Behaviour and resistance of concrete encased embossments in composite slabs

PhD Dissertation

Budapest University of Technology and Economics
Department of Structural Engineering

Author:
Seres, Noémi

Supervisor:
Dr. Dunai, László

Budapest, 2012
# TABLE OF CONTENTS

Abstract.............................................................................................................................................5

Acknowledgement .........................................................................................................................7

1 Introduction .................................................................................................................................8
  1.1 Background of the research ......................................................................................................8
    1.1.1 Composite slabs.................................................................................................................8
    1.1.2 Structural behaviour ..........................................................................................................9
    1.1.3 Testing and design ..............................................................................................................9
    1.1.4 The m-k method ................................................................................................................10
    1.1.5 The partial shear connection method .................................................................................11
  1.2 Research program ..................................................................................................................14
    1.2.1 Aims ..................................................................................................................................14
    1.2.2 Problems to be solved ........................................................................................................14
    1.2.3 Research strategy ..............................................................................................................15

I. Experimental investigations .........................................................................................................17
  2 Pull-out test of an individual enlarged embossment ....................................................................17
    2.1 Test program .......................................................................................................................17
      2.1.1 General ..........................................................................................................................17
      2.1.2 Materials and measurement technology ...........................................................................18
      2.1.3 Preparation of the specimens, test execution .................................................................21
    2.2 Evaluation of test results ......................................................................................................21
      2.2.1 Test results ......................................................................................................................21
      2.2.2 Observed behaviour ........................................................................................................22
    2.3 Summary ................................................................................................................................25
  3 Effect of cold forming of the individual enlarged embossment ..................................................26
    3.1 Extruding .............................................................................................................................26
      3.1.1 Test program ...................................................................................................................26
      3.1.2 Extruding of the embossment .........................................................................................27
  4 Pull-out test of a real size embossment series ............................................................................29
    4.1 Design of the test program ....................................................................................................29
    4.2 Pilot tests ................................................................................................................................32
    4.3 Characteristics of the specimens ..........................................................................................32
    4.4 Test results ...........................................................................................................................34
    4.5 Evaluation of the ultimate behaviour ....................................................................................37

II. Numerical studies .......................................................................................................................39
  5 Simulation of the concrete type behaviour ..................................................................................39
    5.1 Introduction ..........................................................................................................................39
5.1.1 Background of the modelling ................................................................. 39
5.1.2 Concrete model details ............................................................................. 40
5.1.3 Program of development and application .................................................. 41
5.2 Reinforced concrete beam model ................................................................. 41
5.2.1 Test #1: reference beam ............................................................................ 41
5.2.2 Test #2: short beam ................................................................................... 45
5.3 Summary ....................................................................................................... 47
5.4 Local model of fictive embossment ............................................................... 48
5.4.1 Model development .................................................................................. 48
5.4.2 Parametric study on the fictive embossment ................................................. 51
5.5 Local model of circular embossment ........................................................... 52
5.5.1 Model development .................................................................................. 52
5.5.2 Effect of friction coefficient on the behaviour of circular embossment ....... 53
5.5.3 Refined model vs. simplified model .......................................................... 54
5.6 Summary ....................................................................................................... 56
6 Simulation of the steel type behaviour: three-step-model .................................. 58
6.1 Principles of the three-step-model ............................................................... 58
6.1.1 Modelling strategy ................................................................................... 58
6.1.2 Features of the finite element model ......................................................... 59
6.2 Simulation of the manufacturing process ..................................................... 60
6.2.1 Load – displacement relationship .............................................................. 60
6.2.2 Strains ..................................................................................................... 61
6.2.3 Summary .................................................................................................. 62
6.3 Simulation of the pull-out test: enlarged individual embossment .................... 63
6.3.1 Computational strategy ............................................................................ 63
6.3.2 Results of the numerical simulation ......................................................... 63
6.3.3 Summary .................................................................................................. 65
6.4 Simulation of the pull-out test: real size embossment series ............................ 66
6.4.1 Applied model ........................................................................................ 66
6.4.2 Comparison of shell and solid models ...................................................... 67
6.4.3 Application of solid model ....................................................................... 68
6.4.4 Summary .................................................................................................. 70
7 Simulation of the steel type behaviour: parametric study ................................ 71
7.1 Introduction .................................................................................................. 71
7.1.1 Model selection ....................................................................................... 71
7.1.2 Finite element model ............................................................................... 71
7.1.3 Mesh sensitivity analysis ......................................................................... 73
7.2 Verification of the model................................................................. 73
  7.2.1 Numerical and experimental behaviour.................................................... 73
  7.2.2 Characteristics of the load – displacement relationship .................. 74
  7.2.3 Strains ......................................................................................... 75
  7.2.4 Evaluation of the numerical model....................................................... 76
7.3 Effect of the embossment’s parameters on the ultimate behaviour.......... 76
  7.3.1 Coefficient of friction........................................................................ 76
  7.3.2 Plate thickness ................................................................................ 77
  7.3.3 Embossment height ....................................................................... 78
  7.3.4 Size effect ..................................................................................... 80
7.4 Summary............................................................................................... 83
8 Horizontal shear resistance calculation................................................... 85
  8.1 Introduction .......................................................................................... 85
  8.2 Horizontal shear resistance of embossment – concrete failure........... 85
    8.2.1 General...................................................................................... 85
    8.2.2 Calculation aspects of bearing resistance ...................................... 85
  8.3 Horizontal shear resistance of embossment – steel failure............... 87
    8.3.1 General...................................................................................... 87
    8.3.2 Resistance of enlarged embossment .......................................... 87
    8.3.3 Resistance of real size embossment series .............................. 91
9 Conclusions ............................................................................................. 94
  9.1 New scientific results................................................................. 94
    9.1.1 Theses of the dissertation in English ........................................ 94
    9.1.2 Theses of the dissertation in Hungarian .................................. 97
  9.2 Proposal for further research ......................................................... 100
    9.2.1 Experimental investigation and numerical studies ..................... 100
    9.2.2 Semi-empirical simulation based partial shear connection method .... 100
    9.2.3 Enhanced concrete modelling.................................................. 101
References .................................................................................................. 102
Publications on the subject of the thesis .............................................. 105
ABSTRACT

The subject of the ongoing research work is to analyze the composite action of the structural elements of composite floors by experimental and numerical studies with a special focus on the rolled embossments on the steel surface. The mechanical and frictional interlocks result in a complex behaviour and failure under horizontal shear. This is why the design characteristics can be determined only by standardized experiments. The experiments determine the property of the interface interlock of the shear zone and provide a uniform (smeared) value for the calculation regardless of the nature of the failure.

The main aim of the current research is to determine the longitudinal shear resistance which originates from the contribution of rolled embossments for composite floors, by applying new type of pull-out test and advanced numerical model. The current embossments are considered as individual connectors such as shear studs or bolts. The design method of shear fasteners is used which defines the theoretically possible failure modes of the connection and calculates the resistance values accordingly.

The local failure of a concrete encased embossment is defined by three components: (i) failure by the crushing of the concrete on the loaded side of the embossment, (ii) failure of the steel embossment due to yielding and deformation and (iii) friction after delamination of the interface. The first two failures are linked to the local behaviour of the embossment and further analyzed. The failure components are considered independent assuming that the weaker part of the connection fails while the other remains undamaged. The interaction of the failure modes is proposed to determine on a semi-empirical manner by embedded design constants. The analysis is executed in two directions according to the failure modes.

Numerical model is developed for the simulation of the concrete type embossment behaviour. The failure is governed by the concrete damage, the effect of steel failure is ignored. The characteristics of the behaviour are observed and tendencies of different geometrical and physical parameters are derived.

A novel experimental procedure is developed and a composite specimen is designed to study the behaviour of concrete encased embossments on steel strips under shear action. Experimental program is executed on individual enlarged spherical embossments whereon the characteristics of the steel type failure mode are determined. The failure is governed by the plastic failure on the embossment, the concrete damage is found negligible. The experimental procedure is extended to analyze real size embossment series. The tendencies of different geometrical parameters (plate thickness, embossment arrangement and spacing) are determined.
A three-step-model is developed to analyze the different phases of the manufacturing process and loading of the steel spherical embossment according to the experimental program. The extruding of the embossment is completed in the first two steps of the simulation, the pull-out test is performed in the third step. Only the failure of steel is considered in the model, the effect of concrete damage is ignored.

Parametric study is executed by the developed model on the manufacturing process, testing procedure and embossment’s behaviour. The tendencies are derived and evaluated.

On the basis of the characteristics of the steel type embossment failure a calculation method is proposed to determine the resistance of the embossment. The theory of yield lines is applied based on the ultimate experimental deformation of the enlarged individual embossment. The calculation is extended for real size embossment groups by considering the spacing of the embossments.

On the basis of the characteristics of the concrete type embossment failure a calculation method is proposed to determine the bearing resistance of the embossment. The link between the local failure of an embossment and the global failure of embossment series in a small-scale pull-out test can be linked by an empirical reduction factor.
ACKNOWLEDGEMENT

The research work is completed under the partial financial support of the following projects, foundations and cooperations:

- OTKA T049305 project, Hungarian National Scientific Research Foundation, Hungary,
- “Research scholarship for PhD students and young researchers”, founded by the German Academic Exchange Service DAAD,
- cooperation between the Budapest University of Technology and Economics (BME), Hungary and the Bauhaus University of Weimar (BUW), Germany,
- the scientific program of the “Development of quality-oriented and harmonized R+D+I strategy and functional model at BME” project (Project ID: TÁMOP-4.2.1/B-09/1/KMR-2010-0002),
- the material of the test specimens are provided by Lindab Ltd. and the BVM Ltd. which are gratefully acknowledged.

I would like to express my special thanks to my supervisor Prof. László Dunai (Budapest University of Technology and Economics, BME, Department of Structural Engineering) who supported and motivated me along the way and helped to improve the presented research.

I would like to thank Dr. Attila László Joó for his helps and teaching in the modelling field.

I would like to thank László Kaltenbach, Dr. Miklós Kállo and Mansour Kachichian (Structural Laboratory of the Department of Structural Engineering, BME) and all of the laboratory staff for helping me to perform the experimental part of the presented research.

I would like to thank Dr. Salem Georges Nehme, Dr. Rita Nemes (Structural Laboratory of the Department of Construction Materials and Engineering Geology, BME) and the laboratory staff for their helps to prepare the test specimens.

I would like to give special thanks to the members of the Research Training Group 1462 - Evaluation of Coupled Numerical Partial Models in Structural Engineering (Bauhaus University Weimar). I am grateful to them that I could experience research in a very hospitable and supportive international ambience. Personally I would like to thank to Prof. Frank Werner for his professional leadership during my study stay.

I would like to thank all kind of help and support to the members of the Department of Structural Engineering of the Budapest University of Technology and Economics where I completed my PhD studies.

Thanks are also to my family who has supported me during the university years and special thanks to my husband Balázs who provided all the background and emotional support that I needed to successfully carry my research work into execution.
1 INTRODUCTION

1.1 Background of the research

1.1.1 Composite slabs

Composite floor structures are used widely in the industrial or high-rise building constructions. The reinforced concrete floor deck is cast on trapezoidally corrugated steel sheeting which provides the formwork for the slab during the construction and bears the weight of poured concrete and construction loads. After the concrete hardens the profile deck and the concrete bears the loads together and the profile deck represents the part or entire of the tension reinforcement.

The cross-section of the profile deck can be (i) open and (ii) re-entrant – also known as dovetail rib, as shown in Figure 1. The interface interlock’s function is to transfer the horizontal shear between the steel-concrete surfaces. Three different kind of interface interlock is applied, as it is shown in Figure 2: (1) mechanical interlock of indentations or embossments, (2) frictional interlock for dovetail ribs (3) end anchorage (e.g. welded studs or deformation of the ribs at the end). Note that a chemical adhesive bond exists between at the interface which is created by the hardening of the cement but it is not allowed to consider as interlock.

The embossment type mechanical bond has various geometrical shapes like spherical, rectangular, long-shaped (straight or inclined), V-shaped, etc as it is shown in Figure 3.
The diversity of the embossment and profile geometry result in a large number of possible combinations and each arrangement produces different effectiveness in the interface.

1.1.2 Structural behaviour

Composite floor is to design for three typical failure modes (Figure 4): bending, vertical shear and horizontal shear whereof the last is the most common failure type. In this last case the resistance of the slab is defined by the horizontal shear resistance of the interface interlock which is mainly created by the rolled embossments. The embossments increase the roughness of the surface and insure the horizontal shear transfer besides friction after the chemical adhesive bond is lost. The embossments in open profiles work also against vertical separation of the steel sheeting and concrete slab, as it is shown in Figure 5.

![Flexural Failure](image1)

![Vertical Shear Failure](image2)

![Horizontal Shear Failure](image3)

**Figure 4. Failure models of composite floors [7]**

The horizontal shear failure phenomenon results in a longitudinal slip between the concrete and steel and it is a complex combination of the failure of the steel embossments on the sheeting surface and the concrete indentations around them which have an influence on each other all along the loading and failure. The horizontal shear resistance of a composite slab is determined on a semi-empirical manner whereof the characteristics of the interface interlock are determined by experiments.

1.1.3 Testing and design

The first testing procedure of composite slabs is based on full-scale specimens which are one way bended slabs [9], [10]. Full-scale test is also approved and described in the standards. The test specimens are later reduced. Recently an intermediate size specimen is introduced [11] which is four-point-bended beam with the same span than a slab but the cross-section is only one rib wide. The most popular small-scale tests are, however, the pull-out [12] and push-out tests [13], [14] which consist of a concrete encased profile rib subjected to pull or push loading. The drawback of this testing procedure is that it does not take into account the
effect of bending which is acting in the real structure. The test procedures are shown in Figure 6.

Based on full-scale and small-scale tests – whereof the \(m\) and \(k\) design constants or the \(\tau_s\) shear strength is determined – the horizontal shear resistance can be computed by the \(m-k\) method (see in Chapter 1.1.4) or the partial shear connection method (see in Chapter 1.1.5) [1]. Beyond the standards alternative methods are proposed (not discussed in details): the new simplified method was developed by Crisinel and Marimon [15] in 2004 which also uses test data of small scale pull-out tests to define a tri-linear moment-curvature relationship for composite floors. Abdullah and Easterling [11] in 2008 developed a calculation method which considers the slab slenderness as the strength parameter. The shear strength – end slip relationship is derived from beam bending tests using the force equilibrium method.

![Bending tests](image1)
![Pull-out test](image2)
![Push-out test](image3)

Figure 6. Evolution of the test specimens for composite floors: (a.1) full-scale bending test [9] and (a.2) beam bending test [11], small scale (b) pull-out test [12], (c) push-out test [13] [14]

Numerical models are also introduced [16]-[18] to follow the behaviour of composite floors. The layout of the models is practically the same regardless of the used finite element program. The concrete is modelled by solid elements, the steel part is modelled by shell elements and the interlock is modelled by different interface elements whereof the characteristics are described by full-scale and small-scale tests. The main point of the model is always the characterization of the interface interlock. Once the local behaviour is well captured and implemented, the global model behaves well, too.

1.1.4 The \(m-k\) method

The longitudinal shear resistance of composite slabs with mechanical or frictional interlock (no end anchorage) can be determined by the \(m-k\) method of the Eurocode 4 standard [1]. The method is applied when the horizontal shear behaviour of the slab is either brittle or ductile. The behaviour is ductile if the failure load exceeds the load causing a recorded end slips of
0.1 mm by more than 10%. The method gives a limit \( V_{1,Rd} \) in Eq. (1) for a width of slab \( b \) which is to be greater than the maximum design shear force \( V_{Ed} \):

\[
V_{1,Rd} = \frac{b \cdot d_p}{\gamma_{Vs}} \left( \frac{m \cdot A_p}{b \cdot L_s} + k \right) \geq V_{Ed}
\]  

where,

- \( b, d_p \) defined in Figure 7 and are in mm,
- \( A_p \) the nominal cross-section of the sheeting in \( \text{mm}^2 \),
- \( m, k \) design values for the empirical factors in \( \text{N/mm}^2 \) obtained from slab tests meeting the basic requirements of the m-k method,
- \( L_s \) the shear span in mm,
- \( \gamma_{Vs} \) the partial factor for the ultimate limit state.

The design constants are determined from a series of full-scale slab specimens. Two groups of three tests are used (groups A and B in Figure 7). For specimens of group A or B the shear span is as long or as short as possible, respectively, while still providing longitudinal shear failure. The design relationship is formed by the linear regression line through these characteristic values for groups A and B. The value of the representative experimental shear force \( (V_t) \) is calculated from the value of the failure load \( (W_t) \) as follows:

\[
V_{t,ductile} = 0.5 \cdot W_t \quad \text{if the behaviour is ductile, and}
\]

\[
V_{t,brittle} = 0.8 \cdot V_{t,ductile} \quad \text{if the behaviour is brittle.}
\]

![Figure 7. Evaluation of the test results by the m-k method [1]](image)

### 1.1.5 The partial shear connection method

The longitudinal shear resistance of composite slabs with mechanical or frictional interlock (no end anchorage) can be determined by the partial shear connection (PSC) method of the Eurocode 4 standard [1]. The PSC method can be applied if the horizontal shear behaviour of the slab is ductile. If the partial connection method is used it should be proved that at any cross-section the design bending moment \( M_{Ed} \) does not exceed the design resistance \( M_{Rd} \) (Figure 8). At the calculation of the moment resistance \( N_{cf} \) is to be replaced by \( N_c \) given by Eq. (4).
\[ N_c = \tau_{u, Rd} \cdot b \cdot L_x \leq N_{cf} \]  

(4)

where,
- \( N_{cf} \) is the concrete compressive force for plastic bending failure,
- \( \tau_{u, Rd} \) is the design shear strength (definition in Figure 10),
- \( L_x \) is the distance of the considered cross-section to the nearest support.

Neutral axis is above the steel sheeting

Neutral axis is in the steel sheeting

Figure 8. Stress distribution in the cross-section due to bending [1]

The value of \( \tau_{u, Rd} \) is determined from full-scale bending tests by the followings. The bending moment \( M_{test} \) at the cross-section under the point load from the maximum applied loads is to be determined. Using the partial interaction diagram path A → B → C determines the degree of shear connection \( \eta \) (Figure 9). For each test the value of \( \tau_u \) is to be calculated by Eq. (5).

\[ \tau_u = \frac{\eta \cdot N_{cf}}{b \cdot (L_x + L_0)} \]  

(5)

The characteristic shear strength \( \tau_{u, rk} \) should be calculated from the test values and the design shear strength \( \tau_{u, Rd} \) is the characteristic strength \( \tau_{u, rk} \) divided by the partial factor \( \gamma_{vs} = 1.25 \) (recommended). The design relationship of partial interaction (Figure 10) is determined based on the value \( \tau_{u, Rd} \), which gives the bending resistance of the cross-section that is \( L_x \) far from the support.
By Figure 10 it is concluded that:
if $L_x \geq L_{sf}$, the shear connection is full, the bending resistance of the cross-section is relevant,
if $L_x < L_{sf}$, the shear connection is partial, the longitudinal shear resistance is relevant.
The load-bearing criterion requires that the design value of the bending moment from the loads cannot exceed the bending moment resistance. The verification is illustrated in Figure 11.

The value of $\tau_u$ can be directly determined from small-scale pull-out tests, too (e.g. Freire, [32]) as follows:

$$\tau_u = \frac{F_{ult}}{A_s}$$

where,
$F_{ult}$ ultimate load of the pull-out test,
$A_s$ concrete encased steel surface.
1.2 Research program

1.2.1 Aims

In the horizontal shear strength calculation the performance tests are necessary since each steel deck profile has its own unique shear transferring mechanism. The purpose of the tests is to provide data for the ultimate strength design equations. In particular, a series of tests is needed in order to provide ultimate experimental shear resistance for linear regression analysis of the relevant parameters affecting the shear-bond capacity. However, the laboratory tests are time consuming and expensive to make. Furthermore small-scale tests especially need precise manufacturing process and special loading conditions.

The aim of the current research is to determine the horizontal shear resistance at embossment level, as shown in Figure 12. The goal is to understand the local failure phenomena of an embossment and to create a semi-empirical calculation method for the resistance calculation.

![Figure 12. Derivation of the local test from the recent test specimens](image)

1.2.2 Problems to be solved

Embossments are investigated in groups so far: the full scale test analyses the entire structure and the small scale tests analyses a part of the shear zone as detailed in Chapter 1.1.3. Those specimens are not suitable for the analysis of one embossment and they cannot predict the failure mode of it. Experimental background is needed which analyses one embossment individually according to the research aims.

Since the embossment type mechanical bond produces a complicated failure under horizontal shear the phenomenon can be followed by a complex model. The problem is highly nonlinear because it needs to follow material nonlinearity of the steel and also the concrete damage the same time. The complex model is needed in order to predict the mode of failure. The basic idea is to develop a calculation method similarly to other shear fasteners – like shear studs or bolts – by separating the possible failure modes and calculating the resistance values according to them. The interaction of the failure modes can be determined on a semi-empirical or empirical manner by embedded design constants in the local resistance calculations or by interaction equations. The definition of the interaction is not part of the current research, it requires large experimental background.
The failure of individual connectors is defined in the Eurocode 3 (for bolts) [19] and Eurocode 4 (for shear studs) [1] by two possible modes: shear and bearing. The design resistance of the fastener is defined by the relevant failure mode, the minimum of the shear and bearing resistances ($P_{v,Rd}$ and $P_{b,Rd}$). Based on the individual resistance values, assuming plastic re-arrangement of the connector forces the design resistance of a group of connectors is taken as:

$$P_{Rd} = n \cdot \min\left(P_{b,Rd}, P_{v,Rd}\right)$$

where $n$ is the number of fasteners. (7)

The aim is to develop an analogous calculation method for embossments.

**1.2.3 Research strategy**

The first step of the research is to understand the behaviour of an individual embossment. Since a large number of embossment creates the interaction in the structure; the analysis is to be extended to study the behaviour of an embossment series by the description of one embossment. The failure of an individual embossment under horizontal shear is a local phenomenon defined by the failure of the constituents. It is defined by three components: (i) failure by the crushing of the concrete on the loaded side of the embossment, (ii) failure of the steel embossment due to yielding and deformation and (iii) friction after delamination of the interface, as illustrated in Figure 13. The three components are considered independent by assuming that the weaker part of the connection fails while the other remains undamaged. The concrete type failure issues when the concrete is weaker then the steel part so the embossment crushes the indentation. The steel type failure issues when the steel part is relatively weaker then the concrete and the embossment yields and deforms. The delamination type failure is assumed to occur when both components are equally strong and none of them becomes dominant. Note that in the reality both steel and concrete type failures are assumed to involve a certain amount of damage of the other component but only the failure of the governing component is taken into account. In the calculation of the separated failure modes no interaction is considered only the dominant failure.

![Figure 13. Theoretical local failure modes of embossments](image)

The first two failures are linked to the local behaviour of the embossment and subjected to further analysis. The third failure type is analogous with the behaviour of non-embossed plates whereon the connection is realized by the friction. In case of embossed plates it means...
that the embossed type mechanical bond not realized and nor the steel neither the concrete part fails. This failure is not included in the detailed analysis.

In the research the possible failure modes are analyzed on separate models and in this way the models have to follow only one kind of failure (steel or concrete) which is easier to handle in the calculation. The basic models consist of one embossment and the surrounding concrete. According to the research idea it is essential to avoid the failure of one component while the other fails since the dominant failure mode is to follow and the interaction of the failure modes is to neglect.

The main focus of the research work is to establish the background the steel type embossment failure by experimental and numerical studies. The simulations need experimental background of an individual embossment. Embossment samples which are appropriate to individually analyse by pull-out test are not commercially available. Therefore the embossed steel strips are needed to be own made and the geometry of the embossment needs to be easy to manufacture and reproduce. Spherical embossment shape is chosen by the above mentioned reasons for the research. Thin-walled mild steel plates are used to make the embossed plates in the specimens. The concrete part of the specimens is chosen to be relatively stronger in the connection, normal and high performance concretes are used. The concrete type behaviour is analyzed by numerical investigations, experimental analysis is not made.

The steps of the research and the structure of the dissertation are as follows:

- an experimental background is assembled to analyse the behaviour of embossments emphasizing one of the separated failure mode: the steel type failure is supported by own experiments and the concrete type failure is supported by experiments found in the literature;
- the steel type failure is analyzed on an individual enlarged embossment with spherical shape subjected to pull-out test as it is detailed in Chapters 2 and 3, and followed by numerical model in Chapters 6 and 7;
- the test procedure and the developed model is applied to analyze a series of real size embossments to link the local behaviour of one embossment and the global behaviour of the interface made by numerous embossments in the structure as it is found in Chapters 4 and 6;
- the concrete type failure is followed by a numerical model for which a concrete material model is calibrated on global beam models and local embossment models in Chapter 5;
- a calculation method is developed to determine the horizontal shear resistance for steel type and concrete type embossment failures in Chapter 8.
- on the basis of the research five new scientific results are concluded in Chapter 9.
1. EXPERIMENTAL INVESTIGATIONS

In the next three chapters the experimental investigations of the research work are presented. The aim of the research is to analyse the local behaviour of embossments under shear action. The tests are designed to be able to analyse the embossments as individual fasteners. Special pull-out tests are carried out to follow the behaviour of the rolled embossments apart from the structure. The pull-out specimens are designed with the goal to have the steel part weaker than the concrete which insures that the ultimate behaviour of the specimens is governed by the failure of the steel embossment and the concrete damage is negligible. An individual and enlarged embossment is investigated first and followed by detailed measurement, including the analysis of the manufacturing process of the embossment. Then the investigation is applied to analyse a series of real size embossments under the same conditions then the enlarged embossment to determine the relationship between the two configurations.

2. PULL-OUT TEST OF AN INDIVIDUAL ENLARGED EMBOSSMENT

The composite action of embossments is followed by a new composite specimen. The test specimens are designed on the basis of traditional pull-out tests, with the difference that the steel plate is not a half wave of an open trough profile, but a steel strip which has one enlarged embossment on it, as shown in Figure 14. The shape of the embossment is chosen to be spherical. The scope of the enlargement of the embossment is to be able to create the specimen and to follow the failure phenomenon by strain gauge measurement.

2.1 Test program

2.1.1 General

The extruding of the spherical embossment is executed as cold forming on the steel plate with a bearing ball of d = 45 mm, as shown in Figure 15. The steel plate was pinched between two forming plates whereof the upper plate thickness is chosen to be bigger than the half of the diameter of the bearing ball. A 45 mm hole is cut on the upper forming plate to lead the bearing ball. An indentation was cut in the bottom forming plate with identical geometry with the expected embossment shape. The bearing ball is pushed against the steel plate until the height of the embossment (10 mm) was reached and its diameter became 37.4 mm. In this way an about four times bigger connection is formed then a real one [14]. In order to keep the quasi-original geometric ratio of the embossment, the steel plate thickness is chosen to be thicker than the plate thickness in a regular composite floor; 1.5 and 2 mm.

Two embossed plates are placed back-to-back in the middle of a concrete cube. A 6 mm thick spacer plate is installed between the embossed plates. An 80 mm diameter hole is cut on the
spacer plate around the embossment, to leave the area of the connection without restraint inside and to insure the free deformation of the embossment.

![Figure 14. Pull-out specimen](image1)

![Figure 15. Extruding of the embossment](image2)

The thin plates and the spacer plate are connected on their edges with spot welding, and finally the edges of the plate pile are covered with waterproof adhesive tape. In the design of the specimen it was aimed to avoid the global failure of the concrete block splitting, hence closely distributed stirrups (by 30 mm) are applied in the concrete block along the plate.

2.1.2 Materials and measurement technology

The applied concrete material is C25/30 whereof the recipe is designed and used for previous experimental investigation by the author [20]. Since the material parameters are previously determined, only the strength of the new mixture is needed to be checked by uniaxial compression tests on cube specimens of 150 mm edge length. The cube specimens are made of the same mixture of concrete as the pull-out specimens. The concrete recipe is detailed in Table 1. Since the pull-out tests were executed at the 15th day after the casting of concrete, the strength of the concrete is determined on the same day. The compression test is executed on three specimens (details in Table 2).

<table>
<thead>
<tr>
<th>Table 1. Concrete recipe of the enlarged pull-out test</th>
</tr>
</thead>
<tbody>
<tr>
<td>C25/30</td>
</tr>
<tr>
<td>Cement CEM I 42.5 N</td>
</tr>
<tr>
<td>Water v/c=</td>
</tr>
<tr>
<td>Aggregates 0/4</td>
</tr>
<tr>
<td>4/8</td>
</tr>
<tr>
<td>8/16</td>
</tr>
<tr>
<td>Additive SIKA Viscocrete 5 neu</td>
</tr>
</tbody>
</table>

It is found that the calculated average compressive strength of the actual concrete mixture is 43.35 N/mm², and the density of the concrete is 2308.68 kg/m³. The density of the actual
mixture agreed with the previous material test; however the compressive strength is found 15% higher than the expected value [20].

Table 2. Material test for concrete in the enlarged pull-out test

<table>
<thead>
<tr>
<th>Geometry</th>
<th>( b_1 ) [mm]</th>
<th>149.5</th>
<th>149.7</th>
<th>149.9</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>( b_2 ) [mm]</td>
<td>150.0</td>
<td>152.2</td>
<td>151.8</td>
</tr>
<tr>
<td></td>
<td>( h ) [mm]</td>
<td>150.1</td>
<td>150.0</td>
<td>149.6</td>
</tr>
<tr>
<td>Weight</td>
<td>[kg]</td>
<td>7.79</td>
<td>7.88</td>
<td>7.85</td>
</tr>
<tr>
<td>Force</td>
<td>[kN]</td>
<td>987.8</td>
<td>970.6</td>
<td>987.3</td>
</tr>
<tr>
<td>Average compressive strength</td>
<td></td>
<td>43.35 N/mm(^2)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Average density</td>
<td></td>
<td>2308.68 kg/m(^3)</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

The reinforcement and the stirrups are made uniformly from 6 mm diameter B38.24 grade steel. The embossed steel plates are made with 340x120x1.5 mm and 340x120x2 mm geometry. The nominal grade of the steel is S350GD+Z [21] \((f_y = 35.5 \text{ kN/cm}^2, f_u = 51 \text{ kN/cm}^2)\). To obtain exact material data for the steel tensile tests are made on 6 specimens – three pieces from each plate thicknesses – to determine the characteristics of the material. The results of the tensile tests are found in Table 3.

Table 3. Material test for steel plate in the enlarged pull-out test

<table>
<thead>
<tr>
<th>Sign of the specimen</th>
<th>Thick/Depth</th>
<th>( \text{ReH} )</th>
<th>( \text{Rm} )</th>
<th>( \text{A80} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.5 mm nominal plate thickness</td>
<td>mm</td>
<td>N/mm(^2)</td>
<td>%</td>
<td></td>
</tr>
<tr>
<td>1.1</td>
<td>1.53/20.31</td>
<td>450</td>
<td>512</td>
<td>22.5</td>
</tr>
<tr>
<td>1.2</td>
<td>1.53/20.29</td>
<td>429</td>
<td>513</td>
<td>19.0</td>
</tr>
<tr>
<td>1.3</td>
<td>1.53/20.12</td>
<td>452</td>
<td>507</td>
<td>22.0</td>
</tr>
<tr>
<td>2 mm nominal plate thickness</td>
<td>mm</td>
<td>N/mm(^2)</td>
<td>%</td>
<td></td>
</tr>
<tr>
<td>2.1</td>
<td>1.92/20.24</td>
<td>457</td>
<td>532</td>
<td>18.5</td>
</tr>
<tr>
<td>2.2</td>
<td>1.92/20.24</td>
<td>462</td>
<td>536</td>
<td>19.0</td>
</tr>
<tr>
<td>2.3</td>
<td>1.92/20.40</td>
<td>458</td>
<td>533</td>
<td>19.0</td>
</tr>
</tbody>
</table>

The strains on the steel surface are measured with strain gauges of type KMT-LIAS-06-1.5/350-5E whereof the active grid length is 1.5 mm and the nominal measurement limit is 10% of strain. The strain gauges are placed in two arrangements, as shown in Figure 16. The strain gauges are glued on the inner face of the embossment. No strain gauges are put on the outer face, since the safe placement of the strain gauges cannot be ensured because of the posterior concrete casting. Five base gauges are put on all of the embossed plate pairs and on two of them ten supplementary gauges are also installed. In this way four specimens are made using 5 gauges and two specimens are made using 15 gauges. The 5 base gauges are placed in the axis of symmetry of the embossment according to the orientation of the load. Gauges #1 and 5 are placed on the plane surface at the bottom edge of the embossment. Gauges #2 and 4
are placed on the opposite side of gauges #1 and 5, respectively on the curved surface. Gauge #3 is put in the middle of the embossment. The role of the other 10 supplementary gauges is to determine the behaviour in the surrounding area of the base gauges. Hence, the gauges are placed next to the gauges #1, 2, 4 and 5 on the right and left side, and also between the gauges #2 – 3 and 3 – 4 to be able to follow the entire longitudinal deformation of the embossment.

![Figure 16. Strain gauges (a) basic and (b) supplementary (c) wax protection (d) inner side of the specimen](image)

After fixing the strain gauges on the steel surface, they are covered with a thin wax layer, to avoid the water penetration to the strain gauges due to the concrete casting. Altogether six specimens are made, with the details summarized in Table 4. Besides the strains, the relative displacement between the steel plate and the concrete cube is also measured with inductive transducers. The results of the strain measurement as well as the load and the displacement results are used for the verification of the numerical model. The importance of the strain measurement is to provide additional information on the ultimate behaviour and failure mechanism, since it cannot be followed inside the concrete block; only the undamaged and the completely destroyed states are visible.

<table>
<thead>
<tr>
<th>Specimen code</th>
<th>Sheeting thickness [mm]</th>
<th>Strain gauges [pc]</th>
<th>Concrete cube size [cm]</th>
<th>Steel plate size [mm]</th>
<th>Embossment diameter/height [mm]</th>
<th>$f_y/f_u$ * of steel [N/mm²]</th>
<th>$f_{ck}$ ** of concrete [N/mm²]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.1</td>
<td>1.5 mm</td>
<td>5</td>
<td>20x20x20</td>
<td>340x120</td>
<td>37.4/10</td>
<td>444/510</td>
<td>43.35</td>
</tr>
<tr>
<td>1.2</td>
<td>1.5 mm</td>
<td>5</td>
<td>20x20x20</td>
<td>340x120</td>
<td>37.4/10</td>
<td>444/510</td>
<td>43.35</td>
</tr>
<tr>
<td>1.3</td>
<td>1.5 mm</td>
<td>15</td>
<td>20x20x20</td>
<td>340x120</td>
<td>37.4/10</td>
<td>444/510</td>
<td>43.35</td>
</tr>
<tr>
<td>2.1</td>
<td>2 mm</td>
<td>5</td>
<td>20x20x20</td>
<td>340x120</td>
<td>37.4/10</td>
<td>459/534</td>
<td>43.35</td>
</tr>
<tr>
<td>2.2</td>
<td>2 mm</td>
<td>5</td>
<td>20x20x20</td>
<td>340x120</td>
<td>37.4/10</td>
<td>459/534</td>
<td>43.35</td>
</tr>
<tr>
<td>2.3</td>
<td>2 mm</td>
<td>15</td>
<td>20x20x20</td>
<td>340x120</td>
<td>37.4/10</td>
<td>459/534</td>
<td>43.35</td>
</tr>
</tbody>
</table>

* yield stress/ultimate stress  
** compressive strength
2.1.3 Preparation of the specimens, test execution

The test specimens are made in the cooperation of the Structural Laboratory of the Department of Structural Engineering and the Structural Laboratory of the Department of Structural Materials and Engineering Geology.

The pull–out test is executed in a loading frame, where the specimen is hung by loading plates which are fixed on the steel overhang of the specimen. The upper surface of the concrete cube is supported from above, as shown in Figure 17. The load is applied through the loading plates. To insure the centralized and uniform load transfer, and also to correct the concrete surface’s irregularity, ~5 mm thick hard rubber pads are used between the loading frame and the concrete cubes’ supported surface.

2.2 Evaluation of test results

2.2.1 Test results

Figure 18 and Figure 19 show the load-displacement curves of the six specimens. Since the experiment is planned as pilot investigation, the specimens with base gauges are tested first. The order of the specimens in the test is: 1.1, 1.2, 2.1, 2.2, 1.3 and 2.3. During the experiments the results are instantly evaluated and it is found that the load-displacement relationship remains quasi-rigid until the ultimate behaviour. In the followings it is cleared up that the inductive transducer stuck and it couldn’t measure the first phase when the displacements are quite small.

This mistake in the next three tests is corrected. Accordingly the first three tests are used for the evaluation of the ultimate load and the tests four to six are used for the qualitative evaluation of the overall behaviour. The effect of unloading is analysed on the test specimens with supplementary gauges: three and two unloading/reloading cycle is executed on the specimens 1.3 and 2.3, respectively.
2.2.2 Observed behaviour

As the load is introduced, the first mark of the failure is appeared on the concrete block. The first crack is appeared in line with the steel plate at the exterior surface of the concrete block. The first crack is shown at the side, where the steel plate is closer to. The crack is propagated all over the height of the cube and the steel plate is slipped out from the concrete block. The failed specimen is removed from the loading frame, and the crack pattern on the supported side is analyzed. Two kinds of crack can be identified from the full crack pattern. The first one is parallel with the steel plate and propagates from top to bottom in the concrete cube, and arises also on the supported and the side surfaces. The second one is representative on the supported surface and its near area, and the crack propagates from the edge of the steel plate to the corner of the concrete cube.

Figure 20 shows two typical measured load-displacement curves. The response of the specimen is almost rigid for the initial loading (2-3 kN). After the behaviour changes and a short linear phase is followed by a nonlinear part, with gradually decreasing slope. From the experimental observation, the end of the linear phase can be identified by a micro crack propagation, which leads to the appearance of the first crack on the concrete surface. After the steel plate slips, a small amount of load increase can be observed till failure. A significant decrease in the slope can be seen in the curves after the slip of the plate (it indicates the start of the plastic failure) at almost the same displacement level of 8-9 mm on every specimen. The global failure, however, occurs after 30-40 mm slips, which shows significant deformation capacity. In Figure 21/a and Figure 21/b the observed typical crack pattern can be seen. The deformation of the steel plate can be analyzed after removing the concrete cover. The concrete cube is completely destroyed, so the inner concrete failure cannot be seen.
Figure 20. Load – displacement curves of 1.3 and 2.2 specimens

The ultimate deformation of the embossment is shown in Figure 22. Analyzing the deformed shape, the plastic failure is identified by local bending on the embossment’s loaded surface. The failed surface has curved boundary and angular inner yield line pattern (emphasized by black marker on the top deformation) as it is typical for plastic failure of bended plates. Accordingly the ultimate experimental load is referred as plastic limit load.

Figure 21. Crack patterns (a) side cracks (b) top cracks

Figure 22. Ultimate deformation of the embossment (a) side view (b) top view
The point of the first yielding on the steel plate, which is marked on both of the curves, belongs to the strain measurement, when the yielding strain appears at one of the measured points. The value of yielding strain is determined from the material tests.

By the evaluation of the measured strains it is found, that the first yielding in the steel plate appears at very low load level (5–10 kN, as shown in Figure 20) at the bottom of the embossment on the loaded side (at gauge #2 position).

The order of the appearance of yielding at the base gauge positions can be followed in Figure 23. The values of the load levels referring to yielding are the average of the three specimens of the same kind. A typical result of the strain measurement can be seen in Figure 24 (specimens with 2 mm plate thickness, gauge #3).

The curve shows the relationship of the load and the strain in the centre of the embossment. The yielding strain (2300 μm/m) is marked in the diagram, as well. The character of the
curves is the same on the specimens made with 1.5 mm plate thickness, only the load level is smaller with those specimens. Supplementary gauges are put in the specimen to follow the longitudinal deformation at the nearby points of the base gauges. The gauges #2j and #2b are put in equal distance on the left and right hand side from gauge #2, as shown in Figure 16. Figure 25 shows the measured strains in specimen 1.3 at gauge #2 at the bottom of the embossment on the loaded side, where the first yielding is observed.

The supplementary gauges showed similar behaviour with the base gauge, placed in the same cross-section of the embossment. The minimal deviation between the #2j and #2b is assumed as the reason of (i) not exactly centric loading and/or (ii) not perfect positioning and/or (iii) asymmetric deformation of the embossment (keeping in mind that the deformation of the embossment is plastic and irreversible in this case).

2.3 Summary

A new composite test specimen is introduced to analyze the local behaviour of an individual embossments. The specimen is designed to emphasize the failure of the steel embossment and in the same time to avoid global concrete failure. The basic behaviour modes are observed from the tests and the results are evaluated quantitatively.

It is found that the ultimate behaviour is conducted by steel embossment failure due to local bending which results in yielding extension and the appearance of the plastic failure. It is found that the connection is ductile, the failure occurs after large plastic deformation of the embossment. Since the concrete cover is destroyed when removed, the inner concrete failure around the embossment cannot be followed, but the outer crack propagation is well captured on the specimen.

The change of the plate thickness has direct effect on the initial stiffness and the load carrying capacity, but it does not affect the global behaviour. The behaviour of the embossment is followed with strain gauge measurement. Early yielding appears on the plate at ~14% of the plastic limit load at the bottom of the embossment on the loaded side.

The results are used to study the complex phenomenon and further validation of the finite element model developed for the embossment’s behaviour.
3  EFFECT OF COLD FORMING OF THE INDIVIDUAL ENLARGED EMBOSSTMENT

Cold forming is a mechanical operation in which a metal shape is permanently deformed into a new shape, normally at room temperature. Cold forming increases the hardness and strength of the metal. Previous investigations showed that cold forming produces plastic strains and remarkable residual stresses in the material which has later an important influence on the behaviour. Plastic bending, followed by elastic springback, creates a nonlinear through-thickness residual stress distribution, in the direction of bending [22]. The residual stresses are characterized by a nonlinear through-thickness variation.

The magnitude of residual stresses due to cold forming can reach the 60% of the yield stress. The through thickness variation of the residual stresses cannot be measured on thin-walled plates, only surface stresses can be obtained by strain gauge measurement.

3.1 Extruding

The aim of the experiments is to discover the effect of the 3D extruding process on an embossment in the steel plate. The results are to be used for numerical model validation.

3.1.1 Test program

The test program includes two extruding tests of a thin steel plate of 1.5 mm: one without strain measurement for checking purposes and a second with strain measurement. The embossments’ geometry is similar to the enlarged embossment (detailed later in Chapter 3.1.2). The embossment is pressed with the same tool in the thin plate as in the case of the pull-out specimens (see in Chapter 2.1.1).

The experiment was executed in the structural laboratory CIB of the Bauhaus University of Weimar. The device which is used for the measurement is an electromechanical static universal (tensile-compression) testing machine of 250 kN (Figure 26/a).

The strains on the steel surface are measured with HBM strain gauges for high strain, type 1-LD20-6/120. Those strain gages can be used wherever there is extreme strain or compression (extended or shortened with more then 5%). The maximum elongation is

Figure 26. Loading (a) and strain gauge (b)
±100 000 µm/m (±10%). The geometry of the strain gauge and its specification can be seen in Figure 26/b.

### 3.1.2 Extruding of the embossment

A 160x160x1.5 mm plane plate was equipped with 3 strain gauges (Figure 27), on the plate’s bottom plane. A strain gauge (a) is put in the middle of the embossment. A second strain gauge (b) is put orthogonal to the 1st gauge and a 3rd gauge (c) is put with 45° rotation, 15 mm far from the middle. All of the strain gauges were put in a Φ = 42 mm circular area.

![Figure 27. Strain gauge arrangement](image)

Since the strain gauges are put at the bottom plane of the plate, the loading arrangement is needed to redesign (comparing to the original setup in Figure 15) to avoid the failure of the strain gauges due to the compression which may occur between thin the steel plate and the bottom forming plate (Figure 15). Thus, a 45 mm hole is cut in the bottom forming plate similarly to the upper forming plate. To avoid sharp edge around the embossment, a 3 mm/2 mm fillet is cut around the hole (Figure 28/a). The experiment is executed under load control in the testing machine (Figure 28/b, c), whereof the load increment is 0.001 kN.

The extruding process is first executed on a 160x160x1.5 mm steel plate without strain measurement to specify the correct support conditions and to predict the ultimate load which belongs to an embossment of 10 mm. The result of the test is shown in Figure 29; an ultimate load of 27.18 kN is achieved at a displacement of 10 mm. A horizontal slip of the thin plate is
observed during the preliminary experiment which is to be avoided when the main experiment takes place. After the preliminary test is executed the support conditions are modified to secure the plate between the forming plates. The unloading is also registered to determine the residual deformation (Figure 29). The ultimate load then increased to 33.06 kN which belonged to a vertical displacement of 10.23 mm. After the unloading a residual deformation of maximum 9.36 mm was found. Strains are measured during the experiment at three positions (Figure 27). The results of the measurement are summarized in Figure 30. The gauges provide results for a limited load level. Gauge $a$ measured up to 1.56 kN, gauge $b$ measured up to 0.92 kN and gauge $c$ measured up to 7.66 kN (3, 5 and 23 % of the ultimate load, respectively). The gauge $b$ measured for the longest time and gauge $a$ is used up to the reliability limit of the gauge (10% of elongation). The inward and outward deformation of the extruded embossment is shown in Figure 31.

![Load-displacement curve of the experiments](image1)

![Strain measurement data](image2)

![Ultimate deformation](image3)
4 PULL-OUT TEST OF A REAL SIZE EMBOSSMENT SERIES

4.1 Design of the test program

The pull-out test of the real size embossment series (later called as small pull-out test) is designed based on the pull-out test of the individual enlarged embossment (later called as enlarged pull-out test). The height of the spherical embossment is 3.35 mm and the base diameter is 12 mm. Three plate thicknesses are chosen from the currently used composite profile deck thicknesses: \( t = 0.7, 1.0 \) and \( 1.2 \) mm. The grade of the steel is S350GD+Z [21]. The steel part (290x40xt mm) of the specimen composes of a three layer plate pile of two thin plates, separated with a spacer plate. The embossment pattern is cold-extruded in the thin plates. The embossments are formed by the helps of forming plates (Figure 32).

![Figure 32. Forming of the embossments (a) and the extruded embossment (b)](image)

The expected shape of the embossment is cut in the bottom forming plate and leader holes for the bearing balls in the upper forming plates. The bearing balls are pushed through the holes against the thin steel plate which is between the forming plates. The edge of the indentations on the bottom forming plates is not rounded which results having sharp edge around the embossments on the steel plate. The plate pile is kept together by gluing the edge of the plates together with cyanoacrylate. Note that the spot welding of the plates is in this case not possible because of the small size of the specimens.

The size of the concrete part of the specimen is 72x72x144 mm which is casted in a 150x150x150 mm formwork with the help of additional formwork plates. The plans of the specimen and the steel plates are shown in Figure 33. The formwork plates separate the cubic formwork in four equal parts and insure the centric and horizontal position of the plate pile which is embedded in the concrete prism, as it is shown in Figure 34.

The global failure of the concrete is avoided. Internal reinforcement obviously cannot be put in the specimen because of size issues. An external fixing is planned consequently, as shown in Figure 35. The width of the fixing is adjusted to be the same size then the specimen by the helps of wooden lining. The fixing parts are bolted together with M12 10.9 bolts.
Three embossment patterns are analyzed. The maximum number of the embossments is four, the minimum is two (Table 5). The basic pattern includes four embossments located 23 mm far from each other (the distance of the middles is 35 mm). The two patterns including two embossments are formed by (i) keeping the two middle embossments and (ii) keeping only the second and the fourth embossment. The surface of the steel plates is kept “as rolled” (neither cleaned nor oiled).
Table 5. Small pull-out specimen types

<table>
<thead>
<tr>
<th>Nominal plate thicknesses [mm]</th>
<th>Patterns</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td><img src="example.png" alt="Pattern" /></td>
</tr>
<tr>
<td>0.7</td>
<td>07.4</td>
</tr>
<tr>
<td>1.0</td>
<td>10.4</td>
</tr>
<tr>
<td>1.2</td>
<td>12.4</td>
</tr>
</tbody>
</table>

Eurocode 4 [1] determines the loading conditions for small scale push-out tests as follows:
- the load should be applied in increments up to 40% of the expected failure load and cycled 25 times between 5% and 40% of the expected failure load,
- the subsequent load increments should then be imposed such that failure does not occur in less than 15 minutes.

![Test setup of the small pull-out test](example.png)

Figure 36. Test setup of the small pull-out test

The loading of the specimens is determined using the above principles adjusting to the current test characteristics. The test setup is shown in Figure 36. The size of the specimen is relatively small comparing to standard push tests so the number of the cycles is reduced to 10 and the failure load is achieved within not shorter then 10 minutes. The loading is performed under load control; however, the ultimate displacement is controlled by specimen types by the followings:
- the contact between the first embossment that comes out of the concrete block and between the support plates is to avoid,
- the maximum displacement is limited by the distance of the embossments in the pattern.
That means that the tests of the four and two-embossment specimens are stopped at an ultimate displacement of 15-25 / 30-50 mm if other circumstances during the tests do not overwrite those values.

4.2 Pilot tests

The test program is planned to include a total number of 27 specimens. To check the behaviour pilot experiments are executed on 4 specimens with the 4-embossment pattern using two plate thicknesses 0.7 and 1.0. The cyclic loading is not performed in case of the pilot tests the load is monotonically increased until failure. The behaviour and failure agrees well with the expectations based on the enlarged pull-out test: the ultimate behaviour is governed by the failure of the embossments on the steel surface. During the loading a vertical crack becomes first visible which starts from the supported side of the concrete along the steel plate. The crack propagates to the top and meantime the plate pile starts to slip out of the concrete block.

Importance of the transversal compression of the specimen from the fixing occurred by the pilot test and further analyzed by the test series. The transversal compression is controlled by the screws of the fixing. The aim is to reach the minimal screwing magnitude which is enough to avoid the opening of the vertical crack. Higher transversal load is attained by tight screwing the fixing around the specimen. The magnitude of the transversal load is not measured.

4.3 Characteristics of the specimens

It is found by the enlarged pull-out test that the centric loading is essential to get accurate data from the tests. It is essential then to keep the plate pile unmoving during the concrete casting. So the use of general compacting methods like vibro-table is excluded. By this reason the concrete recipe is chosen to be a self compacting high strength concrete (see in Table 6).

<table>
<thead>
<tr>
<th>Ingredient</th>
<th>Specification</th>
<th>kg/m³</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cement</td>
<td>CEM I 42.5N</td>
<td>350</td>
</tr>
<tr>
<td>Limestone powder</td>
<td></td>
<td>220</td>
</tr>
<tr>
<td>Water</td>
<td></td>
<td>158</td>
</tr>
<tr>
<td>Aggregate</td>
<td>0/4 and 4/8</td>
<td>776 and 875</td>
</tr>
<tr>
<td>Additive</td>
<td>Glenium C300</td>
<td>7.35</td>
</tr>
</tbody>
</table>

A total of 27 small pull-out specimens are made by the patterns which are summarized in Table 5. Three specimens are made of each kind. The specimens are removed from the formwork at the age of one day and dried at open air.

With every specimen series three concrete test cubes of 150 mm edge length are made to check the compressive strength of the concrete on the day when the composite specimens are tested (Table 7). The compressive strength values are averaged for every three test cubes.
made one day and the value is further used in the modelling process. The Young’s modulus of the concrete is determined on 70x70x250 mm size specimens from the concrete of the first day. By the average of three specimens the value of the Young’s modulus is 32 553 N/mm$^2$. The steel material is tested to provide material model for the numerical simulations (Table 8). It is found that all of the plate thicknesses are 6-7% less then the nominal plate thickness. It is found also that the 1.2 mm thick plate has 7-8% smaller yield strength and 5-6% smaller ultimate strength then the thinner (0.7 and 1.0 mm) plates, which are to be taken into account in the evaluation of the test results. The summary of the test specimens is found in Table 9 - Table 11.

Table 7. Material test for concrete

<table>
<thead>
<tr>
<th>Mark</th>
<th>1.1</th>
<th>1.2</th>
<th>1.3</th>
<th>2.1</th>
<th>2.2</th>
<th>2.3</th>
<th>3.1</th>
<th>3.2</th>
<th>3.3</th>
<th>4.1</th>
<th>4.2</th>
<th>4.3</th>
</tr>
</thead>
<tbody>
<tr>
<td>$f_c$</td>
<td>54.70</td>
<td>53.00</td>
<td>55.15</td>
<td>54.46</td>
<td>56.07</td>
<td>57.75</td>
<td>60.70</td>
<td>57.22</td>
<td>60.12</td>
<td>57.20</td>
<td>56.91</td>
<td>56.82</td>
</tr>
<tr>
<td>$f_{c,avg}$</td>
<td>54.28</td>
<td>56.09</td>
<td>59.35</td>
<td>56.98</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table 8. Material test for steel

<table>
<thead>
<tr>
<th>Thickness* [mm]</th>
<th>Thickness [mm]</th>
<th>Width [mm]</th>
<th>$f_y$ [N/mm$^2$]</th>
<th>$f_u$ [N/mm$^2$]</th>
<th>$\varepsilon_u$ [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. 0.7</td>
<td>0.64</td>
<td>11.8</td>
<td>412</td>
<td>481</td>
<td>15</td>
</tr>
<tr>
<td>2. 0.7</td>
<td>0.65</td>
<td>12.1</td>
<td>401</td>
<td>473</td>
<td>14.5</td>
</tr>
<tr>
<td>3. 1.0</td>
<td>0.94</td>
<td>12.1</td>
<td>407</td>
<td>487</td>
<td>15</td>
</tr>
<tr>
<td>4. 1.0</td>
<td>0.94</td>
<td>12.3</td>
<td>401</td>
<td>487</td>
<td>16.5</td>
</tr>
<tr>
<td>5. 1.2</td>
<td>1.12</td>
<td>12.4</td>
<td>382</td>
<td>455</td>
<td>18</td>
</tr>
<tr>
<td>6. 1.2</td>
<td>1.12</td>
<td>12.3</td>
<td>373</td>
<td>453</td>
<td>18</td>
</tr>
</tbody>
</table>

*Nominal thickness of the plate

Table 9. Pull-out test specimens with 1.2 mm plate thickness

<table>
<thead>
<tr>
<th>Mark (nr.thickness.type)</th>
<th>Casting [date]</th>
<th>Test [date]</th>
<th>Age [day]</th>
<th>$f_c$ [N/mm$^2$]</th>
<th>$f_y$ [N/mm$^2$]</th>
<th>$f_u$ [N/mm$^2$]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.12.4</td>
<td>2011.07.27</td>
<td>2011.08.10</td>
<td>14</td>
<td>54.28</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2.12.4</td>
<td>2011.07.27</td>
<td>2011.08.11</td>
<td>15</td>
<td>54.28</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3.12.4</td>
<td>2011.07.27</td>
<td>2011.08.11</td>
<td>15</td>
<td>54.28</td>
<td></td>
<td></td>
</tr>
<tr>
<td>1.12.2k</td>
<td>2011.07.27</td>
<td>2011.08.11</td>
<td>15</td>
<td>54.28</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2.12.2k</td>
<td>2011.07.27</td>
<td>2011.08.11</td>
<td>14</td>
<td>56.09</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3.12.2k</td>
<td>2011.07.28</td>
<td>2011.08.11</td>
<td>14</td>
<td>56.09</td>
<td></td>
<td></td>
</tr>
<tr>
<td>1.12.2s</td>
<td>2011.07.28</td>
<td>2011.08.12</td>
<td>14</td>
<td>56.09</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2.12.2s</td>
<td>2011.07.28</td>
<td>2011.08.12</td>
<td>15</td>
<td>56.09</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3.12.2s</td>
<td>2011.07.28</td>
<td>2011.08.12</td>
<td>15</td>
<td>56.09</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Table 10. Pull-out test specimens with 1.0 mm plate thickness

<table>
<thead>
<tr>
<th>Mark (nr.thickness.type)</th>
<th>Casting [date]</th>
<th>Test [date]</th>
<th>Age [day]</th>
<th>$f_c$ [N/mm$^2$]</th>
<th>$f_y$ [N/mm$^2$]</th>
<th>$f_u$ [N/mm$^2$]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.10.4</td>
<td>2011.07.29</td>
<td>2011.08.12</td>
<td>14</td>
<td>59.35</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2.10.4</td>
<td>2011.07.29</td>
<td>2011.08.12</td>
<td>14</td>
<td>59.35</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3.10.4</td>
<td>2011.07.29</td>
<td>2011.08.12</td>
<td>14</td>
<td>59.35</td>
<td></td>
<td></td>
</tr>
<tr>
<td>1.10.2k</td>
<td>2011.08.01</td>
<td>2011.08.15</td>
<td>14</td>
<td>56.98</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2.10.2k</td>
<td>2011.08.02</td>
<td>2011.08.16</td>
<td>14</td>
<td>55.39</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3.10.2k</td>
<td>2011.08.02</td>
<td>2011.08.16</td>
<td>14</td>
<td>55.39</td>
<td></td>
<td></td>
</tr>
<tr>
<td>1.10.2s</td>
<td>2011.08.01</td>
<td>2011.08.15</td>
<td>14</td>
<td>56.98</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2.10.2s</td>
<td>2011.08.01</td>
<td>2011.08.15</td>
<td>14</td>
<td>56.98</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3.10.2s</td>
<td>2011.08.01</td>
<td>2011.08.15</td>
<td>14</td>
<td>56.98</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table 11. Pull-out test specimens with 0.7 mm plate thickness

<table>
<thead>
<tr>
<th>Mark (nr.thickness.type)</th>
<th>Casting [date]</th>
<th>Test [date]</th>
<th>Age [day]</th>
<th>$f_c$ [N/mm$^2$]</th>
<th>$f_y$ [N/mm$^2$]</th>
<th>$f_u$ [N/mm$^2$]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.07.4</td>
<td>2011.08.02</td>
<td>2011.08.16</td>
<td>14</td>
<td>55.39</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2.07.4</td>
<td>2011.08.02</td>
<td>2011.08.16</td>
<td>14</td>
<td>55.39</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3.07.4</td>
<td>2011.08.03</td>
<td>2011.08.17</td>
<td>14</td>
<td>55.49</td>
<td></td>
<td></td>
</tr>
<tr>
<td>1.07.2k</td>
<td>2011.08.03</td>
<td>2011.08.17</td>
<td>14</td>
<td>55.49</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2.07.2k</td>
<td>2011.08.04</td>
<td>2011.08.18</td>
<td>14</td>
<td>58.98</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3.07.2k</td>
<td>2011.08.04</td>
<td>2011.08.18</td>
<td>14</td>
<td>58.98</td>
<td></td>
<td></td>
</tr>
<tr>
<td>1.07.2s</td>
<td>2011.08.03</td>
<td>2011.08.17</td>
<td>14</td>
<td>55.49</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2.07.2s</td>
<td>2011.08.03</td>
<td>2011.08.17</td>
<td>14</td>
<td>55.49</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3.07.2s</td>
<td>2011.08.04</td>
<td>2011.08.18</td>
<td>14</td>
<td>58.98</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

4.4 Test results

From the total set of specimens twenty-five tests are analyzed. The load-displacement curves of the analyzed test results are shown in Figure 37 - Figure 41. The observed behaviour of the specimens agrees well with the pilot test results. The vertical crack which is parallel with the steel plate and propagates from the support along the concrete block emerges on every specimen. Note that this crack does not reach the top of the concrete block in case of ‘2k’ type specimens since there are no embossments on the plate around this region.

Since the magnitude of the transversal compression is not measured qualitative evaluation is made based on the ultimate load and behaviour of the specimens. It can be seen on the test results that all 07.2s, 1.07.4, 3.10.2s, 2.12.2s, 3.12.2s and 2.12.2k specimens (whereon the fixing is tight screwed) produce different character of load-displacement curves, although they have a visibly matching failure mode with the other specimens. The initial phase of the curves ends in a peak and then the load continuously decreases. The ultimate load is always
higher on the tight screwed specimens then on the minimally screwed specimens (definitions in Chapter 4.2).

![Figure 37. Test results of 12.4 and 12.2s specimens](image1.png)

![Figure 38. Test results of 12.2k and 10.4 specimens](image2.png)

![Figure 39. Test results of 10.2s and 10.2k specimens](image3.png)
Comparing to the failure of the enlarged individual embossment the real size embossment does not collapse completely rather it keeps a shape that is deformed at the loaded side and the back part of the embossment remains spherical, as it is shown in Figure 42. This shape of deformation is valid on the enlarged embossments until the ultimate load then the back part crushes as well.

The first embossments do not become as effective as the last three, because of the concrete damage at the supported face. After removing the fixing the internal concrete failure can be studied. A cone-type crack develops on the specimens at the loaded side, as it is seen in Figure 43. The broken concrete part causes softening in front of the first embossment. Beyond the cracking at the supported surface on the concrete block the inner concrete damage in front of the embossments is the same in both tests. A straight scratch can be seen along the path where the embossment is slipping out of the concrete block.
The concrete damage at the supported surface changes with the specimen types (Figure 44). The most significant damage at the supported concrete face occurs in the case of the 4-embossment-specimen (‘4’ type). In the case of 2-embossment-specimens (‘2k’ and ‘2s’ types) the supported surface normally does not undergo local concrete damage. If there is local concrete cracking, it is quasi parallel with the supported face.

The deformation of the steel embossments in the 2-embossment-specimens is visually the same. The difference between the ‘2k’ and ‘2s’ type specimens is the spacing of the embossment which results in individual embossment deformation in the case of ‘2s’ type specimens and grouped embossment deformation in the case of ‘2k’ type specimen where the deformation of the two embossments influences each other.

### 4.5 Evaluation of the ultimate behaviour

As it is expected the number of embossments and the plate thickness influences the load bearing capacity of the specimens. To evaluate the tests, the load-displacement results are compared by specimen types. The ultimate load is evaluated on 9 specimens (one from each kind, whereon the test uncertainties are avoided) and the load-bearing capacity of them is considered as representative value of the specimen kind.

The evaluation of the experimental ultimate load results by specimen type and by plate thickness is shown in Figure 45. In general the expected tendencies are observed. The 4-embossment specimens show higher load-bearing capacity than the 2-embossment ones. The 4-embossment-specimens does not carry twice as much load as the 2-embossment-specimens, but note that the first embossment in the ‘4’ type specimens is less effective then the other
three due to concrete damage at the support. Note that inactive embossments are assumed to be in the real structure, as well at the edge of the slab, but their importance in the load bearing is less important then in the specimen. This effect should be avoided further in the specimens by sufficient concrete cover.

![Comparison of the filtered test results by specimen types](image)

Figure 45. Comparison of the filtered test results by specimen types

The ‘2s’ type specimens show slightly higher load-bearing capacity then the ‘2k’ type ones. The reason of it is identified by the fact that the embossments in the ‘2s’ type specimens behave more individually then the embossments in the ‘2k’ type specimens. It is also observed that the load-bearing capacity decreases by reducing the plate thickness. More accurate evaluation of the tendencies cannot be done due to the small number of test results.
II. NUMERICAL STUDIES

In the next three chapters the numerical investigations of the research work are presented. According to the research strategy the numerical analysis of the rolled embossment splits into two parts: (1) simulation of the concrete damage around the embossment and the (2) simulation of the plastic failure of the steel embossment.

In the concrete damage analysis the concrete is characterised by a homogeneous isotropic linear elastic material associated with the Willam-Warnke failure criterion while the steel sheeting is considered linear elastic, non damageable part of the models. Models are built in two levels: (i) global and (ii) local. The aim of the global models is to calibrate the concrete material model which is used in the further investigations. The aim of the local models is to determine the concrete failure around the embossment.

In the steel failure analysis the sheeting is characterized by linear elastic – hardening plastic material model. The concrete part in this case remains non damageable part of the models. A three-step-model is developed according to the pull-out test of individual enlarged embossment which includes the manufacturing and the loading of the embossment. The model is applied to simulate the pull-out test of real size embossment series. Detailed parametric study is executed by the model of the enlarged embossment to determine the effect of the different geometrical and physical parameters on the behaviour.

The numerical models are developed using ANSYS finite element software [23]. The Newton-Raphson method is used in the nonlinear analysis. The convergence of the solution is checked on the basis of the Euclidean norm of unbalanced force vector by applying a tolerance factor of 0.1%. In the geometric nonlinear analysis large displacements and strains are considered. Automatic time stepping is used which cut a time step size in half whenever equilibrium iterations fail to converge and automatically restart from the last converged sub step. If the halved time step again fails to converge, bisection will again cut the time step size and restart, continuing the process until convergence is achieved or until the minimum time step size is reached.

5 SIMULATION OF THE CONCRETE TYPE BEHAVIOUR

5.1 Introduction

5.1.1 Background of the modelling

The horizontal shear failure on a four-point-bended composite slab composes of three stages:

1. the load reaches the value which causes cracks in the tension zone of the concrete slab then the interlock between the steel deck and the concrete slab depends on the mechanical bond;
2. increasing the load sliding occurs after the critical level of loading is reached, and increasing further the load;
3. the failure of the slab comes off with the initiation of a dominant crack under the concentrated load followed by a relative displacement at the end of the slab between the steel and the concrete decks (end slip).

Since the concrete damage plays a role not only in the local failure of embossment but also in the global failure of the slab it is essential to be able to simulate the adequate behaviour of it.

Concrete is a non homogenous building material mainly consist of aggregates embedded in a mortar matrix. The nonlinear behaviour of the concrete is related to the initiation and propagation of micro and macro cracking/crushing in the structure. The concrete material can be modelled on different levels (macro-, meso-, multi-scale) in the numerical simulations. Most commonly the macro-scale is used where the concrete is treated as homogeneous continuum. This approximation is correct since in the size of the engineering structures is much higher then the representative volume element (RVE) of the material. The chosen finite element program offers a homogeneous continuum approach for concrete modelling which is applied on the analyzed models.

5.1.2 Concrete model details
The SOLID65 element of ANSYS is chosen to model the concrete. The element is defined by eight nodes and isotropic material properties and it is usually applied in three dimensional modelling of brittle solids, especially in concrete application and modelling of reinforced composites. To assign the concrete material properties the uniaxial tensile ($f_t$) and compressive strength ($f_c$), and additionally the shear transfer coefficients for open and closed cracks are the required input data. The material is initially considered isotropic. The cracking is modelled as a smeared band and it brings a modification in the stress-strain relationship by introducing a plane of weakness in a direction normal to the crack face [23]. Typical shear transfer coefficients range from 0 to 1, with 0 representing a smooth crack (complete loss of shear transfer) and 1 representing a rough crack (no loss of shear transfer). When the brittle failure of the material under tension/compression can be neglected the cracking/crushing capability of the material can be disabled by defining $f_t/f_c$ by the value -1, respectively (this option of the material model improves the convergence).

The failure surface for concrete is identified by the Willam-Warnke criterion [24]. The mathematical model of this failure surface is smooth and convex; it gives close fit of experimental data in the operation range and it is defined by a small number of parameters

---

1 RVE is the smallest volume possible whereof the physical state (stress, strain) is yet representative to the material.
which are determined from standard test data. When applying concrete material properties, a total of five input strength parameters ($f_t$, $f_c$, $f_{cb}$, $f_1$ and $f_2$ whereof the first two parameters are detailed above and the last three parameters are the biaxial compressive strength, the compressive strength for a state of biaxial and triaxial compression superimposed on hydrostatic stress state, respectively) are needed to define a 3D failure surface. The criterion, however, can be specified with a minimum of two constants, $f_t$ and $f_c$, the other three constants are calculated as: $f_{cb} = 1.2 \cdot f_c$, $f_1 = 1.45 \cdot f_c$, $f_2 = 1.725 \cdot f_c$.

5.1.3 Program of development and application

The concrete material model is calibrated and verified by two experiments resulting in an adequate model for further investigation. The previously obtained concrete material model is extended to study the effect of concrete damage first around a fictive embossment with simplified rectangular geometry, considering the steel part of the connection linear elastic. A parametric study is accomplished by the fictive model where the effect of different geometrical and physical parameters and the phenomenon are determined. The analysis is extended to study an embossment with refined geometry and the relationship is determined between the simplified/refined geometries.

5.2 Reinforced concrete beam model

5.2.1 Test #1: reference beam

To test the material model such structure is needed which fulfils the following requirements: well-known behaviour for the simple comparison of the results, concrete and steel components to analyze the concrete behaviour and the steel-concrete interaction. Regarding the conditions a simply supported reinforced concrete beam is chosen for the analysis. Accordingly, an experiment on an ordinary reinforced concrete beam with geometry of 155x240x2800 mm [25] is found as verification background for the concrete material model (Figure 46).

![cross-section of the experimental beam](image1.png)

(a) cross-section of the experimental beam

![mesh of the specimen](image2.png)

(b) mesh of the specimen*

* selected concrete elements removed to illustrate internal reinforcement

Figure 46. Details of the reference beam experiment and model [25]
Different models are worked out ($1/a - 1/c$) for testing the change of those parameters of the concrete material model, which are defined in Table 12. The compressive and tensile strength of the beam model are set by experimental values (69 N/mm$^2$ and 5.1 N/mm$^2$). The setup and meshing of the model is shown in Figure 47/a.

The reinforcement can be discrete or smeared. In the first case, it is defined apart, as 3D tension/compression spar element (LINK8 element). In the second case, the reinforcement is defined as modified material property (Figure 47/b and Figure 47/c). The SOLID65 element has up to three rebar materials which are capable of tension and compression, but not shear. Rebar specifications, which are input as real constants, include the volume ratio of the reinforcement and its orientation angles.

![Figure 47. (a) Setup of the numerical model, (b) discrete and (c) smeared reinforcement](image)

Table 12. Characteristics of concrete models of the reference beam

<table>
<thead>
<tr>
<th>Input data</th>
<th>1/a</th>
<th>1/b</th>
<th>1/c</th>
<th>1/d</th>
<th>1/e</th>
<th>1/f</th>
</tr>
</thead>
<tbody>
<tr>
<td>Concrete compressive strength [N/mm$^2$]</td>
<td>-1*</td>
<td>69</td>
<td>69</td>
<td>69</td>
<td>-1*</td>
<td>-1*</td>
</tr>
<tr>
<td>Concrete tensile strength [N/mm$^2$]</td>
<td>5.1</td>
<td>5.1</td>
<td>5.1</td>
<td>5.1</td>
<td>5.1</td>
<td>5.1</td>
</tr>
<tr>
<td>Shear transfer coefficient for open crack</td>
<td>1</td>
<td>1</td>
<td>0.1</td>
<td>1</td>
<td>1</td>
<td>0.15</td>
</tr>
<tr>
<td>Shear transfer coefficient for closed crack</td>
<td>1</td>
<td>1</td>
<td>0.9</td>
<td>1</td>
<td>1</td>
<td>0.3</td>
</tr>
<tr>
<td>Reinforcement (d = discrete, s = smeared)</td>
<td>d</td>
<td>d</td>
<td>s</td>
<td>d</td>
<td>d</td>
<td>d</td>
</tr>
<tr>
<td>Mesh size</td>
<td>30 mm uniform coarse</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Load increment size</td>
<td>1 mm</td>
<td>maximum 0.3 mm, minimum 0.003 mm</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Geometry of the beam</td>
<td>full</td>
<td>full</td>
<td>half</td>
<td>quarter</td>
<td>quarter</td>
<td>quarter</td>
</tr>
</tbody>
</table>

The load, according to the experimental data is applied in a four-point-bending arrangement by a displacement control. The maximum value of displacement is chosen by the maximum midspan deflection of the experimental beam ($e_{max} = 30$ mm). The load is applied on the 1/a
model in 1 mm steps, but convergence problem occurred when tensile crack appeared. Smaller loadsteps are chosen for models 1/b and 1/c: the maximum and minimum loadstep is $e_{\text{max}}/100$ and $e_{\text{max}}/10\ 000$, respectively, during the nonlinear analysis. The starting value of the loadstep is set to $e_{\text{max}}/1\ 000$. The calculation starts with this value and it is increased and reduced automatically between the maximum and minimum values, if needed by the convergence. Bisection is enabled, if the minimum loadstep value does not reach convergence. The runtime of the models is found significantly large. By the evaluation of the model results the following observations are derived:

- using smeared reinforcement instead of discrete have an influence only on the runtime of the model but not on the behaviour. The runtime reduces since there are not additional elements to represent the reinforcement; the programming of the smeared reinforcement, however, is found complicated;
- the value of the shear transfer coefficients should be adjusted to reach convergence after cracks appear.

Having these observations two new models are built (1/d and 1/e in Table 12). The quarter of the single span beam is modelled to reduce computation time and the load is applied in small loadsteps to keep the velocity of crack propagation low. The longitudinal reinforcement cannot transfer shear, so the cracked concrete cross-section is taken into account in the shear transfer (open and closed cracks, too). The shear transfer coefficients can be varied between 0-1. Those values express the shear transfer capacity of the cracked concrete cross-section compared to the undamaged concrete cross-section. Its value can be calibrated by experimental results, if the numerical stability exists. Recently researchers dealing with numerical modelling of reinforced concrete beam structures used a wide range of values for the shear transfer coefficients. The common value of this magnitude for open crack is in the range of 0.05-0.5 and for closed crack in the range of 0.6-0.9 [26]-[28]. However, other authors proposed e.g. 0.12 or 0.22 [29]-[31] for closed cracks. As a conclusion of the extensive literature review, it is found that no exact suggestion exists for setting these parameters. In this case, the maximum value is considered due to the reason of numerical stability.

The quarter beam model composed of 2 240 solid elements and 3 135 nodes; the total number of the degrees of freedom is 9 405. The runtime is 4.5 hours on a computer characterized by 3.6 GHz Pentium 4 dual-processor, 2GB RAM. The used maximum and minimum loadsteps are $e_{\text{max}}/104$ and $e_{\text{max}}/30\ 000$ (due to bisection), respectively. The average of the loadstep sizes is $e_{\text{max}}/4090$ under the total of 1517 steps.
The results of 1/d and 1/e models are compared to experimental results in Figure 48 (model 1/b and 1/c overlaps with 1/d and not presented for this reason). The behaviour of the beam and the numerical models are linear till the first crack appears (at the load level of 16 kN). When crack appears on the model a „disturbed“ phase (Figure 48, enlarged part) can be seen on the curve, that is followed by a quasi-linear phase till the yielding of the reinforcement (at the load level of 62 kN). After yielding of the reinforcement the load carrying capacity does not increase. If the crushing capability of concrete material is not considered it does not decrease neither. The model 1/e shows the case when the descending phase can be observed after crushing. Note that in the published paper the descending branch is not presented.

To analyze the effect of the shear transfer coefficients the 1/f model is built on the basis of 1/e model. The computational time is found considerably high on the beam model 1/e, so a literature review is completed to find an appropriate mesh size to reduce computation time but keep the accuracy of the results. Several suggestions exist in the literature from 25 mm fine uniform mesh [28] to coarser meshes: 50, 75 mm [27] and 80 mm in the longitudinal direction [25]. Since no exact value is advised the mesh is performed so that one volume is meshed by one element, so the mesh size is defined by the position of the reinforcement and the distribution of the stirrups. The coarse mesh gives accurate results: it does not affect the behaviour but the runtime becomes less, so this mesh size is applied further on the 1/f model.

![Figure 48. Numerical results of RC beam – test #1: (a) force-deflection curve (b) start of nonlinearity (magnified)](image)

Changing the value of the shear transfer coefficient it is observed that it has only a slight effect on the behaviour of the model, however, it has a certain minimum value where under the model does not converge. In the case of the 1/f model it is 0.15 and 0.3 for open and closed cracks, respectively (Table 12). It is also observed that the loadstep size could be increased. The model achieved the maximum displacement load (30 mm) under 23 loadsteps,
the average of the step sizes is \( e_{\text{max}}/21.78 \). With the increased load steps the load-displacement curve becomes smoother and the slope of the final branch (after the yielding of the steel bars) slightly decreases (Figure 48). Note that the model with the finer mesh (1/d) does not converge with the same large loadsteps. The maximum and minimum loadsteps wherewith convergence is reached is \( e_{\text{max}}/50 \) and \( e_{\text{max}}/100 \), respectively and the runtime reduces to 40 minutes.

As a conclusion of the numerical studies of the reference RC beam: the type of reinforcement (discrete or smeared) has no effect on the ultimate load or on the behaviour of the models. The numerical results of 1/d and 1/e both show good accordance with experimental results. Due to the small loadsteps the crack propagation becomes traceable. Despite the model with smeared reinforcement requires less computation time, its programming is more complicated. For the sake of simplicity and to keep the models’ flexibility for further geometrical or physical changes, the discretely reinforced model is used in the next steps of the research. By the 1/f model the change of the value of the shear transfer coefficient affects slightly the behaviour, in the analyzed case the effect is negligible. The mesh size has an effect on the maximal/minimal loadstep size: increasing the mesh size the loadstep size can be increased.

### 5.2.2 Test #2: short beam

A reinforced concrete beam model is built in the frame of the current research detailed in [20]. This specimen is selected as a test #2 for concrete modelling. The RC beam with geometry of 150x150x700 mm is built from grade C25/30 concrete. The test is executed at the age of 16 days of the beam, when the compressive strength and the Young’s modulus of the concrete are 37 N/mm\(^2\) and 37.123 GPa, respectively. The reinforcement at the tension zone composes of 2x6 mm diameter of grade B600A steel bars and the reinforcement at the compression zone composes of 2x3 mm of grade B240A steel bars. The shear strength is provided by stirrups (B240A, 3 mm diameter) distributed by 35 mm. The material properties are applied on the model by the material test results because it is found in [25] that the model is sensitive to the Young’s modulus of the concrete and the yield strength of the steel bars. The load is applied in four-point-bending arrangement. By the results of the experimental analysis, the first cracks and the yielding of the reinforcement are observed at the load levels of 20.5 kN and 41.03 kN, respectively. The experimental load carrying capacity of the beam is 45.5 kN.

Taking advantage of the symmetry the quarter of the single span beam is modelled. The same element type and material model for concrete and reinforcement is applied as in the model 1/e: the 3D solid element SOLID65 to model concrete, and LINK8 spar element to model the reinforcement with an elastic – plastic material model. Based on the previous modelling
experiences small loadsteps are applied in the nonlinear solution to make sure that the velocity of the crack propagation stays low to insure numerically stable analysis.

Two mesh sizes are analyzed by the model: (i) a uniform 20 mm mesh is applied first on the model and then (ii) the coarsest mesh possible, when one element represents one volume element in the model (the mesh size is defined by the position of the reinforcement and the distribution of the stirrups). The model and the meshes are shown in Figure 49.

![Figure 49. Numerical model and mesh of RC beam – test #2: (a) elements (b) uniform fine mesh (c) coarsest mesh (d) the mesh which gives the best fit for the experiment](image)

The model with the fine and uniform mesh shows increasing load carrying capacity after the yielding of the steel, as it is shown in Figure 50 (curve 2) and the model had almost an hour runtime. The model with coarse mesh on the other hand gives a good approximation on the load carrying capacity and the runtime significantly reduces to ~10 min. The final branch is still increasing, but its slope becomes much lower (Figure 50, curve 3). The best fit between the experimental and numerical load-displacement curves (Figure 50, curve 4) is found when the mesh in the cross-section is controlled by the edges of the volume elements of which the model composes. In the longitudinal direction every volume element is divided in two elements. This type of mesh sensitivity is not detected in the case of test #1. The fine and the
coarse mesh in this case gave almost exactly the same result. It leads to a conclusion, that the effect of shear is significant, because of the high height/span ratio of the short beam \((h_{\text{beam}} > l_{\text{beam}}/(10\div12))\). In this case the fine mesh gives an artificial stiffness to the model and leads to a spurious load-displacement relationship after the yielding of the reinforcement appears. The shear transfer through the cracked element is controlled by shear transfer coefficients (defined by the user) and it is assumed that the difficulty of handling both the shear from the loading and the shear transfer through the cracked elements results in the obtained phenomenon. Due to the increased element number in a denser mesh causes this effect more significant.

From the point of view of runtime, the coarse mesh turns the beam models time efficient and saves computational time and in the same time the accuracy of the results is kept.

To follow the assumptions of the test #1 the loadsteps are increased to accelerate the calculation. Large loadsteps are applied on the model with coarse mesh. Good agreement is found with the original load-displacement curve (Figure 50) by setting the maximum and minimum loadstep to \(e_{\text{max}}/10\) and \(e_{\text{max}}/230\), respectively \((e_{\text{max}}\) is the maximum displacement by experimental results). The magnitude of the shear transfer coefficients can be adjusted by experimental results, but in the case of the actual investigation its effect is negligible. It is found that care should be taken when setting the mesh density, when the effect of shear is important in the structure.

The final phase of the numerical load-displacement curve (after the steel yielding) remains ascendant. Note that in [25] the numerical curve shows the same character, as well. The common property of the beams is that the bond between the reinforcement and the concrete is considered perfect, which means that no bond slippage occurs. It is assumed that the consideration of the bond slip could adjust the final part of the load-displacement curve, but the results are accurate enough with this simplification of the model, too.

### 5.3 Summary

As a result of the accomplished parametric tests an adequate material model for concrete is calibrated by two experimental results. The applicability of concrete material model is verified by comparing it by test #1 and test #2. The requested loadsteps to insure convergence after the tensile crack appears is determined. The shear transfer coefficients are set by the experimental results. Based on these results, the concrete material model is found to be appropriate to follow the concrete behaviour, and it can be built in the local model of the rolled embossment.
5.4 Local model of fictive embossment

5.4.1 Model development

In the next step of the modelling process, the local model is developed to follow the behaviour of the concrete encased rolled embossments. The steel plate has a pattern of rolled embossments. According to the strategy of the research, only one embossment is modelled to describe the behaviour, which is later extended for all of the embossments. The failure process of one individual embossment is followed by a local model which considers only concrete damage and the nonlinearity of the steel is ignored.

The arrangement of the composite connection to be analysed is shown in Figure 51/a. To check the applicability of the local model, the geometry of the embossment is chosen as a rectangular dishing type, according to a published experimental research [6]. The concrete block’s geometry is 65x65x45 mm and the block’s width is determined by the arrangement of the embossments (Figure 51/b). It can be noted, however, that the model is fictive, because the geometry is simplified, and the rounding on the edges is not considered. Thus a quantitative comparison of the numerical and experimental results cannot be made, but the behaviour tendencies and the failure modes can be determined. The load is applied as uniformly distributed surface load on the back face of the concrete block (Figure 51/a). The purpose of the analysis is to push off the concrete block from the steel sheeting and observe the ultimate behaviour and failure.

![Figure 51](image)

* all the dimensions are noted in mm

Figure 51. (a) Local model’s geometry and arrangement (b) embossments on the plate

The SOLID65 element with crushing capability is applied to model concrete and SHELL63 elastic shell element for the steel sheeting. As interface element, a surface-to-surface contact pair is employed: the CONTA173 is used to represent contact and sliding between 3D target surfaces (TARGE170) and a deformable surface, defined by this element. These elements are applicable to 3D structural and coupled field contact analyses. It has the same geometrical characteristics as the solid or shell element face with which it is connected. Contact occurs
when the element surface penetrates one of the target segment elements on a specified target surface. Isotropic Coulomb friction is applied in the model which is specified by a single coefficient of friction. The local model of mechanical bond is numerically stable due to the contacts on the shell-solid interface.

Since the concrete failure is proved to be a mesh sensitive problem in the analysis of the reinforced concrete beam specimens, a mesh sensitivity analysis is carried out on the fictive model. It is found that the load carrying capacity shown by the model increases when increasing the mesh size (Figure 52/a). Reference value to compare the load carrying capacity results for one embossment does not exist. The only point of view about setting the mesh density is to keep the model time efficient and to be able to follow progressive crack propagation. It is found that by bigger mesh sizes extensive concrete damage occurs suddenly so the initiation and the propagation of it is not traceable. By those assumptions the maximum mesh size is set to 20 mm for further analyses.

The behaviour of rolled embossment in the numerical model can be analyzed by the force-displacement curve, as shown in Figure 52/b. The force is defined as the sum of the horizontal reaction forces and the displacement of the concrete block is accordingly measured in the same direction. The force-displacement relationship remains quasi linear despite the failure belongs to the appearance and propagation of the concrete cracking/crushing behind the embossment at the loaded side. The crack appears firstly around the edges of the embossment (points 1 to 2 on the curve) then it propagates on the interlocking concrete surface and also it spreads towards the loaded face until the maximum load is reached. Then the force falls back to a lower level.

![Figure 52. (a) Mesh sensitivity of the fictive local model and (b) force-displacement curve](image)

The degradation of the concrete on the loaded side of the embossment leads to failure. It is important to notice that the crack propagation’s direction is mesh-dependent: cracks are
spreading not just on the loaded face and in the direction of the loading but also cracks are running upwards and sideways apart. The ultimate load and displacement are 7.964 kN and 0.057 mm, respectively. The crack propagation and the plate deformation are presented in Figure 53, the numbering follows the notation of Figure 52/b.

![Crack patterns and plate deformation of the local model](image)

Figure 53. Crack patterns and plate deformation of the local model

The magnitude of the ultimate load cannot be quantitatively evaluated, since no experimental value nor calculation formula exists to compute the resistance of it. However, the failure mode that the model produces can be supported qualitatively by an experimental investigation of a pull-out test [32]. The test specimen and the specimen parts (profile steel rib and the surrounding concrete block) after the failure are shown in Figure 54. In this test the local crushing of the concrete near the embossments led to final slip and failure of the specimen, whereof the concrete damage can be seen in Figure 54/c.

![Test specimen](image)

Figure 54. Test specimen (a) profile deck rib (b) concrete block after the test (c) [32]
5.4.2 Parametric study on the fictive embossment

The design of composite slabs is currently based on performance test information (full or small scale tests) for a particular sheeting profile. A parametric investigation of the embossed local model is executed by an experimental analysis of small scale specimens [13]. The aim of the published experiment is to determine the effect of different shape, size and location of rolled embossments and different steel thicknesses. The aim of the parametric investigation is to generate the same type of geometric changes (depth, length and sheeting thickness), and analyze the behaviour. The experimental observations showed that the longitudinal shear resistance of the test specimens is significantly affected by the depth of the embossments and the length of the embossment is an influential factor, too. However, when the embossments had a certain length (~40-50 mm), the influence of further increasing is not significant. Additionally it is found by the experiments that the sheeting thickness has a significant effect on the stiffness of the tested specimens. The character of the tendencies (linear, 2nd order, etc.) are not derived from the tests, only qualitative evaluation is made.

A parametric investigation is completed by changing the depth, the length of the embossment and by changing the thickness of the steel sheeting, as it is seen in Figure 55/a. When one geometric parameter is changed on the model the other measures of the embossment are fixed according to the original geometry of the fictive embossment. However the amount of concrete cover (b₁ and b₂ in Figure 55) is kept constant around the embossment in every analyzed case. It means that the length (B) of the concrete block is enlarged, when increasing the length of the embossment and the depth (H) is reduced when decreasing the depth of the embossment. The results of sheeting thickness analysis give the expected results. Six different plate thicknesses are applied in the model. The character of the curves remains the same, but the initial stiffness increases by increasing the sheeting thickness, as shown in Figure 55/b.

![Figure 55. Embossment parameters (a), Effect of the sheeting thickness change (b)]
The results of depth and length analysis are summarized in Figure 56. It is observed that the models show to the same type of concrete failure and propagation but the load carrying capacity increases quasi-linearly by increasing the length/depth of the embossment.

![Figure 56](image_url)

Figure 56. Results of the different depth and length change on the ultimate load

### 5.5 Local model of circular embossment

#### 5.5.1 Model development

Having an experience on the basic behaviour of the fictive model, a refinement of the local model is completed. The embossment is chosen based on a published experiment [14].

![Figure 57](image_url)

Figure 57. Embossment details [14]: (a) cross-section; (b) side view; (c) detail; (d) finite element model

The details of the embossment are presented in Figure 57/a-c. The arrangement of the numerical model is the same as in the case of the fictive local model. The finite element mesh of the circular local model is shown in Figure 57/d. A crushing concrete material with shear transfer coefficient set to 0.3 (definition in Chapter 5.1.2) and a linear elastic-perfectly plastic steel material are applied in the model.

The results of the model are presented in Figure 58. The first crack appears at a load level of 0.215 kN at the upper back part of the embossment at the loaded side. Then the crack spreads around and towards the loaded face. At the point (2) a vertical jump is observed on the curve which belongs to the appearance of a cracked zone on the loaded concrete face. Continuous crack propagation leads then to failure. As it is noticed by the fictive local model the crack
propagation’s direction shows mesh-dependency by this refined model, as well. The ultimate force and displacement is 1.184 kN and 0.03 mm, respectively.

<table>
<thead>
<tr>
<th></th>
<th>[kN]</th>
<th>[mm]</th>
</tr>
</thead>
<tbody>
<tr>
<td>(1)</td>
<td>0.215</td>
<td>0.004</td>
</tr>
<tr>
<td>(2)</td>
<td>0.516</td>
<td>0.011</td>
</tr>
<tr>
<td>(3)</td>
<td>1.184</td>
<td>0.030</td>
</tr>
</tbody>
</table>

Figure 58. Force – displacement curve of the circular embossment, crack patterns and steel deformation.

5.5.2  Effect of friction coefficient on the behaviour of circular embossment

After the basic numerical model is built, a parametric study is accomplished on it, by changing the coefficient of friction between the steel sheeting and the concrete block. The coefficient of friction is a scattered physical magnitude [8] which is influenced by the quality of the contact surfaces.

Figure 59. Effect of coefficient of friction (a) on the stiffness, (b) on the behaviour
In case of concrete the quality of the concrete surface is affected by a large number of uncontrolled factors like the characteristics of the constituents or the manufacturing of the concrete, etc. [8]. Different values of dry friction coefficients can be chosen from 0.2 to 0.6 depending on the parameters mentioned above. Since the deterministic input parameter of the contact finite element is the coefficient of friction, it is important to determine its effect on the model behaviour. Four coefficients of friction are chosen for analysis: 0.3-0.4-0.5-0.6. The curves (Figure 59/b) show that the stiffness of the local model is increasing linearly when the friction of coefficient increases (Figure 59/a). The stiffness of every curve is taken as the quotient of the maximum load and relevant displacement. Besides, the character of the curves does not change, as it is shown by the load-displacement relationships in Figure 59/b.

5.5.3 Refined model vs. simplified model

As it is shown in the mesh sensitivity analysis of the fictive embossment, the mesh size has a significant influence on the load carrying capacity of an embossment. The mesh of the refined model, however, is not flexible for size changing since the geometry defines a certain mesh density. The question arises if it is possible to apply a rough geometry instead of the fine one (Figure 60/a) which has the basic features of the refined model but omits the rounding of the edges (Figure 60/b). To find the relationship between the fine and the simplified geometry the simplified model is built with similar mesh density to be able to compare the results.

The force-displacement diagram of the refined and simplified model together is shown in Figure 61. The ultimate load on the simplified model is 1.15 kN which gives good agreement with the refined models’ ultimate load (1.18 kN). Difference can be seen, however, in the ductility and the stiffness of the models.

The crack patterns and the plate deformation for the comparison are presented in Figure 62. Comparing the crack pattern in the final stage the refined and the simplified model show good accordance, as well.
When the stresses of the steel plate are analyzed it is found that the simplified model shows lower stresses than the refined model but the stress distribution is similar, as shown in Figure 63. The maximum stress value is 302 and 421 N/mm$^2$ on the simplified and on the refined model, respectively, which may mean yielding by lower steel grade (e.g. S235).

When the stresses of the steel plate are analyzed it is found that the simplified model shows lower stresses than the refined model but the stress distribution is similar, as shown in Figure 63. The maximum stress value is 302 and 421 N/mm$^2$ on the simplified and on the refined model, respectively, which may mean yielding by lower steel grade (e.g. S235).
It suggests that interaction can occur between the concrete and steel failures. This effect however, is not followed by the models, since the steel part remains linear elastic along the analysis. The importance of the steel failure is checked by the refined model by applying SHELL181 elastic-plastic shell element and linear elastic – perfectly plastic material model. It is found that the effect of the steel plates’ nonlinearity has negligible influence on the ultimate load and on the behaviour of the model, as it is shown by the load-displacement curves in Figure 61, since the failure is governed by concrete damage.

5.6 Summary

In this chapter, the concrete material model defined by the Willam-Warnke failure criterion is studied. The element type SOLID65 is associated with the material model whereof the cracking and crushing features are used. The input isotropic material parameters are the Young’s modulus and the Poisson’s ratio. The failure surface is specified with two material parameters: the uniaxial tensile and compressive strengths. The shear transfer through the cracked/crushed concrete elements is given by additional material properties by the shear transfer coefficients.

Models are developed for the simulation of concrete encased rolled embossments and the observed numerical difficulties due to the material nonlinearity and the local and global models’ behaviour are solved. As the result of the numerical modelling completed up to present an adequate concrete and reinforced concrete model is obtained, calibrating it by two beam experiments. A numerical local model is worked out for fictive rolled embossments where the basic behaviour modes (crack initiation and propagation) are determined. A parametric study is accomplished as well, according to published experiments [13], [14] and the tendencies are well followed by the local models. Refined embossment geometry is worked out and the relationship between the simplified and the refined geometry is determined. Having the observations on the models the following conclusions are derived.

Reinforced concrete beam models:

- the small loadsteps in every case insure numerically stable analysis. In the current research the applied minimum loadstep size is \( \text{emax}/10000 \). The loadstep size, however, can be increased. Using larger loadstep values result in smoother load-displacement curve and shorter runtime then in case of using small loadsteps;

- the shear transfer coefficients of the concrete model have a slight effect on the concrete models’ behaviour: the coefficients are to be used for detailed model calibration, their magnitude can be adjusted to experimental results;
- the FE model of the short beam is mesh sensitive, the mesh size influences the behaviour of the model. When the shear becomes governing in the behaviour a denser mesh leads to spurious stiffness increment;
- in general a coarse FE mesh (in the current research means one volume is meshed by one element) gives good results and produces a time efficient analysis.

Local models – general:
- the failure mode of the local models was primary conducted by concrete failure and the effect of steel yielding was negligible;
- the failure mode can be justified by experiments of pull-out test results [32];
- the local models are found mesh sensitive, the mesh size influences the load carrying capacity;
- with the denser mesh the ultimate load decreases (note, that the tendency is not strict);
- the geometrical shape of the mesh influences the direction of the crack propagation;
- the qualitative tendencies from changing physical and geometrical parameters on the model show good agreement with experimental results of pull-out tests.

Local models – refined vs. simplified models:
- a simplified local model (with similar mesh density) shows the same failure mode, ultimate load and crack pattern then the refined model;
- the simplified model shows higher stiffness, lower ductility and lower stresses in the steel sheeting;
- the simplified model is accurate enough if the target of the calculation is the ultimate load, and if the concrete behaviour governs the failure.
6 SIMULATION OF THE STEEL TYPE BEHAVIOUR: THREE-STEP-MODEL

6.1 Principles of the three-step-model

6.1.1 Modelling strategy

The aim of the model is to analyze the different phases of manufacturing and loading of the steel embossment. Two experiments are embedded in one model: (i) the extruding and (ii) the pull-out test of the enlarged embossment. In the pull-out experiments the steel plate was relatively weaker comparing to the concrete which means that the steel behaviour dominates in the global failure. Previously the local model composed of a section of steel sheeting with one embossment on its surface (embossed plate), and the surrounding concrete block. The same idea is adapted to the three-step-model with the following characteristics:

- the base of the model is a flat thin steel plate and the concrete block with the indentation of the embossment,
- the concrete block is assumed to be linear elastic and non-damageable part of the model,
- the model includes the extruding related model parts: the bearing ball and a section of the the bottom forming plate,
- the manufacturing process of the embossment is made under two loadsteps: extruding of the embossment (1\textsuperscript{st} step) and removing the extruding related model parts (2\textsuperscript{nd} step),
- the model includes a pull-out test related part: the spacer plate which insures support and symmetry condition for the embossment for the loading procedure,
- the pull-out test is performed in the 3\textsuperscript{rd} step.

The concept of the model and the details of the loadsteps are shown in Figure 64 and Figure 65, respectively. In the simulation certain model parts are multi-functional: the spacer plate and the section of the forming plate together model the bottom forming plate, the concrete part models the upper forming plate of the extruding experiment in the first two loadsteps. In the third loadstep the concrete part and the spacer plate act according to their role in the pull-out test.

![Figure 64. Details of the model](image)

![Figure 65. Details of the loadsteps](image)

The boundary conditions of the cold-forming and the afterwards performed pull-out test, however, are basically different. In the extruding process the flat steel plate is free to move between the forming plates. In the pull-out test the spacer plate and the embossed plate are
connected and move together. Those conditions have to be handled in one model in such a way that the restrictions of one loadstep do not bother the successful execution of another step. Some of these boundary conditions are defined by contact surfaces in the model, so the definition of their type is found essential (whether the contact pairs’ behaviour is frictional, bonded or frictionless). Since the element properties are not changeable during the solution of the model between the loadsteps, they need to be defined at the beginning of the simulation process to be appropriate for all the loadsteps.

The simulation of the extruding process is verified first by experimental results. After the model characteristics (support conditions, contact features) of the manufacturing are determined the complex three-step-model is developed.

### 6.1.2 Features of the finite element model

The numerical models are developed using the SOLID186, 20-node structural solid element having three degrees of freedom per node. The element supports plasticity, large deflection, and large strain capabilities. As interface element, a surface-to-surface contact pair is employed: the CONTA174 is used to represent contact and sliding between 3D target surfaces (TARGE170) and a deformable surface, defined by this element. These elements are applicable to 3D structural and coupled field contact analyses. It has the same geometric characteristics as the solid or shell element face with which it is connected. Contact occurs when the element surface penetrates one of the target segment elements on a specified surface. Isotropic Coulomb friction is applied in the model which is specified by a single coefficient of friction.

A numerical model is built according to the test setup (Chapter 3.1.1) in ANSYS Workbench for the simulation, as shown in Figure 66. The quarter of the specimen is modelled with applying the adequate symmetry conditions. The forming plates are vertically supported and the edge of the steel plate is supported against horizontal displacement. The mesh is formed such way that the steel plate composes of two element layers through the thickness.

![Figure 66. Numerical model](image)
The analysis is completed in two steps: the embossment is extruded first under displacement control until the height of 10 mm is reached. The unloading of the embossed plate is performed afterwards. A multi-linear (linear elastic – hardening plastic) material model is applied on the steel sheeting on the basis of material tests in Chapter 2.1.2. The extruding test is executed on a 1.5 mm thick plate; the material model is derived accordingly. The characteristics of the material model are determined by four points in Figure 67: (1) yield stress, (2) starting of hardening, (3) ultimate stress, (4) ultimate strain.

![Material model for extruding](image)

**Figure 67. Material model for extruding**

### 6.2 Simulation of the manufacturing process

#### 6.2.1 Load – displacement relationship

Figure 68 shows the load-displacement curves from the experiment and the simulation. Both curves show the extruding and the unloading of the plate. The results of the model show good agreement with the experimental curve. The ultimate displacement and load by the model is 10.2 mm and 31.91 kN, respectively (Figure 68). The deformed shape of the plate and the results of the model are shown in Figure 69.

![Load-displacement relationship](image)

**Figure 68. Load-displacement relationship**

It can be seen on the pictures that the deformed shape agrees well (Figure 69), only the sharpness of the edge around the embossment on the top surface is not exactly captured. This
local phenomenon, however, does not affect the global behaviour of extruding, as it is seen from the load-displacement relationships (Figure 68).

![Deformed shape](image)

Figure 69. Deformed shape (a) experiment (b) (c) model

### 6.2.2 Strains

The strain results of the model and the experiment are shown in Figure 70. Good agreement can be seen between the measured and the calculated results. The strain measurement results on gauge \( a \) show good agreement, only a small rate of deviation is observed at a strain level of \( \sim 2100 \, \mu \text{m/m} \) between the load-strain curves (Figure 71).

This difference is assumed to be the reason of the alteration between the experimental and the applied (simplified) material model (Figure 67). The yield strain by the experiments is 2080 \( \mu \text{m/m} \) which is followed by yielding plateau type behaviour, then by a hardening plastic section before the final yielding plateau. In the model’s simplified multi-linear material model the point of yielding is followed by two hardening plastic curve phases, whereof the slope of the first one is close to zero. It is possible then that the material model has a visible effect on the results at this model level.

![Strain measurement and numerical analysis](image)

Figure 70. Strain measurement and numerical analysis

![Strain at gauge a](image)

Figure 71. Strains at gauge \( a \)

Since gauge \( b \) failed at the very beginning of the loading, only the initial phase of the behaviour can be compared with experimental data and it gives good agreement (Figure 72).

In case of gauge \( c \) good agreement is found in the character of the experimental and the numerical curves. However, the numerical and experimental curves are farther from each other then at the two other positions (Figure 73).
The possible explanation of this phenomenon is that the strains from the numerical model are measured on one node, but in the reality the gauge collects data from a certain area. In the case of this position, the deformation under the grid of the gauge is not symmetrical to axis perpendicular to the measurement direction. It means that the edges of the gauge elongate diversely and may provide less accurate measurement data to compare with strains which occur at one node at the model. Despite the observed difference the range and the character of the simulated data are considered as acceptable agreement.

6.2.3 Summary

According to the experiment of 3D cold-forming a numerical model is worked out to follow the load-displacement relationship as well as the strains of the manufacturing process. The numerical model is developed in ANSYS finite element program under the Workbench platform. The model is built up from higher ordered solid elements with experimentally adjusted material models. The finite element model shows good agreement in the load-displacement and in the strains with the experimental data. The model is found to be appropriate to study the 3D cold-forming processes. The connections and support conditions of the model is analyzed and further applied in the three-step-model.
6.3 Simulation of the pull-out test: enlarged individual embossment

6.3.1 Computational strategy

The model is developed based on the principles detailed in Chapter 6.1. Since the cold-forming phenomenon is coupled with the pull-out test, essential problem is the setting of the connection between the multi-functional model parts. Three kinds of contact are considered: frictional, frictionless and bonded. In case of frictional contact the connected surfaces can slide on each other, the characteristic is defined by the coefficient of friction. Frictionless contact represents contact by touching of faces where there is no relative slip. Bonded contact represents rigid bonding (weld, glue) of faces. As for the extruding process the contact should be frictional on the concrete – steel plate – spacer plate interface to insure free deformation for the sheeting between the forming plates. As for the pull-out test the frictional contact is appropriate between the concrete and sheeting. Bonded contact is suggested, however, between the spacer plate and the sheeting for two reasons: (i) those parts are connected on their edges in the experiment, and (ii) the use of bonded contact saves computation time because the status of this connection type does not need to be updated during the calculation.

Analysis is needed to determine if the used bonded contact between the spacer plate and the sheeting has an influence on the extruding step of the simulation. It can be seen in Figure 74 that the stresses due to the extruding process spread beyond the edge of the spacer plate. It means that the bonded approach in this case would create a restriction for the extruding process. Frictional contact is used henceforth between the relevant model parts but the loading of them is performed together.

![Opening of the spacer plate in the pull-out test](image)

Figure 74. Von-Mises stress distribution on the embossed plate after extruding

6.3.2 Results of the numerical simulation

Two models are built for the verification: (i) an only pull-out solid model and (ii) a three-step-model to compare the behaviour of the models with and without extruding to the relevant experimental results. Figure 75 shows the load displacement results of the models and the specimen 1.3 with 1.5 m plate thickness (Chapter 2.2.1); Figure 76 shows the experimental and numerically simulated deformations. Both models show good agreement with the
experimental results in the range of ultimate behaviour. The runtime of the model is found high: depending on the mesh density and substep size ~1.5-2.5 hours. A simpler, only pull-out test model from higher order solid elements requires ~1 hour runtime.

Figure 75. Load-displacement relationship

Comparing the models’ load-displacement curves the initial linear phase on the three-step-model is softer than on the only pull-out model and it gives better fit with the first phase of the experimental curve until the first yielding appears in the plate at the bottom of the embossment on the loaded side due to the pull-out procedure (approx. 5 kN). As it is expected, the extruding does not affect the ultimate behaviour. The initial stiffness of the model curves and the experimental curve is marked on the load-displacement diagram (Figure 75). $S_1$, $S_2$ and $S_{exp}$ mark the stiffness of the only pull-out test model, the three-step-model and the experimental value, respectively. The initial stiffness for every curve is counted as the quotient of the load and the displacement that belong to the first yielding on the steel plate (the position of the measured first yielding coincide with the position of the first yielding on the models). The stiffness values are: $S_1 = 27.97$ kN/mm, $S_2 = 16.49$ kN/mm and $S_{exp} = 13.69$ kN/mm. Comparing the two numerical initial stiffnesses with the experimental one: the three-step-model gives ~20% higher value while the only pull-out model gives ~100% plus.

The stiffness of the numerical curves remains the same until the failure mechanism of the embossment starts, while the experimental curve shows decreasing stiffness after the first yielding on the steel plate. This difference, however, can be determined. Since the only difference between the experiment and the model is the concrete damage – which occurred during the failure of the specimen, but was excluded from the model – the rest of the alteration between the experimental and numerical load-displacement relationship can be identified by the previously assumed local internal concrete crushing around the embossment and by outer concrete cracking.
The importance to learn the effect of the extruding process is not only to find a better approximation for the experimental results, but also to filter out the effect of the concrete damage.

6.3.3 Summary

A three-step-model is developed for the simulation of the pull-out test. The connection between the contacting surfaces is found essential. It is studied in details and solved in the research. By the implementation of the extruding procedure in the pull-out test it is found that the initial stiffness of the model decreases and in the same time the same load carrying capacity is reached as by the model which excludes the effect of cold-forming. The initial stiffness gives a better accordance with the experimental results until the first yielding in the steel plate. The further difference between the experimental and numerical load-displacement curves can be explained by the local concrete damage, which occurred during the experiment. The model is proved to be appropriate for further application.
6.4 Simulation of the pull-out test: real size embossment series

6.4.1 Applied model

The numerical analysis of the small pull-out test splits in two directions: (i) analysis on the quality of the potential models (solid or shell), and (ii) the simulation of the small pull-out test by the selected model. Two types of models are built for the simulation: an 8-node-solid (SOLID 185) model and a 4-node-shell (SHELL181) model, as shown in Figure 77. First the solid model is built as a three-step-model which includes the forming of the embossments and the pull-out test. The quarter of the specimen is modeled using the symmetry of the specimen. It is observed during the experimental tests that in the ultimate behaviour the transversal deflection of the embossed thin plate becomes significant and the two thin plates touch on each other while pulled out from the concrete block. To follow this effect a symmetry plane is built in the model. After the first run it is found that the symmetry plane has negligible influence on the load bearing capacity, because its effect (load increment) comes out later on, when the ultimate load is already achieved. Linear elastic-hardening plastic (bilinear) material model is applied on the steel part of the model which is adjusted to tensile tests (see in Chapter 4.3, Table 8). The concrete part of the model is associated with a linear elastic material model. The shell model is built afterwards in a simpler way neglecting the symmetry plane and representing the concrete as a rigid target surface. The extruding of the embossments is not executed in this model. Three model parts are considered accordingly: (i) an embossed steel plate with linear elastic - hardening plastic material model, (ii) spacer plate with linear elastic material model and (iii) a rigid – non damageable – concrete surface.

In the first analysis, two models are built from each kind (solid and shell) for the comparison of the behaviour of the three-step-model and the only pull-out shell model according to two specimen types, 10.4 and 10.2s (see in Table 5).

In the second analysis, the specimen 2.10.2s (detailed in Chapter 4.3) is chosen for the model calibration for two reasons: (1) the results of the test are not influenced by the transversal pre-stressing and (2) there are no inactive or less effective embossments.
6.4.2 Comparison of shell and solid models

The most significant difference between the models is the runtime which is ~1.5 hours /10 minutes on the solid and shell models, respectively by the applied hardware. The extruding process and the multiple numbers of freedoms of the solid model comparing to the shell model make it time consuming. The advantage of the solid model despite its runtime is its accuracy comparing to the shell model. The load-displacement curves of the developed solid and shell models are shown in Figure 78. It can be seen that the shell model which uses the same material property settings as the solid model underestimates the load bearing capacity by ~10%. The behaviour of the model follows a quasi-bilinear like character while the solid model produces a load-displacement relationship whereon the transition between the initial phase and the ultimate phase shows gradually decreasing slope. The difference is identified by the missing extruding process in the shell model.

Additionally the convergence on the solid model is found to be better; the descending phase of the load-displacement relationship is longer traceable. The solid model is found more appropriate to follow the behaviour of the specimen.

Figure 79 shows the load-displacement results of the simulation and test of 2.10.2s specimen. The results are compared until the first 10 mm of displacement, which means that the embossment completely leaves its indentation in the concrete. Beyond 10 mm the load-displacement diagrams are not relevant since in the real structure there is no transversal constraints to insure further the composite action. Since the cyclic loading (see in Chapter 4.1) is not performed on the model its results are compared with the experimental curve part which belongs to the pull-out test after the last unloading. The model results show good agreement with the experiment and it is suitable for the modelling of the other specimens.

![Figure 78. Comparison of the solid and the shell models](image1)

![Figure 79. Calibration of the solid model](image2)
6.4.3 Application of solid model

A total of nine solid models are built according to the specimen types. The specimens and the material properties that are applied on the models are summarized in Table 13. The steel material properties are determined from material test results by plate thicknesses (Chapter 4.3, Table 8). The load bearing capacity of the models and the specimens are summarized in Table 14.

Table 13. Material properties for the numerical models

<table>
<thead>
<tr>
<th>Type</th>
<th>$E_s$ [N/mm$^2$]</th>
<th>$f_y$ [N/mm$^2$]</th>
<th>$f_u$ [N/mm$^2$]</th>
<th>$E_s$ [N/mm$^2$]</th>
</tr>
</thead>
<tbody>
<tr>
<td>12.4</td>
<td>32 553</td>
<td>378</td>
<td>454</td>
<td>219 177</td>
</tr>
<tr>
<td>12.2k</td>
<td>32 553</td>
<td>378</td>
<td>454</td>
<td>219 177</td>
</tr>
<tr>
<td>12.2s</td>
<td>32 553</td>
<td>378</td>
<td>454</td>
<td>219 177</td>
</tr>
<tr>
<td>10.4</td>
<td>32 553</td>
<td>404</td>
<td>487</td>
<td>217 124</td>
</tr>
<tr>
<td>10.2k</td>
<td>32 553</td>
<td>404</td>
<td>487</td>
<td>217 124</td>
</tr>
<tr>
<td>10.2s</td>
<td>32 553</td>
<td>404</td>
<td>487</td>
<td>217 124</td>
</tr>
<tr>
<td>07.4</td>
<td>32 553</td>
<td>407</td>
<td>477</td>
<td>216 247</td>
</tr>
<tr>
<td>07.2k</td>
<td>32 553</td>
<td>407</td>
<td>477</td>
<td>216 247</td>
</tr>
<tr>
<td>07.2s</td>
<td>32 553</td>
<td>407</td>
<td>477</td>
<td>216 247</td>
</tr>
</tbody>
</table>

Table 14. Experimental vs. numerical model results

<table>
<thead>
<tr>
<th>Thickness [mm]</th>
<th>4 type</th>
<th>2s type</th>
<th>2k type</th>
</tr>
</thead>
<tbody>
<tr>
<td>Test [kN]</td>
<td>Model [kN]</td>
<td>%</td>
<td>Model$^*$ [kN]</td>
</tr>
<tr>
<td>1.2</td>
<td>32.33</td>
<td>44.07</td>
<td>136</td>
</tr>
<tr>
<td>1.0</td>
<td>30.80</td>
<td>38.68</td>
<td>126</td>
</tr>
<tr>
<td>0.7</td>
<td>21.55</td>
<td>29.84</td>
<td>138</td>
</tr>
</tbody>
</table>

Model with 3 active embossment

During the evaluation it is observed that the models with 4 embossments overestimate the load bearing capacity of the specimens. It is found in the experiments that the first embossments are always less effective in the specimens due to concrete damage in front of them. In the models the concrete damage cannot be activated, which results in having 4 equally active connectors in it. As a solution a 3-embossment-model is built on the basis of the 4-embossment-model for every 4-embossment-specimen. The modified models show good agreement with the test, as it is shown in the Model$^*$ column of Table 14.

The highlighted test results in Table 14 are those which are not influenced by transversal compression by experimental observations (see in Chapter 4.4). The biggest differences are found in the cases of 07.2k and 07.2s specimens. Based on experimental observation the transversal compression from the fixing has an effect on the load bearing capacity and on the
behaviour. By the solid model an investigation is made so that the concrete part is pushed (under displacement control) against the embossed plate after the extruding process is done and the pull-out test is performed after this. It is found that by increasing the transversal compression the load bearing capacity of the specimen is increasing. The character of the curve changes as well and the reason of it is well illustrated by the models: when the transversal compression increases it pinches the embossed plates so that it blocks the embossments from moving to the direction of the force. Since the embossments are stuck due to the transversal compression the tensile behaviour of the flat sheet part starts to dominate and causes a stiffer initial phase and a peak at the ultimate load. This load-displacement character appears at all of the 07.2s, 3.10.2s and 2.12.2k specimens. Figure 80 shows the different characters for the load-displacement relationships under transversal compression. The aim of the diagram is to give a qualitative evaluation on the behaviour so the load/displacement values are not marked on it.

![Figure 80. Change of the character of the load-displacement relationship due to transversal loading](image)

In Figure 81 the model results show the same tendencies as the specimens (Figure 45 in Chapter 4.4): the load bearing capacity decreases with decreasing the plate thickness. The 4*-embossment-models (3 active embossment) show higher ultimate load then the 2-embossment-models and the ‘2s’ type models show higher ultimate load then the ‘2k’ type models. The ultimate load of one embossment is derived from the simulations for the comparison. Based on modelling observations the embossments develop individual and grouped failure as it is expected by the experimental results (see in Chapter 4.5). The individual failure is representative on the ‘2s’ type models and the grouped failure is representative on the ‘2k’ and ‘4’ type models. The individual failure of an embossment results in an average of 11% higher ultimate load then the ones from grouped failure. The embossments in the ‘4’ type models (with three active embossments) provide approximately the same ultimate load as the embossments in the ‘2k’ type models.
**Figure 81. Results of the three-step-model of the real size embossment series**

### 6.4.4 Summary

Numerical models are worked out for the simulation of the small pull-out test from 8-node-solid and 4-node-shell elements. The solid models take into account the forming of the embossments on the steel surface (three-step-model) while the shell models simulated only the pull-out tests. None of the models considered the damage of the concrete into account. The solid models are found more accurate comparing to the shell models to follow the experimental behaviour and further used to simulate the experiments. The shell model, however, more time efficient and gives good prediction for the ultimate load. The experiments and the simulations proved the sensitivity of the behaviour on the transversal compression. The transversal load increases the ultimate load and influences the behaviour. According to the experimental investigations the models of the 4-embossment-specimens include the 3 active embossments and the contribution of the less active 4\textsuperscript{th} embossment is neglected.

In general it can be stated that good agreement is found between the experimental and the numerical model results; the tendencies of changing the plate thickness, the number of embossments and the embossment pattern is well predicted by the model.
7 SIMULATION OF THE STEEL TYPE BEHAVIOUR: PARAMETRIC STUDY

7.1 Introduction

7.1.1 Model selection

The parametric study aims to analyse a wide range of geometrical and physical embossment parameters: the plate thickness, the coefficient of friction, the height and the size of the individual enlarged embossment in the model. The study results in a large number of simulations whereof the time efficiency of the applied model is essential. The three-step-model requires considerable runtime (see in Chapter 6.3.2). The aim of the study is to apply a lower order model which is time efficient and precise enough to provide reliable results comparing to experimental results. Comparison is made in Chapter 6.4.2 between the three-step-solid model and an only pull-out shell model where the time efficiency of the shell model is proved (by the runtime of ~10 min.) but the accuracy of the model needs to be further tested.

It is proved in Chapter 6.3.2 that the extruding process has an effect on the initial stiffness of the behaviour and does not influence the ultimate behaviour. For the sake of runtime this step is excluded in the parametric study. Simplification of the model can be made comparing to the solid models by replacing the concrete solid elements by a rigid target (shell) surface in the finite element model: the steel part is modelled with shell elements, the solid part is ignored, and a rigid shell surface with identical geometry like the embossment is considered instead. In the analysis the steel shell is pulled out from the rigid shell, considering the contact and friction between them. Since the extruding process is not simulated the spacer plate and the embossed plate is modelled together taking into consideration their thicknesses.

7.1.2 Finite element model

On the basis of the above idea a shell finite element model is developed, the details and the layout of the model are summarized in Table 15 and in Figure 82. The SHELL181, 4-node finite strain shell element is applied in the model, which is suitable for analyzing thin and moderately thick shell structures. A 5 mm maximum element size is applied in the finite element mesh. The effect of the mesh density is analyzed, as detailed in Chapter 7.1.3. As interface element, surface-to-surface contact pair with friction is employed: the CONTA173 is used to represent contact and sliding between 3D target surfaces (TARGE170) and a deformable surface, covered by this element. The interface elements have the same features as in the three-step-model (Chapter 6.1.2).

A quarter of the full specimen (Figure 14) is modelled, and it is composed of two surfaces: (i) the embossed shell surface with steel material properties and (ii) the finite element mesh of target surface, which represents the rigid concrete block. The target surface is rigid by
restricting all of its DOF’s. The contact finite element mesh is defined on the steel part of the interface. The embossed plate is composed of two parts: (i) the area of the embossment and (ii) the area of the spacer plate. The area of the embossment is left without restraint, while the transversal displacement is avoided along the spacer plate’s surface. The load is applied on the model by prescribed displacements at the end of the steel plate.

![Figure 82. The FE model](image)

**Table 15. Details of the FE model**

<table>
<thead>
<tr>
<th>Model part</th>
<th>Steel</th>
<th>Concrete</th>
</tr>
</thead>
<tbody>
<tr>
<td>Embossment area</td>
<td>SHELL181 +CONTA173</td>
<td>SHELL181 +CONTA173</td>
</tr>
<tr>
<td>Spacer plate</td>
<td>SHELL181 TARGE170</td>
<td>SHELL181 TARGE170</td>
</tr>
<tr>
<td>Interface area</td>
<td>Multi-linear</td>
<td>Multi-linear</td>
</tr>
<tr>
<td>Position</td>
<td>Under target</td>
<td>Under target</td>
</tr>
<tr>
<td>Overhang</td>
<td>Target</td>
<td></td>
</tr>
<tr>
<td>Mat. model</td>
<td>Multi-linear</td>
<td>Multi-linear</td>
</tr>
<tr>
<td>DOF</td>
<td>No constraint</td>
<td>Transversal fixed</td>
</tr>
<tr>
<td>All fixed</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

A multi-linear (linear elastic – hardening plastic) material model is applied for the steel plate. The material model is determined for both 1.5 mm and 2 mm thick plate; the characteristics can be found in Figure 83. The material of steel is considered as homogenous, isotropic and described with the multi-linear model, all over the steel surface.

![Figure 83. Material model for steel](image)

<table>
<thead>
<tr>
<th>Curve points</th>
<th>Stress [N/mm²]</th>
<th>Strain [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.5 mm plate</td>
<td></td>
<td></td>
</tr>
<tr>
<td>(1)</td>
<td>444</td>
<td>0.208</td>
</tr>
<tr>
<td>(2)</td>
<td>444</td>
<td>3.204</td>
</tr>
<tr>
<td>(3)</td>
<td>510</td>
<td>11.285</td>
</tr>
<tr>
<td>(4)</td>
<td>510</td>
<td>21.17</td>
</tr>
<tr>
<td>2.0 mm plate</td>
<td></td>
<td></td>
</tr>
<tr>
<td>(1)</td>
<td>459</td>
<td>0.230</td>
</tr>
<tr>
<td>(2)</td>
<td>459</td>
<td>2.880</td>
</tr>
<tr>
<td>(3)</td>
<td>534</td>
<td>8.790</td>
</tr>
<tr>
<td>(4)</td>
<td>534</td>
<td>18.83</td>
</tr>
</tbody>
</table>
7.1.3 Mesh sensitivity analysis

In the first step, the model is meshed with uniform finite elements with 5 mm edges. The obtained load – displacement curve can be seen in Figure 84. The curve shows waving character, what is caused by the sudden change of the contact. To check the effect of discretisation, a mesh sensitivity analysis is carried out with refining the mesh around the area of the embossment.

Four mesh sizes are studied by a maximum element edge length of 5 mm to 2 mm. The results in Figure 84 show that the global characters of the curves remain the same; the ultimate loads are practically the same, but the curves become smoother by decreasing the mesh size. Based on the results it is concluded that the behaviour of the interface elements is influenced by the mesh size, which locally disturbs the characteristics of the load – displacement curve. The 2 mm mesh size is used further in the model verification.

![Figure 84. Mesh sensitivity analysis](image)

7.2 Verification of the model

7.2.1 Numerical and experimental behaviour

Nonlinear analysis is completed on test model by applying kinematic load, with 20 mm maximum displacement. This displacement is identified as the start of the development of plastic failure in the tests. The results show good agreement with the experiments, as it can be seen in the load – displacement curves in Figure 85.

The initial quasi-linear phase is longer in the numerical result, and its slope is higher, compared to the experimental curve. The yielding plateau type behaviour is obtained thereafter. Two effects which play role in the experiment are not considered in the model: (a) the effect of the extruding process of the embossment is not taken into account in the material model and (b) the concrete is simplified in the model and its damages are not considered.
Based on experimental observation, the decreasing of the slope of the experimental curve overlaps with the appearance of the cracking on the concrete surface. Serious delamination between the concrete and the steel plate is not observed, since the crack opening is relatively small around the plate due to the stirrups, which hold the concrete block in a whole. Since the applied model does not contain concrete, this local phenomenon cannot be followed. The reason of the initial deviation between the experimental and the numerical curve is the combined effect of the internal concrete damages and the cold forming.

Good agreement can be seen, however, between the experimental and the numerical curves in the range of ultimate behaviour. In this phase the source of the nonlinearity is the steel yielding of the embossment whereof the applied model gives good prediction.

### 7.2.2 Characteristics of the load – displacement relationship

A typical load – displacement curve (Model: 1.5 mm curve from Figure 85) is presented in Figure 86, showing its characteristic points. The points indicate the relevant yield pattern, evaluated by von-Mises plastic strain distribution, and the embossment’s deformation, which are presented in Figure 87.

A quasi-linear first phase can be observed on the curve till first yielding appears on the loaded side of the embossment (1\textsuperscript{st} point). Increasing the load, the yielding zone spreads around the embossment from front to back. As the yielding zone reaches halfway around the embossment (2\textsuperscript{nd} point), it starts to expand from the bottom to the top of the embossment, on the loaded face. This phenomenon is followed by a significant decrease in the slope. The 3\textsuperscript{rd} point on the diagram belongs to the appearance of a new plastic zone, on the plane surface, in front of the embossment’s loaded side. The propagation of the yielding zones leads to the failure. The 4\textsuperscript{th}
point represents the ultimate load (28.88 kN at a displacement of 11.87 mm). The 5\textsuperscript{th} point shows the load belonging to the ultimate displacement (20 mm).

![Figure 86. Load-displacement relationship](image)

7.2.3 Strains

The evaluation of the strain results are made in the specified nodes. Typical results of the strain measurement can be seen in Figure 88. The curves show the relationship of the load and the strain in the center of the embossment comparing it to the experimental results (gauge #3). The global characteristics of the models’ load-strain curve agree well with the measurement. Difference can be observed, however, at the initial phase in the slopes.

The difference can explained both by the model and by the measurement. With the extruding process a significant residual strain is introduced in the steel plates around the embossment.
which is not considered in the model. Beside this, most of the strain gauges are put on curved surface, and the precision of the measured values can be influenced by this fact.

![Graph showing strains in the middle of the embossment](image)

**Figure 88.** Strains in the middle of the embossment

By comparing the strain results of the measurement and the numerical model, it is found that the arrangement of the strain gauges is appropriate to provide results for the verification, and the strains follow the behaviour of the embossment’s failure.

### 7.2.4 Evaluation of the numerical model

From the previous results it can be seen that the model can follow accurately the ultimate behaviour if it is governed by the plastic failure of the steel embossment. Despite the differences in the numerical and experimental behaviour in the first phase of the load – displacement relationship the ultimate load and deformations are well predicted. The computational efficiency of the model is good; the running time of a typical model with 5 mm/2 mm of maximum element edges having 10 506/27 066 DOF’s is 7/31 minutes (Intel P4 3 GHz, 2 GB RAM).

On this basis the model is found to be accurate and efficient enough to study the ultimate behaviour of this type of embossment.

### 7.3 Effect of the embossment’s parameters on the ultimate behaviour

An extensive parametric investigation is carried out by the developed model to study the ultimate behaviour of an embossment by changing the geometric and physical characteristics: the plate thickness, the coefficient of friction and the height and size of the embossment in the model.

#### 7.3.1 Coefficient of friction

The analysis is carried out on three plate thicknesses, with increasing the value of the coefficient of friction from 0 – 0.6 by steps of 0.1. These values are applied on the 1.5 mm thick plate to derive the tendency, and three of them are chosen for the other two plate
thicknesses to support the results. Three plate thicknesses are analyzed: 1.5 and 2 mm, according to the experiment and 3.5 mm which is considered as the maximum of the thin-walled plate thicknesses.

It is found that the load carrying capacity increases approximately threefold with increasing the coefficient of friction from the minimum to the maximum value, and the tendency is quasi-linear, as shown in Figure 89.

![Figure 89. Effect of the coefficient of friction on the load carrying capacity.](image)

### 7.3.2 Plate thickness

Based on previous experimental investigations [6] the sheeting thickness is an important influential factor of the load carrying capacity and also of the initial stiffness. Seven plate thicknesses are applied on the model from 0.5 to 3.5 mm, in a step of 0.5 mm.

![Figure 90. Effect of the plate thickness on the load carrying capacity.](image)

The load carrying capacity is more than eightfold higher on the 3.5 mm thick plate compared to the 0.5 mm thick plate. Figure 90 illustrates, that the relationship between the load carrying capacity and the sheeting thickness can be approximated with a 2nd order function what reflects the dominant bending type failure of the plate.
7.3.3 Embossment height

The effect of changing the height of the embossment is analyzed by taking the maximum height of the embossment as 10 mm (experimental value) and it is decreased by taking 2.5 mm sections to the 2.5 mm minimum value of the embossment, as shown in Figure 91. In the experiment, the ratio between the embossment diameter and the diameter of the spacer plate’s hole is 80 mm/37.4 mm = 2.14. It determines the area of the embossment which is left without restraints compared to the embossment’s diameter. The original ratio between the diameters is kept in case of every embossment height.

![Figure 91. Section levels](image)

Effect on the behaviour

The behaviour is determined for 1.5 mm plate thickness. The results are illustrated by the load – displacement curves in Figure 92, and by the plastic zones in Table 16. The evaluation is based on the characteristic curve points of 10 mm height embossment, as introduced in Chapter 7.2.2. The tendencies of the behaviour are explained as follows:

Load – displacement relationship

![Figure 92. Effect of the height of the embossment on the behaviour](image)

The load-displacement curves of the embossments of 5/7.5/10 mm heights are composed of four parts, namely (i) a linear phase, (ii) a nonlinear part with gradually decreasing slope, (iii) a yielding accompanied with large displacement and (iv) a final descending phase. The load-displacement curve of the embossment of 2.5 mm height on the other hand composes of three
phases. The above mentioned third curve part is missing, so the descending phase follows immediately the end of the nonlinear part. It means that the appearance of the additional plastic zone brings also the reach of the ultimate load, as shown in Figure 92.

**Yielding propagation**

Concerning the yielding propagation, the followings are observed: in every case the end of the linear phase on all of the load displacement curves belongs to the appearance of the first yielding at the bottom of the embossment on the loaded side (as in Figure 87/detail 1 and also in the 1st column of Table 16). The end of the nonlinear part belongs to the phenomenon, when the yielding zone reaches halfway around the embossment (as in Figure 87/detail 2).

When the appearance and the propagation of yielding is analysed on the models, it is observed that for the embossments of 5/7.5/10 mm heights an additional plastic zone appears on the plane surface in front of the embossment’s loaded side, as it is shown in the second column of Table 16. This additional yielding zone for the embossment of 2.5 mm height appears in the middle of the embossment.

<table>
<thead>
<tr>
<th>Height</th>
<th>End of the linear phase, yielding (1)</th>
<th>Additional plastic zone (3)</th>
<th>Ultimate load Yield pattern (4)</th>
<th>Ultimate deformation</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.5 mm</td>
<td><img src="image1.png" alt="Image" /></td>
<td><img src="image2.png" alt="Image" /></td>
<td><img src="image3.png" alt="Image" /></td>
<td><img src="image4.png" alt="Image" /></td>
</tr>
<tr>
<td>5 mm</td>
<td><img src="image5.png" alt="Image" /></td>
<td><img src="image6.png" alt="Image" /></td>
<td><img src="image7.png" alt="Image" /></td>
<td><img src="image8.png" alt="Image" /></td>
</tr>
<tr>
<td>7.5 mm</td>
<td><img src="image9.png" alt="Image" /></td>
<td><img src="image10.png" alt="Image" /></td>
<td><img src="image11.png" alt="Image" /></td>
<td><img src="image12.png" alt="Image" /></td>
</tr>
<tr>
<td>10 mm</td>
<td><img src="image13.png" alt="Image" /></td>
<td><img src="image14.png" alt="Image" /></td>
<td><img src="image15.png" alt="Image" /></td>
<td><img src="image16.png" alt="Image" /></td>
</tr>
</tbody>
</table>

For the embossment of 2.5 mm height the plastic zone propagates practically equally from the loaded side to the back. When increasing the height from 2.5 mm to 10 mm the separation between the plastic zone on the front of the embossment and the plastic zone around the embossment is getting more significant.

The load – displacement relationship in case of 2.5 mm height decreases after the end of the first nonlinear phase, while in the case of the other heights the load is increasing after this
point (curve sections between the markers (3) and (4) in Figure 92). The load increment is accompanied with large deformation of the embossment. The phenomena are well illustrated by the deformed shape at ultimate load level in the last column of Table 16. In case of 2.5 mm height, the embossment keeps almost its original shape till ultimate load, while the other embossment sustains significant deformations before reaching the ultimate load.

**Effect on the load carrying capacity**

The four embossment heights are analyzed by applying three plate thicknesses to determine the tendencies on the load carrying capacity. It is found that the load carrying capacity increases quasi-linearly with increasing the height of the embossment, as it is shown in Figure 93. The most dominant increment in load carrying capacity is observed in the case of 1.5 mm plate thickness.

![Figure 93. Effect of the height of the embossment on the load carrying capacity](image)

### 7.3.4 Size effect

**The purpose of the study**

After the effect of basic parameters is determined, the relationship between the experimental enlarged embossment and a real sized embossment with similar geometric ratios is analysed. The height of the embossment is reduced proportionally with its diameter (to keep the original embossment height/diameter ratio), from the experimental size to a real embossment size [14]. Four embossment sizes are taken from 10 mm to 2.5 mm height, in a step of 2.5 mm, as shown in Figure 94.

Comparing the results to the height analysis, same heights of embossment are analysed with 1.5 mm plate thickness and a value of coefficient of friction 0.5. Comparing to the height analysis, the surface of the embossment is always smaller and the embossing slope is always higher by the size effect analysis, as shown in Figure 95. The reference embossment
height/size is always the experimental embossment geometry \((h = 10 \text{ mm}, d = 37.4 \text{ mm}, \alpha = 56^\circ)\). The size effect analysis in this way involves the slope effect, too.

![Figure 94. Embossment size reduction](image)

![Figure 95. Embossing slope for (a) height and (b) size effect analysis](image)

**Size effect on the behaviour**

The effect of the embossment size (considering the embossing slope effect) can be analyzed by the load – displacement curves, shown in Figure 96. Comparing to the load – displacement relationships of the height analysis the character of the curves remains similar.

![Figure 96. Size effect on the behaviour](image)

The same evaluation method is chosen for the size effect analysis as in the case of height analysis. The end of the linear phase (1) which is identified with the appearance of the first yielding on the steel plate is followed by a nonlinear curve section which belongs to the spreading of the plastic zone in front of the embossment. The end of the nonlinear part (2) in case of 2.5 mm size is identical with the ultimate load, and in case of the other heights the load is increasing after this point while the displacements become large. In case of 5 mm size the end of the nonlinear phase (2) is identical with the appearance of an additional plastic
zone (3) on the top of the embossment. In case of 7.5 – 10 mm heights the additional plastic zone appears on the plane surface in front of the embossment (3). Point (4) represents the ultimate load which appears besides increasing displacements when increasing the size of the embossment.

The propagation of the plastic zones can be followed by the pictures of Table 17. In case of 2.5 mm and 5 mm size the plastic zone concentrates on the front side of the embossment. In case of size 7.5 mm (and also 10 mm) the plastic zone separates on two parts and it is propagating around the bottom of the embossment and on the front side. The deformation at ultimate load can be seen in the last row of Table 17. It is shown on the pictures that the smallest embossment fails after relatively small deformations while the other sizes fail after large deformations.

<table>
<thead>
<tr>
<th>Height</th>
<th>End of the linear phase, yielding (1)</th>
<th>Additional plastic zone (3)</th>
<th>Ultimate load Yield pattern (4)</th>
<th>Ultimate deformation</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.5 mm</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>5 mm</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>7.5 mm</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>10 mm</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
</tbody>
</table>

**Table 17.** Behaviour of the embossments for different size

**Effect on the load carrying capacity**

Analysing the four embossment sizes it is found that the load carrying capacity increases quasi-linearly with increasing the size of the embossment, as it is shown in Figure 97 in the same diagram with the height analysis results. The observed load carrying capacity is always higher for the same height in case of size effect results compared to the results of the height analysis. It is found that the highest difference in the load carrying capacity occurs where the difference between the embossing slopes is the biggest. Based on prior research by Ferrer et al. [8] the embossing slope is the most important parameter and the dependency of sliding resistance is significant.
By the experimental observations the failure of the embossment occurs by local bending on the embossment, which means plastic failure of the material. The height and size effect results show that the height and size reduction involves change in the behaviour although the ultimate load can be scaled quasi-linearly. Its reason may be the change in the nature of the failure: the plastic limit state transforms/interacts with plastic buckling by smaller embossment sizes. Extended studies and analysis of the phenomena is required in further research.

7.4 Summary
A shell finite element model is developed and verified to be used for a parametric study on the basis of experimental results of an individual embossed mechanical bond. The model is developed on the assumption that in the ultimate behaviour the steel part dominates (as in the experiments) ignoring the concrete damages, and replacing the concrete part with a fix and rigid surface. By the comparison of the experimental and numerical results it is found that:
- the results show good agreement in the range of ultimate behaviour,
- the deformations, stress distribution and load carrying capacity can be predicted with good accuracy.
A parametric study is accomplished by the developed model, to determine the behaviour by different embossment parameters, namely: the sheeting thickness, the coefficient of friction, the height and the size of the embossment. The size effect analysis is important since the experiment is executed on an enlarged embossment; hence a size effect analysis needed to determine the relationship between the experimental embossment size and the real size of a same shape. From the results the following achievements can be written:
- the load carrying capacity is influenced linearly by the change of the coefficient of friction and the height of the embossment,
- the load carrying capacity also depends linearly on the embossment size and a same type of difference in the behaviour is found like in the case of the height analysis,
- the dependency on the plate thickness is second order type,
- in the height analysis different ultimate behaviour occur by decreasing the height of the embossment.

On the basis of the height and size effect analysis it can be concluded, that scaling up the problem influences the behaviour and the failure type of the connection. The deformation capacity is larger with the bigger embossment sizes. By the evaluation of the models the real size embossment has not this kind of ductility. However, the load carrying capacity can be scaled back from the enlarged connection to the real size connection; but the difference in the behaviours must be further analysed.


8 HORIZONTAL SHEAR RESISTANCE CALCULATION

8.1 Introduction

The aim of the current research is to calculate the resistance of the embossment based on the method which is used for the shear fasteners e.g. in [1] and [19]. The embossments are considered as individual shear connectors whereof the possible failure modes are determined as: (i) bearing resistance of concrete and (ii) yielding mechanism of steel. The bearing resistance represents the failure of the indentation on the loaded side of the embossment and the yielding mechanism represents the plastic failure of the steel embossment on the sheeting. The concrete type failure (further denoted the resistance by \( P_{b,e} \)) is analogous with the bearing resistance and it emerges by concrete crushing on the loaded side of the embossment. The steel type embossment failure is local bending on the embossments surface with the appearance of a yielding mechanism (further denoted the resistance by \( P_{y,e} \)).

The resistance values are to be used later in the calculation of the horizontal shear strength of the interlock \( \tau_u = F_{ult}/A_s \) which is defined by Eq. (6) according to the evaluation of the pull-out tests. The value of \( F_{ult} \) is to be determined based on the local failure of the embossments whereof: \( F_{ult} = f(P_{b,e}, P_{y,e}) \). The resistance values can be determined semi-empirically.

8.2 Horizontal shear resistance of embossment – concrete failure

8.2.1 General

Based on experimental observations [32] the concrete type embossment failure is followed by concrete crushing in front of the embossment while the steel part of the embossment visibly does not fail. The observed failure is well illustrated by numerical simulation of local models. The failure type is analogous with the bearing failure of shear fasteners, so the bearing resistance of the embossment can be computed from the cross-section in compression of the embossment and the concrete compressive strength. The details of the calculation, however, need further investigation with a special focus on the definition of the cross-section in compression. The experimental background of the calculation is based on small-scale pull-out test data which provide to investigate the relationship between local failure of an embossment and global failure of an embossed plate.

8.2.2 Calculation aspects of bearing resistance

The numerical simulation which is detailed in Chapter 5.4 followed the described behaviour; continuous concrete damage propagation starting from the front of the embossment leads to failure. The ultimate bearing load of the circular embossment is 1.18 kN by the model. To calculate the bearing resistance of the embossment, the area of the embossment under
compression is to be determined. As a first approach the middle cross-section ($A = 21 \text{ mm}^2$) of the embossment is considered (Figure 98/a) as bearing area. Using the concrete compressive strength $f_c = 37 \text{ N/mm}^2$ which is applied in the model, the calculated bearing resistance is 0.78 kN which is found too conservative. Based on the numerical simulation of the circular embossment it can be seen that a bigger area is exposed to compression by the extension of the maximum stresses on the surface of the embossment, as it is shown in Figure 98/b ($A = 29.5 \text{ mm}^2$). By this assumption the bearing resistance is calculated by considering one third of the embossments nappe (Figure 98/c). The resistance value becomes 1.09 kN which is a more realistic approximation comparing to the model.

![Figure 98. Definition of the area of the embossment under compression:](image)

In the next step the calculation is applied to the results of pull-out tests [32]. The geometry of the embossment is shown in Figure 99. The failure according to the test observations is pure concrete failure. The bearing resistance of the embossment is calculated using the cross-section of the middle part which is previously proved to be a conservative approach. The contribution of the embossments in the specimen is considered equal, so the bearing resistance is calculated as:

$$P_{b,e,\text{sum}} = n \cdot P_{b,e}$$

where,

$$n$$ the number of the concrete encased embossments,

$$P_{b,e} = A_{k-k} \cdot f_c$$ the bearing resistance of one embossment.

![Figure 99. Pull-out test and embossment details](image)

The calculation gives a resistance value $P_{b,e,\text{sum}} = 45.09 \text{ kN}$ and the test result is $P_{\text{test}} = 19.13 \text{ kN}$. It can be seen on the concrete shell of the specimen (Figure 99) that the concrete damage is not as deep as the embossment so the bearing does not develop entirely.
Note that the deformation of the embossment in the structure is affected by the deformation of the steel plate whereon the connection is formed [14]. It means that the embossment can step out of the indentation due to the deformation of the steel plate and the force transfer fulfils at another relative position of the embossment and the indentation, as it is shown in Figure 100.

![Diagram](image)

Figure 100. Horizontal shear failure in the structure (a) and in the pull-out tests (b) by [14]

The numerical model analyses an ideal situation of the force transfer and the calculation provides the limit load accordingly, but the embossment does not reach the “ideal” resistance value in the structure. Consequently the calculated bearing resistance of the embossment cannot be directly applied to determine the horizontal shear resistance although the failure mode shows agreement. A reduction factor is suggested to apply on the calculation which defines the portion of the resistance which is taken into account as:

$$P_{b,e} = \beta \cdot A_{k-k} \cdot f_c, \text{ where } \beta < 1.0$$  \hspace{1cm} (9)

It is assumed that the value $\beta$ depends on the material properties of the constituents of the connection. Based on the calculation of shear studs there is likely a relationship with the Young’s modulus of the concrete and based on the observations of [14] there is a relationship with the stiffness of the steel plate, as well. By the current test results [32] $\beta \sim 0.40$ is expected, but the exact definition of the reduction factor needs further investigation.

8.3 Horizontal shear resistance of embossment – steel failure

8.3.1 General

By experimental observations of the enlarged individual embossment the failure is governed by the appearance of the plastic failure on the embossment. The yield line pattern which evolves on the shape of the ultimate deformation of the embossment is found to be typical for the spherical embossment shape. A calculation method is proposed in the followings which take into consideration the pattern of the failure surface.

8.3.2 Resistance of enlarged embossment

During the enlarged pull-out test the loading of specimen 2.2 is stopped at the maximum load. By this specimen the embossment’s deformation at the plastic limit load is available. The shape of the failure surface is rather angular then smooth and the deformation concentrates to the compressed face of the embossment (Figure 101). The top of the embossment is not yet
damaged at this phase. The failure surface is surrounded by a curved boundary and composes of angular-like sections. The pattern of the failure surface is shown in Figure 102.

By experimental observation the failure of the specimen originates from the failure of the steel embossment, which is defined by the plastic failure along the lines of the failure pattern. The phenomenon shows analogy with the plastic failure of bended plates. The plastic failure of the spherical embossment can be approximated by the failure of a plane circular plate under bending: the development of plastic mechanism of yield lines.

![Figure 101. Ultimate deformation of the enlarged embossment side (a) and top view (b).](image)

![Figure 102. Yield line boundaries on the ultimate deformation](image)

![Figure 103. Initial (a) and ultimate (b) deformation on the model](image)

The plastic theory of plates proposes a solution by yield – line theory of slabs by Johansen [33]. The theory is based on plastic behaviour occurring by a pattern of yield lines whereof the location depends on load and boundary conditions. The plastic strength of the slab is evaluated by the equality of the internal virtual work of the yield lines on the rotations and external virtual work of the external forces on the displacements.

The failure surface develops an envelope shape under the loading. The failed spherical surface is replaced in the calculation by a flat surface (substituent plate) with a circular boundary which fits well with the experimental boundary shape. The area of the substituent circular plate is adjusted to be equal to the failed embossment (spherical) surface. The yield lines are defined on the substituent plate in the calculation.

Since the top of the embossment does not fail at the plastic limit load by experimental observation, the position of the substituent plate is defined to connect the bottom with the top...
of the embossment (Figure 105). On the model, the deformation at the plastic limit load supports this assumption: the failed surface of the embossment is between the initial top and bottom and the top of the embossment remains the highest point of the embossment, as it is shown in Figure 103. The area of the failure surface is calculated on the specimen, too, by measuring two perpendicular diameters on it (one in force direction and one perpendicular to it) and counting the base diameter as the average of the measured data. The measured failure surface is 1.25 times bigger then the one which is used in the calculation but the calculation which is supported by the numerical model remains on the safe side by this assumption.

The yield line pattern is semi-empirically determined. The load is introduced on the embossment by the edge of the concrete indentation in front of the embossment. The imprint of this edge is recognized at the upper part of the yield line pattern, as it is shown in Figure 102/a (red dot line). The inner yield lines are determined by the followings (Figure 104):

- (1) the base circle of the embossment is offset to the center of the substituent plate,
- (2) based on the measurement of the experimental yield lines, the arc section of the two circles is replaced by a tri-linear curve,
- (3) the inclined chord is rotated by 90° to get the last two inner yield lines.

![Figure 104. Definition of the yield line geometry.](image)

The internal work of the yield lines on the rotations is written on the basis of Figure 105:

\[
\Pi_i = M_{pl} \sum_i \Theta_i \cdot \Theta_i
\]  

(10)

where,

\[
M_{pl} = \frac{t^2 \cdot f_y}{4}
\]  

is the plastic moment of the plate (\(t\) is the thickness, \(f_y\) is the yield strength),

\(l\) is the length of the yield line section, and

\(\Theta\) is the angle of rotation under the yield line.

The external work of the external forces on the displacements is written by Eq. (11), assuming that the load is introduced by the edge of the concrete indentation in front of the embossment which is replaced by a tri-linear curve in the calculation.
The reaction force which is to be compared to the experimental results is calculated by

$$P_y = \frac{p_m \cdot (2 \cdot \ell_a + \ell_m)}{\sin(\alpha)} + \frac{\mu \cdot p_m}{\cos(\alpha)}$$  \hspace{1cm} (12)

where
\[
\alpha = \text{the definition of } \alpha \text{ is shown in Figure 105,}
\]
\[
\mu = \text{the value of the coefficient of friction (} \mu = 0.2 \text{ is considered in the calculation).}
\]

In the literature various values are measured and proposed for the dry coefficient of friction of steel and concrete interface. Relatively high friction values are found as 0.47 in [34] and 0.57-0.7 in [35]. Open source literature suggests a value of 0.45 (http://www.supercivilcd.com), however, the numerical models developed in Chapter 6 and Chapter 7 show good agreement with test results using 0.3 – 0.4 for the coefficient of friction. Knowing that the friction coefficient is sensitive to manufacturing and surface conditions, as it is detailed in Chapter 5.5.2, the magnitude of this parameter is the most unpredictable. Since the calculation aims not to overestimate the contribution of an uncertain parameter, the value of 0.2 is used, which is consistent with the minimum value proposed in [8]. The results of the calculation, the test and model results are summarized in Table 18.

The plastic limit load from the experiment is calculated as the mean value of the results of the three test specimens by plate thickness. The calculated values show good agreement with the experimental and model results.
Table 18. Calculation vs. test and model results by the enlarged pull-out test

<table>
<thead>
<tr>
<th>Plate thickness [mm]</th>
<th>( P_y ) [kN]</th>
<th>Test [kN]</th>
<th>( P_y ) / ( F_{test} ) [%]</th>
<th>Model [kN]</th>
<th>( P_y ) / ( F_{model} ) [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.5</td>
<td>24.58</td>
<td>29.66</td>
<td>82.9</td>
<td>28.58</td>
<td>86.0</td>
</tr>
<tr>
<td>2</td>
<td>39.77</td>
<td>39.68</td>
<td>100.2</td>
<td>44.30</td>
<td>89.8</td>
</tr>
</tbody>
</table>

The calculation comparing to the model results is always on the safe side by 10-14%. Comparing to the test results the deviation shows less consistency; by 1.5 mm plate thickness the calculation gives ~17% less, by 2 mm plate thickness it gives 0.2% more. However, it is understandable that the model contains less uncertainty than the experiments from the point of view of material and mechanical properties.

8.3.3 Resistance of real size embossment series

The calculation is applied to determine the plastic limit load of the pull-out tests of real size embossment series, as well. The definition of the substituent plate and the yield line pattern is made alike then in the case of the enlarged pull-out test. Based on the experimental observations the embossment have similar failure surfaces, consequently equal contribution in the load bearing, so the total ultimate load \( (P_{y,sum}) \) is calculated by multiplying the ultimate load of one embossment \( (P_y) \) by the number of active embossments in the specimen. The calculation results are compared to the test and model results in Table 19 (columns 1-5).

Using this approach the spacing of the embossments is not taken into account so the ‘2k’ and ‘2s’ type specimens have the same ultimate load.

Table 19. Calculation vs. test and model results by the small pull-out test

<table>
<thead>
<tr>
<th>Specimen</th>
<th>( P_{y,sum} ) [kN]</th>
<th>Test [kN]</th>
<th>( P_{y,sum} ) / ( F_{test} ) [%]</th>
<th>Model [kN]</th>
<th>( P_{y,sum} ) [%]</th>
<th>( P_y ) [kN]</th>
<th>( P_{y,mod} ) [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>12.4</td>
<td>37.30</td>
<td>32.33</td>
<td>115</td>
<td>33.05</td>
<td>113</td>
<td>32.94</td>
<td>100</td>
</tr>
<tr>
<td>12.2s</td>
<td>24.87</td>
<td>23.48</td>
<td>106</td>
<td>27.02</td>
<td>92</td>
<td>24.87</td>
<td>92</td>
</tr>
<tr>
<td>12.2k</td>
<td>24.87</td>
<td>21.72</td>
<td>115</td>
<td>23.31</td>
<td>107</td>
<td>21.96</td>
<td>94</td>
</tr>
<tr>
<td>10.4</td>
<td>27.68</td>
<td>30.80</td>
<td>90</td>
<td>28.24</td>
<td>98</td>
<td>25.77</td>
<td>91</td>
</tr>
<tr>
<td>10.2s</td>
<td>18.46</td>
<td>22.69</td>
<td>81</td>
<td>22.73</td>
<td>81</td>
<td>18.46</td>
<td>81</td>
</tr>
<tr>
<td>10.2k</td>
<td>18.46</td>
<td>20.89</td>
<td>89</td>
<td>20.59</td>
<td>87</td>
<td>17.18</td>
<td>83</td>
</tr>
<tr>
<td>07.4</td>
<td>13.67</td>
<td>21.55</td>
<td>63</td>
<td>17.57</td>
<td>78</td>
<td>13.37</td>
<td>76</td>
</tr>
<tr>
<td>07.2s</td>
<td>9.11</td>
<td>18.33</td>
<td>51</td>
<td>13.30</td>
<td>69</td>
<td>9.11</td>
<td>68</td>
</tr>
<tr>
<td>07.2k</td>
<td>9.11</td>
<td>18.24</td>
<td>50</td>
<td>12.36</td>
<td>74</td>
<td>8.91</td>
<td>72</td>
</tr>
</tbody>
</table>

The experiments, however, showed that the ‘2s’ type specimens developed individual failure by embossments and in the ‘2k’ type specimens the embossments affected each others deformation consequently the ‘2k’ specimens gave slightly smaller (max ~8%) ultimate load. The individual and grouped deformation is well illustrated by the model in Figure 106. It is
observed that the longitudinal extension of the deformation is approximately the same for two individual and for the group of two embossments a transversal delamination, however, can be seen between the steel and the concrete interface for the group of embossments. Due to the delamination the embossment comes out of its indentation and the load transfer point shifts towards to the top of the embossment. The phenomenon can be seen on the plastic strain distribution of the embossment (Figure 107): the frontal face of the embossment is loaded in the case of individual failure and the top region of the loaded side contributes in the load bearing capacity in the case of grouped failure.

![Figure 106. Individual and grouped embossment deformation](image)

![Figure 107. Plastic strain distribution on the embossment in case of (a) individual (b) grouped failure](image)

![Figure 108. Relationship of the plastic limit load ratio of individual and grouped failure](image)

The numerical simulations show that the ratio of the plastic limit loads of ‘2k’ and ‘2s’ type models is not constant but the difference is bigger by thicker plates. It suggests that in case of softer plates the effect due to the interaction of the deformations in the embossment group becomes less significant. The relationship between the plate thickness and the resistance reduction can be expressed by Eq. (13). The function represents the effect of the plate stiffness and it shows good accuracy with the model results (Figure 108).

\[
P_{y,2} = \left(1 - \frac{t^3}{12}\right)P_{y,1}
\]  

(13)
where,
\[ t \] the plate thickness in [mm],
\[ P_{y,g} \] is the plastic limit load of one embossment in the group, and
\[ P_{y,i} = P_y \] the plastic limit load on the individual embossment failure.

In the studies individual and grouped failure modes are observed. The mode of failure depends mainly on the spacing of the embossment. The longitudinal extension of the ultimate deformation for the individual failure is \(~39\) mm (the base diameter of the embossment is \(12\) mm). In the ‘2k’ and ‘2s’ specimens the embossments are formed 35 and 70 mm far from each other, respectively (the distance is measured between the top of the embossments). It is clear that in the ‘2k’ specimens the embossments are formed just close enough not to develop individual failure. The effect of the embossment distance is found an influential factor so it is subjected to further investigations.

The total plastic limit load of the specimens developing grouped embossment failure is calculated using the relationship by Eq. (14). The plastic limit load of the specimens showing individual embossment failure is calculated by Eq. (15). The modified calculation results remain on the safe side and they show good agreement with the model results (Table 19, column 6-7).

\[
P_{y,mod,i} = n \cdot P_{y,g} = n \cdot \left(1 - \frac{13}{12}\right) P_{y,i} \quad (14)
\]

\[
P_{y,mod,g} = n \cdot P_{y,i} \quad (15)
\]

The relationship gives a limit to the application: over a certain plate thickness (2.3 mm) the embossment yielding cannot be taken into account, other behaviour is to be expected. The validity of the relationship, however, needs further investigation since the experimental and modelling background is yet narrow.

Currently the calculation assumes that the mode of failure (grouped or individual) is a-priori known from tests or models. The failure mode depends on the distance of the embossments, however, only two specific distances are analysed during the tests, so the relationship is not clarified. Accordingly the equations (13) – (15) is not yet suitable to predict the failure mode, their improvement is the subject of further investigation.

In Chapter 8.2.2 it is proved that the horizontal shear resistance of a pull-out test cannot be determined directly from the local resistance of the embossments on the steel surface, because the global behaviour of the steel ribs affects the failure of the embossments in the structure. It is expected that this assumption is true on the steel type resistance, as well. The relationship is to be determined by further investigation.
9 CONCLUSIONS

Results of the presented research work are summarized in the following new scientific results.

9.1 New scientific results

9.1.1 Theses of the dissertation in English

1. Thesis

I developed a novel experimental procedure and designed a new composite specimen to study the behaviour of concrete encased embossment of steel strips under shear action for composite slabs with profiled steel decking.

1(a) I executed an experimental program on individual enlarged spherical embossments by the developed procedure; on the basis of the test results the followings can be stated:

- The method and the specimen are appropriate to study the local behaviour of the embossments.
- By the applied parameter range the observed failure mode is governed by the steel behaviour, the effect of concrete damage is negligible.
- The steel type of failure is characterized by the extension of yielding zones due to local bending and followed by large deformations (ductile behaviour).

1(b) I executed an experimental program on real size spherical embossments by the developed procedure; on the basis of the test results the followings can be stated:

- The significant effect of the transversal compression force on the behaviour is showed.
- The same steel type of failure mode in each embossment of the series in the specimen is observed as in the case of individual embossments.
- The effect of the spacing of the embossments on the ultimate behaviour is showed and explained.

Publications linked to the thesis: [41], [42], [44], [45], [46].
2. Thesis

I developed numerical model for the simulation of the concrete type embossment behaviour. The failure is governed by the concrete damage; the effect of steel failure is ignored. I completed push-out test simulations on two different shapes of individual embossments. On the basis of the results the followings can be stated:

- The adopted concrete material model is calibrated with two experimental investigations whereof the mesh sensitivity and the load-increment dependency of the results is determined.
- The ultimate behaviour is rigid, crack initiation and propagation on the loaded face of the embossment leads to the final failure.
- It is proved that simplification of the rounded edge to sharp edge of spherical embossment in the model does not influence the ultimate behaviour but it increases the stiffness.
- The tendencies of changing geometrical and physical parameters are determined; the observations are verified by published experimental results.

Publications linked to the thesis: [38], [39], [40], [48].

3. Thesis

I developed a three-step numerical model for the analysis of the different phases of the manufacturing and loading of steel spherical embossment’s behaviour, as follows: (1) extruding of the embossment on the steel strip by spherical tool, (2) separation of the extruding tool from the sheet, and (3) completing the pull-out test. In phase (3) only the failure of the steel is considered; the effect of concrete damage is ignored. The developed model is proved to have appropriate accuracy and efficiency for the simulation of the behaviour and plastic limit load of one enlarged and the real size embossment series.

Publications linked to the thesis: [43], [47].
4. Thesis

I applied the developed numerical models for parametric studies on the simulation of the manufacturing process, testing procedure and embossments’ behaviour.

- By the simulation of the manufacturing process it is proved that it has significant effect on the stiffness but does not influence the resistance of the embossment.
- By the simulation of the pull-out test procedure the sensitivity of the behaviour and ultimate load on the applied transversal compression force is proved. This finding can be used in the design of small-size pull-out testing method.
- By the simulation of the behaviour of the embossments the tendencies of the (i) geometrical parameters (height, plate thickness, number and arrangement of embossments), the (ii) friction coefficient, and the (iii) size effect are determined.

Publications linked to the thesis: [45], [47].

5. Thesis

I proposed to determine the resistance of an embossment on the basis of the local failure components as follows: (i) failure from the crushing of the concrete on the loaded side of the embossment, (ii) failure of the steel embossment due to yielding and deformation, and (iii) friction after delamination of the interface. The failure components are considered independent.

I determined the characteristics of the steel type embossment failure by experimental and numerical investigations. I developed a calculation method for the resistance of an individual embossment using the theory of yield lines based on the ultimate deformation (plastic mechanism). I extended the method for real size embossment groups by considering the spacing.

I determined the characteristics of the concrete type embossment failure by numerical investigations. I proposed a calculation method for the bearing resistance of an individual embossment. An empirical reduction factor can be derived to link the local failure of an embossment and the global failure of embossment series on a pull-out test.

Publications linked to the thesis: [49]
9.1.2 Theses of the dissertation in Hungarian

1. Tézis

Kísérleti eljárást dolgoztam ki, valamint újszerű öszvér kísérleti próbatestet terveztem profillemezes öszvérődének betonba ágyazott, az acéllemez felületén domborítással létrehozott mechanikus nyírt kapcsolatának (a továbbiakban domborítás) acélszalagon történő vizsgálatára.

1(a) A eljárás felhasználásával kísérleti programot hajtottam végre különálló, felnagyított, gömbsüveg alakú domborításon, amelyek eredményei alapján a következőket állapítottam meg:

- A kísérleti eljárás és a próbatest alkalmas a domborítás lokális vizsgálatára.
- Az alkalmazott paraméterek mellett acél tönkremenetelt tapasztaltam, a beton tönkremenetel elhanyagolható volt.
- Az acél tönkremenetel a domborítás lokális hajlításaként jött létre képlekeny zónák terjedése és összekapcsolódása után, nagy alakváltozások mellett (a viselkedés duktilis).

1(b) A kidolgozott kísérleti eljárással kísérleti programot hajtottam végre valós méretű, gömbsüveg alakú domborításokon. A kísérlet eredményei alapján a következőket állapítottam meg:

- Az oldalirányú összeszorító erőnek jelentős hatása van a próbatest viselkedésére.
- A valós méretű domborítások mindegyike ugyanazt az acél tönkremeneteli módot mutatta, mint a különálló felnagyított domborítás.
- A domborítások számának és kiosztásának teherbírásra gyakorolt hatását bemutattam és megmagyaráztam.

A tézishoz kapcsolódó publikációk: [41], [42], [44], [45], [46].
2. Tézis

Numerikus modellt dolgoztam ki a domborítás beton oldali viselkedésének modellezésére és vizsgálatára. A tönkremenetelt a beton károsodás okozta, az acél tönkremenetel hatását elhanyagoltam.

Szimulációt hajtottam végre különböző alakú domborítások kinyomó vizsgálatára. Az eredmények alapján a következőket állapítottam meg:

- Az alkalmazott beton anyagmodellt két kísérlet segítségével kalibráltam, melyeken a végselem háló, illetve a teherlépcső hatását az eredményekre meghatároztam.
- A tönkremenetel rideg, a domborítás terhelt felületén a betonban megjelenő repedés terjedése okozza.
- Bebizonyítottam, hogy a domborítás élein a lekerekítés elhanyagolása nem befolyásolja a kapcsolat teherbírását, azonban megnöveli annak merevségét.
- A geometriai és fizikai paraméterek változtatásából adódó tendenciák jó egyezést mutatnak a szakirodalomban publikált kísérleti eredményekkel.

A tézishez kapcsolódó publikációk: [38], [39], [40], [48].

3. Tézis

Háromlépéses numerikus modellt dolgoztam ki gömbsüveg alakú acél domborítás megmunkálásának és terhelésének leírására a következők szerint: (1) domborítás besajtolása az acél szalagba, (2) a megmunkáláshoz kapcsolódó modellrések eltávolítása és (3) kihúzóvizsgálat végrehajtása. A harmadik lépésben csak acél tönkremenetelt vettem figyelembe, a beton tönkremenetel elhanyagoltam.

A kidolgozott modellről bebizonyítottam, hogy kellő pontossággal és hatékonysággal követi a különálló felnagyított és a valós méretű domborítás sorozat viselkedését.

A tézishez kapcsolódó publikációk: [43], [47].
4. Tézis

A kidolgozott modellek segítségével paraméteres vizsgálatot hajtottam végre a domborítás megmunkálásával, a kísérleti eljárással és a kapcsolat viselkedésével kapcsolatosan.

- Bebizonyítottam, hogy a domborítás megmunkálása jelentős hatással van a kezdeti merevségre, de nem befolyásolja a kapcsolat teherbírását.
- Bebizonyítottam, hogy a kihúzóvizsgálat érzékeny az oldalirányú összeszorító erő hatására, amely a viselkedésben és a teherbírásban is megnyilvánul. Ez a hatás kiküszöbölendő kiselemes kihúzóvizsgálat eljárásának tervezésekor.
- Tendenciákat határoztam meg a kapcsolat viselkedése és a (i) domborítás geometriája (magasság, lemezvastagság, domborítások száma elrendezése és száma), (ii) a súrlódási tényező, és a (iii) mérethatás között.

A tézishez kapcsolódó publikációk: [45], [47].

5. Tézis

Javaslatot tettem az acéllemez felületén domborítás alapú mechanikus nyírt kapcsolat ellenállásának meghatározására annak lokális tönkremeneteli módjai alapján, mint: (i) a domborítás terhelt oldalán létrejövő lokális beton tönkremenetel, (ii) a domborítás tönkremenetele megfolyás és deformáció következtében, és (iii) súrlódásos erőátadás az érintkező felületek elválása (delamináció) után. A tönkremeneteli módokat egymástól függetlennek tekintettem.

Meghatároztam az acél oldali tönkremenetel jellemzőit kísérleti és numerikus vizsgálatok alapján. Számítási módszert dolgoztam ki a képlekkeny törésvonal elmélet alkalmazásával a domborítás ellenállásának számítására, a domborítás teherbírásához tartozó deformációja alapján (képlekkeny mechanizmus). A számítást kiterjesztettem domborítás csoportok számítására, figyelembe véve a domborítások kiosztását.

Meghatároztam a beton oldali domborítás tönkremenetet jellemzőt numerikus vizsgálatok segítségével. Számítási módszert javasoltam különálló domborítás palástnyomási ellenállásának meghatározására. Kísérlet alapú csökkentő tényezőt javasoltam a domborítás lokális tönkremenetele és kiselemes kihúzóvizsgálatokban létrejövő domborítás sorozat globális tönkremenetele közötti kapcsolat leférésére.

A tézishez kapcsolódó publikációk: [49]
9.2 Proposal for further research

9.2.1 Experimental investigation and numerical studies

The current research has limited validity regarding the shape and the spacing of the embossments. The experimental procedure is proved to be appropriate to analyse one specific embossment shape. Furthermore, the calculation is worked out for the circular embossment shape and for two specific distances where the embossment show individual or grouped failure. It is known that the shape and spacing of the embossments has large diversity. The experiment is to be verified for other embossment geometries. Parametric study is needed furthermore to investigate the effect of spacing of the embossments on the load carrying capacity to determine the relationship for other (closer/farther) spacing conditions then the analysed distances.

The experimental investigation and the numerical study of the current research based on a novel composite specimen. Difficulties arose during the preparation of the specimen as casting the concrete and keep the steel strips in position, avoid the global concrete failure and control the transversal compression, etc. Specification is needed about the preparation of the specimens and look for other options to make the specimens better. Further specification is needed also on the details of the numerical model, as well. The model is now appropriate to follow the actual experiments and the goal is to define the general guidelines to make it available for further analysis.

9.2.2 Semi-empirical simulation based partial shear connection method

A calculation is proposed for two local failure components: concrete type and steel type embossment failure ($P_{b,e}$, $P_{y,e}$). The next step of the research is to establish the relationship between the local failure and pull-out test results which makes the calculation adoptable for the partial shear connection method. In the thesis a proposal is made by defining a reduction factor $\beta$ to get the description of the global behaviour ($\tau_u$) from the local failure. Parallel experimental and numerical studies are needed to determine the parameters (physical, material, etc.) that affect the relationship and also the properties of the effects.

Note that there is a lack of experimental data of pull-out test regarding the details. If pull-out test results are presented in the literature the most commonly given data is the value $\tau_u$. The presentation of the embossment features: geometry, spacing is usually missing and the mode of failure is not taken care of. Those data, however, are needed to develop the embossment’s resistance.
9.2.3 Enhanced concrete modelling

A further goal of the research is to avoid the observed mesh sensitivity in the local models. To be able to create accurate local models of rolled embossments and get better data for the ultimate load and crack propagation a higher scale material model can be used. This model scale can be the mesoscale, where the internal structure of the material is considered: the concrete is modelled by the coarse aggregates, the surrounding mortar matrix and by the interface elements on the aggregate-matrix boundary. Recently researchers proposed several types of models in two fields for this material scale: discontinuous and continuous models e.g. [36], [37]. Generally it can be told that this field of material modelling is a dynamically developing and promising research area and its models give very good accordance with experimental results (examples are given in Figure 109 and Figure 110). Further ambition is to improve the embossment model by mesoscale material model.

Figure 109. Mesoscale discrete element model results: uniaxial compression test [36]

Figure 110. Mesoscale finite element model results: uniaxial tension test – nominal stress-strain relationship and final damage distribution [37]
REFERENCES


[23] ANSYS® v11.0, Canonsburg, Pennsylvania, USA.


PUBLICATIONS ON THE SUBJECT OF THE THESIS


