATA", "Bar", "INTEGR", "HIGH" )
259
260  #Composed Line
261  #addGeometry( "Composed Line", "LINE", "COMLIN", [] )
262  #setParameter( "GEOMET", "Composed Line", "DISTAN/THICK", 250 )
263  #setElementClassType( "SHAPE", [ "Composed Line" ], "COMLIN" )
264  #assignGeometry( "Composed Line", "SHAPE", [ "Composed Line" ] )
265  #resetElementData( "SHAPE", [ "Composed Line" ] )
266
267  #Tyings
```

```
createPointTying( "Tying Top", "Tying" )
setParameter( "GEOMETRYTYING", "Tying Top", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYTYING", "Tying Top", "TRANSL", [ 1, 1, 1 ] )
setParameter( "GEOMETRYTYING", "Tying Top", "ROTATI", [ 1, 1, 1 ] )
attachTo( "GEOMETRYTYING", "Tying Top", "SLAVE", "LG1", [ [ XM, YM, 165.5 ] ] )
attachTo( "GEOMETRYTYING", "Tying Top", "MASTER", "LG1-mb", [ [ XM, YM, 165.5 ] ] )

createPointTying( "Tying Bot", "Tying" )
setParameter( "GEOMETRYTYING", "Tying Bot", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYTYING", "Tying Bot", "TRANSL", [ 1, 1, 1 ] )
setParameter( "GEOMETRYTYING", "Tying Bot", "ROTATI", [ 1, 1, 1 ] )
attachTo( "GEOMETRYTYING", "Tying Bot", "SLAVE", "Anchor", [ [ XM, YM, T1 ] ] )
attachTo( "GEOMETRYTYING", "Tying Bot", "MASTER", "LG1", [ [ XM, YM, T1 ] ] )

#Assign Supports
addSet( "GEOMETRYSUPPORTSET", "Supports" )

createSurfaceSupport( "Support X", "Supports" )
setParameter( "GEOMETRYSUPPORT", "Support X", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Support X", "TRANSL", [ 1, 0, 0 ] )
setParameter( "GEOMETRYSUPPORT", "Support X", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Support X", "SlabBottom", [ [ X1, 344.1438, 19.402428 ], [ X2, 344.1438, 26.097572 ] ] )
attach( "GEOMETRYSUPPORT", "Support X", "SlabTop", [ [ X1, 344.1438, 55.947461 ], [ X2, 344.1438, 59.552538 ] ] )
attach( "GEOMETRYSUPPORT", "Support X", "Pressure layer", [ [ X1, 344.1438, 146.75686 ], [ X2, 344.1438, 173.24314 ] ] )

createSurfaceSupport( "Support Y", "Supports" )
setParameter( "GEOMETRYSUPPORT", "Support Y", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Support Y", "TRANSL", [ 0, 1, 0 ] )
setParameter( "GEOMETRYSUPPORT", "Support Y", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Support Y", "SlabBottom", [ [ 344.1438, Y1, 19.402428 ], [ 344.1438, Y2, 26.097572 ] ] )
attach( "GEOMETRYSUPPORT", "Support Y", "SlabTop", [ [ 344.1438, Y1, 55.947461 ], [ 344.1438, Y2, 59.552538 ] ] )
attach( "GEOMETRYSUPPORT", "Support Y", "Pressure layer", [ [ 344.1438, Y1, 146.75686 ], [ 344.1438, Y2, 173.24314 ] ] )

createSurfaceSupport( "Support Z", "Supports" )
setParameter( "GEOMETRYSUPPORT", "Support Z", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Support Z", "TRANSL", [ 0, 0, 1 ] )
setParameter( "GEOMETRYSUPPORT", "Support Z", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Support Z", "Pressure layer", [ [ XM, YM, 250 ] ] )

createSurfaceSupport( "Deform", "Supports" )
setParameter( "GEOMETRYSUPPORT", "Deform", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Deform", "TRANSL", [ 0, 0, 1 ] )
setParameter( "GEOMETRYSUPPORT", "Deform", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Deform", "SlabBottom", [ [ XM, YM, 0 ] ] )

createPointSupport( "Support Y beam", "Supports" )
setParameter( "GEOMETRYSUPPORT", "Support Y beam", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Support Y beam", "TRANSL", [ 0, 1, 0 ] )
setParameter( "GEOMETRYSUPPORT", "Support Y beam", "ROTATI", [ 1, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Support Y beam", "LG1-mb", [ [ XM, Y1, 165.5 ], [ XM, Y2, 165.5 ] ] )

#Assign Loads
addSet( "GEOMETRYLOADSET", "Load" )

# createLineLoad( "Load", "Load" )
# setParameter( "GEOMETRYLOAD", "Load", "LODTYP", "DEFORM" )
# setParameter( "GEOMETRYLOAD", "Load", "DEFORM/TR/VALUE", 0.1 )
# setParameter( "GEOMETRYLOAD", "Load", "DEFORM/TR/DIRECT", 1 )
# attach( "GEOMETRYLOAD", "Load", "Pressure Layer", [[ 200, 100, 250 ]] )

createSurfaceLoad( "Load", "Load" )
setParameter( "GEOMETRYLOAD", "Load", "LODTYP", "DEFORM" )
setParameter( "GEOMETRYLOAD", "Load", "DEFORM/TR/VALUE", -0.1 )
setParameter( "GEOMETRYLOAD", "Load", "DEFORM/TR/DIRECT", 3 )
```

```
attach( "GEOMETRYLOAD", "Load", "SlabBottom", [ [ XM, YM, 0 ] ] )

#Set mesh properties
setElementSize( [ "Anchor" ], 4, -1, True )
clearMesherType( [ "Anchor" ] )
setElementSize( [ "SlabTop" ], 8, -1, True )
clearMesherType( [ "SlabTop" ] )
setElementSize( [ "Pressure layer", "SlabBottom" ], 20, -1, True )
clearMesherType( [ "Pressure layer", "SlabBottom" ] )
generateMesh( [] )
hideView( "GEOM" )
showView( "MESH" )

#Analysis
addAnalysis( "Analysis1" )
addAnalysisCommand( "Analysis1", "NONLIN", "Structural nonlinear" )
renameAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "EXECUT(1)",
"Load" )
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/STEPTY", "AUTOMA" )
# setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/STEPS/AUTOMA/SIZES", 250 )
# setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/STEPS/AUTOMA/MAXSIZ", 0.01 )
# setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/STEPS/AUTOMA/MINSIZ", 0.00001 )
# setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/STEPS/AUTOMA/CUTBCK", 0.25 )
# setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/STEPS/AUTOMA/MAXSTP", 100 )
# setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/STEPS/AUTOMA/ARCLEN", True )
# addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/STEPS/AUTOMA/ARCLEN/REGULA/SET" )
# setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/STEPS/AUTOMA/ARCLEN/REGULA/SET(1)/NODES(1)/RNGNRS", "1-9361" )
# setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/STEPS/AUTOMA/ARCLEN/REGULA/SET(1)/DIRECT", 2 )
setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/STEPS/EXPLIC/SIZES", "0.2(15)" )
# setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN", True )
# addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN/REGULA/SET" )
# setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN/REGULA/SET(1)/NODES(1)/RNGNRS", "3 4 9 10 41-96
237-242 349-354 569-2000(53) 2001-2113 2133-2653(20) 3541-9848(53) 9992 10047-10109
10163 10202 10223-10227 10281-10283 10323 10324 10364 10365 10405 10406 10446 10447
10487 10488 10528 10529 10569 10570 10610 10611 10651 10652 10692 10693 10733 10734
10774 10775 10815 10816 10856 10857 10897 10898 10938 10939 10979 10980 11020 11021
11061 11062 11102 11103 11143 11144 11184 11185 11225 11226 11266 11267 11307 11308
11348 11349 11389 11390 11430 11592-11847 12237 12239 12344 12346 12451 12453 12558
12560 12665 12667 12772 12774 12879 12881 12986 12988 13093 13095 13200 13202 13307
13309 13414 13416 13521 13523 13628 13630 13735 13737 13842 13844 13949 13951 14056
14058 14163 14165 14270 14272 14377 14379 14484 14486 14591 14593 14698 14700 14805
14807 14912 14914 15019 15021 15126 15128 15181-15183 15289 15290 15396 15397 15503
15504 15610 15663 15664 15770 15771 15877 15878 15984 15985 16091 17825-17827
17985-17987 18145-18147 18305-18307 18465 18518-18520 18678-18680 18838-18840
18998-19000 19158 19211-19213 19371-19373 19531-19533 19691-19693 19851 19904-19906
20064-20066 20224-20226 20384-20386 20544 20597-20599 20757-20759 20917-20919
21077-21079 21237 21290-21292 21450-21452 21610-21612 21770-21772 21930 21983-21985
22143-22145 22303-22305 22463-22465 22623 22676-22678 22836-22838 22996-22998
23156-23158 23316 23369-23371 23529-23531 23689-23691 23849-23851 24009 24062-24064
24222-24224 24382-24384 24542-24544 24702 24755-24757 24915-24917 25075-25077
25235-25237 25395 25448-25450 25608-25610 25768-25770 25928-25930 26088 26141-26143
26301-26303 26461-26463 26621-26623 26781 26834-26836 26994-26996 27154-27156
27314-27316 27474 27527-27529 27687-27689 27847-27849 28007-28009 28167 28220-28222
```

```
28380-28382 28540-28542 28700-28702 28860 28913-28915 29073-29075 29233-29235
29393-29395 29553 29606-29608 29766-29768 29926-29928 30086-30088 30246 30299-30301
30459-30461 30619-30621 30779-30781 30939 30992-30994 31152-31154 31312-31314
31472-31474 31632 31685-31687 31845-31847 32005-32007 32165-32167 32325 32378-32380
32538-32540 32698-32700 32858-32860 33018 33071-33073 33231-33233 33391-33393
33551-33553 33711 33764-33766 33924-33926 34084-34086 34244-34246 34404 34457-34459
34617-34619 34777-34779 34937-34939 35097 35150-35152 35310-35312 35470-35472
35630-35632 35790 35843-35845 36003-36005 36163-36165 36323-36325 36483 36536-36538
36696-36698 36856-36858 37016-37018 37176-37441(53) 37443 37445-37472 37501 37554
37659-39090(53) 39091 39095 39183-39210 39262 39263 39315 39316 39611-40772(43)
40773-40801 40854-42338(53) 42420-42449 42503 42504 42548 42551-42579 42650 42704
42705 42749 42844 42846 42931 42933 43018 43020 43105 43107 43192 43194 43279 43281
43366 43368 43453 43455 43540 43542 43627 43629 43714 43716 43801 43803 43888 43890
43975 43977 44062 44064 44149 44151 44236 44238 44323 44325 44410 44412 44497 44499
44584 44586 44671 44673 44758 44760 44845 44847 44932 44934 45019 45021 45106 45108
45193 45195 45238-45323 45950 45951 46057 46110 46111 46217 46374 46376 46481 46483
46588 46590 46695 46697 46802 46804 46909 46911 47016 47018 47123 47125 47230 47232
47337 47339 47444 47446 47551 47553 47658 47660 47765 47767 47872 47874 47979 47981
48086 48088 48193 48195 48300 48302 48407 48409 48514 48516 48621 48623 48728 48730
48835 48837 48942 48944 49049 49051 49156 49158 49263 49265 49318-49321 49479
49532-49534 49692 49745-49747 49905 49958-49960 50118 50171-50173 50331 50384-50386
50544 50597-50599 50757 50810-50812 50970 51023-51025 51183 51236-51238 51396
51449-51451 51609 51662-51664 51822 51875-51877 52035 52088-52090 52248 52301-52303
52461 52514-52516 52674 52727-52729 52887 52940-52942 53100 53153-53155 53313
53366-53368 53526 53579-53581 53739 53792-53794 53952 54005-54007 54165 54218-54220
54378 54431-54433 54591 54644-54646 54804 54857-54859 55017 55070-55072 55230 55283
57181 57228-57230 57269-57271 57310-57312 57351-57353 57392-57394 57433-57435
57474-57476 57583-57585 57692-57694 57801-57803 57910-57912 58019-58021 58128-58130
58237-58239 58346-58348 58455-58457 58564-58566 58673-58675 58782-58784 58891-58893
59000-59002 59109-59111 59218-59220" )
# setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN/REGULA/SET(1)/DIRECT", 3 )
setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/ITERAT/MAXITE", 25 )
#setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/ITERAT/LINESE", True )
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/FORCE", 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/TOLCON", 0.01 )
setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/ITERAT/CONVER/FORCE/NOCONV", "CONTIN" )
#setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/ITERAT/METHOD/METNAM", "SECANT" )

setAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "OUTPUT(1)/SELTYP",
"USER" )
addAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "OUTPUT(1)/USER" )

## OUTPUT ##
addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"OUTPUT(1)/USER/DISPLA(1)/TOTAL" )

addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"OUTPUT(1)/USER/FORCE(1)/REACTI" )

addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"OUTPUT(1)/USER/STRAIN(1)/TOTAL/GREEN" )
addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"OUTPUT(1)/USER/STRAIN(2)/TOTAL/GREEN/PRINCI" )
addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"OUTPUT(1)/USER/STRAIN(3)/CRKWDT" )
addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
```

```
    "OUTPUT(1)/USER/STRAIN(4)/CRACK/GREEN" )
389 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(5)/CRKWDT/GREEN/PRINCI" )
390 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(6)/TOTAL/TRACTI/LOCAL" )
391 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(7)/TOTAL/GREEN/LOCAL" )
392
393 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(1)/TOTAL/CAUCHY" )
394 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(2)/TOTAL/CAUCHY/PRINCI" )
395 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(3)/TOTAL/FORCE/LOCAL" )
396 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(4)/TOTAL/MOMENT/LOCAL" )
397 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(5)/TOTAL/TRACTI/LOCAL" )
398 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(6)/TOTAL/CAUCHY/LOCAL" )
399
400 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(1)/TOTAL/GREEN/GLOBAL/LOCATI", "INTPNT" )
401 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(1)/TOTAL/CAUCHY/GLOBAL/LOCATI", "INTPNT" )
402 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(6)/TOTAL/CAUCHY/LOCAL/LOCATI", "INTPNT" )
403 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(7)/TOTAL/GREEN/LOCAL/LOCATI", "INTPNT" )
404
405 #Run Analysis
406 # runSolver( [] )
407 # showView( "RESULT" )
408 fitAll( )
409
410 # renameViewSettingsDirectory( "view setting", "RESULT/CONTOU/CLIPS/PLANES(1)",
    "X=260" )
411 # setViewSettingValue( "view setting", "RESULT/CONTOU/CLIPS/PLANES(1)/LOCATI", "260
    0.000000 0.000000" )
412 # renameViewSettingsDirectory( "view setting", "RESULT/CONTOU/CLIPS/PLANES(2)",
    "X=340" )
413 # setViewSettingValue( "view setting", "RESULT/CONTOU/CLIPS/PLANES(2)/LOCATI", "340
    0.000000 0.000000" )
414 # renameViewSettingsDirectory( "view setting", "RESULT/CONTOU/CLIPS/PLANES(4)",
    "Y=50" )
415 # setViewSettingValue( "view setting", "RESULT/CONTOU/CLIPS/PLANES(4)/NORMAL",
    "0.00000 1.000000 0.000000" )
416 # setViewSettingValue( "view setting", "RESULT/CONTOU/CLIPS/PLANES(4)/LOCATI", "0
    100.000000 0.000000" )
417
418 #setViewSettingValue( "view setting", "RESULT/CONTOU/AUTRNG", "LIMITS" )
419 #setViewSettingValue( "view setting", "RESULT/CONTOU/LIMITS/MINVAL", -600 )
420 #setViewSettingValue( "view setting", "RESULT/CONTOU/LIMITS/MAXVAL", 600 )
421 #setViewSettingValue( "view setting", "RESULT/CONTOU/EQUIDI/NUMBER", 11 )
422
```

E3 - Lattice girder model bond-slip

```
newProject( "D:/Afstuderen/Python/3D/DIANA Output/ Lattice girder bondslip test", 10 )
setModelAnalysisAspects( [ "STRUCT" ] )
setModelDimension( "3D" )
setDefaultMeshOrder( "QUADRATIC" )
setDefaultMesherType( "HEXQUAD" )
setDefaultMidSideNodeLocation( "ONSHAP" )
setUnit( "LENGTH", "MM" )
setUnit( "FORCE", "N" )
setUnit( "ANGLE", "DEGREE" )

#Parameters
X1=200
X2=400
LX=X2-X1
Y1=0
Y2=200
LY=Y2-Y1
XM=X1+(X2-X1)/2
YM=Y1+(Y2-Y1)/2

#Thickness layers
T1 = 45.5
T2 = 24.5
T3 = 180

#Create Shapes
createBlock( "SlabBottom", [ X1, Y1, 0 ], [ LX, LY, T1 ] )
createBlock( "SlabTop", [ X1, Y1, T1 ], [ LX, LY, T2 ] )
createBlock( "Pressure layer", [ X1, Y1, T1+T2 ], [ LX, LY, T3 ] )

#Create Reinforcement
createLine( "Koppel_M12a", [ X1, 30, 76 ], [ X2, 30, 76 ] )
createLine( "Koppel_M12b", [ X1, 180, 76 ], [ X2, 180, 76 ] )

createLine( "Reinfo_M8a", [ X1, 2.5, 35 ], [ X2, 2.5, 35 ] )
createLine( "Reinfo_M8b", [ X1, 67.5, 35 ], [ X2, 67.5, 35 ] )
createLine( "Reinfo_M8c", [ X1, 132.5, 35 ], [ X2, 132.5, 35 ] )
createLine( "Reinfo_M8d", [ X1, 197.5, 35 ], [ X2, 197.5, 35 ] )

createPolyline( "LG1-mb", [[ XM, 0, 165.5],[ XM, YM, 165.5],[ XM, Y2, 165.5]], False )
createPolyline( "LG1-l", [ [260, Y1, 45.5],[XM, Y1+100, 165.5],[260, Y1+200, 45.5]
],True )
createPolyline( "LG1-r", [ [340, Y1, 45.5],[XM, Y1+100, 165.5],[340, Y1+200, 45.5]
],True )


#Regroup Material Parts
addSet( "GEOMETRYREINFOSET", "Coupling Reinforcement" )
moveToShapeSet( [ "Koppel_M12a","Koppel_M12b" ], "Coupling Reinforcement" )
addSet( "GEOMETRYREINFOSET", "Longitudinal Reinforcement" )
moveToShapeSet( [ "Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d"],
"Longitudinal Reinforcement" )
addSet( "GEOMETRYREINFOSET", "Lattice girders" )
moveToShapeSet( [ "LG1-mb", "LG1-l", "LG1-r"], "Lattice girders" )
rename( "SHAPESET", "Shapes", "Concrete" )

#Material Precast concrete
addMaterial( "Precast concrete", "CONCR", "TSCR", [] )
setParameter( "MATERIAL", "Precast concrete", "LINEAR/ELASTI/YOUNG", 32000 )
setParameter( "MATERIAL", "Precast concrete", "LINEAR/ELASTI/POISON", 0.2 )
setParameter( "MATERIAL", "Precast concrete", "LINEAR/MASS/DENSIT", 2.4e-09 )
setParameter( "MATERIAL", "Precast concrete", "MODTYP/TOTCRK", "ROTATE" )
setParameter( "MATERIAL", "Precast concrete", "TENSIL/TENCRV", "HORDYK" )
setParameter( "MATERIAL", "Precast concrete", "TENSIL/TENSTR", 2.9 )
setParameter( "MATERIAL", "Precast concrete", "TENSIL/GF1", 0.1405 )
setParameter( "MATERIAL", "Precast concrete", "TENSIL/CBSPEC", "ROTS" )
setParameter( "MATERIAL", "Precast concrete", "TENSIL/POISRE/POIRED", "DAMAGE" )
setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMCRV", "MC2010" )
setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMSTR", 38 )
setParameter( "MATERIAL", "Precast concrete", "COMPRS/EPSC1", 0.0023 )
setParameter( "MATERIAL", "Precast concrete", "COMPRS/EPSCU", 0.0035 )
```

```
69  setParameter( "MATERIAL", "Precast concrete", "COMPRS/CONFIN/CNFCRV", "NONE" )
70  # setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMCRV", "CONSTA" )
71  # setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMSTR", 38 )
72
73  addGeometry( "Precast concrete", "SOLID", "STRSOL", [] )
74  setParameter( "GEOMET", "Precast concrete", "LOCAXS", True )
75  setParameter( "GEOMET", "Precast concrete", "LOCAXS/XAXIS", [ 1, 0, 0 ] )
76  setParameter( "GEOMET", "Precast concrete", "LOCAXS/YAXIS", [ 0, 1, 0 ] )
77
78  setElementClassType( "SHAPE", [ "SlabBottom", "SlabTop" ], "STRSOL" )
79  assignMaterial( "Precast concrete", "SHAPE", [ "SlabBottom", "SlabTop" ] )
80  assignGeometry( "Precast concrete", "SHAPE", [ "SlabBottom", "SlabTop" ] )
81
82  #Material Pressure layer
83  addMaterial( "Pressure layer", "CONCR", "TSCR", [] )
84  setParameter( "MATERIAL", "Pressure layer", "LINEAR/ELASTI/YOUNG", 32000 )
85  setParameter( "MATERIAL", "Pressure layer", "LINEAR/ELASTI/POISON", 0.2 )
86  setParameter( "MATERIAL", "Pressure layer", "LINEAR/MASS/DENSIT", 2.4e-09 )
87  setParameter( "MATERIAL", "Pressure layer", "MODTYP/TOTCRK", "ROTATE" )
88  setParameter( "MATERIAL", "Pressure layer", "TENSIL/TENCRV", "HORDYK" )
89  setParameter( "MATERIAL", "Pressure layer", "TENSIL/TENSTR", 2.9 )
90  setParameter( "MATERIAL", "Pressure layer", "TENSIL/GF1", 0.1405 )
91  setParameter( "MATERIAL", "Pressure layer", "TENSIL/CBSPEC", "ROTS" )
92  setParameter( "MATERIAL", "Pressure layer", "TENSIL/POISRE/POIRED", "DAMAGE" )
93  setParameter( "MATERIAL", "Pressure layer", "COMPRS/COMCRV", "MC2010" )
94  setParameter( "MATERIAL", "Pressure layer", "COMPRS/COMSTR", 38 )
95  setParameter( "MATERIAL", "Pressure layer", "COMPRS/EPSC1", 0.0023 )
96  setParameter( "MATERIAL", "Pressure layer", "COMPRS/EPSCU", 0.0035 )
97  setParameter( "MATERIAL", "Pressure layer", "COMPRS/CONFIN/CNFCRV", "NONE" )
98  #setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMCRV", "CONSTA" )
99  #setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMSTR", 38 )
100
101 addGeometry( "Pressure layer", "SOLID", "STRSOL", [] )
102 setParameter( "GEOMET", "Pressure layer", "LOCAXS", True )
103 setParameter( "GEOMET", "Pressure layer", "LOCAXS/XAXIS", [ 1, 0, 0 ] )
104 setParameter( "GEOMET", "Pressure layer", "LOCAXS/YAXIS", [ 0, 1, 0 ] )
105
106 setElementClassType( "SHAPE", [ "Pressure layer" ], "STRSOL" )
107 assignMaterial( "Precast concrete", "SHAPE", [ "Pressure layer"] )
108 assignGeometry( "Precast concrete", "SHAPE", [ "Pressure layer"] )
109
110 #Concrete Interface
111 addMaterial( "Interface Concrete", "INTERF", "FRICTI", [] )
112 setParameter( "MATERIAL", "Interface Concrete", "LINEAR/ELAS6/DSNZ", 60000 )
113 setParameter( "MATERIAL", "Interface Concrete", "LINEAR/ELAS6/DSSX", 6000 )
114 setParameter( "MATERIAL", "Interface Concrete", "LINEAR/ELAS6/DSSY", 6000 )
115 setParameter( "MATERIAL", "Interface Concrete", "COULOM/OPNTYP", "GAP" )
116 setParameter( "MATERIAL", "Interface Concrete", "COULOM/GAPVAL", 0.5 )
117 setParameter( "MATERIAL", "Interface Concrete", "COULOM/COHESI", 0.5 )
118 setParameter( "MATERIAL", "Interface Concrete", "COULOM/PHI", 30 )
119 setParameter( "MATERIAL", "Interface Concrete", "COULOM/PSI", 30 )
120 setParameter( "MATERIAL", "Interface Concrete", "COULOM/FRCHSR/MODE2", 0 )
121
122
123 #addMaterial( "Interface no tension", "INTERF", "NONLIF", [] )
124 #setParameter( "MATERIAL", "Interface no tension", "LINEAR/ELAS6/DSSY", 10000 )
125 #setParameter( "MATERIAL", "Interface no tension", "LINEAR/ELAS6/DSSX", 10000 )
126 #setParameter( "MATERIAL", "Interface no tension", "LINEAR/ELAS6/DSNZ", 100000 )
127
128 addGeometry( "Interface Concrete", "SHEET", "STPLIF", [] )
129 setParameter( "GEOMET", "Interface Concrete", "LOCAXS", True )
130 setParameter( "GEOMET", "Interface Concrete", "LOCAXS/XAXIS", [ 1, 0, 0 ] )
131
132 addElementData( "Interface Concrete" )
133 createConnection( "Interface Concrete", "INTER", "SHAPEFACE", "SHAPEFACE" )
134 setParameter( "GEOMETRYCONNECTION", "Interface Concrete", "MODE", "CLOSED" )
135 setElementClassType( "GEOMETRYCONNECTION", "Interface Concrete", "STPLIF" )
136 assignMaterial( "Interface Concrete", "GEOMETRYCONNECTION", "Interface Concrete" )
137 #assignMaterial( "Interface no tension", "GEOMETRYCONNECTION", "Interface Concrete" )
138 assignGeometry( "Interface Concrete", "GEOMETRYCONNECTION", "Interface Concrete" )
139 assignElementData( "Interface Concrete", "GEOMETRYCONNECTION", "Interface Concrete" )
```

```
140 setParameter( "GEOMETRYCONNECTION", "Interface Concrete", "FLIP", False )
141 attachTo( "GEOMETRYCONNECTION", "Interface Concrete", "SOURCE", "SlabTop", [ [ XM,
    YM, 70 ] ] )
142 attachTo( "GEOMETRYCONNECTION", "Interface Concrete", "TARGET", "Pressure layer", [
    [ XM, YM, 70 ] ] )
143
144 #Material Reinforcement Koppel_M12
145 addMaterial( "Koppel_M12", "REINFO", "REBOND", [ ] )
146 setParameter( "MATERIAL", "Koppel_M12", "REBARS/ELASTI/YOUNG", 200000 )
147 setParameter( "MATERIAL", "Koppel_M12", "REBARS/POISON/POISON", 0.3 )
148 setParameter( "MATERIAL", "Koppel_M12", "REBARS/PLATYP", "VMISES" )
149 setParameter( "MATERIAL", "Koppel_M12", "REBARS/PLASTI/TRESSH", "KAPSIG" )
150 setParameter( "MATERIAL", "Koppel_M12", "REBARS/PLASTI/KAPSIG", [ 0, 550, 0.04725,
    594 ] )
151 setParameter( "MATERIAL", "Koppel_M12", "RESLIP/SHFTYP", "BONDS6" )
152 setParameter( "MATERIAL", "Koppel_M12", "RESLIP/DSNY", 100000 )
153 setParameter( "MATERIAL", "Koppel_M12", "RESLIP/DSSX", 10000 )
154 setParameter( "MATERIAL", "Koppel_M12", "RESLIP/BONDS6/SLPVAL", [ 15.41, 6.16, 0.1,
    1, 2, 10, 0.4 ] )
155 addGeometry( "Koppel_M12", "RELINE", "REBAR", [] )
156 setParameter( "GEOMET", "Koppel_M12", "REITYP", "REITRU" )
157 setParameter( "GEOMET", "Koppel_M12", "REITYP", "CIRBEA" )
158 setParameter( "GEOMET", "Koppel_M12", "CIRBEA/CIRCLE", 12)
159 setParameter( "GEOMET", "Koppel_M12", "ORIENT/ZAXIS", [ 0, 0, 1 ] )
160 setParameter( "GEOMET", "Koppel_M12", "TIPELM/SURFAC", 113.1 )
161 #assignMaterial( "Koppel_M12", "SHAPE", [ "Koppel_M12a","Koppel_M12b" ] )
162 #assignGeometry( "Koppel_M12", "SHAPE", [ "Koppel_M12a","Koppel_M12b" ] )
163 #convertToReinforcement( [ "Koppel_M12a","Koppel_M12b" ] )
164 #addElementData( "Koppel_M12" )
165 #assignElementData( "Koppel_M12", "SHAPE", [ "Koppel_M12a","Koppel_M12b" ] )
166 #setParameter( "DATA", "Koppel_M12", "INTERF", "BEAM" )
167 #setReinforcementDiscretization( [ "Koppel_M12a","Koppel_M12b" ], "ELEMENT" )
168 #setContinuousInInterfaces( [ "Koppel_M12a","Koppel_M12b" ], False )
169 setReinforcementType( "GEOMETRYREINFOSET", "Coupling Reinforcement",
    "BEAM_NUMERICAL_BOND_SLIP" )
170 setContinuousInInterfaces( "GEOMETRYREINFOSET", "Coupling Reinforcement", False )
171 assignMaterial( "Koppel_M12", "GEOMETRYREINFOSET", [ "Coupling Reinforcement"])
172 assignGeometry( "Koppel_M12", "GEOMETRYREINFOSET", [ "Coupling Reinforcement"])
173
174 #Material Reinforcement M8
175 addMaterial( "Reinfo_M8", "REINFO", "REBOND", [] )
176 setParameter( "MATERIAL", "Reinfo_M8", "REBARS/ELASTI/YOUNG", 200000 )
177 setParameter( "MATERIAL", "Reinfo_M8", "REBARS/POISON/POISON", 0.3 )
178 setParameter( "MATERIAL", "Reinfo_M8", "REBARS/PLATYP", "VMISES" )
179 setParameter( "MATERIAL", "Reinfo_M8", "REBARS/PLASTI/TRESSH", "KAPSIG" )
180 setParameter( "MATERIAL", "Reinfo_M8", "REBARS/PLASTI/KAPSIG", [ 0, 550, 0.04725,
    594 ] )
181 setParameter( "MATERIAL", "Reinfo_M8", "RESLIP/SHFTYP", "BONDS6" )
182 setParameter( "MATERIAL", "Reinfo_M8", "RESLIP/DSNY", 100000 )
183 setParameter( "MATERIAL", "Reinfo_M8", "RESLIP/DSSX", 10000 )
184 setParameter( "MATERIAL", "Reinfo_M8", "RESLIP/BONDS6/SLPVAL", [ 15.41, 6.16, 0.1,
    1, 2, 10, 0.4 ] )
185 addGeometry( "Reinfo_M8", "RELINE", "REBAR", [] )
186 setParameter( "GEOMET", "Reinfo_M8", "REITYP", "REITRU" )
187 setParameter( "GEOMET", "Reinfo_M8", "REITYP", "CIRBEA" )
188 setParameter( "GEOMET", "Reinfo_M8", "CIRBEA/CIRCLE", 8)
189 setParameter( "GEOMET", "Reinfo_M8", "ORIENT/ZAXIS", [ 0, 0, 1 ] )
190 setParameter( "GEOMET", "Reinfo_M8", "TIPELM/SURFAC", 50.3 )
191 #assignMaterial( "Reinfo_M8", "SHAPE", [
    "Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d" ] )
192 #assignGeometry( "Reinfo_M8", "SHAPE", [
    "Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d" ] )
193 #convertToReinforcement( [ "Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d"] )
194 #addElementData( "Reinfo_M8" )
195 #assignElementData( "Reinfo_M8", "SHAPE", [
    "Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d"] )
196 #setParameter( "DATA", "Reinfo_M8", "INTERF", "BEAM" )
197 #setReinforcementDiscretization( [
    "Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d" ], "ELEMENT" )
198 #setContinuousInInterfaces( [ "Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d" ],
    False )
```

```
setReinforcementType( "GEOMETRYREINFOSET", "Longitudinal Reinforcement", "BEAM_NUMERICAL_BOND_SLIP" )
setContinuousInInterfaces( "GEOMETRYREINFOSET", "Longitudinal Reinforcement", False )
assignMaterial( "Reinfo_M8", "GEOMETRYREINFOSET", [ "Longitudinal Reinforcement"])
assignGeometry( "Reinfo_M8", "GEOMETRYREINFOSET", [ "Longitudinal Reinforcement"])

#Material Reinforcement M5
addMaterial( "Reinfo_M5", "REINFO", "REBOND", [] )
setParameter( "MATERIAL", "Reinfo_M5", "REBARS/ELASTI/YOUNG", 200000 )
setParameter( "MATERIAL", "Reinfo_M5", "REBARS/POISON/POISON", 0.3 )
setParameter( "MATERIAL", "Reinfo_M5", "REBARS/PLATYP", "VMISES" )
setParameter( "MATERIAL", "Reinfo_M5", "REBARS/PLASTI/TRESSH", "KAPSIG" )
setParameter( "MATERIAL", "Reinfo_M5", "REBARS/PLASTI/KAPSIG", [ 0, 550, 0.04725, 594 ] )
setParameter( "MATERIAL", "Reinfo_M5", "RESLIP/SHFTYP", "BONDS6" )
setParameter( "MATERIAL", "Reinfo_M5", "RESLIP/DSNY", 100000 )
setParameter( "MATERIAL", "Reinfo_M5", "RESLIP/DSSX", 10000 )
setParameter( "MATERIAL", "Reinfo_M5", "RESLIP/BONDS6/SLPVAL", [ 0.63, 0.63, 0.01, 0.02, 0.03, 0.04, 0.5 ] )
addGeometry( "Reinfo_M5", "RELINE", "REBAR", [] )
setParameter( "GEOMET", "Reinfo_M5", "REITYP", "REITRU" )
setParameter( "GEOMET", "Reinfo_M5", "REITYP", "CIRBEA" )
setParameter( "GEOMET", "Reinfo_M5", "CIRBEA/CIRCLE", 5 )
setParameter( "GEOMET", "Reinfo_M5", "ORIENT/ZAXIS", [ 0, 0, 1 ] )
setParameter( "GEOMET", "Reinfo_M5", "TIPELM/SURFAC", 19.6 )

setReinforcementType( "GEOMETRYREINFOSET", "Lattice girders", "BEAM_NUMERICAL_BOND_SLIP" )
setContinuousInInterfaces( "GEOMETRYREINFOSET", "Lattice girders", True )
assignMaterial( "Reinfo_M5", "GEOMETRYREINFOSET", [ "Lattice girders" ] )
assignGeometry( "Reinfo_M5", "GEOMETRYREINFOSET", [ "Lattice girders" ] )

# setParameter( "DATA", "Bar", "NUMINT", [ "GAUSS" ] )
# setParameter( "DATA", "Bar", "INTEGR", "HIGH" )

#Composed Line
#addGeometry( "Composed Line", "LINE", "COMLIN", [] )
#setParameter( "GEOMET", "Composed Line", "DISTAN/THICK", 250 )
#setElementClassType( "SHAPE", [ "Composed Line" ], "COMLIN" )
#assignGeometry( "Composed Line", "SHAPE", [ "Composed Line" ] )
#resetElementData( "SHAPE", [ "Composed Line" ] )

#Tyings
createPointTying( "Tying", "Tying Top" )
setParameter( "GEOMETRYTYING", "Tying", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYTYING", "Tying", "TRANSL", [ 1, 1, 1 ] )
setParameter( "GEOMETRYTYING", "Tying", "ROTATI", [ 1, 1, 1 ] )
attachTo( "GEOMETRYTYING", "Tying", "SLAVE", "LG1-l", [ [ XM, YM, 165.5 ] ] )
attachTo( "GEOMETRYTYING", "Tying", "SLAVE", "LG1-r", [ [ XM, YM, 165.5 ] ] )
attachTo( "GEOMETRYTYING", "Tying", "MASTER", "LG1-mb", [ [ XM, YM, 165.5 ] ] )

#Assign Supports
addSet( "GEOMETRYSUPPORTSET", "Supports" )

createSurfaceSupport( "Support X", "Supports" )
setParameter( "GEOMETRYSUPPORT", "Support X", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Support X", "TRANSL", [ 1, 0, 0 ] )
setParameter( "GEOMETRYSUPPORT", "Support X", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Support X", "SlabBottom", [ [ X1, 344.1438, 19.402428 ], [ X2, 344.1438, 26.097572 ] ] )
attach( "GEOMETRYSUPPORT", "Support X", "SlabTop", [ [ X1, 344.1438, 55.947461 ], [ X2, 344.1438, 59.552538 ] ] )
attach( "GEOMETRYSUPPORT", "Support X", "Pressure layer", [ [ X1, 344.1438, 146.75686 ], [ X2, 344.1438, 173.24314 ] ] )

createSurfaceSupport( "Support Y", "Supports" )
setParameter( "GEOMETRYSUPPORT", "Support Y", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Support Y", "TRANSL", [ 0, 1, 0 ] )
setParameter( "GEOMETRYSUPPORT", "Support Y", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Support Y", "SlabBottom", [ [ 344.1438, Y1, 19.402428 ], [ 344.1438, Y2, 26.097572 ] ] )
```

```
attach( "GEOMETRYSUPPORT", "Support Y", "SlabTop", [ [ 344.1438, Y1, 55.947461 ], [ 344.1438, Y2, 59.552538 ] ] )
attach( "GEOMETRYSUPPORT", "Support Y", "Pressure layer", [ [ 344.1438, Y1, 146.75686 ], [ 344.1438, Y2, 173.24314 ] ] )

createSurfaceSupport( "Support Z", "Supports" )
setParameter( "GEOMETRYSUPPORT", "Support Z", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Support Z", "TRANSL", [ 0, 0, 1 ] )
setParameter( "GEOMETRYSUPPORT", "Support Z", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Support Z", "Pressure layer", [ [ XM, YM, 250 ] ] )

createSurfaceSupport( "Deform", "Supports" )
setParameter( "GEOMETRYSUPPORT", "Deform", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Deform", "TRANSL", [ 0, 0, 1 ] )
setParameter( "GEOMETRYSUPPORT", "Deform", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Deform", "SlabBottom", [ [ XM, YM, 0 ] ] )

createPointSupport( "Support Y beam", "Supports" )
setParameter( "GEOMETRYSUPPORT", "Support Y beam", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Support Y beam", "TRANSL", [ 0, 1, 0 ] )
setParameter( "GEOMETRYSUPPORT", "Support Y beam", "ROTATI", [ 1, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Support Y beam", "LG1-l", [ [ 260, Y1, 45.5 ], [ 260, Y2, 45.5 ] ] )
attach( "GEOMETRYSUPPORT", "Support Y beam", "LG1-r", [ [ 340, Y1, 45.5 ], [ 340, Y2, 45.5 ] ] )
attach( "GEOMETRYSUPPORT", "Support Y beam", "LG1-mb", [ [ XM, Y1, 165.5 ], [ XM, Y2, 165.5 ] ] )

#Assign Loads
addSet( "GEOMETRYLOADSET", "Load" )

# createLineLoad( "Load", "Load" )
# setParameter( "GEOMETRYLOAD", "Load", "LODTYP", "DEFORM" )
# setParameter( "GEOMETRYLOAD", "Load", "DEFORM/TR/VALUE", 0.1 )
# setParameter( "GEOMETRYLOAD", "Load", "DEFORM/TR/DIRECT", 1 )
# attach( "GEOMETRYLOAD", "Load", "Pressure Layer", [[ 200, 100, 250 ]] )

createSurfaceLoad( "Load", "Load" )
setParameter( "GEOMETRYLOAD", "Load", "LODTYP", "DEFORM" )
setParameter( "GEOMETRYLOAD", "Load", "DEFORM/TR/VALUE", -0.1 )
setParameter( "GEOMETRYLOAD", "Load", "DEFORM/TR/DIRECT", 3 )
attach( "GEOMETRYLOAD", "Load", "SlabBottom", [ [ XM, YM, 0 ] ] )

#Set mesh properties
setElementSize( [ "SlabTop" ], 10, -1, True )
clearMesherType( [ "SlabTop" ] )
setElementSize( [ "Pressure layer", "SlabBottom" ], 20, -1, True )
clearMesherType( [ "Pressure layer", "SlabBottom" ] )
generateMesh( [] )
hideView( "GEOM" )
showView( "MESH" )

#Analysis
addAnalysis( "Analysis1" )
addAnalysisCommand( "Analysis1", "NONLIN", "Structural nonlinear" )
renameAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "EXECUT(1)", "Load" )
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/STEPTY", "AUTOMA" )
# setAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "EXECUT(1)/LOAD/STEPS/AUTOMA/SIZES", 250 )
# setAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "EXECUT(1)/LOAD/STEPS/AUTOMA/MAXSIZ", 0.01 )
# setAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "EXECUT(1)/LOAD/STEPS/AUTOMA/MINSIZ", 0.00001 )
# setAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "EXECUT(1)/LOAD/STEPS/AUTOMA/CUTBCK", 0.25 )
```

```
# setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/STEPS/AUTOMA/MAXSTP", 100 )
# setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/STEPS/AUTOMA/ARCLEN", True )
# addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/STEPS/AUTOMA/ARCLEN/REGULA/SET" )
# setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/STEPS/AUTOMA/ARCLEN/REGULA/SET(1)/NODES(1)/RNGNRS", "1-9361" )
# setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/STEPS/AUTOMA/ARCLEN/REGULA/SET(1)/DIRECT", 2 )
setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/STEPS/EXPLIC/SIZES", "0.4(50)" )
# setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN", True )
# addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN/REGULA/SET" )
# setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN/REGULA/SET(1)/NODES(1)/RNGNRS", "3 4 9 10 41-96
237-242 349-354 569-2000(53) 2001-2113 2133-2653(20) 3541-9848(53) 9992 10047-10109
10163 10202 10223-10227 10281-10283 10323 10324 10364 10365 10405 10406 10446 10447
10487 10488 10528 10529 10569 10570 10610 10611 10651 10652 10692 10693 10733 10734
10774 10775 10815 10816 10856 10857 10897 10898 10938 10939 10979 10980 11020 11021
11061 11062 11102 11103 11143 11144 11184 11185 11225 11226 11266 11267 11307 11308
11348 11349 11389 11390 11430 11592-11847 12237 12239 12344 12346 12451 12453 12558
12560 12665 12667 12772 12774 12879 12881 12986 12988 13093 13095 13200 13202 13307
13309 13414 13416 13521 13523 13628 13630 13735 13737 13842 13844 13949 13951 14056
14058 14163 14165 14270 14272 14377 14379 14484 14486 14591 14593 14698 14700 14805
14807 14912 14914 15019 15021 15126 15128 15181-15183 15289 15290 15396 15397 15503
15504 15610 15663 15664 15770 15771 15877 15878 15984 15985 16091 17825-17827
17985-17987 18145-18147 18305-18307 18465 18518-18520 18678-18680 18838-18840
18998-19000 19158 19211-19213 19371-19373 19531-19533 19691-19693 19851 19904-19906
20064-20066 20224-20226 20384-20386 20544 20597-20599 20757-20759 20917-20919
21077-21079 21237 21290-21292 21450-21452 21610-21612 21770-21772 21930 21983-21985
22143-22145 22303-22305 22463-22465 22623 22676-22678 22836-22838 22996-22998
23156-23158 23316 23369-23371 23529-23531 23689-23691 23849-23851 24009 24062-24064
24222-24224 24382-24384 24542-24544 24702 24755-24757 24915-24917 25075-25077
25235-25237 25395 25448-25450 25608-25610 25768-25770 25928-25930 26088 26141-26143
26301-26303 26461-26463 26621-26623 26781 26834-26836 26994-26996 27154-27156
27314-27316 27474 27527-27529 27687-27689 27847-27849 28007-28009 28167 28220-28222
28380-28382 28540-28542 28700-28702 28860 28913-28915 29073-29075 29233-29235
29393-29395 29553 29606-29608 29766-29768 29926-29928 30086-30088 30246 30299-30301
30459-30461 30619-30621 30779-30781 30939 30992-30994 31152-31154 31312-31314
31472-31474 31632 31685-31687 31845-31847 32005-32007 32165-32167 32325 32378-32380
32538-32540 32698-32700 32858-32860 33018 33071-33073 33231-33233 33391-33393
33551-33553 33711 33764-33766 33924-33926 34084-34086 34244-34246 34404 34457-34459
34617-34619 34777-34779 34937-34939 35097 35150-35152 35310-35312 35470-35472
35630-35632 35790 35843-35845 36003-36005 36163-36165 36323-36325 36483 36536-36538
36696-36698 36856-36858 37016-37018 37176-37441(53) 37443 37445-37472 37501 37554
37659-39090(53) 39091 39095 39183-39210 39262 39263 39315 39316 39611-40772(43)
40773-40801 40854-42338(53) 42420-42449 42503 42504 42548 42551-42579 42650 42704
42705 42749 42844 42846 42931 42933 43018 43020 43105 43107 43192 43194 43279 43281
43366 43368 43453 43455 43540 43542 43627 43629 43714 43716 43801 43803 43888 43890
43975 43977 44062 44064 44149 44151 44236 44238 44323 44325 44410 44412 44497 44499
44584 44586 44671 44673 44758 44760 44845 44847 44932 44934 45019 45021 45106 45108
45193 45195 45238-45323 45950 45951 46057 46110 46111 46217 46374 46376 46481 46483
46588 46590 46695 46697 46802 46804 46909 46911 47016 47018 47123 47125 47230 47232
47337 47339 47444 47446 47551 47553 47658 47660 47765 47767 47872 47874 47979 47981
48086 48088 48193 48195 48300 48302 48407 48409 48514 48516 48621 48623 48728 48730
48835 48837 48942 48944 49049 49051 49156 49158 49263 49265 49318-49321 49479
49532-49534 49692 49745-49747 49905 49958-49960 50118 50171-50173 50331 50384-50386
50544 50597-50599 50757 50810-50812 50970 51023-51025 51183 51236-51238 51396
51449-51451 51609 51662-51664 51822 51875-51877 52035 52088-52090 52248 52301-52303
52461 52514-52516 52674 52727-52729 52887 52940-52942 53100 53153-53155 53313
53366-53368 53526 53579-53581 53739 53792-53794 53952 54005-54007 54165 54218-54220
54378 54431-54433 54591 54644-54646 54804 54857-54859 55017 55070-55072 55230 55283
57181 57228-57230 57269-57271 57310-57312 57351-57353 57392-57394 57433-57435
57474-57476 57583-57585 57692-57694 57801-57803 57910-57912 58019-58021 58128-58130
58237-58239 58346-58348 58455-58457 58564-58566 58673-58675 58782-58784 58891-58893
59000-59002 59109-59111 59218-59220" )
# setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
"EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN/REGULA/SET(1)/DIRECT", 3 )
```

```
330 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/MAXITE", 25 )
331 #setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/LINESE", True )
332 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/ENERGY", True )
333 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/DISPLA", False )
334 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/FORCE", True )
335 #setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/ENERGY/TOLCON", 0.001 )
336 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/ENERGY/NOCONV", "CONTIN" )
337 #setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/FORCE/TOLCON", 0.01 )
338 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/FORCE/NOCONV", "CONTIN" )
339 #setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/METHOD/METNAM", "SECANT" )
340
341 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "OUTPUT(1)/SELTYP",
    "USER" )
342 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "OUTPUT(1)/USER" )
343
344 ## OUTPUT ##
345 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/DISPLA(1)/TOTAL" )
346
347 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/FORCE(1)/REACTI" )
348
349 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(1)/TOTAL/GREEN" )
350 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(2)/TOTAL/GREEN/PRINCI" )
351 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(3)/CRKWDT" )
352 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(4)/CRACK/GREEN" )
353 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(5)/CRKWDT/GREEN/PRINCI" )
354 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(6)/TOTAL/TRACTI/LOCAL" )
355 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(7)/TOTAL/GREEN/LOCAL" )
356
357 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(1)/TOTAL/CAUCHY" )
358 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(2)/TOTAL/CAUCHY/PRINCI" )
359 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(3)/TOTAL/FORCE/LOCAL" )
360 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(4)/TOTAL/MOMENT/LOCAL" )
361 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(5)/TOTAL/TRACTI/LOCAL" )
362 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(6)/TOTAL/CAUCHY/LOCAL" )
363
364 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(1)/TOTAL/GREEN/GLOBAL/LOCATI", "INTPNT" )
365 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(1)/TOTAL/CAUCHY/GLOBAL/LOCATI", "INTPNT" )
366 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(6)/TOTAL/CAUCHY/LOCAL/LOCATI", "INTPNT" )
367 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(7)/TOTAL/GREEN/LOCAL/LOCATI", "INTPNT" )
368
369 #Run Analysis
370 # runSolver( [] )
```

```
371  # showView( "RESULT" )
372  fitAll( )
373
374  renameViewSettingsDirectory( "view setting", "RESULT/CONTOU/CLIPS/PLANES(1)",
     "X=260" )
375  setViewSettingValue( "view setting", "RESULT/CONTOU/CLIPS/PLANES(1)/LOCATI", "260
     0.000000 0.000000" )
376  renameViewSettingsDirectory( "view setting", "RESULT/CONTOU/CLIPS/PLANES(2)",
     "X=340" )
377  setViewSettingValue( "view setting", "RESULT/CONTOU/CLIPS/PLANES(2)/LOCATI", "340
     0.000000 0.000000" )
378  renameViewSettingsDirectory( "view setting", "RESULT/CONTOU/CLIPS/PLANES(4)", "Y=50" )
379  setViewSettingValue( "view setting", "RESULT/CONTOU/CLIPS/PLANES(4)/NORMAL",
     "0.00000 1.000000 0.000000" )
380  setViewSettingValue( "view setting", "RESULT/CONTOU/CLIPS/PLANES(4)/LOCATI", "0
     100.000000 0.000000" )
381
382  #setViewSettingValue( "view setting", "RESULT/CONTOU/AUTRNG", "LIMITS" )
383  #setViewSettingValue( "view setting", "RESULT/CONTOU/LIMITS/MINVAL", -600 )
384  #setViewSettingValue( "view setting", "RESULT/CONTOU/LIMITS/MAXVAL", 600 )
385  #setViewSettingValue( "view setting", "RESULT/CONTOU/EQUIDI/NUMBER", 11 )
386
```

E4 - Lattice girder model line-interface

```
newProject( "D:/Afstuderen/Python/3D/DIANA Output/ Lattice girder lineinterface", 10 )
setModelAnalysisAspects( [ "STRUCT" ] )
setModelDimension( "3D" )
setDefaultMeshOrder( "QUADRATIC" )
setDefaultMesherType( "HEXQUAD" )
setDefaultMidSideNodeLocation( "ONSHAP" )
setUnit( "LENGTH", "MM" )
setUnit( "FORCE", "N" )
setUnit( "ANGLE", "DEGREE" )

#Parameters
X1=200
X2=400
LX=X2-X1
Y1=0
Y2=200
LY=Y2-Y1
XM=X1+(X2-X1)/2
YM=Y1+(Y2-Y1)/2

#Thickness layers
T1 = 45.5
T2 = 70 - T1
T3 = 180

#Create Shapes
createBlock( "SlabBottom", [ X1, Y1, 0 ], [ LX, LY, T1 ] )
createBlock( "SlabTop", [ X1, Y1, T1 ], [ LX, LY, T2 ] )
createBlock( "Pressure layer", [ X1, Y1, T1+T2 ], [ LX, LY, T3 ] )

#Create Reinforcement
createLine( "Koppel_M12a", [ X1, 30, 76 ], [ X2, 30, 76 ] )
createLine( "Koppel_M12b", [ X1, 180, 76 ], [ X2, 180, 76 ] )

createLine( "Reinfo_M8a", [ X1, 2.5, 35 ], [ X2, 2.5, 35 ] )
createLine( "Reinfo_M8b", [ X1, 67.5, 35 ], [ X2, 67.5, 35 ] )
createLine( "Reinfo_M8c", [ X1, 132.5, 35 ], [ X2, 132.5, 35 ] )
createLine( "Reinfo_M8d", [ X1, 197.5, 35 ], [ X2, 197.5, 35 ] )

createPolyline( "LG1-mb", [[ XM, 0, 165.5],[ XM, YM, 165.5],[ XM, Y2, 165.5]], False )
createPolyline( "LG1-l", [ [260, Y1, 45.5],[XM, Y1+100, 165.5],[260, Y1+200, 45.5]
],False )
createPolyline( "LG1-r", [ [340, Y1, 45.5],[XM, Y1+100, 165.5],[340, Y1+200, 45.5]
],False )
createLine( "LG1-or",  [340, Y1, 45.5],[340, Y1+200, 45.5]  )
createLine( "LG1-ol",  [260, Y1, 45.5],[260, Y1+200, 45.5]  )

#Imprint
imprintIntersection( "SlabBottom", "LG1-or", True )
imprintIntersection( "SlabBottom", "LG1-ol", True )
imprintIntersection( "SlabTop", "LG1-or", True )
imprintIntersection( "SlabTop", "LG1-ol", True )


#Regroup Material Parts
addSet( "GEOMETRYREINFOSET", "Coupling Reinforcement" )
moveToShapeSet( [ "Koppel_M12a","Koppel_M12b" ], "Coupling Reinforcement" )
addSet( "GEOMETRYREINFOSET", "Longitudinal Reinforcement" )
moveToShapeSet( [ "Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d"],
"Longitudinal Reinforcement" )
addSet( "GEOMETRYREINFOSET", "Lattice girders" )
moveToShapeSet( [ "LG1-mb", "LG1-l", "LG1-r"], "Lattice girders" )
addSet( "SHAPESET", "Reinfo Bar" )
moveToShapeSet( [ "LG1-or", "LG1-ol"], "Reinfo Bar" )
rename( "SHAPESET", "Shapes", "Concrete" )

#Material Precast concrete
addMaterial( "Precast concrete", "CONCR", "TSCR", [] )
setParameter( "MATERIAL", "Precast concrete", "LINEAR/ELASTI/YOUNG", 32000 )
setParameter( "MATERIAL", "Precast concrete", "LINEAR/ELASTI/POISON", 0.2 )
setParameter( "MATERIAL", "Precast concrete", "LINEAR/MASS/DENSIT", 2.4e-09 )
```

```
69  setParameter( "MATERIAL", "Precast concrete", "MODTYP/TOTCRK", "ROTATE" )
70  setParameter( "MATERIAL", "Precast concrete", "TENSIL/TENCRV", "HORDYK" )
71  setParameter( "MATERIAL", "Precast concrete", "TENSIL/TENSTR", 2.9 )
72  setParameter( "MATERIAL", "Precast concrete", "TENSIL/GF1", 0.1405 )
73  setParameter( "MATERIAL", "Precast concrete", "TENSIL/CBSPEC", "ROTS" )
74  setParameter( "MATERIAL", "Precast concrete", "TENSIL/POISRE/POIRED", "DAMAGE" )
75  setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMCRV", "MC2010" )
76  setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMSTR", 38 )
77  setParameter( "MATERIAL", "Precast concrete", "COMPRS/EPSC1", 0.0023 )
78  setParameter( "MATERIAL", "Precast concrete", "COMPRS/EPSCU", 0.0035 )
79  setParameter( "MATERIAL", "Precast concrete", "COMPRS/CONFIN/CNFCRV", "NONE" )
80  # setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMCRV", "CONSTA" )
81  # setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMSTR", 38 )
82
83  addGeometry( "Precast concrete", "SOLID", "STRSOL", [] )
84  setParameter( "GEOMET", "Precast concrete", "LOCAXS", True )
85  setParameter( "GEOMET", "Precast concrete", "LOCAXS/XAXIS", [ 1, 0, 0 ] )
86  setParameter( "GEOMET", "Precast concrete", "LOCAXS/YAXIS", [ 0, 1, 0 ] )
87
88  setElementClassType( "SHAPE", [ "SlabBottom", "SlabTop" ], "STRSOL" )
89  assignMaterial( "Precast concrete", "SHAPE", [ "SlabBottom", "SlabTop" ] )
90  assignGeometry( "Precast concrete", "SHAPE", [ "SlabBottom", "SlabTop" ] )
91
92  #Material Pressure layer
93  addMaterial( "Pressure layer", "CONCR", "TSCR", [] )
94  setParameter( "MATERIAL", "Pressure layer", "LINEAR/ELASTI/YOUNG", 32000 )
95  setParameter( "MATERIAL", "Pressure layer", "LINEAR/ELASTI/POISON", 0.2 )
96  setParameter( "MATERIAL", "Pressure layer", "LINEAR/MASS/DENSIT", 2.4e-09 )
97  setParameter( "MATERIAL", "Pressure layer", "MODTYP/TOTCRK", "ROTATE" )
98  setParameter( "MATERIAL", "Pressure layer", "TENSIL/TENCRV", "HORDYK" )
99  setParameter( "MATERIAL", "Pressure layer", "TENSIL/TENSTR", 2.9 )
100 setParameter( "MATERIAL", "Pressure layer", "TENSIL/GF1", 0.1405 )
101 setParameter( "MATERIAL", "Pressure layer", "TENSIL/CBSPEC", "ROTS" )
102 setParameter( "MATERIAL", "Pressure layer", "TENSIL/POISRE/POIRED", "DAMAGE" )
103 setParameter( "MATERIAL", "Pressure layer", "COMPRS/COMCRV", "MC2010" )
104 setParameter( "MATERIAL", "Pressure layer", "COMPRS/COMSTR", 38 )
105 setParameter( "MATERIAL", "Pressure layer", "COMPRS/EPSC1", 0.0023 )
106 setParameter( "MATERIAL", "Pressure layer", "COMPRS/EPSCU", 0.0035 )
107 setParameter( "MATERIAL", "Pressure layer", "COMPRS/CONFIN/CNFCRV", "NONE" )
108 #setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMCRV", "CONSTA" )
109 #setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMSTR", 38 )
110
111 addGeometry( "Pressure layer", "SOLID", "STRSOL", [] )
112 setParameter( "GEOMET", "Pressure layer", "LOCAXS", True )
113 setParameter( "GEOMET", "Pressure layer", "LOCAXS/XAXIS", [ 1, 0, 0 ] )
114 setParameter( "GEOMET", "Pressure layer", "LOCAXS/YAXIS", [ 0, 1, 0 ] )
115
116 setElementClassType( "SHAPE", [ "Pressure layer" ], "STRSOL" )
117 assignMaterial( "Precast concrete", "SHAPE", [ "Pressure layer"] )
118 assignGeometry( "Precast concrete", "SHAPE", [ "Pressure layer"] )
119
120 #Concrete Interface
121 addMaterial( "Interface Concrete", "INTERF", "FRICTI", [] )
122 setParameter( "MATERIAL", "Interface Concrete", "LINEAR/ELAS6/DSNZ", 60000 )
123 setParameter( "MATERIAL", "Interface Concrete", "LINEAR/ELAS6/DSSX", 6000 )
124 setParameter( "MATERIAL", "Interface Concrete", "LINEAR/ELAS6/DSSY", 6000 )
125 setParameter( "MATERIAL", "Interface Concrete", "COULOM/OPNTYP", "GAP" )
126 setParameter( "MATERIAL", "Interface Concrete", "COULOM/GAPVAL", 0.5 )
127 setParameter( "MATERIAL", "Interface Concrete", "COULOM/COHESI", 0.5 )
128 setParameter( "MATERIAL", "Interface Concrete", "COULOM/PHI", 30 )
129 setParameter( "MATERIAL", "Interface Concrete", "COULOM/PSI", 30 )
130 setParameter( "MATERIAL", "Interface Concrete", "COULOM/FRCHSR/MODE2", 0 )
131
132
133 #addMaterial( "Interface no tension", "INTERF", "NONLIF", [] )
134 #setParameter( "MATERIAL", "Interface no tension", "LINEAR/ELAS6/DSSY", 10000 )
135 #setParameter( "MATERIAL", "Interface no tension", "LINEAR/ELAS6/DSSX", 10000 )
136 #setParameter( "MATERIAL", "Interface no tension", "LINEAR/ELAS6/DSNZ", 100000 )
137
138 addGeometry( "Interface Concrete", "SHEET", "STPLIF", [] )
139 setParameter( "GEOMET", "Interface Concrete", "LOCAXS", True )
```

```
140  setParameter( "GEOMET", "Interface Concrete", "LOCAXS/XAXIS", [ 1, 0, 0 ] )
141
142  addElementData( "Interface Concrete" )
143  createConnection( "Interface Concrete", "INTER", "SHAPEFACE", "SHAPEFACE" )
144  setParameter( "GEOMETRYCONNECTION", "Interface Concrete", "MODE", "CLOSED" )
145  setElementClassType( "GEOMETRYCONNECTION", "Interface Concrete", "STPLIF" )
146  assignMaterial( "Interface Concrete", "GEOMETRYCONNECTION", "Interface Concrete" )
147  #assignMaterial( "Interface no tension", "GEOMETRYCONNECTION", "Interface Concrete" )
148  assignGeometry( "Interface Concrete", "GEOMETRYCONNECTION", "Interface Concrete" )
149  assignElementData( "Interface Concrete", "GEOMETRYCONNECTION", "Interface Concrete" )
150  setParameter( "GEOMETRYCONNECTION", "Interface Concrete", "FLIP", False )
151  attachTo( "GEOMETRYCONNECTION", "Interface Concrete", "SOURCE", "SlabTop", [ [ XM,
     YM, 70 ] ] )
152  attachTo( "GEOMETRYCONNECTION", "Interface Concrete", "TARGET", "Pressure layer", [
     [ XM, YM, 70 ] ] )
153
154  #Material Reinforcement Koppel_M12
155  addMaterial( "Koppel_M12", "REINFO", "REBOND", [ ] )
156  setParameter( "MATERIAL", "Koppel_M12", "REBARS/ELASTI/YOUNG", 200000 )
157  setParameter( "MATERIAL", "Koppel_M12", "REBARS/POISON/POISON", 0.3 )
158  setParameter( "MATERIAL", "Koppel_M12", "REBARS/PLATYP", "VMISES" )
159  setParameter( "MATERIAL", "Koppel_M12", "REBARS/PLASTI/TRESSH", "KAPSIG" )
160  setParameter( "MATERIAL", "Koppel_M12", "REBARS/PLASTI/KAPSIG", [ 0, 550, 0.04725,
     594 ] )
161  setParameter( "MATERIAL", "Koppel_M12", "RESLIP/SHFTYP", "BONDS6" )
162  setParameter( "MATERIAL", "Koppel_M12", "RESLIP/DSNY", 100000 )
163  setParameter( "MATERIAL", "Koppel_M12", "RESLIP/DSSX", 10000 )
164  setParameter( "MATERIAL", "Koppel_M12", "RESLIP/BONDS6/SLPVAL", [ 15.41, 6.16, 0.1,
     1, 2, 10, 0.4 ] )
165  addGeometry( "Koppel_M12", "RELINE", "REBAR", [] )
166  setParameter( "GEOMET", "Koppel_M12", "REITYP", "REITRU" )
167  setParameter( "GEOMET", "Koppel_M12", "REITYP", "CIRBEA" )
168  setParameter( "GEOMET", "Koppel_M12", "CIRBEA/CIRCLE", 12)
169  setParameter( "GEOMET", "Koppel_M12", "ORIENT/ZAXIS", [ 0, 0, 1 ] )
170  setParameter( "GEOMET", "Koppel_M12", "TIPELM/SURFAC", 113.1 )
171  #assignMaterial( "Koppel_M12", "SHAPE", [ "Koppel_M12a","Koppel_M12b" ] )
172  #assignGeometry( "Koppel_M12", "SHAPE", [ "Koppel_M12a","Koppel_M12b" ] )
173  #convertToReinforcement( [ "Koppel_M12a","Koppel_M12b" ] )
174  #addElementData( "Koppel_M12" )
175  #assignElementData( "Koppel_M12", "SHAPE", [ "Koppel_M12a","Koppel_M12b" ] )
176  #setParameter( "DATA", "Koppel_M12", "INTERF", "BEAM" )
177  #setReinforcementDiscretization( [ "Koppel_M12a","Koppel_M12b" ], "ELEMENT" )
178  #setContinuousInInterfaces( [ "Koppel_M12a","Koppel_M12b" ], False )
179  setReinforcementType( "GEOMETRYREINFOSET", "Coupling Reinforcement",
     "BEAM_NUMERICAL_BOND_SLIP" )
180  setContinuousInInterfaces( "GEOMETRYREINFOSET", "Coupling Reinforcement", False )
181  assignMaterial( "Koppel_M12", "GEOMETRYREINFOSET", [ "Coupling Reinforcement"])
182  assignGeometry( "Koppel_M12", "GEOMETRYREINFOSET", [ "Coupling Reinforcement"])
183
184  #Material Reinforcement M8
185  addMaterial( "Reinfo_M8", "REINFO", "REBOND", [] )
186  setParameter( "MATERIAL", "Reinfo_M8", "REBARS/ELASTI/YOUNG", 200000 )
187  setParameter( "MATERIAL", "Reinfo_M8", "REBARS/POISON/POISON", 0.3 )
188  setParameter( "MATERIAL", "Reinfo_M8", "REBARS/PLATYP", "VMISES" )
189  setParameter( "MATERIAL", "Reinfo_M8", "REBARS/PLASTI/TRESSH", "KAPSIG" )
190  setParameter( "MATERIAL", "Reinfo_M8", "REBARS/PLASTI/KAPSIG", [ 0, 550, 0.04725,
     594 ] )
191  setParameter( "MATERIAL", "Reinfo_M8", "RESLIP/SHFTYP", "BONDS6" )
192  setParameter( "MATERIAL", "Reinfo_M8", "RESLIP/DSNY", 100000 )
193  setParameter( "MATERIAL", "Reinfo_M8", "RESLIP/DSSX", 10000 )
194  setParameter( "MATERIAL", "Reinfo_M8", "RESLIP/BONDS6/SLPVAL", [ 15.41, 6.16, 0.1,
     1, 2, 10, 0.4 ] )
195  addGeometry( "Reinfo_M8", "RELINE", "REBAR", [] )
196  setParameter( "GEOMET", "Reinfo_M8", "REITYP", "REITRU" )
197  setParameter( "GEOMET", "Reinfo_M8", "REITYP", "CIRBEA" )
198  setParameter( "GEOMET", "Reinfo_M8", "CIRBEA/CIRCLE", 8)
199  setParameter( "GEOMET", "Reinfo_M8", "ORIENT/ZAXIS", [ 0, 0, 1 ] )
200  setParameter( "GEOMET", "Reinfo_M8", "TIPELM/SURFAC", 50.3 )
201  #assignMaterial( "Reinfo_M8", "SHAPE", [
     "Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d" ] )
202  #assignGeometry( "Reinfo_M8", "SHAPE", [
```

```
"Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d" ] )
#convertToReinforcement( [ "Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d"] )
#addElementData( "Reinfo_M8" )
#assignElementData( "Reinfo_M8", "SHAPE", [
"Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d"] )
#setParameter( "DATA", "Reinfo_M8", "INTERF", "BEAM" )
#setReinforcementDiscretization( [
"Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d" ], "ELEMENT" )
#setContinuousInInterfaces( [ "Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d" ],
False )
setReinforcementType( "GEOMETRYREINFOSET", "Longitudinal Reinforcement",
"BEAM_NUMERICAL_BOND_SLIP" )
setContinuousInInterfaces( "GEOMETRYREINFOSET", "Longitudinal Reinforcement", False )
assignMaterial( "Reinfo_M8", "GEOMETRYREINFOSET", [ "Longitudinal Reinforcement"])
assignGeometry( "Reinfo_M8", "GEOMETRYREINFOSET", [ "Longitudinal Reinforcement"])

#Material Reinforcement M5
addMaterial( "Reinfo_M5", "REINFO", "REBOND", [] )
setParameter( "MATERIAL", "Reinfo_M5", "REBARS/ELASTI/YOUNG", 200000 )
setParameter( "MATERIAL", "Reinfo_M5", "REBARS/POISON/POISON", 0.3 )
setParameter( "MATERIAL", "Reinfo_M5", "REBARS/PLATYP", "VMISES" )
setParameter( "MATERIAL", "Reinfo_M5", "REBARS/PLASTI/TRESSH", "KAPSIG" )
setParameter( "MATERIAL", "Reinfo_M5", "REBARS/PLASTI/KAPSIG", [ 0, 550, 0.04725,
594 ] )
setParameter( "MATERIAL", "Reinfo_M5", "RESLIP/SHFTYP", "BONDS6" )
setParameter( "MATERIAL", "Reinfo_M5", "RESLIP/DSNY", 100000 )
setParameter( "MATERIAL", "Reinfo_M5", "RESLIP/DSSX", 10000 )
setParameter( "MATERIAL", "Reinfo_M5", "RESLIP/BONDS6/SLPVAL", [ 0.63, 0.63, 0.01,
0.02, 0.03, 0.04, 0.5 ] )
addGeometry( "Reinfo_M5", "RELINE", "REBAR", [] )
setParameter( "GEOMET", "Reinfo_M5", "REITYP", "REITRU" )
setParameter( "GEOMET", "Reinfo_M5", "REITYP", "CIRBEA" )
setParameter( "GEOMET", "Reinfo_M5", "CIRBEA/CIRCLE", 5 )
setParameter( "GEOMET", "Reinfo_M5", "ORIENT/ZAXIS", [ 0, 0, 1 ] )
setParameter( "GEOMET", "Reinfo_M5", "TIPELM/SURFAC", 19.6 )

setReinforcementType( "GEOMETRYREINFOSET", "Lattice girders",
"BEAM_NUMERICAL_BOND_SLIP" )
setContinuousInInterfaces( "GEOMETRYREINFOSET", "Lattice girders", True )
assignMaterial( "Reinfo_M5", "GEOMETRYREINFOSET", [ "Lattice girders" ] )
assignGeometry( "Reinfo_M5", "GEOMETRYREINFOSET", [ "Lattice girders" ] )


addMaterial( "Reinfo Bar", "MCSTEL", "TRESCA", [] )
setParameter( "MATERIAL", "Reinfo Bar", "LINEAR/ELASTI/YOUNG", 200000 )
setParameter( "MATERIAL", "Reinfo Bar", "LINEAR/ELASTI/POISON", 0.3 )
setParameter( "MATERIAL", "Reinfo Bar", "LINEAR/MASS/DENSIT", 7.8e-09 )
setParameter( "MATERIAL", "Reinfo Bar", "TREPLA/YIELD", "VMISES" )
setParameter( "MATERIAL", "Reinfo Bar", "TREPLA/TRESSH", "KAPSIG" )
setParameter( "MATERIAL", "Reinfo Bar", "TREPLA/KAPSIG", [ 0, 550, 0.04725, 594 ] )

addGeometry( "Reinfo Bar", "LINE", "CLS3B3", [] )
setParameter( "GEOMET", "Reinfo Bar", "SHAPE/BESHAP", "CIRCLE" )
setParameter( "GEOMET", "Reinfo Bar", "SHAPE/ROUND/CIRCLE", 5 )
setParameter( "GEOMET", "Reinfo Bar", "LOCAXS", True )
setParameter( "GEOMET", "Reinfo Bar", "LOCAXS/ZAXIS", [ 0, 0, 1 ] )
setElementClassType( "SHAPE", [ "LG1-ol" , "LG1-or" ], "CLS3B3" )
assignMaterial( "Reinfo Bar", "SHAPE", [ "LG1-or", "LG1-ol" ] )
assignGeometry( "Reinfo Bar", "SHAPE", [ "LG1-or", "LG1-ol" ] )

# setParameter( "DATA", "Bar", "NUMINT", [ "GAUSS" ] )
# setParameter( "DATA", "Bar", "INTEGR", "HIGH" )

#Composed Line
#addGeometry( "Composed Line", "LINE", "COMLIN", [] )
#setParameter( "GEOMET", "Composed Line", "DISTAN/THICK", 250 )
#setElementClassType( "SHAPE", [ "Composed Line" ], "COMLIN" )
#assignGeometry( "Composed Line", "SHAPE", [ "Composed Line" ] )
#resetElementData( "SHAPE", [ "Composed Line" ] )

#Bar Interface
```

```
addMaterial( "Interface Bar", "INTERF", "NONLIF", [] )
setParameter( "MATERIAL", "Interface Bar", "LINEAR/IFTYP", "LIN3D3" )
setParameter( "MATERIAL", "Interface Bar", "LINEAR/ELAS7/DSNY", 100000 )
setParameter( "MATERIAL", "Interface Bar", "LINEAR/ELAS7/DSSX", 10000 )
setParameter( "MATERIAL", "Interface Bar", "LINEAR/ELAS7/DSSZ", 10000 )
setParameter( "MATERIAL", "Interface Bar", "NONLIN/IFNOTE", "NOTSHR" )
addGeometry( "Interface Bar", "LINE", "LIN3DS", [] )
setParameter( "GEOMET", "Interface Bar", "THICK", 5 )
setParameter( "GEOMET", "Interface Bar", "LOCAXS", True )
setParameter( "GEOMET", "Interface Bar", "LOCAXS/ZAXIS", [ 1, 0, 0 ] )
addElementData( "Interface Bar" )

createConnection( "Interface bottom", "INTER", "SHAPEEDGE", "SHAPEEDGE" )
setParameter( "GEOMETRYCONNECTION", "Interface bottom", "MODE", "CLOSED" )
setElementClassType( "GEOMETRYCONNECTION", "Interface bottom", "LIN3DS" )
assignMaterial( "Interface Bar", "GEOMETRYCONNECTION", "Interface bottom" )
assignGeometry( "Interface Bar", "GEOMETRYCONNECTION", "Interface bottom" )
setParameter( "GEOMETRYCONNECTION", "Interface bottom", "FLIP", False )
attachTo( "GEOMETRYCONNECTION", "Interface bottom", "SOURCE", "SlabBottom", [ [ 340, 25, 45.5 ] ] )
attachTo( "GEOMETRYCONNECTION", "Interface bottom", "SOURCE", "SlabBottom", [ [ 260, 25, 45.5 ] ] )
attachTo( "GEOMETRYCONNECTION", "Interface bottom", "TARGET", "LG1-or", [ [ 260, 25, 45.5 ] ] )
attachTo( "GEOMETRYCONNECTION", "Interface bottom", "TARGET", "LG1-ol", [ [ 340, 25, 45.5 ] ] )


createConnection( "Interface top", "INTER", "SHAPEEDGE", "SHAPEEDGE" )
setParameter( "GEOMETRYCONNECTION", "Interface top", "MODE", "CLOSED" )
setElementClassType( "GEOMETRYCONNECTION", "Interface top", "LIN3DS" )
assignMaterial( "Interface Bar", "GEOMETRYCONNECTION", "Interface top" )
assignGeometry( "Interface Bar", "GEOMETRYCONNECTION", "Interface top" )
setParameter( "GEOMETRYCONNECTION", "Interface top", "FLIP", False )
attachTo( "GEOMETRYCONNECTION", "Interface top", "SOURCE", "LG1-or", [ [ 260, 25, 45.5 ] ] )
attachTo( "GEOMETRYCONNECTION", "Interface top", "SOURCE", "LG1-ol", [ [ 340, 100, 45.5 ] ] )
attachTo( "GEOMETRYCONNECTION", "Interface top", "TARGET", "SlabTop", [ [ 260, 100, 45.5 ] ] )
attachTo( "GEOMETRYCONNECTION", "Interface top", "TARGET", "SlabTop", [ [ 340, 100, 45.5 ] ] )


#Tyings
createPointTying( "Tying 1", "Tying LG" )
setParameter( "GEOMETRYTYING", "Tying 1", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYTYING", "Tying 1", "TRANSL", [ 1, 1, 1 ] )
setParameter( "GEOMETRYTYING", "Tying 1", "ROTATI", [ 1, 1, 1 ] )
attachTo( "GEOMETRYTYING", "Tying 1", "SLAVE", "LG1-l", [[ 260, 0, 45.5 ]] )
attachTo( "GEOMETRYTYING", "Tying 1", "MASTER", "LG1-ol", [[ 260, 0, 45.5 ]] )

createPointTying( "Tying 2", "Tying LG" )
setParameter( "GEOMETRYTYING", "Tying 2", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYTYING", "Tying 2", "TRANSL", [ 1, 1, 1 ] )
setParameter( "GEOMETRYTYING", "Tying 2", "ROTATI", [ 1, 1, 1 ] )
attachTo( "GEOMETRYTYING", "Tying 2", "SLAVE", "LG1-l", [[ 260, 200, 45.5 ]] )
attachTo( "GEOMETRYTYING", "Tying 2", "MASTER", "LG1-ol", [[ 260, 200, 45.5 ]] )

createPointTying( "Tying 3", "Tying LG" )
setParameter( "GEOMETRYTYING", "Tying 3", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYTYING", "Tying 3", "TRANSL", [ 1, 1, 1 ] )
setParameter( "GEOMETRYTYING", "Tying 3", "ROTATI", [ 1, 1, 1 ] )
attachTo( "GEOMETRYTYING", "Tying 3", "SLAVE", "LG1-r", [[ 340, 200, 45.5 ]] )
attachTo( "GEOMETRYTYING", "Tying 3", "MASTER", "LG1-or", [[ 340, 200, 45.5 ]] )

createPointTying( "Tying 4", "Tying LG" )
setParameter( "GEOMETRYTYING", "Tying 4", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYTYING", "Tying 4", "TRANSL", [ 1, 1, 1 ] )
setParameter( "GEOMETRYTYING", "Tying 4", "ROTATI", [ 1, 1, 1 ] )
attachTo( "GEOMETRYTYING", "Tying 4", "SLAVE", "LG1-r", [[ 340, 0, 45.5 ]] )
```

```
attachTo( "GEOMETRYTYING", "Tying 4", "MASTER", "LG1-or", [[ 340, 0, 45.5 ]] )

createPointTying( "Tying 5", "Tying LG" )
setParameter( "GEOMETRYTYING", "Tying 5", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYTYING", "Tying 5", "TRANSL", [ 1, 1, 1 ] )
setParameter( "GEOMETRYTYING", "Tying 5", "ROTATI", [ 1, 1, 1 ] )
attachTo( "GEOMETRYTYING", "Tying 5", "SLAVE", "LG1-l", [ [ XM, YM, 165.5 ] ] )
attachTo( "GEOMETRYTYING", "Tying 5", "SLAVE", "LG1-r", [ [ XM, YM, 165.5 ] ] )
attachTo( "GEOMETRYTYING", "Tying 5", "MASTER", "LG1-mb", [ [ XM, YM, 165.5 ] ] )

#Assign Supports
addSet( "GEOMETRYSUPPORTSET", "Supports" )

createSurfaceSupport( "Support X", "Supports" )
setParameter( "GEOMETRYSUPPORT", "Support X", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Support X", "TRANSL", [ 1, 0, 0 ] )
setParameter( "GEOMETRYSUPPORT", "Support X", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Support X", "SlabBottom", [ [ X1, 344.1438, 19.402428 ], [ X2, 344.1438, 26.097572 ] ] )
attach( "GEOMETRYSUPPORT", "Support X", "SlabTop", [ [ X1, 344.1438, 55.947461 ], [ X2, 344.1438, 59.552538 ] ] )
attach( "GEOMETRYSUPPORT", "Support X", "Pressure layer", [ [ X1, 344.1438, 146.75686 ], [ X2, 344.1438, 173.24314 ] ] )

createSurfaceSupport( "Support Y", "Supports" )
setParameter( "GEOMETRYSUPPORT", "Support Y", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Support Y", "TRANSL", [ 0, 1, 0 ] )
setParameter( "GEOMETRYSUPPORT", "Support Y", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Support Y", "SlabBottom", [ [ 344.1438, Y1, 19.402428 ], [ 344.1438, Y2, 26.097572 ] ] )
attach( "GEOMETRYSUPPORT", "Support Y", "SlabTop", [ [ 344.1438, Y1, 55.947461 ], [ 344.1438, Y2, 59.552538 ] ] )
attach( "GEOMETRYSUPPORT", "Support Y", "Pressure layer", [ [ 344.1438, Y1, 146.75686 ], [ 344.1438, Y2, 173.24314 ] ] )

createSurfaceSupport( "Support Z", "Supports" )
setParameter( "GEOMETRYSUPPORT", "Support Z", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Support Z", "TRANSL", [ 0, 0, 1 ] )
setParameter( "GEOMETRYSUPPORT", "Support Z", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Support Z", "Pressure layer", [ [ XM, YM, 250 ] ] )

createSurfaceSupport( "Deform", "Supports" )
setParameter( "GEOMETRYSUPPORT", "Deform", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Deform", "TRANSL", [ 0, 0, 1 ] )
setParameter( "GEOMETRYSUPPORT", "Deform", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Deform", "SlabBottom", [ [ XM, YM, 0 ] ] )

createPointSupport( "Support Y beam", "Supports" )
setParameter( "GEOMETRYSUPPORT", "Support Y beam", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Support Y beam", "TRANSL", [ 0, 1, 0 ] )
setParameter( "GEOMETRYSUPPORT", "Support Y beam", "ROTATI", [ 1, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Support Y beam", "LG1-ol", [ [ 260, Y1, 45.5 ], [ 260, Y2, 45.5 ] ] )
attach( "GEOMETRYSUPPORT", "Support Y beam", "LG1-or", [ [ 340, Y1, 45.5 ], [ 340, Y2, 45.5 ] ] )
attach( "GEOMETRYSUPPORT", "Support Y beam", "LG1-mb", [ [ XM, Y1, 165.5 ], [ XM, Y2, 165.5 ] ] )

#Assign Loads
addSet( "GEOMETRYLOADSET", "Load" )

# createLineLoad( "Load", "Load" )
# setParameter( "GEOMETRYLOAD", "Load", "LODTYP", "DEFORM" )
# setParameter( "GEOMETRYLOAD", "Load", "DEFORM/TR/VALUE", 0.1 )
# setParameter( "GEOMETRYLOAD", "Load", "DEFORM/TR/DIRECT", 1 )
# attach( "GEOMETRYLOAD", "Load", "Pressure Layer", [[ 200, 100, 250 ]] )

createSurfaceLoad( "Load", "Load" )
setParameter( "GEOMETRYLOAD", "Load", "LODTYP", "DEFORM" )
setParameter( "GEOMETRYLOAD", "Load", "DEFORM/TR/VALUE", -0.1 )
setParameter( "GEOMETRYLOAD", "Load", "DEFORM/TR/DIRECT", 3 )
```

```
391 attach( "GEOMETRYLOAD", "Load", "SlabBottom", [ [ XM, YM, 0 ] ] )
392
393 #Set mesh properties
394 setElementSize( [ "SlabTop" ], 10, -1, True )
395 clearMesherType( [ "SlabTop" ] )
396 setElementSize( [ "Pressure layer", "SlabBottom" ], 20, -1, True )
397 clearMesherType( [ "Pressure layer", "SlabBottom" ] )
398 generateMesh( [] )
399 hideView( "GEOM" )
400 showView( "MESH" )
401
402 #Analysis
403 addAnalysis( "Analysis1" )
404 addAnalysisCommand( "Analysis1", "NONLIN", "Structural nonlinear" )
405 renameAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "EXECUT(1)",
    "Load" )
406 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/LOADNR" )
407 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/LOADNR", 1 )
408 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/STEPTY", "AUTOMA" )
409 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/SIZES", 250 )
410 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/MAXSIZ", 0.01 )
411 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/MINSIZ", 0.00001 )
412 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/CUTBCK", 0.25 )
413 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/MAXSTP", 100 )
414 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/ARCLEN", True )
415 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/ARCLEN/REGULA/SET" )
416 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/ARCLEN/REGULA/SET(1)/NODES(1)/RNGNRS", "1-9361" )
417 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/ARCLEN/REGULA/SET(1)/DIRECT", 2 )
418 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/SIZES", "0.2(30)" )
419 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN", True )
420 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN/REGULA/SET" )
421 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN/REGULA/SET(1)/NODES(1)/RNGNRS", "3 4 9 10 41-96
    237-242 349-354 569-2000(53) 2001-2113 2133-2653(20) 3541-9848(53) 9992 10047-10109
    10163 10202 10223-10227 10281-10283 10323 10324 10364 10365 10405 10406 10446 10447
    10487 10488 10528 10529 10569 10570 10610 10611 10651 10652 10692 10693 10733 10734
    10774 10775 10815 10816 10856 10857 10897 10898 10938 10939 10979 10980 11020 11021
    11061 11062 11102 11103 11143 11144 11184 11185 11225 11226 11266 11267 11307 11308
    11348 11349 11389 11390 11430 11592-11847 12237 12239 12344 12346 12451 12453 12558
    12560 12665 12667 12772 12774 12879 12881 12986 12988 13093 13095 13200 13202 13307
    13309 13414 13416 13521 13523 13628 13630 13735 13737 13842 13844 13949 13951 14056
    14058 14163 14165 14270 14272 14377 14379 14484 14486 14591 14593 14698 14700 14805
    14807 14912 14914 15019 15021 15126 15128 15181-15183 15289 15290 15396 15397 15503
    15504 15610 15663 15664 15770 15771 15877 15878 15984 15985 16091 17825-17827
    17985-17987 18145-18147 18305-18307 18465 18518-18520 18678-18680 18838-18840
    18998-19000 19158 19211-19213 19371-19373 19531-19533 19691-19693 19851 19904-19906
    20064-20066 20224-20226 20384-20386 20544 20597-20599 20757-20759 20917-20919
    21077-21079 21237 21290-21292 21450-21452 21610-21612 21770-21772 21930 21983-21985
    22143-22145 22303-22305 22463-22465 22623 22676-22678 22836-22838 22996-22998
    23156-23158 23316 23369-23371 23529-23531 23689-23691 23849-23851 24009 24062-24064
    24222-24224 24382-24384 24542-24544 24702 24755-24757 24915-24917 25075-25077
    25235-25237 25395 25448-25450 25608-25610 25768-25770 25928-25930 26088 26141-26143
    26301-26303 26461-26463 26621-26623 26781 26834-26836 26994-26996 27154-27156
    27314-27316 27474 27527-27529 27687-27689 27847-27849 28007-28009 28167 28220-28222
    28380-28382 28540-28542 28700-28702 28860 28913-28915 29073-29075 29233-29235
    29393-29395 29553 29606-29608 29766-29768 29926-29928 30086-30088 30246 30299-30301
```

```
    30459-30461 30619-30621 30779-30781 30939 30992-30994 31152-31154 31312-31314
    31472-31474 31632 31685-31687 31845-31847 32005-32007 32165-32167 32325 32378-32380
    32538-32540 32698-32700 32858-32860 33018 33071-33073 33231-33233 33391-33393
    33551-33553 33711 33764-33766 33924-33926 34084-34086 34244-34246 34404 34457-34459
    34617-34619 34777-34779 34937-34939 35097 35150-35152 35310-35312 35470-35472
    35630-35632 35790 35843-35845 36003-36005 36163-36165 36323-36325 36483 36536-36538
    36696-36698 36856-36858 37016-37018 37176-37441(53) 37443 37445-37472 37501 37554
    37659-39090(53) 39091 39095 39183-39210 39262 39263 39315 39316 39611-40772(43)
    40773-40801 40854-42338(53) 42420-42449 42503 42504 42548 42551-42579 42650 42704
    42705 42749 42844 42846 42931 42933 43018 43020 43105 43107 43192 43194 43279 43281
    43366 43368 43453 43455 43540 43542 43627 43629 43714 43716 43801 43803 43888 43890
    43975 43977 44062 44064 44149 44151 44236 44238 44323 44325 44410 44412 44497 44499
    44584 44586 44671 44673 44758 44760 44845 44847 44932 44934 45019 45021 45106 45108
    45193 45195 45238-45323 45950 45951 46057 46110 46111 46217 46374 46376 46481 46483
    46588 46590 46695 46697 46802 46804 46909 46911 47016 47018 47123 47125 47230 47232
    47337 47339 47444 47446 47551 47553 47658 47660 47765 47767 47872 47874 47979 47981
    48086 48088 48193 48195 48300 48302 48407 48409 48514 48516 48621 48623 48728 48730
    48835 48837 48942 48944 49049 49051 49156 49158 49263 49265 49318-49321 49479
    49532-49534 49692 49745-49747 49905 49958-49960 50118 50171-50173 50331 50384-50386
    50544 50597-50599 50757 50810-50812 50970 51023-51025 51183 51236-51238 51396
    51449-51451 51609 51662-51664 51822 51875-51877 52035 52088-52090 52248 52301-52303
    52461 52514-52516 52674 52727-52729 52887 52940-52942 53100 53153-53155 53313
    53366-53368 53526 53579-53581 53739 53792-53794 53952 54005-54007 54165 54218-54220
    54378 54431-54433 54591 54644-54646 54804 54857-54859 55017 55070-55072 55230 55283
    57181 57228-57230 57269-57271 57310-57312 57351-57353 57392-57394 57433-57435
    57474-57476 57583-57585 57692-57694 57801-57803 57910-57912 58019-58021 58128-58130
    58237-58239 58346-58348 58455-58457 58564-58566 58673-58675 58782-58784 58891-58893
    59000-59002 59109-59111 59218-59220" )
422 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN/REGULA/SET(1)/DIRECT", 3 )
423 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/MAXITE", 30 )
424 #setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/LINESE", True )
425 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/ENERGY", True )
426 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/DISPLA", False )
427 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/FORCE", True )
428 #setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/ENERGY/TOLCON", 0.001 )
429 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/ENERGY/NOCONV", "CONTIN" )
430 #setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/FORCE/TOLCON", 0.01 )
431 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/FORCE/NOCONV", "CONTIN" )
432 #setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/METHOD/METNAM", "SECANT" )
433
434 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "OUTPUT(1)/SELTYP",
    "USER" )
435 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "OUTPUT(1)/USER" )
436
437 ## OUTPUT ##
438 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/DISPLA(1)/TOTAL" )
439
440 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/FORCE(1)/REACTI" )
441
442 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(1)/TOTAL/GREEN" )
443 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(2)/TOTAL/GREEN/PRINCI" )
444 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(3)/CRKWDT" )
445 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(4)/CRACK/GREEN" )
446 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
```

```
    "OUTPUT(1)/USER/STRAIN(5)/CRKWDT/GREEN/PRINCI" )
447 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(6)/TOTAL/TRACTI/LOCAL" )
448 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(7)/TOTAL/GREEN/LOCAL" )
449
450 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(1)/TOTAL/CAUCHY" )
451 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(2)/TOTAL/CAUCHY/PRINCI" )
452 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(3)/TOTAL/FORCE/LOCAL" )
453 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(4)/TOTAL/MOMENT/LOCAL" )
454 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(5)/TOTAL/TRACTI/LOCAL" )
455 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(6)/TOTAL/CAUCHY/LOCAL" )
456
457 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(1)/TOTAL/GREEN/GLOBAL/LOCATI", "INTPNT" )
458 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(1)/TOTAL/CAUCHY/GLOBAL/LOCATI", "INTPNT" )
459 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(6)/TOTAL/CAUCHY/LOCAL/LOCATI", "INTPNT" )
460 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(7)/TOTAL/GREEN/LOCAL/LOCATI", "INTPNT" )
461
462 #Run Analysis
463 # runSolver( [] )
464 # showView( "RESULT" )
465 fitAll( )
466
467 renameViewSettingsDirectory( "view setting", "RESULT/CONTOU/CLIPS/PLANES(1)",
    "X=260" )
468 setViewSettingValue( "view setting", "RESULT/CONTOU/CLIPS/PLANES(1)/LOCATI", "260
    0.000000 0.000000" )
469 renameViewSettingsDirectory( "view setting", "RESULT/CONTOU/CLIPS/PLANES(2)",
    "X=340" )
470 setViewSettingValue( "view setting", "RESULT/CONTOU/CLIPS/PLANES(2)/LOCATI", "340
    0.000000 0.000000" )
471 renameViewSettingsDirectory( "view setting", "RESULT/CONTOU/CLIPS/PLANES(4)", "Y=50" )
472 setViewSettingValue( "view setting", "RESULT/CONTOU/CLIPS/PLANES(4)/NORMAL",
    "0.00000 1.000000 0.000000" )
473 setViewSettingValue( "view setting", "RESULT/CONTOU/CLIPS/PLANES(4)/LOCATI", "0
    100.000000 0.000000" )
474
475 #setViewSettingValue( "view setting", "RESULT/CONTOU/AUTRNG", "LIMITS" )
476 #setViewSettingValue( "view setting", "RESULT/CONTOU/LIMITS/MINVAL", -600 )
477 #setViewSettingValue( "view setting", "RESULT/CONTOU/LIMITS/MAXVAL", 600 )
478 #setViewSettingValue( "view setting", "RESULT/CONTOU/EQUIDI/NUMBER", 11 )
479
```

F - Full 3D model script

```
newProject( "D:/Afstuderen/Python/3D/DIANA Output/T13 3D", 10 )
setModelAnalysisAspects( [ "STRUCT" ] )
setModelDimension( "3D" )
setDefaultMeshOrder( "QUADRATIC" )
setDefaultMesherType( "HEXQUAD" )
setDefaultMidSideNodeLocation( "ONSHAP" )
setUnit( "LENGTH", "MM" )
setUnit( "FORCE", "N" )
setUnit( "ANGLE", "DEGREE" )


#Create Shapes
# createSheet( "Pressure Layer", [[ 0, 0, 70 ],[ 1900, 0, 70 ],[ 1900, 0, 250 ],[ 1150, 0, 250 ],[1050, 0, 250 ],[ 0, 0, 250 ]] )
# createSheet( "Slab", [[ 0, 0, 0 ],[ 250, 0, 0 ],[ 350, 0, 0 ],[ 1900, 0, 0 ],[ 1900, 0, 70],[ 0, 0, 70 ]] )

# createSheet( "Support", [[ 250, 0, 0 ],[ 350, 0, 0 ],[ 350, 0, -30 ],[ 300, 0 , -30],[ 250, 0, -30 ]] )
# createSheet( "Loadplate", [[1050, 0, 250 ],[ 1150, 0, 250 ],[ 1150, 0, 280 ],[ 1100, 0, 280 ],[ 1050, 0, 280 ]] )

# extrudeProfile( [ "Pressure Layer" ], [ 0, 100, 0 ] )
# extrudeProfile( [ "Slab" ], [ 0, 100, 0 ] )
# extrudeProfile( [ "Support" ], [ 0, 100, 0 ] )
# extrudeProfile( [ "Loadplate" ], [ 0, 100, 0 ] )

# createBlock( "Slab", [1000, 0, 0], [900, 100, 70] )
# createBlock( "Slab_1", [0, 0, 0], [1000, 100, 70] )

# createBlock( "Pressure Layer", [1000, 0, 70], [900, 100, 180] )
# createBlock( "Pressure Layer_1", [0, 0, 70], [1000, 100, 180] )

createBlock( "Slab", [0, 0, 0], [1900, 100, 70] )
createBlock( "Pressure Layer", [0, 0, 70], [1900, 100, 180] )

createBlock( "Support", [250, 0, -30], [100, 100, 30] )
createBlock( "Loadplate", [1050, 0, 250], [100, 100, 30] )

createLine( "Line Support", [300, 0, -30], [300, 100, -30] )
createLine( "Line Loadplate", [1100, 0, 280], [1100, 100, 280] )

#Create Reinforcement
createLine( "Koppel_M12a", [ 1200, 50, 76 ], [ 1900, 50, 76 ] )

createLine( "Reinfo_M8a", [ 0, 12.5, 35 ], [ 1865, 12.5, 35 ] )
createLine( "Reinfo_M8b", [ 0, 77.5, 35 ], [ 1865, 77.5, 35 ] )

createLine( "LG1-lo", [ 135, 0, 45.5],[ 135, 100, 45.5] )
createLine( "LG1-mb", [ 175, 0, 165.5],[ 175, 100, 165.5] )
createLine( "LG1-ro", [ 215, 0, 45.5],[ 215, 100, 45.5] )
createPolyline( "LG1-l", [[ 135, 0, 45.5],[175, 100, 165.5]],False )
createPolyline( "LG1-r", [[ 215, 0, 45.5],[175, 100, 165.5]],False )

createLine( "LG2-lo", [ 735, 0, 45.5],[ 735, 100, 45.5] )
createLine( "LG2-mb", [ 775, 0, 165.5],[ 775, 100, 165.5] )
createLine( "LG2-ro", [ 815, 0, 45.5],[ 815, 100, 45.5] )
createPolyline( "LG2-l", [[ 735, 0, 45.5],[775, 100, 165.5]],False )
createPolyline( "LG2-r", [[ 815, 0, 45.5],[775, 100, 165.5]],False )

createLine( "LG3-lo", [ 1460, 0, 45.5],[ 1460, 100, 45.5] )
createLine( "LG3-mb", [ 1500, 0, 165.5],[ 1500, 100, 165.5] )
createLine( "LG3-ro", [ 1540, 0, 45.5],[ 1540, 100, 45.5] )
createPolyline( "LG3-l", [[ 1460, 0, 45.5],[1500, 100, 165.5]],False )
createPolyline( "LG3-r", [[ 1540, 0, 45.5],[1500, 100, 165.5]],False )

createLine( "Composed Line", [ 0, 50, 125 ], [ 1900, 50, 125 ] )

createBlock( "LG Mesh", [1440, 0, 20], [120, 100, 50] )
createBlock( "Course Slab", [0, 0, 0], [900, 100, 70] )
createBlock( "Course Pressure Layer", [0, 0, 70], [900, 100, 180] )
```

```
68
69  imprintIntersection( "Support", "Line Support", True )
70  imprintIntersection( "Loadplate", "Line Loadplate", True )
71  imprintIntersection( "Pressure Layer", "Loadplate", True )
72  imprintIntersection( "Pressure Layer", "LG Mesh", True )
73  imprintIntersection( "Slab", "Support", True )
74  imprintIntersection( "Slab", "LG Mesh", True )
75  imprintIntersection( "Slab", "Course Slab", True )
76  imprintIntersection( "Pressure Layer", "Course Pressure Layer", True )
77
78
79  removeShape( [ "Line Support" ] )
80  removeShape( [ "Line Loadplate" ] )
81  removeShape( [ "LG Mesh" ] )
82  removeShape( [ "Course Slab" ] )
83  removeShape( [ "Course Pressure Layer" ] )
84
85  # Splitting Mesh Parts
86
87  #Regroup Material Parts
88  addSet( "GEOMETRYREINFOSET", "Coupling Reinforcement" )
89  moveToShapeSet( [ "Koppel_M12a"], "Coupling Reinforcement" )
90  addSet( "GEOMETRYREINFOSET", "Longitudinal Reinforcement" )
91  moveToShapeSet( [ "Reinfo_M8a","Reinfo_M8b" ], "Longitudinal Reinforcement" )
92  addSet( "GEOMETRYREINFOSET", "Lattice girders" )
93  moveToShapeSet( [ "LG1-lo", "LG1-mb", "LG1-ro", "LG1-l", "LG1-r", "LG2-lo",
    "LG2-mb", "LG2-ro", "LG2-l", "LG2-r", "LG3-lo", "LG3-mb", "LG3-ro", "LG3-l",
    "LG3-r"], "Lattice girders" )
94  addSet( "SHAPESET", "Steel plates" )
95  moveToShapeSet( [ "Loadplate", "Support" ], "Steel plates" )
96  addSet( "SHAPESET", "Precast concrete" )
97  moveToShapeSet( [ "Slab" ], "Precast concrete" )
98  rename( "SHAPESET", "Shapes", "Pressure layer" )
99  addSet( "SHAPESET", "Composed Line" )
100 moveToShapeSet( [ "Composed Line" ], "Composed Line" )
101
102 #Material Precast concrete
103 addMaterial( "Precast concrete", "CONCR", "TSCR", [] )
104 setParameter( "MATERIAL", "Precast concrete", "LINEAR/ELASTI/YOUNG", 33372 )
105 setParameter( "MATERIAL", "Precast concrete", "LINEAR/ELASTI/POISON", 0.2 )
106 setParameter( "MATERIAL", "Precast concrete", "LINEAR/MASS/DENSIT", 2.4e-09 )
107 setParameter( "MATERIAL", "Precast concrete", "MODTYP/TOTCRK", "ROTATE" )
108 setParameter( "MATERIAL", "Precast concrete", "TENSIL/TENCRV", "HORDYK" )
109 setParameter( "MATERIAL", "Precast concrete", "TENSIL/TENSTR", 3.03 )
110 setParameter( "MATERIAL", "Precast concrete", "TENSIL/GF1", 0.1419 )
111 setParameter( "MATERIAL", "Precast concrete", "TENSIL/CBSPEC", "ROTS" )
112 setParameter( "MATERIAL", "Precast concrete", "TENSIL/POISRE/POIRED", "DAMAGE" )
113 setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMCRV", "MC2010" )
114 setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMSTR", 55.9 )
115 setParameter( "MATERIAL", "Precast concrete", "COMPRS/EPSC1", 0.0024 )
116 setParameter( "MATERIAL", "Precast concrete", "COMPRS/EPSCU", 0.0035 )
117 setParameter( "MATERIAL", "Precast concrete", "COMPRS/CONFIN/CNFCRV", "NONE" )
118 # setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMCRV", "CONSTA" )
119 # setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMSTR", 40.1 )
120
121 addGeometry( "Precast concrete", "SOLID", "STRSOL", [] )
122 setParameter( "GEOMET", "Precast concrete", "LOCAXS", True )
123 setParameter( "GEOMET", "Precast concrete", "LOCAXS/XAXIS", [ 1, 0, 0 ] )
124 setParameter( "GEOMET", "Precast concrete", "LOCAXS/YAXIS", [ 0, 1, 0 ] )
125
126 addElementData( "Precast concrete" )
127 setElementClassType( "SHAPE", [ "Slab"], "STRSOL" )
128 assignMaterial( "Precast concrete", "SHAPE", [ "Slab" ] )
129 assignGeometry( "Precast concrete", "SHAPE", [ "Slab" ] )
130 assignElementData( "Precast concrete", "SHAPE", [ "Slab" ] )
131
132 #Material Pressure layer
133 addMaterial( "Pressure layer", "CONCR", "TSCR", [] )
134 setParameter( "MATERIAL", "Pressure layer", "LINEAR/ELASTI/YOUNG", 26045 )
135 setParameter( "MATERIAL", "Pressure layer", "LINEAR/ELASTI/POISON", 0.2 )
136 setParameter( "MATERIAL", "Pressure layer", "LINEAR/MASS/DENSIT", 2.4e-09 )
```

```
137 setParameter( "MATERIAL", "Pressure layer", "MODTYP/TOTCRK", "ROTATE" )
138 setParameter( "MATERIAL", "Pressure layer", "TENSIL/TENCRV", "HORDYK" )
139 setParameter( "MATERIAL", "Pressure layer", "TENSIL/TENSTR", 1.35 )
140 setParameter( "MATERIAL", "Pressure layer", "TENSIL/GF1", 0.1223 )
141 setParameter( "MATERIAL", "Pressure layer", "TENSIL/CBSPEC", "ROTS" )
142 setParameter( "MATERIAL", "Pressure layer", "TENSIL/POISRE/POIRED", "DAMAGE" )
143 setParameter( "MATERIAL", "Pressure layer", "COMPRS/COMCRV", "MC2010" )
144 setParameter( "MATERIAL", "Pressure layer", "COMPRS/COMSTR", 35.6 )
145 setParameter( "MATERIAL", "Pressure layer", "COMPRS/EPSC1", 0.0021 )
146 setParameter( "MATERIAL", "Pressure layer", "COMPRS/EPSCU", 0.0035 )
147 setParameter( "MATERIAL", "Pressure layer", "COMPRS/CONFIN/CNFCRV", "NONE" )
148 # setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMCRV", "CONSTA" )
149 # setParameter( "MATERIAL", "Precast concrete", "COMPRS/COMSTR", 17.6 )
150
151 addGeometry( "Pressure layer", "SOLID", "STRSOL", [] )
152 setParameter( "GEOMET", "Pressure layer", "LOCAXS", True )
153 setParameter( "GEOMET", "Pressure layer", "LOCAXS/XAXIS", [ 1, 0, 0 ] )
154 setParameter( "GEOMET", "Pressure layer", "LOCAXS/YAXIS", [ 0, 1, 0 ] )
155
156 addElementData( "Pressure layer" )
157 setElementClassType( "SHAPE", [ "Pressure Layer" ], "STRSOL" )
158 assignMaterial( "Pressure layer", "SHAPE", [ "Pressure Layer" ] )
159 assignGeometry( "Pressure layer", "SHAPE", [ "Pressure Layer"] )
160 assignElementData( "Pressure layer", "SHAPE", [ "Pressure Layer" ] )
161
162 #Material Steel
163 addMaterial( "Steel", "MCSTEL", "ISOTRO", [] )
164 setParameter( "MATERIAL", "Steel", "LINEAR/ELASTI/YOUNG", 210000 )
165 setParameter( "MATERIAL", "Steel", "LINEAR/ELASTI/POISON", 0.3 )
166 setParameter( "MATERIAL", "Steel", "LINEAR/MASS/DENSIT", 0 )
167 addGeometry( "Plate", "SOLID", "STRSOL", [] )
168 addElementData( "Plate" )
169
170 setElementClassType( "SHAPE", [ "Support", "Loadplate" ], "STRSOL" )
171 assignMaterial( "Steel", "SHAPE", [ "Support", "Loadplate" ] )
172 assignGeometry( "Plate", "SHAPE", [ "Support", "Loadplate" ] )
173 assignElementData( "Plate", "SHAPE", [ "Support", "Loadplate" ] )
174 setParameter( "DATA", "Plate", "INTEGR", "HIGH" )
175
176 #Concrete Interface
177 addMaterial( "Interface Concrete", "INTERF", "FRICTI", [] )
178 setParameter( "MATERIAL", "Interface Concrete", "LINEAR/ELAS6/DSNZ", 60000 )
179 setParameter( "MATERIAL", "Interface Concrete", "LINEAR/ELAS6/DSSX", 6000 )
180 setParameter( "MATERIAL", "Interface Concrete", "LINEAR/ELAS6/DSSY", 6000 )
181 setParameter( "MATERIAL", "Interface Concrete", "COULOM/OPNTYP", "GAP" )
182 setParameter( "MATERIAL", "Interface Concrete", "COULOM/GAPVAL", 0.5 )
183 setParameter( "MATERIAL", "Interface Concrete", "COULOM/COHESI", 0.5 )
184 setParameter( "MATERIAL", "Interface Concrete", "COULOM/PHI", 30 )
185 setParameter( "MATERIAL", "Interface Concrete", "COULOM/PSI", 0 )
186 setParameter( "MATERIAL", "Interface Concrete", "COULOM/FRCHSR/MODE2", 0 )
187
188 addGeometry( "Interface Concrete", "SHEET", "STPLIF", [] )
189 setParameter( "GEOMET", "Interface Concrete", "LOCAXS", True )
190 setParameter( "GEOMET", "Interface Concrete", "LOCAXS/XAXIS", [ 1, 0, 0 ] )
191
192 addElementData( "Interface Concrete" )
193 createConnection( "Interface Concrete", "INTER", "SHAPEFACE", "SHAPEFACE" )
194 setParameter( "GEOMETRYCONNECTION", "Interface Concrete", "MODE", "CLOSED" )
195 setElementClassType( "GEOMETRYCONNECTION", "Interface Concrete", "STPLIF" )
196 assignMaterial( "Interface Concrete", "GEOMETRYCONNECTION", "Interface Concrete" )
197 assignGeometry( "Interface Concrete", "GEOMETRYCONNECTION", "Interface Concrete" )
198 assignElementData( "Interface Concrete", "GEOMETRYCONNECTION", "Interface Concrete" )
199 setParameter( "GEOMETRYCONNECTION", "Interface Concrete", "FLIP", False )
200 attachTo( "GEOMETRYCONNECTION", "Interface Concrete", "SOURCE", "Slab", [ [
    1209.7294, 57.3573, 70 ] ] )
201 attachTo( "GEOMETRYCONNECTION", "Interface Concrete", "SOURCE", "Slab", [ [
    1755.0148, 57.3573, 70 ] ] )
202 attachTo( "GEOMETRYCONNECTION", "Interface Concrete", "SOURCE", "Slab", [ [
    1508.8288, 57.3573, 70 ] ] )
203 attachTo( "GEOMETRYCONNECTION", "Interface Concrete", "SOURCE", "Slab", [ [
    516.2157, 57.3573, 70 ] ] )
```

```
attachTo( "GEOMETRYCONNECTION", "Interface Concrete", "TARGET", "Pressure Layer", [ [ 1704.9852, 57.3573, 70 ] ] )
attachTo( "GEOMETRYCONNECTION", "Interface Concrete", "TARGET", "Pressure Layer", [ [ 1491.1712, 57.3573, 70 ] ] )
attachTo( "GEOMETRYCONNECTION", "Interface Concrete", "TARGET", "Pressure Layer", [ [ 1130.2706, 57.3573, 70 ] ] )
attachTo( "GEOMETRYCONNECTION", "Interface Concrete", "TARGET", "Pressure Layer", [ [ 383.7843, 57.3573, 70 ] ] )

#Steel Interface
addMaterial( "Interface Steel", "INTERF", "NONLIF", [] )
setParameter( "MATERIAL", "Interface Steel", "LINEAR/IFTYP", "SUR3D" )
setParameter( "MATERIAL", "Interface Steel", "LINEAR/ELAS6/DSNZ", 100000 )
setParameter( "MATERIAL", "Interface Steel", "LINEAR/ELAS6/DSSX", 10000 )
setParameter( "MATERIAL", "Interface Steel", "LINEAR/ELAS6/DSSY", 10000 )
setParameter( "MATERIAL", "Interface Steel", "NONLIN/IFNOTE", "NOTSHR" )
setParameter( "MATERIAL", "Interface Steel", "NONLIN/NLEL9/NOTENS", [ 0.001, 0 ] )
setParameter( "MATERIAL", "Interface Steel", "NONLIN/NLEL9/NOSHTE", [ 0.001, 0 ] )
setParameter( "MATERIAL", "Interface Steel", "NONLIN/NLEL9/NOSHT2", [ 0.001, 0 ] )

addGeometry( "Interface Steel", "SHEET", "STPLIF", [] )
setParameter( "GEOMET", "Interface Steel", "LOCAXS", True )
setParameter( "GEOMET", "Interface Steel", "LOCAXS/XAXIS", [ 1, 0, 0 ] )

addElementData( "Interface Steel" )
createConnection( "Interface Steel", "INTER", "SHAPEFACE", "SHAPEFACE" )

setParameter( "GEOMETRYCONNECTION", "Interface Steel", "CONTYP", "INTER" )
setParameter( "GEOMETRYCONNECTION", "Interface Steel", "MODE", "AUTO" )
setElementClassType( "GEOMETRYCONNECTION", "Interface Steel", "STPLIF" )
assignMaterial( "Interface Steel", "GEOMETRYCONNECTION", "Interface Steel" )
assignGeometry( "Interface Steel", "GEOMETRYCONNECTION", "Interface Steel" )
assignElementData( "Interface Steel", "GEOMETRYCONNECTION", "Interface Steel" )
setParameter( "GEOMETRYCONNECTION", "Interface Steel", "FLIP", False )
attachTo( "GEOMETRYCONNECTION", "Interface Steel", "SOURCE", "Support", [[ 300, 50, 0 ]] )
attachTo( "GEOMETRYCONNECTION", "Interface Steel", "SOURCE", "Loadplate", [[ 1100, 50, 250 ]] )
attachTo( "GEOMETRYCONNECTION", "Interface Steel", "TARGET", "Slab", [[ 300, 50, 0 ]] )
attachTo( "GEOMETRYCONNECTION", "Interface Steel", "TARGET", "Pressure Layer", [[ 1100, 50, 250 ]] )

#Material Reinforcement Koppel_M12
# addMaterial( "Koppel_M12", "REINFO", "VMISES", [] )
# setParameter( "MATERIAL", "Koppel_M12", "LINEAR/ELASTI/YOUNG", 200000 )
# setParameter( "MATERIAL", "Koppel_M12", "PLASTI/YLDTYP", "KAPSIG" )
# setParameter( "MATERIAL", "Koppel_M12", "PLASTI/HARDI2/KAPSIG", [] )
# setParameter( "MATERIAL", "Koppel_M12", "PLASTI/HARDI2/KAPSIG", [ 0, 550, 0.04725, 594 ] )
addMaterial( "Koppel_M12", "REINFO", "REBOND", [ ] )
setParameter( "MATERIAL", "Koppel_M12", "REBARS/ELASTI/YOUNG", 200000 )
setParameter( "MATERIAL", "Koppel_M12", "REBARS/POISON/POISON", 0.3 )
setParameter( "MATERIAL", "Koppel_M12", "REBARS/PLATYP", "VMISES" )
setParameter( "MATERIAL", "Koppel_M12", "REBARS/PLASTI/TRESSH", "KAPSIG" )
setParameter( "MATERIAL", "Koppel_M12", "REBARS/PLASTI/KAPSIG", [ 0, 550, 0.04725, 594 ] )
setParameter( "MATERIAL", "Koppel_M12", "RESLIP/SHFTYP", "BONDS6" )
setParameter( "MATERIAL", "Koppel_M12", "RESLIP/DSNY", 100000 )
setParameter( "MATERIAL", "Koppel_M12", "RESLIP/DSSX", 10000 )
setParameter( "MATERIAL", "Koppel_M12", "RESLIP/BONDS6/SLPVAL", [ 15.41, 6.16, 0.1, 1, 2, 10, 0.4 ] )
addGeometry( "Koppel_M12", "RELINE", "REBAR", [] )
# setParameter( "GEOMET", "Koppel_M12", "REITYP", "REIEMB" )
# setParameter( "GEOMET", "Koppel_M12", "REIEMB/CROSSE", 113.1 )
setParameter( "GEOMET", "Koppel_M12", "REITYP", "REITRU" )
setParameter( "GEOMET", "Koppel_M12", "REITYP", "CIRBEA" )
setParameter( "GEOMET", "Koppel_M12", "CIRBEA/CIRCLE", 12)
setParameter( "GEOMET", "Koppel_M12", "ORIENT/ZAXIS", [ 0, 0, 1 ] )
setParameter( "GEOMET", "Koppel_M12", "TIPELM/SURFAC", 113.1 )
#assignMaterial( "Koppel_M12", "SHAPE", [ "Koppel_M12a","Koppel_M12b" ] )
```

```
#assignGeometry( "Koppel_M12", "SHAPE", [ "Koppel_M12a","Koppel_M12b" ] )
#convertToReinforcement( [ "Koppel_M12a","Koppel_M12b" ] )
#addElementData( "Koppel_M12" )
#assignElementData( "Koppel_M12", "SHAPE", [ "Koppel_M12a","Koppel_M12b" ] )
#setParameter( "DATA", "Koppel_M12", "INTERF", "BEAM" )
#setReinforcementDiscretization( [ "Koppel_M12a","Koppel_M12b" ], "ELEMENT" )
#setContinuousInInterfaces( [ "Koppel_M12a","Koppel_M12b" ], False )
setReinforcementType( "GEOMETRYREINFOSET", "Coupling Reinforcement",
"BEAM_NUMERICAL_BOND_SLIP" )
setContinuousInInterfaces( "GEOMETRYREINFOSET", "Coupling Reinforcement", False )
assignMaterial( "Koppel_M12", "GEOMETRYREINFOSET", [ "Coupling Reinforcement"])
assignGeometry( "Koppel_M12", "GEOMETRYREINFOSET", [ "Coupling Reinforcement"])

#Material Reinforcement M8
# addMaterial( "Reinfo_M8", "REINFO", "VMISES", [] )
# setParameter( "MATERIAL", "Reinfo_M8", "LINEAR/ELASTI/YOUNG", 200000 )
# setParameter( "MATERIAL", "Reinfo_M8", "PLASTI/YLDTYP", "KAPSIG" )
# setParameter( "MATERIAL", "Reinfo_M8", "PLASTI/HARDI2/KAPSIG", [] )
# setParameter( "MATERIAL", "Reinfo_M8", "PLASTI/HARDI2/KAPSIG", [ 0, 550, 0.04725,
594 ] )
addMaterial( "Reinfo_M8", "REINFO", "REBOND", [] )
setParameter( "MATERIAL", "Reinfo_M8", "REBARS/ELASTI/YOUNG", 200000 )
setParameter( "MATERIAL", "Reinfo_M8", "REBARS/POISON/POISON", 0.3 )
setParameter( "MATERIAL", "Reinfo_M8", "REBARS/PLATYP", "VMISES" )
setParameter( "MATERIAL", "Reinfo_M8", "REBARS/PLASTI/TRESSH", "KAPSIG" )
setParameter( "MATERIAL", "Reinfo_M8", "REBARS/PLASTI/KAPSIG", [ 0, 550, 0.04725,
594 ] )
setParameter( "MATERIAL", "Reinfo_M8", "RESLIP/SHFTYP", "BONDS6" )
setParameter( "MATERIAL", "Reinfo_M8", "RESLIP/DSNY", 100000 )
setParameter( "MATERIAL", "Reinfo_M8", "RESLIP/DSSX", 10000 )
setParameter( "MATERIAL", "Reinfo_M8", "RESLIP/BONDS6/SLPVAL", [ 15.41, 6.16, 0.1,
1, 2, 10, 0.4 ] )
addGeometry( "Reinfo_M8", "RELINE", "REBAR", [] )
# setParameter( "GEOMET", "Reinfo_M8", "REITYP", "REIEMB" )
# setParameter( "GEOMET", "Reinfo_M8", "REIEMB/CROSSE", 50.3 )
setParameter( "GEOMET", "Reinfo_M8", "REITYP", "REITRU" )
setParameter( "GEOMET", "Reinfo_M8", "REITYP", "CIRBEA" )
setParameter( "GEOMET", "Reinfo_M8", "CIRBEA/CIRCLE", 8)
setParameter( "GEOMET", "Reinfo_M8", "ORIENT/ZAXIS", [ 0, 0, 1 ] )
setParameter( "GEOMET", "Reinfo_M8", "TIPELM/SURFAC", 50.3 )
#assignMaterial( "Reinfo_M8", "SHAPE", [
"Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d" ] )
#assignGeometry( "Reinfo_M8", "SHAPE", [
"Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d" ] )
#convertToReinforcement( [ "Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d"] )
#addElementData( "Reinfo_M8" )
#assignElementData( "Reinfo_M8", "SHAPE", [
"Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d"] )
#setParameter( "DATA", "Reinfo_M8", "INTERF", "BEAM" )
#setReinforcementDiscretization( [
"Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d" ], "ELEMENT" )
#setContinuousInInterfaces( [ "Reinfo_M8a","Reinfo_M8b","Reinfo_M8c","Reinfo_M8d" ],
False )
setReinforcementType( "GEOMETRYREINFOSET", "Longitudinal Reinforcement",
"BEAM_NUMERICAL_BOND_SLIP" )
setContinuousInInterfaces( "GEOMETRYREINFOSET", "Longitudinal Reinforcement", False )
assignMaterial( "Reinfo_M8", "GEOMETRYREINFOSET", [ "Longitudinal Reinforcement"])
assignGeometry( "Reinfo_M8", "GEOMETRYREINFOSET", [ "Longitudinal Reinforcement"])

#Material Reinforcement M5
addMaterial( "Reinfo_M5", "REINFO", "REBOND", [] )
setParameter( "MATERIAL", "Reinfo_M5", "REBARS/ELASTI/YOUNG", 200000 )
setParameter( "MATERIAL", "Reinfo_M5", "REBARS/POISON/POISON", 0.3 )
setParameter( "MATERIAL", "Reinfo_M5", "REBARS/PLATYP", "VMISES" )
setParameter( "MATERIAL", "Reinfo_M5", "REBARS/PLASTI/TRESSH", "KAPSIG" )
setParameter( "MATERIAL", "Reinfo_M5", "REBARS/PLASTI/KAPSIG", [ 0, 550, 0.04725,
594 ] )
setParameter( "MATERIAL", "Reinfo_M5", "RESLIP/SHFTYP", "BONDS6" )
setParameter( "MATERIAL", "Reinfo_M5", "RESLIP/DSNY", 100000 )
setParameter( "MATERIAL", "Reinfo_M5", "RESLIP/DSSX", 10000 )
setParameter( "MATERIAL", "Reinfo_M5", "RESLIP/BONDS6/SLPVAL", [ 0.63, 0.63, 0.01,
```

```
    0.02, 0.03, 0.04, 0.5 ] )
addGeometry( "Reinfo_M5", "RELINE", "REBAR", [] )
setParameter( "GEOMET", "Reinfo_M5", "REITYP", "REITRU" )
setParameter( "GEOMET", "Reinfo_M5", "REITYP", "CIRBEA" )
setParameter( "GEOMET", "Reinfo_M5", "CIRBEA/CIRCLE", 5 )
setParameter( "GEOMET", "Reinfo_M5", "ORIENT/ZAXIS", [ 0, 0, 1 ] )
setParameter( "GEOMET", "Reinfo_M5", "TIPELM/SURFAC", 19.6 )

setReinforcementType( "GEOMETRYREINFOSET", "Lattice girders",
"BEAM_NUMERICAL_BOND_SLIP" )
setContinuousInInterfaces( "GEOMETRYREINFOSET", "Lattice girders", True )
assignMaterial( "Reinfo_M5", "GEOMETRYREINFOSET", [ "Lattice girders" ] )
assignGeometry( "Reinfo_M5", "GEOMETRYREINFOSET", [ "Lattice girders" ] )

#Composed Line
addGeometry( "Composed Line", "LINE", "COMLIN", [] )
setParameter( "GEOMET", "Composed Line", "DISTAN/THICK", 250 )
setElementClassType( "SHAPE", [ "Composed Line" ], "COMLIN" )
assignGeometry( "Composed Line", "SHAPE", [ "Composed Line" ] )
resetElementData( "SHAPE", [ "Composed Line" ] )

##Tyings
#createPointTying( "Tying LG1-l", "Tying LG" )
#setParameter( "GEOMETRYTYING", "Tying LG1-l", "AXES", [ 1, 2 ] )
#setParameter( "GEOMETRYTYING", "Tying LG1-l", "TRANSL", [ 1, 1, 1 ] )
#setParameter( "GEOMETRYTYING", "Tying LG1-l", "ROTATI", [ 1, 1, 1 ] )
#attachTo( "GEOMETRYTYING", "Tying LG1-l", "SLAVE", "LG1-l", [[ 135, 0, 45.5 ]] )
#attachTo( "GEOMETRYTYING", "Tying LG1-l", "MASTER", "LG1-lo", [[ 135, 0, 45.5 ]] )
#
#createPointTying( "Tying LG1-r", "Tying LG" )
#setParameter( "GEOMETRYTYING", "Tying LG1-r", "AXES", [ 1, 2 ] )
#setParameter( "GEOMETRYTYING", "Tying LG1-r", "TRANSL", [ 1, 1, 1 ] )
#setParameter( "GEOMETRYTYING", "Tying LG1-r", "ROTATI", [ 1, 1, 1 ] )
#attachTo( "GEOMETRYTYING", "Tying LG1-r", "SLAVE", "LG1-r", [[ 215, 0, 45.5 ]] )
#attachTo( "GEOMETRYTYING", "Tying LG1-r", "MASTER", "LG1-ro", [[ 215, 0, 45.5 ]] )
#
#createPointTying( "Tying LG1-m", "Tying LG" )
#setParameter( "GEOMETRYTYING", "Tying LG1-m", "AXES", [ 1, 2 ] )
#setParameter( "GEOMETRYTYING", "Tying LG1-m", "TRANSL", [ 1, 1, 1 ] )
#setParameter( "GEOMETRYTYING", "Tying LG1-m", "ROTATI", [ 1, 1, 1 ] )
#attachTo( "GEOMETRYTYING", "Tying LG1-m", "SLAVE", "LG1-r", [[ 175, 100, 165.5]] )
#attachTo( "GEOMETRYTYING", "Tying LG1-m", "SLAVE", "LG1-l", [[ 175, 100, 165.5]] )
#attachTo( "GEOMETRYTYING", "Tying LG1-m", "MASTER", "LG1-mb", [[ 175, 100, 165.5]] )
#
#createPointTying( "Tying LG2-l", "Tying LG" )
#setParameter( "GEOMETRYTYING", "Tying LG2-l", "AXES", [ 1, 2 ] )
#setParameter( "GEOMETRYTYING", "Tying LG2-l", "TRANSL", [ 1, 1, 1 ] )
#setParameter( "GEOMETRYTYING", "Tying LG2-l", "ROTATI", [ 1, 1, 1 ] )
#attachTo( "GEOMETRYTYING", "Tying LG2-l", "SLAVE", "LG2-l", [[ 735, 0, 45.5 ]] )
#attachTo( "GEOMETRYTYING", "Tying LG2-l", "MASTER", "LG2-lo", [[ 735, 0, 45.5 ]] )
#
#createPointTying( "Tying LG2-r", "Tying LG" )
#setParameter( "GEOMETRYTYING", "Tying LG2-r", "AXES", [ 1, 2 ] )
#setParameter( "GEOMETRYTYING", "Tying LG2-r", "TRANSL", [ 1, 1, 1 ] )
#setParameter( "GEOMETRYTYING", "Tying LG2-r", "ROTATI", [ 1, 1, 1 ] )
#attachTo( "GEOMETRYTYING", "Tying LG2-r", "SLAVE", "LG2-r", [[ 815, 0, 45.5 ]] )
#attachTo( "GEOMETRYTYING", "Tying LG2-r", "MASTER", "LG2-ro", [[ 815, 0, 45.5 ]] )
#
#createPointTying( "Tying LG2-m", "Tying LG" )
#setParameter( "GEOMETRYTYING", "Tying LG2-m", "AXES", [ 1, 2 ] )
#setParameter( "GEOMETRYTYING", "Tying LG2-m", "TRANSL", [ 1, 1, 1 ] )
#setParameter( "GEOMETRYTYING", "Tying LG2-m", "ROTATI", [ 1, 1, 1 ] )
#attachTo( "GEOMETRYTYING", "Tying LG2-m", "SLAVE", "LG2-r", [[ 775, 100, 165.5]] )
#attachTo( "GEOMETRYTYING", "Tying LG2-m", "SLAVE", "LG2-l", [[ 775, 100, 165.5]] )
#attachTo( "GEOMETRYTYING", "Tying LG2-m", "MASTER", "LG2-mb", [[ 775, 100, 165.5]] )
#
#createPointTying( "Tying LG3-l", "Tying LG" )
#setParameter( "GEOMETRYTYING", "Tying LG3-l", "AXES", [ 1, 2 ] )
#setParameter( "GEOMETRYTYING", "Tying LG3-l", "TRANSL", [ 1, 1, 1 ] )
#setParameter( "GEOMETRYTYING", "Tying LG3-l", "ROTATI", [ 1, 1, 1 ] )
#attachTo( "GEOMETRYTYING", "Tying LG3-l", "SLAVE", "LG3-l", [[ 1460, 0, 45.5 ]] )
```

```
#attachTo( "GEOMETRYTYING", "Tying LG3-l", "MASTER", "LG3-lo", [[ 1460, 0, 45.5 ]] )
#
#createPointTying( "Tying LG3-r", "Tying LG" )
#setParameter( "GEOMETRYTYING", "Tying LG3-r", "AXES", [ 1, 2 ] )
#setParameter( "GEOMETRYTYING", "Tying LG3-r", "TRANSL", [ 1, 1, 1 ] )
#setParameter( "GEOMETRYTYING", "Tying LG3-r", "ROTATI", [ 1, 1, 1 ] )
#attachTo( "GEOMETRYTYING", "Tying LG3-r", "SLAVE", "LG3-r", [[ 1540, 0, 45.5 ]] )
#attachTo( "GEOMETRYTYING", "Tying LG3-r", "MASTER", "LG3-ro", [[ 1540, 0, 45.5 ]] )
#
#createPointTying( "Tying LG3-m", "Tying LG" )
#setParameter( "GEOMETRYTYING", "Tying LG3-m", "AXES", [ 1, 2 ] )
#setParameter( "GEOMETRYTYING", "Tying LG3-m", "TRANSL", [ 1, 1, 1 ] )
#setParameter( "GEOMETRYTYING", "Tying LG3-m", "ROTATI", [ 1, 1, 1 ] )
#attachTo( "GEOMETRYTYING", "Tying LG3-m", "SLAVE", "LG3-r", [[ 1500, 100, 165.5]] )
#attachTo( "GEOMETRYTYING", "Tying LG3-m", "SLAVE", "LG3-l", [[ 1500, 100, 165.5]] )
#attachTo( "GEOMETRYTYING", "Tying LG3-m", "MASTER", "LG3-mb", [[ 1500, 100, 165.5]] )

#Assign Supports
addSet( "GEOMETRYSUPPORTSET", "Support" )

createLineSupport( "Support", "Support" )
setParameter( "GEOMETRYSUPPORT", "Support", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Support", "TRANSL", [ 0, 0, 1 ] )
setParameter( "GEOMETRYSUPPORT", "Support", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Support", "Support", [[ 300, 50, -30 ]] )

addSet( "GEOMETRYSUPPORTSET", "Sym_X" )

createSurfaceSupport( "Sym_X", "Sym_X" )
setParameter( "GEOMETRYSUPPORT", "Sym_X", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Sym_X", "TRANSL", [ 1, 0, 0 ] )
setParameter( "GEOMETRYSUPPORT", "Sym_X", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Sym_X", "Pressure Layer", [[ 1900, 50, 160 ]] )

createPointSupport( "Symp_X", "Symp_X" )
setParameter( "GEOMETRYSUPPORT", "Symp_X", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Symp_X", "TRANSL", [ 1, 0, 0 ] )
setParameter( "GEOMETRYSUPPORT", "Symp_X", "ROTATI", [ 0, 1, 0 ] )
attach( "GEOMETRYSUPPORT", "Symp_X", "Koppel_M12a", [[ 1900, 50, 76 ]] )

createPointSupport( "Symp_Y", "Symp_Y" )
setParameter( "GEOMETRYSUPPORT", "Symp_Y", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Symp_Y", "TRANSL", [ 0, 1, 0 ] )
setParameter( "GEOMETRYSUPPORT", "Symp_Y", "ROTATI", [ 1, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Symp_Y", "LG1-lo", [[ 135, 0, 45.5 ]] )
attach( "GEOMETRYSUPPORT", "Symp_Y", "LG1-ro", [[ 215, 0, 45.5 ]] )
attach( "GEOMETRYSUPPORT", "Symp_Y", "LG1-mb", [[ 175, 0, 165.5 ]] )
attach( "GEOMETRYSUPPORT", "Symp_Y", "LG1-lo", [[ 135, 100, 45.5 ]] )
attach( "GEOMETRYSUPPORT", "Symp_Y", "LG1-ro", [[ 215, 100, 45.5 ]] )
attach( "GEOMETRYSUPPORT", "Symp_Y", "LG1-mb", [[ 175, 100, 165.5 ]] )
attach( "GEOMETRYSUPPORT", "Symp_Y", "LG2-lo", [[ 735, 0, 45.5 ]] )
attach( "GEOMETRYSUPPORT", "Symp_Y", "LG2-ro", [[ 815, 0, 45.5 ]] )
attach( "GEOMETRYSUPPORT", "Symp_Y", "LG2-mb", [[ 775, 0, 165.5 ]] )
attach( "GEOMETRYSUPPORT", "Symp_Y", "LG2-lo", [[ 735, 100, 45.5 ]] )
attach( "GEOMETRYSUPPORT", "Symp_Y", "LG2-ro", [[ 815, 100, 45.5 ]] )
attach( "GEOMETRYSUPPORT", "Symp_Y", "LG2-mb", [[ 775, 100, 165.5 ]] )
attach( "GEOMETRYSUPPORT", "Symp_Y", "LG3-lo", [[ 1460, 0, 45.5 ]] )
attach( "GEOMETRYSUPPORT", "Symp_Y", "LG3-ro", [[ 1540, 0, 45.5 ]] )
attach( "GEOMETRYSUPPORT", "Symp_Y", "LG3-mb", [[ 1500, 0, 165.5 ]] )
attach( "GEOMETRYSUPPORT", "Symp_Y", "LG3-lo", [[ 1460, 100, 45.5 ]] )
attach( "GEOMETRYSUPPORT", "Symp_Y", "LG3-ro", [[ 1540, 100, 45.5 ]] )
attach( "GEOMETRYSUPPORT", "Symp_Y", "LG3-mb", [[ 1500, 100, 165.5 ]] )

createSurfaceSupport( "Sym_Y", "Sym_Y" )
setParameter( "GEOMETRYSUPPORT", "Sym_Y", "AXES", [ 1, 2 ] )
setParameter( "GEOMETRYSUPPORT", "Sym_Y", "TRANSL", [ 0, 1, 0 ] )
setParameter( "GEOMETRYSUPPORT", "Sym_Y", "ROTATI", [ 0, 0, 0 ] )
attach( "GEOMETRYSUPPORT", "Sym_Y", "Slab", [[ 500, 0, 35 ],[ 1200, 0, 50 ],[ 1500,
0, 50 ]] )
attach( "GEOMETRYSUPPORT", "Sym_Y", "Slab", [[ 500, 100, 35 ],[ 1200, 100, 50 ],[
1500, 100, 50 ]] )
```

```
462 attach( "GEOMETRYSUPPORT", "Sym_Y", "Pressure Layer", [[ 500, 0, 160 ],[ 1200, 0,
    160 ]] )
463 attach( "GEOMETRYSUPPORT", "Sym_Y", "Pressure Layer", [[ 500, 100, 160],[ 1200, 100,
    160 ]] )
464 attach( "GEOMETRYSUPPORT", "Sym_Y", "Support", [[ 300, 0, 15 ]] )
465 attach( "GEOMETRYSUPPORT", "Sym_Y", "Support", [[ 300, 100, 15 ]] )
466 attach( "GEOMETRYSUPPORT", "Sym_Y", "Loadplate", [[ 1100, 0, 265 ]] )
467 attach( "GEOMETRYSUPPORT", "Sym_Y", "Loadplate", [[ 1100, 100, 265 ]] )
468
469 addSet( "GEOMETRYSUPPORTSET", "Load point" )
470
471 createLineSupport( "Load point", "Load point" )
472 setParameter( "GEOMETRYSUPPORT", "Load point", "AXES", [ 1, 2 ] )
473 setParameter( "GEOMETRYSUPPORT", "Load point", "TRANSL", [ 0, 0, 1 ] )
474 setParameter( "GEOMETRYSUPPORT", "Load point", "ROTATI", [ 0, 0, 0 ] )
475 attach( "GEOMETRYSUPPORT", "Load point", "Loadplate", [[ 1100, 50, 280 ]] )
476
477 #Assign Loads
478 addSet( "GEOMETRYLOADSET", "Load" )
479
480 createLineLoad( "Load", "Load" )
481 setParameter( "GEOMETRYLOAD", "Load", "LODTYP", "DEFORM" )
482 setParameter( "GEOMETRYLOAD", "Load", "DEFORM/TR/VALUE", -1 )
483 setParameter( "GEOMETRYLOAD", "Load", "DEFORM/TR/DIRECT", 3 )
484 attach( "GEOMETRYLOAD", "Load", "Loadplate", [[ 1100, 50, 280 ]] )
485
486 #Set mesh properties
487 setElementSize( [ "Loadplate", "Support" ], 20, -1, True )
488 clearMesherType( [ "Loadplate", "Support" ] )
489 clearMidSideNodeLocation( [ "Loadplate", "Support" ] )
490 setElementSize( "Pressure Layer", 2, [ [ 1900, 57.3573, 146.75686 ], [ 1473.573, 0,
    146.75686 ], [ 1580.1798, 57.3573, 250 ], [ 1107.3573, 57.3573, 250 ], [ 1704.9852,
    57.3573, 70 ], [ 1491.1712, 57.3573, 70 ], [ 1130.2706, 57.3573, 70 ], [ 1473.573,
    100, 173.24314 ], [ 986.03595, 57.3573, 250 ] ], 20, 0.5, True )
491 setElementSize( "Pressure Layer", 2, [ [ 383.7843, 57.3573, 70 ], [ 516.2157, 100,
    173.24314 ], [ 0, 57.3573, 173.24314 ], [ 516.2157, 57.3573, 250 ], [ 516.2157, 0,
    146.75686 ] ], 50, 0.5, True )
492 setCurrentShapeSet( "Precast concrete" )
493 show( "SHAPESET", [ "Precast concrete" ] )
494 hide( "SHAPESET", [ "Pressure layer" ] )
495 setCurrentShapeSet( "Pressure layer" )
496 setElementSize( "Slab", 2, [ [ 1508.8288, 57.3573, 70 ], [ 1508.8288, 100, 52.94292
    ], [ 1508.8288, 0, 47.05708 ] ], 8, 0.5, True )
497 setElementSize( "Slab", 2, [ [ 1209.7294, 0, 40.15011 ], [ 1209.7294, 57.3573, 70 ],
    [ 1755.0148, 57.3573, 70 ], [ 1326.427, 100, 29.84989 ], [ 1900, 57.3573, 29.84989
    ], [ 1326.427, 57.3573, 0 ] ], 20, 0.5, True )
498 setViewPoint( "FRONT" )
499 setElementSize( "Slab", 2, [ [ 106.60675, 57.3573, 0 ], [ 516.2157, 100, 40.15011 ],
    [ 0, 57.3573, 40.15011 ], [ 584.53485, 57.3573, 0 ], [ 292.6427, 57.3573, 0 ], [
    516.2157, 57.3573, 70 ], [ 516.2157, 0, 29.84989 ] ], 35, 0.5, True )
500
501 generateMesh( [] )
502 hideView( "GEOM" )
503 showView( "MESH" )
504
505 #Analysis
506 addAnalysis( "Analysis1" )
507 addAnalysisCommand( "Analysis1", "NONLIN", "Structural nonlinear" )
508 renameAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "EXECUT(1)",
    "Load" )
509 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/LOADNR" )
510 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/LOADNR", 1 )
511 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/STEPTY", "AUTOMA" )
512 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/SIZES", 250 )
513 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/MAXSIZ", 0.01 )
514 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
```

```
    "EXECUT(1)/LOAD/STEPS/AUTOMA/MINSIZ", 0.00001 )
515 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/CUTBCK", 0.25 )
516 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/MAXSTP", 100 )
517 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/ARCLEN", True )
518 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/ARCLEN/REGULA/SET" )
519 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/ARCLEN/REGULA/SET(1)/NODES(1)/RNGNRS", "1-9361" )
520 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/AUTOMA/ARCLEN/REGULA/SET(1)/DIRECT", 2 )
521 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/SIZES", "0.05(100)" )
522 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN", True )
523 # addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN/REGULA/SET" )
524 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN/REGULA/SET(1)/NODES(1)/RNGNRS", "628-631 684-701
    790-808 974-992 1106-3420(89) 3421-3581 5735-5751 5907 5909 5910 5912 5920-5926(2)
    5927 5931 5933 5935 5951-5953 5959 5962 5964 5967 5969 5971 5975 5980 5981 6005 6007
    6008 6010 6017 6022-6024 6029 6034 6035 6037 6044 6049 6050 6052 6084-6086 6088 6089
    6103 6105 6106 6108 6113 6119 6121 6123 6124 6132 6139 6141 6142 6177 6181 6183-6185
    6201 6204 6206 6207 6218 6221 6222 6224 6225 6234 6242 6244 6245 6283 6287 6289-6291
    6309 6312 6314 6315 6327 6330-6333 6352 6353 6355 6356 6397 6401 6403-6405 6424 6428
    6430 6431 6446 6447 6449-6451 6470 6472 6474 6475 6519 6523 6525-6527 6547 6551 6553
    6554 6571-6574 6593 6597 6598 6643 6647 6649 6650 6671 6674 6676 6694-6696 6714 6718
    6761 6765 6767 6788 6790 6809 6810 6827 6871 6875 6897 6916 6977 19524 19525
    19535-19544 19633-20434(89) 20525 20526 20548-20556 20645-20651 20740-20800
    21329-21334 21863-21868 22661-22669 22727 22728 22730 22731 22740 22742 22743 22745
    22751-22754 22770 22775 22777 22780 22781 22783 22793 22796-22800 22825 22829 22831
    22835 22838 22839 22849 22857 22859-22862 22893 22895 22897 22901 22904 22905 22925
    22927 22928 22930 22961 22962 22967 22971 22993 22994 23018 23031 27554" )
525 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/LOAD/STEPS/EXPLIC/ARCLEN/REGULA/SET(1)/DIRECT", 3 )
526 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/MAXITE", 30 )
527 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/LINESE", False )
528 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/ENERGY", True )
529 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/DISPLA", False )
530 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/FORCE", True )
531 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/ENERGY/TOLCON", 0.001 )
532 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/ENERGY/NOCONV", "CONTIN" )
533 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/FORCE/TOLCON", 0.01 )
534 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/CONVER/FORCE/NOCONV", "CONTIN" )
535 # setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/METHOD/METNAM", "SECANT" )
536 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "EXECUT(1)/ITERAT/METHOD/METNAM", "NEWTON" )
537
538 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "OUTPUT(1)/SELTYP",
    "USER" )
539 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear", "OUTPUT(1)/USER" )
540
541 ## OUTPUT ##
542 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/DISPLA(1)/TOTAL" )
543
544 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/FORCE(1)/REACTI" )
545
```

```
546 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(1)/TOTAL/GREEN" )
547 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(2)/TOTAL/GREEN/PRINCI" )
548 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(3)/CRKWDT" )
549 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(4)/CRACK/GREEN" )
550 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(5)/CRKWDT/GREEN/PRINCI" )
551 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(6)/TOTAL/TRACTI/LOCAL" )
552 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(7)/TOTAL/GREEN/LOCAL" )
553
554 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(1)/TOTAL/CAUCHY" )
555 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(2)/TOTAL/CAUCHY/PRINCI" )
556 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(3)/TOTAL/FORCE/LOCAL" )
557 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(4)/TOTAL/MOMENT/LOCAL" )
558 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(5)/TOTAL/TRACTI/LOCAL" )
559 addAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(6)/TOTAL/CAUCHY/LOCAL" )
560
561 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(1)/TOTAL/GREEN/GLOBAL/LOCATI", "INTPNT" )
562 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(1)/TOTAL/CAUCHY/GLOBAL/LOCATI", "INTPNT" )
563 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRESS(6)/TOTAL/CAUCHY/LOCAL/LOCATI", "INTPNT" )
564 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(7)/TOTAL/GREEN/LOCAL/LOCATI", "INTPNT" )
565 setAnalysisCommandDetail( "Analysis1", "Structural nonlinear",
    "OUTPUT(1)/USER/STRAIN(2)/TOTAL/GREEN/PRINCI/LOCATI", "INTPNT" )
566
567 # #Run Analysis
568 # # runSolver( [] )
569 # # showView( "RESULT" )
570 fitAll( )
571
572 # hide( "SHAPE", [ "Pressure Layer" ] )
573 # hide( "SHAPE", [ "Loadplate" ] )
574 # hide( "SHAPE", [ "Slab" ] )
575 # hide( "SHAPE", [ "Support" ] )
576
```