Improving Robustness Assessment Methodologies for Structures Impacted by Missiles (IRIS_2012)

Final Report

This document only exists in PDF format.

JT03359994

Complete document available on OLIS in its original format

This document and any map included herein are without prejudice to the status of or sovereignty over any territory, to the delimitation of international frontiers and boundaries and to the name of any territory, city or area.
ORGANISATION FOR ECONOMIC CO-OPERATION AND DEVELOPMENT

The OECD is a unique forum where the governments of 34 democracies work together to address the economic, social and environmental challenges of globalisation. The OECD is also at the forefront of efforts to understand and to help governments respond to new developments and concerns, such as corporate governance, the information economy and the challenges of an ageing population. The Organisation provides a setting where governments can compare policy experiences, seek answers to common problems, identify good practice and work to co-ordinate domestic and international policies.

The OECD member countries are: Australia, Austria, Belgium, Canada, Chile, the Czech Republic, Denmark, Estonia, Finland, France, Germany, Greece, Hungary, Iceland, Ireland, Israel, Italy, Japan, Luxembourg, Mexico, the Netherlands, New Zealand, Norway, Poland, Portugal, the Republic of Korea, the Slovak Republic, Slovenia, Spain, Sweden, Switzerland, Turkey, the United Kingdom and the United States. The European Commission takes part in the work of the OECD.

OECD Publishing disseminates widely the results of the Organisation’s statistics gathering and research on economic, social and environmental issues, as well as the conventions, guidelines and standards agreed by its members.

NUCLEAR ENERGY AGENCY

The OECD Nuclear Energy Agency (NEA) was established on 1 February 1958. Current NEA membership consists of 31 countries: Australia, Austria, Belgium, Canada, the Czech Republic, Denmark, Finland, France, Germany, Greece, Hungary, Iceland, Ireland, Italy, Japan, Luxembourg, Mexico, the Netherlands, Norway, Poland, Portugal, the Republic of Korea, the Russian Federation, the Slovak Republic, Slovenia, Spain, Sweden, Switzerland, Turkey, the United Kingdom and the United States. The European Commission also takes part in the work of the Agency.

The mission of the NEA is:

– to assist its member countries in maintaining and further developing, through international co-operation, the scientific, technological and legal bases required for a safe, environmentally friendly and economical use of nuclear energy for peaceful purposes, as well as
– to provide authoritative assessments and to forge common understandings on key issues, as input to government decisions on nuclear energy policy and to broader OECD policy analyses in areas such as energy and sustainable development.

Specific areas of competence of the NEA include the safety and regulation of nuclear activities, radioactive waste management, radiological protection, nuclear science, economic and technical analyses of the nuclear fuel cycle, nuclear law and liability, and public information.

The NEA Data Bank provides nuclear data and computer program services for participating countries. In these and related tasks, the NEA works in close collaboration with the International Atomic Energy Agency in Vienna, with which it has a Co-operation Agreement, as well as with other international organisations in the nuclear field.

This document and any map included herein are without prejudice to the status of or sovereignty over any territory, to the delimitation of international frontiers and boundaries and to the name of any territory, city or area.

Corrigenda to OECD publications may be found online at: www.oecd.org/publishing/corrigenda.
© OECD 2014

You can copy, download or print OECD content for your own use, and you can include excerpts from OECD publications, databases and multimedia products in your own documents, presentations, blogs, websites and teaching materials, provided that suitable acknowledgment of the OECD as source and copyright owner is given. All requests for public or commercial use and translation rights should be submitted to rights@oecd.org. Requests for permission to photocopy portions of this material for public or commercial use shall be addressed directly to the Copyright Clearance Center (CCC) at info@copyright.com or the Centre français d'exploitation du droit de copie (CFC) contact@cfcopies.com.
THE COMMITTEE ON THE SAFETY OF NUCLEAR INSTALLATIONS

“The Committee on the Safety of Nuclear Installations (CSNI) shall be responsible for the activities of the Agency that support maintaining and advancing the scientific and technical knowledge base of the safety of nuclear installations, with the aim of implementing the NEA Strategic Plan for 2011-2016 and the Joint CSNI/CNRA Strategic Plan and Mandates for 2011-2016 in its field of competence.

The Committee shall constitute a forum for the exchange of technical information and for collaboration between organisations, which can contribute, from their respective backgrounds in research, development and engineering, to its activities. It shall have regard to the exchange of information between member countries and safety R&D programmes of various sizes in order to keep all member countries involved in and abreast of developments in technical safety matters.

The Committee shall review the state of knowledge on important topics of nuclear safety science and techniques and of safety assessments, and ensure that operating experience is appropriately accounted for in its activities. It shall initiate and conduct programmes identified by these reviews and assessments in order to overcome discrepancies, develop improvements and reach consensus on technical issues of common interest. It shall promote the co-ordination of work in different member countries that serve to maintain and enhance competence in nuclear safety matters, including the establishment of joint undertakings, and shall assist in the feedback of the results to participating organisations. The Committee shall ensure that valuable end-products of the technical reviews and analyses are produced and available to members in a timely manner.

The Committee shall focus primarily on the safety aspects of existing power reactors, other nuclear installations and the construction of new power reactors; it shall also consider the safety implications of scientific and technical developments of future reactor designs.

The Committee shall organise its own activities. Furthermore, it shall examine any other matters referred to it by the Steering Committee. It may sponsor specialist meetings and technical working groups to further its objectives. In implementing its programme the Committee shall establish co-operative mechanisms with the Committee on Nuclear Regulatory Activities in order to work with that Committee on matters of common interest, avoiding unnecessary duplications.

The Committee shall also co-operate with the Committee on Radiation Protection and Public Health, the Radioactive Waste Management Committee, the Committee for Technical and Economic Studies on Nuclear Energy Development and the Fuel Cycle and the Nuclear Science Committee on matters of common interest.”
APPENDIX II

Numerical Simulation Reports
1.0 Introduction

A round robin exercise (RRE) on ‘Improving the Robustness assessments methodologies for structures Impacted by missiles (IRIS-2012)’ was instituted jointly by IRSN, France and CNSC, Canada based on experiments conducted by VTT, Finland. Outcome of the RRE (1st phase) completed in 2010 was not encouraging. The RRE (2nd phase) was floated again in the year 2012 to improve the simulation results close to the experiments. A team from AERB, India is also participating in IRIS-2012 for the first time. The simulations consist of bending and punching test of reinforced concrete (RCC) slab with soft and hard/rigid missiles respectively. The main objective of this exercise is to improve capability regarding damage assessment of NPP structure by commercial airplanes crashes. As this type of analysis is extremely complex in nature, experiments as well as simulations had been done using simple structures like RCC slab as target and steel cylinders as missiles, hard/soft. The objective of the round robin analysis is to develop computational capability to simulate impact analysis.

2.0 Objective

The objective of this study is to benchmark the bending as well as punching impact test results through numerical simulation using finite element technique. Thus to get an opportunity to participate in this round robin analysis and contribute in recommendations regarding the simulations and the assessment of damage to reinforced concrete slabs by missiles at medium velocity.

3.0 Scope

The scope of this exercise is limited to finite element simulation of bending test and punching test conducted using soft and hard missiles respectively.

4.0 Structure of the report

In item no. 5.0 the details of bending tests and analyses are presented and item no. 6.0 describes the simulation of punching test. All the references are furnished in item 7.0.
5.0 Simulation of Bending Test for IRIS-2012

5.1 General

Bending test with soft missile impact on a Reinforced Concrete (RC) Slab was conducted at VTT, Finland. Present study is aimed at benchmarking the test results through FE simulation of the impact problem.

5.2 FE Model

The target was a 2.1m x 2.1m x 0.25m RC slab. Target was modelled using solid elements, whereas individual reinforcing steel including shear links were modelled using truss elements as shown in Fig. 5.1(a)&(b).

![Fig 5.1 FE model of RC Slab (a) and Reinforcement (b)](image)

The missile was modelled as shell. Fig. 5.2 shows the sectional view of the missile.

![Fig. 5.2 Sectional view of soft missile.](image)
Number of elements across the slab thickness plays an important role in capturing the strain rate occurring in the model. Mesh-Sensitivity study was done and 15 mm was taken as the mesh size so as to give 10 elements across the slab. Same element size was adopted for meshing the missile and reinforcing steel. Hourglass control and adaptive meshing option was utilized for numerical stability of the solution procedure.

5.3 Material Model

5.3.1 Concrete
Nonlinear behaviour of concrete was modelled using the concrete damage plasticity (CDP) model in ABAQUS. A strain rate sensitive elasto-viscoplastic material model was employed. Stress-strain data in compression for different confinement pressures were provided as input. European CEB (Comite Euro-International du Beton, 1993) recommends dynamic increase factors (DIF formulas) for concrete in both tension and compression [1]. These relations were used to convert the given confining pressures to strain rate. Cracking stress value was provided for concrete tension. Bi-linear stress-displacement curve proposed by Gylltoft (1983) [2] was used, which is based on stress-crack opening relationship. Fracture energy (\(G_f\)) was considered as 110 N/m assuming 20 mm as maximum aggregate size [3]. Figure 5.3 (a) & (b) shows both the compression and tension models for different strain rates.

![Fig 5.3 Stress-strain curves of concrete at different strain rates, in compression (a) and in tension (b)](image)

5.3.2 Steel
Both reinforcing and carbon steel (missile) were modelled using elasto-plastic stress-strain relations. Strain rate effects were taken into account in reinforcement steel material model by specifying yield stress ratios, a factor, by which the yield strength of material increases with given strain rate.

5.4 Load
Missile velocity of 110 m/s was given as a predefined field.

5.5 Boundary Condition
Earlier studies [4] on soft missile impact revealed that supporting frame is sufficiently rigid to prevent large rigid body displacement of the test slab and can be neglected in modelling.
Out of plane restraint was applied on both front and rear face along all four edges 50 mm before the slab boundary to simulate simple support condition. Figure 5.4 shows the slab with the applied boundary conditions.

![Fig. 5.4 RC Slab with applied boundary conditions.](image)

5.6 Methodology and Assumptions

(i) Supporting steel frame arrangement was not modelled in the study.
(ii) Reinforcing steel was assumed to be embedded into concrete and bond-slip behaviour was not modelled.
(iii) Strain rate effects were not considered for missile.
(iv) Element erosion could not be simulated with the available concrete material model.
(v) Damage at multiple strain rates could not be modelled.
(vi) Missile to slab interaction was modelled through hard contact in normal direction. Friction between the two was neglected.

5.7 Analysis
The analysis ran for 100 milli-secs and output results were written in 0.25 milli-secs interval. The stable time increment, observed throughout the analysis, was of the order of 1.12E-6 s.

5.8 Results

5.8.1 Missile
The displacement and load time history of missile is shown in Fig. 5.5(a) and (b) respectively.

![Fig.5.5 Displacement time history of the missile (a). Load time history between missile and target (b).](images)
5.8.2 Target

The crack pattern for front and back side is shown in Figure 5.6 (a) and (b) respectively by plotting contour of maximum principle strain. The minimum threshold value of strain was fixed as ten times the initial cracking strain.

![Front side and back side crack pattern](image)

**Fig. 5.6** Front side (a) and back side (b) crack pattern of the Slab after impact.

The displacement time history at the rear end of the slab (at centre of the slab) and reaction force time history generated at support are shown in Figure 5.7 (a)& (b) respectively.

![Displacement and Force time history](image)

**Fig. 5.7** Displacement (a) and Force (b) time history of the target.

5.9 Conclusion

(i) Missile was deformed to half of its original length.
(ii) The impact happened within 22 milli-secs after which the missile loose contact with the slab and started returning back.
(iii) The total impulse generated in this phenomena was estimated to be 5.52 kN.s
(iv) The above observations were found to be in good agreement with the studies conducted by other researchers using Riera method [4].
(v) The maximum slab displacement was observed to be 30mm, which matched well with the experimental results.
(vi) The maximum value of compressive strain in the front end of the slab and the peak value of support reactions were in good agreement with the results observed experimentally.
(vii) The maximum strain rate generated was of the order of 250-300 per second.
(viii) The free-vibration behaviour of the slab, post impact, could not be captured properly resulting in lower recovery of slab displacement.
(ix) The scabbing and spalling behaviour could not be studied with the available material model.

6.0 Simulation of Punching Test for IRIS-2012

6.1 General
The punching test of IRIS model was done using a reinforced concrete slab with bending reinforcements only subject to impact load of hard missile at a velocity of 135 m/s.

6.2 Geometric Model
The target was a 2.1m x 2.1m x 0.25m reinforced concrete slab. Target was modelled as solid, where as individual reinforcing steel was modelled using line geometry, Fig. 6.1(a)&(b).

![Fig. 6.1 Geometric model of target, (a) concrete and (b) reinforcements](image)

In case of rigid missile, the outer steel layer as well as light weight concrete was modelled using solid geometry. Fig. 6.2 shows the sectional view of outer steel layer 2(a) and light weight concrete 2(b) respectively. Tail of aluminium pipe was not modelled for simplicity.
Fig. 6.2 Sectional view of outer steel layer (a) and light weight concrete fill (b) of hard missile

6.3 Material Model

6.3.1 Target
Concrete models available in ABAQUS/Explicit are concrete damage plasticity model (CDP), smeared crack model and brittle crack model. However, except brittle crack model, other constitutive models of concrete do not have the capability to simulate ‘element erosion’ [5]. In case of brittle model, the compressive behaviour is always assumed to be elastic. Modelling of ‘element erosion’ is necessary for simulation of penetration, perforation, spalling and scabbing of target in case of punching test. So, elastic-plastic constitutive relation (compression as well as tension) is used as concrete material of the target. Two different failure criteria, such as ‘shear failure strain’ and ‘tensile failure hydrostatic stress (cut-off stress)’ are incorporated in that model to simulate ‘element erosion’ of the target. The reinforcement steel is modelled as elastic-plastic material as per given data with ‘shear failure strain’ criteria. Fig. 6.3 shows the constitutive relationship of concrete slab 6.3(a) and its reinforcements 6.3(b).

(a) (b)

Fig. 6.3 Stress-strain curves of slab concrete (a) and reinforcing steel (b)

6.3.2 Missile
Outer steel layer of the missile is modelled as elastic-plastic material without any failure criteria and the filler concrete is modelled using CDP material with low density (1520 Kg/m³). Fig. 6.4 shows the constitutive relationship of outer steel layer 6.4(a) and light weight concrete fill 6.4(b) of rigid missile.
The numerical simulation is performed in commercially available finite element software ABAQUS/Explicit Code. The total model is made in different parts, such as target slab, reinforcements, missile outer steel layer and filled light weight concrete and assembled based on their position and interaction. The outer surface of light weight concrete fill is ‘tied’ with inner surface of steel layer of missile. The reinforcing steels are ‘embedded’ into concrete slab, thus maintain perfect bond between concrete and steel. Missile and target (except rebar) are modelled using linear hexahedral (type C3D8R) reduced integration solid elements. The rebar or reinforcing steel of the target is modelled using truss linear line elements (type T3D2). The average size of elements is 15mm. Details of elements and nodes are tabulated below, Table 6.1.

<table>
<thead>
<tr>
<th>Part Name</th>
<th>Number of Elements</th>
<th>Number of Nodes</th>
</tr>
</thead>
<tbody>
<tr>
<td>Missile outer steel layer</td>
<td>1873</td>
<td>3613</td>
</tr>
<tr>
<td>Missile light weight concrete fill</td>
<td>4485</td>
<td>5280</td>
</tr>
<tr>
<td>Concrete target</td>
<td>305984</td>
<td>329817</td>
</tr>
<tr>
<td>Reinforcing steels of target</td>
<td>15552</td>
<td>14964</td>
</tr>
</tbody>
</table>

**6.5 Boundary Condition**

The supporting frame is not modelled. The target is simply supported at four sides of front as well as back faces, 50 mm from edges. The target is also simply supported at bottom, mid thickness line. The movement of missile is allowed only in perpendicular direction of target, thus restrained from any rigid body rotation during penetration/perforation.

**6.6 Methodology and Assumptions**

The target slab is ‘partitioned’ based on its failure criteria due to missile impact [6]. Generally from literature on impact mechanics [7], [8], it is understood that thickness near
to impact zone of target undergoes compaction, whereas at other side or back side of target experiences tensile stress and creates scabbing. The compressive shock wave returns from back faces into tensile shock wave and creates spalling at front face. So, it is assumed that 1/4\textsuperscript{th} thickness of the target slab near impact zone fails in shear (simulate penetration) and remaining areas of target fail in hydrostatic tensile stress (simulate perforation) or cut-off stress. The ‘shear failure’ zone is cylindrical with diameter nearly equal to missile diameter and the length is equal to 1/4\textsuperscript{th} thickness of the slab. The concrete at boundary zone (50 mm from edge) is assumed to be elastic without any failure to ascertain numerical stability of the analysis preventing ‘element erosion’ at support locations.

The ‘shear failure’ of reinforcement steel is assumed as 18% (uniaxial failure strain of this steel is 20% as per supplied test data). The uniaxial tensile stress of target concrete is 4 MPa, where as supplied experimental data shows an enhancement of this tensile stress nearly 20 MPa under high ‘strain rate’ 100 s\textsuperscript{-1}. Thus, cut-off stress of concrete target is considered as 20 MPa. The ‘shear failure’ of concrete of target is defined as 18% after simulating a large number of iterations with these failure criteria to achieve desire responses.

6.7 Analysis

The missile is placed just in contact with target before analysis, thus to simulate the analysis immediately at the impact. ABAQUS/Explicit Code is used with automatic time steps, where ‘stable time increment’ is 1.4 microsecond. The analysis is simulated for 100 millisecond (ms). The responses of missile and target are extracted for 25 ms and 100 ms respectively. The missile velocity of 135 m/s is applied in ‘predefined field’ of initial step.

6.8 Results

6.8.1 Missile

The trail velocity of the missile is found to be 33.4 m/s. The trail velocity of the missile is measured at the time, when the missile has just perforated the target fully (estimated as 2.75 ms). Fig. 6.5 shows the missile velocity 6.5(a) and displacement 6.5(b) time histories.

![Velocity time history of the rear of the missile during impact](image1)

![Displacement time history of the rear of the missile during impact](image2)

(a) (b)

Fig. 6.5 Time histories of missile velocity (a) and displacement (b)

The total impact duration is found to be nearly 5.0 ms after that no contact force is observed. Fig. 6.6 shows the load time history between missile and target 6.6(a) and impulse time history received by the target 6.6(b).
6.8.2 Target

The detailed response of the target is furnished in Table 6.2 below.

<table>
<thead>
<tr>
<th>Penetration (yes/no)</th>
<th>Perforation (yes/no)</th>
<th>Spalling (yes/no)</th>
<th>Scabbing (yes/no)</th>
<th>Cone cracking (yes/no)</th>
<th>Radial cracking (yes/no)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table 6.2 Response details of the target

- Horizontal width of cracked area: 0.2 m
- Vertical width of cracked area: 0.2 m
- Horizontal width of spalling: 0.2 m
- Vertical width of spalling: 0.2 m
- Crushed depth: 0.078 m

Fig. 6.7 shows the comparative study of spalling of target in experiment 6.7(a) and simulation 6.7(b)

(a) (b)

**Fig. 6.6 Load time history between missile and target (a) and impulse time history of the target (b)**

**Fig. 6.7 Spalling of target at front face; experiment (a) and simulation (b) (black areas are eroded elements)**
Fig. 6.8 shows the comparative study of scabbing of target in experiment 6.8(a) and simulation 6.8(b).

**Fig. 6.8 Scabbing of target at back face; experiment (a) and simulation (b)**

Fig. 6.9 shows the comparative study of damage in horizontal section in experiment 6.9(a) and numerical simulation 6.9(b).

**Fig. 6.9 Horizontal cut sectional view of experiment (one quadrant) (a) and simulation (b)**

Fig. 6.10 shows the comparative study of damage in vertical section in experiment 6.10(a) and numerical simulation 6.10(b).

**Fig. 6.10 vertical cut sectional view of experiment (one quadrant) (a) and simulation (b)**
6.9 Conclusion

The following conclusion may be drawn based on this simulation.

(i) The responses of the target are highly dependent on failure criteria of different material, such as concrete and reinforcement steel.

(ii) The mesh should be sufficiently fine to capture all the responses of target due to impact of hard missile.

(iii) As CDP model does not have the facility of ‘element erosion’, the alternative is to assume concrete as elastic-plastic material with ‘failure criteria’. Another option can be incorporating new concrete material with efficient constitutive relations to capture all these phenomena.

(iv) Holmquist-Johnson-Cook (HJC) constitutive model for concrete (inbuilt model in ABAQUS 6.10 or later version) is another option. The given test data of concrete are not sufficient for HJC model [8].

7.0 Reference


[3] JSCE Guidelines for Concrete


REPORT FOR PARTICIPATION IN IRIS_2012 POST-TEST BENCHMARK ANALYSES

Prepared for
Nuclear Energy Agency
Organization for Economic Cooperation and Development
Committee on the Safety of Nuclear Reactors

Prepared by
ANATECH Corp., San Diego, CA
Randy J. James
Liping Zhang, PhD

ANA-R-12-0886
September 2012
1. INTRODUCTION

This report summarizes the work performed by ANATECH in association with the IRIS 2012 Post-Test Benchmark Analyses Round Robin organized by the Committee on the Safety of Nuclear Installations (CSNI) for the Nuclear Energy Agency (NEA) of the Organization for Economic Cooperation and Development (OECD). The current activity is a follow-on effort to the IRIS 2010 program for “Improving Robustness Assessment of Structures Impacted by Missiles”. In the 2010 program, pre-test predictions for impact tests on reinforced concrete slabs were performed by a variety of participants. ANATECH was not aware of this program and thus did not participate. ANATECH was contacted by the organizers of the 2012 project and invited to participate in the post-test program. The stated goals of this post-test effort are: 1) to give the opportunity to IRIS 2010 participants (+ new comers if any) to update and improve the simulations with the knowledge of the tests results and with the experience gained by one’s own 1st computation and by others’ computations using a single set of material properties; 2) to give the opportunity to each participants to develop, test and share the means and tools for alternative approaches (“simplified models”); and 3) to gather all the results from the new simulations and all the proposals of simplified models and to issue a new set of recommendations.

Since ANATECH did not participate in the pre-test predictions, simulations of the tests were first performed using our “standard” modeling procedures for impact analyses. For these types of impact tests, the response of the test fixture restraining the specimen relative to the analytical boundary conditions applied can have an effect on the computed results because these boundary conditions are in relatively close proximity to the loading. These initial ANATECH “Pre-Test” simulations assumed simple roller support conditions at the boundaries of the slab, which appears to be the basic intent of the test fixture configuration. These results are discussed in Section 7 Parameter Sensitivity along with other parameter variations. For the “Post-Test” analyses, the boundary conditions were modified to better capture the test apparatus response by incorporating a “bearing frame” and benchmarking the calculated reaction forces to the measured reaction forces for the VTT bending mode test. The intent of this approach is to minimize the effects of the boundary conditions on the analytical response and concentrate on the structural response of the slab to the impact loading. These “Post-Test” simulation results are discussed in Section 5 for the VTT Bending Mode Tests and in Section 6 for the VTT Punching Mode Tests. These results are also the data supplied in the Excel Template files requested by the organizers.

2. MODELING METHODS

2.1 Analysis Software

These analyses use the ANACAP-U concrete constitutive model [1] developed by ANATECH Corp. This model was developed as an implicit formulation for use with the ABAQUS/Standard general purpose finite element program [2]. Following the events of September 11, 2001, this concrete model was used in an extensive EPRI research program to assess the performance of structures at nuclear plants for impact of a large commercial aircraft [3,4,5,6]. During this time, the ANACAP-U concrete material model was also converted for use in explicit dynamics formulations, such as ABAQUS/Explicit [7], and tested against a variety of test data for impulsive loading on reinforced concrete [8,9,10]. The results of this EPRI research effort were
incorporated into the structural methods section of a methodology report [11] developed by Nuclear Energy Institute (NEI) for assessing aircraft impact at nuclear plants for new regulations being imposed by U. S. Nuclear Regulatory Commission (USNRC) for new plant designs to be constructed in the U. S. The NEI methodology guidance has been accepted by USNRC in Regulatory Guide 1.217 [12] for use in aircraft impact assessments required under 10 CFR 50.150. The ANACAP-U concrete material model has also been incorporated into the TeraGrande explicit dynamics software [13] developed by ANATECH and used to assess aircraft impact for new nuclear plant designs. One example with some reported preliminary work that is publically available is the small modular reactor under development by NuScale Power [14].

For the current study described herein, the static analyses to simulate concrete cylinder tests are performed using the ANACAP-U concrete material model coupled with the ABAQUS/Standard general purpose finite element software. For the VTT and Meppen impact tests, the analyses are performed with the ANACAP-U concrete material model incorporated into the TeraGrande explicit dynamics finite element software. Due to the impulsive nature of the loading, where the duration of the load is small compared to the fundamental period of the structure, and the need to include the impacting missile in the analyses, the finite element method using an explicit dynamics formulation is best suited to establish the structural performance. However, it is difficult to establish static equilibrium using explicit dynamics procedures, and thus the static tests are simulated with the implicit version of the concrete model.

2.2 Concrete Material Modeling

The behavior of concrete is highly nonlinear with a small tensile strength, shear performance that depends on crack widths, and compressive plasticity. The main components of the concrete model for these analyses are tensile cracking, post-cracking shear performance, and compressive yielding when the compressive strength is reached. A summary description of the modeling for concrete behavior used in this software is described below.

Tensile cracking in the concrete is governed by the magnitude of the load in the directions of principal strain. Cracks are assumed to form perpendicular to the directions of largest tensile strains. Multiple cracks are allowed to form at each material point, but they are constrained to be mutually orthogonal. If cracking occurs, the normal stress across the crack is reduced to zero and the distribution of stresses around the crack is recalculated. This allows stress redistribution and load transfer to reinforcement or other load paths in the structure. For the explicit formulation, the normal stress across a crack is gradually reduced to zero over increasing normal strains up to about 5 times the fracture strain. This is generally referred to as a tension stiffening model accounting for the time needed to crack completely between the integration points. Once a crack forms, the direction of the crack remains fixed and can never heal. However, a crack can close, resist compression and shear, and re-open under load reversals. The cracking criterion is based on an interaction of both stress and strain as illustrated in Figure 2-1. The model predicts cracking when the generalized (principal) stress and strain state exceeds the limit state shown. Thus, biaxial and triaxial stress states are treated consistently with uniaxial conditions, but the associated cracking will now occur at a slightly higher stress and slightly lower strain. Split cracking, for example near a free edge under high compressive stress, occurs at near zero stress and a tensile strain approximately twice that of uniaxial tensile cracking.
The surfaces of cracks that develop due to tensile stress in concrete are usually rough and irregular. When a shear force is applied along a crack surface, tangential sliding occurs and this causes displacements normal to the crack surface to develop as the crack surfaces ride up on each other. When this normal displacement is restrained by reinforcement crossing the crack, tensile stresses will develop in the steel bars, which will then induce compressive stresses across the crack in the concrete. The resistance to sliding is provided by the frictional force generated by the compressive stress across the crack. The crack width is the primary variable affecting this mechanism of shear transfer. Smaller crack widths correspond to greater shear stiffness and strength. Aggregate type and size, reinforcement design, and concrete strength are other important factors. In order to account for the effect of cracking on shear stiffness, a reduced shear modulus is retained in the stress-strain matrix when a crack forms. The shear modulii in the plane of the crack are immediately reduced by 60% when a crack forms. The shear stiffness is further reduced using a hyperbolic variation with the opening strain normal to the crack, as illustrated in Figure 2-2.
Perhaps the most important feature of concrete modeling is the ability to capture the shear capacity in cracked concrete. The ANACAP model is equipped with a shear-shedding feature to model the shear stress capacity across an open crack. The shear retention model reduces the incremental shear modulus across an open crack as discussed above. The shear stress capacity for an open crack is a function of the crack opening strain, as illustrated in Figure 2-3. The shear-shedding feature reduces the shear stresses previously supported across an open crack if the crack continues to open. Recall that cracks form in the principal strain directions so that, in general, there is no shear across a crack when it first opens. However, continued loading resulting in shear deformations will be carried in shear across the crack if possible.

![Figure 2-3. Example of Shear Stress Capacity Across Open Cracks](image)

In the compression regime, the continuous stress-strain curve is defined from uniaxial test data, which is then generalized to multi-axial stress/strain states using the uniaxial equivalence of the multi-axial state, namely, the effective stress and the effective strain. The uniaxial behavior is generalized to multi-axial behavior, within the analytical framework of isotropic hardening plasticity formulation, using a Drucker-Prager surface to represent the loading surface under multi-axial compression. In this formulation, the loading surface is a function of the hydrostatic pressure, the second invariant of the deviatoric stress tensor, and the yield strength. This type of formulation incorporates the effects of low to moderate confinement stress levels, which typifies the behavior of civil structures. These relations allow for linear behavior for compressive stresses below about 50% of the compressive strength, and then strain hardening behavior until the compressive strength is reached, as illustrated in Figure 2-4. The ANACAP material model is not currently formulated to account for the effects of highly confined concrete. For these type situations, for example, impact loading on pre-stressed concrete containment structures, an elastic-perfectly plastic model in compression is generally employed to capture the increased compressive strain that can be accommodated before compressive failure.
Typically, the steel reinforcing bars are modeled as sub-elements embedded within the concrete elements at the appropriate locations. The stress and stiffness due to the rebar sub-elements are superimposed on the concrete element in which the rebar resides. The strain in the rebar is determined from the parent concrete element at the location of the rebar and transformed to be in the direction along the segment of the rebar at that location. Yielding in the rebar material is treated using the classical J2 or Von Mises plasticity formulation with isotropic hardening. This formulation uses the effective stress and effective strain for defining increasing yield stress with plastic strain and assumes linear unloading. Reinforcement steel can fail in a variety of ways, reaching the strain ductility limit in tension, buckling under compression, experiencing excessive shear stress, or due to bond slip and loss of anchorage with the concrete. In addition, large stress and strain concentrations can develop in the rebar where the concrete has cracked. While rebar steel shows ductility limits around 12% in tensile tests, a tensile ductility limit around 5% is generally used in failure analyses of large structures because the mesh refinement is typically not adequate to capture the high strain concentration at the discontinuity of the crack. The data supplied by the organizers is used to construct the stress-strain relation for the reinforcement steel without any rate effects. Note that a rebar failure is enforced when plastic strain reaches 5%.

The steel components for the missile are modeled using plate bending elements. The material model for these components also employs the J2 or Von Mises plasticity formulation with isotropic hardening. Stress-Strain relations for the steel material are used based on yield stress, ultimate strength, and uniaxial elongation. Self-contact is included but ignoring the thickness of the shells.
2.4 Material Properties

The benchmark analyses were performed using the material data in the EXCEL files and related documents provided by the organizers. In cases where information is lacking, material properties are determined from visual observation or scaling of the plots provided by the organizers. Table 2-1 summarizes the material properties used for these analyses.

For the ANACAP-U concrete model, the fracture strain is the input variable for use in determining crack initiation. This is defined as the concrete tensile strength divided by modulus as a uniaxial value. In all the dynamic analyses, the tensile strength of concrete is calculated from the relation \( f_t = 1.7 \times (f'_c)^{2/3} \) developed by Raphael [15], where \( f'_c \) is the compressive strength of concrete. If test data for the concrete modulus is not available, it is calculated using the standard ACI formula [16], \( E = 57000 \sqrt{f'_c} \), based on the compressive strength. Note that these relations use English units (psi) for concrete compressive strength.
Table 2-1. Material Properties Used in Explicit Dynamic Analyses

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Units</th>
<th>VTT-Bending</th>
<th>VTT-Punching</th>
<th>Meppen II-4</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Concrete</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Comp. Strength</td>
<td>MPa</td>
<td>55.2</td>
<td>60.0</td>
<td>37.2</td>
</tr>
<tr>
<td>Modulus</td>
<td>GPa</td>
<td>30.0</td>
<td>30.0</td>
<td>29.1</td>
</tr>
<tr>
<td>Tensile Strength</td>
<td>MPa</td>
<td>4.7</td>
<td>5.0</td>
<td>3.6</td>
</tr>
<tr>
<td>Fracture Strain</td>
<td>x E-6</td>
<td>156.4</td>
<td>165.3</td>
<td>124.1</td>
</tr>
<tr>
<td>Poisson’s Ratio</td>
<td></td>
<td>0.2</td>
<td>0.2</td>
<td>0.22</td>
</tr>
<tr>
<td><strong>Reinforcement</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Modulus</td>
<td>GPa</td>
<td>200</td>
<td>200</td>
<td>200</td>
</tr>
<tr>
<td>Yield Stress</td>
<td>MPa</td>
<td>600</td>
<td>540</td>
<td>430</td>
</tr>
<tr>
<td>Tensile Strength</td>
<td>MPa</td>
<td>715</td>
<td>605.3</td>
<td>620</td>
</tr>
<tr>
<td>Elongation</td>
<td>%</td>
<td>3.0</td>
<td>11.4</td>
<td>3.0</td>
</tr>
<tr>
<td><strong>Missile</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Modulus</td>
<td>GPa</td>
<td>175.4</td>
<td>200</td>
<td>210</td>
</tr>
<tr>
<td>Yield Stress</td>
<td>MPa</td>
<td>350.7 (@0.2%)</td>
<td>314.5</td>
<td>413</td>
</tr>
<tr>
<td>Tensile Strength</td>
<td>MPa</td>
<td>633</td>
<td>413.8</td>
<td>607</td>
</tr>
<tr>
<td>Elongation</td>
<td>%</td>
<td>45</td>
<td>20</td>
<td>25</td>
</tr>
<tr>
<td>Poisson’s Ratio</td>
<td></td>
<td>0.3</td>
<td>0.3</td>
<td>0.3</td>
</tr>
</tbody>
</table>

2.5 Structural Failure Criteria

For the reinforced concrete, material limit states, such as cracking in tension, post cracking shear capacity, and compressive plasticity, are included in the material modeling for the concrete. Structural failure occurs either when excessive tensile loads cause rebars to rupture, or when the shear forces across a section exceed the shear capacity. Under bending loads, concrete crushing may develop on the compressive side, but failure of the section can still be attributed to rebar rupture on the tension side of the section. Section shear failure develops when sufficient cracking extends across a section such that the shear capacity of the concrete is reduced below that needed to support the shear demands. Section shear failure can also occur when compressive struts develop due to arch action within a member and initiate concrete failure in compression, which then also rapidly degrades the shear capacity of the section.

In addition to the material limit states imposed on the concrete through the concrete material model, element deletion criteria are activated for concrete elements. Concrete elements are removed from the calculation when there are two or more cracks at any material point and the associated strains are greater than 5% in tension or 10% in compression or 20% shear component. At these levels of strain the concrete material can no longer carry any load.

The rupture strain for reinforcing bars is based on the elongation limit of the material, factored to account for strain concentration factors that are not captured by the finite element modeling, which is based on smeared cracking. In reinforced concrete structures, cracking generally
develops in a series of discrete cracks, and the strain in the rebar is intensified at the intersection of these discrete cracks. From previous experience with similar modeling, this strain concentration factor has a median value of about 2.0, and the calculated strain at which rebar rupture can occur is generally taken to be 5% for the modeling methodology employed here. Here, any rebar segment reaching 5% plastic strain is removed from the model to simulate rupture of the bar.

The failure criteria for the steel missile materials is set relatively high so that the accordion folding deformation will develop rather than the elements being removed from the analysis.
3. CONCRETE CYLINDER TESTS

3.1 Brazilian Test - Modeling and Results

As shown in Figure 3-1, a 3D half-length symmetric model is developed where the concrete cylinder, modeled with 8-node solid elements, is placed between two rigid surfaces. The bottom rigid surface is fixed and the top platen is pushed down to simulate the Brazilian test. The point loading for this geometry generally causes a high stress concentration in the element under the applied load. To simulate the local deformation that occurs in the test as the plates are loaded, the 2 elements immediately under the load concentration on both tangent points are treated as linear response. The compressive strength, Young’s modulus and Poisson’s ratio for the concrete cylinder are 60 MPa, 29429 MPa and 0.2 respectively.

The tensile strength of the concrete is provided as 4.04 MPa. In the ANACAP-U concrete model, the fracture strain, defined as the tensile strength divided by the Modulus, is the input parameter establishing the crack initiation criteria as discussed in Section 2. For the Brazilian test, the commonly used formula to compute tensile strength from the test is

\[
\sigma_t = \frac{2P}{\pi DL}
\]

where \( P \) is the load at failure, \( D \) the diameter of the cylinder and \( L \) its length. This formula, however, has known inadequacies, mainly because it is based on plane stress assumptions and linear response up to the point of collapse. A modified equation can be found in the literature [17], which gives the effective tensile stress as

\[
\sigma_e = \frac{2P}{\pi DL} (1 + 3\nu)
\]

where \( \nu \) is the Poisson’s ratio. For a poisson’s ratio of .2, this represents a correction factor of 1.6 on the tensile strength calculated from Equation 1. In addition, other researchers [18] have determined that the result for tensile strength obtained from the Brazilian test is sensitive to the length to diameter ratio of the specimen. They propose a specimen shape correction factor of \([0.2621(L/D) +1]\) on the tensile strength calculated from equation 1, which for our geometry is a factor of 1.52.

It is noted that, fortunately, for analytical modeling of reinforced concrete for extensive damage, the tensile strength or fracture strain of concrete is not a critical input parameter because failure of structural components happens at loads well past the cracking strength of concrete. The more critical modeling requirements are load transfer to reinforcement, compressive strength and shear performance and capacity after cracking.

For this simulation, two analyses are performed: 1) with the input fracture strain based on the 4.04 MPa tensile strength provided, and 2) with the fracture strain based on 4.04*(1+3\( \nu \)) = 6.46 Mpa, which should be more consistent with the 3D analytical modeling assumptions. For case 1, the fracture strain is input as 137.3E-6 and for Case 2 the fracture strain is input as 219.6E-6. Figure 3-2 shows the contours of maximum principal strains after failure viewed from the free surface side for Case 2. Figure 3-3 provides a plot of applied force versus the strain at the center of the specimen for the 2 cases. The cylinder can take approximately 179 kN load before it splits...
apart at the center for the assumed tensile strength of 4.04 MPa and about 257 kN for failure with an assumed tensile strength of 6.46 MPa.

Figure 3-1. Finite Element Model, Brazilian Test

Figure 3-2. Maximum Principal Strain Contours at Failure, Brazilian Test
3.2 Uniaxial and Tri-axial Tests - Modeling and Results

Finite element model for uniaxial and tri-axial tests is shown in Figure 3-4 where a half-symmetric concrete cylinder is placed between two rigid surfaces. The bottom rigid surface is fixed and the top one is pushed down. Symmetry boundary conditions (fixed displacements normal to the cut surface) are applied at the symmetry plane. The concrete cylinder is modeled with 8-node solid elements. Depending on different specimens, the compressive strength varies from 66.93 MPa to 69.0 MPa. Young’s modulus and Poisson’s ratio are about 29.7 GPa and 0.22 respectively, with very little change between specimens. For this simulation, the given tensile strength is 4.04 MPa.

The existing ANATECH concrete constitutive model is used to simulate the uniaxial test. Figure 3-5 shows the load vs. global vertical strain for both experiment and analysis. The global vertical strain for the analysis is computed from the vertical displacement of the top rigid surface divided by the original height of the specimen. The only adjustment made to the concrete material model is setting the value of compressive strain where the compressive strength is reached. The default value in the concrete model is .0018, whereas the test data shows .005. The ANACAP model which was developed based on other test data, shows higher residual capacities than the uni-axial test data provided. No adjustments to the ANACAP concrete model are made for the impact simulations.

As mentioned previously, the ANATECH concrete constitutive model currently does not have a formulation for high confinement effects. The model is modified as a first pass at a confinement model to simulate the triaxial tests. The failure surface, as well as the strain value at onset of crushing, is modified as function of confinement pressure. Figure 3-6 shows load vs. global vertical strain for different confinement pressures. It is noted that this confinement model is currently under development and this effect is not included in any of the post-test analyses.
For uniaxial test, failure is initiated by split cracks. For triaxial tests, failures are more likely initiated by shear. As the vertical load increases, the compressive stress in the vertical direction increases and the compressive stress in the hoop direction decreases, especially at the mid-section near the surface. Radial stress near the surface basically remains unchanged and equal to the confinement pressure. Using Mohr circle concept, the increasing difference in vertical and hoop stresses will eventually cause catastrophic shear failure. The X-shaped shear failure planes tend to start at the weakest spot in the mid-section near the surface since the real specimen is never exactly axisymmetric or homogeneous. Once shear crack initiates, it quickly propagates causing final catastrophic failure. The finite element model is perfectly symmetric and homogeneous. As an illustration for this mechanism that leads to shear failure of the specimen, a demonstration analysis was performed, and the contour of maximum principal strain is plotted in Figure 3-7. In this demonstration analysis, a weak spot is seeded in two elements at the surface where compressive strength for these two elements is reduced by about 15%. As load increases, X-shaped failure planes develop. Since our concrete constitutive model requires that cracks at any spot be orthogonal to each other, shear cracks in our simulation initiate in a 45° pattern, leading to failure planes that are roughly 45° with respect to the vertical. The seeded location was moved to different locations around the circumference, including right at the symmetry plane, and the shear failure initiates where ever the weakest spot is located.

![Finite Element Model, Uniaxial and Tri-axial Test](image)

**Figure 3-4.** Finite Element Model, Uniaxial and Tri-axial Test
Figure 3-5. Load vs. Global Vertical Strain, Uniaxial Test

Figure 3-6. Load vs. Global Vertical Strain, Tri-axial Tests
Figure 3-7. Illustration of Shear Failure Pattern, Tri-axial Tests
4. MEPPEN II-4 TEST

4.1 Modeling Description

The 3D model for Meppen II-4 test is shown in Figure 4-1 where no simplification was made to take advantage of symmetry of the configuration. The concrete slab, 6.5 x 6.0 x 0.7m, is modeled with 8-node solid elements. The element size near the center of impact is about 36.8 x 37.5 x 43.8mm. The missile is mostly modeled with 4-node shell elements except for the end plate which is modeled with solid elements. The concrete slab is simply supported at 48 counter bearings at the back face. Figure 4-2 illustrates the reinforcement included in the model.

For the Meppen test, the recommended material property for the missile is a yield stress of about 275 MPa and tensile strength of about 405 MPa. As pointed out by some teams in the pre-test analyses, the rate effect is of primary importance to obtain correct crushed length of the missile because of the high strain rates and large strains that develop in the accordion type deformation. Therefore, following the lead of these researches in considering strain rate effects, the yield stress and tensile strength are increased to 413 MPa and 607 MPa, respectively, for the missile in the Meppen simulation to account for rate effects in an average sense.

![Figure 4-1. Finite Element Model, Meppen II-4 Test](image-url)
4.2 Results Discussion

The 1016 kg missile strikes the concrete slab at an initial velocity of 247.7 m/s. The velocity at the tail of the missile is shown in Figure 4-3. Upon impact, missile velocity rapidly decreases and both experiment and analysis results show that the missile is basically arrested by about 35 ms. The displacement histories at sensors W3~W6 on the back face are shown in Figure 4-4. Although the computed peak displacement is in good agreement with experiment measurement, some difference in final residual displacement is indicated. It is noted that the displacement under the point of impact for the simulation is significantly larger than for the points plotted. This implies a contribution from a punching mode of deformation, so the simulation has damage concentrated more around the immediate area of the impact rather than at the locations where measurements were taken. Figure 4-5 shows the histories of reaction forces at load cells K1~K4 where the computed peak reaction forces are in relatively good agreement with the experiment results. It is noted that in the test, the load cells have some axial stiffness and deformation, while, in our simulation, displacements at those 48 counter bearings are assumed to be completely fixed, thus leading to larger tensile reaction forces. Maximum principal strains in concrete are shown in Figures 4-6 where the contour limit, the red color region, is set at 2% to highlight the areas with more extensive cracking. The slab has heavy damage near the center as well as cracks that radiate from the center. Figure 4-7 shows the plastic strains in main bars at the end of the analysis. Some main bars are ruptured from the impact based on the imposed 5% plastic strain as the bar rupture criteria. Again, the simulation shows significant damage in the slab, but sufficient resistance to just stop the missile without perforation. The deformed shape of the missile after impact is shown in Figure 4-8. The final remaining length of the missile is approximately 1400mm which is smaller than the experimental value of 2090~2180mm. Some of this difference can be attributed to the contact modeling used which does not account for the thickness of the shell in the self-contact.

![Figure 4-2. Illustration of Reinforcement Modeling, Meppen II-4 Test](image-url)
Figure 4-3. History of Velocity at Tail of Missile, Meppen II-4 Test

Figure 4-4. History of Displacement at Rear Face of Slab, W3~W6, Meppen II-4 Test

Figure 4-5. History of Reaction Force at Load Cells K1~K4, Meppen II-4 Test
Figure 4-6. Damage in Concrete Slab, View from Back Side, Meppen II-4 Test

Figure 4-7. Plastic Strains in Main Bars, Time = 0.05 Seconds, Meppen II-4 Test
Figure 4-8. Missile Deformed Shape After Impact, Meppen II-4 Test
5. VTT BENDING MODE TEST

5.1 Modeling Description

A 3D model is developed and shown in Figure 5-1. The concrete slab, 2.1 x 2.1 x 0.15m, is modeled with 8-node solid elements. The element size near the center of impact is about 18.8 x 16.4 x 15mm. The 50.5 kg pipe missile is modeled with 4-node shell elements for the pipe and end-cap and 8-node solid elements for the steel plate at the tail. As mentioned in the Introduction, for these Post-Test simulations, the reaction frame and support used in the test is modeled with a simplified support system for the analysis and calibrated to the measured reaction forces to minimize the effects and uncertainty of the applied boundary conditions on the slab response. The concrete slab is clamped between two 2”-thick bearing supports to simulate the contact interaction between the slab and the supporting frames which are not modeled in current analysis. Only compressive load is transmitted between the slab and the bearing supports. Displacement boundary conditions are enforced at the outside surfaces of the bearing supports. Elastic material is used for the bearing plates and the stiffness is adjusted in such a way that the total reaction force roughly matches the experimental result. This boundary condition is then used directly for the Punching Mode Test simulation. The reinforcement included in the modeling is illustrated in Figure 5-2 where the bars are treated as truss-like sub-elements embedded in the concrete solid elements. Note that the missile impact location is slightly off center by 20mm (horizontal) x 9mm (vertical).

Figure 5-1. Finite Element Model, VTT Flexural Mode Test
5.2 Results Discussion

The 50.5 kg “soft” pipe missile strikes the concrete slab at an initial velocity of 110.15 m/s. Upon impact, the front part of the missile crumples up and the missile velocity rapidly decreases as shown in Figure 5-3, which plots the history of the velocity at the tail of the missile. The velocity plot indicates that some axial ringing deformation develops in the missile. It can be observed that the missile is basically stopped by 0.018 seconds and then starts to bounce back. The computed duration of the impact, 0.018 seconds, is in good agreement with the estimate from the experiment. Figure 5-4 shows the history of total reaction force and the corresponding impulse. The total reaction force is used to calibrate the effective stiffness of the support frame for the boundary conditions. The displacement histories at two sensors on the back side of the slab are shown in Figure 5-5. This shows that the computed displacements are in good agreement with the experiment measurement. The histories of 45° diagonal strains on the front face of the concrete for 2 locations are compared to test measurements in Figure 5-6. These are also in good agreement with the measured data. It is noted that the analyses show that the maximum strain at these locations decrease with distance from the center of the slab (R2>R1>R3), while the measured data shows the largest value at R1 with R2 and R3 being almost equal, although the differences in magnitudes are relatively small. The calculated damage to the slab is illustrated in Figure 5-7 for maximum principal strains in the concrete and in Figure 5-8 for plastic strains in rebar. The cut-off limit in the maximum principal strain plot is set at 2% to highlight the areas with more extensive cracking damage. When reaching maximum displacement, the slab develops significant cracking on its back face, and these cracks close as the slab rebounds. Some reinforcement sustains yielding and plastic deformation from the impact, but no bars are ruptured in the simulation. Comparison of the calculated rebar strains with the measured rebar strains for 2 locations is provided in Figure 5-9. This plot indicates that the response of the slab is captured by the simulation. The peak values of rebar strain are very sensitive to the exact locations where the measurement is taken and where the point is chosen.
from the simulation because the strain is highly affected by local concentrations due to cracking. It is noted that the contour plot of accumulated plastic strain shows peak plastic strains close to 5% in the bending reinforcement, which indicates that the simulation is indeed calculating significant rebar strains and, in fact, may be conservative relative to the measured data. The deformed shape of the missile after impact is shown in Figure 5-10. The final remaining length of the missile is computed to be approximately 1180mm which is in reasonably good agreement with the experimental value of 1130~1150mm.

![Figure 5-3. Velocity History at Tail of Missile, VTT Flexural Mode Test](image)

![Figure 5-4. Histories of Total Reaction Force and Impulse, VTT Flexural Mode Test](image)
Figure 5-5. Displacement Histories on Back Side of Slab, VTT Flexural Mode Test

Figure 5-6. Strain Histories on Front Side of Slab, VTT Flexural Mode Test
Figure 5-7. Max. Principal Strains, View from Back Side, VTT Flexural Mode Test

Figure 5-8. Plastic Strain in Rebars, Time = 0.10 Seconds, VTT Flexural Mode Test
Figure 5-9. Comparison of Measured and Computed Rebar Strains

Figure 5-10. Missile Deformed Shape After Impact, VTT Flexural Mode Test
6. VTT PUNCHING MODE TEST

6.1 Modeling Description

Although the configuration for this test has two axes of symmetry, no simplification was applied as is shown in Figure 6-1 for the 3D model. The concrete slab, 2.1 x 2.1 x 0.25m, is modeled with 8-node solid elements. The element size near the center of impact is about 18.8 x 16.4 x 16.7mm. The missile is also modeled with 8-node solid elements. The aluminum pipe at the tail of the missile is ignored. The density for the light weight concrete inside the steel casing is adjusted so that the total mass of the missile is 47.38 kg. As in the case for the bending mode test, the concrete slab is clamped between two bearing supports to simulate the contact interaction between the slab and the supporting frames. The stiffness for the bearing supports is the same as that used for the bending mode simulation (to meet the requirement by the organizer that “a single set of boundary conditions” be used). Figure 6-2 illustrates the reinforcement included in the model.

![Figure 6-1. Finite Element Model, VTT Punching Mode Test](image)

Figure 6-1. Finite Element Model, VTT Punching Mode Test
6.2 Results Discussion

The 47.38 kg “hard” missile strikes the concrete slab at an initial velocity of 135.85 m/s. Figure 6-3 shows the velocity of the missile. The residual velocity is computed to be 35.1 m/s, which is in good agreement with the experimental measurement. The missile punches through the slab in about 4 ms. Damage to the back side of the slab is shown in Figure 6-4 where heavy scabbing and cracking is evident. Displacement histories at two transducers on the front face of the slab are plotted in Figure 6-5. The computed slab displacement for the punching mode is somewhat larger than that measured in the experiment although the overall peak displacement is still quite small, just about 7 mm. It is noted that the test shows higher displacements during the rebound after the perforation while the calculation shows the peak displacement occurs during the initial deformation with smaller peaks during the free vibration after perforation. Figure 6-6 shows the histories of the total reaction force and the corresponding impulse for the simulation and the test data. The shape of the computed reaction force suggests that the assumed simplified supports might be too stiff for this case, meaning there could be some plastic response or damping in the actual supporting frame system that is not captured in the current analysis. The impulse from the calculated reaction forces is consistent with the test performance where the slab cannot resist the total applied impulse. It is noted that the impulse for the measured reaction forces eventually exceeds the initial momentum of the missile. The damage to rebar is shown in Figure 6-7 where the computed result shows more bar ruptures than what is observed in experiment. Again, comparisons of calculated rebar strains to measured strains are very sensitive to the location of the gauge and the point selected in the analysis relative to the cracking, especially in the area where rebar rupture occurs. However, the extent of calculated damage again appears conservative relative to the test, although the exit velocity is well simulated. After impact, the missile has noticeable plastic deformation at the front part, as shown in Figure 6-8, which is in agreement with experiment observation.
Figure 6-3. History of Velocity of Missile, VTT Punching Mode Test

Figure 6-4. Damage to Back Side of Slab, VTT Punching Mode Test

Figure 6-5. History of Displacement at Selected Transducers, VTT Punching Mode Test
Figure 6-6. Total Reaction Force And Impulse, VTT Punching Mode Test

Figure 6-7. Plastic Strains in Rebars, Time = 0.015 Seconds, VTT Punching Mode Test

Figure 6-8. Missile Deformed Shape After Impact, VTT Punching Mode Test
7. PARAMETER SENSITIVITY

Results from a few analyses with parameter variations are provided in this section as a first look at sensitivity in the analytical modeling. These parameter variations include boundary conditions for the reaction frame, a multiplier on concrete compressive strength for rate effects, and mesh refinement.

7.1 “Pre-Test” Conditions

As a “pre-test” type simulation, the VTT Bending Mode test was first simulated with a roller support condition and concrete modeling generally used at ANATECH for impulsive loading analyses on reinforced concrete. In lieu of a model for localized high rate and confining effects that can develop under the impulsive loading conditions, a 25% increase in the compressive strength is used to account for loading rate and confinement effects. In addition, if the impactor is included in the analysis (rather than an equivalent pressure loading), experience is that using the elastic-perfectly plastic model for compressive yielding in the concrete provides a better simulation for the increase in compressive strain capacity due to the confinement effects under the impactor. For the “pre-test” simulation, the line of nodes on both sides of the slab in contact with the bearing in the test fixture is fixed in the direction transverse to the slab as a fixed roller support. This ignores any flexibility in the reaction frame used in the test to support the target slab. This analysis was performed both with the concrete strength provided and with the 1.25 multiplier on concrete strength. Figure 7-1 provides the comparison of reaction forces for these analyses along with the simplified support results from Section 5 and the test data. Figure 7-2 provides the comparisons for displacements at Sensor W1. These plots indicate that the roller boundary conditions provide relatively good results, except the free vibration response has a higher frequency when the stiffness of the frame support is ignored. Figure 7-3 provides contour plot for accumulated plastic strain in the bending reinforcement for the “Pre-Test” conditions with the 1.25 factor on compressive strength. This plot shows that the damage in the slab is very similar to the results reported in Section 5 and discussed relative to the test data. These plots also indicate that the 25% increase in concrete compressive strength is not significant in this case.

Figure 7-1. Comparison of Reaction Forces for “Pre-Test” Conditions
7.2 Mesh Refinement

It is noted that simulations for experimental tests generally use very refined meshing that is much more refined than can be accommodated in applications to large practical problems. As a sensitivity study for mesh sizing, the VTT Bending Mode test is also simulated with a coarser mesh, roughly 2:1 to that used for the reported results and also uniform over the extent of the slab. This coarser mesh is illustrated in Figure 7-4. The modeling used for the missile and for the boundary conditions are unchanged from the reported results. Comparisons of the calculated reaction forces for the coarser mesh modeling are compared to the finer mesh results and to the measured data in Figure 7-5. Comparisons of the calculated displacements for the coarser mesh modeling are compared to the finer mesh results and to the measured data in Figure 7-6. Figure
7-7 provides a contour plot for the accumulated plastic strain in the bending reinforcement for the coarser mesh modeling. This plot is a good indicator of the damage sustained by the slab since it is accumulated plastic strain. Comparing with the same plot for the finer mesh presented in Section 5, it is clear that the coarser mesh plot captures the same response as the finer mesh.

Figure 7-4. Coarser Mesh Model for Bending Test

Figure 7-5. Comparisons of Reaction Forces for Mesh Sensitivity
Figure 7-6. Comparisons of Displacements for Mesh Sensitivity

Figure 7-7. Accumulated Plastic Strain in Rebar for Coarser Mesh
REFERENCES


7. ABAQUS/Explicit, Version 6.x, Dassault Systemes Americas Corp, Waltham, MA.


Nonlinear Behaviour of Reinforced Concrete Slabs under Missile Impact Loading

Jianming Cao and Yuxin Liu
Candu Energy Inc., Mississauga, Canada, L5K 1B2

1. Introduction

This document summarises the work made by Candu Energy Inc. for the IRIS_2012 program. Three non-linear simulations are carried out using the explicit dynamic finite element software LS-DYNA_971 (explicit method) [1, 2]. Concrete simulation (IRSN-CNSC-VTT) [3] was conducted using constitutive laws and pre-known uniaxial and triaxial results from several tests to calibrate concrete damage model of LS-DYNA. The calibrated concrete model was applied for Punching Simulation (IRSN-CNSC-VTT-Punching Failure, P1) and Bending Simulation (IRSN-CNSC-VTT-Flexural Failure, B1), and comparison with the test results is performed.

The ultimate objective of the IRIS_2012 project is to update and improve the numerical simulation using LS-DYNA software to predict the response of reinforced concrete slabs impacted by soft (deformable) and hard (non-deformable) missiles with the knowledge of the test results and with the experience gained by IRIS_2010 program for structural modelling and analysis as well as design criteria for structures impacted by a missile. The work presented in this report focuses on calibrating the concrete damage model and then using the calibrated concrete damage model to simulate the concrete structures impacted by hard and soft missiles (P1&B1).

As with the IRIS_2010 program, LS-DYNA software was selected for conducting research of the simulations. Explicit analysis scheme was chosen in conducting the three simulations since the time step required for the analysis is very small and implicit analysis could lead to divergence and unstable calculations. The investigation that was carried out for this benchmark includes, developing the analyses models, performing sensitivity studies to decide on the key models parameters, applying the impact loadings, and sifting through the enormous output results.

2. Calibration of concrete damage model

For simulating the cylinder concrete specimens, the material properties are based on the VTT tested uniaxial and triaxial results provided by the IRIS_2012 committee, as shown in Table 1.

<table>
<thead>
<tr>
<th>Table 1 Concrete Property for Model Calibration</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density, $\rho$</td>
</tr>
<tr>
<td>Compressive Strength, $\sigma_c$</td>
</tr>
<tr>
<td>Young’s Modulus, $E$</td>
</tr>
<tr>
<td>Poisson Ratio, $\nu$</td>
</tr>
<tr>
<td>Aggregate Size, $A$</td>
</tr>
</tbody>
</table>

2.1. Results for a single LS-DYNA element model

To simulate the actual cylinder concrete specimens, a single LS-DYNA cubic element with material properties in Table 1 is considered first to compare the LS-DYNA concrete damage
models. To this end, the LS-DYNA damage modes, *MAT_84 and *MAT_159 were selected. For *MAT_159 model, the *MAT_CSCM_CONCRETE (159A) define some main parameters embedded in the LS-DYNA program, and the *MAT_CSCM (159B) allows the users to input their own parameters. The single element cubic model with 1 inch side length is applied to match IRSN-CNSC-VTT uniaxial and triaxial concrete test results. According to stress-stain relationship, as shown in Figure 1, *MAT_CSCM (159B) with different softening parameters compared to 159A is chosen in the following simulation due to smaller difference to the VTT test data. However, the current version of concrete *MAT_CSCM has up to 37 input parameters, with a minimum of 19 parameters that must be fit to data [4-5]. The stress-strain curve can be calibrated through those parameters.

![Stress-strain curve](image)

Figure 1 Calculated axial stress-strain curve of cubic specimen (unconfined)

### 2.2 Results for concrete specimens

Based on the analysis results shown in Figure 1, the LS-DYNA concrete damage model *MAT_CSCM (159B) is applied to simulate each VTT concrete cylinder specimen, Figure 2a, where the total element number from cylinder only is 768. The contact surface is modeled between each end caps and the concrete cylinder using fractional coefficients of 0.3, with contact type *CONTACT_CONSTRAINT_NODES_TO_SURFACE [4]. The cylinder is loaded in compression by applying a constant axial velocity to the layer of nodes along top of the top end cap. The bottom end cap is constrained from motion and rotation along the layer of end cap nodes. Figure 2b shows the stress distribution in a loading state. Calculated stress-strain relationship of cylinder surface under unconfined compression together with test result is shown in Figure 3a. Figure 3b shows the simulation results of different confined conditions.

![Concrete cylinder models](image)

Figure 2 Concrete cylinder models with end caps, before and after unconfined compression
3. Modelling of concrete slab specimens

3.1. Material input data for punching and bending simulation

For simulation of punching, the concrete material properties are from the tests reported for cylindrical test pieces (W-T-P1-1). Whilst, steel material properties for reinforcement rebar and missile are used based on the data provided in “IRSN-VTT Tests: punching tests” documents.

For simulation of bending, the concrete material properties are from the test reported for cylindrical test pieces (W-P-B1-1). The stainless steel material properties of the missile and the steel properties of the reinforcement bars and the missile are used based on the data provided in “IRSN-CNSC-VTT Tests: bending rupture tests” document.

Table 2 summarizes the concrete and steel parameters used in the simulations. The concrete material parameters (*MAT_CSCM) are input to the program based on the calibrated concrete damage model and the concrete material properties from cylinder test specimens.

<table>
<thead>
<tr>
<th>Target</th>
<th>Concrete</th>
<th>Rebar</th>
<th>Missile</th>
</tr>
</thead>
<tbody>
<tr>
<td>Punching simulation (P1)</td>
<td>ρ 2,261 kg/m³</td>
<td>ρ 7,850 kg/m³</td>
<td>ρsteel 7,700 kg/m³</td>
</tr>
<tr>
<td></td>
<td>‾c 67.60 MPa</td>
<td>y 540 MPa</td>
<td>ysteel 260 MPa</td>
</tr>
<tr>
<td></td>
<td>E 29.43 GPa</td>
<td>E 200 GPa</td>
<td>Esteel 200 GPa</td>
</tr>
<tr>
<td></td>
<td>ν 0.22</td>
<td></td>
<td>ρconcrete 1,567 kg/m³</td>
</tr>
<tr>
<td></td>
<td>A 19.0 mm</td>
<td>A</td>
<td>EConcrete 30 GPa</td>
</tr>
<tr>
<td>Bending simulation (B1)</td>
<td>ρ 2,261 kg/m³</td>
<td>ρ 7,850 kg/m³</td>
<td>ρstainless 8,000 kg/m³</td>
</tr>
<tr>
<td></td>
<td>‾c 62.7 MPa</td>
<td>y 600 MPa</td>
<td>yStainless 300 MPa</td>
</tr>
<tr>
<td></td>
<td>E 26.92 GPa</td>
<td>E 200 GPa</td>
<td>EStainless 200 GPa</td>
</tr>
<tr>
<td></td>
<td>ν 0.22</td>
<td></td>
<td>ρsteel 7,850 kg/m³</td>
</tr>
<tr>
<td></td>
<td>A 19.0 mm</td>
<td>A</td>
<td>ysteel 300 MPa</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>Esteel 200 GPa</td>
</tr>
</tbody>
</table>
3.2. Contact modelling

The LS-DYNA contact card used for the simulations (both punching and bending simulations) was *CONTACT_AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE. In both simulation, the missile represents the slave part and the target slab represents the master parts as well as the rebars. In punching simulation, perfect coupling is assumed between edges of the slab and the supporting frame, there is no contact card between the slab target and the supporting frame. The one-way contact is chosen as it is computationally more efficient since it has close to has half the cost of the two-way treatment.

The contact card *CONTACT_AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE instead of the perfect coupling method as used in IRIS_2010 is applied to link the slab target including the rebars with the supporting frame to improve the simulation.

3.3. Support structure modelling

Support structure members are assumed to behave in a perfectly elastic manner. As such, an elastic material (*MAT_ELASTIC) with material properties corresponding to steel is introduced for the support structure system.

3.4. Punching simulation

A 3-D finite element model is developed to simulate the reinforced concrete target slab with its full dimensions $2.1 \times 2.1 \times 0.25$ m as shown in Figure 4. The concrete cover is 20 mm from top and bottom sides. The supporting system is modelled explicitly using the arrangement provided in the project document and test results. Perfect coupling is assumed between edges of the slab and the supporting frame.

The model includes about 318,144 elements, among which total solid elements for concrete slab are 304,704 and total beam elements for reinforcement steel are 13,440. The concrete slab is modelled using constant stress SOLID (brick) element (ELFORM = 1) with mesh size $14.8 \times 14.8 \times 15.6$ mm (16 elements through the concrete wall thickness), which is same as IRIS_2010. Reinforcement rebar are modelled using BEAM elements (ELFORM =1) with a mesh size of 15 mm (Figure 5). The diameter of longitudinal horizontal and vertical rebars at the front and rear faces is 10 mm with a total number of rebars in each direction per each face 24 bars. The nodes of the rebars are coupled to the nodes of the concrete slab through *CONSTRAINED_LAGRANGE_IN_SOLID keyword.
The constitutive model used to model the concrete slab is *MAT_CSCM, which is calibrated based on concrete compressive test data, while the constitutive material model used to model the reinforcement rebars is *MAT_PIECEWISE_LINEAR_PLASTICITY, in which the rebar’s stress-strain curve is obtained from reinforcement test. The MAT_CSCM damage concrete model is used to model the concrete because 1) the model fits the concrete test data better; and 2) the model can be calibrated according to the test data. Concrete elements are removed from the calculation when element the stress damage index exceeds 0.99 (full damage is 1.0), and the maximum principal strain exceeds 5% by setting ERODE=1.05 in the concrete model. This erosion criterion is chosen based on comprehensive sensitivity analysis report by LS-DYNA [4].

3.5. Bending simulation

A 3-D finite element model is developed to simulate the reinforced concrete target slab with its full dimensions 2.1 × 2.1 × 0.15 m (Figure 6) with concrete cover 15 mm. The supporting system is modelled explicitly using the arrangement provided in the program document.

The model includes about 363,388 elements, among which total solid elements for concrete slab are 338,688 and total beam elements for reinforcement steel are 24,700. Concrete slab is modelled using constant stress SOLID (brick) element (ELFORM = 1) with mesh size 12.5 × 12.5 × 12.5 mm (12 elements through the concrete wall thickness). This mesh size is chosen based on comprehensive mesh discretization sensitivity analysis using four different mesh sizes. Reinforcement rebars are modelled using BEAM (Truss) elements (ELFORM =3) with a mesh size 18.3 mm (Figure 7). The diameter of longitudinal horizontal and vertical rebars at the front and rear faces is 6 mm with a total number of rebars in each direction per each face 38 bars. The transverse rebars of 4.41 mm diameter are modeled with total density of 50.10 cm²/m². The nodes of the longitudinal and transverse reinforcement rebars are coupled to the nodes of the concrete slab through *CONSTRAINED_LAGRANGE_IN_SOLID keyword.

The constitutive model used to model the concrete slab is *MAT_CSCM, while the constitutive material model used to model the reinforcement rebars is *MAT_PLASTIC_KINEMATIC. The MAT_CSCM damage material model is used to model the concrete because 1) the model fits the concrete test data better; and 2) the model can be calibrated according to the test data. Concrete elements are removed from the calculation when element stress damage index exceeds 0.99 (full damage is 1.0) and the maximum principal strain exceeds 5% by setting ERODE=1.05 in the concrete model. This erosion criteria is chosen based on comprehensive sensitivity analysis report from LS-DYNA.
4. Missiles modelling

4.1. Hard missile

A 3-D finite element model is developed for the whole missile (Figure 8). The model includes about 85,032 elements, among which solid elements for the solid steel nose, shell elements for the steel pipe and the rear plate, and solid elements for the light-weight concrete. The solid steel nose of the missile is modelled using SOLID (brick) element with a mesh size does not exceed 8.0 mm. The missile steel pipe and rear plate are modelled using SHELL elements with mesh size 8.0 mm. The light-weight concrete that is used to fill the missile pipe is modelled using SOLID (brick) elements. Perfect coupling is assumed between different parts of the missile. The constitutive material model used to model the steel parts of the missile is *MAT_PLASTIC_KINEMATIC, while the constitutive model used to model the concrete filling is *MAT_ELASTIC. Strain rate effect is not included since the strength of hard missile is not sensitive.

4.2. Soft missile

A 3-D finite element model is developed for the whole missile (Figure 9). The model includes about 118,920 elements, among which shell elements for the stainless steel pipe and the rear carbon steel pipe and plate, and beam elements for the screws connecting carbon steel pipe to the stainless steel pipe. The stainless steel parts of the missile as well as the rear plates and the carbon steel pipe are modelled using SHELL elements with mesh size of 4.0 mm. The carbon steel pipe is connected to the stainless steel pipe using 6 bolts along the length of the steel pipe every quarter of the perimeter. The bolts are modelled using BEAM (truss) elements. The constitutive material model used to model the stainless steel is *MAT PIECEWISE_LINEAR_PLASTICITY, in which the stress-strain curve is obtained from stainless steel test. The constitutive material model used to model the other parts of the missile is *MAT_PLASTIC_KINEMATIC except the stainless steel. Strain rate effect is not included in modelling the missile.

5. Simulation results

5.1 Response due to hard missile impact

The time step that is used in the calculation is \(3.68 \times 10^{-4}\) ms and the total elapsed running time is around 100 Hrs for a total simulation period of 100ms. At approximately 5.0 ms, the missile starts to rebound and it nearly loses the contact with the slab at 8.0ms. The displacement time history of the front face of the slab is shown in Figure 10 (left). Displacement comparison of IRIS_2010 and IRIS_2012 is listed in Figure 10 (right). During the simulation, the missile is found to be very rigid as it experiences very little shortening in its solid steel front nose (about
2.6 mm). The deformation and velocity of the missile nose are shown in Figure 11. The crack pattern of the target slab is shown in Figure 12. A middle section view of the whole system simulation under punching at 25ms is shown in Figure 13.

In the current simulation, the slab maximum displacement (23mm) is bigger than the test result from same position (4mm), but smaller than IRIS_2010 (70mm without erosion or 32mm with adding erosion). By employing an erosion criteria (ERODE=1.05), the target experiences a penetration of about 105 mm of the slab depth associated with spalling of diameter 800 mm in the front face and scabbing of diameter 764 mm in the rear face (Figure 12). A punching cone formation with a punching angle 42~45° can be observed from a section view (Figure 13). Unlike the test result, there is no missile residual velocity (Figure 11).

Figure 10 Displacement Time History of the Front Face (left), comparison of displacement of IRIS2010 and IRIS2012 (right)

Figure 11 Deformation (left) and velocity (right) of the missile during impact

Figure 12 Cracking pattern of 25 ms, front (left) and rear (right) of the slab
5.2 Bending simulation

The time step that was used in the calculation is $3.10 \times 10^{-4}$ ms and the total elapsed running time is around 120 Hrs for a period of 100ms. At around 17.5 ms, the missile reached its maximum deformation of 983.5 mm and starts to rebound around 20ms till the contact between target and missile completely diminishes. During the simulation, the missile showed to be very soft (crushable) as it experiences very large shortening in its stainless steel front nose. The final shorten of the missile in this simulation is 973 mm (Figure 14), which is pretty close to the bending test results of 971 mm. The crack patterns of the missile are shown in Figure 15.

On the other hand, the target experiences almost no damage in both the front face and the rear face, as shown in Figure 16. The displacement time history of the front face of the slab is shown in Figure 17 (left). Displacement comparison of IRIS_2010 and IRIS_2012 is listed in Figure 17 (right). The maximum deflection recorded at the centre of the rear face is about 16 mm, which is smaller than the test result of 28mm. For both IRIS_2010 and IRIS_2012, the target slab time history displacements are almost same.

Figure 14 Soft Missile Tail Displacement and Velocity
6. Observations and lessons learnt

This document summarises the main results of the LS-DYNA simulations conducted by Candu Energy Inc. for the IRIS_2012 benchmark program. In general, reasonable agreement with the known results of the concrete cylinder test; however the nonlinear performance of concrete damage model is sensitive to too many parameters. Calibrated LS-DYNA *MAT_CSCM damage model applied to punching model with improvement in the displacement of the slab and the deformation of the missile than IRIS_2010. By changing the boundary condition between the target slab and the supporting frame, deformation of the soft missile is almost same as the bending test results. However, the slab displacement in both punching and bending simulations are different with the punching and bending test results.

The following observations and lessons are learned from IRIS_2012 Assignment:

- Calibrated MAT_159 parameters applied to cylinder model shows good match with cylinder test results on both uniaxial unconfined and tri-axial confined conditions. For
uniaxial unconfined condition, the stress-strain curves are not smooth due to large deformation of concrete damage model.

- Calibrate concrete damage model *MAT_CSCM (159) with erosion option is better than enforced concrete model *MAT_WINFRITH_CONCRETE_REINFORCEMENT (84), which is used in IRIS_2010. However, the current version of concrete material model (MAT_159) has up to 37 input parameters, with a minimum of 19 parameters that must be fit during calibration. Calibrating the MAT_159 damage model is a challenge.

- The maximum displacement at given points from punching simulation (P1) is smaller than IRIS_2010, but still larger than IRIS_VTT punching test results.

- The shortening of the missile from bending simulation (B1) is almost same as IRIS_VTT bending test results. However, the displacements from bending simulation of the slab are smaller than those from the bending test.

- Changing constrained condition and/or contact method of target concrete and side channel/support affects simulation results. Finding a proper method/condition between target concrete and supporting system is another key issue of matching the test result.

- IRIS2012 punching simulation has not successfully match the behaviour of the missile which goes through the target concrete and has a residual velocity of 33.8mm/sec.

It is worthy to highlight a few issues as recommendations for future improvements:

- It is recommended to carry out more sensitivity studies on the key modelling parameters that affect the analysis results, such as material constitutive model, erosion criteria and type of contact.

- The transient analysis undertaken here takes extensive computational resources and as a result a very long time to execute, especially for complex large 3D model. It is recommended to develop an effective method to approach numerical simulation (i.e. analytical tools or numerical models with reduced number of degrees of freedom).

7. References


1. INTRODUCTION

The IRIS 2012 benchmark is devoted to compare various modelling of impact on reinforced concrete slabs. Its aim is to improve the methods used and develop guidance to assess the integrity of structures impacted by missiles.

The CEA and the European Commission (EC – JRC Ispra) are the owners of EUROPLEXUS, a general finite element code for fast transient analysis of structures. This software was jointly developed by the owners, EDF and other partners. EUROPLEXUS formulation, fully explicit, is well suited for fast transient calculation and this is the reason why it has been chosen for this benchmark.

The Global constitutive Law for Reinforced Concrete model [Koechlin 2007] developed by EDF, was chosen to perform IRIS 2010 benchmark for its simplicity to use and its robustness. This model devoted to shell elements represents the flexural behaviour of slabs. Concrete cracking is modelled through damage theory and inelastic strains through plasticity theory for shell. The yield criterion is an extension of [Johansen 1962] bending yield criterion and it takes into account membrane effects. Interaction diagrams (that give ultimate bending moment versus normal force), are calculated by
following EC2 prescriptions and with the supplementary hypothesis that steel rebars are elastic and perfectly plastic. The shear behaviour of the shell is supposed to be elastic.

Comparison with the experimental results showed that the calculation was conservative and over predicts displacements and steel deformations due to the fact that strain rate effects were not accounted for. For the punching test, the global approach was unable to give the residual velocity of the projectile after its penetration through the slab.

All these reasons conducted us to develop in EUROPLEXUS a new 3D model of concrete behaviour devoted to industrial purposes, which means that it must be robust, efficient and easy to use.

This new constitutive law, called DPDC (Damage Plastic Dynamic Concrete), has been elaborated and introduced in EUROPLEXUS in less than one year with the help of an internship. Therefore, some aspects are still missing, in particular, the introduction of rate effects and erosion of distorted elements. Nevertheless, this version was used to compute the flexural test.

After a short theoretical overview of the model, the calculation is described and comparison with test is discussed in the following paper.

2. THEORETICAL OVERVIEW OF THE CONCRETE MODEL

The model is inspired by the LS_DYNA Concrete Material Model 159 [Murray 2007]. It is an isotropic damage plastic model for concrete failure. The effective stresses (stresses of the undamaged material) are calculated with a plastic update. The initial damage threshold surface is identical to the shear yield surface and the damage model is driven by strain terms. The behaviour of concrete is elastic until the yield surface is reached. This kind of concrete model is widely used because plasticity and damage are simply coupled and mathematically leads to a well posed problem [Grassl 2006]. Furthermore, calibration procedure of parameters is relatively easy to deal with and it is also simple to change plastic or damage formulation.

2.1. Plasticity model

The plasticity model is formulated in a 3D framework with a pressure sensitive shear (failure) convex surface, a hardening cap to close the yield surface on the compression axis and a nonassociated plastic flow to control volume expansion.

The yield surface is described by means of the three stress invariants:

\[ J_1 = Tr(\sigma) = 3P \]
Any point of the yield surface is represented by $J_1$ the abscissa along the pressure axis, and the polar coordinates in the deviatoric plane: the radius of the yield surface $\sqrt{J'_2}$ and the Lode angle defined as follows:

$$\cos 3\theta = \frac{3\sqrt{3}J'_3}{2J'_2\sqrt{J'_2}}$$

(The yield surface exhibits symmetry of $2\pi/3$ around the pressure axis).

The yield function is the product of three functions:

$$f(J_1, J'_2, \theta, \kappa) = \sqrt{J'_2} - \mathcal{R}(J_1, \theta) F_f(J_1) \sqrt{F_c(J_1, \kappa)}$$ (1)

with

- $F_f$ describes the failure surface,
- $\mathcal{R}$ is the William-Warnke function,
- $F_c$ is the cap function and $\kappa$ its hardening parameter.

Isovalue 0 represents the yield surface. Concrete behavior is elastic as long as stress state is strictly inside the surface (i.e. $f<0$). The behavior becomes plastic if stress state reaches the surface (i.e. $f=0$).

The plastic potential reads:

$$g(J_1, J'_2, \theta, \kappa) = \sqrt{J'_2} - \mathcal{R}(J_1, \theta) c_p F_f(J_1) \sqrt{F_c(J_1, \kappa)}$$ (2)

with $c_p$ coefficient to reduce dilation. $c_p = 1$ when $\frac{\partial f}{\partial J_1} \geq 0$

### 2.1.1. Failure function
Shear failure function only depends on the first stress invariant $J_1$:

$$F_f(J_1) = \alpha_1 - \lambda_1 e^{\theta_1 J_1} - \theta_1 J_1 \quad (3)$$

where $\alpha_1, \beta_1, \lambda_1$ and $\theta_1$ are input parameters of the model.

### 2.1.2. $R$ function

The $R$ function depends on two invariants: $J_1$ and $\theta$. For the DPDC model, William-Warnke formulation has been chosen [Grassl 2006]:

$$R(J_1, \theta) = \frac{2(1-e^2) \cos(\theta) + (2e-1)\sqrt{4(1-e^2) \cos^2(\theta) + 5e^2 - 4e}}{4(1-e^2) \cos^2(\theta) + (2e-1)^2} \quad (4)$$

with

$$e = \alpha_2 - \lambda_2 e^{\beta_2 J_1} - \theta_2 J_1$$

$e$ is also called “eccentricity parameter”. It controls the shape of the deviatoric section of the yield surface according to the level of pressure: from tensile pressure to low confinement, the deviatoric section is triangular, whereas it is circular at high confinement. Geometrically, $e$ is the ratio between the value of $R$ function at triaxial extension TXE (i.e. $\theta = 0$) and its value at triaxial compression TXC (i.e. $\theta = \frac{\pi}{3}$), that is always equal to 1 (see figure 1 below).

$\alpha_2, \beta_2, \lambda_2$ and $\theta_2$ are four input parameters.

$e$ has to be bounded:

$$0.5 \leq e \leq 1$$

The following figure presents the shape of the William-Warnke function in the deviatoric plane, for various values of pressure.
2.1.3. Cap function

For confining pressures from moderate to high, concrete behavior is modeled by the combination of failure and cap functions. This cap allows simulating pore compaction in order to control volumetric strain.

The cap is described by a two-part function [Schwer 1994]. In traction and for low confining pressures, the function is equal to one. For higher confinement, it describes a part of an ellipse. The general formulation of the function is:

\[
F_C(J_1, \kappa) = 1 + (J_1 - L(\kappa)) \frac{|J_1 - L(\kappa)| - J_1 + L(\kappa)}{2(X - L(\kappa))^2}
\]

(5)

with: \( L(\kappa) = \min(\kappa; \kappa_0) \) and \( X = L(\kappa) - RF_f(\kappa) \)

CEA Saclay
The representation of $F_c$ is presented on the figure below:

![Graph showing the representation of $F_c$](image)

**Figure 2: initial cap surface**

Geometrically, $X$ is the value of $J_1$ at which $F_c$ cuts the pressure-axis. $\kappa$ is the value of $J_1$ at the intersection between the failure surface and the cap surface. (see figure 2 above).

The cap depends on $\kappa$, the isotropic hardening parameter which determines the size of the cap. It is controlled by the plastic volumetric strain $\varepsilon^p_v$ [Schwer 1994] according to the following law:

$$
\varepsilon^p_v = -W\left(1 - e^{D_2(X(\kappa)-X_0) - D_2(X(\kappa)-X_0)^2}\right)
$$

(6)

The graphic representation of this law is given below:
Cap shape $R$, initial cap location $X_0$, maximum plastic volume reduction $W$, linear hardening $D_1$ and quadratic hardening $D_2$ are five input parameters.

Combining the three previous functions, it is possible to plot the yield surface in principal stress space for a given hardening parameter $\kappa$:

**Figure 3:** pressure as a function of total volumetric strain, during isotropic compression

**Figure 4:** Shape of the yield surface in principal stress space.

*(The black arrow represents the pressure axis from compression to tension)*

2.1.4. Plastic update

CEA Saclay
When the stresses lay outside the yield surface, the cutting plane algorithm [Fichant 1996] is used to return on the yield surface and calculate plastic deformation increment. A few iterations (<10) is needed, but strain increment must be limited to avoid divergence of plastic iterations [Jirasek 2001].

When the strain increment exceeds a maximum strain limit defaulted by the model, subincrementation is used. This case is typically encountered when rate effects are higher than 10 per second. Furthermore, although $\partial R/\partial \theta$ is discontinuous for $\theta = \frac{\pi}{3}$ and $\epsilon=0.5$, the algorithm always converges.

2.2. Damage model

The damage formulation models two phenomena: strength reduction and Young modulus reduction.

In this model, damage is isotropic and controlled by a scalar damage parameter $d$. Stress tensor with damage is expressed as follows:

$$\sigma^d = (1 - d)\sigma$$

(7)

where $\sigma$, the effective stress tensor is updated by the plasticity algorithm.

The damage parameter ranges from 0 to 1 and increases as damage accumulates. $d = 0$ for no damage and $d = 1$ for complete damage. Damage starts when strain-based energy terms exceed the damage threshold. Moreover damage cannot initiate on the cap if plastic volume strain is compactive.

The parameter $d$ is expressed via two different formulations, called brittle damage and ductile damage which are used for simplicity (even though thermodynamically not admissible).

First, let us introduce the level of stress triaxiality $T_x$, defines as:

$$T_x = \frac{J_1}{\sqrt{3} J_2}$$

$T_x$ equals 1 in tension, 0 in pure shear and -1 in compression.

CEA Saclay
It is assumed that brittle damage accumulates in tension but also in compression with low confinement (i.e. $T_x > -1$).

It depends on the maximum principal strain $\epsilon_{\text{max}}$ by mean of an energy-type term $\tau_b$:

\[
\tau_b = \sqrt{E \epsilon_{\text{max}}^2}
\]  

(8)

Brittle damage accumulates as soon as the failure surface is reached. This hypothesis defines the initial threshold $\tau_{0b}$.

2.2.2. Ductile damage

Ductile damage accumulates when the pressure is compressive (i.e. $T_x < 0$).

It is driven by an energy-type term $\tau_d$:

\[
\tau_d = \sqrt{\frac{E}{\gamma}} \tilde{\epsilon}
\]  

(9)

Ductile damage accumulates as soon as the shear failure surface is reached where plastic volume strain is dilative (damage does not initiate on the cap where plastic volume strain is compactive). This hypothesis defines the initial threshold $\tau_{0d}$.

2.2.3. Damage accumulation

When $\tau$ become greater than its value at previous time step, damages accumulate according to the following function:

CEA Saclay
For each kind of damage, A and B are two shape parameters that depend on material properties and element size. Even if the function is unique for ductile and brittle damage, Figure 5 shows that A and B parameters change the shape of damage evolution with $\tau-\tau_0$.

\[
d(\tau) = \frac{d_{\text{max}}}{B} \left( \frac{1 + B}{1 + Be^{-A(\tau-\tau_0)}} - 1 \right)
\]

(10)

Typically, the blue curve represents the evolution of ductile damage because initial slope is equal to zero. On the contrary, the green one, with steep descent upon initiation of damage, is typical to brittle damage evolution.

In this model, to avoid numerical difficulties (mesh tangling ...) it was chosen to use $d_{\text{max}} = 0.999$ for brittle damage. For ductile damage, the expression reads:

$$d_{\text{max}} = \min \left( 0.999; -\frac{1}{f_x} \right)$$

to simulate the diminution of ductile damage with confinement.

2.2.4. Transition zone
The previous definition of brittle and ductile damage exhibits a range of pressure in which both kinds of damage are active, called “transition zone” and defined as:

\[ T_x \in ]-1; 0[ \]

In this domain, damage parameter is defined as a combination of both ductile and brittle formulations, using the following expression:

\[ d = d^b + T_x(d^b - d^d) \]  \hspace{1cm} (11)

Thus, there is no sharp transition between brittle and ductile damage.

Crack closure remains to be introduced in the damage model.

2.2.5. Reducing mesh size sensitivity

The softening model described above would definitely be mesh size dependent. Damage would preferentially accumulate in the smallest element. According to the energy method due to [Hillerborg 1976], the proposed solution to tackle mesh size dependency is to maintain constant fracture energy regardless of element size. More precisely, this is done by introducing the element length \( L \) (cube root of the element volume) and a fracture energy term \( G_f \) that depends of damage formulation (brittle or ductile).

Fracture energy is equal to the area under stress-displacement curve after peak. It is defined with the following integral form:

\[ G_f = \int_{x_0}^{\infty} (1 - d) f_{ps} \, dx \]  \hspace{1cm} (12)

with \( x \) the crack opening, \( x_0 \) the crack opening at peak strength \( f_{ps} \).

In this expression, fracture energy is defined according to the shape parameters \( A \) and \( B \). Integration is performed with \( d_{max} = 1 \) and \( \tau - \tau_0 \) is expressed by the following expressions:

CEA Saclay
\[
\tau_b - \tau_{0b} = \sqrt{E} \frac{X - X_0}{L} \quad \text{for brittle damage}
\]

\[
\tau_d - \tau_{0d} = \sqrt{\frac{f_{ct}}{L}} (\sqrt{X} - \sqrt{X_0}) \quad \text{for ductile damage}
\]

In this way, shape parameter \( A \) depends of \( G_f \) and \( L \).

### 2.3. Strain rate effects

In this model, rate effects are accounted for with viscoplasticity instead of plasticity, increase of damage threshold and eventually fracture energies. As this part of the model is not completely achieved, and consequently not used for the IRIS benchmark, it is not described here.

### 2.4. Input parameters

This model needs 35 input parameters that must be fitted to data. For easy use of the model, default material parameters are calculated internally by using correlations based on experimental data in a large range of ordinary concrete strength. Finally, only two parameters are mandatory: the unconfined strength of concrete in compression and the aggregate size. Correlations demand that SI units must be used.

### 3. CYLINDER TESTS

#### 3.1. Modelling

The concrete cylinder is meshed irregularly to avoid preferential orientation of the damage zone.

Finite elements are 8 nodes solid elements with reduced integration (only one integration point) because with complete integration (8 Gauss points), inclined shear bands are never obtained. A possible explanation for this numerical behavior is a shear or volumetric locking because reduced integration solves the problem. Nevertheless this point remains to be investigated.
The concrete cylinder is compressed between two rigid plates of which one is moving at constant velocity. Ends of the cylinder are clamped on the plates.

The characteristic data used for concrete are: $E_c=29670$ MPa, $\nu_c=0.22$, $\rho_c=2400$ kg/m$^3$, $f_c=69$ MPa and aggregate size (diameter) is 0.8 cm. All others parameters are calculated by the DPDC model.

For this exploratory study, the eccentricity factor $e$ is fixed and two values have been used: 0.5 and 0.7 the first one leading to a triangular yield shape in the deviatoric plane and the second leading to a rounded yield shape.

3.2. First results on compression tests

For $e = 0.5$, after an approximate X shaped shear band formed just before the instability, only one branch of the X grows and an inclined shear band (48° with the axis of symmetry) appears (figure 6). For $e = 0.7$, a horizontal strain localization band is obtained (figure 7).

In any case, the instability is very sudden (figure 8). Even if the pressure effect (via the eccentricity factor $e$) is qualitatively reproduced (shear band is less and less inclined when the pressure increases) no vertical shear bands have been obtained. Perhaps, this is due to the boundary conditions between the plates and the cylinder and this point remains to be investigated.
Damage field after instability has occurred

Damage just before the instability

Damage just after the instability

At the onset of shear band formation

Later, brittle damage appears

Figure 6: Damage field of the cylinder for $e=0.5$

Figure 7: Damage field of the cylinder for $e=0.7$
4. FLEXURAL TEST

4.1. Basic choices

Missile is modelled because it gives more precise impact force than approximated method like Riera’s one.

Even missile buckling and slab cracking do not follow strictly the symmetry pattern, only a quarter of the shock configuration is discretized to save computation time. So, boundary conditions of symmetry type were introduced for missile and slab.

In the first run, a lot of ductile damage was created under the head cap of the missile and the calculation was interrupted by tangling mesh after 0.18 ms of physical time. This was probably due to the lack of rate effects which strengthens the slab. So, to get rid of this difficulty, it was decided to cancel ductile damage. In this way, the next run was completed.

The coordinate system is the following: x is the horizontal axis of the slab, y is the vertical one and z is perpendicular to the slab.

CEA Saclay
Rebar nodes and concrete nodes are not coincident which allows smearing damage over concrete elements that include rebar nodes.

4.2. Concrete slab modelling

Even if Hillerborg method is used, it seems preferable to have a regular spatial discretization. This is why a structured mesh of 123350 elements and 138240 nodes is used (element size along x or y: \( l_{xy} \equiv 1.1 \text{ cm} \) and element size along z: \( l_z \equiv 1.07 \text{ cm} \)). These sizes are carefully chosen to have rebars and stirrups inside concrete elements and not near concrete nodes (see later steel concrete bond modelling).

Finite elements are 8 nodes solid elements with only one integration point.

4.3. Rebars

6099 beam elements are used to discretized longitudinal rebars (element size \( l_b \equiv 1.375 \text{ cm} \)) and 1859 beam elements are used to mesh stirrups (element size \( l_z \)). Diameter and position are those of [Vepsä 2010]. Lap splices are not represented. Stirrups are modelled by straight beams along axis z connecting horizontal rebars. Horizontal rebars and vertical ones are not connected.

Perfect bond is assumed between steel and concrete and realized by means of kinematic constraints because steel nodes and concrete nodes are not coincident.

4.4. Support modelling

The U-shaped plate, named corner plate, which surrounds the slab is meshed with 5428 shell elements. Bond between slab and concrete is supposed perfect and consequently corner plate nodes and concrete slab nodes are merged. For this reason, corner plate width is reduced to 8.5 cm (in place of 9 cm).

The corner plate is maintained between 2 frames made of I-shaped beams bolted together. The supporting frame has to be modelled because, due to the flexibility of the back pipes which support the frames, the slab displacement is not completely negligible during the impact (the lowest eigen period of the slab clamped in the support is about 11 ms and corresponds to the load duration – the slab displacement under the impact load can be evaluated to 0.65 mm). Furthermore, as the supporting “I” beams are very rigid, the supporting frame can be simply modelled by a one degree of
freedom system $w_{zsupt}$. For a quarter, its mass is assessed as $M_{zsupt}=745$ kg and its rigidity, these of one back pipe, is $K_{zsupt}=375$ MN/m.

Finally, the slab is simply supported on the frame (hence along metallic log bearings $w_z=w_{zsupt}$).

4.5. Missile modelling

The missile is meshed with 17640 Q4GS (shell element with 4 nodes for flexure and shear). The size of elements is about 0.5 cm in order to have a fine description of buckles.

4.6. Contact modelling

Contact modelling is performed with master nodes and slave elements method between missile and slab. For the missile, self-contact is activated to avoid the covering of buckles. All these conditions are modelled with unilateral kinematic constraints which are solved implicitly.

4.7. Material data

4.7.1. Missile

The characteristic data used for the mild steel are: $E_s=210000$ MPa, $\nu_s=0.3$, $\rho_s=7591.5$ kg/m$^3$.

Strain rate can be higher than 1000 s$^{-1}$ and generates a strong increase of steel yield stress (x3). This effect is of primary importance to reproduce the crush length of the missile and the duration of impact force.

The Johnson-Cook constitutive model was chosen to serve this purpose [Vedantam 2005]. It reads:

$$\sigma = (\sigma_y + B\dot{\varepsilon}_p^n)(1 + C \ln \left( \frac{\dot{\varepsilon}_p}{\dot{\varepsilon}_{p0}} \right))$$

CEA Saclay
where $\sigma$ is the effective stress, $\varepsilon_p$ is the effective plastic strain, and $\dot{\varepsilon}_p$ is the strain rate. Other variables are the following constants: $\sigma_y = 300$ MPa, $B = 703$ MPa, $C = 0.03$, $n = 0.4475$ and $\dot{\varepsilon}_{p0} = 10^{-4}$ (that corresponds to static case). Yield stress $\sigma_y$, hardening parameters $B$ and exponent $n$ are calibrated by mean of the static stress-strain curve. $C$ is adjusted to find impact duration of about 18 ms.

### 4.7.2. Concrete

The characteristic data used for concrete are: $E_c = 27000$ MPa, $\nu_c = 0.2$, $\rho_c = 2300$ kg/m$^3$, $f_c = 64$ MPa and aggregate size (diameter) is 0.8 cm. All others parameters are calculated by the DPDC model.

### 4.7.3. Rebars

The characteristic data used for steel rebar are: $E_s = 210000$ MPa, $\nu_s = 0.3$, $f_{yk} = 600$ MPa, $\rho_s = 7800$ kg/m$^3$. Steel is supposed elastic plastic with isotropic hardening. Stress-strain curved is issued from [Vepsä 2010]. No rate effect is included.

### 4.7.4. Corner plate

The characteristic data used for corner plate are: $E_s = 210000$ MPa, $\nu_s = 0.3$, $f_{yk} = 500$ MPa, $\rho_s = 7800$ kg/m$^3$. Steel is supposed elastic perfectly plastic.

### 4.8. Comparison with test

Time step of this calculation is about $3.10^{-7}$ s. It is fixed by the missile elements. Duration of the run is about 32 h (CPU).

The comparison shows that slab deflection (and obviously deformation) is overestimated (see figures 9 and 10). The order of magnitude of support force is correct, but not its time history (figure 11).

Cracking is severe, in particular on the back side one (figures 12-14) and very often follows reinforcement. It is important to mention that diagonal cracks appear on the front side only when...
the slab goes back due to its elasticity. In this return phase, damage of the back side does not evolve significantly because crack closure is not introduced yet in the DPDC model.

Plastic deformation of rebars, localized onto back side diagonal is about 2 % (figure 15).

The effective deformation rate is drawn on figures 16 and 17. It is localized on back side of the slab and under the impact zone. It is particularly important in damaged elements. It varies from 200 s\(^{-1}\) at the beginning of impact to a little more than 1 s\(^{-1}\) at the end of calculation. It is well known that brittle damage, in particular, is strongly reduced in that ranges of effective deformation rate [Brara 2001]. So, the lack of rate effects leads to a strong over-estimation of damage and slab deflection.

Besides, the IRIS 2010 flexural test has been calculated again with the impact force calibrated on experimental results. Then, maximum displacements given by GLRC and DPDC models which have no rate effect compare satisfactorily.

![Figure 9: Displacement W2 (0.25, 0.25, -0.075)](image-url)
Figure 10: Strain R2 in concrete (0.229, 0.229, 0.075)

Figure 11: Support force (1/4)
Figure 12: Representation of the front side crack pattern

Figure 13: Representation of the back side crack pattern

Figure 14: General view of damage in the slab (with a cross section showed by the red plane)

Figure 15: Plastic deformation of rebars is localized on back side diagonal
5. CONCLUSION

Main lessons learned during our IRIS 2010 and 2012 simulations are the following:

- The GLRC approach using a shell approach which was used in 2010 is well suited for industrial applications dealing with soft impact on reinforced concrete structures. The main drawback is the modelling of shear behaviour that is supposed to be elastic. The lack of deformation rate leads to conservative results. It is worth to note that adaptation to this model to treat hard impact is not straightforward.

- Even though DPDC behaviour law for 3D Finite Element is not implemented completely in EUROPLEXUS, its robustness was shown in this IRIS 2012 calculation: plasticity algorithm always converged and the model was able to deal with severe damage. Furthermore, time calculation is reasonable.

Besides, this kind of model was used with success by numerous IRIS participants whereas more sophisticated approaches are costly and not always convincing. This is why development concerning mainly rate effect and erosion of fully damaged and distorted elements will be pursued until DPDC model reaches state of the art.
6. ACKNOWLEDGEMENTS

IRIS benchmark is a great opportunity to test various models of reinforced concrete structures under dynamic loadings and to discuss about such a challenging and crucial issue for safety. Gratitude is expressed to VTT, IRSN and CNSC for financing the tests and NEA/CSNI for organising the benchmark. Technical support of EUROPLEXUS team, and in particular, of H. Bung, F. Bliard and V. Faucher at CEA Saclay, is gratefully acknowledged.

7. REFERENCES

Experimental and numerical study of concrete at high strain rates in tension.

[Fichant 1996] Fichant S.
Endommagement et anisotropie induite du béton de structures. Modélisations approchées.
Thèse d’Etat, LMT, ENS Cachan, France, 1996.

Damage-plastic model for concrete failure.

[Hillerborg 1976] Hillerborg A., Mooder M., Patersson P.
Analysis of crack formation and crack growth in concrete by means of fracture mechanics and finite elements.
Cement and Concrete Research, 1976.

CEA Saclay
IRIS 2012 NUMERICAL SIMULATION REPORT

Inelastic Analysis of Structures.

Yield line theory.

[Koechlin 2007]  Koechlin P.
Modèle de comportement membrane-flexion et critère de perforation pour l’analyse de structures minces en béton armé sous choc mou.

[Murray 2007]  Murray Y.
Users’ manual for LS-DYNA Concrete Material Model 159.

[Schwer 1994]  Schwer L., Murray Y.
A three-invariant smooth cap model with mixed hardening.

[Vedantam 2005]  Vedantam K., Bajaj D., Brar N. S., Hill S.
Johnson-Cook strength models for mild and DP 590 steels.

[Vepsä 2010]  Vepsä A.
CEA Saclay
Experimental tests for bending and punching behavior of reinforced concrete walls under impact loading.

Research report VTT-R-05587-10.
1. Introduction
The current report describes the FE model and simulation results produced by CNSC Team I (former Team II for the IRIS_2010) for all 3 benchmark tests included in IRIS_2012 project. The geometry, material modelling and software used for two impact tests were identical to the IRIS 2010 benchmark. However, some adjustment of concrete properties was employed based on test results. The same material model and software were also used for the modelling of the additional tri-axial static test.

2. Improvements done for IRIS_2012
Based on test results and additional modelling conducted in 2011 and 2012, the following improvements were made in all FE models:

- Strain-rate effect in Winfrith concrete model was excluded. Practically, Winfrith concrete model MAT084 was changed to MAT085. As a result, computed dynamic strength of the concrete slab was significantly decreased. To compensate for this decrease, significantly different erosion criteria were introduced as follows:
  - Concrete element erodes when either maximum or minimum principal strain reaches 70% instead of 5% in IRIS_2010 benchmark
  - Identical erosion criteria were applied for all 3 benchmark tests.

This decision resulted in better correspondence between FE and test results and was based on the following observations:

- Including strain-rate in concrete material model resulted in significantly lower slab maximum and residual displacements obtained in IRIS_2010 benchmark
- Including strain-rate in concrete also resulted in significantly higher peak stress in tri-axial tests, see Fig. 11 in section 8.3.
- Previous values of erosion ±5% were significantly lower than values used by most of IRIS_2010 participants
- As mentioned by concrete expert Len Schwer (Schwer Engineering & Consulting Services, Windsor CA USA, http://www.schwer.net/SECS/) MAT084 have some energy conservation problems and even a simple modeling of clamped-clamped beam under self-weight using MAT084 did not display a symmetric response in the crack patterns.

3. Material Input Data
CNSC Team I found that material input data provided are consistent. The only one discrepancy found was the difference in peak locations for the tri-axial tests without confining pressure on slides 1 and 2 in file IRSN tests results.pptx.

However, highly non-linear material models used in current FE analysis require some additional data that were not provided. Unfortunately, most of these additional parameters require special high-precision tests to identify them. This option was clearly out of the scope for the current project. We would like to mention the following non-defined material data:

(i) Strain-rate effects for missile, rebars and concrete. In the absence of these data, the widely used Cowper-Symonds strain-rate law was selected as follows [1]:

A III.5 Team #8 CNSC-Team I
IRIS_2010 NUMERICAL SIMULATION REPORT
CNSC TEAM I
\[ \sigma_{y, \text{dynamic}} = \sigma_{y, \text{static}} \cdot (1 + \epsilon/C)^{1/p} \text{, where } C=40 \text{ s}^{-1}, \ p=5 \]

The selected values of parameters C and p are widely used for a typical mild steel and were implemented for both missile and rebars in all FE models.

(ii) Winfrith concrete model used in all our FE simulations requires additional material properties that were not provided, such as full pressure - volumetric strain curve and fracture energy. These properties cannot be obtained from the additional tri-axial test. Therefore, the default values were selected for these properties in all FE simulations.

(iii) No data were provided for the scatter on all material properties. The bounding box (min/max values) was also not provided.

However, CNSC Team I fully understands that this deficiency is not a result of some negligence from the Organizing Committee but rather is the result of the very complex nature of the problem.

4. Selection of Modelling Type
CNSC Team I used the explicit commercial FE code LS-DYNA ls971d R4 64-bit version on Windows XP 64-bit OS. This is exactly the same version that was used in IRIS_2010 benchmark. This selection was motivated by the common understanding of LS-DYNA advantages in modelling fast impact-type events with highly non-linear material behaviour using explicit FE algorithms. In addition, LS-DYNA has high-fidelity non-linear constitutive models for both steel and concrete allowing adequate modelling of penetration and perforation of concrete target in the required range of model parameters. To be consistent, tri-axial tests were also modelled using the same software and material model even LS-DYNA is not the best choice for quasi-static modelling.

The CNSC Team I consists of one person working on IRIS_2012 project part-time (~20%) for 5 months. However, this person has spent considerable time working on IRIS_2010 and another similar project involving modelling of other VTT tests with different test conditions. The most serious constraints encountered for this project were as follows:

(i) Very short time period and only one person available part-time to conduct the simulations required, and

(ii) Limited computer power (only one PC with one CPU license installed on it).

These limitations resulted in limited amount of cases analyzed by CNSC Team I. Many cases with sensitivity analysis of numerous model parameters were dropped due to lack of time and computer power. However, all 3 cases required in IRIS_2012 were modelled and results are provided in corresponding Excel files. Additionally, the effect of most sensitive parameters, such as strain-rate in both concrete and reinforcement, reinforcement fracture strain and concrete erosion criteria were examined.

5. Concrete Slab Modelling
5.1 FE Model selection
Due to the nature of the tests a full 3-D FE model was created for all 3 tests. Based on IRIS_2010 and other work, one quarter of the entire slab with appropriate symmetry conditions and mesh density were selected for the impact tests, see Excel file with results. Due to a smaller size, the entire concrete cylinder was modelled in tri-axial tests.

5.2 Choice of FE for the concrete and for the reinforcing steel
FE selected in IRIS_2012 were identical to FE used in IRIS_2010 as follows:
(i) 3-D 8-noded FE with reduced 1-point integration (constant stress FE) was selected for concrete slab. This is the default solid FE in LS-DYNA and is compatible with selected material model. Due to low order shape functions and reduced integration for this FE, CNSC Team I avoided excessive element distortion, even for zones near the impact area
(ii) 2-D 4-noded shell FE with Belytschko-Tsay formulation and 2 integration points through thickness (LS-DYNA default) was selected for steel frame used in impact tests
(iii) Both lateral and vertical reinforcements were modelled explicitly using 2-noded beam FE (Hughes-Liu formulation with 2x2 Gauss integration points, LS-DYNA default). Spherical joints connections with coincident nodes (no moment transfer) were assumed between reinforcement layers in x- y- and z- directions and between reinforcement and concrete. Despite a computational penalty, CNSC Team I decided to use coincident concrete-reinforcement nodes instead of commonly used approach with separate FE nodes for concrete and reinforcement meshes tied using kinematic constrains. We believe that the latest approach could lead to inaccurate results in cases involving FE erosion.

5.3 Constitutive laws for the concrete and for the reinforcing steel
The following constitutive laws were selected for the concrete and for the reinforcing steel for all benchmark tests:
(i) Winfrith material model MAT085 without strain-rate effects (*MAT_WINFRITH_CONCRETE in LS-DYNA) was selected for the concrete slab for all tests due to the following reasons:
   • The model is suitable for high-speed impact analysis involving cracking and crushing
   • The model allows visualize crack patterns
   • The model uses default pressure versus volumetric strain curve in case when this curve is not defined explicitly
   • Input data provided by the Organizing Committee are sufficient if default pressure versus volumetric strain curve is used. The only missing parameter is the value of crack width $w$ at which crack-normal tensile stress goes to zero. This value was calculated based on fracture energy $G_p$ using the following relationship: $w = \frac{2 \cdot G_p}{UTS}$, where UTS is uniaxial compressive strength. Fracture energy $G_p$ was calculated by LS-DYNA using another material model MAT159 (*MAT_CSCM). Averaged compressive and tensile splitting strengths values provided to benchmark participants were treated as uniaxial compressive and tensile strengths (UCS and UTS) values needed for Winfrith model.
(ii) Bi-linear steel model MAT003 in LS-DYNA (*MAT_PLASTIC_KINEMATIC) was selected for the slab reinforcement as follows:

- Simplified bi-linear elastic-plastic model was selected due to lack of full stress-strain curves for the expected loading range. Plastic tangent modulus $E_t$ was calculated based on provided average values of yield stress $\sigma_y$, ultimate tensile strength $\sigma_t$ and relative ultimate elongation at break.

- Material failure was introduced for stresses exceeding Ultimate Tensile Strength (UTS) using failure strain parameter $\varepsilon_f$ for defining eroding FE. Values of relative averaged ultimate elongation provided to participants were used as failure strain for eroding elements.

- Pure isotropic hardening ($\beta=1$) was selected for all cases. Varying $\beta$ from 0 to 1 corresponds to varying between pure kinematic and isotropic hardenings. The value $\beta=1$ was selected based on authors previous experience for large deformation of steel structures. Due to non-cyclic nature of the benchmark tests, the effect of $\beta$ selection is expected to be limited.

- Cowper-Symonds strain-rate law with parameters $C=40$ s$^{-1}$ and $p=5$ (see section 3) was selected for all cases.

5.4 Material and FE erosion

Numerous runs conducted earlier show that FE erosion is needed for adequate modelling in cases involving target penetration and/or perforation for the following reasons:

- To prevent excessive FE deformation in the impact zone leading to non-convergence
- To allow missile moving though the target for Lagrange FE formulation
- To model material damage and failure

Erosion for reinforcement FE and missile was incorporated in the correspondent material model MAT003 as described earlier in Section 5.3. Winfrith concrete model does not have erosion inside the material constitutive law. Moreover, as pointed by L. Schwer (Schwer Engineering & Consulting Services, Windsor CA USA, http://www.schwer.net/SECS/), this model also does not have material softening after concrete strength in uniaxial compression is exceeded. LS-DYNA option for including erosion (*MAT_ADD_EROSION) has in total 14 different erosion criteria. Unfortunately, none of them has the direct physical correlation with the concrete damage during impact. Based on earlier work, CNSC Team I selected the following two criteria:

- Maximum principal stress at failure $\varepsilon_1>\varepsilon_{\text{max}}$ (positive value) that governs erosion in tension, and
- Minimum principal stress at failure $\varepsilon_3<\varepsilon_{\text{min}}$ (negative value) that governs erosion in compression

As mentioned above in Section 2, erosion criteria $\varepsilon_{\text{max}}=70\%$ and $\varepsilon_{\text{min}}=-70\%$ were selected for all FE models in IRIS_2012 benchmark. Please notice, that different criteria or their combinations could be also selected. In case with equivalent absolute values of $\varepsilon_{\text{max}}$ and $\varepsilon_{\text{min}}$, the selected criteria could be approximately reduced to only one: maximum shear stress at failure $\gamma_{\text{max}}=\varepsilon_{\text{max}}=\varepsilon_{\text{min}}$ since $\gamma=(\varepsilon_1-\varepsilon_3)/2$. However, the CNSC Team I
found that for some benchmark cases using both $\varepsilon_{\text{max}}$ and $\varepsilon_{\text{min}}$ instead of $\gamma_{\text{max}}$ produced better results. Please, notice that no concrete elements were eroded in flexural and tri-axial tests for the selected erosion criteria $\pm70\%$.

6. **Missiles Modelling**

Adequate FE models were created for missiles in all impact benchmark cases as follows:
- A quarter model was selected for the missile.
- A 2-D 4-noded shell FE with Belytschko -Tsay formulation and 2 integration points through thickness (LS-DYNA default) were selected for missile side pipes and back plates. The equivalent “effective” material density was calculated for the back plate in both impact tests to match the total mass of the part. The same FE type was also used for “soft” missile head in the bending test.
- A 3-D 8-noded FE with reduced 1-point integration identical to those used for concrete slab were selected for “hard” missile head in punching test.
- Winfrith concrete model identical to that used for slab was selected for “hard” missile filling in punching test. Material properties for the light-weight filling were not provided to benchmark participants. They were selected by scaling slab properties as follows: $\rho=1120$ kg/m$^3$ - to match the total mass of the filling, $E=3.22$ GPa, $\nu=0.17$, UTS=1 MPa, UCS=3 MPa.
- FE erosion was not introduced for missile filling
- Bi-linear elastic-plastic steel model identical to that used for slab reinforcement was selected for missile material
- Missile mass and initial velocity were selected according to B1 and P1 tests data

7. **Contact Modelling**

Full contact interactions during missile-target impact were modeled using several types of contact elements available in LS-DYNA as follows:
- Surface-to-surface contact with erosion was assumed between missile and concrete slab
- Surface-to-nodes contact was assumed between missile and reinforcement
- Self-contact was assumed for the “soft” missile in bending test to model missile buckling adequately
- Typical values were selected for contact friction coefficients as follows: $f_1=0.45$ for steel-on-concrete [2] and $f_2=0.74/0.57$ for static/dynamic steel-on-steel friction [3]

8. **Modeling Results**

As was mentioned earlier, all simulations were conducted using LS-DYNA with default value of time step. The simulation time for all three benchmark tests was selected according to guidelines provided: 50 ms for “hard” and 100 ms for “soft” missiles. For the tri-axial tests modeling the simulation time of 10 s was selected to avoid any dynamic effects as follows:
- 0.5 s for the first stage (applying confining pressure linearly from zero to maximum value), and
9.5 s for the second stage (applying compressive displacements on the specimen top surface linearly from zero to maximum value while keeping confining pressure constant). The maximum value was selected to provide axial compression of 57% for the specimen. This value represents the maximum compression that could be achieved in simulation. Practically, in all cases the simulation was terminated earlier upon achieving the global specimen crushing as depicted in graphs provided in Excel results file.

To stabilize the solution, viscous contact damping of 10% was selected as recommended in LS-DYNA manual. No additional material or stiffness damping was introduced since all simulation results show reasonably low residual oscillations for all output variables examined. As expected, the largest oscillations occur in the test with mostly flexural target behavior. It is worth to mention that crack patterns displayed in figures should be treated with certain caution. LD-DYNA post-processor results show all cracked FE including very small micro-cracks that are practically invisible. These cracked FE could be filtered by selecting the relative crack width limit for display. Unfortunately, this limit does not have direct correlation with the real crack size. Therefore, we decided to display all cracks in our Figures.

To simplify the process of obtaining displacements at given locations they were calculated at the closest nodes instead of employing some interpolation procedure. Authors believe that this simplification does not lead to significant errors. Since the length and exact position of strain gauges on the rebars were not provided, average axial FE strains were calculated at provided sensor locations Author believes that this simplification could result in significant deviation from experimental results that are recorded somewhere on the rebar surface and averaged over the gauge length. Additionally, the local bonding condition between concrete and reinforcement could also have a significant effect on the strain gauges measurements.

The results of FEA conducted were presented in the correspondent Excel files. Current report presents only main simulation results together with some additional information deemed important by CNSC Team I for understanding target and missile behavior. The results are provided in three subsections for benchmark tests #1 to 3 consequently.

8.1 Benchmark test #1 (punching test)
Fig. 1 shows the final shape of the practically un-deformed missile at the end of simulation time t=50 ms. Dislocations and partial crushing were observed only for the missile filling. Fig. 2 shows time history of the missile velocity predicted at the center of the missile head. The missile has perforated the slab with residual velocity ~38.3 m/s versus ~52 m/s for the IRIS_2010 model and 33.8 ± 1.4 m/s for the test P1. Fig. 3 shows the time history of the total contact force between missile and slab in the impact direction.

Finally, Figs. 4 and 5 show the final shape of the fully perforated target at the end of simulation time. No significant target damage occurred at the front face of the target around penetration hole. However, the back surface of the target is heavily cracked with concrete cover completely destroyed around the penetration hole. Figs. 4 and 5 show also intensive cracking through the entire target thickness with conical crack patterns in impact area. Some cracking also occurs around slab boundaries due to slab-frame interaction. As
was mentioned at the beginning of this section, crack patterns should be treated with
certain caution since not all cracks displayed are real visible cracks.

Please notice, that different output variables were obtained in a different way: some were
obtained directly from saved ASCII results files with time resolution 0.01 ms and some –
from plot files with time resolution 0.25 ms. The Author feels that any kind of
filtering/averaging is un-necessary and even misleading for these results. Therefore, they
are provided as obtained in FE modeling.

The results show clearly that FE predictions given by IRIS_2012 model are overall better
for outputs selected in Figs. 1 -5 as well as for all other output variables, such as support
forces, crack patterns and slab and reinforcement strains.

---

Fig. 1 "Hard" Missile shape at the end of simulation (t=50 ms)

Fig. 2 Time history of the velocity at the rear of the missile

Fig. 3 Time history of the total contact force in impact direction

Fig. 4 Slab cross-sections with crack patterns at the end of the simulation (t=50 ms)
8.2 Benchmark test #2 (flexural/bending test)

Fig. 6 shows the final shape of the missile at the end of simulation time t=100 ms. Fig. 7 shows the time history of the missile velocity recorded at the rear plate. The missile rebounded with residual velocity ~6 m/s in the IRIS_2012 model. Figs. 8 and 9 show the time history of the total contact force in the impact direction and slab displacements at the center of the rear surface respectively. Clearly, the predicted values for the maximum and residual displacements are much better for the IRIS_2012 model. Similar to punching test, the IRIS_2012 FE prediction are also overall better for all other output variables, such as support forces, crack patterns and slab and reinforcement strains. Finally, Fig. 10 shows the cross-section of the target at the end of simulation time t=100 ms. Localized target damage and limited cracks could be observed on this Fig. As was mentioned at the beginning of this section, crack patterns should be treated with certain caution since not all cracks displayed are real visible cracks.

The detailed analysis shows that relatively flexible missile head quickly transforms from convex to concave shape (“pop-in”) during the initial impact state. After this, side surface of the missile with relatively small contact area and large axial stiffness impact target directly creating some denting prior to softening due to buckling. Side surface buckling also leads to an increase in contact area, therefore limiting target penetration during the remaining impact duration.
Since no damage occurred in the vicinity of all sensors, obtained displacement and strain time histories are deemed to be representative for this benchmark test, see Excel file with results.

![Missile shape at the end of simulation (t=100 ms)](image1)

**Fig. 6** Missle shape at the end of simulation (t=100 ms)

![Time history of the velocity at the rear of the missile](image2)

**Fig. 7** Time history of the velocity at the rear of the missile

![Time history of the total contact force in impact direction](image3)

**Fig. 8** Time history of the total contact force in impact direction

![Time history of the slab displacements at the center of the rear surface](image4)

**Fig. 9** Time history of the slab displacements at the center of the rear surface

![Horizontal cross-section with crack patterns at the end of the simulation (t=100ms)](image5)

**Fig. 10** Horizontal cross-section with crack patterns at the end of the simulation (t=100ms)
8.3 Benchmark test #3 (quasi-static test)
The main purpose of this additional test was to provide a possibility to obtain additional properties of concrete used in tests. However, the results of FE modelling show that unidentified parameters of Winfrith model used could not be characterized using these tests. Moreover, Winfrith model was developed and fitted to dynamic test results [4, 5]. Therefore, as expected, this model did not produced very good agreement for the quasi-static tests, see Fig. 11. However, this Fig. show clearly that, again, modified IRIS_2012 concrete material model works better than IRIS_2010 model. Despite this discrepancy, the Author feels strongly that FE models developed are fully adequate for the impact modelling according to the main objective of IRIS benchmark projects.

Fig. 11 Stress-strain curves for tri-axial tests

9. Conclusions and Recommendations
The following conclusions were made by CNSC Team I based on FE modeling conducted during IRIS_2012 benchmark project:

1. LS-DYNA FE code was successfully used for modeling all three benchmark tests
2. Winfrith concrete model in LS-DYNA is capable of adequate modeling of slab behavior and inflicted damage for all impact tests. Excluding stain-rate effects in concrete resulted in improved FE predictions for both flexural and punching tests. However, Winfrith model is not fully suitable for modeling of quasi-static concrete behavior as required in tri-axial tests.
3. Even small scale FE models for impact tests #1 and 2 require significant computer resources: 35 hrs and 80 hrs respectively on Dell Xeon X5335@2.66 GHz with 8 CPU (only one CPU was employed due to CNSC license limitation). Some significant simplification or reduction methods should be employed in future to be able to analyze the larger structures.

4. FE and material erosion should be introduced in FE model to get adequate results in cases involving deep penetration and/or perforation. Unfortunately, FE erosion criteria could not be either measured nor justified based on physical considerations. CNSC Team I believes strongly that in order to define erosion criteria properly, sufficient number of experimental tests should be conducted first for the desired impact velocities and damage types. The type and values of erosion criteria should be determined based on these tests and used in further analysis of large scale structures.

References


A III.6 Team #9 CNSC-Team II

IRIS_2012 SIMULATION REPORT
CNSC Team II (University of Toronto)

Introduction and Modeling Approach

The VTT-B1 flexural and VTT-P1 punching specimens were modeled using the programs VecTor2 and VecTor3. The purpose of these simulations was to gauge the capabilities of the VecTor programs, in their current state, in modeling soft and hard impacts on reinforced concrete targets. Compared to programs such as LS-DYNA and ABAQUS, the VecTor programs are simplified tools, and one of the goals of the simulations is to compare the accuracy of the results of simplified models to experimental results and more complex hydrocodes, and to determine how these simplified models can be improved.

In keeping with the objective of employing simplified modeling procedures, in both VecTor2 and VecTor3 simulations, default material behaviour models were used in the majority of cases. Moreover, the only material properties specified were those provided in the experimental data; program-defined default material properties were used for properties that were not provided. No attempt was made to refine the analyses by adjusting the material models or material properties.

Modeling of Triaxial Specimens

IRSN Specimens 2, 4, 7, 8, and 9 were modeled in VecTor2, both at the material and structural levels. At the material level, a single element was used and at the structural level, a mesh of 2x2 mm elements was used. For confined strength of concrete, the two material models examined were the Kupfer/Richart and Montoya/Ottosen models; four different concrete dilatation models were examined. A quarter of the cylinder was modeled in VecTor2, with confinement being applied using nodal forces for the in-plane confinement and smeared steel reinforcement for the out-of-plane confinement; the smeared steel was prestressed to achieve the appropriate level of confinement for each specimen from the onset of the simulation. In terms of input material properties, the average properties provided in the IRSN triaxial test summary were used. At the material level, both confinement models captured an increase in peak pressure as confinement increased, and an appreciable amount of post-peak response was observed. At higher levels of confinement, the Montoya/Ottosen model tended to underestimate the peak stress, while the Kupfer/Richart model was able to match the experimental peak stress. The best results at the material level were achieved using the Kupfer/Richart confinement model and the Montoya 2003 concrete dilatation model. Those results, along with the material level model, are shown below.

Table 1: Material Level Results for Kupfer/Richart confinement and Montoya 2003 dilatation

<table>
<thead>
<tr>
<th>Specimen</th>
<th>f_{cc-exp} (MPa)</th>
<th>f_{cc-calc} (MPa)</th>
<th>\frac{f_{cc-calc}}{f_{cc-exp}}</th>
</tr>
</thead>
<tbody>
<tr>
<td>4</td>
<td>128</td>
<td>124.4</td>
<td>0.972</td>
</tr>
<tr>
<td>7</td>
<td>165</td>
<td>162.5</td>
<td>0.985</td>
</tr>
<tr>
<td>8</td>
<td>232</td>
<td>219.8</td>
<td>0.947</td>
</tr>
<tr>
<td>9</td>
<td>400</td>
<td>424.4</td>
<td>1.061</td>
</tr>
<tr>
<td>Mean</td>
<td></td>
<td></td>
<td>0.991</td>
</tr>
<tr>
<td>COV</td>
<td></td>
<td></td>
<td>0.049</td>
</tr>
</tbody>
</table>

Figure 1: Material level model for triaxial specimens
At the structural level, the Montoya/Ottosen confinement model captured the most post-peak behaviour when used with the Kupfer dilatation model; post-peak response was not captured at the structural level when the Kupfer/Richart confinement model was used. Note that the results of the triaxial modeling were not used to calibrate any input material parameters for the subsequent modeling work.

**Modeling of VTT-B1**

Modeling for the VTT-B1 specimen was done in both VecTor2 and VecTor3. Firstly, the missile was explicitly modeled in VecTor2 using a combination of steel elements and compression-only truss bars. Compression-only truss bars were used due to the fact that the VecTor suite of programs does not yet have contact elements. The forces in the compression-only truss bars were then used in the VecTor3 modeling of the specimen. The missile was not explicitly modeled in VecTor3, but nodal forces were used.

**Missile Modeling in VecTor2**

**Mesh and Simplifying Assumptions**

The model used in VecTor2, including both the missile and the target, is shown in Figure 2.

![VecTor2 mesh including missile and target](image)

The soft missile was modeled using structural steel elements, the missile forces were transferred to the target using compression-only truss bars, and the concrete slab was modeled using concrete elements. [This study is the first time VecTor2 is being used to model a soft impact.] Buckling of thin-walled steel elements is not rigorously considered in the VecTor programs, and thus instability issues arose when modeling the missile. In order to capture more of the response, element erosion in the missile was used; once the ultimate strain in the element was exceeded, the element was eroded (i.e., rendered inactive). Making this change did not greatly affect the slab displacement; a decrease in slab peak displacement of approximately 2 mm was observed when element erosion was introduced. This was determined to be acceptable for these preliminary analyses, since the results had been largely unstable with no missile erosion.

The missile itself was modeled in three sections. The back end of the missile was assigned a thickness in the out-of-plane direction of 254 mm, the full diameter of the missile, since that part of the missile was a solid plate. The thickness of the carbon pipe near the back of the missile was 29 mm, twice the wall thickness; similarly, the thickness of the remainder of the missile was 4 mm. The missile properties are summarized in Table 2.
A series of simulations was carried out in order to determine the effect of varying the area of the compression-only truss bars, in order to determine the proper load profile to use for the VecTor3 simulations. The effect of the compression-only truss bar area on slab displacement and missile damage are discussed in the next section.

**Determination of Load Profile from Truss Bar Forces**

In order to determine the appropriate load profile to use for the VecTor3 analyses, different combinations of compression-only truss bar areas was used in the VecTor2 simulations. The effect of the truss area on missile damage was assessed in terms of displacement at the back of the missile. It was observed that increasing the area of the truss bars increased the displacement of the back of the missile.

Changing the truss bar area had virtually no effect on the displacement profile of the slab; in all cases, the maximum displacement of the slab was approximately 49 mm, with a deviation of about 1 mm. While the slab displacement in each case was almost identical, the maximum force in the truss bars dramatically increased as the area increased. Despite the increase in initial force, the truss bar forces always returned to the same backbone load curve regardless of area. Because of this, the load profile was chosen such that the displacement obtained using nodal forces in VecTor2, for the same target, was 49 mm. That is, an impulse load using nodal forces was defined such as to give the same response as obtained when the missile was explicitly modeled.

The load profile used in the VecTor3 analyses is shown in Figure 3 below, along with the force profile from the VecTor2 analysis where the missile was modeled explicitly. The end of the impact was determined to happen when the nose of the missile rebounded from the slab; this occurred at approximately 20.2 ms.

**Figure 3:**
Load profile from truss bars and for VecTor3 analyses
The results for the displacement and velocity of the back of the missile are shown in the excel file.

**Modeling of Impact Using Nodal Forces**

Analyses were done in VecTor2 and VecTor3 using the load profile derived from truss bar forces. The finer VecTor2 model was used to help determine the correct load profile; the coarser VecTor2 mesh was used to illustrate the effect of mesh density on analysis results.

**VecTor2 Models**

The finer VecTor2 model has the same mesh density as the model where the missile was modeled explicitly; the coarse model has the same mesh density as the VecTor3 model. Both are shown in Figure 4. For the fine mesh, the element size is 10x12 mm on average; the coarse mesh has an element that is approximately 15x54 mm.

In the preliminary models, the concrete cover was 30 mm. The peak displacements were 48.7 mm at 22.4 ms and 44.2 mm at 20 ms for the fine and coarse meshes, respectively.

**VecTor3 Model**

The concrete target alone was modeled in VecTor3; the load profile defined in the VecTor2 analyses by modeling the missile explicitly was adapted for the VecTor3 analyses. A total of 4851 nodes, 4000 8-node rectangular concrete elements, and 1520 2-node truss bar elements were used in the modeling of the flexural specimen. The longitudinal reinforcement was modeled using truss bar elements, while the transverse reinforcement was modeled as smeared through the concrete elements. The concrete cover to the longitudinal reinforcement in the front and back was 15 mm. For the supports, only the degree of freedom in the direction of impact was restrained. Double symmetry was assumed, and those faces were restrained in the appropriate directions. The mesh and reinforcement layout are shown in Figure 5. The nodes loaded with the impulse force and the total load applied to the quarter slab is illustrated in Figure 6.
Comparison of Displacements

Table 3 below compares the VecTor3 peak displacement with the displacement from VecTor2 using the same mesh density. In VecTor3, a quarter of the slab was modeled; in VecTor2, half of the slab was modeled, but was modeled as simply supported since the analysis is limited to 2D.

Table 3: Displacement results for VecTor2 and VecTor3 simulations using nodal loads

<table>
<thead>
<tr>
<th>Sensor</th>
<th>Experimental</th>
<th>VecTor2 (Coarse Mesh)</th>
<th>VecTor3 20x20x10</th>
<th>VecTor3 20x20x10 w/ Strain Rate</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Max Displacement (mm)</td>
<td>28.89</td>
<td>40.1</td>
<td>30.43</td>
</tr>
<tr>
<td></td>
<td>Time of Max. Disp. (ms)</td>
<td>13.50</td>
<td>19.1</td>
<td>10.50</td>
</tr>
</tbody>
</table>

A detailed comparison of VecTor3 results to experimental results is shown in the excel file: reinforcement strains, concrete strains, displacements, and support forces are compared in that file for the base analysis. An analysis was also done considering strain rate effects for both the concrete and the steel; the displacements from the two VecTor3 analyses are compared to experimental peak displacements at each displacement sensor location.

Table 4: Comparison of displacements with and without strain rate effects

<table>
<thead>
<tr>
<th>Sensor</th>
<th>Peak Displacements</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Experimental</td>
</tr>
<tr>
<td>W1</td>
<td>28.89</td>
</tr>
<tr>
<td>W2</td>
<td>20.38</td>
</tr>
<tr>
<td>W3</td>
<td>21.96</td>
</tr>
<tr>
<td>W4</td>
<td>15.27</td>
</tr>
<tr>
<td>W5</td>
<td>19.53</td>
</tr>
</tbody>
</table>
Modeling of VTT-P1

VecTor2 Missile Modeling and Results

Similar to the flexural specimen, the punching specimen was modeled in both VecTor2 and VecTor3. An initial model was constructed in VecTor2 to determine the truss forces; this model is shown in Figure 7, and the material properties are listed in Table 5.

![Figure 7: VecTor2 mesh including missile and target](image)

### Table 5: Missile Properties

<table>
<thead>
<tr>
<th>Missile Truss Properties</th>
<th>1</th>
<th>2</th>
<th>3</th>
</tr>
</thead>
<tbody>
<tr>
<td>$A_{\text{truss}}$ (mm$^2$)</td>
<td>1235</td>
<td>$F_y$ (MPa) 355</td>
<td>$f'_c$ (MPa) 60</td>
</tr>
<tr>
<td>Number of Trusses</td>
<td>9</td>
<td>$F_u$ (MPa) 600</td>
<td>$f'_t$ (MPa) 4.04</td>
</tr>
<tr>
<td>$E_{\text{truss}}$ (MPa)</td>
<td>200000</td>
<td>$e_{th}$ (me) 3</td>
<td>$E$ (MPa) 29429</td>
</tr>
<tr>
<td>$F_y$ (MPa)</td>
<td>355</td>
<td>$e_u$ (me) 220</td>
<td>$e_u$ (me) 220</td>
</tr>
<tr>
<td>$F_u$ (MPa)</td>
<td>600</td>
<td>$E$ (MPa) 200000</td>
<td>$E$ (MPa) 200000</td>
</tr>
<tr>
<td>$e_{th}$ (me)</td>
<td>3</td>
<td>Thickness (mm) 80</td>
<td>Thickness (mm) 80</td>
</tr>
<tr>
<td>$e_u$ (me)</td>
<td>220</td>
<td>b/t (buckling) N/A</td>
<td>b/t (buckling) N/A</td>
</tr>
</tbody>
</table>

The area of the compression-only truss bars sum to the cross-sectional area of the missile; softer impacts were not examined.
While the VecTor2 analysis cannot simulate behaviour in situations where the missile punches completely through the target, the crack pattern can be examined to determine whether punching is likely to occur. The crack pattern obtained for VTT-P1, showing maximum damage to the slab, is shown in Figure 8.

The truss forces from the VecTor2 analysis and the load profile used for the VecTor3 analysis are shown below in Figure 9. The VecTor2 analysis was done with a timestep of 0.00001 seconds, and the forces were filtered to a timestep of 0.0001 seconds for VecTor3.

**VecTor3 Model**

A total of 6336 nodes, 5290 rectangular elements for the concrete, and 1104 truss elements for truss elements were used to model the punching specimen. Only the target was modeled; the impact was applied to the slab using the nodal force profiles determined from VecTor2. For the supports, only the degree of freedom in the direction of impact was restrained. Double symmetry was assumed, and those faces on the axes of symmetry were restrained in the appropriate directions. The mesh and reinforcement layout are shown in Figure 10. The cover used was 25 mm for the front and back faces. The nodes loaded with the impulse force and the total load applied to the quarter slab is illustrated in Figure 11.
VecTor3 results are compared to experimental results in the excel files; displacements, reinforcement, and concrete strains are shown. Since perforation cannot be modeled in VecTor3, the deformed mesh is examined in Figure 12 to illustrate the behaviour observed in the early stages of the simulation.

Figure 11:
Punching specimen loaded nodes

Figure 12:
Displaced mesh for load stage 5 illustrates large displacements in the centre of the slab with minimal displacements outside of the impact region (indicating punching)
The displacements along the length of the slab can also be used to determine if punching is occurring. Figures 13 and 14 illustrate the displacements along the horizontal line of symmetry at different times during the simulation. The displacements of both the front and back of the slab are shown. In both, punching is evident in the early stages of the simulation.

**Figure 13:**
Displacement profile along top of slab for different stages in the simulation

**Figure 14:**
Displacement profile along the back of the slab for different stages in the simulation
1. Introduction

The IRIS activity began in 2009 thanks to CSNI-IAGE,

The objective of the OECD IRIS_2010 Program was to make blind simulations on two kind of impact tests realized at VTT, a research center located at Espoo, Finland.

In the first one, a deformable missile had impacted a 15 cm thickness reinforced concrete slab at around 110 m/s (two repetitions). In the second one, a very rigid missile had impacted a 25 cm thickness reinforced concrete slab at around 135 m/s (three repetitions).

In a first step of the benchmark, it was asked to the various participants to model one of the so-called MEPPEN test II/4, in order to calibrate the model.

An important scattering was observed in the calculation results, much more than in the tests.

As a consequence, it was asked to the participants to calibrate their models based on the known experimental results, and to make other calculations in 2012. Sensitivity studies are expected and potential recommendations for further calculations.

This document synthesizes:

- The lessons learnt from the blind simulations made in 2010
- An analysis on the tests
- The sensitivity studies which were done in 2012
- The final calculation for the IRIS_2012 phase, based on the fitted values

2. References


[2] « Punching behaviour test of one 250 mm thick reinforced concrete wall under hard impact loading », note VTT-R-05588-10, 24.08.2010, VEPSA Ari


3. Flexural tests B1 and B2

3.1 Analysis on B1 and B2 test results

The test results B1 and B2 are detailed in the document [1]. Nevertheless, we make some comments on both tests in this section.

3.1.1 Test repeatability

The repeatability of both tests was quite good, concerning:

- Maximum displacements (around 2.8 cm to 3.2 cm for the maximum value):

  The figure below shows a comparison of maximum displacements between B1 and B2 tests, for each sensor. A 15 % to 20 % difference is observed:

- Post-damage oscillation frequency (around 35 Hz for both tests)

- Maximum concrete strains (around 2.5 ‰ to 3.0 ‰ for the maximum value):

  The figure below shows a comparison of maximum concrete strains between B1 and B2 tests, for each sensor. A 15 % to 25 % difference is observed:

Concerning the strains in the rebars, the test results are not as good as the displacement ones, and a comparison between B1 and B2 is difficult. The main reasons are the following:

- Some gauges failed during the missile impact, in particular on the test B1
• Some other gauges were contaminated by noise (in both tests)
As a consequence, only 7 gauges out of 18 can be properly compared.
Only 2 gauges on these 7 provide comparable results between B1 and B2. The maximum strain value is measured for sensor number 4, and the results are varying from 1.4 % (B2) to 3.7 % (B1):

![Maximum rebar strains](image)

3.1.2 Discussion on the strain gauges
The previous paragraph highlights that it would be very difficult to rely on the strains measured in the rebars. The potential reasons to explain such a result are:
• A lack of efficiency of this kind of gauge, in dynamic tests?
• Uncertainties on the measures in the rebars, due to the location of the cracks in the concrete thickness?

On the figure below, we observe the quarter cut slabs, after the impact. As we can see, the location of the cracks due to the bending behavior of the structure is not the same in the tests B1 and B2:
This point could explain the difference observed between both tests, concerning the strain in the rebars. Indeed, if a crack passes through an area where a rebar is embedded into the concrete, the crack’s opening can locally increase the strain and the steel lengthening. Near the impact, the amount of cracks can be important, but their location may vary a lot between two quite similar tests.

3.2 IRIS_2010 : B1-B2 impact modelling

3.2.1 FE software

The 2010 blind calculations were done with the transient dynamic computer program EUROPLEXUS (explicit integration scheme). This code is the property of CEA (Commissariat à l’Energie Atomique) and JRC (Joint Research Centre of the European Commission).

As a member of the EUROPLEXUS consortium, EDF makes its proper developments in the code, in particular in the domain of non-linear behavior of reinforced concrete structures. It developed the methodology which is used here, to simulate the IRIS flexural tests.

3.2.2 Impact modelling

The impact was modelled in terms of the pressure time history using the Riera method. The computations were done with the EUROPLEXUS finite element code (ref [3]).

3.2.2.1 Finite element mesh

The dimensions of the real slab are 2.1 m x 2.1 m x 0.15 m.

In our modelling, the outer dimensions of the mesh correspond to the span dimensions of the slab, they are given below:

- width : 2.0 m
- height : 2.0 m

The slab is meshed with 4-node thick shell elements.

The main characteristics of the finite element formulation are:

- thick shell element based on the Reissner-Mindlin theory with 6 d.o.f. per node: it has 4 integration points in the plane and 1 integration point through the thickness
- combined with the stress-resultant constitutive model (“GLRC DAMA” [6])

The slab is represented thanks to 4096 thick shell elements and 4299 nodes.

The average length of each finite element is 3.1 cm.
3.2.2.2 Material model and parameters

A stress-resultant constitutive law is used in our calculations, based on a through-the-thickness homogenisation of the non-linear behaviours of concrete and steel (one layer only).

This global model allows to take into account both membrane and bending non-linear behaviours of the slab. It is called GLRC_DAMA i.e. GLocal Reinforced Concrete_DAMAge.

This modelling strategy deals with damage and plasticity mechanisms. The cracking of concrete is modelled in the frame of the damage theory with two isotropic damage variables : one for the positive bending and the other one for negative bending (the distinction is made accordingly to the sign of the curvature). The plasticity is modelled with the double Johansen threshold surfaces.

The figure below shows the bending moment – curvature curve, and evidences both energy dissipation mechanisms of the law : 

The main material parameters of the global reinforced concrete law are given below :

<table>
<thead>
<tr>
<th>Material</th>
<th>Definition</th>
<th>Numerical value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Concrete</td>
<td>Concrete density</td>
<td>2500 kg/m$^3$</td>
</tr>
<tr>
<td></td>
<td>Young’s modulus</td>
<td>26 915 MPa</td>
</tr>
<tr>
<td></td>
<td>Poisson’s ratio</td>
<td>0.2</td>
</tr>
<tr>
<td></td>
<td>Slab thickness</td>
<td>0.15 m</td>
</tr>
<tr>
<td></td>
<td>Tensile strength</td>
<td>3.71 MPa</td>
</tr>
<tr>
<td></td>
<td>Compression strength</td>
<td>63.9 MPa</td>
</tr>
<tr>
<td>Bending reinforcement</td>
<td>Number of reinforcement layers</td>
<td>2</td>
</tr>
<tr>
<td></td>
<td>Young’s modulus</td>
<td>200 000 MPa</td>
</tr>
<tr>
<td></td>
<td>Yield stress</td>
<td>600 MPa</td>
</tr>
<tr>
<td></td>
<td>Steel density</td>
<td>7800 kg/m$^3$</td>
</tr>
<tr>
<td></td>
<td>Effective area of bending reinforcement per unit length for the upper layer</td>
<td>5.14 cm$^2$/m</td>
</tr>
<tr>
<td></td>
<td>Effective area of bending reinforcement per unit length for the lower layer</td>
<td>5.14 cm$^2$/m</td>
</tr>
<tr>
<td></td>
<td>Nondimensional coordinate of the upper reinforcement layer $\rho_{sup}$</td>
<td>+0.72</td>
</tr>
<tr>
<td></td>
<td>Nondimensional coordinate of the lower reinforcement layer $\rho_{inf}$</td>
<td>-0.72</td>
</tr>
</tbody>
</table>
Some GLRC parameters need to be calibrated by a non-linear quasi-static beam analysis. These parameters are described below:

- Prager coefficients in membrane $C_i N_j$ (for positive and negative bending and $j=[1,3]$ for each direction)
  - Function of elastic concrete and steel main parameters (young modulus, Poisson ratio, concrete thickness and effective area of reinforcement)

- Prager coefficients in bending $C_i M_j$ (for positive and negative bending and $j=[1,3]$ for each direction)
  - Function of the plastic slope of the moment-curvature curve

- Ratio (slope after cracking / slope before cracking) = $Q_p$

The numerical values for these parameters are the following:

<table>
<thead>
<tr>
<th>GLRC parameter</th>
<th>Numerical value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Prager coefficients in membrane $C_i N_j$</td>
<td>$2.16e6$ N/m</td>
</tr>
<tr>
<td>Prager coefficients in bending $C_i M_j$</td>
<td>$4.18e4$ N.m/m</td>
</tr>
<tr>
<td>Ratio $Q_p$</td>
<td>$0.145$</td>
</tr>
</tbody>
</table>

### 3.2.2.3 Boundary conditions

The slab is simply supported on its vertical and horizontal edges, so the displacements parallel to the impact axis (Z) are set to zero without any constraints on the rotations or on the tangential displacements (X, Y):

![Boundary conditions diagram](image)

### 3.2.2.4 Load description

The force-time history curve was applied exactly in the center of the slab, using the Riera method hypotheses for the missile.
An average mass by unit length is obtained by dividing the total mass of the missile by the missile length:

The crushing strength of the missile is obtained using the crushing force formula for folding shape:

\[
\frac{P_c}{\sigma_{y,\text{stat}} \cdot h \sqrt{h + r}} = 8.462 \times \left[ 1 + \left( \frac{\dot{\varepsilon}}{D} \right)^\frac{1}{q} \right]
\]

[Jones, 1989]

With a strain rate effect based on Cowper-Symonds constitutive model

\[
\frac{\sigma_{y,dyn}}{\sigma_{y,stat}} = 1 + \left( \frac{\dot{\varepsilon}}{D} \right)^\frac{1}{q}
\]

with

\[
\dot{\varepsilon} = \frac{V}{4r}
\]

and:

- \( q = 9 \text{ s}^{-1} \) and \( D = 100 \text{ s}^{-1} \) (for stainless steel pipe)
- \( \sigma_{y,\text{stat}} = 350 \text{ MPa} \)

The time evolution of the loading impact is shown on the figure below:

3.2.2.5 Damping

We introduce in the calculation of the non linear slab response a 2% mass proportional damping \( [C] = \alpha[M] \), on the 45 Hz frequency corresponding to the first bending mode of the slab, after damage.
3.2.3 Blind calculation results

The calculation time period is 100 ms.

The calculations were done with the sequential version of the EUROPLEXUS finite element code.

We give below the time characteristics of the calculation:

- Time step: $7.18 \times 10^{-6}$ sec
- CPU time: 22 min

3.2.3.1 Displacements results

In this paragraph, we compare the displacement curves associated with B1 and B2 experimental results (sensor 1) and the 2010 blind calculation results:

Concerning the calculation/test comparison, we can focus on the main following points:

- The first displacement peak is well represented by the model, concerning the magnitude value, which slightly overestimates the test values, and the instant when this maximum displacement is reached.
- The after-damage oscillations frequency is more important in the calculation slab response (around 45 Hz) than in the real one (about 35 Hz): this difference can be explained by an underestimation of the stiffness reduction of the slab.
- In the test, the damping is very important between the first displacement peak and the second one (around 10 % - 12 %), much more than in the calculation. The measured oscillations is slightly more damped after their first peak too.
After the first peak, the damping is equal to 2.5 %. It is calculated thanks to the following formula:

$$\zeta = \frac{1}{2\pi} \ln \frac{u_i}{u_{i+j}}$$

Where \( j \) is the number of oscillation periods

- The displacement amplitudes after the first peak and the residual displacement are strongly overestimated (factor 1.5 to 2)

These comments are similar for the other sensors.

**3.2.3.2 Concrete strain results**

The concrete strains are correctly reproduced, with a little overestimation of the blind calculation, compared with the test results (strain gauge n°2):
3.2.3.3 Rebar strain results

In the paragraph 3.1.1, we talked about the scattering of the experimental results between the tests B1 and B2, concerning the strain in the rebars. We did not make a detailed comparison calculation/test, for each gauge.

In the blind calculation, the maximum strain in the rebar is less than 2.0 %, and in the test it was around 3.7 % (strain gauge n°4):

3.2.3.4 Conclusion of the blind study

Except for the rebar strains, the maximum values (displacements, concrete strains) were correctly reproduced in the blind calculation.

But the results agree less well at the stage of free vibrations. The dominant frequency is not simulated very well, with an error of 30%, and the amplitudes are overestimated in the calculation (the damping is strongly underestimated in the calculation). This remark can be applied to the major part of the participants, as we can see it in the table below:

<table>
<thead>
<tr>
<th>Maximum displacement [cm]</th>
<th>Post-damage frequency [Hz]</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
</tr>
</tbody>
</table>
As a consequence, a sensitivity study based on the main parameters of the model has been done. The main goal is to reproduce better the vibratory behavior of the slab after the impact.

### 3.3 IRIS_2012 : B1-B2 impact modeling

#### 3.3.1 Sensitivity study on the material parameters

In this section, we study the sensitivity on the displacement results (sensor n°1) to the main concrete parameters of the global model ("GLRC_DAMA") described at paragraph 3.2.2.2.

The table below gives a comparison between the values used in the 2010 blind calculation and the ones which are used in the 2012 sensitivity studies:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>2010 blind calculation</th>
<th>2012 sensitivity study</th>
</tr>
</thead>
<tbody>
<tr>
<td>Unconfined compressive strength [MPa]</td>
<td>63.9</td>
<td>50.0</td>
</tr>
<tr>
<td>Tensile concrete strength [MPa]</td>
<td>3.71</td>
<td>3.0</td>
</tr>
<tr>
<td>Young’s modulus [MPa]</td>
<td>26 915</td>
<td>20 000</td>
</tr>
<tr>
<td>Poisson’s ratio</td>
<td>0.2</td>
<td>0.0</td>
</tr>
</tbody>
</table>
In the calculations whose results are presented below, all the other numerical data are identical to the ones defined in the 2010 analysis:

- Steel properties of the rebars
- Force-time history curve for the missile
- Boundary conditions of the slab
- Structural damping

We can observe that decreasing the concrete compressive or tensile strengths by a factor of 20% does not modify significantly the maximum displacement and post-damage oscillation frequency of the slab (less than 10% difference):

The observation is quasi the same concerning the Poisson ratio:
We can conclude that the blind calculation made in 2010 is not very sensitive to the main concrete parameters of the model. As a consequence, we feel that it is necessary to continue the analysis with the “numerical” parameters of the GLRC_DAMA material law.

### 3.3.2 Sensitivity study on the specific “GLRC_DAMA” parameters

In this section, we study the sensitivity of the displacement results (sensor n°1) to the specific parameters of the material law.

The table below gives a comparison between the values used in the 2010 blind calculation and the ones which are used in the 2012 sensitivity studies:

<table>
<thead>
<tr>
<th></th>
<th>2010 blind calculation</th>
<th>2012 sensitivity study</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Prager coefficients in membrane</strong> C_{inj}</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Dependant on the elastic concrete and steel main parameters (young’s modulus, Poisson ratio, concrete thickness and effective area of reinforcement)</td>
<td>2.16e6 N/m</td>
<td>1.8e6 N/m</td>
</tr>
<tr>
<td><strong>Prager coefficients in bending</strong> C_{imj}</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Function of the plastic slope of the moment-curvature curve</td>
<td>4.18e4 N.m/m</td>
<td>0.0 N.m/m</td>
</tr>
<tr>
<td><strong>Ratio Qp</strong> (slope after)</td>
<td>0.145</td>
<td>0.1</td>
</tr>
</tbody>
</table>
In the calculations whose results are presented below, all the other numerical data are identical to the ones defined in the 2010 analysis:

- Concrete properties of the slab
- Steel properties of the rebars
- Force-time history curve for the missile
- Boundary conditions of the slab
- Structural damping

**CiNi parameter**

The figure below highlights that decreasing the parameter CiNi by 20% does not modify significantly the global behavior of the slab, in particular its maximum displacement and its post-damage oscillation frequency (≈ 5 % difference). This observation allows to say that the membrane effects are not very important in these flexural tests:

A modification of this parameter could physically be linked to a modification of an elastic characteristic of concrete and/or steel, for example the Young’s modulus.

**CiMi parameter**

The figure below evidences the influence of setting this parameter to zero. From a physical point of view, it amounts at considering a plastic slope equal to zero (no strain hardening for steel properties). As expected, if we do not consider the strengthening in the rebars due to plastic deformation, the behavior of the slab is less rigid: the maximum displacement increases (≈ 10 %) and the damage too. Indeed we can
observe an little decrease of the stiffness of the slab, compared with the blind calculation, materialized by a 5% difference in the post-damage oscillation period:

$$Q_p$$ parameter

The figure below evidences that decreasing the parameter $$Q_p$$ by 30% modify significantly the global behavior of the slab, in particular its maximum displacement ($\approx 40\%$ increase) and its post-damage oscillation frequency ($\approx 20\%$ decrease):

From a physical point of view, it amounts at decreasing the elastic properties of steel and/or increasing the elastic properties of concrete (for example their Young’s modulus). This parameter can also be linked to:

- The behavior of cracked concrete before the plastification of the rebars
- The concrete/steel bonding

3.3.3 Sensitivity study on damping parameters

In the various analyses which have been done in the previous sections, we noted that one parameter allowed us to fit the correct post-damage frequency (“$$Q_p$$”). In this section, we analyze the sensitivity on the residual displacement to the damping parameters.

In the blind calculation, we introduced a 2% mass proportional damping $$[C] = \alpha[M]$$ on the 45 Hz frequency (cf. paragraph 3.2), during all the calculation. On the experimental curves presented in section 3.2.3.1, we observed that the damping is important between
the first and the second displacement peaks (about 10 % to 12 %), and equal to 2.5 % on the shaking phase of the structure.

In this section, we make a calculation with a 10 % mass proportional damping, which allows to fit the experimental residual displacement:

![Graph showing experimental data and calculations](image)

3.3.4 2012 final calculation

In this section we make a final calculation, based on the fitted values we studied in the previous sections (GLRC parameters and damping), and on the concrete data provided by the IRIS_2012 committee:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>2010 blind calculation</th>
<th>2012 final calculation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Unconfined compressive strength [MPa]</td>
<td>63.9</td>
<td>66.93 [5]</td>
</tr>
<tr>
<td>Young's modulus [MPa]</td>
<td>26 915</td>
<td>29 670 [5]</td>
</tr>
<tr>
<td>Poisson's ratio</td>
<td>0.2</td>
<td>0.223 [5]</td>
</tr>
<tr>
<td>CiMj parameter [N.m/m]</td>
<td>4.18*10e4</td>
<td>0.0</td>
</tr>
<tr>
<td>Qp parameter</td>
<td>0.145</td>
<td>0.09</td>
</tr>
<tr>
<td>Damping</td>
<td>2 %</td>
<td>10 %</td>
</tr>
</tbody>
</table>

The tri-axial test results on concrete samples [5] were not used in the 2012 final calculation, mainly because our methodology is based on a thick shell mesh coupled with a global material law for reinforced concrete. As a consequence, the behaviours of concrete and steel in tension can not be dissociated (homogenization of the material behaviors). In compression, the effect due to the confinement seems not to be much important, because the damage in compression is low in these flexural tests (maximum strain $\approx 2.5 \%$).

The calculation time period is 100 ms.

The calculations were done with the sequential version of the EUROPLEXUS finite element code.

We give below the time characteristics of the calculation:

- Time step: $7.18 \times 10^6$ sec
On the figure given below (displacement at sensor n°1), we observe that the post-damage oscillation frequency and the residual displacement of the slab had been correctly fitted, much more than in the 2010 blind calculation:

4. Punching tests P1, P2 and P3

4.1 IRIS_2012 : P1-2&3 analysis

4.1.1 Description of the methodology

For the IRIS-VTT punching tests, the methodology used for bending (decoupling projectile and slab behaviors with Riera formula, and applying a force-time history curve on the structure meshed with thick shell elements) can not be re-employed. Indeed, the missile is a very hard penetrator, and contact actions and target reactions are strongly coupled. As a consequence, the local phenomena in the slab thickness are not correctly reproduced with this kind of model. In this case, we need a more detailed model (3D solid approach) combined with a damage law for concrete, and with the missile modeling.

As this kind of calculation is not at the moment an industrial practice at EDF, we based our analysis on empirical considerations.

In this section, we study the sensitivity on the residual velocity of the missile to some parameters of a usual empirical formula.

In a first part we have to calculate the just-perforation velocity of the missile \( V_p \), with a given empirical formula. We decide to use the CEA-EDF formula modified by Fullard [6] :

\[
V_p = 1.3 \rho_c^{1/6} f_c^{0.5} \left( \frac{pH_0^2}{\pi M} \right)^{2/3} (r + 0.3)^{0.5}
\]

Where :

- \( \rho_c \) is the concrete density 2500 kg/m^3
- \( f_c \) is the concrete compressive strength
- \( p = D^*\pi \) is the perimeter of the missile, with \( D = 0.168 \) m its diameter
- Ho is the thickness of the slab 0.25 m
- M is the mass of the missile 47.5 kg
- r is the reinforcement ratio each way each face ≈ 0.35 % for this slab

The validity domain of the formula is respected in this case, except for the concrete compressive strength, which is often limited to 45 MPa [8].

The hypothesis which is made here is that the dissipated energy during perforation is not dependant on the impact velocity of the missile.

In a second part, we use the just-perforation velocity of the missile $V_p$ to calculate its residual velocity $V_R$ after perforation, by (energy balance):

$$V_R = \sqrt{\frac{V_o^2 - V_p^2}{1 + \frac{M_c}{M}}}$$

Where :
- $V_o$ is the impact velocity of the missile 136 m/s
- $M_c$ is the mass of the concrete plug, dependant on the angle of the punching cone cracks $\theta$

$$\theta = \frac{45^\circ}{\left(\frac{120^\circ}{45^\circ}\right)} \leq 60^\circ \quad [3]$$

### 4.1.2 Sensitivity study on the concrete compressive strength

In this section we calculate the just-perforation velocity of the missile with the EDF-CEA formula modified by Fullard, as a function of the concrete compressive strength.

Then we calculate its residual velocity as a function of the concrete compressive strength, for a single cracks angle ($\theta=40^\circ[3]$) :

As we can see, the residual velocity of the missile after perforation is very sensitive to the concrete compressive strength we use in the formula.
If we use the compressive strength measured on a cubic sample, on a cylinder sample, if we take into account the increase of the mechanical properties due to ageing or strain rate effect, the calculated residual velocity can change significantly, as we can see on the table given below:

<table>
<thead>
<tr>
<th>Position on the figure</th>
<th>Concrete compressive strength</th>
<th>Residual velocity obtained with CEA-EDF formula</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Cubic sample at 43 days (hall conditions) – P2&amp;P3</td>
<td>64.7 MPa [1]</td>
</tr>
<tr>
<td>2</td>
<td>Cylinder sample at 43 days (hall conditions) – P2&amp;P3</td>
<td>57.4 MPa [1]</td>
</tr>
<tr>
<td>3 (cf. figure)</td>
<td>Cubic sample at 56 days (100 % humidity) – P2&amp;P3</td>
<td>68.4 MPa [1]</td>
</tr>
<tr>
<td>4 (cf. figure)</td>
<td>Cubic sample at 27 days (hall conditions) – P1</td>
<td>67.1 MPa [2]</td>
</tr>
<tr>
<td>5 (cf. figure)</td>
<td>Cylindrical sample at 27 days (hall conditions) – P1</td>
<td>60.0 MPa [2]</td>
</tr>
<tr>
<td>6 (cf. figure)</td>
<td>Validity range of the formula</td>
<td>45 MPa [8]</td>
</tr>
</tbody>
</table>

4.1.3 Sensitivity study on the angle of the punching cone cracks

In this section, the just-perforation of the missile is calculated with a given concrete compressive strength (60.0 MPa), and we calculate its residual velocity as a function of the angle of the punching cone cracks:

![Diagram of punching cone cracks](image)
We observe that the definition of this angle have a significant influence on the residual velocity of the missile, as we can see on the table given below:

<table>
<thead>
<tr>
<th>Position on the figure</th>
<th>Angle of the punching cone cracks</th>
<th>Residual velocity obtained with CEA-EDF formula</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 (cf. figure)</td>
<td>$\theta = 45^\circ (t_w/D)^{1/3} \approx 40^\circ$ defined by [3]</td>
<td>44 m/s</td>
</tr>
<tr>
<td>2 (cf. figure)</td>
<td>$20^\circ$ (experimental observation)</td>
<td>55 m/s</td>
</tr>
<tr>
<td>3 (cf. figure)</td>
<td>$70^\circ$ (experimental observation)</td>
<td>23 m/s</td>
</tr>
</tbody>
</table>

A detailed analysis on this parameter seems to be necessary, all the more so as it is not clearly observable in the tests. In the figures given below, we can see a first phase with a deep penetration in the concrete (the cracks angle is small, we have a quasi straight rupture) and a second phase where the cracks angle increase significantly:

![Image of a fracture in concrete]
This remark is valid for the three tests, in particular because there is no transverse reinforcement in the slab. If there was transverse reinforcement, we may have fewer uncertainties on the cracking angle of the cone plug.

If we combine the uncertainties on the cracking angle and on the concrete compressive strength, we obtain a range of residual velocities included between 10 m/s and 70 m/s :

![Graph showing residual velocities vs. angle of punching cone cracks]

We can make another comment and say that if we had use another kind of empirical formula, the range of the calculated residual velocities would increase :
Concrete compressive strength & Angle of the punching cone cracks & Residual velocity of the missile

<table>
<thead>
<tr>
<th></th>
<th>Concrete compressive strength</th>
<th>Angle of the punching cone cracks</th>
<th>Residual velocity of the missile</th>
</tr>
</thead>
<tbody>
<tr>
<td>CEA-EDF formula [6]</td>
<td>60.0 MPa</td>
<td>$\theta = 45^\circ(t_w/D)^{1/3} \approx 40^\circ$ defined by [3]</td>
<td>44 m/s</td>
</tr>
<tr>
<td>Modified NDRC [6]</td>
<td>60.0 MPa</td>
<td>$\theta = 45^\circ(t_w/D)^{1/3} \approx 40^\circ$ defined by [3]</td>
<td>67 m/s</td>
</tr>
</tbody>
</table>

The list of parameters studied in this section is not exhaustive, and the following ones may have a significant influence on the residual velocity calculated with empirical formulas:

- Deformability of the missile (perfectly rigid or not)
- Nose shape of the missile
- Penetration thickness in the concrete, before perforation (cf. figures above showing the quarter cut slabs of P1-2&3 tests), which can also have an influence on the calculated scabbed area:

As a consequence, such an analysis seems to be necessary even when we use a simple approach.

5. Conclusion on the IRIS_2012 calculations

In accordance with the IRIS_2010 recommendations, we will say that making various sensitivity studies is an essential step before doing a Finite Elements calculation. This kind of studies must focus on:

- The material parameters, for which we may have some uncertainties associated with:
  - The measures (kind of sample used, scattering…)
  - The modification of their properties due to the kind of loading applied (uniaxial, tri-axial…), to the dynamic effects (strain rate), to the ageing…
- The numerical parameters, specific to a FE software or to a material law, which can have a significant influence on the results, without having necessarily a physical signification

The more complex is the modeling, the more difficult to make is this step.
In this document we highlighted that the maximum displacement of the slab and the maximum strain in the rebars are not the only important values that a numerical calculation must fit. In the field of impacts, the vibratory response of the structure after the shock is as important as its local response. That is why fitting the experimental post-damage oscillation frequency, structural damping and residual displacement is necessary.

Finally, we feel that making sensitivity studies is an important step even if we use simple approaches, like empirical formulas for perforation. First of all, the range of this kind of formulas is large in the literature, and they provide results which can be very different from one to another. What's more, they are very dependent on the various parameters which make them up (concrete properties, rupture mode of the slab which is rarely easy to assess, deformability of the missile, etc...).
A III.8 Team #11 ENSI

IRIS_2012 NUMERICAL SIMULATION REPORT
Team ENSI (ENSI¹, Principia², SPI³)

¹Swiss Federal Nuclear Safety Inspectorate ENSI, Brugg, Switzerland
²Principia Ingenieros Consultores, Madrid, Spain
³Stangenberg und Partner Ingenieur-GmbH, Bochum, Germany

1. Goal

This document is a summary of methods, assumptions, and overall results from the SOFiSTiK and Abaqus codes applied to the IRSN-VTT flexural test B1 and punching test P1, respectively. The work described in the present report is a result of a collaboration of ENSI, Principia, and SPI.

2. Introduction

Within the benchmark project, numerical calculations were carried out by SPI for the IRSN-VTT flexural test B1 and by Principia for the IRSN-VTT punching test P1. SPI had participated in IRIS_2010. Principia had already collaborated with SPI in the past, specifically in the numerical modelling of rigid impacts against reinforced concrete walls. Motivated by that experience, SPI invited Principia for investigating some of the issues of IRIS_2012.

3. Material input data

The material input data were introduced as they were provided in the benchmark IRIS_2012. Strain-rate effects have not been taken into account.

4. Basic choices

The program utilized for the non-linear dynamic FE analysis of flexural test B1 is SOFiSTiK, cf. SOFiSTiK AG (2010). The Abaqus (SIMULIA, 2011) concrete damaged plasticity model was the one adopted for the present punching test P1 calculations.

5. Concrete slabs modelling

IRSN-VTT flexural test B1
As finite element for modelling the r/c plates for the soft impact computation, a shell element is used based on a layered concrete model regarding the reinforcement at both sides. Shear deformations of shell/plate elements are approximately included in SOFiSTiK. The elements shear forces are limited by the ultimate shear resistance specified with respect to the punching resistance of the concerning r/c structure. Damping was introduced by Rayleigh parameters adjusted to 1 % of critical damping for the relevant frequency range 30 – 80 Hz. The FE model is shown in Fig. 1. The total system of r/c plate, steel frame and back pipes of the VTT flexural tests has been considered in a coupled model. The used mesh size for the r/c plate is 50 x 50 mm.
IRSN-VTT punching test P1

For the P1 Abaqus computation, a decision was made to incorporate two of the symmetries displayed by the problem, thus reducing the model to one quarter of the actual configuration. The global geometry of the model, taking into account that symmetry, appears in Fig. 2. As apparent in the figure, the reinforcing bars are included in the model.

Based on the geometry described, a finite element model was generated to represent the concrete, the reinforcing bars and the missile, which was considered to be undeformable. A detail of the finite element mesh can be seen in Fig. 3, which indicates the level of refinement employed in the discretisation. The concrete near the outer edge, which was reinforced with steel members, has simply been taken to perform elastically, which is the reason why it appears with a different colour in Fig. 2 and Fig. 3.
6. Missiles modelling

For flexural test B1, the missiles have not been modelled explicitly. Instead, the loading function has been derived by an improved Riera method. As it has already been mentioned, the missile was considered to be undeformable for the punching test P1.

7. Contact modelling

This item does not apply for flexural test B1, since the impact is considered by use of load-time functions. For the case of the punching test P1 the general contact algorithm in Abaqus/Explicit including initial internal surfaces contactable after eroding has been defined.

8. Calculation

Concrete Tests – Concrete Material Model in Abaqus

For those more demanding situations, Abaqus (SIMULIA, 2011) has a number of models of varying complexity that might be suitable for modelling the behaviour of mass concrete, with the reinforcing bars embedded in the concrete:

- **Smeared cracking** (available only in the implicit solver Abaqus/Standard).
- **Brittle cracking** (available only in the explicit solver Abaqus/Explicit).
- **Concrete damaged plasticity** (available in both solvers). It is a more recent and robust material model, intended to provide a general capability for the analysis of concrete structures. It is described in the following section. This model was the one adopted for the present calculations.
- **Drucker-Prager cap model** (available in both solvers).

The details of the available models are described in the program manuals, but specific attention will be paid here to the damaged plasticity model, which is considered to be the more appropriate one for investigating the problems of interest.

This material model provided a general capability for the analysis of concrete structures under monotonic, cyclic, and/or dynamic loading. It includes a scalar (isotropic) damage model with tensile cracking and compressive crushing modes of failure. The model...
accounts for the stiffness degradation mechanisms associated with the irreversible damage that occurs during the fracturing process.

Fig. 4 shows the response of the model under monotonically increasing uniaxial strains; the upper graph in the figure presents the expected brittleness of the tensile response, while the lower graph describes the behaviour under uniaxial compression. The post-failure response is controlled by a fracture energy criterion. Due account is taken of the effects of multiaxial compression, as shown by the yield surface is also presented in the Fig. 4, which corresponds to a plane stress situation.

![Fig. 4 Response to uniaxial loading in (a) tension, (b) compression and (c) multiaxial](image)

It is also interesting to note the cyclic behaviour of the model, which is depicted in Fig. 5. It can be observed that unloading from a post-peak situation does not necessarily return the load path to the origin, giving rise to some permanent strains. This point will be recalled later on when discussing some of the results obtained.

![Fig. 5 Uniaxial load cycle](image)
Nevertheless, the cyclic response is of only lesser importance in cases such as those considered here, which essentially consists of a single monotonic loading followed by the corresponding unloading. The situation could potentially be rather different if the problem being studied was one of high earthquake demands instead of impact.

The model actually used in the simulations is not exactly that described in the Abaqus manuals as the damaged plasticity model, which Principia has slightly modified for what we believe to constitute a better representation of the response as commented in the following section.

A number of tests were provided for calibration of the parameters of the concrete model (IRSN, 2012). Some general parameters are initially provided:
- Density: 2273 kg/m$^3$
- Young’s modulus: 29.7 GPa
- Poisson’s ratio: 0.223

Results of a uniaxial unconfined compression test, depicting both the longitudinal and transverse strains induced at each load applied during the test, where provided by the benchmark specification. And also similar information for triaxial tests performed at progressively increasing levels of confining pressure: 15.5, 26, 47 and 100 MPa (respectively specimens 4, 7, 8 and 9).

The concrete damaged plasticity model uses a non-associated flow potential. The flow potential $G$ used for this model is the Drucker-Prager hyperbolic function, which neglecting damage can be approximately expressed as:

$$G = q - p \tan \psi$$  \hspace{1cm} (1)

where:
- $q$ is the von Mises stress,
- $p$ is the pressure invariant of the stress tensor, and
- $\psi$ is the dilation angle.

For triaxial tests it can easily be shown that the dilation angle can be obtained from:

$$\tan \psi = \frac{3}{2} \frac{2\nu_p - 1}{1 + \nu_p}$$  \hspace{1cm} (2)

where:
- $\nu_p$ is the apparent Poisson’s ratio for yielding.

When the above calculations were conducted, using the results of the triaxial tests performed under confinements of 15.5, 26, 47 MPa, the average value obtained for the dilation angle was 30º. Fig. 6 gathers the stress-strain responses observed in all the previous tests, which allows observing how the increasing levels of confining pressure are responsible for increasing the peak stress and reducing the post-peak softening of the response. The strength characteristics observed in the tests at different confining pressures can also be plotted by means of the corresponding Mohr circles. This has been done in Fig. 7 and the resulting friction angle turns out to be $\phi = 30.8^\circ$. 
The yield condition for undamaged concrete tends to be defined for high confinement pressures by the surface:

\[ F = q - 3\alpha \varphi + \gamma \sigma_{\text{max}} \]  \hspace{1cm} (3)

where: \( \alpha \) is a material parameter related with the ratio of the biaxial stress \( \sigma_b \) to the uniaxial stress \( \sigma_c \) through

\[ \frac{\sigma_b}{\sigma_c} = \frac{1 - \alpha}{1 - 2\alpha} \]  \hspace{1cm} (4)

\( \sigma_{\text{max}} \) is the maximum principal stress (being negative for compression), and \( \gamma \) is defined by

\[ \gamma = \frac{3(1 - K_c)}{2K_c - 1} \]  \hspace{1cm} (5)

where \( K_c \) is a material parameter between 0.5 and 1.0. A value of 0.75 will be assumed here.

It can also be shown that \( \alpha \) is related with the former friction angle \( \varphi \) through:

\[ \alpha = \frac{3K_c + K_c \sin \varphi + \sin \varphi - 3}{6K_c + \sin \varphi - 2K_c \sin \varphi - 3} \]  \hspace{1cm} (6)
Thus the value $\alpha = 0.117$ is obtained and the ratio of the biaxial to the tensile compressive stress is 1.15.

On the other hand, going back to equation (3), if the effects of the effective compressive cohesion stress $\sigma_c$ are not neglected, the actual yield condition can be expressed as:

$$F = q - 3\alpha p + \gamma \sigma_{\text{max}} + \sigma_c(\varepsilon_p)$$  \hspace{1cm} (7)

The compressive cohesion stress, as resulting using the above equation ($F = 0$) for each of the five reference tests supplied, has been plotted in Fig. 8. To allow a good calibration of all these results, the compressive cohesion stress has been defined as a function not only of the equivalent plastic strain but also of the maximum principal stress.

Additionally, the Brazilian tests conducted indicated that the tensile strength of the concrete is approximately 4 MPa. No fracture energy was provided, but the value typically ranges between 40 N/m for a concrete with 20 MPa compressive strength and 120 N/m for one with 40 MPa. When this information is extrapolated to a concrete with a compressive strength of 69 MPa, the fracture energy obtained is 228 N/m. This is the value adopted for the calculations, though it should be clarified that this parameter has only a moderate effect in the results.

Although some information was also supplied on the strain rate dependence of the material behaviour, this dependence was not considered to be sufficiently significant and hence the decision was made not to incorporate it in the constitutive description.

**Simulation of the tests**

Having determined all the parameters that enter the constitutive model adopted for the concrete, the various calibration tests were reproduced to evaluate the consistency with the observations.

The results of the simulations of the tests conducted at the five different levels of confining pressure have been plotted together with the experimental data in Fig. 9, where the solid lines represent the analytical results and the dots indicate the experimental values.

As can be seen in the figure, the match is not perfect since some deviations appear, particularly at the higher end of the range of confining pressures tested. However, for the purposes of the present exercise, this representation was considered sufficient and was the one finally adopted for simulating the effects of the punching tests.

![Fig. 8 Compressive cohesion stress](image-url)
Fig. 9  Comparison between tests and concrete material model

IRSN-VTT flexural test B1
The time step of the computation is 0.05 ms, the duration of calculation is 100 ms. The load-time function used for the computation is an idealized function derived from Riera model. In addition, a load-time function in the sense of contact forces between missile and target derived by GRS within an explicit simulation with ANSYS AUTODYN, see contribution in IRSN (2011), was used, cf. Fig. 10. Both load functions led to a good agreement of measured and calculated load duration and of measured and calculated displacement and strain time histories, cf. Fig. 10 through Fig. 12. The influence of the oscillations of the GRS load function on the computed results, which is correlated with local buckling of the missile in comparison with the Riera load function, obviously is small.

Fig. 10  Flexural test B1, time histories of load functions (left) and displacements (right, centre of the slab)
IRIS_2012  Team ENSI (ENSI, Principia, SPI)

Fig. 11  Flexural test B1, time histories of concrete strain gauge 2 (left) and steel strain gauge 3 (right)

Fig. 12  Flexural test B1, time histories of steel strain gauges 4 (left) and 5 (right)

**IRSN-VTT punching test P1**

In order to avoid excessive and unrealistic distortions, the elements are removed from the mesh when the equivalent plastic strain reaches 0.3. This is accomplished with an Abaqus user’s subroutine. Note that the stiffness contribution of the concrete when plastic strains exceeds about 0.13 becomes negligible, but a premature deletion of elements may produce inadequate results.

For the reinforcing steel, an elastoplastic model was used, based on the properties supplied by the organisers. In the course of the impact a number of contacts take place between the various materials involved. The coefficient of friction was again taken to be 0.3 at all such interfaces.

Under the conditions of the punching test the missile clearly perforates the concrete wall. The resulting situation, as determined by the simulations at 10 ms after the onset of impact, is presented in Fig.13. In that figure, the large displacements that appear in the back face of a seemingly solid wall should not be taken literally; they are simply the result of the inconvenience of deleting concrete elements in a situation in which the reinforcement of the back face loses its concrete cover.

An outer view of the distribution of crushing strains appears in Fig. 14. However, this figure may be somewhat less useful because, as already mentioned, the outer concrete
cover is lost in some areas; as a consequence Fig. 15 was prepared, which depicts similar information following removal of the outer cover from the mesh.

The cracked zones determined by the calculations and observed in the actual test can be visually compared in Fig. 16 and Fig. 17. Although the comparison is rather of a qualitative nature, it is clear that the simulation leads to very realistic results for the cracking configurations and extents developed in both the front and back faces of the wall. The effects produced in the reinforcing bars are shown in Fig. 18. The yielding of rebars is fairly localised in the front face and less so in the back face; large deformations and failure of rebars occurs as the missile perforates the concrete wall.

Finally, the kinetic energy of the emerging missile affords a very relevant comparison between the simulation and test results. As can be seen in Fig. 19 (red curve), the energy dissipated in the test while perforating the concrete wall was 408 kJ, while that determined by the calculations was 358 kJ. The approximation is considered very reasonable, particularly when taking into account that the materials have been assumed to go suddenly to zero stress at the end of the stress-strain curves provided as reference; any elongation of those curves, or a more gradual decrease towards zero, would lead to a somewhat greater dissipation of energy during the perforation process.

Additionally, a conversion from finite element to smoothed particle hydrodynamic methods (SPH) has been activated. This is a new technique supported in Abaqus version 6.12. SPH is a fully Lagrangian modeling scheme permitting the discretization of a prescribed set of continuum equations by interpolating the properties directly at a discrete set of points distributed over the solution domain without the need to define a spatial mesh. In our analysis each conventional brick element is automatically converted to eight SPH particles when the equivalent plastic strain reaches 0.2. The resulting deformed configuration with contours of equivalent plastic strains at 10 ms is shown in Fig. 20. The kinetic energy of the missile is also presented in Fig. 19.

Overall, as a conclusion from the simulation exercise conducted, it can be stated that the constitutive model adopted for the concrete and the material parameters determined from the tests conducted on laboratory samples have allowed performing a very realistic simulation of the punching test and its effects on the target wall.
Fig. 14 Crushing strains after the punching test

Front  Back

Fig. 15 Crushing after the punching test below the concrete cover

Front  Back

Fig. 16 Crack zones after the punching test (front)
Fig. 17  Crack zones after the punching test (back)

Fig. 18  Plastic strains in rebars after the punching test (-)

Fig. 19  Kinetic energy history of the missile during punching
9. Conclusion

IRSN-VTT flexural test B1

The non-linear dynamic analysis of a plate model in connection with a simple Riera load function carried out with the program SOFiSTiK is well-suited for the numerical simulation of impact tests on r/c structures with dominating bending behaviour.

IRSN-VTT punching test P1

Based on the work conducted the following conclusions and recommendations can be offered:

a) The damaged plasticity model available in Abaqus has proved to be a very adequate model for describing concrete behaviour in the laboratory tests, the bending test, and the punching test. It is recommended that this model be adopted for further work in the context of IRIS_2012.

b) The modification introduced by Principia in the damaged plasticity model, which consisted in making the compressive cohesion stress to depend also on the maximum signed principal stress, has also been shown to be successful.

c) The model used is considered to be particularly suitable for primarily impulsive situations, such as contemplated in IRIS_2012, in which the concrete is subjected to an essentially monotonic loading path; the unloading that subsequent follows has only a minor influence on the results.

The differences observed between the calculation and test results (wall centre displacements upon unloading in the bending test, kinetic energy of the emerging missile in the punching test) are relatively minor and can be traced to known features of the model, which could be improved if warranted. The computed residual velocity using conventional
elements is 58 m/s and for the SPH model it is 43 m/s, while the measured residual velocity was 34 m/s.

10. References


INTRODUCTION - LESSONS LEARNED FROM IRIS-2010

The authors took part in the IRIS-2010 blind simulation exercise for the flexural and punching mode tests (Ref. 1). The approach was to use experimental results from the Meppen II-4 test (Ref. 2 and 3) for calibration of a general methodology that was then exported to the new tests.

Methodology was based on simulation with ABAQUS/Explicit (Ref. 4), with a 3D finite element representation of missile and target. Thin shell and solid elements were used to represent the missile for the flexural test and the punching test, respectively. On the other hand, a segregated approach was used to represent the reinforced concrete of the targets. The concrete was introduced by means of solid elements, whereas the rebars were represented as separate beam elements. The beam elements did not introduce additional nodes. They shared the nodes with the solid elements representing the concrete. Therefore, perfect bonding between concrete and rebars was assumed. The pattern of elements representing the rebars tried to match the actual pattern of rebars, without grouping bars.

Constitutive modeling of the steel at the missiles was based on a conventional von Mises elastic-plastic model, with plasticity and strain rate dependence according to the Cowper-Symonds model. The reinforcing steel at the targets was also modeled using von Mises plasticity, using the same rate dependence as given for the rebars in the Meppen test.

For modeling of the concrete, a Drucker-Prager plasticity model with isotropic hardening was used. It was thought that the behavior of the concrete during impacts was going to be governed mainly by cracking. Hence, plastic softening was chosen based on uniaxial tensile strength parameters. Note that in the classical Drucker-Prager model the hardening-softening behavior is governed by a single parameter. The model uses a single yield surface, both for tension and compression.

When the results of the tests were made available, comparison with blind predictions showed that performance of the missiles was predicted relatively well. However, experimental results for the target slabs yielded significant differences with respect to predictions. In the case of the flexural test, the response was predicted only qualitatively, computed displacements and strains were significantly smaller than those measured in the test. In the case of the punching test, simulation totally failed to predict the penetration of the missile.

Careful examination of the computed stress states in the concrete around the impacted area showed unrealistically high compression levels which were not adequately limited by the Drucker-Prager model. This was therefore believed to be the root cause of too stiff a response and the reason also of the failure to predict penetration in the punching test.

For the present IRIS-2012 exercise, the same general simulation methodology of IRIS-2010 has been maintained. The only exception is that the concrete constitutive model has been changed to the ABAQUS Concrete Damaged Plasticity model, which is based on the work by Lubliner et al (Ref. 5), as complemented by Lee and Fenves (Ref. 6). In this model, the evolution of the yield surface is controlled by two hardening variables (i.e. not just one as in the Drucker-Prager model). The two variables are linked to the failure mechanisms under tension and compression loading, respectively.

This constitutive model still has some limitations. The most significant is that softening-hardening behavior under compression cannot be varied as a function of the hydrostatic pressure (first stress invariant). However, it represents a significant modeling improvement with respect to the Drucker-Prager model.
MATERIAL INPUT DATA

Steel

Regarding steel, input information provided for the benchmark corresponds to what is usually available in a regular design environment. The authors have missed information about strain rate sensitivity of mechanical properties. The Cowper-Symonds model has been used to introduce the strain rate sensitivity of the yield stress at the missile steel:

\[
\sigma_y = \sigma_{y0} \left( 1 + \frac{\dot{\varepsilon}}{D} \right)
\]

Where \( \dot{\varepsilon} \) is the strain rate and \( q \) and \( D \) are material constants.

Parameters \( q \) and \( D \) have been taken from the literature (Ref. 8):

\[ q = 10, \quad D = 100 \text{ s}^{-1} \]

for the EN 1.4432 stainless steel of the flexure test missile.

For concrete rebars, strain rate sensitivity given for the Meppen test in IRIS-2010 has been extended to the other tests.

Concrete

With respect to IRIS-2010, very detailed information about the concrete has been supplied, courtesy of IRSN. The purpose was to reduce epistemic uncertainties. Results of a series of triaxial tests of concrete cylinders and the results of a Brazilian test have been given by the organizers. These results have been used to adjust material model parameters.

CONCRETE SPECIMENS FOR TRIAXIAL AND BRAZILIAN TESTING

For simulating the tests involving plain concrete cylinders, 3D reduced integration solid finite elements have been used (C3D8R elements of ABAQUS). Figure 1 gives a general view of the meshes. There are 24 elements in a diameter and 30 elements in the length. Note that softening behavior in tension and compression can lead to strong localization of deformation in plain concrete models. In the absence of special purpose elements within the ABAQUS/Explicit element library, reduced integration elements are used together with Hillerborg regularization based on fracture energy in the concrete constitutive model.

Figure 1. Finite element models for Brazilian and triaxial tests simulation
CONCRETE SLAB MODELLING

For the two impact tests, a 3D finite element representation of missile and target has been used. It is assumed that the deformation and failure mode will have two orthogonal planes of symmetry and, consequently, only 1/4 of the missile and target are represented. Figure 2 gives a general view of the finite element models.

Figure 2. General view of finite element models for impact test simulations
Supporting frames are not represented in the models. At the points or lines of support of the target, both at front and rear faces, velocities in the direction of the support are set to zero. On the other hand, the corresponding translational and rotational velocities are set to zero at the planes of symmetry.

A segregated approach has been used to represent the reinforced concrete: the concrete is introduced by means of solid elements, whereas the rebars are represented as separate beam elements. The beam elements do not introduce additional nodes. They share the nodes with the solid elements representing the concrete. Therefore, perfect bonding between concrete and rebars is assumed. The pattern of elements representing the rebars tries to match the actual pattern of rebars, without grouping bars.

For the IRSN-VTT flexural mode test, the mesh is the same as in the IRIS 2010 exercise. A total of 59290 eight-node reduced integration solid elements are used to represent the target (C3D8R in ABAQUS). As before, the size of the elements has been selected to have 10 elements through the thickness, which gives a maximum size in the order of 1.5 cm. A total of 66924 nodes have been used to represent the target. The rebars are represented using 8740 two-node shear deformable beam elements (B31 in ABAQUS). The beam elements do not introduce additional nodes.

For the IRSN-CNSC-VTT punching mode test, the mesh of the slab has been somewhat refined with respect to the IRIS 2010 round robin. A total of 96276 eight-node reduced integration solid elements are used to represent the slab (C3D8R in ABAQUS). The size of the elements has been selected to have 17 elements through the thickness, instead of 10 (IRIS 2010), which gives a maximum size in the order of 1.5 cm. A total of 104706 nodes have been used to represent the target. The rebars are represented using 3540 two-node shear deformable beam elements (B31 in ABAQUS). The beam elements do not introduce additional nodes.

Behaviour of steel in the rebars is represented using a von Mises plasticity model with strain rate dependence taken from Meppen tests and linear or bilinear hardening (figure 3). Yield stress and hardening modulus at different strain rates are computed from the material input data. Note that log strain-true stress laws are used.

![Figure 3. Stress-strain law for reinforcing steel](image)

The concrete behaviour is represented using the concrete damaged plasticity (CDP) model of ABAQUS (Ref. 4 and 9). The CDP model is a modification of the Drucker-Prager strength criterion in order to accommodate different hardening-softening behavior in tension and compression (Ref. 5 and 6). It is a continuum, plasticity-based damage model for concrete. It assumes that the main two failure mechanisms are tensile cracking and compressive crushing. The evolution of the yield surface is controlled by two hardening variables, linked to failure mechanisms under tension and compression, respectively. Additionally, two damage variables, one in tension and one in compression, are introduced to account for degradation of elastic stiffness during loading and unloading cycles. The constitutive relations for elasto-plastic responses are decoupled from the degradation damage responses. Consequently, by setting the damage evolution to zero, the model can be used to simulate a purely elasto-plastic response.
Figure 4 shows a schematic view of the yield surface of the CDP. Note that the model allows for dependence on the third stress invariant, since the shape of the deviatoric cross section of the surface can be varied from circular \((K_c = 1)\) to triangular \((K_c = 0.5)\). Note also that two softening curves are needed to define post-yield behavior, one in tension and one in compression. One of the model parameters is the dilation angle \((\psi)\), which allows for non-associated plastic flow.

After calibration with the given experimental information, the following parameters have been used for feeding the concrete CDP model used for the target slabs:

- \(E_0\) = initial Young’s modulus = 50.88 GPa (secant modulus at 0.4 \(\sigma_{cu}\))
- \(v\) = Poisson’s ratio = 0.223
- \(\sigma_{cu}\) = compressive strength = 69 MPa (hardening curve using Sargin model with \(A=2, D=1\))
- \(\sigma_{to}\) = tensile strength = 4.04 MPa (softening based on Hillerborg regularization with fracture energy \(G_f = 150 \text{ N/m}\))
- \(\psi\) = dilation angle = 40°
- \(\varepsilon\) = flow potential eccentricity = 0.30 (smoothing of flow potential surface near the vertex)
- \(\sigma_{cc}/\sigma_c\) = biaxial/uniaxial strength ratio = 1.16
- \(K_c\) = shape factor for deviatoric plane cross section = 0.72

Since no cycles of loading-unloading are expected, the damage part of the constitutive model has been deactivated (i.e. damage variables assumed to be zero all over the analyses).

Figure 5 shows the uniaxial tension and compression stress-strain curves used for the concrete in the analyses. These curves are used for expansion or shrinkage of the yield surface due to hardening-softening.

**MISSILE MODELLING**

For the IRSN-VTT flexural test, the missile is modeled using four-node reduced integration shell elements \((S4R\text{ in ABAQUS})\), with seven integration points through the thickness. The size of the elements has been selected in order to have at least 4 elements in each fold of the shell when it collapses in a concertina mode.
The size of the folds has been estimated using the approximate formula given in the book by W. Johnson (Ref. 8). For the IRSN-VTT flexural test, the resulting size is 6 mm (figure 6). A total 12112 four-node reduced integration shell elements (S4R in ABAQUS) and 12498 nodes are used.

![Concrete tenstile stress-crack displacement law](image1)

![Concrete compressive stress-strain law](image2)

Figure 5. Uniaxial stress-displacement and stress-strain curves used for concrete

Figure 6. IRSN-VTT flexural test - Missile finite element model

The missile material is represented using a von Mises plasticity model with strain rate dependence according to the Cowper-Symonds model, as mentioned above. Stress-strain curves are given in figure 7.

In the IRSN-CNCS-VTT punching mode test, the missile is represented using a total of 4740 eight-node reduced integration solid elements (C3D8R in ABAQUS). The size of the elements has been selected in order to have at least five elements in the thickness of the dome. This gives an element size of about 10 mm. A total of 5873 nodes are used for modelling 1/4 of the missile. In figure 8, elements representing the steel case are drawn in green, whereas the elements of the light-weight concrete are in grey. Perfect bonding has been assumed between the steel case and the light-weight concrete.

For the IRSN-CNCS-VTT punching mode test, the behavior of the steel parts of the missile is represented using a von Mises plasticity model with linear hardening and no rate effects (figure 9). The behavior of light-weight concrete is introduced with a linear elastic model with a Young’s modulus of 10 GPa (1/21 times of steel modulus) and a Poisson’s ratio of 0.20.
Figure 7. Stress-strain law for missile steel at IRSN-VTT flexural test

Figure 8. IRSN-CNSC-VTT punching test - Longitudinal section of the missile finite element model

Figure 9. Stress-strain law for missile steel at IRSN-CNSC-VTT punching test
As in the target representations, symmetry boundary conditions are set at the orthogonal planes limiting the missile models. The corresponding translational and rotational velocities are set to zero at the planes of symmetry.

CONTACT MODELLING

Cylinder testing

Analytical rigid surfaces are used to represent the steel loading platens in the Brazilian and triaxial tests. A friction coefficient of 0.25 has been assumed at the contact with the specimen.

Impacts

No interaction force is assumed between the missiles and the targets. A contact pair is declared between them. Self-contact is also declared for the missile shells. Coefficients of friction of 0.3 and 0.4 have been assumed for steel-steel contact and steel-concrete contact, respectively.

Interaction at contacts is dealt with using the penalty formulation (i.e. not the kinematical formulation).

CALCULATIONS

Simulation of triaxial tests

Figure 10 presents the comparison between experimental and simulation results for several confining pressures \( p \) (magenta= experiment, red= simulation).

Figure 10. Stress-strain results in the simulated triaxial tests (magenta-test, red-simulation)
Note that the stress peaks and residual stresses (stress plateau at large strains) are reasonably well captured at all confining pressures. However, the strain corresponding to the stress peaks can be very different from value obtained in the tests, especially at large confining pressures. This is a known limitation of the CDP constitutive model, since the compressive hardening-softening can not be made dependent on the hydrostatic pressure (third stress invariant). In the simulations, the hardening-softening has been assumed to be that in the unconfined or conventional uniaxial compression test (CS). This is believed to be the common practice, since triaxial testing is not generally available for day-to-day projects.

**IRSN-VTT flexure mode test**

Impact lasts for about 15 ms (figure 11). Time step is between 0.32 and 0.33 \( \mu \text{s} \). Computer runs were led up to 30 ms, to allow for rebound and free vibration. The missile bounces back with a velocity between 5 and 6 m/s (impact velocity is 110.15 m/s). Computer time is in the order of 4 CPU hours on a Linux based PC. Momentum balance is met exactly. Energy balance is met approximately, since the numerical integration produces some artificial loss of energy. No energy spikes due to instability were obtained.

Missile damage can be seen in figure 12. The missile collapses in a “concertina” mode. The final length of the missile is 1.39 m. Interaction forces and impulse are basically the same as in IRIS-2010 exercise (figure 11).

Figure 13 shows the displacement time histories at point W3 in the rear of the slab. The approximation to the test peak displacement is now much better than in the IRIS 2010 exercise. However, residual displacements are overestimated by the simulations. Inelastic deformation is larger in the model than in the test. Note also that the first natural frequency in the model (~150 Hz) is in the order of four times the natural frequency of the actual test set-up including frames, columns, etc. (~35 Hz).

![Velocity time history of the rear of the missile during impact](image1)

![Load time history between the missile and the target](image2)

Figure 11. IRSN-VTT flexure test - Velocity at the rear of the missile and contact force during impact

![Damage to the missile at the end of the impact](image3)

Figure 12. IRSN-VTT flexure test - Damage to the missile at the end of the impact

Maximum strain at the monitored rebars is in the order of 3 %, much less than ultimate strain (10 %) (figure
This indicates that slight damage occurs at the reinforcing steel grid and, consequently, that the target sustains the impact with some margin. Maximum computed strains are smaller than experimental values (3.7 %). However, computed plastic strains are much larger (2.7 % vs. 2.1 %).

Figure 13. IRSN-VTT flexure test – Displacement at rear (point W3) and maximum strain at rebars (D4)

Figure 14. IRSN-VTT flexure test – Damage to the target (permanent effective crushing strains; grey color indicates effective strain above 0.5 %)

Figure 15. IRSN-VTT flexure test – Damage to the target (permanent effective cracking strains; grey color indicates effective strain above 1.0 %)

Figures 14 and 15 show the contours of effective crushing and cracking strains in the concrete, respectively. The contours suggest the formation of a long diagonal crack in the rear face of the target. Cone-shape damage develops at the impact zone. Computed residual displacement at the rear of the target is in the order of 35 mm. This is far more than the value measured in the experiment (12 mm) and it suggests again that permanent deformation is larger in the simulation than in the test.
IRSN-CNSC-VTT punching mode test

The main part of the impact lasts only for 1 ms (figure 16). At this time, rupture at the target has taken place, since no interaction force is predicted and the missile still has a velocity in the order of 35 m/s. The interpretation is that perforation will occur. However, the software is not equipped with an erosion algorithm that would allow elimination of failed concrete elements. Element distortion makes results less reliable as deformation progresses beyond this point. The software is not able to predict the rupture of the steel bars and the sharp separation of the concrete plug in front of the missile from the rest of the slab that remains in place.

Missile damage is very small (figure 17). The final total length is 63 cm, only 1 cm less than the initial length. The steel nose has some small permanent deformation.

Figure 18 gives two representative displacement and rebar strain time histories. Note that computed results compare well with the experiment up to 5 ms, approximately. Then, the experimental results show a drop in displacement and strain and an elastic oscillation, as if the part of the slab that stays in place had been unloaded or left without interaction with the missile. This “unloading” effect does not appear in the computed results. As mentioned above, the simulation reproduces the onset of the penetration process, but it is not able to reproduce the penetration itself with accuracy.

Figures 19 and 20 show the contours of effective crushing and cracking plastic strains in the concrete. The contours suggest localized damage, with severe cone cracking at the impact zone.

Figure 16. IRSN-CNSC-VTT punch test - Velocity at the rear of the missile and contact force during impact

Figure 17. IRSN-CNSC-VTT punch test - Damage to the missile at the end of the impact
Figure 18. RSN-CNSC-VTT punch test - Displacement at rear (point W1) and maximum strain at rebars (D1)

Figure 19. RSN-CNSC-VTT punch test - Damage to the target (permanent effective crushing strains; grey color indicates effective strain above 0.5 %)

Figure 20. RSN-CNSC-VTT punch test - Damage to the target (permanent effective cracking strains; grey color indicates effective strain above 1.0 %)
CONCLUSIONS

After the results of the flexural and punching mode tests were made available, an analysis of the differences with the results of the blind simulations made by the authors within the IRIS-2010 exercise was carried out. The analysis pointed out the limitations of the Drucker-Prager model used to represent the concrete behaviour. For the present IRIS-2012 exercise, the same general simulation methodology of IRIS-2010 has been maintained. The only exception is that the concrete constitutive model has been changed to the ABAQUS Concrete Damaged Plasticity model. In this model, the evolution of the yield surface is controlled by two hardening variables (i.e. not just one as in the Drucker-Prager model). The two variables are linked to the failure mechanisms under tension and compression loading, respectively.

After calibrating the parameters of this material model using the Brazilian and triaxial test results supplied by the organizers, a significant improvement with respect to the results obtained with the Drucker-Prager model has been obtained.

The flexural test is reproduced reasonably well, taking into account that the support structure is not represented in the finite element model and double symmetry has been assumed to build the finite element model.

The punching mode test is more difficult to reproduce due to the complex nature of the penetration process. The software that has been used does not have an “erosion” facility to eliminate the ruptured concrete. The simulation reproduces the onset of the penetration process, but it is not able to reproduce the penetration itself with accuracy.

REFERENCES


The present numerical simulation report is the contribution of the team constituted by IRSN and the research unit LMT from the ENS Cachan academic institution (France).

1 Introduction

The simulations presented in this rapport were made in an "industrial" condition. Once the model parameters chosen to best represent the experimental datas, calculations were made directly without setting particular parameters (default values of the code have then been used).

The Finite Element code used for the simulations was LS-DYNA version 971d. This code is well known for solving highly nonlinear transient problems. Moreover the contact algorithms, the element and materials libraries are large enough to be able to correctly model the problems we want to deal with. We used the explicit solver of the code due to the nature of the simulations. The simulations have been fully modeled in 3D.

2 Concrete slabs modeling

For the concrete slabs we used under-integrated cubic elements (1 Gauss point per element). To avoid problems with this type of elements we used the Flanagan-Belytschko stiffness form with exact volume integration Hourglass control type. The mesh size has been chosen to obtain a final problem with acceptable dimensions, while respecting the minimal element number associated with the finite element method to describe non-linear problems. The boundaries conditions are imposed on the slab in order to best comply with the experimental ones.

The reinforcements have been modeled by 1D bar elements coupled (rather than merged) to the surrounding cubic elements of the concrete slab. This coupling was achieved using the CONSTRAINED_LAGRANGE_IN_SOLID (see also [Murray et al 2007]) without friction feature in LS-DYNA. Using this method, the reinforcements can be placed anywhere inside the concrete elements without any special mesh accommodation. The experimental geometry and position of the reinforcement have been approximately respected but for sake of simplicity some arrangements may have been done with respect to the real steel/concrete ratio.

2.1 Constitutive laws

Material models are essential to accurately studying such problems. For our study we used the continuous surface cap model (CSCM, i.e. MAT_159) implemented in LS-DYNA. The model includes isotropic constitutive equations, yield and hardening surfaces, and damage formulations to simulate softening and Young modulus reduction. A rate effects formulation allows increase strength with strain rate. For a complete theoretical description one can refer to [Murray 2007] and the LS-DYNA manual.

The basic model has a set of parameters that are important to identify. However, the simplified version CONCRETE allows to only input the unconfined compressive strength,
maximum aggregate size (the other parameters are determined by default) and the use of a material rate effect. This model has been tested and successfully validated in [Murray et al 2007].

All steel reinforcement was modeled using an elastoplastic with kinematic hardening material model. No rate effect has been used due to the expectable low strain rate in the rebars.

2.2 Finite element model for the VTT Flexural and punching tests

![Figure 1: VTT tests boundary conditions](image1)

The cylindrical sections, the metal plate and U-shaped profile forming the supports of the slab (see Figure 1a) weren’t modeled in these simulations contrary to IRIS_2010. With regards to the first benchmark results, the supports were only modeled by blocking the corresponding nodal displacements (white triangle in Figure 1), on both contact planes from the supporting frame.

Figure 2 shows the FE model used for the flexural test simulation. We obtained 589,440 cubic elements with an average size of 5 mm x 5 mm x 7.5 mm in the middle of the slab and 12.5 mm x 12.5 mm x 7.5 mm elsewhere.

Figure 3 shows the FE model used for the punching test simulation. We obtained 741,600 cubic elements with an average size of 5 mm x 5 mm x 7.5 mm in the middle of the slab and 12.5 mm x 12.5 mm x 7.5 mm elsewhere. Indeed, to represent the phenomenon of perforation we used two erosion techniques. The first technique involves removing the elements of the FE model where and when a criterion (here a damage of 1,05) is reached. We coupled this technique with a criterion on shear strain where the elements of the FE model are deleted when the shear strain is higher than 0.55. If the size of deleted elements is too large we obtain a result that is highly dependent on the mesh used. That's why we use a very fine mesh in order to limit the mesh dependency of the results.

The parameters introduced in the concrete model were: RO=0.00243 g/mm³, FPC=56 MPa, FT=3.75 MPa and DAGG=8 mm for the flexural test and RO=0.00243 g/mm³, FPC=58.1 MPa, FT=4.0 MPa and DAGG=8 mm for the punching one.
3 Projectiles modeling

The projectiles were meshed to represent the different thicknesses of steel tubes used and a realistic weight distribution. Finite elements used were the default shell elements of LS-DYNA: Belytschko-Tsay elements with the standard LS-DYNA viscous form hourglass control. The mesh size of the nose has been chosen to properly manage the contact between the projectile and the slab. Other elements have a size that allows fully capturing the local buckling of the projectile tubes and thus obtaining the most realistic deformation as possible.
Figure 5 gives an overview of the final mesh obtained for the projectile used in the VTT flexural test. The mesh size is fairly uniform and is about 10 mm x 10 mm. We obtain 24,000 shell elements. The projectile was meshed into several parts to reflect the geometry of it as accurately as possible. Each color corresponds to a given shell thickness, 12.5 mm for yellow, 25 mm for brown and 2 mm for blue and green. The heel (brown) was also detailed to properly take into account the mass distribution in the projectile. The constitutive law used for the simulation is plotted in Figure 5. The constitutive law used for the calculation is a classical elastic plastic with kinematic hardening with no strain rate effect. We used yield strength of 350 MPa, a Young's modulus of 175 GPa and a strain hardening modulus of 2.33 GPa.

Figure 6 shows the mesh used for the projectile of the VTT punching test. The projectile was meshed into several parts to reflect the geometry of it: the nose in yellow (solid elements), the shaft in brown (shell elements), the lightweight concrete (solid elements) and the heel (shell elements). The mesh size is small and fairly homogeneous in the nose (3 mm x 3 mm x 10 mm) to manage the contact with the slab and the eroded elements. We obtain 111,216 cubic elements and 6,912 shell elements. The parameters introduced in the lightweight concrete model were: RO=0.000345 g/mm³, FPC=25 MPa and DAGG=8 mm. For the steel, we used the same constitutive law than for the projectile of the VTT flexural test. We used yield strength of 355 MPa, a Young's modulus of 210 GPa and a strain hardening modulus of 1.646 GPa.
4 VTT Flexural simulation

The total physical time of the study was 100 ms, the time step of calculation was $5.3 \times 10^{-4}$ ms. The total CPU time of calculation was 39h46min for a single processor.

![Figure 7: a) Kinetic energy vs time, b) Internal energy vs time](image)

Figure 7 shows the energy balance of the VTT Flexural numerical simulation. It is clear again that almost all the initial kinetic energy is converted into internal energy. At the end of the calculation the ratio of the total/initial energy was 98.2% with an hourglass energy accounting for only 0.2% of the total energy that is very acceptable.

![Figure 8: Time history of the contact force between the projectile and the target](image)

Figure 8 shows the impact force imposed by the projectile on the target. One can see the classic shape of a load due to a deformable projectile. The maximum force reached is 900 kN for a total duration less than 30 ms. The evolution the velocity of the rear face of the projectile shows a rebound of it, with a velocity of about 5 m/s.
Figure 9: a) Maximal deflection of the slab (x10), b) W1 displacement vs time

Figure 9a shows the maximum deflection of the slab (t = 11 ms) with an amplification factor of 10. It shows that there is no central area very distorted due to an important damage of the concrete. We can therefore conclude that in this case there will be no scabbing of the rear face of the slab. Figure 9b shows the evolution of the displacement of point W1 versus time. One can see that 16.8 mm maximum displacement is reached in about 11 ms and that oscillations occur around a final displacement of 14 mm.

Figure 10: Damage map of the front, rear faces and sectional of the slab

Figure 10 shows the damage of the slab at the end of the simulation. On the front face of the slab, we can see that there are almost no cracks that appear. On the rear face the cracked zone has a diameter of 2.5 m. It also confirms that there is no scabbing in this simulation.
Figure 11 shows the projectile at the end of the simulation. We obtain LT=0.7 m and HT=0.195 m.

5 VTT Punching simulation

The total physical time of the study was 50 ms, the time step of calculation was $1.58 \times 10^{-4}$ ms. The total CPU time of calculation was 80h for a single processor.

Figure 12 shows the energy balance of the VTT Punching numerical simulation. It is clear again that almost all the initial kinetic energy is converted into internal energy but it then decreases over time. At the end of the calculation the ratio of the total/initial energy was 92.8% with an hourglass energy accounting for 0.2% of the total energy. One can note here the limitations of the erosion technique used, which lose energy in the calculation. The results are however still considered acceptable.

Figure 13 shows the impact force applied by the projectile on the target. One can see the classic shape of a load due to a rigid projectile with a very marked peak. The maximum force reached is $6.26 \times 10^3$ kN for a total duration of 2.75 ms.
Figure 13: Time history of the contact force between the projectile and the target

Figure 14: Damage map of the slab in the central section
Figure 15 shows the damage of the slab at the end of the simulation. It shows very clearly the perforation of the central area of the slab. On the rear face of the slab the scab has a diameter of 0.5 m. On the front face, we obtain a cracked zone of about 0.3 m in diameter. The velocity evolution of the rear face of the projectile is shown in Figure 15. We can notice that its speed is reduced to 40 m/s at the end of the simulation.
Figure 16 shows the shape of the projectile (slightly deformed) at the end of the simulation.

6 Conclusions

The results presented in this contribution are in a good agreement with the experimental ones. Nevertheless, we made others simulations (not presented here) in order to see the influence of important parameters (concrete tensile strength, slab mesh size, erosion coefficients). As expected, the results strongly depend on the concrete tensile strength. This could have detrimental consequences for a parameter so uncertain. Also our results show that the choice of the erosion parameter can be made only according to a given mesh size and not in a objective way. It seems then difficult to be able to fix this one a priori what is an important problem to be able to obtain predictive simulations of perforation.

All these important points must be more studied and discussed during the workshop.

7 References

1 INTRODUCTION

The simulation of the concrete tests and of the motion of B1 slab and P1 slab are made by using the computer code RADIOSS, using a fast dynamics explicit solver.

2 SIMULATION OF TESTS ON CONCRETE SAMPLES

2.1 MESH, LIMIT CONDITIONS AND MATERIAL LAW

Simulations of triaxial tests and Brazilian test are done to validate the concrete material law (Mat. 24 of Radioss). Only 5 parameters must be fixed by the analyst\(^1\). In the present study the following values are chosen: density 2.4 \(\text{g/cm}^3\); Young’s modulus 29,000 MPa; Poisson’s coefficient 0.2; uniaxial concrete strength 60 MPa; tensile strength 4 MPa.

2.1.1 TRIAXIAL TEST

For the triaxial tests simulations, the concrete samples are modelized using brick elements. On figure 1, deformations (extension) along circumferential direction are presented on the left side and deformations (contraction) along axial direction are presented on the right side.

Fig. 1a. Triaxial tests: comparison of tests results with the numerical simulations

\(^1\) The other parameters are determined by default
2.1.2 BRAZILIAN TEST

Simulations with brick elements and with SPH (Smoothed-Particle Hydrodynamics) are done: the value of the applied diametral force at splitting is consistent with the concrete limit in tension. Sensitivity to mesh refinement is checked.

---

Fig. 2. Brazilian test simulation with the brick model. View of the damage in the concrete sample

Fig. 3. Brazilian test simulation with the SPH model. View of the density of the concrete sample: blue colour for lower density
3 SIMULATION OF SLAB B1 MOTION

3.1 PROJECTILE

3.1.1 MODEL: MESH, LIMIT CONDITIONS AND MATERIAL LAW

The steel projectile is modeled using 29,820 shell elements (size 8 mm) and the Johnson & Cook material law. For sensitivity purposes, both brittle and ductile failure modes are considered.

![Stress-strain curve with linear damage model in the tensile regime](image1)

Fig. 4. Deformable missile: a) stress-strain curve with linear damage model in the tensile regime; b) view of the mesh

3.1.2 MOTION AND LOADING

The projectile is uniformly decelerated from 110 m/s to 0 in $t_{\text{shock}} = 20$ ms and then rebounds with a velocity of $-5$ m/s. LT(H) = 1,164 mm (131 mm) at $t_{\text{shock}}/2$ and 850 mm (190 mm) at $t_{\text{shock}}$.

![Simulation of slab B1 motion](image2)

The loading function after a 10 000 Hz filtering. The peak value is about 0.7 MN. The loading duration is about 21 ms. The impulse transmitted by the projectile is about 5.8 kN.s which is greater than $m.V_0 = 5.5$ kN.s. This comes from the effect of the 5m/s rebound velocity.

The 3 first ms are used for imposing, by dynamic relaxation, the tightening force along the limits of the slab.

![Results of the simulation](image3)

Fig. 5. Results of the simulation: crushing length and impact load of the missile

The final length fitting leads to consider strain hardening and dynamic increase factors (DIF) in missile material.
3.1.3 CHECKS AND SENSITIVITY STUDIES

The sensitivity of the results is studied along two aspects of the missile’s material law:

- the failure mode of the metal: brittle or ductile;
- the strain rate effect.

The sensitivity studies show that if both aspects influence the damage pattern of the missile and its crushed length, they do not change much the loading diagram on the target.

3.2 SLAB B1

3.2.1 MODEL BY FINITE ELEMENTS: BRICKS AND TRUSSES

3.2.1.1 Limits conditions

The contacts of the slab along 4 lines parallel to the edges on front and back faces are tightened with an imposed uniformly distributed force (1,75MN), the displacements parallel to the slab plane being free but submitted to in-plane friction forces according to a Coulomb law (friction coefficient f=0.3).

3.2.1.2 Concrete mesh and behavior law

The concrete mesh contains 330,000 brick elements of Belytschko type (typical size: 15 mm × 15 mm × 8.8 mm). The concrete behavior law is based on Ottosen approach (References: 1. Ottosen N.S. “Non Linear Finite Element Analysis of Concrete Structures” and 2. D.J. Han, W.H. Chen “Plasticity Model for Concrete in Mechanics of Materials”). The values of parameters fixed by the analyst are those listed in § 2.1.

3.2.1.3 Rebars mesh and behavior law

Reinforcement is modeled by 21,500 individual rebars composed by 15 mm long truss elements. Kinematic interfaces are arranged between rebars and concrete nodes. The rebar steel behavior law follows the Johnson & Cook law:

$$\sigma = (a + b \varepsilon_p^n) \left(1 + c \ln \frac{\varepsilon}{\varepsilon_0}\right)$$

with $a=650$ MPa, $b=1,667$ MPa, $c=0$ and $n=1$ $(\varepsilon_p$: plastic deformation)

Elastic characteristics: Young’s modulus $E=200,000$ MPa and Poisson’s ratio $\nu=0.3$

Yield limit $f_y=700$ MPa and rupture deformation $\varepsilon_{ru}=10\%$
3.2.2 **SLAB MOTION**

### 3.2.2.1 Displacements

The displacements in the simulation match fairly the ones measured during the test. It can be however observed that the period of the local extrema and the damping of the vibrations seem higher in the simulation.

The simulation underestimates the first eigenfrequency of the slab, this may be due to the limit conditions: the slab has to be more restrained in the supporting frame. The damping is overestimated, this has to be investigated.

### 3.2.2.2 Concrete and rebars strains

The simulation provides rebars strain values smaller than the measured ones: typical value of simulated plastic strain is about 0.15% against a measured value of few percent. This is to be investigated. A hypothesis is that the measure is actually the one of rebar that may slip within the concrete, specialy in the presence of the gauge itself, and it is compared to a punctual value of strain given by the computation. It can be outlined that the simulation strain values of the concrete are more consistent with the experimental values (maximum value of R1, R2 and R3 around 0.2%).
3.2.2.3 Support reaction

Some differences appear between simulated and measured total force in the tubes that support the frame. This is due to the fact that the supporting frame itself and the tube are not modeled and their filtering effect is not taken into consideration. The order of magnitude of the peaks (first peak excepted which is a numerical artifact) is correct and the integral of $F(t)$ during the slab vibration time gives the impulse communicated by the projectile.

3.2.2.4 Damaging

Figures 7 and 8 show the damage of the slab after impact. The color variations show the concrete iso-densities that underline the slab damaging. The cone shaped damaging under the impact area and the diagonal hinges are well apparent. They are consistent with the experimental results.

Fig. 7. Damaging of the slab after impact: variations of material density. Front and back views.

Fig. 8. Damaging of the slab after impact: variations of material density. Cross sections.
3.2.3 CHECKS AND SENSITIVITY STUDIES

The checks done on the impulse conservation and the energy transformation give satisfactory results (the low hourglass energy consumption is modest). Sensitivity studies on the limit conditions: i) the modeling of the cylindrical bars ensuring the contact slab/supporting does not modify the slab motion and ii) the higher is the slab tightening force the lower is the amplitude of the slab motion. Sensitivity study on concrete cracking energy: its value modifies the frequency and the amplitude of the slab motion, but not the residual damping.

4 SIMULATION OF SLAB P1 MOTION

4.1 PROJECTILE

4.1.1 MODEL: MESH, LIMIT CONDITIONS AND MATERIAL LAW

Concrete modeled by 1760 bricks, steel envelope by 2796 shells (typical size 11 mm x 11 mm). Brittle Plastic material law (Johnson & Cook). Total number of nodes: 16252.

4.1.2 MOTION

![Fig. 9. Results on the missile: impact force and velocity](image)

The impulse plateau, which absolute value is 5.07 kN/s, is consistent with a 47 kg projectile hitting the target at 135 m/s with a residual velocity about 27 m/s: 5.07 ≈ 47.(135 – 27).

4.1.3 CHECKS AND SENSITIVITY STUDIES

![Fig. 10. Simulation results: impulse received by the target](image)
4.2 SLAB P1

4.2.1 MODEL BY SPH

4.2.1.1 Limit conditions

The slab model is fixed along 4 lines of nodes parallel to the edges of the slab.

4.2.1.2 Concrete mesh and behavior law

The meshing of the slab is of the type “meshless” based on the particle topology named “Smooth Particle Hydrodynamics” (SPH). The typical size of the particles is 15x15x15 mm$^3$. The behavior law of the concrete is same as the one used for B1 slab, with the same values of parameters.

4.2.1.3 Rebars mesh and behavior law

The rebars are modeled by 6600 15 mm long truss elements. The rebar/concrete interaction is obtained by a “merging” procedure that enables a common deformation. The rebars behavior law is similar to the one of B1 slab.

Fig. 11. Model of slab P1 showing the rebars

4.2.2 SLAB MOTION

4.2.2.1 Displacements

The computed displacements of the slab are far above tests results (see Fig. 12). There is no clear explanation of that result.

Fig. 12. Simulated versus experimental displacements of the slab P1
4.2.2.2 Concrete and rebars strains

No concrete strains were provided by the computation. Only plastic strains in the rebars are figured out on the figure 14.

Fig. 14. Slab P1. Strains in the rebars. Comparison test-simulation

4.2.2.3 Support reaction

No support reaction was provided by the simulation, due to the limit conditions chosen: blocking of the displacements of the edges nodes.
4.2.2.4 Damaging

Fig. 15. Slab P1. Map of density variations in the slab, illustrating the damage of the target.

4.2.3 CHECKS AND SENSITIVITY STUDIES

Kinetic energy (dashed red line) is progressively transformed into internal energy (plain blue line). The total energy curve (dotted magenta line) is almost constant all along the motion. Slab modeling with FE brick element gives residual velocity of about 11 m/s (without erosion).

5 CONCLUSIONS

The simulations of the triaxial and Brazilian concrete tests and of the B1 and P1 slab tests made by the code RADIOSS provide rather reliable and consistent results.

The simulation of the slab whose motion is mainly due to flexural deformation (B1) is rather close to the experimental one. However some points are still to be investigated, in particular the limit condition modeling.

Concerning the simulation of the slab with perforation (P1), the model by particle topology named “Smooth Particle Hydrodynamics” (SPH) correctly predicts the perforation and damaging of the slab and the residual velocity of the missile. The displacements and strains are however less accurate and differ from the experimental values. Simulated displacements are up to ten times higher than the measured ones whereas the simulated rebar strains are lower than their experimental values. There is no clear explanation of this observation.
IRIS_2012 NUMERICAL SIMULATION REPORT

Institut de Radioprotection et de Sureté Nucléaire

Team 3

S. Kevorkian

1 Introduction

The IRIS_2012 Benchmark on Improving Robustness Assessment Methodology for Structures Impacted by a Missile is based on IRIS_2010 results and recommendations. The goal of this benchmark is to continue the work on understanding and improving simulation of structural impact. This report introduces the results obtained for the simulation of the VTT-IRSN-CNSC Punching P1 test and the VTT-IRSN bending B1 test. These calculations take into account the results from IRSN, as well as the results from the Brazilian concrete tensile test, to calibrate concrete constitutive models to be used in the benchmark.

2 Basic choices

For the two simulations punching and bending, the Finite Element code chosen is LS-DYNA version 971 Release 5.1.

The supporting frame is not modelled but taken into account with the boundary conditions. The edges of the slab are embedded. The rebars are modelled with beam elements. Not only the rebars are merged to the concrete but the junctions between the rebars are also merged. The constitutive law of the rebars is *MAT_PIECEWISE_LINEAR_PLASTICITY.

The contact is modelled in the same way for the punching and bending calculations. The contact between the missile and the reinforced concrete is modelled with *CONTACT_ERODING_SINGLE_SURFACE.
3 Modelling

3.2 Punching

3.2.1 Concrete slab

Concrete is modelled with 125316 solid elements (1.5 cm*1.5 cm in the middle of the target).
The constitutive law of the concrete is *MAT_CONCRETE_DAMAGE_REL3 (*MAT_72R3).
The rebars are modelled with 12768 beam elements (for BH and BV: 3600 beam elements, for FV and FH 2724 beam elements). The size of the beam element is 1.5 cm.

3.2.2 Missile

The missile is modelled with solid elements for the dome and the light-weight concrete and with shell elements for the steel plate and the steel pipe. The constitutive law for the steel elements and the light-weight concrete is *MAT_PLASTIC_KINEMATIC (*MAT03). For the light-weight concrete the law takes into account the compression strength. The weight of the modelled missile is equal to 47.04 kg.
3.3 Bending

3.3.1 Concrete slab

Concrete is modelled with 51984 solid elements (2.75cm*2.75 cm) all over the target. The rebars and stirrups are modelled with 16012 beam elements (for BH and BV: 3192 beam elements, for FV and FH 2812 beam elements and 4004 beam elements for the stirrups). The size of the beam element is 2.75 cm. The constitutive law of the concrete is *MAT_WINFRITH_CONCRETE.

![Concrete](image1)
![Rebars and stirrups](image2)

3.3.2 Missile

The missile is modelled with shell elements for the steel plate and the steel pipe. The constitutive law for the steel elements is *MAT_PLASTIC_KINEMATIC (*MAT03). The weight of the modelled missile is equal to 50.07 kg.

![Bending missile](image3)

4 Calculations

4.2 Punching
Two calculations are performed for punching. In the first calculation erosion is not considered, for the second calculation erosion has been taken into account with *MAT_ADD_EROSION associated to the concrete.

4.2.1 Damages
The figures hereafter show the comparison between the calculation without erosion and the calculation with erosion for the damages of the slab.

Figure 7: Punching without erosion

Figure 8: Punching with erosion

Figure 9: Front of the slab: Punching without erosion

Figure 10: Front of the slab: Punching with erosion
For the calculation without erosion, plastic erosion was considered in order to obtain an estimation of the diameter of the perforation. In figure 15, the elements with a plastic strain greater than 2, were suppressed from the mesh. The diameter obtained with both calculations is about 30 cm.

The diameter obtained by both calculations is about 30 cm, regardless whether there is erosion or not.

Considering the displacements of the sensors, for the sensors close to the impact, the behaviour is very different from the tests whatever the calculation. For the
other sensors, there is an oscillation around a small permanent displacement in compliance with the punching test.

4.2.2 Missile

The residual velocity of the missile is presented in the next figure. The residual velocity obtained after a calculation without erosion (26 m/s) proved to be less important than the residual velocity after a calculation with erosion i.e. 32.2 m/s. The duration of the impact is about 19 ms.
The impulse shall be compared to the initial momentum of the missile which is equal to 6.345 kN/s : (135 m/s * 47 kg). The impulse obtained by the calculation without modelling the erosion is the closest value to the initial momentum.

4.2 Bending

The missile impacts the slab and bounces. The missile is crushed during the impact.

4.2.1 Damages

There is no serious damage for the target like scabbing or spalling.
The next figures represent the impact force between the missile and the concrete and the impulse.

Figure 24: Front of the slab: Bending

Figure 25: Back of the slab: Bending

Figure 26: Load force between missile and concrete
The impulse shall be compared to the initial momentum of the missile which is equal to 5.5 kN/s : (110 m/s * 50 kg). The impulse obtained with the calculation is quite similar even if the value is higher than the initial momentum. This is due to the residual velocity after the bounce on the slab (5 m/s).

Figure 27: Impulse

Figure 28: Displacement time history of the slab (W1)

There is a peak of the displacement at the end of the shock and the slab oscillates around a small permanent displacement (1 mm). The displacements obtained for the different sensors are less important than the ones of the bending test.

4.2.2 Missile

The missile bounces on the slab, with a residual velocity equal to -5 m/s.
The missile is crushed during the impact.

The next figure shows the evolution of the length of the blue part of the missile during the impact. The diminution is equal to 53 cm.
5 Conclusion

Punching mode: two different calculations were performed with and without erosion. A perforation is obtained with both calculations. The residual velocity of the missile is in compliance with the punching test in both cases. There is an oscillation of the slab after the perforation around a small permanent displacement.

Bending mode: the calculation shows that the missile doesn't create important damage to the slab (neither spalling nor scabbing). During the impact the missile is crushed and bounces on the slab. The slab oscillates around a small permanent displacement.

The calculations carried out by IRSN using LS-DYNA and its concrete models proved to be a way to simulate correctly the behavior of concrete for missile impacts, both in bending and punching.
1 INTRODUCTION

The report relates to the simulation of B1 slab motion by a home-made code named PENTABLOC.

2 METHODOLOGY OF COMPUTATION

2.1 LOGIGRAM

The simulation is based on the use of several programs by spreadsheet according to the following logigram.

The 3 programs RIERA, PENTABLOC and 2-DOF are briefly presented here below.
2.2 PROJECTILE SIMULATION AND LOADING

The RIERA program is based on the method developed by the Professor Riera in 1968 [1].

2.3 SIMULATION OF SLAB MOTION UNDER FLEXURAL BEHAVIOR

The simulation is based on the method developed by Rambach and Tarallo in 2008 [2].

2.4 SIMULATION OF WHOLE MOTION OF SLAB AND SUPPORTING FRAME

The simulation is based on the resolution of the equations of motion of a set of 2 successive mass-spring sets (2 degrees of freedom) that is classically exposed in the academic literature.

3 SIMULATION OF SLAB B1 MOTION

3.1 PROJECTILE

3.1.1 MODEL: MESH, LIMIT CONDITIONS AND MATERIAL LAW

The projectile is modeled according to the Riera's method. The method needs the determination of the buckling resistance density Rc and of the mass density dm/dx along the axis of the projectile.

The density of buckling resistance Rc is supposed constant and its value is given by the product of the static buckling resistance of the tube Pm by a dynamic coefficient DIF: Rc=DIF.Pm.

The static buckling resistance Pm is given by Jone's formula [3]:

\[ P_m = 2 \cdot (\pi \cdot H)^{3/2} \cdot R^{1/2} \cdot \sigma_0 (1/3)^{1/4} \]

where H is the thickness of tube wall, R the tube radius and \( \sigma_0 \) the yield limit of the tube material.

The dynamic coefficient DIF is determined by the adjustment of the final length of the tube (after Riera’s formula use) with respect to the experimental measured one. The dynamic coefficient taken into account is about 1.35; that value is consistent with values coming from other similar tests.

The mass density is determined according to the drawings for projectile construction provided by VTT.

3.1.2 MOTION

3.1.2.1 Loading and velocity

Fig. 2 Loading by the projectile
Fig. 3 Time history of the velocity of the rear of the projectile
3.1.3 CHECKS AND SENSITIVITY STUDIES

The impulse plateau at whose absolute value is 5.51 kN/s is consistent with a 50 kg projectile hitting the target at 110 m/s: \(5.51 \times 10^3 = 50 \times 110\)

Fig. 4 Time history of the impulse received by the target

3.2 SLAB B1

3.2.1 MODEL

3.2.1.1 Limit conditions

The slab is supposed simply resting along its 4 edges.

3.2.1.2 Concrete mesh and behavior law

The meshing of the slab is made of 4 elements. The PENTABLOC programs allows a meshing by 5 bocks, the fifth block being located in the centre of the slab, the 4 other blocks being with a trapezoidal shape.

Fig. 5 Typical scheme of PENTABLOC with 5 blocks

Fig. 6 Scheme used, with suppression of the central block
3.2.1.3 Rebars mesh and behavior law

The rebars are not explicitly modeled, but their action within the concrete is modeled through elastic-plastic spiral springs all along the hinges diagonally located. The behavior law is of type “bending moment – curvature” obtained under the classical assumptions of reinforced concrete that consider that the plane sections when submitted to axial force and bending moment remain plane and that there is no tension in concrete.

![Behavior law of rebar steel and concrete](image)

The mechanical values that are used are those ones provided by VTT value, excepted for the Young modulus of the concrete specimen that is taken according to the value proposed by EC2 code (E=27 000 MPa).

3.2.2 SLAB MOTION

3.2.2.1 Introduction

The slab motion simulation is done with the knowledge of the results: the sensitive parameters have been established in order to match as close as possible to the measured displacements with credible value for those characteristics. This exercise is the occasion to test the sensitivity of the governing parameters. It appears, as already mentioned in preceding paper (Rambach, 2007, Ref. 4), that the governing parameters are: the bending moment plastic limit, the Young modulus and the damping ratio before the impact and after the impact and, specifically for slabs, the choice of the hinges location (along diagonals or with a central block).

The best fitting of the mechanical characteristics gives following values for the governing parameters:

- E=17 000 MPa, $\xi=5\%$ before the shock
- E=13 016 MPa, $\xi=25\%$ after the shock

The limit bending moment value is unchanged: it is directly the value coming from the experimental tests on the concrete and on the rebar of the slab, it can be seen on the figure here above: $Mu=0.039 \text{ MN}^\text{m}/\text{m}$. 
3.2.2.2 Displacements

The maxima and at rest displacement values are consistent with the measured ones, however the frequency of the post-shock vibrations are slightly underestimated, it means that the rigidity of the simulated slab is lower than of the tested slab. The sensitivity tests regarding the rigidity show it is not possible to get simultaneously the coincidence with the extrema and with the free vibration frequency. This is probably due to the modeling itself that uses only one degree of freedom.

3.2.2.3 Concrete and rebar strains

No concrete strain is directly provided by the computation. The use of bending moment-curvature relationship does not provide strain time histories in line with the measured ones (this statement is similar to the one from more sophisticated FEM models).
3.2.2.4 Support reaction

The support reaction time history is given by a 2-Degree-of-freedom model: a 1st DOF (purely elastic) represents the supporting frame motion and the second one (within an elastic plastic domain) the motion of the active moving part of the slab. The following characteristics were used:

DOF #1: M=4600kg, elastic coefficient K=1430MN/m (elasticity of the 4 tubes in static axial compression), damping ratio $\xi=4\%$

DOF #2: M=500kg (1/3 slab mass), elastic coefficient before plasticity K=41.3 MN/m, elastic coefficient after plasticity K=14.4 MN/m, yield limit for the plastic force Fp=0.468 MN, damping ratio $\xi=4\%$

The simulation of the total force at the support gives rather fair results. The value of the total impulse at support oscillates around a final value of 5.5 kN.s that corresponds to 50 kg x 110 m/s. It can be noticed that the total impulse of the support force derived from the test results does not oscillates around a line parallel to the abscissa axis: its absolute value continues to increase with the time. Our opinion is that is not a derive of the measuring devices, but this impulse increase does come from the air pressure (outing from the projectile launcher) that still pushes onto the slab, after projectile impact: an increase of impulse of 5 kN.s during 60 ms gives a constant applied force of $5/60.10^3 \approx 83.3$ kN, for an exposed surface of about $2.1x2.1$ m². The corresponding air pressure would be about $83.3/2.1x2.1 \approx 19$ kPa, which is a credible value and deserves to be measured.

3.2.2.5 Damaging

Damaging may be evaluated by the evolution of the Young modulus to be considered:

Young modulus of the concrete alone: E=27,000 MPa

Young modulus of the slab in reinforced concrete before the shock: 17,000 MPa

Young modulus of the slab in reinforced concrete after the shock: 5,840 MPa

4 CONCLUSIONS

The use of simple analytical model for simulating such type of impact is possible, even for non elastic motion. The displacements, in terms of maximal displacement and permanent displacement and the induced reaction force are correctly predicted.
5 REFERENCES


1. Introduction

The main target of the IRIS2012 project is to validate the evaluation techniques used in assessments of integrity of reinforced concrete (RC) structures impacted by missiles or aircrafts, etc. In order to accomplish the target, analysis and evaluation methods were studied in this paper based on simulation analyses compared with the test results of RC slabs (punching and bending) impacted by missiles in the IRIS2010.

Some concrete fracture tests were newly conducted in the IRIS2012 for the purpose of improvement of simulation analysis methods to the missile impact tests conducted in the IRIS2010. Therefore the material constitutive model of concrete was improved based on comparison with the concrete fracture tests under compression and tension loadings, and missile impact test simulation analyses were conducted using the improved concrete constitutive model. As a result the improved simulation results were in relatively good agreement with the test results.

2. Outline of this study

The flow of this study is shown in Fig.-1. First, improvement of concrete analysis model was studied by comparing with concrete fracture test results under compression and tension loadings. And the improved concrete constitutive model was inserted into the missile impact analysis models. Next using the missile impact analysis models, simulation analyses were conducted, and finally simulation results obtained were compared with the test results to validate an accuracy of the impact analysis models.

3. Study on material constitutive model of concrete

Simulation analyses were conducted and its results were compared with the concrete fracture tests under compression and tension (splitting) loads. Outline of the compressive and tensile tests are shown in Fig.-2. The compressive tests were conducted in uni-axial and tri-axial conditions, beside the tensile test was conducted as splitting test (Brazilian test). Concrete constitutive models used in this study (the IRIS2012) and the IRIS2010 are respectively shown in Fig.-3 and Table-1. The green, red and blue lines in Fig.-3 show an elastic limit, a failure surface and a residual surface of the concrete constitutive models of the IRIS2010 and the IRIS2012, respectively. The failure surface represents a hardening between the elastic limit and the peak of yield stress, and the residual surface represents a softening after the peak of yield stress. The failure surface shown as the red line in Fig.-3 can be changed into the red dot-line or the red dash-line by using the coefficients of "a" and "b" as shown in Fig.-3, and the residual surface shown as the blue line in Fig.-3 can be also changed into the blue dot-line by using the coefficient of "ε". As shown in

Fig.-1 Study flow
Table-1, several coefficients in the concrete constitutive model of the IRIS2012 were changed from the IRIS2010. Simulation analysis results of the compressive tests (uni-axial and tri-axial) are shown in Fig.-4 in comparison with simulation results using the IRIS2010's concrete constitutive model. Simulation results of this study (the IRIS2012) were in better agreement with the test results than results of the IRIS2010's concrete constitutive model. Tensile strength of simulation results using concrete constitutive models of the IRIS2012 and the IRIS2010 are shown in Table-2 in comparison with tensile strength of the test. The tensile strength 4.04(MPa) obtained from the tensile test was inputted to the both concrete constitutive models. Table-2 shows that the both simulation results were higher than tensile strength of the test. The main reason of the tensile strength deference is considered that the tensile fracture model in these concrete constitutive models is a simple conventional model only using spall fracture criteria, besides, it might also be affected by the general tendency that tensile strength of a splitting test is higher than strength of a direct tensile test.

(a) Compressive test

(b) Tensile test

Fig.-2 Compressive and tensile tests of concrete (provided by OECD/NEA)
Table 1  Material properties of concrete constitutive models

<table>
<thead>
<tr>
<th>Material properties</th>
<th>Symbols</th>
<th>IRIS2010</th>
<th>IRIS2012</th>
</tr>
</thead>
<tbody>
<tr>
<td>Failure surface constant</td>
<td>$a$</td>
<td>0.97</td>
<td>0.98</td>
</tr>
<tr>
<td>Biaxial strength constant</td>
<td>$b$</td>
<td>1.0</td>
<td>1.2</td>
</tr>
<tr>
<td>Pressure dependency constant</td>
<td>$n$</td>
<td>2.4</td>
<td>2.1</td>
</tr>
<tr>
<td>Residual strength factor</td>
<td>$\tau$</td>
<td>0.7</td>
<td>0.5</td>
</tr>
</tbody>
</table>

*1 The failure surface constant $a$ defines the size of the failure surface.
*2 The biaxial strength constant $b$ defines the slope of the failure surface in the high pressure region.
*3 The constant $n$ is used to consider the pressure dependency of the compressive strain $\epsilon_c$, which is used to calculate the hardening level ($0 \leq \text{Hard} \leq 1$) from the elastic limit surface to the failure surface. $\epsilon_c = \epsilon_{c0} \left( \frac{p}{f'_c} \right)^n$. $\text{Hard} = \epsilon_c / \epsilon_{c0}$, where $\epsilon_{c0}$ denotes a static compressive strain when a stress state reaches a static compressive strength $f'_c$; $\epsilon_p$ and $p$ are plastic strain and pressure, respectively.
*4 The residual strength factor $\tau$ defines the size of the residual surface $\sigma_r$ by the following equation, $\sigma_r = \tau \sigma_f$.

Fig.-3 Concrete constitutive model and its modified material properties
- $\sigma$, $\varepsilon$: stress and strain in the loading (black line) or longitudinal (red line) direction.
- Obs. (LVDT): stress and strain in the loading direction measured by LVDT gauge.
- Obs. (Jo): stress and strain in the lateral direction measured by gauge attached on the specimen.
- $P$: Confining pressure

Fig. 4 Simulation analysis results of the concrete compressive tests

Table 2 Simulation analysis results of the splitting (Brazilian) test

<table>
<thead>
<tr>
<th></th>
<th>IRIS2010</th>
<th>IRIS2012</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tensile strength $f_t$(Obs.)$^1$ (MPa)</td>
<td>4.04</td>
<td>4.04</td>
</tr>
<tr>
<td>Tensile strength $f_t$(Calc.)$^2$ (MPa)</td>
<td>7.40</td>
<td>6.69</td>
</tr>
</tbody>
</table>

$^1$ Tensile strength obtained by the splitting (Brazilian) test.
$^2$ Tensile strength obtained by simulating the splitting test with the input value of the tensile strength $f_t$(Obs.) 4.04MPa. The values of $f_t$(Calc.) were calculated by the equation of $f_t$(Calc.) = (2・$p$)/( $\pi$・$d$・$l$), where, $p$ is the maximum load obtained by the simulation, $d$ is the diameter of the specimen and $l$ is the length of the specimen.
In consideration of the difference between the simulation result of the IRIS2012 and the tensile test result, in case of only using the spall fracture criteria as tensile fracture criteria in any missile impact analysis, it should be calibrated by Eq.(1) below.

\[
ft' = C \cdot ft(\text{obs.}) \quad (1)
\]

- \(ft'\): calibrated tensile strength used in the missile impact analysis
- \(C\): coefficient aimed to calibrate concrete tensile strength, \(C = ft(\text{obs.}) / ft(\text{calc.})\)
- \(ft(\text{obs.})\): concrete tensile strength observed from splitting test, characteristics of the concrete specimen used in the splitting test is almost the same as the concrete characteristics of the RC slab used in the missile impact test.
- \(ft(\text{calc.})\): concrete tensile strength calculated by the simulation analysis to the splitting test aimed to obtain \(ft(\text{obs.})\).

It was considered that the calibrated tensile strength (\(ft'\)) should be used in the concrete constitutive models of missile impact analyses.

4. Validation of impact analysis models using the improved concrete constitutive model

Simulation analyses were conducted using the improved concrete constitutive model and its results were compared with test results of the missile impact tests. The impact tests (punching and bending) were conducted in the IRIS2010. In the punching test, the missile passed through the RC slab, on the other hand in the bending test, the missile did not penetrate into the RC slab. Outline of the impact test results are shown in Fig.-5 and Fig.-6.

Outline of our impact analysis models and material models are shown in Fig.-7 and Fig.-8, respectively. In the impact analyses, the improved concrete constitutive model and \(ft'\) calculated by Eq.(1) were used.

![Concrete crack pattern](image1)

(a) Concrete crack pattern

Fig.-5 Punching impact test (P1) results (provided by OECD/NEA)

![Concrete crack pattern](image2)

(a) Concrete crack pattern

Fig.-6 Bending impact test (B1) results (provided by OECD/NEA)
Rebar and Missile: Formulation of the Plastic Stain based on the nature strain (N.S.)

Main rebar

Transverse rebar

Beam element

Concrete (in-house program)

There is no transverse rebar.

Bending test simulation model

(a) Punching test simulation model

Concrete (in-house program) Rebar (punching and bending in common) (Example of bending)

(a) Material constitutive relation

- Concrete: Formulation of the “Fujimoto-Yamaguchi”
- Rebar and Missile: Formulation of the “Johnson-Cook”

(b) Strain rate dependency

- Concrete: based on the equivalent strain ($\varepsilon_{eq}$)

$$\varepsilon_{eq} = \frac{2}{3}\left[\varepsilon_{eq}^2 + \varepsilon_{eq}^{-2} + \varepsilon_{eq}^{\infty} - (\varepsilon_{eq}^2 + \varepsilon_{eq}^{-2} + \varepsilon_{eq}^{\infty} + \varepsilon_{eq}^{-N} + \varepsilon_{eq}^{\infty} + \varepsilon_{eq}^{-N} + \varepsilon_{eq}^{\infty}) + 3\left(\varepsilon_{eq}^2 + \varepsilon_{eq}^{-2} + \varepsilon_{eq}^{\infty}\right)^2\right]^{1/2}$$

The values of $\varepsilon_{eq}$ were set based on an experience. (the value “2.0” was set for the punching test)

- Rebar: based on the fracture strain of the rebar

(c) Numerical erosion

Fig. -8 Material models in the impact analysis models

Fig. -7 Impact analysis models (punching and bending)

Support (beam, shell and solid element)

RC slab (solid element)

Missile (shell element)

Overview

Main rebar

Transverse rebar

Beam element

There is no transverse rebar.

Yield Stress

Spall Fracture

Elastic Limit

Hardening

Softening

Residual Surface

Dynamic Drucker-Prager Failure Surface

Strain rate : 100 [1/s]

Strain rate : 1000 [1/s]
(1) Punching test simulation

Concrete crack patterns of the RC slab and crushed state of the missile are shown in Fig.-9. The behaviour of the RC slab and the missile obtained by the simulation analysis to the punching test were similar to the test behaviour. Residual velocity time histories of the missile, examples of the displacement time histories of the RC slab and the rebar strain time histories are shown in Fig.-10, 11 and 12, respectively. All the analysis results were in better agreement with the test results than analysis results of the IRIS2010.

![Fig.-9 Fracture states of the impact analysis simulation results (P1)](image)

![Fig.-10 Residual velocity time histories of the missile (P1)](image)
Test states after the crush

Displacement (W2)

Displacement (W3)

Strain of the rebar (D5)

Strain of the rebar (D6)

Fig. 11 Displacement time histories of the RC slab (P1)

Fig. 12 Strain time histories of the rebar (P1)
(2) Bending test simulation

Concrete crack patterns of the RC slab and crushed state of the missile are shown in Fig.-13. The behaviour of the RC slab and the missile obtained by the simulation analysis to the bending test were similar to the test behaviour. Examples of the displacement time histories of the RC slab and the rebar strain time histories are shown in Fig.-14 and 15, respectively. All the analysis results were in better agreement with the test results than analysis results of the IRIS2010.

Fig.-13 Fracture states of the impact analysis simulation results (B1)

Fig.-14 Displacement time histories of the RC slab (B1)
5. Conclusion

Improving the concrete constitutive model of the IRIS2010 based on the concrete fracture test results newly provided by OECD/NEA, simulation analyses to the missile impact tests were conducted. As a result, these impact analysis results were in better agreement with the test results than the analysis results of the IRIS2010.

It seemed to show that in case of only using the spall fracture criteria as a concrete tensile fracture in an impact analysis model, setting of concrete constitutive model is particularly important and concrete tensile strength obtained by splitting (Brazilian) test should be calibrated by a calibration formula such as Eq.(1) proposed in this paper.
1. Introduction

This report describes the procedures and results of the IRIS-2012 benchmark project which is the continued project of IRIS-2010 planned to improve the existing FE models and develop simplified approach.

Within the scope of the work, uniaxial and tri-axial concrete tests were simulated using three concrete material models such as concrete damage rel. 3(#72r3), Winfrith concrete(#84), and CSCM concrete(#159). Then, impact simulations of IRIS-2010 experiments (VTT-IRSN-CNSC Punching P1 and VTT-IRNS Bending B1) were re-performed to improve the accuracy of the simulation results and reduce the computation time.

To reduce the computation time, one quarter of the specimen is modelled considering symmetric condition, and loading function with loading plate is adopted instead of modelling of missiles. With this approach, computation time could be drastically reduced without significant sacrifice of accuracy of the simulation results.

2. Simulation of uniaxial and tri-axial concrete test

Uniaxial and tri-axial concrete tests were simulated using three concrete material models such as concrete damage rel. 3(#72r3), Winfrith concrete(#84), and CSCM concrete(#159). As shown in the Figure 1, one quarter of the specimen is modelled. Loading is applied to the model using the *BOUNDARY_PRESCRIBED_MOTION command, and confining pressure is applied using the *LOAD_SEGMENT command.

Stress-strain curve is obtained using the applied displacement and reaction output. Calculated
stress-strain curves of various concrete models and confining pressure are shown in Figure 2 with the test results. As shown in the figure, concrete damage rel.3 model shows the most similar trend with the test results.

![Stress-strain curves of concrete test](image)

(a) Concrete damage rel.3(#72r3) model
(b) Winfrith concrete(#84) model
(c) CSCM concrete(#159) model

Figure 2. Stress-strain curves of concrete test

3. Impact simulation

3.1 Results of the previous simulation

The calculated displacement responses of previous simulations performed during the IRIS-2010 project are shown with the measured responses in Figure 3. As mentioned in the benchmark synthesis report authored by IRSN and as shown in the Figure 3, maximum displacement values are similar between the calculated and measured responses. However the differences of residual displacements are relatively larger than those of the maximum displacements. Also, the vibration frequencies of calculated responses are much higher than that of the measured responses. Therefore, the main objective of the continued work was reducing the differences of residual displacement and vibration frequency. Another objective was simplifying the FE model for reducing the computation time.
3.2 Simplification of FE model

To reduce required computation time, the existing model previously built during IRIS-2010 project was reduced to one quarter of the full model, and symmetric boundary condition is applied. After confirming that the two models, full model and 1/4 model, give reasonably consistent results, loading function with rigid loading frame replacing missile modeling is adopted.

Figure 4(a) and (b) show the shapes of the adopted rigid loading plates. Fictitious rigid loading plate is devised to make the loaded area similar with that of missile modeling approach regardless of the mesh size of the slab model. Also it is possible to load the slab continuously even if the elements under load are eroded with this loading plate approach.

In the Figure 4(c) and (d), loading function of bending test and punching test are shown with load history acquired from missile modeling approach.

Calculated slab displacements of three cases such as 1) full modeling with missile model, 2) 1/4 modeling with missile model, and 3) 1/4 modeling with loading function, are comparatively shown in Figure 5. CSCM concrete model is utilized in these analyses. Maximum values of each case are listed in Table 1, and consumed computation time of each case are listed in Table 2. As shown in the Figure 5, Table 1, and Table 2, computation time could be drastically reduced without significant sacrifice of accuracy of the simulation results.
Figure 4. loading plate and loading function

Figure 5. Comparison of the displacement response
Table 1. Comparison of the maximum displacements (mm)

<table>
<thead>
<tr>
<th>Case</th>
<th>Full, missile</th>
<th>1/4, missile</th>
<th>1/4, loading fn.</th>
<th>Test</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bending</td>
<td>25.0</td>
<td>30.7</td>
<td>29.3</td>
<td>27.0</td>
</tr>
<tr>
<td>Punching</td>
<td>3.7</td>
<td>3.6</td>
<td>4.9</td>
<td>5.2</td>
</tr>
</tbody>
</table>

Table 2. Comparison of the computation time (hrs)

<table>
<thead>
<tr>
<th>Case</th>
<th>Full, missile</th>
<th>1/4, missile</th>
<th>1/4, loading fn.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bending</td>
<td>33</td>
<td>14</td>
<td>1</td>
</tr>
<tr>
<td>Punching</td>
<td>441</td>
<td>42</td>
<td>3</td>
</tr>
</tbody>
</table>

3.3 Consideration of the supporting structure

Although the computation time is largely reduced by 1/4 modeling and loading function approach, the vibration frequency is still considerably higher than the experimental results. To match the vibration frequency of the simulation result with that of experimental result, supporting frame and column is considered in the FE model. With this consideration, global vibration of supporting structure mainly caused by the axial deformation of the columns could be included in the impact response of the slab.

FE Model with supporting structure is shown in Figure 6. As shown in the Figure, supporting structure is considered in the simplest way with single beam element and one point mass element. The input value of the point mass is acquired from the document provided from the organizing committee (Figure 7). Since the supporting frame has much higher stiffness than the slab, constraint condition is applied instead of modeling the frames as shown in the Figure 6.

![Figure 6. Modeling of the supporting structure](image-url)
The displacement results of the two models, the model with and without supporting structure, are compared in the Figure 8. Also CSCM concrete model was utilized in these analyses. As expected, major vibration frequency is lowered and closed to the experimental results.

![Figure 8](image)

**Figure 8. Effect of supporting structure modeling (bending test case)**

3.4 Effect of material model of concrete

1/4 model with loading function and supporting structure is decided as the final model, and the three concrete models previously considered are applied to the model to observe the effect of concrete model. Since the erosion function is included only in the CSCM model, additional *add_erosion* command is used for the other two concrete models, concrete damage rel.3 and Winfrith concrete model. The element deletion criterion is set as 0.10 for compressive strain and 0.05 for tensile strain. In case of CSCM concrete, included erosion capability is utilized with the input value of erode parameter 1.4.

Resulting time histories of displacement responses are shown in the Figure 9 and failure shapes are show in the Figure 10 and 11. As shown in the figures, concrete damage rel.3
model shows excessive damage for bending case, and CSCM concrete model shows larger residual displacement and lower rebound behavior.

Although the effect element deletion criterion is not clearly identified, Winfrith concrete model shows most similar pattern of displacement response with the measured results so far. However, in view of damage pattern, CSCM concrete model shows the most realistic results.

![Figure 9. Effect of concrete model on displacement responses](image)

![Figure 10. Effect of concrete model on damage pattern (bending test)](image)
Figure 10. Effect of concrete model on damage pattern (bending test) (continued)

Figure 11. Effect of concrete model on damage pattern (punching test)
Figure 11. Effect of concrete model on damage pattern (punching test) (continued)

4. Lessons learned

Tentative conclusions and lessons learned through the previously mentioned procedure are as follows.

1) Computation time could be reduced considerably without significant sacrifice of accuracy of the simulation results by adopting 1/4 model and loading function with loading plate approach.

2) More realistic vibration frequency of the slab displacement response could be simulated by considering support columns and frame mass. Therefore, it is reasonable to think that the major vibration frequency of the slab response is governed by the vibration of the supporting structure rather than vibration of the slab itself.

3) In case of concrete test simulation, concrete damage rel.3 model showed most similar behaviour with the test results, however highly overestimated damage pattern was observed with this concrete model with additional *mat_add_erosion command.

4) By using Winfrith concrete model, the most reliable displacement responses were obtained in view of both maximum value and residual value. However, damage extent or scabbing area was relatively underestimated.

5) CSCM concrete model could effectively be utilized for estimating maximum displacement and failure mode such as penetration, perforation, and scabbing. However CSCM model did not give reasonable results of residual displacement, or rebound behaviour.

6) For the considering simulation cases, element deletion criterion is presumed to be one of the most important factors that affect the simulation result. Therefore an in-depth study about the element deletion criterion is planned.
1. Introduction

This document provides a brief summary of the modeling assumptions and simulation results for the analyses performed by Sandia National Laboratories (SNL) for the United States (US) Nuclear Regulatory Commission (NRC) in support of the IRIS 2012 workshop. The report is broken down into three major sections. The first section discusses the concrete cylinder test modeling. The second section discusses the detailed modeling of the two missile impact tests performed for the IRIS 2010 (Vepsä, 2010a), (Vespä, 2010b). The third section discusses a simplified methodology developed for modeling the two missile impact tests.

2. Concrete Cylinder Test Model

The concrete cylinder test model was assembled to calibrate the concrete material constitutive model used in the detailed missile impact simulations with concrete test data provided by the IRIS organizing committee (CSNI, 2012). The data was obtained from three unconfined compression tests and six tri-axial compression tests. The details of the model constructed and the analyses performed are outlined next.

2.1. Finite Element Model

The explicit transient dynamics finite element code LS-DYNA (LSTC, 2010) was used to perform the concrete cylinder test modeling. While the use of an explicit transient dynamics code does not lend itself to load regimes that are quasi-static in nature, LS-DYNA was used because it is the code that was utilized for the detailed missile impact modeling discussed later.

2.1.1. Mesh

The concrete cylinder finite element model employed quarter symmetry about the x-z and y-z planes and was constructed with 1536 8-noded hexahedral single point under-integrated elements (Figure 2.1). The average element volume was 87 mm$^3$, and the element aspect ratio varied from 1.03 to 1.56. The finite element model had a length equal to that of the standard concrete cylinder test (140 mm) and had an equivalent radial dimension (represented in quarter symmetry) equal to the test specimen’s diameter (70 mm).
2.1.2. Loads and Boundary Conditions

Symmetry boundary conditions were applied to the symmetry planes of the model. Confining pressures (when present) were applied to all exterior surfaces of the cylinder with exception of the two symmetry planes. Axial loads were applied via enforced displacements to the end surfaces of the cylinder, which also had displacement constraints applied that allowed the nodes on those surfaces to move only in the z-direction.

2.1.3. Material Model Input

The Karagözian & Case (K&C) concrete material model (*MAT_072R3) was chosen to represent the concrete material (LSTC, 2010). The K&C model is a three-invariant model that makes use of three shear failure surfaces (i.e., yield, maximum, and residual) to describe the concrete behavior over a range of confining pressures. Figure 2.2 shows the shear failure surfaces used in the model along with the other material model input parameters. No strain rate effects were included in the concrete material model.

![Concrete Properties Table](image)

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Unconfined Compressive Stress</td>
<td>69.0</td>
</tr>
<tr>
<td>Compressive Stress</td>
<td>79.0</td>
</tr>
<tr>
<td>Density</td>
<td>2260</td>
</tr>
<tr>
<td>Unconfined Tensile Stress</td>
<td>4.04</td>
</tr>
<tr>
<td>Young's Modulus</td>
<td>39315</td>
</tr>
<tr>
<td>Specific Fracture Energy</td>
<td>103.8</td>
</tr>
</tbody>
</table>

2.1.4. Analysis

One unconfined compression and four triaxial compression tests, with confining pressures of 15.5, 26.0, 47.0, and 100.0 MPa, were performed. In all tests except the unconfined compression test, the confining pressure was applied first, followed by the application of the axial load. The axial load was applied at a rate of \(1.0 \times 10^{-4}\) \(1/\text{msec}\) which is not quasi-static, but allowed the analysis to proceed in a timely fashion without deviating excessively from a quasi-static load application rate.

2.2. Results

Results of the concrete cylinder test numerical simulations are shown in Figure 2.3. As seen in the figure the fit between the test data and numerical predictions is reasonably good. There are some important discrepancies to note, particularly at higher confining pressures, where the onset of yield occurs at too high a stress. This discrepancy is directly attributable to the poor fit of the yield failure curve with the test data over the range of confining pressures as seen in Figure 2.2. In addition, there are suspect numerical features that arise at higher confining pressures that may be attributable to the high strain rate imparted to the specimen \(1.0 \times 10^{-4}\) \(1/\text{msec}\) when the axial load is applied.
2.3. Conclusions

In general, the K&C material constitutive model input parameters determined during the concrete cylinder model do a reasonable good job of predicting the response of the concrete under unconfined and triaxial test conditions. The material input parameters determined during this phase were used to define the concrete material properties for the detailed missile impact modeling discussed next.

3. Detailed Missile Impact Model

Detailed finite element models representing the two VTT missile impact tests (flexural and punching) were constructed in order to simulate each test. The details pertaining to these models and the analyses performed are outlined next.

3.1. Finite Element Model

The explicit transient dynamics finite element code LS-DYNA was used to perform the detailed missile impact modeling. The details of the finite element model are outlined below.

3.1.1. Mesh

The VTT flexural and punching test models were both quarter-symmetric. The flexural test model incorporated a soft missile comprised of 12,256 4-noded Belytschko-Tsay shell elements, with five integration points through the thickness, and 3,264 under-integrated 8-noded hexahedral elements, whereas the punching test model missile was constructed of 21,681 under-integrated 8-noded hexahedral elements (Figure 3.1). The concrete targets in both models were constructed using under-integrated 8-noded hexahedral elements. The target in the flexural test comprised 162,000 elements, whereas the target in the punching model comprised 267,300 elements (See Figure 3.2 for the punching model target. The flexural model target is not shown but is similar). Both targets utilized Hughes-Liu beam elements with a 2x2 Gauss quadrature scheme to represent the reinforcing steel (See Figure 3.2 for the punching model reinforcing steel. The flexural model reinforcing steel is not shown but is similar but adds transverse-direction reinforcing bars). The flexural model made use of 3,112 elements to represent the reinforcing steel, whereas the punching model made use of 1,104 elements. The reinforcing steel beam elements were imbedded in the target using the *CONSTRAINED_LAGRANGE_IN_SOLID option available in LS_DYNA.
3.1.2. **Loads and Boundary Conditions**

Both the VTT flexural and punching model targets were constrained by applying three degree-of-freedom displacement constraints to regions of nodes on the top at bottom surfaces of the target along their outer edges (Figure 3.3). Because both models were quarter-symmetric, symmetry boundary conditions were applied to all target and missile surfaces and edges lying on the symmetry planes. The missile components of each numerical impact model were given an initial velocity equal to the missile velocity specified for each test (i.e., flexural, missile initial velocity = 110.9 m/sec; and punching, missile initial velocity = 135.0 m/sec). Loads were generated in the model via contact between the missile and target.
3.1.3. Material Model Input

The impact scenario target concrete, target reinforcing steel, and missile steel properties used in the calculations are listed in Table 3-1. Material property data used was taken from the VTT Reports (Vepsä, 2010a), (Vespä, 2010b). Three different concrete material models were used to simulate the response of the concrete within the target. These models are the K&C (*MAT_072R3), Winfrith without strain-rate effects (*MAT_085), and the Continuous Surface Cap Model (*MAT_159). Using the concrete model inputs developed from the cylinder test modeling, a complete input parameter set was constructed for the *MAT_72R3 and *MAT_159 concrete material models. The *MAT_85 concrete model input parameters were automatically generated by LS_DYNA for the unconfined compressive strength, unconfined tensile strength, Young's modulus, and density values for the concrete. Figure 3.4 shows the ultimate compressive strength curves utilized for each concrete model and Figure 3.5 shows the volumetric response for each model. Both the *MAT_085 and the *MAT_159 concrete models did not include strain rate effects, whereas the *MAT_072R3 did. Table 3-2 lists the dynamic increase factors used with the *MAT_072R3 material model.

Table 3-1 Impact Scenario Model Properties

<table>
<thead>
<tr>
<th>Target Concrete</th>
<th>VTT Flexural</th>
<th>VTT Punching</th>
</tr>
</thead>
<tbody>
<tr>
<td>Unconfined Compressive Stress, ( f'_{c} ) (MPa) [cylindrical specimen]</td>
<td>69.0</td>
<td>69.0</td>
</tr>
<tr>
<td>Unconfined Compressive Stress, ( f'_{c} ) (MPa) [cubic specimen]</td>
<td>79.0</td>
<td>79.0</td>
</tr>
<tr>
<td>Density, ( \rho ) [kg/m³]</td>
<td>2260</td>
<td>2260</td>
</tr>
<tr>
<td>Poisson's Ratio, ( \nu )</td>
<td>0.22</td>
<td>0.22</td>
</tr>
<tr>
<td>Unconfined Tensile Stress, ( f'_{t} ) (MPa)</td>
<td>4.04</td>
<td>4.04</td>
</tr>
<tr>
<td>Young's Modulus(^{1}), ( E ) (MPa)</td>
<td>39315</td>
<td>39315</td>
</tr>
<tr>
<td>Specific Fracture Energy, ( G_{F} ) (N/m)</td>
<td>103.8</td>
<td>103.8</td>
</tr>
<tr>
<td>Target Reinforcing Steel</td>
<td>VTT Flexural</td>
<td>VTT Punching</td>
</tr>
<tr>
<td>Material Designation</td>
<td>A500 HW</td>
<td>A500 HW</td>
</tr>
<tr>
<td>Young's Modulus, ( E ) (MPa)</td>
<td>219000</td>
<td>210000</td>
</tr>
<tr>
<td>Density, ( \rho ) [kg/m³]</td>
<td>7843</td>
<td>7843</td>
</tr>
<tr>
<td>Poisson's Ratio, ( \nu )</td>
<td>0.29</td>
<td>0.29</td>
</tr>
<tr>
<td>Yield Stress, ( \sigma_{y} ) (MPa)</td>
<td>600.0</td>
<td>535</td>
</tr>
<tr>
<td>Ultimate True Stress, ( \sigma_{uts} ) (MPa)</td>
<td>747</td>
<td>666</td>
</tr>
<tr>
<td>Failure Strain (%)</td>
<td>12</td>
<td>12</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Missile</th>
<th>VTT Flexural</th>
<th>VTT Punching</th>
</tr>
</thead>
<tbody>
<tr>
<td>Material Designation</td>
<td>Stainless Steel EN 1.4432</td>
<td>Carbon Steel S355J2H</td>
</tr>
<tr>
<td>Young's Modulus, ( E ) (MPa)</td>
<td>200000</td>
<td>200000</td>
</tr>
<tr>
<td>Density, ( \rho ) [kg/m³]</td>
<td>7850</td>
<td>7738</td>
</tr>
<tr>
<td>Poisson's Ratio, ( \nu )</td>
<td>0.29</td>
<td>0.29</td>
</tr>
<tr>
<td>Yield Stress, ( \sigma_{y} ) (MPa)</td>
<td>302</td>
<td>500</td>
</tr>
<tr>
<td>Ultimate True Stress, ( \sigma_{uts} ) (MPa)</td>
<td>513</td>
<td>N/A</td>
</tr>
</tbody>
</table>

\(^{1}\)Young's Modulus, \( E \), was computed from the American Concrete Institute formula \( E = 5.7 \times 10^{4} \cdot \sqrt{f'_{c}} \), where \( f'_{c} \) is the cylindrical specimen unconfined compressive strength specified in units of lb/in².
Figure 3.4 Ultimate Compressive Strength Response for the Concrete Material Models.

Figure 3.5 Volumetric Response for the Concrete Material Models.
Table 3-2 Dynamic Increase Factors for the *MAT_072R3 Concrete Model.

<table>
<thead>
<tr>
<th>Strain Rate (1/msec)</th>
<th>Dynamic Increase Factor</th>
<th>VTT Flexural</th>
<th>VTT Punching</th>
</tr>
</thead>
<tbody>
<tr>
<td>-30</td>
<td>8.85593</td>
<td>8.85593</td>
<td></td>
</tr>
<tr>
<td>-0.3</td>
<td>8.8593</td>
<td>8.8593</td>
<td></td>
</tr>
<tr>
<td>-0.1</td>
<td>5.9347</td>
<td>5.9347</td>
<td></td>
</tr>
<tr>
<td>-0.03</td>
<td>3.9729</td>
<td>3.9729</td>
<td></td>
</tr>
<tr>
<td>-0.01</td>
<td>2.7547</td>
<td>2.7547</td>
<td></td>
</tr>
<tr>
<td>-0.003</td>
<td>1.8441</td>
<td>1.8441</td>
<td></td>
</tr>
<tr>
<td>-0.001</td>
<td>1.2787</td>
<td>1.2787</td>
<td></td>
</tr>
<tr>
<td>-0.0001</td>
<td>1.2273</td>
<td>1.2273</td>
<td></td>
</tr>
<tr>
<td>-1.00E-05</td>
<td>1.17818</td>
<td>1.17818</td>
<td></td>
</tr>
<tr>
<td>-1.00E-06</td>
<td>1.1308</td>
<td>1.1308</td>
<td></td>
</tr>
<tr>
<td>-1.00E-07</td>
<td>1.0854</td>
<td>1.0854</td>
<td></td>
</tr>
<tr>
<td>-1.00E-08</td>
<td>1.0418</td>
<td>1.0418</td>
<td></td>
</tr>
<tr>
<td>0</td>
<td>1</td>
<td>1</td>
<td></td>
</tr>
<tr>
<td>3.00E-08</td>
<td>1.0186</td>
<td>1.0186</td>
<td></td>
</tr>
<tr>
<td>1.00E-07</td>
<td>1.0551</td>
<td>1.0551</td>
<td></td>
</tr>
<tr>
<td>1.00E-06</td>
<td>1.0929</td>
<td>1.0929</td>
<td></td>
</tr>
<tr>
<td>0.0001</td>
<td>1.1321</td>
<td>1.1321</td>
<td></td>
</tr>
<tr>
<td>0.001</td>
<td>1.1726</td>
<td>1.1726</td>
<td></td>
</tr>
<tr>
<td>0.003</td>
<td>1.1925</td>
<td>1.1925</td>
<td></td>
</tr>
<tr>
<td>0.01</td>
<td>1.2146</td>
<td>1.2146</td>
<td></td>
</tr>
<tr>
<td>0.03</td>
<td>1.2352</td>
<td>1.2352</td>
<td></td>
</tr>
<tr>
<td>0.1</td>
<td>1.8451</td>
<td>1.8451</td>
<td></td>
</tr>
<tr>
<td>0.3</td>
<td>2.6611</td>
<td>2.6611</td>
<td></td>
</tr>
<tr>
<td>30</td>
<td>2.6611</td>
<td>2.6611</td>
<td></td>
</tr>
</tbody>
</table>

The lightweight fill concrete used in the punching test impact missile was modeled using *MAT_016. This material model takes as input the unconfined compressive *cylindrical specimen* strength which was assumed to be 20 MPa. A lightweight concrete density was selected such that the specified total missile mass equaled 47.0 kg. A Young’s Modulus of 10,120 MPa was also assumed.

The reinforcing steel was modeled with the *MAT_024 material model. Figure 3.6 shows the true stress vs. strain relationships used in the simulations. This model allows for strain rate effects to be included through a Cowper-Symonds rate model as \( \sigma_{\text{dynamic}} / \sigma_{\text{static}} = 1 + (d\varepsilon/dt) / C \), where \( \sigma_{\text{dynamic}} \), \( \sigma_{\text{static}} \), \( d\varepsilon/dt \), \( C \), and \( P \) are the dynamic yield strength, static yield strength, strain rate, first Cowper-Symonds parameter, and second Cowper-Symonds parameter, respectively. The strain-rate effect was only included for the flexural test model. The missile steel material was also modeled using the *MAT_024 material model. Figure 3.6 shows the true stress vs. equivalent plastic strain relationships used. The S355J2H carbon steel component of the flexural test model missile was modeled using *MAT_003. Both missile steel material models (*MAT_003 and *MAT_024) allow strain rate effects to be included using a Cowper-Symonds relationship. Table 3-3 shows the rate parameters used in each case.
Figure 3.6 Steel Material True Stress vs. Effective Plastic Strain Curves.

Table 3-3 Missile Finite Element Model Material Strain-Rate Parameters.

<table>
<thead>
<tr>
<th>Missile</th>
<th>VTT Flexural</th>
<th>VTT Punching</th>
</tr>
</thead>
<tbody>
<tr>
<td>Material Type</td>
<td>Stainless Steel EN 1.4432</td>
<td>Carbon Steel S355J2H</td>
</tr>
<tr>
<td>Material Model</td>
<td>*MAT_024</td>
<td>*MAT_003</td>
</tr>
<tr>
<td>Cowper-Symonds, C (1/msec)</td>
<td>0.1</td>
<td>4.04 x 10^{-2}</td>
</tr>
<tr>
<td>Cowper-Symonds, P</td>
<td>10</td>
<td>5</td>
</tr>
</tbody>
</table>
3.2. **Analysis**

Two sets of analyses were performed. For each test one model was analyzed for each of the three concrete constitutive models discussed earlier (*MAT_072R3, *MAT_085, and *MAT_159).

3.3. **Results**

3.3.1. **VTT Flexural Test**

Three simulations were completed. All of these simulations included strain-rate effects for the reinforcing steel (Cowper-Symonds parameters $C = 40.4 \times 10^{-3}$ 1/msec and $P = 5$), but only the *MAT_72R3 model include concrete strain rate effects. Table 3-4 lists key model response values, Figure 3.7 shows the missile deformation at the shock termination time, Figure 3.8 shows target displacements at several locations, Figure 3.9 shows the concrete strains in the target at several locations, Figure 3.10 shows strains in the steel reinforcing at several locations, and Figure 3.11 shows damage in the targets at the shock termination time.

<table>
<thead>
<tr>
<th>Table 3-4 Key Results from the VTT Flexural Test Simulations.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Response</td>
</tr>
<tr>
<td>-----------</td>
</tr>
<tr>
<td>Rebound Velocity (m/sec)</td>
</tr>
<tr>
<td>$t_{shock} = $ Bounce/Shock Duration (msec)</td>
</tr>
<tr>
<td>$L_T$ of Missile (mm)</td>
</tr>
<tr>
<td>$H_T$ of Missile (mm)</td>
</tr>
<tr>
<td>$L_T + H_T$ (mm)</td>
</tr>
</tbody>
</table>

Figure 3.7 VTT Flexural Model Missile Deformation.
Figure 3.8 VTT Flexural Test Model Target Displacements.
Figure 3.9 VTT Flexural Test Model Target Concrete Strains.
Figure 3.10. VTT Flexural Test Model Steel Reinforcing Strains.
In general, the numerical simulations under predict the missile crush-up (Table 3-4 and Figure 3.7). Both the *MAT_072R3 and *MAT_159 models accurately predict the target displacements and frequency of response (Figure 3.8), whereas the *MAT_085 model is in less agreement with the test data. Concrete strains (Figure 3.9) are not well predicted by any of the models, whereas steel reinforcement strains are in somewhat better agreement (Figure 3.10). Finally, damage to the target is generally consistent with test data (Figure 3.11) although the damage plots are always subject to some interpretation.

3.3.2. VTT Punching Test

Three simulations were completed. None of these simulations included strain-rate effects for the reinforcing steel and only the *MAT_72R3 model included strain rate effects in the concrete. Several preliminary simulations were performed that included strain rate effects for the steel reinforcement but in all cases the missile either bounced off the target after some amount of target penetration or
became stuck in the target. Two of the target concrete constitutive laws (*MAT_072R3 and *MAT_085) do not permit implicit erosion control. The *MAT_159 material model has several means of controlling erosion. In order to equilibrate in some fashion this control between the three different target concrete constitutive models, the LS-DYNA control, *MAT_ADD_EROSION was used. This method allows for over thirteen different criteria, or any combination thereof, to invoke erosion of the target elements. The *MAT_072R3 and *MAT_159 models used erosion controls based on the maximum shear strain, $\varepsilon_{pssh} = 0.6$ (i.e., 60%), whereas, the *MAT_085 model used a value of 0.5 (note that a value of $\varepsilon_{pssh} = 0.6$ caused massive element erosion, and deletion of the entire target). Table 3-5 lists key model response values, Figure 3.12 shows the missile deformation after impact, Figure 3.13 shows target displacements at several locations, Figure 3.14 shows strains in the steel reinforcing at several locations, and Figure 3.15 shows damage in the targets at the 11 msec.

### Table 3-5 Key Results From the VTT Punching Test Simulations.

<table>
<thead>
<tr>
<th>Response</th>
<th>Experiment</th>
<th>Simulations</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>P1</td>
<td>P2</td>
</tr>
<tr>
<td>Exit Velocity (m/sec)</td>
<td>33.8</td>
<td>45.3</td>
</tr>
<tr>
<td>Missile Nose Bulging</td>
<td>YES</td>
<td>YES</td>
</tr>
</tbody>
</table>

*MAT_072R3

*MAT_085

*MAT_0159

Figure 3.12 VTT Punching Test Missile Deformation.
Figure 3.13 VTT Punching Test Model Target Displacements.
Figure 3.14 VTT Punching Test Model Reinforcing Steel Strains.
Figure 3.15 VTT Punching Test Model Target Damage.

*MAT_072R4 (t = 11.0 msec)

*MAT_85 (t = 4.25 msec)

*MAT_159 (t = 11.0 msec)
In general, the numerical simulations did a good job of predicting the deformation response of the missile (Figure 3.12) and a decent job at predicting its exit velocity (Table 3-5). Target displacement were only available for the *MAT_072R3 model and it under predicted the target displacement by a significant amount (Figure 3.13). Steel reinforcement strains predictions were in modest agreement with test measured values but varied significantly in some instances (Figure 3.14). Finally, damage to the target was generally consistent with test data (Figure 3.15).

3.4. Detailed Modeling Conclusions

In many instances, detailed response parameters were not precisely reproduced by the model. However, the detailed finite element methodology was able to reproduce relatively accurately the general response characteristics of the missile and target, and to do so despite the fact that many of the critical model input parameters were unknown to the analyst. One particular area of uncertainty seems to be related to how material degradation and failure should be represented in these numerical methods. Often, the analyst is left to tune damage or failure parameters for some failure model that are not directly relatable to any measurable quantity simply to achieve a good visual fit with test observed behavior. This problem is particularly acute for situations in which significant damage is imparted to the impacted structure (punching test model). Also of concern is the large amount of material response data required to construct these models. In particular, high strain rate material response data is often not available and for complex materials such as concrete, test data covering the range of possible response scenarios is difficult if not impossible to attain. Despite these limitations, the methodology does appear to produce results reasonable enough to make accurate and useful predictions.

4. Simplified Missile Impact Model

The use of detailed finite element models to predict the response of moderately complex steel reinforced concrete structure is not practical. For this reason, simplified methods are required. The investigation of one such methodology is presented below.

4.1. Outline of Simplified Methodology

The simplified procedure consists of the steps listed below.

1) Determine the one-way bending moment-curvature relationship for a unit width section of the target.

2) Establish parameters for a bi-linear material that when used to represent the target will approximately mimic the moment-curvature response calculated in step one.

3) Create a simplified model of the target using shell elements, assigning the bi-linear material response calculated in step 2 to the target elements.

4) Assign a maximum equivalent plastic strain failure criteria to the target elements that corresponds with the curvature at failure of the target in bending.

5) Assume the missile is rigid and calculate the failure load and force versus missile penetration depth using the empirical equations outlined by Li and Tong (1994) and Forrestal et. al. (2003).

6) Assign a maximum through thickness shear stress failure criteria to the target elements based on the impacting missile’s outer diameter, the target’s undamaged thickness, and the failure load determined in step 4.

7) Create an analytical representation of the impacting missile.

   a. For Rigid Missiles – Represent the missile as analytically rigid. Insert a crushable volume between the rigid missile and the target that when crushed by the missile will result in the correct force versus penetration depth profile calculated in step 5 being applied to the target.
b. For Soft Missiles – Represent the missile with a sufficient number of deformable elements (preferably shells elements for their reduced computational costs) to accurately capture the force versus crush response of the missile.

8) Run the analysis.

4.1.1. Moment-Curvature Relationships and Simplified Target Material Parameters

For the flexural and punching targets in the VTT tests the moment curvature relationships show in Figure 4.1 were calculated. The relationships were constructed using the concrete and steel material response characteristics illustrated in Figure 4.2 which are unmodified from the data provided by the IRIS organizing committee. The simple bi-linear material response relationships shown in Figure 4.3 were created to represent the calculated moment-curvature relationships.

Figure 4.1 VTT Missile Impact Test Target Moment-Curvature Relationships.

Figure 4.2 Concrete and Reinforcing Steel Material Responses Used in Calculation of the Target Moment-Curvature Relationships.
4.1.2. Rigid Missile Impact Response Characteristics

For each of the VTT tests the missile impacting the target was assumed to be rigid and a missile penetration depth versus resisting load relationship calculated using the equations outlined by Li and Tong (Li & Tong, 2003) and Forrestal et. al. (Forrestal, Altman, Gargile, & Hanchak, 1994). Failure was assumed to occur when the axial resistance on the end of the penetrating rigid missile exceeded the shear resistance capability of the remaining undamaged portion of the concrete target and any shear and bending reinforcement bridging the assumed shear failure surface. The target response characteristics calculated via this methodology are shown in Figure 4.4 and Figure 4.5 for the VTT bending and punching tests, respective.
4.2. **Finite Element Model**

Two separate analysis codes were utilized. For the flexural test model version 6.10-1 of the commercially available explicit dynamics finite element program ABAQUS (ABAQUS, 2011) was used. For the punching test model version 4.25.9 of Sandia National Laboratories’ in-house developed explicit dynamics finite element analysis code Sierra/SolidMechanics (Sierra/SM, 2012) was used. Shown in Figure 4.6 are the flexural and punching finite element models produced for each test. Note that the actual models are quarter-symmetric and that a full model has been displayed only for visualization purposes.

![Flexural and Punching Models](image)

**Figure 4.6 VTT Flexural and Punching Test Finite Element Models.**

4.2.1. **Mesh**

The quarter-symmetry VTT flexural model is composed of 6,184 four node reduced integration shell elements (S4R). The missile accounts for the majority of these elements with 4,420 of the total (Figure 4.7). The average element size for the missile shell elements is approximately 10 mm. The target is composed of 1,764 elements with a characteristic length of 25 mm (Figure 4.8). The quarter-symmetry VTT punching model is composed of 3,300 four node reduced integration shell elements and eight node reduced integration hexahedral elements. The missile accounts for nearly half of these elements with 1,536 of the total (Figure 4.7). Of these 1,536 elements, 756 are shell elements and 780 are hexahedral. The missile is made analytically rigid during the analysis. The average element size for the missile elements is approximately 10 mm. The target is composed of 1,764 elements with a characteristic length of 25 mm (Figure 4.8).

![Missile Models](image)

**Figure 4.7 VTT Test Missile Finite Element Model Representations.**
4.2.2. Boundary Conditions

Each quarter-symmetric model target is supported along its outer edge in the vertical direction (z-direction) only. In addition, both models employ appropriate symmetry boundary condition displacement and rotational constraints on both the target and missile along each symmetry boundary plane. Figure 4.8 shows the displacement and rotational boundary conditions applied to the target in each model. For the flexural model an initial velocity equal to 110.15 m/s is applied to the missile. For the punching model an initial velocity equal to 135.9 m/s is applied to the missile. Finally, for the punching model, the missile was made analytically rigid.

![Flexural Boundary Conditions](image1)

![Punching Boundary Conditions](image2)

Figure 4.8 Finite Element Model Target Boundary Conditions.

4.2.3. Material Model Input

Missile steel materials were modeled in a fashion identical to how they were modeled for the detailed simulations discussed above, with the exception that no strain rate dependence was included. For the flexural model target the bi-linear material response in Figure 4.3 was implemented using the Concrete Damage Plasticity Model with the damage parameters defined such that the elastic modulus of the material was reduced as the material deforms plastically. Representing the response of the target in this manner allowed the closure of cracks within the concrete slab to be captured. For the punching model target the bi-linear material response in Figure 4.3 was implemented using a simple elastic-plastic material model. Also for the punching test model a crushable volume was introduced to capture the energy absorbed by the damage induced locally in the target at the missile’s point of contact (Figure 4.7). The response characteristics of this material were determined from the missile axial-penetration-force versus penetration-depth relationship in Figure 4.5. To recreate the missile axial force versus penetration depth profile calculated above the crush strength versus compaction relationship shown in Figure 4.9 was derived and implemented in the model.
4.3. Analysis
Two analyses were performed; one analysis for the VTT flexural test and one analysis for the VTT punching test.

4.4. Results

4.4.1. VTT Flexural Test Model
Table 4-1 lists pertinent response characteristics for the missile and target for the VTT flexure impact case. Figure 4.10 illustrates the model predictions for the response of the missile, and Figure 4.11 illustrates the model predictions for the displacement response of the target plotted alongside the test measured values. Figure 4.12 shows the deformation of the missile and target at the time of shock termination (see Table 4-1). The target is shown with several sets of contours overlain. The central two contour plots show the front and back face tensile equivalent plastic strains. The scales have been set such that any region of the target that is not colored black will be expected to experience some level of crack formation in the concrete. The final target contour plot shows the equivalent plastic strain in the target at the extreme fiber. The scale on this plot has been set such that the peak value listed corresponds with the value at which a bending failure would occur.

Table 4-1 VTT Flexure Model Response Summary.

<table>
<thead>
<tr>
<th>Missile Results</th>
<th>Baseline Model</th>
<th>Test Measured Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shock Duration</td>
<td>21 msec</td>
<td>Not Reported</td>
</tr>
<tr>
<td>Peak Load</td>
<td>842 kN</td>
<td>Not Reported</td>
</tr>
<tr>
<td>Total Impulse</td>
<td>5.59 kN-sec</td>
<td>Not Reported</td>
</tr>
<tr>
<td>Missile Residual Velocity</td>
<td>-6.0 m/sec (Rebound)</td>
<td>Rebound</td>
</tr>
<tr>
<td>LT at Shock Termination</td>
<td>671 mm</td>
<td>955 mm</td>
</tr>
<tr>
<td>HT at Shock Termination</td>
<td>250 mm</td>
<td>185 mm</td>
</tr>
<tr>
<td>Target Results</td>
<td>Baseline Model</td>
<td>Test Measured Value</td>
</tr>
<tr>
<td>Initial Max W1 Displacement</td>
<td>38.6 mm at 14.0 msec</td>
<td>28.9 mm at 13.5 msec</td>
</tr>
<tr>
<td>Initial Max W2 Displacement</td>
<td>29.9 mm at 14.5 msec</td>
<td>20.4 mm at 13.3 msec</td>
</tr>
<tr>
<td>Initial Max W3 Displacement</td>
<td>33.2 mm at 14.0 msec</td>
<td>22.0 mm at 12.9 msec</td>
</tr>
<tr>
<td>Initial Max W4 Displacement</td>
<td>19.9 mm at 15.0 msec</td>
<td>15.3 mm at 12.0 msec</td>
</tr>
<tr>
<td>Initial Max W5 Displacement</td>
<td>25.6 mm at 14.5 msec</td>
<td>19.5 mm at 13.1 msec</td>
</tr>
<tr>
<td>Total Max Support Force</td>
<td>1,783 kN at 9.0 msec</td>
<td>862 kN at 13.7 msec</td>
</tr>
<tr>
<td>Total Max Support Impulse</td>
<td>~12.0(^a) kN-sec</td>
<td>~6.5(^b) kN-sec</td>
</tr>
<tr>
<td>Target Failure (Yes/No)?</td>
<td>No</td>
<td>No</td>
</tr>
<tr>
<td>Target Failure Mode</td>
<td>N/A</td>
<td>N/A</td>
</tr>
</tbody>
</table>

\(^a\) Approximate average of min and max value in 50 to 100 msec range.

\(^b\) Approximate average of min and max value in 20 to 40 msec range.
Figure 4.10 VTT Flexural Model Missile Response.

Figure 4.11 VTT Flexural Model Target Displacement Response.
Figure 4.12 VTT Flexural Model Missile and Target Response.
In general the model does a good job of predicting the VTT flexural test response. The model correctly predicts that the target will withstand the missile’s impact without failure by either punching or bending. The type (cracking of the target concrete without failure by bending or punching) and extent of the damage (localized to the central portion of the back side of the target and to the front side of the target at the supports) predicted by the model is generally consistent with that observed during the test (see Figure 4.12). The model also does a reasonably good job of predicting the magnitude and frequency of the target displacements. Comparison of the model predictions with those measured during the test (see Table 4-1) shows that the model over predicts the displacement of the target by 30% to 50%, but the frequency of the model response is in excellent agreement with the test measured values (see Figure 4.11). What is not well captured by the model is the degradation in the dynamic response over time. In general, the model response is damped out too slowly in comparison with what was observed during the test. This highlights a shortcoming with the simplified method in that the model is not good at capturing the effects that the complex failure mechanisms in steel reinforced concrete have on the response of the structure. Finally, the model did only a modest job at predicting the response of the missile. This is evidenced by the nearly 25% over-crush of the missile in the model versus that observed in the test (see Figure 4.12 and Table 4-1). This discrepancy is likely attributable to the fact that no strain rate dependence was included in the modeling of the missile steel.

4.4.2. VTT Punching Test Model

Table 4-2 lists pertinent response characteristics for the missile and target for the VTT punching impact case. Figure 4.13 illustrates the model predictions for the response of the missile, and Figure 4.14 illustrates the model predictions for the displacement response of the target plotted alongside the test measured values. Figure 4.15 shows the deformation of the missile at the time of shock termination (see Table 4-2) and deformation of the target at the time of peak vertical displacement at location W2. The missile is shown with contours indicating the amount of crush (defined as \( V_{current}/V_{initial} \)) imparted to the crushable volume at the leading end of the missile. Because the missile in modeled as perfectly rigid no deformation of the missile occurs during the analysis. The target is shown with contours indicating the equivalent plastic strain in the target at the extreme fiber. The scale has been set such that the peak value listed corresponds with the value indicating failure of the target in bending.

<table>
<thead>
<tr>
<th>Missile Results</th>
<th>Baseline Model</th>
<th>Test Measured Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shock Duration</td>
<td>0.5 msec</td>
<td>Not Reported</td>
</tr>
<tr>
<td>Peak Load</td>
<td>7,669 kN</td>
<td>Not Reported</td>
</tr>
<tr>
<td>Total Impulse</td>
<td>2.35 kN-sec</td>
<td>Not Reported</td>
</tr>
<tr>
<td>Missile Residual Velocity</td>
<td>77.7 m/sec</td>
<td>33.8 m/sec</td>
</tr>
<tr>
<td>LT at Shock Termination</td>
<td>N/A – Rigid Missile</td>
<td>Not Reported</td>
</tr>
<tr>
<td>HT at Shock Termination</td>
<td>N/A – Rigid Missile</td>
<td>Not Reported</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Target Results</th>
<th>Baseline Model</th>
<th>Test Measured Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Initial Max W1 and W3 Displacement</td>
<td>5.99 mm at 4.5 msec</td>
<td>3.60 mm at 3.6 msec</td>
</tr>
<tr>
<td>Initial Max W2 Displacement</td>
<td>10.97 mm at 4.5 msec</td>
<td>3.96 mm at 5.5 msec</td>
</tr>
<tr>
<td>Initial Max W4 Displacement</td>
<td>4.19 mm at 5.0 msec</td>
<td>3.31 mm at 4.1 msec</td>
</tr>
<tr>
<td>Initial Max W5 Displacement</td>
<td>3.10 mm at 5.0 msec</td>
<td>2.50 mm at 3.3 msec</td>
</tr>
<tr>
<td>Total Max Support Force</td>
<td>2,446 kN at 3.0 msec</td>
<td>1,095 kN at 8.5 msec</td>
</tr>
<tr>
<td>Total Max Support Impulse</td>
<td>~2.0 kN-sec</td>
<td>~4.0 kN-sec</td>
</tr>
<tr>
<td>Target Failure (Yes/No)?</td>
<td>Yes</td>
<td>Yes</td>
</tr>
<tr>
<td>Target Failure Mode</td>
<td>Bending</td>
<td>Punching</td>
</tr>
</tbody>
</table>

* Value based on impulse at 7.5 msec.
Figure 4.13 VTT Punching Model Missile Response.

Figure 4.14 VTT Punching Model Target Displacement Response.
In general the simplified method does a poor job of predicting the VTT punching test response. The model correctly predicts that the target will fail when the missile impacts but the failure mode is by bending and not by punching. The extent of the damage is also poorly predicted (see Figure 4.15). Whereas the model does indicate significant damage to the central portion of the target, including perforation of the target by the missile, the extensive damage observed on the back side of the target in the test is not directly captured by the simplified model and cannot be easily represented in the simplified model framework. The model does a poor job of predicting the magnitude and frequency of the target displacements. Comparison of the model predictions for the maximum target displacements with those measured during the test (see Table 4-2) shows that the model significantly over predicts the displacement of the target, with over predictions being by more than 100% in some instances. In addition, the model predicts a response frequency approximately double that observed in the test (see Figure 4.14). The degradation in the response over time is also not well characterized with the model response not damping out quickly enough. Finally, the model did only a modest job at predicting the response of the missile, with the residual velocity of the missile being approximately 130% higher than observed in the test (see Table 4-2).
4.5. *Simplified Methodology Conclusion*

The simplified methodology for predicting the response of steel reinforced concrete structures to the impact of soft and rigid missiles outlined here has been shown to be relatively accurate in predicting the response characteristics of soft missile impacts into targets in which the target response is dominated by bending and bending type failures, and the target suffers only modest damage. For rigid missile impacts in which punching failure in the target is expected, the simplified methodology as envisioned here appears to be poorly suited. The range of situations over which the methodology is accurate is difficult to predict given the limited data set used in this investigation. For example, it is not clear that this method would work for soft missile impacts involving significant damage to the target. In determining the punching failure mode strength to use in the simplified model, some test response data was utilized to define key input parameters. This data would likely not be available in real world situations.


As with many numerical analyses, there was a significant evolution and progression in developing robust models used to simulate the response of two blind impact scenarios (i.e., flexural and punching) during the course of nearly three years. The first published numerical results from the NRC Team I, simulating the response of these impact scenarios (and the Meppen IV impact scenario (Meppen, 2010)), during year 2010, involved two different numerical techniques and approaches: (1) A traditional finite element analysis, using the LS-DYNA code (LSTC, 2010), employing the Riera method (involving a load function representing the missile impact (Riera, 1968), (Sugano, Tsubota, Kasai, Koshika, Oru, & von Riesman, 1992); and (2) a separate set of numerical analyses based on the peridynamic theory of continuum mechanics implemented in the SNL’s EMU code (Silling & Askari, 2005). Both of these numerical analyses were documented (Rath, 2010), (Silling S., 2010) and presented to the IRIS 2010 participants. The next set of numerical results were generated during the 2011 calendar year, and involved only a finite element approach, but implemented an explicit representation of the missile for each of the three impact scenarios (i.e., Meppen IV, flexural, and punching). The 2011 year numerical analysis effort was fully documented, however not made public to the IRIS participants, since nothing was required during this time period (Rath, IRIS 2011 Numerical Simulation Report, NRC Team I, 2011). The group of numerical analyses performed by the NRC Team I committee, during calendar year 2012, incorporated new concrete target test data, in order to improve numerical models and allow some common calibration to the concrete target response. This effort, year 2012, involved a repeat of the previous calendar year 2011 finite element calculations, using the new concrete target data calibrated models (Rath & Bignell, IRIS 2012 Numerical Simulation Report, NRC Team I, 2012). Also included in the 2012 year analysis effort was a simplified missile impact approach, discussed in the previous section of this report. In order to help organize and clarify all of these numerical analyses, the NRC Team I IRIS numerical analyses are shown below in Table 5-1.

The simulation results from the 2012 effort were significantly better than those produced in 2010. In general, the predictions made by the SNL/NRC teams for the 2010 IRIS round robin were not overly accurate. In the 2010 LS-DYNA model, the use of the Riera method eliminated the ability to predict missile response. In addition, target displacements were often significantly over predicted (both LS-DYNA and EMU simulations) as was the missile exit velocity in the EMU punching simulation. Reinforcing steel strain predictions were also generally not in good agreement with the test measured values. Following the IRIS 2010 workshop significant changes were made to the LS-DYNA model (EMU was not included in the 2012 exercise) that contributed directly to the observed improvement in the 2012 results. The most significant of these changes are the abandonment of the Riera method in favor of an explicit model of the missile as well as the use of improved concrete material model input parameters derived from previously unavailable concrete test data.
<table>
<thead>
<tr>
<th>Designation ID</th>
<th>Impact Scenario</th>
<th>Computational Method</th>
<th>Year</th>
<th>Code</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>MR2010</td>
<td>Meppen IV</td>
<td>Finite Element Method + Riera Method</td>
<td>2010</td>
<td>LS-DYNA</td>
<td>No explicit finite element representation of missile component</td>
</tr>
<tr>
<td>PR2010</td>
<td>Punching</td>
<td>Finite Element Method + Riera Method</td>
<td>2010</td>
<td>LS-DYNA</td>
<td>No explicit finite element representation of missile component</td>
</tr>
<tr>
<td>MP2010</td>
<td>Meppen IV</td>
<td>Peridynamics theory of Continuum Mechanics</td>
<td>2010</td>
<td>EMU</td>
<td>Meshfree</td>
</tr>
<tr>
<td>FP2010</td>
<td>Flexural</td>
<td>Peridynamics theory of Continuum Mechanics</td>
<td>2010</td>
<td>EMU</td>
<td>Meshfree</td>
</tr>
<tr>
<td>PP2010</td>
<td>Punching</td>
<td>Peridynamics theory of Continuum Mechanics</td>
<td>2010</td>
<td>EMU</td>
<td>Meshfree</td>
</tr>
<tr>
<td>MF2011</td>
<td>Meppen IV</td>
<td>Finite Element Method</td>
<td>2011</td>
<td>LS-DYNA</td>
<td>All components represented explicitly using the FE code</td>
</tr>
<tr>
<td>FF2011</td>
<td>Flexural</td>
<td>Finite Element Method</td>
<td>2011</td>
<td>LS-DYNA</td>
<td>All components represented explicitly using the FE code</td>
</tr>
<tr>
<td>PF2011</td>
<td>Punching</td>
<td>Finite Element Method</td>
<td>2011</td>
<td>LS-DYNA</td>
<td>All components represented explicitly using the FE code</td>
</tr>
<tr>
<td>FF2012</td>
<td>Flexural</td>
<td>Finite Element Method</td>
<td>2012</td>
<td>LS-DYNA</td>
<td>All components represented explicitly using the FE code + Updated/Calibrated Concrete Model</td>
</tr>
<tr>
<td>PF2012</td>
<td>Punching</td>
<td>Finite Element Method</td>
<td>2012</td>
<td>LS-DYNA</td>
<td>All components represented explicitly using the FE code + Updated/Calibrated Concrete Model</td>
</tr>
<tr>
<td>FS2012</td>
<td>Flexural</td>
<td>Simplified Missile Impact Model</td>
<td>2012</td>
<td>ABAQUS</td>
<td>See Section 4.1 of this report</td>
</tr>
<tr>
<td>PS2012</td>
<td>Punching</td>
<td>Simplified Missile Impact Model</td>
<td>2012</td>
<td>SNL/SIERRA/SM</td>
<td>See Section 4.1 of this report</td>
</tr>
</tbody>
</table>
Specific lessons learned during the course of the 2010 and 2012 IRIS analysis efforts are listed below.

1. Material models employed in any detailed finite element model need to be able to reproduce the salient characteristics of the response of the materials that they are being used to represent. This includes the behavior of the material under the conditions that they will be subject to (high strain rates, varying levels of confinement, etc.). This is particularly true for the material model used to represent the target concrete in the missile impact simulations, and to a lesser extend any material model used to represent the reinforcing and missile steels. Salient response characteristics include material response as well as degradation and failure behavior. The best way to ensure the suitability of a material model is to compare its response against material test data spanning the entire range of conditions that are applicable for the test case being modeled. This highlights the need for having a sufficient amount of quality material test data.

2. The choice of numerical models used to represent the concrete, and the assumptions inherent to those material models, does have a significant effect on the predicted response, even for concrete models that are considered approximately equivalent (in other words, even when the material models all include the ability to account for confinement of the concrete or rate effects and they are used with a consistent set of material model input parameters they still may produce different response predictions).

3. Non-traditional numerical approaches such as peridynamics or even simplified methods can produce results that are at least as accurate as the more traditional finite element approach (employed in most commercially available codes such as LS-DYNA).

4. The Riera approach is not well suited for simulating penetration (i.e., punching) type scenarios.

5. Damage is a difficult thing to quantify. Contour plots illustrating crack patterns, damage variables, or tensile principal strains require a substantial level of interpretation to relate them back to outcomes in a real case. Better metrics that are generally agreed upon within the technical community need to be developed for quantifying damage, degradation, and failure.

6. Idealizations of actual configuration details can result in significant differences between predicted results and test observed behavior. As an example, modeling all of the reinforcing steel within the slab in a single plane with junctions between transverse and longitudinal bars occurring at coincident nodes results in a largely symmetric damage cone on the back face of the target for the punching test, whereas the test produced a clear directional bias in the failure cone due to the layering of the two directions of reinforcing steel bars.

7. The number of elements used to represent the target must be sufficient enough to adequately capture its response. This is particularly true in situations involving material degradation and failure. In addition, mesh size is important when trying to correctly capture the concertina type buckling in the missile of the soft missile impacts.

8. Methods for capturing material degradation and failure are not generally agreed upon, yet the choices made with respect to these parameters can have a significant effect on the response predicted by the model. Additional work in this area to develop guidance on the selection of such parameters is needed.

9. Significant model and methodology improvements were realized after the 2010 IRIS workshop as a direct result of the information exchange between IRIS participants that occurred at that event, which underscores the beneficial nature of such exercises.
6. References


IRIS_2012 NUMERICAL SIMULATION REPORT

1. Introduction

Report summarises changes of models from previous version used for analyses done in 2010. Input data for new set of analyses have been extended by more detail knowledge of concrete material (triaxial tests) and by results of impact tests.

In comparison with IRIS_2010 analyses, the following changes in models have been made:

- refinement of mesh of concrete in central part of slab and of a missile body – the size of mesh is one half of the original size
- change of slab supports at both sides into supports at bottom side only acting only in Z direction – comparison of vibration of slab after impact shows that originally applied supports are too stiff and in reality clutch of slab enables small motion of slab in supports
- changes in concrete material model and model parameters – for IRIS_2012 more concrete material parameters is available and material model have been changed. Now, used material model enables also plastic deformations of concrete in compression.

2. Material input data

Steel (both reinforcement and missile)
- used Abaqus classical metal plasticity
- stress-strain dependency taken from test data
- strain-rate effect calculated using the provided stress-strain curves and following formulas:
  \[ \sigma = \sigma \times \alpha \]
  for the yield stress:
  \[ \alpha_{fy} = 0,074 - 0,040 \times \frac{f_{y}}{414} \]
  for the ultimate stress:
  \[ \alpha_{fu} = 0,019 - 0,009 \times \frac{f_{y}}{414} \]

(taken from Dynamic increase factors for steel reinforcing bars; L. Javier Malvar, John E. Crawford; 28. DDESB Seminar, Orlando, FL, August 98)

Concrete
- used Abaqus Concrete Damaged Plasticity model (model enables plastic behaviour of concrete in compression and cracking in tension; doesn’t allow removal of elements based on failure criterion)
- elastic parameters, plastic parameters in compression and tensile strength used according to test data; fracture energy estimated according to strength class of concrete (used fracture energy 150N/m)
- parameters of light-weight concrete (filling of the missile in punching test) calculated according to Eurocode 2, paragraph 11.3. In first version of calculation non-linear behaviour of concrete was expected, in later calculations linear behaviour was used (filling has little effect on missile response).

3. **Basic choices**
   Basic choices are the same for all tests. Calculation code Abaqus in version 6.12 has been used. All calculations were done by Explicit solver (fast dynamic process with contacts of bodies).

4. **Concrete slabs modelling**
   **CNSC VTT punching**
   The slab was made as a 3D quarter model
   Symmetry BC at axis of symmetry, clamping by steel frame modelled by displacement BC in axis Z at bottom side of RC slab.
   Concrete - hexahedral elements of type C3D8R, size 22.5 mm, decreased at size 11.25mm in middle part (aprox. 0.5x0.5m)
   Rebars - line elements of type B31, size 25 mm, embedded in concrete

   **VTT flexural**
   The slab was made as a 3D quarter model
   Symmetry BC at axis of symmetry, clamping by steel frame modelled by displacement BC in axis Z at bottom side of RC slab.
   Concrete - hexahedral elements of type C3D8R, size 27.5 mm, decreased at size 13.75mm in middle part (aprox. 0.5x0.5m)
   Rebars - line elements of type B31, size 25 mm, embedded in concrete
5. Missiles modelling

*CNST VTT punching*
Elements of type S4R/S3R (steel parts) and C3D8R (concrete filling and steel dome), size aprox. 20 mm
Full contact interaction

*VTT flexural*
Elements of type S4R/S3R, size aprox. 12 mm
Full contact interaction
6. **Contact modelling**
   In all tests full general contact has been used.

7. **Calculation results**

   **CNSC VTT punching**
   Results with close agreement between calculation and test haven’t been obtained until elaboration of this report. Strong deformation of slab in area of impact causes distortion of elements and consequential decrease of time step or loss of numerical stability. Ideal way of solving of this problem would be possibility of element removal based on degree of material damage but it is not allowed for type of material used in analyses. Therefore, the problem is solved by applying of smoothed particle hydrodynamics but in this way the only numerical stability problems are well solved, not general response of concrete in compression.

   **VTT flexural**
   Duration of impact 17.5 ms.
   Time step - automatic incrementation - aprox. 5e-8, duration of the calculation 100 ms
   Damage of target – during process of impact creation of cracks in form of cone but without perforation of slab (cone supported by reinforcement).
   Damage of missile – at the beginning of impact crushing of missile body in the front area, in later period of impact increasing ratio of buckling with effect along the length of missile tube.

   In the following figure, the history of displacement in the middle of slab is presented for different analyses:
   - Analysis 1 correspondes to the original IRIS_2010 calculation.
   - In case of Analysis 2 the same model parameters have been used except material model of concrete - Concrete Damaged Plasticity model instead of Cracking Model for Concrete. Due to ability of model to simulate plastic deformation of concrete in compression, there are permanent deformation of slab after impact.
   - In case of Analysis 3, concrete material model parameters have been set by test results, mesh have been refinement and boundary conditions of slab have been changed. These changes conduced to extension of slab vibration period and to increasing of deformation (vibration period is closer to the time of impact). Rapid increase of deformation in the centre of slab is caused by cone cracking of slab in the area of impact. In comparison with test, there is still higher eigen frequency of slab after impact. It could by affected by material model parameters used for damage and stiffness recovery but this option haven’t been check out in the simulations. In this case, the experimental verification of slab eigen frequencies would be useful, both for slab condition before and after the impact test.
1 INTRODUCTION

The following report describes the modeling approach and major results of numerical simulations carried out within the framework of the IRIS-Benchmark project. The simulations were performed by the Chair of Structural Statics and Dynamics, Aachen University on behalf of Swissnuclear. The simulations are carried out for the following three missile impact tests:

- Meppen II/4 – flexural test
- VTT – flexural test
- VTT – punching test

The aim of the IRIS-Benchmark project is a comparison of simulation and test results for predefined solution points and output quantities. Because the solution results depend strongly on the applied nonlinear material models for steel and concrete, the flexural tests are executed with two combinations of material models. The application of two material models will improve the validity of the performed benchmarks with respect to the specific characteristics of the applied material models. The calculations are carried out with LS-DYNA. The present report contains both flexural tests and the VTT-punching test. It has to be pointed out, that the verification of the applied material models was not carried out, because the data of the compression strength were not provided at the time of simulations. For this reason standard values are applied.

2 FLEXURAL TESTS

2.1 General remarks

The simulation of the impact tests Meppen II/4 and VTT flexural is based on three-dimensional finite element models build up with LS-DYNA (Version: ls971sR5.1.1.). The computing time is reduced by simulating just a quarter of the entire model. The gravity acceleration is disregarded because of its low influence and to reduce the computing time. For all simulations a time step of 0.001 ms and a total simulation time of 100 ms are applied.
2.2 Applied material models

The concrete slab of the flexural tests was represented by the material models MAT_159_CSCM_CONCRETE and MAT_084/085_WINFRITH_CONRETE. The applied material models for the slab reinforcement and missile wall are MAT_003_PLASTIC_KINEMATIC and MAT_024 PIECEWISE_LINEAR_PLASTICITY. A detailed description of the models can be found in the theory manual of LS-DYNA. The simulations are carried out with different material combinations and their results are discussed and compared.

2.3 Flexural test: Meppen II/4

2.3.1 Simulation model

2.3.1.1 Concrete slab

The concrete slab is discretized with a total number of 54600 solid elements with 8 nodes each. The model and a detail of the edge are shown Figure 1. Due to the simplification to a quarter model, the dimensions are 3.25 m in x-direction, 3.0 m in y-direction and 0.7 m in z-direction.

![Concrete slab model with detail A](image)

**Figure 1:** Concrete slab model with detail A

2.3.1.2 Reinforcement

The bending and shear reinforcement bars are modeled by beams according to the “Standard-Hughes-Liu” formulation. Bending and shear reinforcement are connected by coincident nodes and the dimensions correspond to the centerlines of the reinforcement bars (Figure 2).
2.3.1.3 Missile

The missile is modeled with 7680 shell elements. The thickness varies between 5 to 10 mm. The geometry and the finite element shell model are shown in Figure 3. Each color of the model represents a shell thickness.

2.3.1.4 Boundary conditions

The simplification to a quarter model requires the definition of additional symmetry conditions for the slab and the missile. The necessary boundary conditions are shown in Figure 4.
2.3.1.5 Contact formulation

The contact within the simulation is described by two contact formulations. The self-contact of the missile wall is represented by the contact type CONTACT_AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE. The contact between the concrete slab and the missile wall is modeled by the contact type CONTACT_AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE, whereas the slab is the target-body and the missile is the slave-body. The reinforcement bars and the concrete slab are directly connected by using the coincident nodes.

2.3.1.6 Summary of the Meppen simulation input data

The simulation input data are summarized in Table 1, Table 2 and Table 3.

Table 1: Reinforcement

<table>
<thead>
<tr>
<th>Element type</th>
<th>Beam elements BEAM</th>
</tr>
</thead>
<tbody>
<tr>
<td>Element formulation</td>
<td>1 ELFORM 1, MAT_003</td>
</tr>
<tr>
<td>Material model</td>
<td>003, 024 MAT_003, MAT_024</td>
</tr>
</tbody>
</table>

Material model MAT_003

<table>
<thead>
<tr>
<th>Mass density ρ</th>
<th>7850 kg/m³ RO</th>
</tr>
</thead>
<tbody>
<tr>
<td>Young’s modulus E</td>
<td>210000 MPa E</td>
</tr>
<tr>
<td>Poisson’s ratio ν</td>
<td>0.2 PR</td>
</tr>
<tr>
<td>Yield stress f_y</td>
<td>500 SIGY</td>
</tr>
<tr>
<td>Tangent modulus E_tan</td>
<td>3600 ETAN</td>
</tr>
<tr>
<td>Hardening parameter β</td>
<td>1.0 BETA</td>
</tr>
<tr>
<td>Cowper Symonds Parameter C</td>
<td>40 SRC</td>
</tr>
<tr>
<td>Cowper Symonds Parameter P</td>
<td>5 SRP</td>
</tr>
</tbody>
</table>

Stress-strain curve

Material model MAT_024

<table>
<thead>
<tr>
<th>Mass density ρ</th>
<th>7850 kg/m³ RO</th>
</tr>
</thead>
<tbody>
<tr>
<td>Young’s modulus E</td>
<td>210000 MPa E</td>
</tr>
<tr>
<td>Poisson’s ratio ν</td>
<td>0.2 PR</td>
</tr>
<tr>
<td>Yield stress f_y</td>
<td>0 SIGY</td>
</tr>
<tr>
<td>Tangent modulus E_tan</td>
<td>0 ETAN</td>
</tr>
<tr>
<td>Cowper Symonds Parameter C</td>
<td>40 s^{-1} C</td>
</tr>
<tr>
<td>Cowper Symonds Parameter P</td>
<td>5 P</td>
</tr>
<tr>
<td>Strain rate effects with Cowper-Symonds</td>
<td>-1 VP</td>
</tr>
</tbody>
</table>

Stress-Strain Curve (Stress [MPa]/Strain[%])
### Table 2: Concrete slab

<table>
<thead>
<tr>
<th>Value/ Notation</th>
<th>Notation in LS-DYNA</th>
</tr>
</thead>
<tbody>
<tr>
<td>Concrete slab</td>
<td></td>
</tr>
<tr>
<td>Element type</td>
<td>Solid elements</td>
</tr>
<tr>
<td>Element formulation</td>
<td>1</td>
</tr>
<tr>
<td>Hourglass-Type</td>
<td>10</td>
</tr>
<tr>
<td>Material model</td>
<td>084/085, 159</td>
</tr>
<tr>
<td>Material Model MAT_084/085</td>
<td></td>
</tr>
<tr>
<td>Mass density $\rho$</td>
<td>2600 kg/m³</td>
</tr>
<tr>
<td>Initial tangent modulus of concrete $E_t$</td>
<td>29100 MPa</td>
</tr>
<tr>
<td>Poisson’s ratio $\nu$</td>
<td>0.2</td>
</tr>
<tr>
<td>Uniaxial compressive strength $f_c$</td>
<td>37.2 MPa</td>
</tr>
<tr>
<td>Uniaxial tensile strength $f_t$</td>
<td>4.8 MPa</td>
</tr>
<tr>
<td>Fracture energy</td>
<td>85.5 J/m</td>
</tr>
<tr>
<td>Aggregate size (radius)</td>
<td>0.04 m</td>
</tr>
<tr>
<td>Material Model MAT_159</td>
<td></td>
</tr>
<tr>
<td>Mass density $\rho$</td>
<td>2600 kg/m³</td>
</tr>
<tr>
<td>Compression strength $f_c$</td>
<td>37.2 MPa</td>
</tr>
</tbody>
</table>

### Table 3: Summary of the missile

<table>
<thead>
<tr>
<th>Value/ Notation</th>
<th>Notation in LS-DYNA</th>
</tr>
</thead>
<tbody>
<tr>
<td>Missile</td>
<td></td>
</tr>
<tr>
<td>Element type</td>
<td>Shell elements</td>
</tr>
<tr>
<td>Element formulation</td>
<td>16</td>
</tr>
<tr>
<td>Hourglass-Type</td>
<td>8</td>
</tr>
<tr>
<td>Velocity</td>
<td>247.7 m/s</td>
</tr>
<tr>
<td>Material model</td>
<td>003, 024</td>
</tr>
<tr>
<td>Material model MAT_003</td>
<td></td>
</tr>
<tr>
<td>Mass density $\rho$</td>
<td>10100 kg/m³</td>
</tr>
<tr>
<td>Young’s modulus $E$</td>
<td>200000 MPa</td>
</tr>
<tr>
<td>Poisson’s Ratio $\nu$</td>
<td>0.3</td>
</tr>
<tr>
<td>Yield stress $f_y$</td>
<td>280</td>
</tr>
<tr>
<td>Tangent modulus $E_{tan}$</td>
<td>800</td>
</tr>
<tr>
<td>Hardening parameter $\beta$</td>
<td>1.0</td>
</tr>
<tr>
<td>Cowper Symonds Parameter C</td>
<td>40 s⁻¹</td>
</tr>
<tr>
<td>Cowper Symonds Parameter P</td>
<td>5</td>
</tr>
<tr>
<td>Stress-strain curve</td>
<td>[Graph]</td>
</tr>
<tr>
<td>Material model MAT_024</td>
<td></td>
</tr>
<tr>
<td>Mass Density $\rho$</td>
<td>10100 kg/m³</td>
</tr>
<tr>
<td>Young’s modulus $E$</td>
<td>200000 MPa</td>
</tr>
<tr>
<td>Poisson’s ratio $\nu$</td>
<td>0.3</td>
</tr>
<tr>
<td>Yield stress $f_y$</td>
<td>0</td>
</tr>
<tr>
<td>Tangent modulus $E_{tan}$</td>
<td>0</td>
</tr>
<tr>
<td>Cowper Symonds Parameter C</td>
<td>40 s⁻¹</td>
</tr>
<tr>
<td>Cowper Symonds Parameter P</td>
<td>5</td>
</tr>
<tr>
<td>Strain rate effects with Cowper-Symonds</td>
<td>-1</td>
</tr>
<tr>
<td>Stress-Strain Curve (Stress [MPa]/Strain[%])</td>
<td>[Graph]</td>
</tr>
</tbody>
</table>
2.3.2 Calculation time

The applied combinations of material models and the corresponding calculation times are summarized in Table 4. The comparison of the calculation times show clearly, that the computational effort is much higher if the more complex concrete material model 085/084 is used.

Table 4: Applied material combinations and calculation times

<table>
<thead>
<tr>
<th>Material Model</th>
<th>Simulation Model</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Target</td>
</tr>
<tr>
<td>159</td>
<td>003</td>
</tr>
<tr>
<td>159</td>
<td>024</td>
</tr>
<tr>
<td>084/085</td>
<td>003</td>
</tr>
<tr>
<td>084/085</td>
<td>024</td>
</tr>
</tbody>
</table>

*(2 processors, Intel(R) Xeon(R) E5620, 2.40 GHz, 64-Bit Windows Server 2008)*

2.3.3 Calculation results

Within this chapter selected calculation results are presented and discussed. A more detailed description of the results can be found in the Excel-Sheets, filled out by all groups joining the benchmark.

2.3.3.1 Displacements of the concrete slab

The displacements between impact test and simulation are compared for the control points W6 and W8. The location of the control points are shown in Figure 5.

Figure 5: Control points W6 and W8
Figure 6 and Figure 7 show the displacement curves for the applied material combinations in comparison to the impact test results. In general, a satisfactory agreement between the simulated and measured displacement curves is obtained. However, the material model MAT 159 shows a better agreement for calculations time greater than ~30ms.

**Figure 6:** Displacement curve: Control point W6

**Figure 7:** Displacement curve: Control point W8
2.3.3.2 Crack pattern of the concrete slab

Because the material model MAT 159 is not able to investigate the crack pattern distribution, the crack pattern of the impact test is compared with the simulation results using the material model MAT 084/085. Figure 8 shows the crack pattern of the overall quarter model. Figure 9 shows the crack pattern distribution for the rear side of the concrete plate and Figure 10 shows the crack pattern over the slab thickness for sections in x- and y-direction. The resulting crack pattern plots show a satisfactory agreement between simulation and test.

Figure 8: Crack Pattern: Concrete slab, 100 ms (Simulation with MAT 084/085 + 024)

Figure 9: Crack Pattern of the concrete slab rear side: Impact test (left) and simulation (right) after 100 ms (Simulation with MAT 084/085 + 024). The red lines corresponds to the experimental crack pattern
2.3.3.3 Displacements of the missile

Figure 11 shows the displacements of the missile. The duration of the impact in the simulations varies between 30 and 33 ms for the investigated material combination and corresponds to the measured duration of 26 ms. The total simulated deformation varies between 3.8 and 4.1 m and agrees to the measured of about 4 m. Figure 12 shows the highly deformed missile after 100 ms.
2.4 Flexural test: VTT

2.4.1 Simulation model

2.4.1.1 Concrete slab

The concrete slab is discretized with a total number of 80,864 solid elements with 8 nodes each. The model and a detail of the edge are shown in Figure 13. Due to the quarter model, the dimensions are 1.041 m in x-direction, 1.041 m in y-direction and 0.15 m in z-direction.

![Concrete slab model with detail A and dimensions [cm]](image)

2.4.1.2 Reinforcement

The bending and shear reinforcement bars are modeled by beams according to the “Standard-Hughes-Liu” formulation. Bending and shear reinforcement are connected by coincident nodes and the dimensions correspond to the centerlines of the reinforcement bars (Figure 14).

![Reinforcement model with detail B and dimensions [cm]](image)
2.4.1.3 U-Profile

The surrounding U-Profile is modeled with shell elements. The model of the U-Profile is shown in Figure 15.

![Figure 15: Model of the U-Profile with shell elements with Detail C and dimensions [cm]]

2.4.1.4 Missile

The missile is modeled with shell elements. The thickness varies between 2 to 12 mm. The geometry and the finite element shell model are shown in Figure 16. Each color of the model represents a shell thickness.

![Figure 16: Missile geometry and finite element model with shell elements]

The missile consists of three different material parts (steel and carbon). Therefore three materials are applied within the simulation model.

![Figure 17: Parts of the missile]
2.4.1.5 Boundary conditions

The boundary conditions of the missile are applied in the same way as already described for the Meppen simulation (see chapter 2.3.1.4). Figure 18 shows the boundary conditions of the target and the U-Profile for the quarter model of the VTT-flexural test.

![Figure 18: Boundaries of the concrete slab (Target) and the U-Profile](image)

2.4.1.6 Contact formulation

The contact within the simulation is described by two contact formulations. The self-contact of the missile wall is represented by the contact type CONTACT_AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE. The contact between the concrete slab and the missile wall is modeled by the contact type CONTACT_AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE, whereas the slab is the target-body and the missile is the slave-body. The reinforcement bars and the concrete slab are directly connected by using the coincident nodes. The U-Profile is connected to the concrete slab with the contact CONTACT_TIED NODES_TO_SURFACE_OFFSET.

2.4.1.7 Summary of the VTT simulation input data

The simulation input data are summarized in Table 5 to Table 9.
Table 5: Reinforcement

<table>
<thead>
<tr>
<th>Reinforcement</th>
<th>Value/ Notation</th>
<th>Notation in LS-DYNA</th>
</tr>
</thead>
<tbody>
<tr>
<td>Element type</td>
<td>Beam elements</td>
<td>BEAM</td>
</tr>
<tr>
<td>Element formulation</td>
<td>1</td>
<td>ELMFORM_1</td>
</tr>
<tr>
<td>Material model</td>
<td>003, 024</td>
<td>MAT_003, MAT_024</td>
</tr>
<tr>
<td><strong>Material model MAT_003</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Mass density $\rho$</td>
<td>7850 kg/m³</td>
<td>RO</td>
</tr>
<tr>
<td>Young's modulus $E$</td>
<td>200000 MPa</td>
<td>E</td>
</tr>
<tr>
<td>Poisson's ratio $\nu$</td>
<td>0.3</td>
<td>PR</td>
</tr>
<tr>
<td>Yield stress $f_y$</td>
<td>600</td>
<td>SIGY</td>
</tr>
<tr>
<td>Tangent modulus $E_{tan}$</td>
<td>2900</td>
<td>ETAN</td>
</tr>
<tr>
<td>Hardening parameter $\beta$</td>
<td>1.0</td>
<td>BETA</td>
</tr>
<tr>
<td>Cowper Symonds Parameter $C$</td>
<td>40 s$^{-1}$</td>
<td>SRC</td>
</tr>
<tr>
<td>Cowper Symonds Parameter $P$</td>
<td>5</td>
<td>SRP</td>
</tr>
<tr>
<td>Stress-strain curve</td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>Material model MAT_024</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Mass density $\rho$</td>
<td>7850 kg/m³</td>
<td>RO</td>
</tr>
<tr>
<td>Young's modulus $E$</td>
<td>200000 MPa</td>
<td>E</td>
</tr>
<tr>
<td>Poisson's ratio $\nu$</td>
<td>0.3</td>
<td>PR</td>
</tr>
<tr>
<td>Yield stress $f_y$</td>
<td>0</td>
<td>SIGY</td>
</tr>
<tr>
<td>Tangent modulus $E_{tan}$</td>
<td>0</td>
<td>ETAN</td>
</tr>
<tr>
<td>Cowper Symonds Parameter $C$</td>
<td>40 s$^{-1}$</td>
<td>C</td>
</tr>
<tr>
<td>Cowper Symonds Parameter $P$</td>
<td>5</td>
<td>P</td>
</tr>
<tr>
<td>Strain rate effects with Cowper-Symonds</td>
<td>-1</td>
<td>VP</td>
</tr>
<tr>
<td>Stress-Strain Curve (Stress [MPa]/Strain[%])</td>
<td></td>
<td>EPS, ES</td>
</tr>
</tbody>
</table>
**Table 6: Concrete slab**

<table>
<thead>
<tr>
<th>Target</th>
<th>Value/ Notation</th>
<th>Notation in LS-DYNA</th>
</tr>
</thead>
<tbody>
<tr>
<td>Element type</td>
<td>Solid elements</td>
<td>SOLID</td>
</tr>
<tr>
<td>Element formulation</td>
<td>1</td>
<td>ELFORM 1</td>
</tr>
<tr>
<td>Hourglass-Type</td>
<td>10</td>
<td>IHQ</td>
</tr>
<tr>
<td>Material model</td>
<td>084/085, 159</td>
<td>MAT_084/085, MAT_159</td>
</tr>
</tbody>
</table>

**Material model MAT_084/085**

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
<th>Notation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mass density $\rho$</td>
<td>2320 kg/m³</td>
<td>RO</td>
</tr>
<tr>
<td>Initial tangent modulus of concrete $E_t$</td>
<td>27986 MPa</td>
<td>TM</td>
</tr>
<tr>
<td>Poisson's ratio $\nu$</td>
<td>0.2</td>
<td>PR</td>
</tr>
<tr>
<td>Uniaxial compressive strength $f_c$</td>
<td>64.0 MPa</td>
<td>UCS</td>
</tr>
<tr>
<td>Uniaxial tensile strength $f_t$</td>
<td>3.73 MPa</td>
<td>UTS</td>
</tr>
<tr>
<td>Fracture energy</td>
<td>85.5 J/m</td>
<td>FE</td>
</tr>
<tr>
<td>Aggregate size (radius)</td>
<td>0.004 m</td>
<td>ASIZE</td>
</tr>
</tbody>
</table>

**Material model MAT_159**

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
<th>Notation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mass density $\rho$</td>
<td>2320 kg/m³</td>
<td>RO</td>
</tr>
<tr>
<td>Compression strength $f_c$</td>
<td>64.0 MPa</td>
<td>FPC</td>
</tr>
<tr>
<td>Aggregate size</td>
<td>0.008 m</td>
<td>DAGG</td>
</tr>
</tbody>
</table>
Table 7: U-Profile

<table>
<thead>
<tr>
<th>U-Profile</th>
<th>Value/ Notation</th>
<th>Notation in LS-DYNA</th>
</tr>
</thead>
<tbody>
<tr>
<td>Element type</td>
<td>Shell elements</td>
<td>SHELL</td>
</tr>
<tr>
<td>Element formulation</td>
<td>16</td>
<td>ELMFORM 16</td>
</tr>
<tr>
<td>Hourglass-Type</td>
<td>8</td>
<td>IHQ</td>
</tr>
<tr>
<td>Material model</td>
<td>003, 024</td>
<td>MAT_003, MAT_024</td>
</tr>
</tbody>
</table>

**Material model MAT_003**

- Mass density $\rho$ 7850 kg/m³  
- Young’s modulus $E$ 200000 MPa  
- Poisson’s ratio $\nu$ 0.3  
- Yield stress $f_y$ 355  
- Tangent modulus $E_{tan}$ 8400  
- Hardening parameter $\beta$ 1.0  
- Cowper Symonds Parameter C 40 s⁻¹  
- Cowper Symonds Parameter P 5  

**Material model MAT_024**

- Mass density $\rho$ 7850 kg/m³  
- Young’s modulus $E$ 200000 MPa  
- Poisson’s ratio $\nu$ 0.3  
- Yield stress $f_y$ 355  
- Tangent modulus $E_{tan}$ 8400  
- Cowper Symonds Parameter C 40 s⁻¹  
- Cowper Symonds Parameter P 5  
- Strain rate effects with Cowper-Symonds -1  

Stress-Strain Curve (Stress [MPa]/Strain[%])  

![Stress-Strain Curve](image)
<table>
<thead>
<tr>
<th>Value/ Notation</th>
<th>Notation in LS-DYNA</th>
</tr>
</thead>
<tbody>
<tr>
<td>Element type</td>
<td>Shell elements</td>
</tr>
<tr>
<td>Element formulation</td>
<td>16</td>
</tr>
<tr>
<td>Hourglass-Type</td>
<td>8</td>
</tr>
<tr>
<td>Velocity</td>
<td>110.0 m/s</td>
</tr>
<tr>
<td>Material model</td>
<td>003, 024</td>
</tr>
</tbody>
</table>

**Part 1 (Steel EN 1.4432)**

<table>
<thead>
<tr>
<th>Material model MAT_003</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mass density ρ</td>
</tr>
<tr>
<td>Young's modulus E</td>
</tr>
<tr>
<td>Poisson's ratio ν</td>
</tr>
<tr>
<td>Yield stress fy</td>
</tr>
<tr>
<td>Tangent modulus E_tan</td>
</tr>
<tr>
<td>Hardening parameter β</td>
</tr>
<tr>
<td>Cowper Symonds Parameter C</td>
</tr>
<tr>
<td>Cowper Symonds Parameter P</td>
</tr>
</tbody>
</table>

**Stress-strain curve**

<table>
<thead>
<tr>
<th>Material model MAT_024</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mass density ρ</td>
</tr>
<tr>
<td>Young’s modulus E</td>
</tr>
<tr>
<td>Poisson’s ratio ν</td>
</tr>
<tr>
<td>Yield stress fy</td>
</tr>
<tr>
<td>Tangent modulus E_tan</td>
</tr>
<tr>
<td>Cowper Symonds Parameter C</td>
</tr>
<tr>
<td>Cowper Symonds Parameter P</td>
</tr>
<tr>
<td>Strain rate effects with Cowper-Symonds</td>
</tr>
</tbody>
</table>

**Stress-Strain Curve (Stress [MPa]/Strain[%])**
Table 9: Summary of the missile Part 2 & 3

<table>
<thead>
<tr>
<th>Value/ Notation</th>
<th>Notation in LS-DYNA</th>
</tr>
</thead>
<tbody>
<tr>
<td>Missile</td>
<td></td>
</tr>
<tr>
<td>Element type</td>
<td>Shell elements</td>
</tr>
<tr>
<td></td>
<td>SHELL</td>
</tr>
<tr>
<td>Element formulation</td>
<td>16</td>
</tr>
<tr>
<td>Hourglass-Type</td>
<td>8</td>
</tr>
<tr>
<td>Velocity</td>
<td>110,0 m/s</td>
</tr>
<tr>
<td>Material model</td>
<td>003, 024</td>
</tr>
<tr>
<td></td>
<td>MAT_003, MAT_024</td>
</tr>
<tr>
<td>Part 2 &amp; 3 (Carbon)</td>
<td></td>
</tr>
<tr>
<td>Material model</td>
<td>MAT_003</td>
</tr>
<tr>
<td>Mass Density $\rho$</td>
<td>B2: 6752.5 kg/m³</td>
</tr>
<tr>
<td></td>
<td>B3: 8013.58 kg/m³</td>
</tr>
<tr>
<td>Young's modulus $E$</td>
<td>205000 MPa</td>
</tr>
<tr>
<td>Poisson's ratio $\nu$</td>
<td>0.3</td>
</tr>
<tr>
<td>Yield stress $f_y$</td>
<td>355</td>
</tr>
<tr>
<td>Tangent modulus $E_{tan}$</td>
<td>5800</td>
</tr>
<tr>
<td>Hardening parameter $\beta$</td>
<td>1.0</td>
</tr>
<tr>
<td>Cowper Symonds Parameter C</td>
<td>100 s⁻¹</td>
</tr>
<tr>
<td>Cowper Symonds Parameter P</td>
<td>5</td>
</tr>
<tr>
<td>Stress-strain curve</td>
<td></td>
</tr>
</tbody>
</table>

Material model MAT_024

<table>
<thead>
<tr>
<th>Value/ Notation</th>
<th>Notation in LS-DYNA</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mass density $\rho$</td>
<td>B2: 6752.5 kg/m³</td>
</tr>
<tr>
<td></td>
<td>B3: 8013.58 kg/m³</td>
</tr>
<tr>
<td>Young's modulus $E$</td>
<td>14250 MPa</td>
</tr>
<tr>
<td>Poisson's ratio $\nu$</td>
<td>0.3</td>
</tr>
<tr>
<td>Yield stress $f_y$</td>
<td>355</td>
</tr>
<tr>
<td>Tangent modulus $E_{tan}$</td>
<td>5800</td>
</tr>
<tr>
<td>Cowper Symonds Parameter C</td>
<td>100 s⁻¹</td>
</tr>
<tr>
<td>Cowper Symonds Parameter P</td>
<td>10</td>
</tr>
<tr>
<td>Strain rate effects with Cowper-Symonds</td>
<td>-1</td>
</tr>
<tr>
<td>Stress-Strain Curve (Stress [MPa]/Strain[%])</td>
<td>EPS, ES</td>
</tr>
</tbody>
</table>
2.4.2 Calculation time

The applied combinations of material models and the corresponding calculation times are summarized in Table 10. The comparison of the calculation times show clearly, that the computational effort is much higher if the more complex concrete material model 085/084 is used.

Table 10: Applied material combinations and calculation times

<table>
<thead>
<tr>
<th>Material Model</th>
<th>Simulation Model</th>
<th>Calculation Time</th>
</tr>
</thead>
<tbody>
<tr>
<td>159/003</td>
<td>Target</td>
<td>2 Hrs. 12 Min.</td>
</tr>
<tr>
<td>159/024</td>
<td>Target</td>
<td>2 Hrs. 22 Min.</td>
</tr>
<tr>
<td>084/085/003</td>
<td>Target</td>
<td>7 Hrs. 8 Min.</td>
</tr>
<tr>
<td>084/085/024</td>
<td>Target</td>
<td>8 Hrs. 19 Min.</td>
</tr>
</tbody>
</table>

(2 processors, Intel(R) Xeon(R) E5620, 2.40 GHz, 64-Bit Windows Server 2008)

2.4.3 Calculation results

Within this chapter selected calculation results are presented and discussed. A more detailed description of the results can be found in the Excel-Sheets, filled out by all groups joining the benchmark.

2.4.3.1 Displacements of the concrete slab

The displacements between impact test and simulation are compared for the applied material combinations for the control points 1 and 2. The corresponding displacement curves are shown in Figure 19 and Figure 20.

![Figure 19: Displacement at control point 1](image-url)
The displacement curves for the material combinations MAT 084/085 + MAT 024 and MAT 084/085 + MAT 003 show a good agreement with the test results, but the Model MAT 159 is not able to represent the dynamic behavior after the impact in the time range greater than 25 ms. For this reason most the following results are just presented for the material combination MAT 084/085 + MAT 024.

2.4.3.2 Crack pattern of the concrete slab

Figure 21 shows the crack pattern of the overall quarter model. Figure 22 and Figure 23 show the crack pattern distribution for the rear and front side of the concrete slab. Furthermore Figure 24 shows the crack pattern over the slab thickness for sections in x- and y-direction. The simulated crack pattern distributions show a satisfactory agreement between simulation and test.
Figure 22: Crack Pattern of the front side: Impact test (left) and simulation (right) after 100 ms (MAT 084/085 + 024)

Figure 23: Crack Pattern of the rear side of the concrete slab: Impact test (left) and simulation (right) after 100 ms (Simulation with MAT 084/085 + 024)
2.4.3.3 Displacement of the missile

The simulated impact duration varies between 18 and 22 ms for the investigated material combination and corresponds to the measured impact duration of 18 ms. The total simulated deformation varies between 1.5 and 1.4 m depending on the applied material model and shows a satisfactory agreement with the maximum measured deformation of 1.15 m. Figure 25 shows the deformed missile after 100 ms.

Figure 25: Deformed missile after 100 ms
3  PUNCHING TEST: VTT

3.1  General remarks

The simulation of the VTT punching test is based on a three dimensional finite element model build up with LS-DYNA (Version: ls971sR5.1.1.). The computing time is reduced by simulating just a quarter of the entire model. The gravity acceleration is disregarded because of its low influence and to reduce the computing time. For the simulation a time step of 0.00005 ms and a total simulation time of 100 ms are applied.

3.2  Applied material models

The concrete slab and the light-weight concrete filling of the missile is represented by the material model MAT_159_CSCM_CONCRETE. The applied material model for the slab reinforcement and missile steel cover is MAT_024 PIECEWISE LINEAR PLASTICITY. A detailed description of the models can be found in the theory manual of LS-DYNA.

3.3  Simulation model

3.3.1  Concrete slab

The concrete slab is discretized with a total number of 457504 solid elements with 8 nodes each. The model and a detail of the edge are shown in Figure 26. Due to the quarter model, the dimensions are 1.041 m in x-direction, 1.041 m in y-direction and 0.25 m in z-direction.

![Concrete slab model with detail A and dimensions [mm]](image)

3.3.2  Reinforcement

The bending and shear reinforcement bars are modeled by beams according to the “Standard-Hughes-Liu” formulation. Bending and shear reinforcement are connected by coincident nodes and the dimensions correspond to the centerlines of the reinforcement bars (Figure 27).
3.3.3 U-Profile

The U-Profile of the experimental test set-up is neglected and not considered in the calculation model.

3.3.4 Missile

The modeling of the missile was carried out with volume elements. The three dimensional model of the missile is shown in Figure 28.

3.3.5 Boundary conditions

The boundary conditions of the concrete slab and the missile are applied in the same way as already described for the Meppen and VTT simulation. Detailed information can be found in chapter 2.3.1.4 and 2.4.1.5. Figure 29 shows the boundary conditions for the quarter model of the punching test.
### 3.3.6 Contact formulation

The contacts within the simulation model are described by two contact formulations. The contact between the concrete slab and the missile is modeled by the contact type CONTACT_ERODING_SURFACE_TO_SURFACE, whereas the slab is the target-body and the missile is the slave-body. The contact between the reinforcement bars and the missile is modeled by the contact type CONTACT_ERODING_NODES_TO_SURFACE, whereas the missile is the target-body and the reinforcement is the slave-body. The reinforcement bars and the concrete slab are directly connected by using the coincident nodes.

### 3.3.7 Erosion Criteria

For simulating the perforation the LS-DYNA card MAT_ADD_EROSION is used, whereas the shear strain at failure is defined by EPSSH = 0.6. Thereby elements are eroded, if the shear strain exceeds this value.

### 3.3.8 Summary of the VTT simulation input data

The simulation input data are summarized in Table 11 to Table 13.
Table 11: Reinforcement

<table>
<thead>
<tr>
<th>Reinforcement</th>
<th>Value/ Notation</th>
<th>Notation in LS-DYNA</th>
</tr>
</thead>
<tbody>
<tr>
<td>Element type</td>
<td>Beam elements</td>
<td>BEAM</td>
</tr>
<tr>
<td>Element formulation</td>
<td>1</td>
<td>ELFORM 1</td>
</tr>
<tr>
<td>Material model</td>
<td>024</td>
<td>MAT_003, MAT_024</td>
</tr>
</tbody>
</table>

Material model MAT_024

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value/ Notation</th>
<th>Notation in LS-DYNA</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mass density ( \rho )</td>
<td>7850 kg/m(^3)</td>
<td>RO</td>
</tr>
<tr>
<td>Young's modulus ( E )</td>
<td>210000 MPa</td>
<td>E</td>
</tr>
<tr>
<td>Poisson's ratio ( \nu )</td>
<td>0.3</td>
<td>PR</td>
</tr>
<tr>
<td>Yield stress ( f_y )</td>
<td>0</td>
<td>SIGY</td>
</tr>
<tr>
<td>Tangent modulus ( E_{tan} )</td>
<td>0</td>
<td>ETAN</td>
</tr>
<tr>
<td>Failure [%]</td>
<td>10</td>
<td>FAIL</td>
</tr>
<tr>
<td>Cowper Symonds Parameter ( C )</td>
<td>40 s(^{-1})</td>
<td>C</td>
</tr>
<tr>
<td>Cowper Symonds Parameter ( P )</td>
<td>5</td>
<td>P</td>
</tr>
<tr>
<td>Strain rate effects with Cowper-Symonds</td>
<td>-1</td>
<td>VP</td>
</tr>
<tr>
<td>Stress-Strain Curve (Stress [MPa]/Strain [%])</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table 12: Concrete slab

<table>
<thead>
<tr>
<th>Target</th>
<th>Value/ Notation</th>
<th>Notation in LS-DYNA</th>
</tr>
</thead>
<tbody>
<tr>
<td>Element type</td>
<td>Solid elements</td>
<td>SOLID</td>
</tr>
<tr>
<td>Element formulation</td>
<td>1</td>
<td>ELFORM 1</td>
</tr>
<tr>
<td>Hourglass-Type</td>
<td>10</td>
<td>IHQ</td>
</tr>
<tr>
<td>Material model</td>
<td>159</td>
<td>MAT_159</td>
</tr>
</tbody>
</table>

Material model MAT_159

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value/ Notation</th>
<th>Notation in LS-DYNA</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mass density ( \rho )</td>
<td>2320 kg/m(^3)</td>
<td>RO</td>
</tr>
<tr>
<td>Compression strength ( f_c )</td>
<td>60.0 MPa</td>
<td>FPC</td>
</tr>
<tr>
<td>Aggregate size</td>
<td>0,008 m</td>
<td>DAGG</td>
</tr>
</tbody>
</table>
**Table 13: Summary of the missile**

<table>
<thead>
<tr>
<th>Missile</th>
<th>Value/ Notation</th>
<th>Notation in LS-DYNA</th>
</tr>
</thead>
<tbody>
<tr>
<td>Element type</td>
<td>Solid elements</td>
<td>SHELL</td>
</tr>
<tr>
<td>Element formulation</td>
<td>1</td>
<td>ELFORM 16</td>
</tr>
<tr>
<td>Velocity</td>
<td>135.0 m/s</td>
<td>INITIAL_VELOCITY</td>
</tr>
<tr>
<td>Material model</td>
<td>024, 159</td>
<td>MAT_024, MAT_159</td>
</tr>
</tbody>
</table>

**Light-weight concrete**

<table>
<thead>
<tr>
<th>Material model MAT_159</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Mass density $\rho$</td>
<td>1567.12 kg/m³</td>
</tr>
<tr>
<td>Compression strength $f_c$</td>
<td>25.0 MPa</td>
</tr>
<tr>
<td>Aggregate size</td>
<td>0.008 m</td>
</tr>
</tbody>
</table>

**Steel cover**

<table>
<thead>
<tr>
<th>Material model MAT_024</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Mass density $\rho$</td>
<td>7850 kg/m³</td>
</tr>
<tr>
<td>Young's modulus $E$</td>
<td>200000 MPa</td>
</tr>
<tr>
<td>Poisson's ratio $\nu$</td>
<td>0.3</td>
</tr>
<tr>
<td>Yield stress $f_y$</td>
<td>355</td>
</tr>
<tr>
<td>Tangent modulus $E_{tan}$</td>
<td>8400</td>
</tr>
<tr>
<td>Cowper Symonds Parameter $C$</td>
<td>40 s$^{-1}$</td>
</tr>
<tr>
<td>Cowper Symonds Parameter $P$</td>
<td>5</td>
</tr>
<tr>
<td>Strain rate effects with Cowper-Symonds</td>
<td>-1</td>
</tr>
</tbody>
</table>

Stress-Strain Curve (Stress [MPa]/Strain [%])

![Stress-Strain Curve](image)
3.3.9 Calculation time

The calculation time for the applied material combination is given in Table 14.

Table 14: Applied material combination and calculation time

<table>
<thead>
<tr>
<th>Material Model</th>
<th>Target 159</th>
<th>Reinforcement 024</th>
<th>Missile 024, 159</th>
<th>Calculation Time 122 Hrs. 16 Min.</th>
</tr>
</thead>
</table>

(2 processors, Intel(R) Xeon(R) E5620, 2.40 GHz, 64-Bit Windows Server 2008)

3.4 Calculation results

Within this chapter selected calculation results are presented and discussed. A more detailed description of the results can be found in the Excel-Sheets, filled out by all groups joining the benchmark.

3.4.1 Displacements of the concrete slab

The displacements between test and simulation are compared for the control points 4 and 5. The corresponding displacement curves for these control points are shown in Figure 30 and Figure 31.

![Displacement at control point 4](image_url)
The displacement curves show a good agreement for the first peak value, but the dynamic behavior after the impact in the time range greater than 15 ms shows an oscillation comparable to the flexural test results, if the concrete model MAT 159 is applied. However, the MAT 159 model was applied for the punching test for minimizing the calculation time.

3.4.2 Crack pattern and failure mechanism of the concrete slab

Figure 32 shows the crack pattern of the overall quarter model after 100 ms on the rear side of the slab. The simulated crack pattern distribution shows a satisfactory agreement to the experimental results.
Figure 33 shows the missile impact with the damage propagation for the calculation time from 0.8 to 51 ms. The simulation results of the dominant failure mechanism of the concrete slab and the reinforcement look reasonable with a satisfactory agreement to the experimental test results.

Figure 33: Failure mechanism of the concrete slab between 0.8 and 51 ms

Figure 34, left shows the simulated crack pattern and concrete spalling of the rear slab side after 100 ms. The comparison with the experimental tests (Figure 34, right) shows a good agreement of the damage area dimension.
3.4.3 Displacement of the missile

The rigid missile shows no significant deformations during the impact and the perforation. Figure 35 shows the deformed missile with the corresponding plastic strains after 100 ms.

Figure 34: Damage area in terms of plastic strain of the rear side: Simulation (left) and test (right) after 100 ms

Figure 35: Plastic Strains of the missile after 100 ms
4 CONCLUSION

The simulations results lead to the following conclusions:

- The steel material model MAT 024 yields to better results than MAT 003.

- The concrete model MAT 159 is numerical much more stable than the material model MAT 084/085, which can be used for the simulation of the crack pattern. However, the calculation of the crack pattern with MAT 084/085 is quite time consuming.

- The usage of the concrete model MAT 159 leads to an oscillation of the simulation results after the first impact peak. The oscillation can be observed in the simulation results of the VTT flexural and punching test. The reason for the oscillation cannot be explained and needs further investigations.

- The sensitivity of the results with respect to the erosion criteria should be investigated with systematic simulations.
5 REFERENCES AND REPORTS


Schneider, K.-J. (2002). Bautabellen für Ingenieure. Werner Verlag.

Schwer, L. (2010). An Introduction to the Winfrith Concrete Model.


A III.19 Team #25 UJF

XUAN-DUNG VU*, YANN MALECOT*, LAURENT DAUDEVILLE*, MATTHIEU BRIFFAUT*, BERTRAND CIREE†

*University Joseph Fourier – Grenoble 1, Grenoble INP, CNRS UMR 5521
3SR Laboratory, BP53, 38041 Grenoble Cedex, France
Email: xuan-dung.vu@hmg.inpg.fr
†Institut de Radioprotection et de Sureté Nucléaire,
DSR/SAMS/BAGCS, 92260 Fontenay-aux-Roses, France

1. Introduction

After the success of the workshop IRIS_2010 organized in Paris – France in 2010, we continue improving our constitutive model of concrete behaviour and taking into account all recommendations issue from scientific committee in the simulation of two impact tests.

This report begins by the presentation and analyze of triaxial tests performed up to 100 MPa of confinement. Some results of test carried out at very high confinement (600 MPa) will also be presented for clarifying the effect of saturation ratio. These tests were simulated with the coupled damage plasticity model PRM. Necessary changes of this model to obtain a better prediction will be presented and the influence of these changes will be highlighted by comparing the two versions of the model and experimental results.

Then the new numerical results of the flexion test which produces the flexion mode of concrete plate and the punching test which produces the scabbing and spalling of concrete plate by the perforation of projectile will be compared to the experimental results.

2. Triaxial Tests

2.1 Experimental device

The GIGA press (figure 1) was designed to study the behavior of concrete under high confinement in a partnership between the 3SR laboratory and the CEA Gramat. It allows testing cylindrical concrete specimens under triaxial compression at a maximum confining pressure of 0.85 GPa and a maximum axial stress of 2.3 GPa [1-6].

![Figure 1: GIGA device and specimen dimensions](image)

2.2 Composition of concrete

The studied high performance concrete was used in the benchmark project “Improving the Robustness of Assessment Methodologies for Structures Impacted by Missiles (IRIS)” of the Nuclear Energy Agency (NEA) of OECD [8]. Concrete samples were manufactured by VTT Finnish laboratory for the IRIS project. The composition and properties of VTT concrete are given in Table 1.
Table 1: Composition and properties of VTT concrete

<table>
<thead>
<tr>
<th>Component</th>
<th>Value (kg/m^3)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Gravel (0.5/8)</td>
<td>925.9</td>
</tr>
<tr>
<td>Sand</td>
<td>646.1</td>
</tr>
<tr>
<td>Water</td>
<td>215</td>
</tr>
<tr>
<td>Cement (CEM II B 42.5)</td>
<td>489</td>
</tr>
<tr>
<td>Fly ash</td>
<td>88</td>
</tr>
<tr>
<td>Water-reducing agent</td>
<td>6.33</td>
</tr>
<tr>
<td>Cement paste volume</td>
<td>0.375</td>
</tr>
<tr>
<td>Density</td>
<td>2370</td>
</tr>
<tr>
<td>Compressive strength (MPa)</td>
<td>67</td>
</tr>
<tr>
<td>Porosity accessible to water</td>
<td>12%</td>
</tr>
<tr>
<td>Cement paste volume</td>
<td>0.375</td>
</tr>
<tr>
<td>E/C ratio</td>
<td>0.44</td>
</tr>
</tbody>
</table>

2.3 Experimental results

The concrete used in this study is representative of the containment of a nuclear power plant. Its strengths under unconfined compression and tension are about 67 MPa and 4.5 MPa respectively. The samples were tested under triaxial confining pressure varying from 15 MPa to 100 MPa. The porosity and the degree of saturation were measured prior to testing. The degree of saturation of concrete in the first series of tests is about 60%.

The performed triaxial tests consist in applying a hydrostatic pressure all around the specimen at 0.5 MPa/s up to pressure value $p_{conf}$. A constant displacement rate of 14 $\mu$m/s of the axial jack and a constant confining pressure $p_{conf}$ on the lateral face are then imposed.

Figure 2 shows the evolutions of the axial stress in function of the axial and circumferential strains for different confining pressures $p_{conf}$. The circumferential strain is the average measure of two gauges. The axial strain is obtained by means of an axial gauge and compared to the strain given by a LVDT sensor. These two measurements of the axial strain gave similar results indicating that the samples deform homogeneously. Figure 3 shows the volumetric behavior of concrete during the triaxial tests.

The analysis of tests highlights that stiffness and strength increase with confinement. This phenomenon is explained by the irreversible closure of porosity (compaction) with the mean stress increase. It is worth noting that with a confinement pressure less than 50 MPa, there is a stress peak in the axial behavior, while this phenomenon is not observed for the test at 100 MPa of confinement. At this confinement level, the reached limit state corresponds to a transition from contraction to dilatancy with no softening (figure 3).

![Figure 2: Axial stress vs. axial and circumferential strains for various confining pressures](image-url)
2.4 Influence of saturation ratio on concrete behavior at moderate and high confining pressures

A second triaxial test at 50 MPa of confinement has been performed on a saturated concrete specimen to study the influence of the saturation ratio ($Sr$). The procedure for testing saturated samples is described in [4]. The axial strain is measured by the LVDT sensor only. Figure 4 shows a comparison between the evolution of the axial stress as a function of axial strain obtained in this second test ($Sr = 100\%$) and the one obtained on a wet concrete ($Sr = 60\%$) at 50 MPa of confinement. The maximum axial stress is about 240 MPa in the two tests. This result is in agreement with the one obtained on a standard concrete [4]. Thus, the saturation ratio seems to have a low influence on the triaxial behavior of the tested high performance concrete at moderate confining pressures.

Figure 4: Comparison of axial behavior at 50 MPa of confining pressure for two different saturation ratios of concrete specimens

Because concrete may be submitted to very high triaxial stresses in case of impact, some additional triaxial tests were performed at high confining pressure.

Figure 5 shows the comparison between two hydrostatic tests at high confinement with different saturation ratios.
The two curves on figure 5 are confounded at a mean stress lower than 100 MPa, this zone corresponds to the elastic behavior of VTT concrete. Beyond this zone, the closure of porosity of concrete begins. Due to the important volume of paste cement (Table 1), the influence of creep may be important and strongly dependent on the saturation ratio [9]. That explains why the volumetric deformation of saturated concrete is higher than one of dried concrete when the porosity of concrete begins to be enclosed. But at high mean stress, water is compressed and because the compressibility of water is higher than the one of air, the volumetric deformation of saturated concrete is lower than the one of dried concrete (figure 5) at high mean stress.

2.5 Limit state of VTT concrete under triaxial compression

The material limit state is defined as the maximum volumetric strain reached during a test; it corresponds to a transition from a contraction behavior to a dilatancy one. At moderate confinement, this transition also corresponds to the peak stress. Figure 6 shows the limit states in the deviatoric stress / mean stress plane for the various tests described earlier.

On figure 6, it can be noted that, for a given mean stress, the maximum deviatoric stress reached during the test strongly depends on the saturation ratio of the concrete specimen. The presence of free water limits the admissible shear stress of concrete under confinement.

Figure 7 shows a zoom of the limit state curve of wet concrete at moderate confinement.
For a moderate confining pressure (lower than 50MPa), the influence of free water on concrete behavior seems to be low, whereas the influence of the saturation degree is significant for a confinement pressure of the order of 500MPa. This difference may have an important effect on the response of a concrete structure submitted to an impact and should be taken into account in simulations.

3. Material model description

3.1. PRM coupled model

We used the PRM coupled model \([1]\) (PONTIROLI-ROUQUAND-MAZAR) to simulate the concrete behaviour under impact. This constitutive model of concrete behaviour is a coupling between damage model and plastic model. This damage model is developed by C. PONTIROLI from J. MAZAR’s model \([3]\), it allows reproducing degradation mechanism and cracking of concrete at low confinement. In this model, two damage variables (Dt for tensile stress and Dc for compression stress) are used. The unilateral characteristic of concrete is taken into account to simulate the concrete behaviour under seismic solicitation \([2]\). It means that the stiffness of concrete structure can be restored when it state change from tensile stress to compression stress. The illustration of this model is presented in Figure 8:

![Figure 8. damage model with two scalar variables](image)
The damage model is very efficient to simulate the behavior of concrete for unconfined or low confined cyclic loading (Rouquand 2005) [5]. For very high dynamic loads leading to a higher pressure level, an elastic plastic model is more appropriate. For example, the impact of a projectile striking a concrete plate at 300 m/s induces local pressures near the projectile nozzle of several hundred MPa. The previous damage model cannot simulate the pore collapse phenomena rising at this pressure level. It also cannot model the shear plastic strain occurring in this pressure range. To overcome these limitations, the elastic and plastic model proposed by Krieg (1978) [4] has been chosen to simulate this kind of problem. From this simple elastic and plastic model, A. Rouquand made two improvements [6]. The first improvement has been introduced in order to simulate the nonlinear elastic behavior encountered during an unloading and reloading cycle under a high pressure level. A second improvement has been made to account for the water content effects introducing an effective stress theory as described by C. Mariotti (2002) [7]. This effect induces change on the pressure volume curve and on the shear plastic stress limit.

![Figure 9 modified elastic and plastic model](image)

The coupling of two models ensures a perfect continuity between the two model responses. The predicted stresses correspond to the damage model response if the maximum pressure is too low to start the pore collapse phenomena or if the shear stress is too low to reach the shear yield stress. If not, the plastic model is activated and pilots the evolutions until the extensions sufficiently increase to lead to a damage failure.

3.2. Improvement of PRM coupled model

Although the model coupled PRM allows obtaining a good prediction of concrete behavior under various load paths, some shortcomings exist and this study aims at fixing them.

3.2.1. Influence of the deviatoric stress into the volumetric behavior

The plasticity model assumes that inelastic volumetric and shear strains are obtained independently. The volumetric strain ($\varepsilon_v$) is assumed to depend on the mean stress ($\sigma_m$) only and the strain deviator tensor is obtained by means of a perfectly plastic model. One of the shortcomings of the present PRM coupled model is that it does not take into account the effect of the deviatoric stress $q$ on the volumetric behavior of the concrete. The present model assumes that the compaction curve, i.e. the volumetric strain ($\varepsilon_v$) vs. the mean stress ($\sigma_m$), as material data independent on the load path. Figure 3 shows that the inelastic volumetric strain depends on both $q$ and $\sigma_m$. It is then necessary to include the influence of $q$.
into the compaction curve of the material ($\varepsilon_v = \text{function} (\sigma_m, q)$).

To improve the PRM model, the original idea of two models of plasticity to calculate the inelastic strains is conserved. According to the test results [8-13], it is assumed that the maximum compaction is obtained under oedometric loading path, i.e. in uniaxial strain condition. The compaction curve of an oedometric test is then added as a second input data. The interest is on the one hand, this data is easily accessible to measurement and, on the other hand, that the oedometric test is the one which maximizes the compaction of concrete because the dilatancy is prevented.

The construction of the modified model is based on the following assumptions:
The curve of volumetric behavior of concrete is not supposed to be bijective. It is instead assumed bounded by the hydrostatic curve (Figure 10- opaque upper curve) and the oedometric curve (Figure 10 - dotted lower curve).

The variation of the mean stress $\sigma_m$, between the bounded curves is given by:

$$d\sigma_m = \alpha d\varepsilon_v$$

With:

$$\alpha = \alpha_H + (\alpha_O - \alpha_H)\min\left(\frac{dq}{d\sigma_m}, 1\right)$$

Where (figure 8):

- $\alpha_H = \frac{d\sigma_m}{d\varepsilon_v}$ for a hydrostatic path ;
- $\alpha_O = (\frac{d\sigma_m}{d\varepsilon_v})_O$ for un oedometric path ;
- $dq / d\sigma_m =$ load path direction ;
- $(dq/d\sigma_m)_O =$ oedometric load path direction.

With formulae (1) and (2), the volumetric strain $\varepsilon_v$ depends on both the mean stress $\sigma_m$ and the deviatoric shear stress $q$, the compaction is then increased in presence of shear compared with volumetric strain obtained with the initial model.

---

3.2.2. Influence of the water saturation ratio into the volumetric behavior

Two kinds of approaches exist to characterize the behavior of a porous medium scale
homogenized according to its properties at the microscopic level. The “mixing law” approaches take into account, at the microscopic level, the interaction between the two phases (liquid + solid) by means of simple rheological models for each phase associated in series or in parallel. Poromechanical approaches [14] assume that the concepts of mechanics in continuous media are valid at the macroscopic scale when the two phases (liquid + solid) overlap.

In the present PRM coupled model, the concept of effective stress is used to take into account the presence of water in confined concrete using the first approach. The drawback of this approach is that the behavior of the material becomes elastic after reaching the consolidation point (closure of all open pores), which is not observed experimentally. In the improved model, the second poro-mechanical approach is used to take into account the effect of free water.

The studied porous medium is assumed to be composed of a solid phase (skeleton) and a fluid phase occupying the voids [14]. The concept of the effective stress is introduced to separate the fluid pressure in the calculation of the total pressure

$$\sigma_{\text{tot}} = \sigma_M + bp$$  \hspace{1cm} (3)

With \(\sigma_{\text{tot}}\) the total stress, \(\sigma_M\) transmitted by the matrix at a macroscopic scale, \(p\) the pore pressure, and \(b\) the Biot coefficient which depends on the nature of the porosity.

The calculation of pore pressure \(p\) is based on the Mie Gruneisen equation of state:

$$P = \frac{\rho C_0^2 (\varepsilon_V - \varepsilon_{Vp})}{(1 - s(\varepsilon_V - \varepsilon_{Vp}))^3} \left[ 1 - \frac{\Gamma_0 (\varepsilon_V - \varepsilon_{Vp})}{\rho_0 E_M} \right] + \Gamma_0 \rho_0 E_M$$ \hspace{1cm} (4)

Where \(C_0\) is the sound celerity (\(C_0 = 1500\,\text{m/s}\)), \(\rho_0\) is the density (\(\rho_0 = 1000\,\text{kg/m}^3\) for water), \(s\) and \(\Gamma_0\) are two Mie Gruneisen coefficients (\(s = 1.75\) and \(\Gamma_0 = 0.28\) for water). \(E_M\) is the internal energy per unit mass. This energy is considered negligible for water temperature and ambient pressure.

\(\sigma_M\) and \(b\) can be obtained by the following formulae [14]:

$$\sigma_M = K_0 \varepsilon_v$$ \hspace{1cm} (5)

$$b = 1 - \frac{K_0}{K_S}$$ \hspace{1cm} (6)

\(K_0\) is the modulus of the material drained \(\varepsilon_V\) is the volumetric strain at homogenized scale, \(K_S\) the compressibility modulus of the skeleton. From equation (6), in the particular case where \(K_0 \ll K_S\), \(b\) is close to 1, which simplifies the equation (3) and becomes \(\sigma_{\text{tot}} = \sigma_M + p\) (Terzaghi formula). In contrast, when \(K_0 \approx K_S\) (dry concrete case), \(b\) tends to 0. In the end, thanks to the technical of homogenization of the drained porous medium [14] the ratio \(K_0/K_S\) can be estimated as follows:

$$\frac{K_0}{K_S} = (1 - \varphi)^2$$ \hspace{1cm} (7)

Where \(\varphi\) is the porosity of the porous medium at the current state.
With this new hypothesis, as the material reaches the point of consolidation (void pores are closed), the volumetric behavior remains nonlinear due to the fact that the voids filled with water continue to be compressed under compaction. Another advantage of this improvement is the unique point of consolidation instead of two points in the original model [Figure 11].

### 3.2.3. Improvement of coupling variable

When we used the coupling variable proposed in PRM coupled model to simulate concrete behavior of impact test, it presented an effect of checkerboard in the damage simulation. It means that this method couldn’t generate a homogeneity or continuity of stress in the impact zone. To remedy this shortcoming, we proposed to change this coupling variable formula by the other one that is formulated using a dual criterion and calculated as a function of the three stress invariants $I_1$, $J_2$, and $\theta$ [15]. Three modes of failure can be identified. For the pure cracking mode, the maximum principal stress must be positive, that is, $\sigma_1 > 0$, and it is assumed that the maximum principal strain is compressive, that is, $\varepsilon_1 < 0$, for the pure crushing mode. A crushing coefficient is subsequently defined as follows by combining these two conditions and using the stress invariants $I_1$, $J_2$, and $\theta$. $I_1$ is the first invariant of the stress tensor, $J_2$ is the second invariant of the stress tensor, and $\theta$ is the angle of similarity that is a function of the third invariant $J_3$ of the stress tensor.

$$\alpha = - \frac{I_1}{3.464 \sqrt{J_2} \cos \theta} \quad \text{for} \quad 0^\circ \leq \theta \leq 60^\circ$$

(8)

The failure modes can then be identified as follows:
- Pure cracking = $\alpha < 1$
- Pure crushing = $\alpha > \frac{(1 + \nu)}{(1 - 2\nu)}$
- Mixed failure mode involving cracking and crushing = $1 \leq \alpha \leq \frac{(1 + \nu)}{(1 - 2\nu)}$

In the previously mentioned relationships, $\nu$ is Poisson’s ratio. Figure 12 shows the failure zones in the octahedral shear and normal stress plane.
3.3. Identification of PRM coupled model

In our simulations, we want to use only one constitutive model of concrete behaviour to simulate different types of dynamic solicitation as impact case. All parameters identified here are used in two impact tests that we will present in the next part of this report.

Table gives parameters of the PRM damage model. Table 4 gives parameters of the PRM crash model. Yellow background rows correspond to measured material data. Other values are deduced from the previous one or are representative of a standard concrete. PRM compressive strength parameter refers to cylindrical concrete specimens.

<table>
<thead>
<tr>
<th>N°</th>
<th>Parameters</th>
<th>Designation</th>
<th>Unit (MKS)</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>E₀ (&gt;0)</td>
<td>Young Modulus</td>
<td>N/m²</td>
<td>2.8 (10^{10})</td>
</tr>
<tr>
<td>2</td>
<td>ν₀ (&gt;0)</td>
<td>Poisson ratio</td>
<td>-</td>
<td>0.1875</td>
</tr>
<tr>
<td>3</td>
<td>σ₉ (&lt;0)</td>
<td>Tensile strength</td>
<td>N/m²</td>
<td>4.7 (10^{6})</td>
</tr>
<tr>
<td>4</td>
<td>σ₉ (&gt;0)</td>
<td>Compressive strength</td>
<td>N/m²</td>
<td>-67.0 (10^{6})</td>
</tr>
<tr>
<td>5</td>
<td>εₑ₉ erosion (&gt;0)</td>
<td>maximum strain before erosion</td>
<td>-</td>
<td>0.23</td>
</tr>
<tr>
<td>6</td>
<td>σₑ₀ final (&gt;0)</td>
<td>Residual tensile stress</td>
<td>N/m²</td>
<td>0.0</td>
</tr>
<tr>
<td>7</td>
<td>β (&gt; 0)</td>
<td>Mazars coefficient</td>
<td>-</td>
<td>1-1.05</td>
</tr>
<tr>
<td>8</td>
<td>σᵣ₀ (&lt;0)</td>
<td>Initial closure stress</td>
<td>N/m²</td>
<td>-4.7 (10^{6})</td>
</tr>
<tr>
<td>9</td>
<td>σₑ₀ (&gt;0)</td>
<td>Focus point of unloading in compression</td>
<td>N/m²</td>
<td>67.0 (10^{6})</td>
</tr>
<tr>
<td>11</td>
<td>Gₑ (&gt;0)</td>
<td>Fracture energy</td>
<td>N/m</td>
<td>120.0</td>
</tr>
<tr>
<td></td>
<td></td>
<td>First tensile strain rate coefficient</td>
<td>-</td>
<td>0.823</td>
</tr>
<tr>
<td>---</td>
<td>---</td>
<td>--------------------------------------</td>
<td>---</td>
<td>-------</td>
</tr>
<tr>
<td>14</td>
<td>b₁ (&gt;0)</td>
<td>Second tensile strain rate coefficient</td>
<td>-</td>
<td>0.329</td>
</tr>
<tr>
<td>15</td>
<td>a₃ (&gt;0)</td>
<td>First compression strain rate coefficient</td>
<td>-</td>
<td>0</td>
</tr>
<tr>
<td>16</td>
<td>b₃ (&gt;0)</td>
<td>Second compression strain rate coefficient</td>
<td>-</td>
<td>0</td>
</tr>
<tr>
<td>17</td>
<td>β₁ (&gt;0)</td>
<td>Damping coefficient in the elastic domain</td>
<td>-</td>
<td>0.02</td>
</tr>
<tr>
<td>18</td>
<td>β₂ (&gt;0)</td>
<td>2&lt;sup&gt;nd&lt;/sup&gt; damping coefficient induced by damage</td>
<td>-</td>
<td>0.05</td>
</tr>
<tr>
<td>19</td>
<td>ρ₀ (&gt;0)</td>
<td>Density</td>
<td>Kg/m³</td>
<td>2272.0</td>
</tr>
</tbody>
</table>

Table 3: Material data of elastic plastic model

### Plastic threshold

<table>
<thead>
<tr>
<th>a₀ (Pa²)</th>
<th>a₁ (Pa)</th>
<th>a₂</th>
<th>P&lt;sub&gt;ca&lt;/sub&gt; elas tens (Pa)</th>
<th>e&lt;sub&gt;max&lt;/sub&gt; (Pa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>7.0E+14</td>
<td>2.0E+08</td>
<td>1.0E+00</td>
<td>1.000E+01</td>
<td>6.000E+09</td>
</tr>
</tbody>
</table>

### Volumetric deformation- mean stress curve of hydrostatic test (24 points)

<table>
<thead>
<tr>
<th>p₁ elas (Pa)</th>
<th>e₁₁ elas</th>
<th>p₂(Pa)</th>
<th>e₁₂</th>
<th>p₃(Pa)</th>
<th>e₁₃</th>
<th>p₄(Pa)</th>
<th>e₁₄</th>
<th>p₅(Pa)</th>
<th>e₁₅</th>
<th>p₁₆(Pa)</th>
<th>e₁₆</th>
</tr>
</thead>
<tbody>
<tr>
<td>7.03E+07</td>
<td>-4,7100E-03</td>
<td>1.13E+08</td>
<td>-8,0600E-03</td>
<td>1.68E+08</td>
<td>-1,3000E-02</td>
<td>2.02E+08</td>
<td>-1,6770E-02</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>p₅(Pa)</td>
<td>e₁₅</td>
<td>p₄(Pa)</td>
<td>e₁₆</td>
<td>p₃(Pa)</td>
<td>e₁₇</td>
<td>p₁₄(Pa)</td>
<td>e₁₈</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2.31E+08</td>
<td>-2,0800E-02</td>
<td>2.62E+08</td>
<td>-2,5840E-02</td>
<td>2.95E+08</td>
<td>-3,1490E-02</td>
<td>3.35E+08</td>
<td>-3,9200E-02</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>p₃(Pa)</td>
<td>e₁₉</td>
<td>p₁₀(Pa)</td>
<td>e₁₁₀</td>
<td>p₁₁(Pa)</td>
<td>e₁₁₁</td>
<td>p₁₂(Pa)</td>
<td>e₁₁₂</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3.75E+08</td>
<td>-4,6950E-02</td>
<td>4.20E+08</td>
<td>-5,5780E-02</td>
<td>4.64E+08</td>
<td>-6,4810E-02</td>
<td>5.07E+08</td>
<td>-7,3050E-02</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>p₁₁(Pa)</td>
<td>e₁₁₃</td>
<td>p₁₄(Pa)</td>
<td>e₁₁₄</td>
<td>p₁₅(Pa)</td>
<td>e₁₁₅</td>
<td>p₁₆(Pa)</td>
<td>e₁₁₆</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>5.86E+08</td>
<td>-8,6800E-02</td>
<td>6.69E+08</td>
<td>-1,0070E-01</td>
<td>7.48E+08</td>
<td>-1,1330E-01</td>
<td>8.70E+08</td>
<td>-1,3230E-01</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>p₂₁(Pa)</td>
<td>e₁₁₇</td>
<td>p₁₈(Pa)</td>
<td>e₁₁₈</td>
<td>p₁₉(Pa)</td>
<td>e₁₁₉</td>
<td>p₂₀(Pa)</td>
<td>e₁₂₀</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>9.76E+08</td>
<td>-1,4660E-01</td>
<td>1.12E+09</td>
<td>-1,6020E-01</td>
<td>1.31E+09</td>
<td>-1,7540E-01</td>
<td>1.55E+09</td>
<td>-1,9070E-01</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>p₂₂(Pa)</td>
<td>e₁₂₁</td>
<td>p₂₃(Pa)</td>
<td>e₁₂₂</td>
<td>p₂₄(Pa)</td>
<td>e₁₂₃</td>
<td>p₂₅(Pa)</td>
<td>e₁₂₄</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1.79E+09</td>
<td>-2,0100E-01</td>
<td>2.06E+09</td>
<td>-2,1050E-01</td>
<td>2.33E+09</td>
<td>-2,1970E-01</td>
<td>2.6589E+09</td>
<td>-2,2980E-01</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

### Volumetric deformation- mean stress curve of oedometric test (24 points)

<table>
<thead>
<tr>
<th>p₁ elas (Pa)</th>
<th>e₁₁ elas</th>
<th>p₂(Pa)</th>
<th>e₁₂</th>
<th>p₃(Pa)</th>
<th>e₁₃</th>
<th>p₄(Pa)</th>
<th>e₁₄</th>
<th>p₅(Pa)</th>
<th>e₁₅</th>
<th>p₁₆(Pa)</th>
<th>e₁₆</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.0000E+07</td>
<td>-1,3400E-03</td>
<td>4.3E+07</td>
<td>-4,1300E-03</td>
<td>5.70E+07</td>
<td>-8,8800E-03</td>
<td>7.37E+07</td>
<td>-1,4670E-02</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>p₅(Pa)</td>
<td>e₁₅</td>
<td>p₄(Pa)</td>
<td>e₁₆</td>
<td>p₃(Pa)</td>
<td>e₁₇</td>
<td>p₈(Pa)</td>
<td>e₁₈</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>9.42E+07</td>
<td>-2,1810E-02</td>
<td>1.13E+08</td>
<td>-2,8550E-02</td>
<td>1.32E+08</td>
<td>-3,4800E-02</td>
<td>1.54E+08</td>
<td>-4,2400E-02</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
### 3.4. Comparison between experimental results and simulations of tests.

The simulation results obtained with the original PRM coupled model and the modified model are compared to experimental results on figures 10 to 12.

#### 3.4.1. Wet concrete

Figure 13 and Error! Not a valid bookmark self-reference. show results for a concrete specimen with a saturation degree of 60% and submitted to triaxial compression with confining pressures varying from 15 to 100 MPa. At this saturation ratio and because of a moderate confining pressure, there is not the effect of free water on concrete behavior. The initial PRM coupled model allows a good prediction of the maximum stress but strains are significantly underestimated. Taking into account the influence of the deviatoric stress on the volumetric behavior of concrete significantly improves the prediction of the volumetric strain.

### Oedometric q₀ curve in relation with 24 points of mean stress p (24 points)

<table>
<thead>
<tr>
<th>p (Pa)</th>
<th>e₁₀</th>
<th>p₁₀ (Pa)</th>
<th>e₁₁₀</th>
<th>p₁₁ (Pa)</th>
<th>e₁₁₁</th>
<th>p₁₂ (Pa)</th>
<th>e₁₁₂</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.84E+08</td>
<td>-5.1450E-02</td>
<td>2.13E+08</td>
<td>-6.0170E-02</td>
<td>2.49E+08</td>
<td>-7.0400E-02</td>
<td>2.92E+08</td>
<td>-8.1800E-02</td>
</tr>
<tr>
<td>p₁₃ (Pa)</td>
<td>e₁₃</td>
<td>p₁₄ (Pa)</td>
<td>e₁₄</td>
<td>p₁₅ (Pa)</td>
<td>e₁₅</td>
<td>p₁₆ (Pa)</td>
<td>e₁₆</td>
</tr>
<tr>
<td>3.51E+08</td>
<td>-9.7300E-02</td>
<td>4.03E+08</td>
<td>-1.1090E-01</td>
<td>4.62E+08</td>
<td>-1.2510E-01</td>
<td>5.54E+08</td>
<td>-1.4150E-01</td>
</tr>
<tr>
<td>p₁₇ (Pa)</td>
<td>e₁₇</td>
<td>p₁₈ (Pa)</td>
<td>e₁₈</td>
<td>p₁₉ (Pa)</td>
<td>e₁₉</td>
<td>p₂₀ (Pa)</td>
<td>e₂₀</td>
</tr>
<tr>
<td>7.08E+08</td>
<td>-1.5570E-01</td>
<td>8.92E+08</td>
<td>-1.6750E-01</td>
<td>1.12E+09</td>
<td>-1.7830E-01</td>
<td>1.37E+09</td>
<td>-1.8780E-01</td>
</tr>
<tr>
<td>p₂₁ (Pa)</td>
<td>e₂₁</td>
<td>p₂₂ (Pa)</td>
<td>e₂₂</td>
<td>p₂₃ (Pa)</td>
<td>e₂₃</td>
<td>p₂₄ (Pa)</td>
<td>e₂₄</td>
</tr>
<tr>
<td>1.61E+09</td>
<td>-1.9630E-01</td>
<td>1.93E+09</td>
<td>-2.0690E-01</td>
<td>2.33E+09</td>
<td>-2.1970E-01</td>
<td>2.66E+09</td>
<td>-2.2980E-01</td>
</tr>
</tbody>
</table>

**Loading stiffness, unloading stiffness, porosity, water content**

<table>
<thead>
<tr>
<th>Kgrain cons (Pa)</th>
<th>Kgrain unload (Pa)</th>
<th>Porsec</th>
<th>eau Sr</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.80E+10</td>
<td>3.80E+10</td>
<td>0.11E+00</td>
<td>6.00E-01</td>
</tr>
</tbody>
</table>
Figure 13 Axial stress vs. axial and circumferential strains: Comparisons of experimental results with simulation results obtained with the initial (PRM-i) and new model (PRM-n) for a wet concrete specimen under moderate confinement.

Figure 14 Mean stress vs. volumetric strain: Comparisons of experimental results with simulation results obtained with the initial (PRM-i) and new model (PRM-n) for a wet concrete specimen under moderate confinement.

3.4.2. Saturated concrete

Figure 15 shows experimental volumetric behavior of saturated and dry concrete and their comparison with simulation results obtained by both the initial and modified PRM coupled model. The initial model considers an elastic behavior after consolidation (closure of voids); while the modified model gives a simulation result closer to the experimental one.
3.5. Johnson Cook plastic and damage model

The behavior of the metallic parts of the structure, the steel reinforcement and the metallic projectile tube is simulated using the Johnson Cook dynamic failure model. In this model, the plastic stress $\bar{\sigma}$ is related to the plastic strain $\bar{\varepsilon}^{pl}$ to the plastic strain rate $\dot{\varepsilon}^{pl}$ and to the damage variable $D$ via the following expression:

$$\bar{\sigma} = (1 - D) \left[ A + B \left( \bar{\varepsilon}^{pl} \right)^n \right] \left[ 1 + C \ln \left( \frac{\dot{\varepsilon}^{pl}}{\varepsilon_0^{pl}} \right) \right]$$

$\varepsilon_0^{pl}$ is the plastic threshold. The Damage $D$ is also related to the plastic strain as follow:

$$D = \min \left( \frac{L_e \max \left( \sum \Delta \bar{\varepsilon}^{pl}_i - \bar{\varepsilon}^{pl0}, 0 \right)}{L_0 \bar{\varepsilon}^{pl}_i}, 1 \right)$$

$\bar{\varepsilon}^{pl0}$ is the plastic threshold. The Damage variable $D$ becomes to increase when the cumulated plastic strain $\sum \Delta \bar{\varepsilon}^{pl}_i$ becomes greater than the plastic threshold $\bar{\varepsilon}^{pl0}$. The plastic threshold $\bar{\varepsilon}^{pl0}$ is given by:

$$\bar{\varepsilon}^{pl0} = d_1 \left[ 1 + d_2 \ln \left( \frac{\dot{\varepsilon}^{pl}}{\varepsilon_0^{pl}} \right) \right]$$

$d_1$ and $d_2$ are material parameters. When $d_2$ is positive the plastic threshold increases with the plastic strain rate.

$L_e$ is the element characteristic length. This size is defined by the distance between the two nodes forming the element (1D finite element), by the square root of the element surface (2D finite element) or by the characteristic length of an equivalent 2D element.
finite element) or by the cubic root of the finite element volume (3D finite element). $L_0 \varepsilon_{pl}^f$ is the failure displacement. This failure displacement can be related to the plastic failure strain $\varepsilon_{pl}^f$ using the internal material length $L_0$. Figure 16 shows an example of the stress strain curve obtained with the Johnson Cook dynamic failure model. In this example we have the following material parameters: $A = 5.6 \times 10^8$, $B = 2.3 \times 10^8$, $n = 0.36$, $C = 0.0141$, $d_1 = 0.045$, $d_2 = 0.203$, $L_0 = 0.2$. In this figure four curves are plotted. They correspond to different constant strain rate values given in the legend.

![Figure 16 Example of a stress strain curve (Johnson Cook dynamic failure model)](image)

3.6. Simulation of impact test

3.6.1. Flexion test

3.6.1.1. Choice of finite element mesh

A 3-D finite element model with solid brick and shell elements is used to simulate the VTT flexural test. The projectile shown on figure 63 is a 254 mm diameter steel tube. Its length is 2.088m including 0.088m for the front and the caped part. A total of 48 x 134 = 6 432 (4 nodes) shell elements are used to mesh the projectile. The total projectile weight is 49.99 kg.
Figure 17 Projectile characteristics

Figure 18 shows the projectile mesh. The shell thickness is 2mm except in the rear part (yellow part) where it is equal to 18.215 mm.

Figure 18 Projectile mesh

The target size is 2.1 m by 2.1 m. The thickness is 15 cm (instead of 25 cm in the punching test). Figure 19 shows the mesh used for the concrete plate (left part) and for the steel ring and the steel reinforcement. The concrete plate is composed of 84 by 84 by 10 = 70560 “C3D8R” finite elements (8 nodes solid elements with reduced integration). The reinforcement is modelled using 2 nodes beam elements with a circular cross section.
Figure 19 Mesh of the reinforced concrete slab

All the nodes along the red line (Figure 19) have their displacements in the z direction set to zero. The metallic frame around the concrete plate (U shape) is modeled using 6 520 shell elements (4 nodes reduced integration shell S8R elements). The thickness of the metallic ring is 12 mm. The frame weight is 314.4 kg.

The reinforcement is modeled using $84 \times 38 \times 2 \times 2 = 12,768$ beam elements and the rebar diameter is 6 mm. The space between 2 rebars is 55 mm. The rebar length is 2,035 m. Transverse rebars are also introduced using $28 \times 28 \times 6 = 4,704$ beam elements. The transverse rebar diameter is also 6 mm. The distance between two transverse rebar is 75 mm along the 2 orthogonal directions. The concrete cover is 15 mm.

<table>
<thead>
<tr>
<th>Target geometry</th>
<th>Modeling</th>
<th>Experimental</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Longitudinal vertical rebars</td>
<td>Front face</td>
<td>Rear face</td>
<td>Front face</td>
</tr>
<tr>
<td>Diameter of rebars</td>
<td>6</td>
<td>6</td>
<td>6</td>
</tr>
<tr>
<td>Number of vertical rebars</td>
<td>38</td>
<td>38</td>
<td>38</td>
</tr>
<tr>
<td>Concrete cover</td>
<td>20</td>
<td>20</td>
<td>20</td>
</tr>
<tr>
<td>Longitudinal horizontal rebars</td>
<td>Front face</td>
<td>Rear face</td>
<td>Front face</td>
</tr>
<tr>
<td>Diameter of rebars</td>
<td>6</td>
<td>6</td>
<td>6</td>
</tr>
<tr>
<td>Number of horizontal rebars</td>
<td>38</td>
<td>38</td>
<td>38</td>
</tr>
<tr>
<td>Concrete cover</td>
<td>20</td>
<td>20</td>
<td>20</td>
</tr>
<tr>
<td>Transverse rebars</td>
<td>Modeling</td>
<td>Experimental</td>
<td>Unit</td>
</tr>
<tr>
<td>Diameter of rebars</td>
<td>6</td>
<td>6</td>
<td>6</td>
</tr>
<tr>
<td>Number of rebars / m³</td>
<td>176</td>
<td>176</td>
<td>/m³</td>
</tr>
<tr>
<td>Density</td>
<td>50.0</td>
<td>50.0</td>
<td>cm²/m²</td>
</tr>
</tbody>
</table>

3.6.1.2. Reinforcement material parameters
The Johnson Cook dynamic failure model presented in chapter 1 is used. Table 11 gives the reinforcement (steel) material parameters. Figure 20 gives the corresponding stress strain curves at constant strain rate of respectively $5.0 \times 10^{-5}$, $2.0 \times 10^{-1}$, $2.0 \times 10^{0}$, $8.5 \times 10^{0}$. These curves correspond to $L_0 = 0.2$. These material parameters are deduced from the experimental data shown on Figure 21.

Table 5 reinforcement material data of the Johnson Cook dynamic failure model

<table>
<thead>
<tr>
<th>N°</th>
<th>Parameters</th>
<th>Designation</th>
<th>Unit</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>$E_0$</td>
<td>Young Modulus</td>
<td>N/m$^2$</td>
<td>$2.0 \times 10^{11}$</td>
</tr>
<tr>
<td>2</td>
<td>$\nu_0$</td>
<td>Poisson ratio</td>
<td></td>
<td>0.3</td>
</tr>
<tr>
<td>3</td>
<td>$A$</td>
<td>Initial yield stress</td>
<td>N/m$^2$</td>
<td>$6.0 \times 10^{8}$</td>
</tr>
<tr>
<td>4</td>
<td>$B$</td>
<td>Hardening yield stress</td>
<td>N/m$^2$</td>
<td>$1.53 \times 10^{8}$</td>
</tr>
<tr>
<td>5</td>
<td>$n$</td>
<td>Johnson Cook exponent</td>
<td></td>
<td>0.36</td>
</tr>
<tr>
<td>6</td>
<td>$C$</td>
<td>Strain rate effect coefficient</td>
<td></td>
<td>0.0141</td>
</tr>
<tr>
<td>7</td>
<td>$\dot{\varepsilon}_0$</td>
<td>Reference strain rate</td>
<td></td>
<td>$5.0 \times 10^{-5}$</td>
</tr>
<tr>
<td>8</td>
<td>$d_1$</td>
<td>Plastic strain at damage initiation</td>
<td></td>
<td>0.045</td>
</tr>
<tr>
<td>9</td>
<td>$d_2$</td>
<td>Coefficient of the strain rate effect on the failure strain</td>
<td></td>
<td>0.203</td>
</tr>
<tr>
<td>11</td>
<td>$L_0 \hat{\varepsilon}_f$</td>
<td>Failure displacement</td>
<td></td>
<td>0.004</td>
</tr>
</tbody>
</table>

Figure 20 Stress strain curve for the Johnson Cook dynamic failure model (reinforcement of the concrete slab, VTT flexural test).
3.6.1.3. Projectile material parameters

The Johnson Cook dynamic failure model presented in chapter 1 is also used. Table 6 gives the material parameters of the steel used in the projectile. Figure 22 gives the corresponding stress strain curves at constant strain rate of respectively $5.0 \times 10^{-5}$, $5 \times 10^{-2}$, $5 \times 10^{1}$, $1 \times 10^{3}$. These curves correspond to $L_0 = 0.2$. These material parameters are deduced from the experimental data shown on Figure 23. This experimental stress strain curve is supposed to be obtained at a very low strain rate of $5 \times 10^{05}$.

![Figure 21 measured static stress strain curve of the reinforcement](image)

Table 6 projectile material data of the Johnson Cook dynamic failure model

<table>
<thead>
<tr>
<th>N°</th>
<th>Parameters</th>
<th>Designation</th>
<th>Unit</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>$E_0$</td>
<td>Young Modulus</td>
<td>N/m²</td>
<td>$2.0 \times 10^{11}$</td>
</tr>
<tr>
<td>2</td>
<td>$\nu_0$</td>
<td>Poisson ratio</td>
<td>-</td>
<td>0.3</td>
</tr>
<tr>
<td>3</td>
<td>A</td>
<td>Initial yield stress</td>
<td>N/m²</td>
<td>$2.7 \times 10^{8}$</td>
</tr>
<tr>
<td>4</td>
<td>B</td>
<td>Hardening yield stress</td>
<td>N/m²</td>
<td>$6.66 \times 10^{8}$</td>
</tr>
<tr>
<td>5</td>
<td>n</td>
<td>Johnson Cook exponent</td>
<td>-</td>
<td>0.384</td>
</tr>
<tr>
<td>6</td>
<td>C</td>
<td>Strain rate effect coefficient</td>
<td>-</td>
<td>0.025</td>
</tr>
<tr>
<td>7</td>
<td>$\dot{\varepsilon}_0$</td>
<td>Reference strain rate</td>
<td>-</td>
<td>$5.0 \times 10^{-5}$</td>
</tr>
<tr>
<td>8</td>
<td>$d_1$</td>
<td>Plastic strain at damage initiation</td>
<td>-</td>
<td>0.18</td>
</tr>
<tr>
<td>9</td>
<td>$d_2$</td>
<td>Coefficient of the strain rate effect on the failure strain</td>
<td>-</td>
<td>0.24</td>
</tr>
<tr>
<td>11</td>
<td>$L_0 \varepsilon_f^{pl}$</td>
<td>Failure displacement</td>
<td>-</td>
<td>0.005</td>
</tr>
</tbody>
</table>
3.6.1.4. Results

3.6.1.4.1. Concrete plate results – comparison with results obtained in workshop 2010

Displacement of concrete plate

In the workshop IRIS_2010, we simulated a quarter of the structural system due its symmetry [Figure 24]. The movement of the concrete plate is prevented at red lines [Figure 24] in movement directions of projectile (U3 = 0) For better understanding the influence of the strain rate effect of steel, we first didn’t take into account the effect of strain rate in both projectile and rebars reinforced in concrete plate in this simulation. Only the strain rate of
concrete was activated. The original PRM coupled constitutive model was used to simulate this test.

Figure 24 model of a quarter of structural system in workshop 2010

Figure 25 computed displacements at points w1, w2, w3, w4 and w5 _ workshop 2010

Displacement time histories at the rear of the slab

Figure 25 computed displacements at points w1, w2, w3, w4 and w5 _ workshop 2010
In this simulation, the displacements of 5 points computed on the rear face of the concrete plate were obtained and presented on the Figure 25. Theses computed displacements were very low in comparison with those measured in the test [Figure 26] (less than 5 times). In the next step of simulation, we took into account the strain rate in both the projectile and the rebars. The results obtained in this step were really better those obtained in the first step of simulation [Figure 27] (less than 2 times).
Although the displacements of 5 points computed on concrete plate were improved when we took into account the strain rate of both projectile and rebars in the simulation, we couldn’t reproduce displacements measured on in the flexion test. In the workshop 2012, after improving the PRM coupled model of concrete that we presented above, the results obtained in this simulation approached the results of tests [Figure 28]
Figure 28 Comparison between computed displacements and measured displacements at points w1, w2, w3, w4 and w5 – with strain rate in projectile and rebars – modified PRM coupled model _ workshop 2012

Figure 29 Damage on the front face and experimental front face after the impact
The damage obtained on the front face and the rear face of concrete plate simulated by modified PRM coupled model are very similar this one of experimental results.

3.6.1.4.2. Projectile results

Figure 31 shows the projectile shape at time $t = 40$ ms (at the end of the numerical simulation). The computed residual projectile length is 1.029 m in comparison with 1.14 m of the measured residual length of projectile. If the folded part isn’t taken into account, the computed residual projectile length is 0.996 m in comparison with 0.97 m of the measured residual length of projectile [Figure 32]. To obtain a better predict of the computed residual length of projectile, a little change of steel behavior should be realized.

3.6.2. Punching test

3.6.2.1. Finite element mesh

The projectile is a 168 mm diameter steel tube filled with concrete [Figure 33]. Its length is 64 cm including 5 cm for the front and the caped part. In our simulation, 3-D finite elements (8 nodes solid brick elements) are used to mesh the projectile because the concrete material inside the steel tube strongly prevents the possible projectile deformations. A total of 1400 3-D elements are used to mesh the projectile. The total projectile weight is 47 kg.
The reinforced concrete target of 2.1 m by 2.1 m by 0.25 m is inserted between two stiff metallic frames as shown on Figure 34. The right part of this figure shows a detailed view of the boundary conditions for the square concrete slab. The concrete is casted inside a metallic ring (yellow part in Figure 34). This metallic ring is meshed using 3-D solid brick elements.

The reinforced concrete target of 2.1 m by 2.1 m by 0.25 m is inserted between two stiff metallic frames as shown on Figure 34. The right part of this figure shows a detailed view of the boundary conditions for the square concrete slab. The concrete is casted inside a metallic ring (yellow part in Figure 34). This metallic ring is meshed using 3-D solid brick elements.

The left part of Figure 35 shows a view of the 3D finite element model of the concrete slab. The mesh is composed of 84 by 84 by 10 = 70,560 solid (8 nodes) brick elements. The movement of concrete plate is prevented from the movement direction of projectile by 8 steel rods. The right part of Figure 35 shows the metallic ring around the concrete plate (yellow part). Elements are used for the ring structure that shares the same nodes than the concrete plate. The thickness of the metallic ring is 25 mm and its total mass is 708 kg. The beam
elements used to simulate the reinforcement can also be seen on the right part of Figure 35. This reinforcement is composed of a total of $24 \times 2$ (horizontal + vertical) $\times 2$ (front face and rear face) = 96 rebars. There is no transverse rebar to connect the front face and the rear face layers. The rebar diameter is 10 mm. The rebar space is 90 mm and the concrete cover is 30 mm.

![Figure 35 view of the finite element mesh of the reinforced concrete plate](image)

**3.6.2.2. Reinforcement material parameters**

The Johnson Cook dynamic failure model is used. Table 7 gives the reinforcement (steel) material parameters. Figure 36 gives the corresponding stress strain curves at constant strain rate of respectively $5.0 \times 10^{-5}$, $2.0 \times 10^{-1}$, $2.0 \times 10^{0}$, $8.5 \times 10^{0}$. These curves correspond to $L_0 = 0.2$. These material parameters are deduced from the experimental data shown on Figure 37.

<table>
<thead>
<tr>
<th>N°</th>
<th>Parameters</th>
<th>Designation</th>
<th>Unit</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>$E_0$</td>
<td>Young Modulus</td>
<td>N/m²</td>
<td>$2.0 \times 10^{11}$</td>
</tr>
<tr>
<td>2</td>
<td>$\nu_0$</td>
<td>Poisson ratio</td>
<td>-</td>
<td>0.3</td>
</tr>
<tr>
<td>3</td>
<td>A</td>
<td>Initial yield stress</td>
<td>N/m²</td>
<td>$5.8 \times 10^{8}$</td>
</tr>
<tr>
<td>4</td>
<td>B</td>
<td>Hardening yield stress</td>
<td>N/m²</td>
<td>$2.3 \times 10^{8}$</td>
</tr>
<tr>
<td>5</td>
<td>n</td>
<td>Johnson Cook exponent</td>
<td>-</td>
<td>0.36</td>
</tr>
<tr>
<td>6</td>
<td>C</td>
<td>Strain rate effect coefficient</td>
<td>-</td>
<td>0.0141</td>
</tr>
<tr>
<td>7</td>
<td>$\dot{\varepsilon}_0$</td>
<td>Reference strain rate</td>
<td>-</td>
<td>$5.0 \times 10^{5}$</td>
</tr>
<tr>
<td>8</td>
<td>$d_1$</td>
<td>Plastic strain rate at damage initiation</td>
<td>-</td>
<td>0.045</td>
</tr>
<tr>
<td>9</td>
<td>$d_2$</td>
<td>Coefficient of the strain rate effect on the failure strain</td>
<td>-</td>
<td>0.203</td>
</tr>
<tr>
<td>11</td>
<td>$L_0 \tilde{\varepsilon}_f^pl$</td>
<td>Failure displacement</td>
<td>-</td>
<td>0.005</td>
</tr>
</tbody>
</table>
3.6.2.3. Projectile material parameters

We use the same parameters that we presented in the flexion test to simulate the steel behaviour of projectile in the punching test.

3.6.2.4. Results
Projectile results

Figure 38 shows the evolution of the displacement of a point located on the rear part of the missile. The maximum displacement is about 110 cm at the end of the simulation (25 ms). The initial projectile velocity is 135 m/s.

![Displacement time history of the rear of the missile during impact](image)

Figure 38 computed displacement versus time of the rear part of the projectile

Figure 39 shows the evolution of the computed projectile velocity versus the time. The velocity is measured on a point located on the rear part of the projectile. This velocity is strongly decreasing in the first 5 ms. In the last 20 ms the projectile deceleration becomes much lower. The computed residual velocity of projectile is 38 m/s, which is very close to the measured one.

![Velocity time history of the rear of the missile during impact](image)

Figure 39 evolution of the computed projectile velocity versus time

Figure 40 shows the evolution of the computed force versus the time. This force is exerted by the projectile on the reinforced concrete target during the impact.
The maximum force is around $2.0 \times 10^6$ N at 1.3 ms after the beginning of the impact. After time $t = 3.5$ ms, the impact force becomes and remains very low.

Figure 41 shows the evolution of the force impulse exerted by the projectile during the crash on the reinforced concrete plate. This curve is the integral of the previous curve shown on Figure 40.

Results obtained on the reinforced concrete plate
Figure 42 shows the computed deformed shape of the target and the projectile position at the end of the numerical simulation ($t = 25$ ms). The contours plotted on the reinforced concrete target give the maximum value reached by the principal strains. A comparison with experimental results are realized and also presented.
The reinforced concrete plate is perforated by the projectile and its residual velocity is closed to 38 m/s. The lack of transverse rebars between the two reinforcement layers is responsible of the propagation of a large crack located along the back face reinforcement layer. The diameter of the numerical spalled area (rear face) is about 0.52 m.

Figure 43 shows the displacement histories of the 5 points located on the front face of the concrete plate. The upper part of this figure shows the point locations. The maximum displacement reaches 22 mm at point W2.
Displacement at the front of the slab: W1
Displacement at the front of the slab: W2

Displacement at the front of the slab: W3
In the comparison with experimental measured displacements, we observed that the computed displacements simulated by modified PRM coupled model are more important, especially in first period of measured displacement. But we also observed that the measured displacements in this period are much closer to those of the second period of measure.

Left part of Figure 44 shows the positions and the orientations of the strain gages bonded on the reinforcement. Strains are plotted on Figure 51 to Figure 51. Table gives the coordinates
of these strain gages. The precise location of the strain gage on the rebar is unknown [Figure 44]. In some particular situations, significant differences can be observed between each position. The plotted curves are an average of the four possible locations.

Several positions are possible for strain gages on the rebar.

The plotted curves are an average of the 4 strain gage locations.

![Figure 44 strain gage location and orientation.](image)

Table 8 strain gage location and orientation

<table>
<thead>
<tr>
<th>Position relative to the center</th>
<th>D1</th>
<th>D2</th>
<th>D3</th>
<th>D4</th>
<th>D5</th>
<th>D6</th>
<th>D7</th>
<th>D8</th>
</tr>
</thead>
<tbody>
<tr>
<td>x (mm)</td>
<td>45</td>
<td>45</td>
<td>405</td>
<td>405</td>
<td>80</td>
<td>440</td>
<td>270</td>
<td>440</td>
</tr>
<tr>
<td>y (mm)</td>
<td>470</td>
<td>270</td>
<td>470</td>
<td>110</td>
<td>405</td>
<td>405</td>
<td>45</td>
<td>45</td>
</tr>
<tr>
<td>z (mm)</td>
<td>-85</td>
<td>-85</td>
<td>-85</td>
<td>-85</td>
<td>-95</td>
<td>-95</td>
<td>-95</td>
<td>-95</td>
</tr>
<tr>
<td>direction</td>
<td>y</td>
<td>y</td>
<td>y</td>
<td>y</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>position</td>
<td>rearrearrearrearrearrearrearrear</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

![Figure 45 strain gage history at D1](image)

Strain of the bending reinforcement: D1
Figure 46 strain gage history at D3

Figure 47 strain gage history at D4
Figure 48 strain gage history at D5

Figure 49 strain gage history at D6
4. Conclusion

This paper has presented the main features of the PRM model and its proposed improvement. The modified PRM model takes into account the influence of deviatoric stress on the volumetric behavior. The influence of the saturation ratio on the behavior of concrete under triaxial compression is also modified thanks to new approach. Therefore, the consolidation point is updated. The modification of coupling variable is also realized to
generate a better continuity between the two model responses. These changes improve significantly the prediction of concrete behavior under triaxial compression with the PRM coupled model. The modified PRM coupled model was then used to simulate the flexion test and the punching test of the IRIS 2012 benchmark of the Nuclear Energy Agency (NEA) of OECD. Compared to experimental results of two tests, we validated the improvement of this model. The advantage of this model is that we can use the same parameters of concrete behavior to simulate different types of dynamic solicitation as impact case. At low confinement, the model allows reproducing the degradation mechanism and cracking of concrete plate, while it can predict the pore collapse phenomena of concrete at high confinement. In the two tests that we simulated, the maximum pressure is generally not high, about 150 MPa in shock domain and 100 MPa elsewhere all over. Only some elements in contact with projectile present high pressure (300 MPa). The erosion option was likewise employed in both two tests. It is worth noting that the disappearance of elements can lead the loss of element mass. The later will lead to lose momentum quantity, which can influence the calculation performance.

Reference:


1. Introduction

After the success of the workshop IRIS_2010 organized in Paris – France in 2010, and the workshop IRIS_2012 organized in Ottawa – Canada in 2012, we continue improving our constitutive model of concrete behaviour and taking into account all recommendations issue from scientific committee in the simulation of two impact tests.

This report begins by the presentation and analyze of triaxial tests performed up to 100 MPa of confinement. Some results of test carried out at very high confinement (600 MPa) will also be presented for clarifying the effect of saturation ratio. These tests were simulated with the coupled damage plasticity model PRM. Necessary changes of this model to obtain a better prediction will be presented and the influence of these changes will be highlighted by comparing the two versions of the model and experimental results.

Then the new numerical results of the flexion test which produces the flexion mode of concrete plate and the punching test which produces the scabbing and spalling of concrete plate by the perforation of projectile will be compared to the experimental results.

2. Triaxial Tests

2.1. Experimental device

The GIGA press (figure 1) was designed to study the behavior of concrete under high confinement in a partnership between the 3SR laboratory and the CEA Gramat. It allows testing cylindrical concrete specimens under triaxial compression at a maximum confining pressure of 0.85 GPa and a maximum axial stress of 2.3 GPa [1-6].

![GIGA device and specimen dimensions](image)

2.2. Composition of concrete

The studied high performance concrete was used in the benchmark project “Improving the Robustness of Assessment Methodologies for Structures Impacted by Missiles (IRIS)” of the...
Nuclear Energy Agency (NEA) of OECD [8]. Concrete samples were manufactured by VTT Finnish laboratory for the IRIS project. The composition and properties of VTT concrete are given in Table 1.

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Gravel (0.5/8) (kg/m³)</td>
<td>925.9</td>
</tr>
<tr>
<td>Sand (kg/m³)</td>
<td>646.1</td>
</tr>
<tr>
<td>Water (kg/m³)</td>
<td>215</td>
</tr>
<tr>
<td>Cement (CEM II B 42.5) (kg/m³)</td>
<td>489</td>
</tr>
<tr>
<td>Fly ash (kg/m³)</td>
<td>88</td>
</tr>
<tr>
<td>Water-reducing agent (kg/m³)</td>
<td>6.33</td>
</tr>
<tr>
<td>Cement paste volume (m³)</td>
<td>0.375</td>
</tr>
<tr>
<td>Density (kg/m³)</td>
<td>2370</td>
</tr>
<tr>
<td>Compressive strength (MPa)</td>
<td>67</td>
</tr>
<tr>
<td>Porosity accessible to water (%)</td>
<td>12</td>
</tr>
<tr>
<td>Cement paste volume (m³)</td>
<td>0.375</td>
</tr>
<tr>
<td>E/C ratio</td>
<td>0.44</td>
</tr>
</tbody>
</table>

2.3. Experimental results

The concrete used in this study is representative of the containment of a nuclear power plant. Its strengths under unconfined compression and tension are about 67 MPa and 4.5 MPa respectively. The samples were tested under triaxial confining pressure varying from 15 MPa to 100 MPa. The porosity and the degree of saturation were measured prior to testing. The degree of saturation of concrete in the first series of tests is about 60%.

The performed triaxial tests consist in applying a hydrostatic pressure all around the specimen at 0.5 MPa/s up to pressure value $p_{conf}$. A constant displacement rate of 14 μm/s of the axial jack and a constant confining pressure $p_{conf}$ on the lateral face are then imposed.

Figure 2 shows the evolutions of the axial stress in function of the axial and circumferential strains for different confining pressures $p_{conf}$, the circumferential strain is the average measure of two gauges. The axial strain is obtained by means of an axial gauge and compared to the strain given by a LVDT sensor. These two measurements of the axial strain gave similar results indicating that the samples deform homogeneously. Figure 3 shows the volumetric behavior of concrete during the triaxial tests.

The analysis of tests highlights that stiffness and strength increase with confinement. This phenomenon is explained by the irreversible closure of porosity (compaction) with the mean stress increase. It is worth noting that with a confinement pressure less than 50 MPa, there is a stress peak in the axial behavior, while this phenomenon is not observed for the test at 100 MPa of confinement. At this confinement level, the reached limit state corresponds to a transition from contraction to dilatancy with no softening (figure 3).
2.4. Influence of saturation ratio on concrete behavior at moderate and high confining pressures

A second triaxial test at 50 MPa of confinement has been performed on a saturated concrete specimen to study the influence of the saturation ratio (Sr). The procedure for testing saturated samples is described in [4]. The axial strain is measured by the LVDT sensor only. Figure 4 shows a comparison between the evolution of the axial stress as a function of axial strain obtained in this second test (Sr = 100%) and the one obtained on a wet concrete (Sr = 60%) at 50 MPa of confinement. The maximum axial stress is about 240 MPa in the two tests. This result is in agreement with the one obtained on a standard concrete [4]. Thus, the saturation ratio seems to have a low influence on the triaxial behavior of the tested high performance concrete at moderate confining pressures.

Figure 4. Comparison of axial behavior at 50 MPa of confining pressure for two different saturation ratios of concrete specimens

Because concrete may be submitted to very high triaxial stresses in case of impact, some additional triaxial tests were performed at high confining pressure.

Figure 5 shows the comparison between two hydrostatic tests at high confinement with different saturation ratios.
The two curves on figure 5 are confounded at a mean stress lower than 100 MPa, this zone corresponds to the elastic behavior of VTT concrete. Beyond this zone, the closure of porosity of concrete begins. Due to the important volume of paste cement (Table 1), the influence of creep may be important and strongly dependent on the saturation ratio [9]. That explains why the volumetric deformation of saturated concrete is higher than one of dried concrete when the porosity of concrete begins to be enclosed. But at high mean stress, water is compressed and because the compressibility of water is higher than the one of air, the volumetric deformation of saturated concrete is lower than the one of dried concrete (figure 5) at high mean stress.

2.5. Limit state of VTT concrete under triaxial compression

The material limit state is defined as the maximum volumetric strain reached during a test; it corresponds to a transition from a contraction behavior to a dilatancy one. At moderate confinement, this transition also corresponds to the peak stress. Figure 6 shows the limit states in the deviatoric stress / mean stress plane for the various tests described earlier.

On figure 6, it can be noted that, for a given mean stress, the maximum deviatoric stress reached during the test strongly depends on the saturation ratio of the concrete specimen. The presence of free water limits the admissible shear stress of concrete under confinement. Figure 7 shows a zoom of the limit state curve of wet concrete at moderate confinement.
For a moderate confining pressure (lower than 50MPa), the influence of free water on concrete behavior seems to be low, whereas the influence of the saturation degree is significant for a confinement pressure of the order of 500MPa. This difference may have an important effect on the response of a concrete structure submitted to an impact and should be taken into account in simulations.

3. Material model description

3.1. PRM coupled model

We used the PRM coupled model [1] (PONTIROLI-ROUQUAND-MAZAR) to simulate the concrete behaviour under impact. This constitutive model of concrete behaviour is a coupling between damage model and plastic model. This damage model is developed by C. PONTIROLI from J. MAZAR’s model [3], it allows reproducing degradation mechanism and cracking of concrete at low confinement. In this model, two damage variables (Dt for tensile stress and Dc for compression stress) are used. The unilateral characteristic of concrete is taken into account to simulate the concrete behaviour under seismic solicitation [2]. It means that the stiffness of concrete structure can be restored when it state change from tensile stress to compression stress. The illustration of this model is presented in Figure 8:

The damage model is very efficient to simulate the behavior of concrete for unconfined or low confined cyclic loading (Rouquand 2005) [5]. For very high dynamic loads leading to a
higher pressure level, an elastic plastic model is more appropriate. For example, the impact of a projectile striking a concrete plate at 300 m/s induces local pressures near the projectile nozzle of several hundred MPa. The previous damage model cannot simulate the pore collapse phenomena rising at this pressure level. It also cannot model the shear plastic strain occurring in this pressure range. To overcome these limitations, the elastic and plastic model proposed by Krieg (1978) [4] has been chosen to simulate this kind of problem. From this simple elastic and plastic model, A. Rouquand made two improvements [6]. The first improvement has been introduced in order to simulate the nonlinear elastic behavior encountered during an unloading and reloading cycle under a high pressure level. A second improvement has been made to account for the water content effects introducing an effective stress theory as described by C. Mariotti (2002) [7]. This effect induces change on the pressure volume curve and on the shear plastic stress limit.

The coupling of two models ensures a perfect continuity between the two model responses. The predicted stresses correspond to the damage model response if the maximum pressure is too low to start the pore collapse phenomena or if the shear stress is too low to reach the shear yield stress. If not, the plastic model is activated and pilots the evolutions until the extensions sufficiently increase to lead to a damage failure.

3.2. Improvement of PRM coupled model

Although the model coupled PRM allows obtaining a good prediction of concrete behavior under various load paths, some shortcomings exist and this study aims at fixing them.

3.2.1. Influence of the deviatoric stress into the volumetric behaviour

The plasticity model assumes that inelastic volumetric and shear strains are obtained independently. The volumetric strain ($\varepsilon_v$) is assumed to depend on the mean stress ($\sigma_m$) only and the strain deviator tensor is obtained by means of a perfectly plastic model.

One of the shortcomings of the present PRM coupled model is that it does not take into account the effect of the deviatoric stress $q$ on the volumetric behavior of the concrete. The present model assumes that the compaction curve, i.e. the volumetric strain ($\varepsilon_v$) vs. the mean stress ($\sigma_m$), as material data independent on the load path. Figure 3 shows that the inelastic volumetric strain depends on both $q$ and $\sigma_m$. It is then necessary to include the influence of $q$ into the compaction curve of the material ($\varepsilon_v = \text{function} (\sigma_m, q)$).
To improve the PRM model, the original idea of two models of plasticity to calculate the inelastic strains is conserved. According to the test results [8-13], it is assumed that the maximum compaction is obtained under oedometric loading path, i.e. in uniaxial strain condition. The compaction curve of an oedometric test is then added as a second input data. The interest is on the one hand, this data is easily accessible to measurement and, on the other hand, that the oedometric test is the one which maximizes the compaction of concrete because the dilatancy is prevented.

The construction of the modified model is based on the following assumptions:

The curve of volumetric behavior of concrete is not supposed to be bijective. It is instead assumed bounded by the hydrostatic curve (Figure 10- opaque upper curve) and the oedometric curve (Figure 10 - dotted lower curve).

The variation of the mean stress $\sigma_m$, between the bounded curves is given by:

$$d\sigma_m = \alpha d\varepsilon_v$$  \hspace{1cm} (1)

With:

$$\alpha = \alpha_H + (\alpha_O - \alpha_H)\frac{dq}{d\sigma_m} \cdot \frac{1}{\alpha_H}$$  \hspace{1cm} (2)

Where (figure 8):

$\alpha_H = d\sigma_m/d\varepsilon_v$ for a hydrostatic path ;  
$\alpha_O = (d\sigma_m/d\varepsilon_v)O$ for un oedometric path ;

$\frac{dq}{d\sigma_m} = \text{load path direction} ;$

$(dq/d\sigma_m)_O = \text{oedometric load path direction}.$

With formulae (1) and (2), the volumetric strain $\varepsilon_v$ depends on both the mean stress $\sigma_m$ and the deviatoric shear stress $q$, the compaction is then increased in presence of shear compared with volumetric strain obtained with the initial model.

![Figure 10. Hydrostatic behavior, oedometric behavior and consolidated behavior of concrete: mean stress in function of volumetric strain.](image)

### 3.2.2. Influence of the water saturation ratio into the volumetric behavior

Two kinds of approaches exist to characterize the behavior of a porous medium scale homogenized according to its properties at the microscopic level. The “mixing law”
approaches take into account, at the microscopic level, the interaction between the two phases (liquid + solid) by means of simple rheological models for each phase associated in series or in parallel. Poromechanical approaches [14] assume that the concepts of mechanics in continuous media are valid at the macroscopic scale when the two phases (liquid + solid) overlap.

In the present PRM coupled model, the concept of effective stress is used to take into account the presence of water in confined concrete using the first approach. The drawback of this approach is that the behavior of the material becomes elastic after reaching the consolidation point (closure of all open pores), which is not observed experimentally. In the improved model, the second poro-mechanical approach is used to take into account the effect of free water.

The studied porous medium is assumed to be composed of a solid phase (skeleton) and a fluid phase occupying the voids [14]. The concept of the effective stress is introduced to separate the fluid pressure in the calculation of the total pressure

\[ \sigma_{\text{tot}} = \sigma_{M} + bp \] (3)

With \( \sigma_{\text{tot}} \) the total stress, \( \sigma_{M} \) transmitted by the matrix at a macroscopic scale, \( p \) the pore pressure, and \( b \) the Biot coefficient which depends on the nature of the porosity.

The calculation of pore pressure \( p \) is based on the Mie Gruneisen equation of state:

\[ P = \frac{\mu C_{0}^{2} (\phi_{0} - \phi)}{1 - s (\phi_{0} - \phi)} \left[ 1 - \frac{\Gamma_{0} (\phi_{0} - \phi)}{2} \right] + \Gamma_{0} \rho_{0} E_{M} \] (4)

Where \( C_{0} \) is the sound celerity (\( C_{0} = 1500 \text{m/s} \)), \( \phi_{0} \) is the density (\( \phi_{0} = 1000 \text{ kg/m}^{3} \) for water), \( s \) and \( \Gamma_{0} \) are two Mie Gruneisen coefficients (\( s = 1.75 \) and \( \Gamma_{0} = 0.28 \) for water). \( E_{M} \) is the internal energy per unit mass. This energy is considered negligible for water temperature and ambient pressure.

\( \sigma_{M} \) and \( b \) can be obtained by the following formulae [14]:

\[ \sigma_{M} = K_{0} \varepsilon_{V} \] (5)

\[ b = 1 - \frac{K_{0}}{K_{S}} \] (6)

\( K_{0} \) is the modulus of the material drained \( \varepsilon_{V} \) is the volumetric strain at homogenized scale, \( K_{S} \) the compressibility modulus of the skeleton. From equation (6), in the particular case where \( K_{0} \ll K_{S} \), \( b \) is close to 1, which simplifies the equation (3) and becomes \( \sigma_{\text{tot}} = \sigma_{M} + p \) (Terzaghi formula). In contrast, when \( K_{0} \approx K_{S} \) (dry concrete case), \( b \) tends to 0. In the end, thanks to the technical of homogenization of the drained porous medium [14] the ratio \( K_{0}/K_{S} \) can be estimated as follows:

\[ \frac{K_{0}}{K_{S}} = (1 - \phi)^{2} \] (7)

Where \( \phi \) is the porosity of the porous medium at the current state.
With this new hypothesis, as the material reaches the point of consolidation (void pores are closed), the volumetric behavior remains nonlinear due to the fact that the voids filled with water continue to be compressed under compaction. Another advantage of this improvement is the unique point of consolidation instead of two points in the original model [Figure 11].

3.2.3. Improvement of coupling variable

When we used the coupling variable proposed in PRM coupled model to simulate concrete behavior of impact test, it presented an effect of checkerboard in the damage simulation. It means that this method couldn’t generate a homogeneity or continuity of stress in the impact zone. To remedy this shortcoming, we proposed to change this coupling variable formula by the other one that is formulated using a dual criterion and calculated as a function of the three stress invariants $I_1$, $J_2$, and $\theta$ [15]. Three modes of failure can be identified. For the pure cracking mode, the maximum principal stress must be positive, that is, $\sigma_1 > 0$, and it is assumed that the maximum principal strain is compressive, that is, $\varepsilon_1 < 0$, for the pure crushing mode. A crushing coefficient is subsequently defined as follows by combining these two conditions and using the stress invariants $I_1$, $J_2$, and $\theta$. $I_1$ is the first invariant of the stress tensor, $J_2$ is the second invariant of the stress tensor, and $\theta$ is the angle of similarity that is a function of the third invariant $J_3$ of the stress tensor.

$$\alpha = -\frac{I_1}{3.464 \sqrt{J_2} \cos \theta} \quad \text{for} \quad 0^\circ \leq \theta \leq 60^\circ$$

(8)

The failure modes can then be identified as follows:
- Pure cracking = $\alpha < 1$
- Pure crushing = $\alpha > (1 + \nu)/(1 - 2\nu)$
- Mixed failure mode involving cracking and crushing = $1 \leq \alpha \leq (1 + \nu)/(1 - 2\nu)$

In the previously mentioned relationships, $\nu$ is Poisson’s ratio. Figure 12 shows the failure zones in the octahedral shear and normal stress plane.
3.2.4. Identification of PRM coupled model

In our simulations, we want to use only one constitutive model of concrete behaviour to simulate different types of dynamic solicitation as impact case. All parameters identified here are used in two impact tests that we will present in the next part of this report.

Table gives parameters of the PRM damage model. Table 4 gives parameters of the PRM crash model. Yellow background rows correspond to measured material data. Other values are deduced from the previous one or are representative of a standard concrete. PRM compressive strength parameter refers to cylindrical concrete specimens.

Table 2: PRM material data

<table>
<thead>
<tr>
<th>N°</th>
<th>Parameters</th>
<th>Designation</th>
<th>Unit (MKS)</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>$E_0$ (&gt;0)</td>
<td>Young Modulus</td>
<td>N/m$^2$</td>
<td>$2.8 \times 10^{10}$</td>
</tr>
<tr>
<td>2</td>
<td>$\nu_0$ (&gt;0)</td>
<td>Poisson ratio</td>
<td>-</td>
<td>0.1875</td>
</tr>
<tr>
<td>3</td>
<td>$\sigma_t$ (&lt;0)</td>
<td>Tensile strength</td>
<td>N/m$^2$</td>
<td>$4.7 \times 10^6$</td>
</tr>
<tr>
<td>4</td>
<td>$\sigma_c$ (&gt;0)</td>
<td>Compressive strength</td>
<td>N/m$^2$</td>
<td>$-67.0 \times 10^6$</td>
</tr>
<tr>
<td>5</td>
<td>$\varepsilon_{\text{erosion}}$ (&gt;0)</td>
<td>maximum strain before erosion</td>
<td>-</td>
<td>0.23</td>
</tr>
<tr>
<td>6</td>
<td>$\sigma_{\text{final}}$ (&gt;0)</td>
<td>Residual tensile stress</td>
<td>N/m$^2$</td>
<td>0.0</td>
</tr>
<tr>
<td>7</td>
<td>$\beta$ (&gt; 0)</td>
<td>Mazars coefficient</td>
<td>-</td>
<td>1-1.05</td>
</tr>
<tr>
<td>8</td>
<td>$\sigma_{\text{f0}}$ (&lt;0)</td>
<td>Initial closure stress</td>
<td>N/m$^2$</td>
<td>$-4.7 \times 10^6$</td>
</tr>
<tr>
<td>9</td>
<td>$\sigma_{\text{fc}}$ (&gt;0)</td>
<td>Focus point of unloading in compression</td>
<td>N/m$^2$</td>
<td>$67.0 \times 10^6$</td>
</tr>
<tr>
<td>11</td>
<td>$G_f$ (&gt;0)</td>
<td>Fracture energy</td>
<td>N/m</td>
<td>120.0</td>
</tr>
<tr>
<td>13</td>
<td>$a_t$ (&gt;0)</td>
<td>First tensile strain rate coefficient</td>
<td>-</td>
<td>0.823</td>
</tr>
</tbody>
</table>
Iris 2012 International Benchmark, Impact of projectiles on a reinforced concrete plate

<table>
<thead>
<tr>
<th></th>
<th></th>
<th>Second tensile strain rate coefficient</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>14</td>
<td>b_t (&gt;0)</td>
<td>-</td>
<td>0.329</td>
</tr>
<tr>
<td>15</td>
<td>a_c (&gt;0)</td>
<td>First compression strain rate coefficient</td>
<td>-</td>
</tr>
<tr>
<td>16</td>
<td>b_c (&gt;0)</td>
<td>Second compression strain rate coefficient</td>
<td>-</td>
</tr>
<tr>
<td>17</td>
<td>β_1 (&gt;0)</td>
<td>Damping coefficient in the elastic domain</td>
<td>-</td>
</tr>
<tr>
<td>18</td>
<td>β_2 (&gt;0)</td>
<td>2nd damping coefficient induced by damage</td>
<td>-</td>
</tr>
<tr>
<td>19</td>
<td>ρ_0 (&gt;0)</td>
<td>Density Kg/m^3</td>
<td>2272.0</td>
</tr>
</tbody>
</table>

Table 3: Material data of elastic plastic model

Plastic threshold

<table>
<thead>
<tr>
<th>a_0 (Pa²)</th>
<th>a_1 (Pa)</th>
<th>a_2</th>
<th>P_{cut elas tens} (Pa)</th>
<th>q_{max} (Pa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>7.0E+14</td>
<td>2.0E+08</td>
<td>1.0E+00</td>
<td>1.000E+01</td>
<td>6.000E+09</td>
</tr>
</tbody>
</table>

Volumetric deformation- mean stress curve of hydrostatic test (24 points)

<table>
<thead>
<tr>
<th>p_v (Pa)</th>
<th>ε_v1</th>
<th>p_v2(Pa)</th>
<th>ε_v2</th>
<th>p_v3(Pa)</th>
<th>ε_v3</th>
<th>p_v4(Pa)</th>
<th>ε_v4</th>
</tr>
</thead>
<tbody>
<tr>
<td>7.03E+07</td>
<td>4.7100E-03</td>
<td>1.13E+08</td>
<td>8.0600E-03</td>
<td>1.68E+08</td>
<td>1.3000E-02</td>
<td>2.02E+08</td>
<td>-1.6770E-02</td>
</tr>
<tr>
<td>2.31E+08</td>
<td>2.0800E-02</td>
<td>2.62E+08</td>
<td>2.5840E-02</td>
<td>2.95E+08</td>
<td>3.1490E-02</td>
<td>3.35E+08</td>
<td>-3.9200E-02</td>
</tr>
<tr>
<td>3.75E+08</td>
<td>4.6950E-02</td>
<td>4.20E+08</td>
<td>5.5780E-02</td>
<td>4.64E+08</td>
<td>6.4810E-02</td>
<td>5.07E+08</td>
<td>-7.3050E-02</td>
</tr>
<tr>
<td>5.86E+08</td>
<td>8.6800E-02</td>
<td>6.69E+08</td>
<td>1.0070E-01</td>
<td>7.48E+08</td>
<td>1.1330E-01</td>
<td>8.70E+08</td>
<td>-1.3230E-01</td>
</tr>
<tr>
<td>9.76E+08</td>
<td>1.4660E-01</td>
<td>1.12E+09</td>
<td>1.6020E-02</td>
<td>1.31E+09</td>
<td>1.7540E-01</td>
<td>1.55E+09</td>
<td>-1.9070E-01</td>
</tr>
<tr>
<td>1.79E+09</td>
<td>2.0100E-01</td>
<td>2.06E+09</td>
<td>2.1050E-01</td>
<td>2.33E+09</td>
<td>2.1970E-01</td>
<td>2.6589E+09</td>
<td>-2.2980E-01</td>
</tr>
</tbody>
</table>

Volumetric deformation- mean stress curve of oedometric test (24 points)

<table>
<thead>
<tr>
<th>p_v (Pa)</th>
<th>ε_v1</th>
<th>p_v2(Pa)</th>
<th>ε_v2</th>
<th>p_v3(Pa)</th>
<th>ε_v3</th>
<th>p_v4(Pa)</th>
<th>ε_v4</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.0000E+07</td>
<td>1.3400E-03</td>
<td>4.3E+07</td>
<td>4.1300E-03</td>
<td>5.70E+07</td>
<td>8.8800E-03</td>
<td>7.37E+07</td>
<td>1.4670E-02</td>
</tr>
</tbody>
</table>
### Iris 2012 International Benchmark, Impact of projectiles on a reinforced concrete plate

<table>
<thead>
<tr>
<th>( p_5 ) (Pa)</th>
<th>( \varepsilon_{v5} )</th>
<th>( p_6 ) (Pa)</th>
<th>( \varepsilon_{v6} )</th>
<th>( p_7 ) (Pa)</th>
<th>( \varepsilon_{v7} )</th>
<th>( p_8 ) (Pa)</th>
<th>( \varepsilon_{v8} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>9.42E+07</td>
<td></td>
<td>1.13E+08</td>
<td></td>
<td>1.32E+08</td>
<td></td>
<td>1.54E+08</td>
<td></td>
</tr>
<tr>
<td>5.35E+07</td>
<td></td>
<td>1.81E+08</td>
<td></td>
<td>2.13E+08</td>
<td></td>
<td>2.49E+08</td>
<td></td>
</tr>
<tr>
<td>7.32E+08</td>
<td></td>
<td>9.730E-02</td>
<td></td>
<td>1.03E+08</td>
<td></td>
<td>1.25E+08</td>
<td></td>
</tr>
<tr>
<td>1.61E+09</td>
<td></td>
<td>1.93E+09</td>
<td></td>
<td>2.05E+09</td>
<td></td>
<td>2.29E+09</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>( q_01 ) (Pa)</th>
<th>( q_02 ) (Pa)</th>
<th>( q_03 ) (Pa)</th>
<th>( q_04 ) (Pa)</th>
<th>( q_05 ) (Pa)</th>
<th>( q_06 ) (Pa)</th>
<th>( q_07 ) (Pa)</th>
<th>( q_08 ) (Pa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.60E+07</td>
<td></td>
<td>4.432E+07</td>
<td></td>
<td>5.761E+07</td>
<td></td>
<td>6.918E+07</td>
<td></td>
</tr>
<tr>
<td>1.572E+08</td>
<td></td>
<td>1.082E+08</td>
<td></td>
<td>2.346E+08</td>
<td></td>
<td>3.298E+08</td>
<td></td>
</tr>
<tr>
<td>1.61E+09</td>
<td></td>
<td>1.93E+09</td>
<td></td>
<td>2.05E+09</td>
<td></td>
<td>2.29E+09</td>
<td></td>
</tr>
</tbody>
</table>

### Oedometric \( q_0 \) curve in relation with 24 points of mean stress \( p \) (24 points)

<table>
<thead>
<tr>
<th>( q_01 ) (Pa)</th>
<th>( q_02 ) (Pa)</th>
<th>( q_03 ) (Pa)</th>
<th>( q_04 ) (Pa)</th>
<th>( q_05 ) (Pa)</th>
<th>( q_06 ) (Pa)</th>
<th>( q_07 ) (Pa)</th>
<th>( q_08 ) (Pa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.60E+07</td>
<td></td>
<td>4.432E+07</td>
<td></td>
<td>5.761E+07</td>
<td></td>
<td>6.918E+07</td>
<td></td>
</tr>
<tr>
<td>1.572E+08</td>
<td></td>
<td>1.082E+08</td>
<td></td>
<td>2.346E+08</td>
<td></td>
<td>3.298E+08</td>
<td></td>
</tr>
<tr>
<td>1.61E+09</td>
<td></td>
<td>1.93E+09</td>
<td></td>
<td>2.05E+09</td>
<td></td>
<td>2.29E+09</td>
<td></td>
</tr>
</tbody>
</table>

### Loading stiffness, unloading stiffness, porosity, water content

<table>
<thead>
<tr>
<th>( K_{\text{grain}} ) cons (Pa)</th>
<th>( K_{\text{grain}} ) unload (Pa)</th>
<th>( \text{Porsec} )</th>
<th>( \text{eau Sr} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.80E+10</td>
<td>3.80E+10</td>
<td>0.11E+00</td>
<td>6.00E-01</td>
</tr>
</tbody>
</table>

#### 3.2.5. Comparison between experimental results and simulations of tests.

The simulation results obtained with the original PRM coupled model and the modified model are compared to experimental results on figures 10 to 12.

#### 3.2.5.1. Wet concrete

Figure 13 and Erreur ! Référence non valide pour un signet. show results for a concrete specimen with a saturation degree of 60% and submitted to triaxial compression with confining pressures varying from 15 to 100 MPa. At this saturation ratio and because of a moderate confining pressure, there is not the effect of free water on concrete behavior. The initial PRM coupled model allows a good prediction of the maximum stress but strains are significantly underestimated. Taking into account the influence of the deviatoric stress on the volumetric behavior of concrete significantly improves the prediction of the volumetric strain.
Figure 13. Axial stress vs. axial and circumferential strains: Comparisons of experimental results with simulation results obtained with the initial (PRM-i) and new model (PRM-n) for a wet concrete specimen under moderate confinement.

Figure 14. Mean stress vs. volumetric strain: Comparisons of experimental results with simulation results obtained with the initial (PRM-i) and new model (PRM-n) for a wet concrete specimen under moderate confinement.

3.2.5.2. Saturated concrete

In the final report sent by organization committee after the workshop IRIS_2010, we found that the age of all concrete slabs tested in impact tests were at about 50 days after the cast day. It means that the concrete slabs were nearly saturated when they were tested. So, it is interested to study the saturated concrete behaviour under confinement when the concrete slabs are subjected to the impact.

Figure 15 shows experimental volumetric behaviour of saturated and dry concrete under high confinement. We observed that the saturated concrete presents an important creep in compared to the dry concrete under confinement. To improve this property of concrete in our numerical simulation, we need to take into account the creep behaviour in the model, that we
Iris 2012 International Benchmark, Impact of projectiles on a reinforced concrete plate

don’t try to do and present in this report. The reason is that under impact as punching test, the
depth of concrete slab is small (25cm), the dynamic effect is dominated, and then the creep of
cement isn’t important. In this figure, we a comparison with simulation results obtained by
both the initial and modified PRM coupled model were realized. The initial model considers
an elastic behaviour after consolidation (closure of voids); while the modified model gives a
simulation result closer to the experimental one.

Figure 15. Mean stress vs. volumetric strain: Comparisons of experimental results with simulation results
obtained with the initial (PRM-i) and new model (PRM-n) for saturated and dry concrete under high
confinement.

3.3. Johnson Cook plastic and damage model

The behavior of the metallic parts of the structure, the steel reinforcement and the metallic
projectile tube is simulated using the Johnson Cook dynamic failure model. In this model, the
plastic stress \( \sigma \) is related to the plastic strain \( \varepsilon^{pl} \) to the plastic strain rate \( \dot{\varepsilon}^{pl} \) and to the
damage variable \( D \) via the following expression:

\[
\sigma = (1-D)\left[A + B\left(\varepsilon^{pl}\right)^n\right] \left[1 + C\ln\left(\frac{\varepsilon^{pl}}{\varepsilon_0}\right)\right]
\]  \( (9) \)

where \( \varepsilon_0 \) is a reference strain rate. \( A, B, n \) and \( C \) are material parameters. The Damage \( D \) is also
related to the plastic strain as follow:

\[
D = \min\left(\frac{L_x \max\left(\frac{\sum \Delta \varepsilon_i^{pl} - \varepsilon_i^{pl(0)}}{L_0 \varepsilon_i^{pl}}, 0\right)}{L_0 \varepsilon_i^{pl}}, 1\right)
\]  \( (10) \)

\( \varepsilon^{pl(0)} \) is the plastic threshold. The Damage variable \( D \) becomes to increase when the cumulated
plastic strain \( \sum \Delta \varepsilon_i^{pl} \) becomes greater than the plastic threshold \( \varepsilon_i^{pl(0)} \). The plastic
threshold \( \varepsilon^{pl(0)} \) is given by:
\[ \bar{\varepsilon}^{pl0} = d_1 \left[ 1 + d_2 \ln \left( \frac{\bar{\varepsilon}^{pl}}{\varepsilon_0} \right) \right] \]  

(11)

d_1, and d_2 are material parameters. When d_2 is positive the plastic threshold increases with the plastic strain rate.

L_e is the element characteristic length. This size is defined by the distance between the two nodes forming the element (1D finite element), by the square root of the element surface (2D finite element) or by the cubic root of the finite element volume (3D finite element). L_e \bar{\varepsilon}^{pl}_f is the failure displacement. This failure displacement can be related to the plastic failure strain \( \bar{\varepsilon}^{pl}_f \) using the internal material length \( L_0 \). Figure 16 shows an example of the stress strain curve obtained with the Johnson Cook dynamic failure model. In this example we have the following material parameters: \( A = 5.6 \times 10^8 \), \( B = 2.3 \times 10^8 \), \( n = 0.36 \), \( C = 0.0141 \), \( d_1 = 0.045 \), \( d_2 = 0.203 \), \( L_0 = 0.2 \). In this figure four curves are plotted. They correspond to different constant strain rate values given in the legend.

![Figure 16. Example of a stress strain curve (Johnson Cook dynamic failure model)](image)

**3.4. Simulation of impact test**

**3.4.1. Flexion test**

**3.4.1.1. Choice of finite element mesh**

A 3-D finite element model with solid brick and shell elements is used to simulate the VTT flexural test. The projectile shown on figure 63 is a 254 mm diameter steel tube. Its length is 2.088m including 0.088m for the front and the caped part. A total of 48 x 134 = 6 432 (4 nodes) shell elements are used to mesh the projectile. The total projectile weight is 49.99 kg.
Figure 17. Projectile characteristics

Figure 18 shows the projectile mesh. The shell thickness is 2mm except in the rear part (yellow part) where it is equal to 18.215 mm.

Figure 18. Projectile mesh

The target size is 2.1 m by 2.1 m. The thickness is 15 cm (instead of 25 cm in the punching test). Figure 19 shows the mesh used for the concrete plate (left part) and for the steel ring and the steel reinforcement. The concrete plate is composed of 84 by 84 by 10 = 70 560 “C3D8R” finite elements (8 nodes solid elements with reduced integration). The reinforcement is modelled using 2 nodes beam elements with a circular cross section.
All the nodes along the red line (Figure 19) have their displacements in the z direction set to zero. The metallic frame around the concrete plate (U shape) is modeled using 6,520 shell elements (4 nodes reduced integration shell S8R elements). The thickness of the metallic ring is 12 mm. The frame weight is 314.4 kg.

The reinforcement is modeled using $84 \times 38 \times 2 \times 2 = 12,768$ beam elements and the rebar diameter is 6 mm. The space between 2 rebars is 55 mm. The rebar length is 2,035 m. Transverse rebars are also introduced using $28 \times 28 \times 6 = 4,704$ beam elements. The transverse rebar diameter is also 6 mm. The distance between two transverse rebar is 75 mm along the 2 orthogonal directions. The concrete cover is 15 mm.

Table 4 reinforcement characteristics (VTT flexural test).

<table>
<thead>
<tr>
<th>Target geometry</th>
<th>Modeling</th>
<th>Experimental</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Longitudinal vertical rebars</td>
<td>Front face</td>
<td>Rear face</td>
<td>Front face</td>
</tr>
<tr>
<td>Diameter of rebars</td>
<td>6</td>
<td>6</td>
<td>6</td>
</tr>
<tr>
<td>Number of vertical rebars</td>
<td>38</td>
<td>38</td>
<td></td>
</tr>
<tr>
<td>Concrete cover</td>
<td>20</td>
<td>20</td>
<td>20</td>
</tr>
<tr>
<td>Longitudinal horizontal rebars</td>
<td>Front face</td>
<td>Rear face</td>
<td>Front face</td>
</tr>
<tr>
<td>Diameter of rebars</td>
<td>6</td>
<td>6</td>
<td>6</td>
</tr>
<tr>
<td>Number of horizontal rebars</td>
<td>38</td>
<td>38</td>
<td></td>
</tr>
<tr>
<td>Concrete cover</td>
<td>20</td>
<td>20</td>
<td>20</td>
</tr>
<tr>
<td>Transverse rebars</td>
<td>Modeling</td>
<td>Experimental</td>
<td>Unit</td>
</tr>
<tr>
<td>Diameter of rebars</td>
<td></td>
<td>6</td>
<td>6</td>
</tr>
<tr>
<td>Number of rebars / m²</td>
<td>176</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Density</td>
<td>50.0</td>
<td>50.0</td>
<td>cm²/m²</td>
</tr>
</tbody>
</table>

3.4.1.2. Reinforcement material parameters

The Johnson Cook dynamic failure model presented in chapter 1 is used. Table 11 gives
the reinforcement (steel) material parameters. Figure 20 gives the corresponding stress strain curves at constant strain rate of respectively $5.0 \times 10^{-5}$, $2.0 \times 10^{-1}$, $2.0 \times 10^{0}$, $8.5 \times 10^{0}$. These curves correspond to $L_0 = 0.2$. These material parameters are deduced from the experimental data shown on Figure 21.

Table 5 reinforcement material data of the Johnson Cook dynamic failure model

<table>
<thead>
<tr>
<th>N°</th>
<th>Parameters</th>
<th>Designation</th>
<th>Unit</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>$E_0$</td>
<td>Young Modulus</td>
<td>N/m²</td>
<td>$2.0 \times 10^{11}$</td>
</tr>
<tr>
<td>2</td>
<td>$\nu_0$</td>
<td>Poisson ratio</td>
<td>-</td>
<td>0.3</td>
</tr>
<tr>
<td>3</td>
<td>A</td>
<td>Initial yield stress</td>
<td>N/m²</td>
<td>$6.0 \times 10^{8}$</td>
</tr>
<tr>
<td>4</td>
<td>B</td>
<td>Hardening yield stress</td>
<td>N/m²</td>
<td>$1.53 \times 10^{8}$</td>
</tr>
<tr>
<td>5</td>
<td>n</td>
<td>Johnson Cook exponent</td>
<td>-</td>
<td>0.36</td>
</tr>
<tr>
<td>6</td>
<td>C</td>
<td>Strain rate effect coefficient</td>
<td>-</td>
<td>0.0141</td>
</tr>
<tr>
<td>7</td>
<td>$\dot{\varepsilon}_0$</td>
<td>Reference strain rate</td>
<td>-</td>
<td>$5.0 \times 10^{-3}$</td>
</tr>
<tr>
<td>8</td>
<td>$d_1$</td>
<td>Plastic strain at damage initiation</td>
<td>-</td>
<td>0.045</td>
</tr>
<tr>
<td>9</td>
<td>$d_2$</td>
<td>Coefficient of the strain rate effect on the failure strain</td>
<td>-</td>
<td>0.203</td>
</tr>
<tr>
<td>11</td>
<td>$L_0\varepsilon_f^{pl}$</td>
<td>Failure displacement</td>
<td>-</td>
<td>0.004</td>
</tr>
</tbody>
</table>

Figure 20. Stress strain curve for the Johnson Cook dynamic failure model (reinforcement of the concrete slab, VTT flexural test).
3.4.1.3. Projectile material parameters

The Johnson Cook dynamic failure model presented in chapter 1 is also used. Table gives the material parameters of the steel used in the projectile. Figure 22 gives the corresponding stress strain curves at constant strain rate of respectively $5.0 \times 10^{-5}$, $5 \times 10^{-2}$, $5 \times 10^{-1}$, $1 \times 10^{0}$. These curves correspond to $L_0 = 0.2$. These material parameters are deduced from the experimental data shown on Figure 23. This experimental stress strain curve is supposed to be obtained at a very low strain rate of $5 \times 10^{-5}$.

Table 6 projectile material data of the Johnson Cook dynamic failure model

<table>
<thead>
<tr>
<th>No</th>
<th>Parameters</th>
<th>Designation</th>
<th>Unit</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>$E_0$</td>
<td>Young Modulus</td>
<td>N/m²</td>
<td>$2.0 \times 10^{11}$</td>
</tr>
<tr>
<td>2</td>
<td>$\nu_0$</td>
<td>Poisson ratio</td>
<td>-</td>
<td>0.3</td>
</tr>
<tr>
<td>3</td>
<td>A</td>
<td>Initial yield stress</td>
<td>N/m²</td>
<td>$2.7 \times 10^{8}$</td>
</tr>
<tr>
<td>4</td>
<td>B</td>
<td>Hardening yield stress</td>
<td>N/m²</td>
<td>$6.66 \times 10^{8}$</td>
</tr>
<tr>
<td>5</td>
<td>n</td>
<td>Johnson Cook exponent</td>
<td>-</td>
<td>0.384</td>
</tr>
<tr>
<td>6</td>
<td>C</td>
<td>Strain rate effect coefficient</td>
<td>-</td>
<td>0.025</td>
</tr>
<tr>
<td>7</td>
<td>$\dot{\varepsilon}_0$</td>
<td>Reference strain rate</td>
<td>-</td>
<td>$5.0 \times 10^{-3}$</td>
</tr>
<tr>
<td>8</td>
<td>$d_1$</td>
<td>Plastic strain at damage initiation</td>
<td>-</td>
<td>0.18</td>
</tr>
<tr>
<td>9</td>
<td>$d_2$</td>
<td>Coefficient of the strain rate effect on the failure strain</td>
<td>-</td>
<td>0.24</td>
</tr>
<tr>
<td>11</td>
<td>$L_0\varepsilon_f^{pl}$</td>
<td>Failure displacement</td>
<td>-</td>
<td>0.005</td>
</tr>
</tbody>
</table>
3.4.1.4. Results

3.4.1.4.1. Concrete plate results – comparison with results obtained in workshop 2010

Displacement of concrete plate

In the workshop IRIS_2010, we simulated a quarter of the structural system due its symmetry [Figure 24]. The movement of the concrete plate is prevented at red lines [Figure 24] in movement directions of projectile (U3 = 0) For better understanding the influence of the strain rate effect of steel, we first didn’t take into account the effect of strain rate in both...
projectile and rebars reinforced in concrete plate in this simulation. Only the strain rate of concrete was activated. The original PRM coupled constitutive model was used to simulate this test.

Figure 24. Model of a quarter of structural system in workshop 2010

Figure 25. Computed displacements at points w1, w2, w3, w4 and w5 in workshop 2010
In this simulation, the displacements of 5 points computed on the rear face of the concrete plate were obtained and presented on the Figure 25. Theses computed displacements were very low in comparison with those measured in the test [Figure 26] (less than 5 times). In the next step of simulation, we took into account the strain rate in both the projectile and the rebars. The results obtained in this step were really better those obtained in the first step of simulation [Figure 27] (less than 2 times).
Although the displacements of 5 points computed on concrete plate were improved when we took into account the strain rate of both projectile and rebars in the simulation, we couldn’t reproduce displacements measured on in the flexion test. In the workshop 2012, after improving the PRM coupled model of concrete that we presented above, the results obtained in this simulation approached the results of tests [Figure 28]
Displacement at the rear of the slab: W2

Displacement at the rear of the slab: W3
Figure 28. Comparison between computed displacements and measured displacements at points w1, w2, w3, w4 and w5 – with strain rate in projectile and rebars – modified PRM coupled model _ workshop 2012
The damage obtained on the front face and the rear face of concrete plate simulated by modified PRM coupled model are very similar this one of experimental results.

3.4.1.4.2. Projectile results

Figure 31 shows the projectile shape at time $t = 40$ ms (at the end of the numerical simulation). The computed residual projectile length is 1.029 m in comparison with 1.14 m of the measured residual length of projectile. If the folded part isn’t taken into account, the computed residual projectile length is 0.996 m in comparison with 0.97 m of the measured residual length of projectile [Figure 32]. To obtain a better predict of the computed residual length of projectile, a little change of steel behavior should be realized.
3.4.2. Punching test

3.4.2.1. Finite element mesh

The projectile is a 168 mm diameter steel tube filled with concrete [Figure 33]. Its length is 64 cm including 5 cm for the front and the caped part. In our simulation, 3-D finite elements (8 nodes solid brick elements) are used to mesh the projectile because the concrete material inside the steel tube strongly prevents the possible projectile deformations. A total of 1400 3-D elements are used to mesh the projectile. The total projectile weight is 47 kg.

The reinforced concrete target of 2.1m by 2.1 m by 0.25 m is inserted between two stiff metallic frames as shown on Figure 34. The right part of this figure shows a detailed view of the boundary conditions for the square concrete slab. The concrete is casted inside a metallic ring (yellow part in Figure 34). This metallic ring is meshed using 3-D solid brick elements.
Figure 34. support device of the reinforced concrete slab

The left part of Figure 35 shows a view of the 3D finite element model of the concrete slab. The mesh is composed of 84 by 84 by 10 = 70,560 solid (8 nodes) brick elements. The movement of concrete plate is prevented from the movement direction of projectile by 8 steel rods. The right part of Figure 35 shows the metallic ring around the concrete plate (yellow part). Elements are used for the ring structure that shares the same nodes than the concrete plate. The thickness of the metallic ring is 25 mm and its total mass is 708 kg. The beam elements used to simulate the reinforcement can also be seen on the right part of Figure 35. This reinforcement is composed of a total of 24 x 2 (horizontal + vertical) x 2 (front face and rear face) = 96 rebars. There is no transverse rebar to connect the front face and the rear face layers. The rebar diameter is 10 mm. The rebar space is 90 mm and the concrete cover is 30 mm.

Figure 35. View of the finite element mesh of the reinforced concrete plate
3.4.2.2. Reinforcement material parameters

The Johnson Cook dynamic failure model is used. Table gives the reinforcement (steel) material parameters. Figure 36 gives the corresponding stress strain curves at constant strain rate of respectively $5.0 \times 10^{-5}$, $2.0 \times 10^{-1}$, $2.0 \times 10^{0}$, $8.5 \times 10^{0}$. These curves correspond to $L_0 = 0.2$. These material parameters are deduced from the experimental data shown on Figure 37.

<table>
<thead>
<tr>
<th>N°</th>
<th>Parameters</th>
<th>Designation</th>
<th>Unit</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>$E_0$</td>
<td>Young Modulus</td>
<td>N/m$^2$</td>
<td>$2.0 \times 10^{11}$</td>
</tr>
<tr>
<td>2</td>
<td>$\nu_0$</td>
<td>Poisson ratio</td>
<td>-</td>
<td>0.3</td>
</tr>
<tr>
<td>3</td>
<td>$A$</td>
<td>Initial yield stress</td>
<td>N/m$^2$</td>
<td>$5.8 \times 10^{8}$</td>
</tr>
<tr>
<td>4</td>
<td>$B$</td>
<td>Hardening yield stress</td>
<td>N/m$^2$</td>
<td>$2.3 \times 10^{8}$</td>
</tr>
<tr>
<td>5</td>
<td>$n$</td>
<td>Johnson Cook exponent</td>
<td>-</td>
<td>0.36</td>
</tr>
<tr>
<td>6</td>
<td>$\dot{\varepsilon}_0$</td>
<td>Strain rate effect coefficient</td>
<td>-</td>
<td>0.0141</td>
</tr>
<tr>
<td>7</td>
<td>$\ddot{\varepsilon}_{pl}$</td>
<td>Reference strain rate</td>
<td>-</td>
<td>$5.0 \times 10^{-3}$</td>
</tr>
<tr>
<td>8</td>
<td>$d_1$</td>
<td>Plastic strain at damage initiation</td>
<td>-</td>
<td>0.045</td>
</tr>
<tr>
<td>9</td>
<td>$d_2$</td>
<td>Coefficient of the strain rate effect on the failure strain</td>
<td>-</td>
<td>0.203</td>
</tr>
<tr>
<td>11</td>
<td>$L_0\varepsilon_{f}^{pl}$</td>
<td>Failure displacement</td>
<td>-</td>
<td>0.005</td>
</tr>
</tbody>
</table>

Figure 36. Stress strain curve for the Johnson Cook dynamic failure model (reinforcement of the concrete slab)
3.4.2.3. Projectile material parameters

We use the same parameters that we presented in the flexion test to simulate the steel behaviour of projectile in the punching test.

3.4.2.4. Results

3.4.2.4.1. Projectile results

Figure 38 shows the evolution of the displacement of a point located on the rear part of the missile. The maximum displacement is about 110 cm at the end of the simulation (25 ms). The initial projectile velocity is 135 m/s.

Figure 39 shows the evolution of the computed projectile velocity versus the time. The velocity is measured on a point located on the rear part of the projectile. This velocity is strongly decreasing in the first 5 ms. In the last 20 ms the projectile deceleration becomes much lower. The computed residual velocity of projectile is 38 m/s, which is very closed to

![Figure 38. Computed displacement versus time of the rear part of the projectile](image)

![Figure 39. Displacement time history of the rear of the missile during impact](image)

![Figure 37. Measured static stress strain curve of the reinforcement.](image)
The measured one.

![Velocity time history of the rear of the missile during impact](image)

**Figure 39. Evolution of the computed projectile velocity versus time**

Figure 40 shows the evolution of the computed force versus the time. This force is exerted by the projectile on the reinforced concrete target during the impact.

![Load time history between the missile and the target](image)

**Figure 40. Evolution of the computed projectile impact force**

The maximum force is around $2.0 \times 10^6$ N at 1.3 ms after the beginning of the impact. After time $t = 3.5$ ms, the impact force becomes and remains very low.

Figure 41 shows the evolution of the force impulse exerted by the projectile during the crash on the reinforced concrete plate. This curve is the integral of the previous curve shown on Figure 40.
3.4.2.4.2. *Results obtained on the reinforced concrete plate*

Figure 42 shows the computed deformed shape of the target and the projectile position at the end of the numerical simulation \((t = 25 \text{ ms})\). The contours plotted on the reinforced concrete target give the maximum value reached by the principal strains. A comparison with experimental results are realized and also presented.
The reinforced concrete plate is perforated by the projectile and its residual velocity is closed to 38 m/s. The lack of transverse rebars between the two reinforcement layers is responsible of the propagation of a large crack located along the back face reinforcement layer. The diameter of the numerical spalled area (rear face) is about 0.52 m.

Figure 43 shows the displacement histories of the 5 points located on the front face of the concrete plate. The upper part of this figure shows the point locations. The maximum displacement reaches 22 mm at point W2.
Iris 2012 International Benchmark, Impact of projectiles on a reinforced concrete plate

Displacement at the front of the slab: W1
Displacement at the front of the slab: W2

Displacement at the front of the slab: W3
In the comparison with experimental measured displacements, we observed that the computed displacements simulated by modified PRM coupled model are more important, especially in first period of measured displacement. But we also observed that the measured displacements in this period are much closer to those of the second period of measure.

Left part of Figure 44 shows the positions and the orientations of the strain gages bonded on the reinforcement. Strains are plotted on Figure 51 to Figure 51. Table gives the
coordinates of these strain gages. The precise location of the strain gage on the rebar is
unknown [Figure 44]. In some particular situations, significant differences can be observed
between each position. The plotted curves are an average of the four possible locations.

![Figure 44. Strain gage location and orientation.](image)

Table 8 strain gage location and orientation

<table>
<thead>
<tr>
<th>Position relative to the center</th>
<th>D1</th>
<th>D2</th>
<th>D3</th>
<th>D4</th>
<th>D5</th>
<th>D6</th>
<th>D7</th>
<th>D8</th>
</tr>
</thead>
<tbody>
<tr>
<td>x (mm)</td>
<td>45</td>
<td>45</td>
<td>405</td>
<td>405</td>
<td>80</td>
<td>440</td>
<td>270</td>
<td>440</td>
</tr>
<tr>
<td>y (mm)</td>
<td>470</td>
<td>270</td>
<td>470</td>
<td>110</td>
<td>405</td>
<td>405</td>
<td>45</td>
<td>45</td>
</tr>
<tr>
<td>z (mm)</td>
<td>-85</td>
<td>-85</td>
<td>-85</td>
<td>-85</td>
<td>-95</td>
<td>-95</td>
<td>-95</td>
<td>-95</td>
</tr>
<tr>
<td>direction</td>
<td>y</td>
<td>y</td>
<td>y</td>
<td>y</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>position</td>
<td>rear</td>
<td>rear</td>
<td>rear</td>
<td>rear</td>
<td>rear</td>
<td>rear</td>
<td>rear</td>
<td>rear</td>
</tr>
</tbody>
</table>

![Figure 45. Strain gage history at D1](image)
Figure 46. Strain gage history at D3

Figure 47. Strain gage history at D4
Figure 48. Strain gage history at D5

Figure 49. Strain gage history at D6
Figure 50. Strain gage history at D7

Figure 51. Strain gage history at D8
4. Conclusion

This paper has presented the main features of the PRM model and its proposed improvement. The modified PRM model takes into account the influence of deviatoric stress on the volumetric behavior. The influence of the saturation ratio on the behavior of concrete under triaxial compression is also modified thanks to new approach. Therefore, the consolidation point is updated. The modification of coupling variable is also realized to generate a better continuity between the two model responses. These changes improve significantly the prediction of concrete behavior under triaxial compression with the PRM coupled model. The modified PRM coupled model was then used to simulate the flexion test and the punching test of the IRIS 2012 benchmark of the Nuclear Energy Agency (NEA) of OECD. In the new simulations, we took into account all the recommendations from IRIS_2010 workshop and IRIS_2012 Workshop to improve our computed results. Compared to experimental results of two tests, we validated the improvement of this model. The advantage of this model is that we can use the same parameters of concrete behavior to simulate different types of dynamic solicitation as impact case. At low confinement, the model allows reproducing the degradation mechanism and cracking of concrete plate, while it can predict the pore collapse phenomena of concrete at high confinement. In the two tests that we simulated, the maximum pressure is generally not high, about 150 MPa in shock domain and 100 MPa elsewhere all over. Only some elements in contact with projectile present high pressure (300 MPa). The erosion option was likewise employed in both two tests. It is worth noting that the disappearance of elements can lead the loss of element mass. The later will lead to lose momentum quantity, which can influence the calculation performance.

Reference:


Iris 2012 International Benchmark, Impact of projectiles on a reinforced concrete plate


A III.20 Team #26 VTT-Team I

IRIS_2012 NUMERICAL SIMULATION REPORT:
VTT_BEND TEST, VTT, Shell model

1. Finite element model

These analyses were carried out with Abaqus/Explicit finite element code (Version 6.11-1). Due to symmetry only one quarter of the slab was modelled. The model is shown in FIG. 1. Four-noded 3-D shell elements, with reduced integration were used. Element length is 5 cm. Rebars were modelled as smeared layers. Loading area and simply supported nodes are indicated by red colour. Loading area is defined by assuming a 45 degrees spreading angle through the slab thickness direction.

FIG. 1. One quarter shell element model.

Loading

m=50.5 kg, v=110 m/s

FIG. 2. Loading function by the Riera method.
The load function was calculated the Riera method assuming folding by viscoplastic crushing mechanism. Force time function is depicted in FIG. 2. This same loading function was used also in the earlier study in IRIS2010.

2. Material properties

**Reinforcement steel**

This slab is reinforced using bars with a diameter of 6 mm (φ6). For the IRIS2010 only the stress-strain curve for the reinforcement of φ8 was given. There is some difference in these stress-strain curves. Tensile test results carried out with φ6 rebar are shown in FIG. 3.

In this IRIS_2012 study the measured stress-strain curve of the 6 mm diameter rebar was applied without any further manipulations related to smeared cracking assumptions.

![Stress-strain curve (A500HW, φ 6 mm)](image)

**FIG. 3. Tensile test result of reinforcement steel φ=6mm.**

Stress strain curves applied for IRIS_2010 and IRIS_2012 studies are presented in FIG. 4. The increase of the yield strength at elevated strain rates was taken into consideration by the Cowper-Symonds visco-plastic model, adopting parameter values D=40 1/s and q=5.
Concrete

Nonlinear behaviour of concrete was modelled with so called concrete damaged plasticity model. A schematic view of the compression crushing and tensile cracking behaviour is presented in FIG. 5. Recovery stiffness after compression crushing and tensile cracking is illustrated: $w_\mathrm{c} = 0$ corresponds to no recovery as load changes from tension to compression and $w_\mathrm{c} = 1$ corresponds to complete recovery as the load changes from tension to compression, default value is $w_\mathrm{c} = 1$. $w_\mathrm{t} = 0$ corresponds to no recovery as the load changes from compression to tension.
changes from compression to tension and $w_t = 1$ corresponds to complete recovery as the load changes from compression to tension, the default value is $w_t = 0$.

The concrete test results for the same concrete pouring badge were given in April 20th 2010. The shape of the compression crushing behaviour curve was defined according to the material test results provided later for the concrete with compression strength of 69 MPa.

![FIG. 6. Assumed compression crushing behaviour of concrete.](image)

Stress-strain relationship for concrete under compressive stress is presented in FIG. 6. In the damage plasticity model the recovery stiffness is reduced with the applied damage parameter. After concrete crushing the recovery stiffness is reduced by multiplying Young’s modulus of concrete by the factor of $(1 - DC)$, where DC is the compression damage factor. Based on the material tests, the stiffness of the concrete is not decreasing much during the compression crushing process.

![FIG. 7. Exponential tensile cracking behaviour.](image)

Only the tensile splitting material test data was available. This means in practice the value of the static tensile strength. The increase of tensile strength due to elevated strain...
rate was taken into account by multiplying the measured static tensile strength by a factor of 1.4 which corresponds a strain rate of 1/s. The fracture energy was assumed to be 200 N/m and the stress strain curve was predicted for an element size of 5 cm. The exponential stress strain curve is presented in FIG. 7.

After tensile cracking the recovery stiffness is reduced by multiplying Young’s modulus of concrete by the factor of (1-DT), where DT is the tensile damage factor.

Since there were no detailed material tests data on tensile behaviour of concrete available, some sensitivity studies were carried out by varying the tensile damage assumption. Different tensile damage assumptions used in the sensitivity studied are presented in as a function of the cracking strain in FIG. 8.

In these studies the recovery stiffness obtained after tensile cracking was applied also under compressive behaviour. It should be noted that this is not the default assumption in the input data.

![FIG. 8. Tensile damage assumptions.](image)

**FIG. 8. Tensile damage assumptions.**

3. Calculation results

Displacements at all the five displacement sensor locations calculated using the base line assumption “DT_V1” are presented with the corresponding measured values in FIG. 9.
Sensitivity studies on tensile damage assumption

Central deflections calculated with the tensile damage assumptions shown in FIG. 8 are presented in FIG. 10 with the corresponding result recorded during the test. The tensile damage assumption affects mainly to the bending vibration behaviour of the wall.
Further sensitivity studies were carried out by varying the shape of the assumed tensile cracking behaviour and assuming the static tensile strength. A bilinear and an exponential assumption were applied. The fracture energy is the same in all the tensile cracking curves presented in FIG. 11. In these analyses the linear assumption “DT_V1” was used for the tensile damage parameter.

![Tensile strength](image)

**FIG. 11. Assumptions for tensile cracking.**

The calculated corresponding central deflections are presented in FIG. 12 with the measured result. Central deflection calculated by using the static bilinear assumption overestimates both the maximum and the permanent deflections. When the exponential assumption was used, the maximum tensile strength value is not affecting the calculated result that much.

![Central deflections](image)

**FIG. 12. Central deflections as a function of time with different tensile cracking assumptions.**
**Sensitivity studies on compression crushing assumption**

Some sensitivity studies were carried out also for the compression crushing behaviour assumptions. The compression strength of concrete measured from the same batch as the test slab was 64 MPa. The corresponding value for concrete used in the supplementary material tests was 69 MPa. In the analyses presented above the original compression strength was used for concrete and the corresponding calculated central deflection result is presented with the solid line in FIG. 13. The dashed line shows the result obtained using the compression strength of 69 MPa with a similar compression damage assumption as presented in Fig FIG. 6. According to the supplementary material test results the recovery strength is not decreasing as strongly as assumed in FIG. 6. In the material test simulation the compression damage parameter was predicted according to the available measurement data information. This compression damage assumption was used with the higher compression strength value of 69 MPa in calculation the bending behaviour of the test slab. The corresponding deflection is presented with the dotted line in FIG. 13. The base line tensile damage assumption DT_V1 was used in these analyses.

![FIG. 13. Central deflection as a function of time with different compression crushing assumptions.](image-url)
4. Conclusions and lessons learned

The calculated new baseline central deflection and the result of the IRIS_2010 benchmark are presented with the recorded result in FIG. 14.

The assumed recovery stiffness of concrete as the cross section becomes under compression after tensile cracking affects the bending vibration behaviour as well as the permanent deflection of the slab. With this assumption also the calculated results are in better agreement with the measured ones.

The manipulated stress strain curve used in the earlier study for the rebar in order to take into account the smeared cracking approach is not appropriate in the case of a two way supported wall. Originally that method was developed for nonlinear analyses of reinforced concrete beams.

Compression crushing behaviour do not affect the result much. The reason for this is the fact that only a rather narrow part of the cross section is under compression.
A III.21 Team #27 VTT-Team II

**IRIS_2012 NUMERICAL SIMULATION REPORT:**

*VTT punching case simulated by VTT Team II, 3D solid model (Kim Calonius)*

**Introduction**

The punching test is simple enough to allow concentration on the essential phenomena. Based on the test results already known, the behaviour of the slab is mainly punching which leads to perforation. In more detail, the local compression strength of the concrete is exceeded first and the missile penetrates into the target slab in a tunnel phase. Furthermore, at a certain stage the contact force surpasses the remaining shear capacity of the slab which leads into a formation of a punching cone. This ultimately leads to perforation with the missile having a residual velocity. The previous model by VTT gave good results within the blind benchmark especially when considering the main question: Does the missile perforate the slab, and if it does, what is the residual velocity? The FE model had a relatively coarse calculation mesh and the material model for the concrete was very simple compared to many other models used by the other teams. The important explanations for the good results besides the numerical modelling expertise, were the wide expertise within VTT team on the physical phenomena included in hard missile impacts, utilization of simplified methods, and the knowledge of some similar kinds of tests done at VTT in the near past.

The new model prepared for this second benchmark is basically just a more detailed version of the previous one. The changes to the model are not based on any calibration on the known test results but on real physical properties and material test results. The same model geometry has been used. Element mesh of the target slab is denser (50 elements instead of 20 elements through the thickness of the slab). The main difference is the new material model used for the concrete of the slab. It is coded by the user as a subroutine (VUMAT) within Abaqus code [1]. In the previous model, the slab had to be divided into two sections: one section for the tunnelling phase and one for the shear cone. Now, only one section is needed and the model is substantially more universal. In fact, it has been found to give relatively good results also in cases with only partial penetration and with different types of reinforcement. The main new features in the concrete model are strain rate dependency and different behaviour in tension and compression.

For the rebars, linear beam elements are used instead of linear truss elements. Contact definitions are somewhat better. The most recent Abaqus [1] version is used (6.12-1). Processed “true stress - true strain” curves are used for the metals instead of direct material test results. The material input data provided by the organizing committee was sufficient this time.

Abaqus was chosen as the calculation code, since it has been used both at VTT and in many other companies as well for decades and it has shown its competence many times. It is one of the few all-purpose explicit codes there are available. Explicit code is a natural choice for a highly dynamic and fast case such as this punching case. Convergence is more likely to be reached in explicit than in implicit time integration. Time steps can get very small, but the
overall analysis time is also short. This second benchmark case of punching behaviour was studied by one person for approximately 1 month including some sensitivity studies and reporting. As previously mentioned, the model was not created from the scratch but the one used in the first benchmark (IRIS_2010) was improved.

FE model

The model is a 3D quarter model (first quarter seen from the front side) utilizing symmetry conditions. This is reasonable, since the whole structure is nearly symmetric both horizontally and vertically and the missile hits nearly the centre of the slab in a straight angle. However, a whole model would give somewhat more accurate results and some minor phenomena may be lost in the symmetry assumption.

The reinforced concrete slab (Figures 1 and 2) is modelled with solid 3D elements with 8 nodes, reduced integration and size of 5 mm. There are 50 element layers through the thickness of the wall. If the dominant mode was bending, approximately 10 elements would be sufficient. In a punching case like this with element erosion, the solution gets more accurate with larger amount of elements and it has been found that approximately 50 element layers are needed somewhat depending on the material model. There are approximately 4 element layers outside the outer rebars, which should ensure quite realistic scabbing of concrete cover. The total numbers of elements and nodes for the slab are 2 000 000 and 2 060 451, respectively.

Only the span width of the wall is modelled and the supports are simplified to merely restraining the displacements of the back surface edge nodes in the impact direction. The supporting frame is not modelled in any way. The boundary conditions have a negligible effect on the main damage modes in this case. Otherwise the model corresponds to the real geometry.

The bending reinforcement bars inside the slab (shown in green colour in Figures 1 and 2) are modelled with linear beam elements with 2 nodes and size of 12.5 mm. The rebars are embedded rigidly inside the concrete, in the exact location they are initially in relation to the concrete elements. They are not tied to each other. There is not any really accurate interaction between the bars and concrete, but in case the concrete elements around the bars get eroded, the bars have contact conditions with the exposed new concrete surfaces (as well as with the missile and other rebars). The total numbers of beam elements and nodes are 3 520 and 3 564, respectively. The rebar ends also have symmetrical boundary conditions.

The missile (shown in Figures 1 and 3) is modelled with solid 3D elements with 8 nodes, reduced integration and approximate size of 10 mm. The number of elements and nodes are 2 127 and 2 850, respectively. An initial velocity of 136 m/s is given to the missile. The missile model has an approximate mass of 12 kg. The slab has a mass of 597 kg, thus 2 388 kg when multiplied by four.
Not many element types are offered by Abaqus for explicit analyses. These are the most reliable and efficient elements for this type of analysis. There are contact definitions between every free element facets in the model. Various friction coefficients of magnitudes from 0.3 to 0.5 depending on the materials are used for contact in tangential direction. Contact in normal condition is hard, except there is some nonlinear physical softening in the pressure-overclosure relationship between the missile and concrete (stiffness $k$ is approximately 33 GPa).

Figure 1. The initial configuration of the whole FE model showing the symmetry planes in the front. The support zone is indicated by purple colour. The rebars are also separately shown in green colour (left).

Figure 2. A detail of the FE mesh of the rebars (left) and slab (right).
The constitutive law for the reinforcement steel is an elastic-plastic material model with von Mises yield surface and isotropic hardening. This model has successfully been used in many types of analyses. The strain rate dependency is taken into account by Cowper-Symonds formula, where the parameters are $D = 40$ 1/s and $q = 5$. There is a failure criterion of 15% plastic strain. After reaching that level, the rebar elements are removed from the model. To be more precise, damage is first initiated at ductile criterion of 10% equivalent plastic strain. The damage is evolved until the ultimate failure takes place at 5% plastic strain after the initiation. Figure 4 shows the stress-plastic strain curve for zero plastic strain rate for the rebar steel.

The constitutive law for the missile steel (cyan colour in Figure 3) is the same used for the reinforcement steel. Stress-plastic strain curve for zero plastic strain rate for the missile steel is also shown in Figure 4. The strain rate dependency is taken into account by Cowper-Symonds formula, where the parameters are $D = 40$ 1/s and $q = 5$. There is no element erosion. Density, Young’s modulus and Poisson ratio are 7 850 kg, 200 GPa and 0.3, respectively, for both steel materials.

![Figure 3. The FE mesh of the missile. Steel and light-weight concrete are shown in cyan and orange colours, respectively.](image)

![Figure 4. Stress-plastic strain curve for zero plastic strain rate for the missile and rebar steel.](image)
The light weight concrete inside the missile has the same constitutive model than the steel materials. The yield point for 0.2% plastic strain is 6.5 MPa. The dynamic increase factor for strain rate of 100 is 2.5. Density, Young’s modulus and Poisson ratio are 1 573 kg, 3 GPa and 0.2, respectively.

The constitutive law for the concrete of the target slab is the most important issue and thus a very difficult one as well. During the discussions of IRIS_2010 workshop it was concluded that the numerical modelling of concrete behaviour in fast loading cases remains a challenging task. When modelling perforation, element erosion is a necessity, unless so-called hydrocodes are considered.

Due to limitations of the previously used material model of concrete a new material model development was initiated and it has been implemented as a VUMAT user subroutine in Abaqus by Juha Kuutti from VTT. In the present stage of development, the material model applied in the simulations utilizes linear elasticity combined with traditional von Mises plasticity (J2) with linear plastic hardening and softening-erosion criteria for tensile and compressive failure. Apparent strength increase as a result of high strain rates is taken into account by utilizing Ceb-Fib Model Code 1990 [2]. Small strain framework is assumed in the model. Strain rate is calculated from the average volumetric strain rate in the analyses.

Tensile failure is assumed to be governed by stress invariant \( I_1 \) and to be such an instantaneous event that no softening occurs and the element fulfilling this criterion is eroded instantly from the analysis. The compressive failure is based on the equivalent plastic strain and it includes some post-failure softening as is often observed in compressive stress-strain behaviour of concrete. After equivalent plastic strain has fulfilled the initiation criterion the material begins to soften linearly. The effect of softening is applied to the effective stress tensor in an isotropic manner often found in continuum damage mechanics models.

The stress-strain relationship in this model is sketched in Figure 5. Values of 29.43 GPa, 0.22, 60 MPa, 4.04 MPa, 5 GPa, 0.003 and 0.045 are used for \( E \), \( \nu \), \( \sigma_y \), \( f_t \), \( H \), \( \varepsilon_{\text{init}}^{\text{p}} \) and \( \varepsilon_{\text{fail}}^{\text{p}} \) (\( B = 15 \)) in the analysis, respectively. The 5 cm deep support zone (shown in purple colour in Figure 1) has substantially higher erosion criteria; \( \sigma_y = 600 \) MPa and \( f_t = 40.4 \) MPa.

![Figure 5. Material model for concrete of the target slab; stress-strain relationship.](image-url)
Results

The analysis time is 20 ms. The impact starts after 1 ms. The time increment size was $1.27 \times 10^{-7}$ s. There were 156,948 time increments. Eight CPUs were used and the CPU time was 29 hours.

Hourglassing can be a problem with first-order, reduced-integration elements. The integral viscoelastic form of hourglass control is used. It generates more resistance to hourglass forces early in the analysis step where sudden dynamic loading is more probable. Linear bulk viscosity that introduces damping associated with volumetric straining is used. Its purpose is to improve the modelling of high-speed dynamic events.

Figure 6 shows the energy balance which is well maintained throughout the analysis. The total energy and initial kinetic energy are 107 kJ which are in agreement with theory. Artificial strain energy is associated with constraints used to remove singular modes (such as hourglass control). It rises during the start of the impact to a value of 17 kJ, which is relatively high. Figure 7 shows the velocity of the missile rear. The residual velocity after the impact is approximately 52 m/s and it is reached 8 ms after the start of the impact.

![Energy balance](image)

**Figure 6. Energy balance in the 3D simulation of punching case by VTT.**

Figure 8 shows the deformed shape of the whole model 1 ms and 19 ms after the impact (after the instant the missile hits the wall surface). The deformation of the missile is minimal. The front part of the steel pipe behind the missile nose is slightly bulged out and the total shortening is 5 mm. As a mesh sensitivity study, the same analysis was done with a coarser mesh (20 elements through the wall thickness). The residual velocity was 32 m/s. The deformed shape of the whole model 19 ms after the impact with a coarser mesh is shown in Figure 9. The missile is more deformed in this case.
Figure 7. Velocity of the missile rear

Figure 8. Deformed shape of the whole model 1 ms (right) and 19 ms (left) after the impact.

Figure 10 shows the eroded elements (left) and deformed rebars (right). The eroded elements show illustratively the cracks inside the slab. The rebar deformation, bending and shear cutting, is in very good agreement with the test. The locations of strain measurements are shown in red colour.

Figures 11 and 12 show the deformed and damaged slab from different angles of view 19 ms after the impact. The simulated damage is in fairly good agreement with the test results. It shows the tunnelling phase, back surface scabbing and shear cone formation. There is clear scabbing on the back surface with an approximate radius of 0.6 m, but the cracks extend much further.
Figure 9. Deformed shape with a coarser mesh 19 ms after the impact (residual velocity was 32 m/s).

Figure 10. Eroded elements (left) and deformed rebars (right). The locations of strain measurements are shown in red colour.

Figure 13 shows the reaction force in the supports and the contact force between the missile and the wall (including rebars). The values are multiplied by four to have a result for a whole structure. The duration of the whole impact is 8 ms, but in fact the perforation almost takes place in 3 ms and after that only the centre back face rebars and some minor contact with concrete particles are resisting the missile movement. The peak values of the contact force and the reaction force both are approximately 5 MN.

Figure 14 shows the displacements of the wall in the impact direction at the deflection sensor locations. The maximum deflection in these locations is 3 mm. The frequency of the vibration is approximately 140 Hz. Figure 15 shows the vertical strains on the front surface of the slab at the locations of the strain gauges. The maximum value is 0.06%. Figure 16 shows the rebar strains from the strain gauge locations for the first 10 ms. The maximum value is almost 7%, but mainly the values are much lower.
Figure 11. Deformed and damaged front surface and vertical section of the wall 19 ms after the impact.

Figure 12. Deformed and damaged back surface of the wall (bottom part) 19 ms after the impact.

Figure 13. The reaction force in the supports and the contact force between the missile and the wall.
Figure 14. Deflections of the wall at the deflection sensor locations for the first 20 ms.

Figure 15. Vertical strains on the front surface of the slab at the locations of the strain gauges.

Figure 16. Rebar strains from the strain gauge locations for the first 10 ms.
Conclusions
The new simulation results agreed well with the test results although the residual velocity of the missile is slightly higher in the simulation. The old model predicted that with a better accuracy, but that is partly just a coincidence. The new model is clearly more detailed compared to the old one used in IRIS_2010, which especially shows in the different types of simulated damage in the slab. It was not yet shown, whether the mesh is now of the optimal size, but the mesh size has a clear effect on the results. It is difficult to say in a few sentences what has been learned. The reliability of a model is not verified with a single benchmark case but only after a series of different types of cases with the same model. Some main results are dependent on many different parameters in a model. The main model characteristics governing the behaviour in a punching case are the erosion criteria.

The triaxial concrete compression test results could not be optimally used for the FE model, since the constitutive model used does not take the dilatancy of concrete into account. The behaviour in compression cannot be determined as dependent on the confining pressure. However, the stress strain curve in compression was calibrated using the results of merely the unconfined compression test. The triaxial tests were simulated with the model described here. The simulated stress strain curves were not in very good agreement with the corresponding test results. The ultimate capacity of the specimen however increased with the confining pressure, even if not as much as in reality.

As already concluded in IRIS_2010 benchmark, the element erosion has many problems, but correctly used is still probably the most reliable way of modelling perforation. In its crudeness it undermines the finesse of sophisticated material behaviour modelling. The simulated failure planes remained horizontal with every confining pressure, which is not in agreement with the test results.

Some more aspects of the calculation model are described in Ref. [3].

References
A III.22 Team #28 VTT-Team III

Missile impact on simply supported concrete plate

Stainless steel missile impact on square slab

The dimensions of the simply supported slab used in the dry stainless steel missile impact tests are: the span $l = 2.0$ m, the thickness $h = 0.15$ m and the effective thickness $h_e = 0.131$ m.

The properties of reinforcement are: the modulus of elasticity $E_s = 210$ GPa, the yield stress $\sigma_y = 600$ MPa, the ultimate stress $\sigma_u = 670$ MPa, reached at a strain value of $\varepsilon_u = 0.1$. The reinforcement areas in two orthogonal directions on top and bottom faces are: $A_{sx} = 0.000514$ m$^2$/m and $A_{sy} = 0.000514$ m$^2$/m.

The material properties of the concrete slab used in computations are:

- the modulus of elasticity of concrete is $E_c = 26900$ MPa,
- the Poisson ratio is $\nu = 0.2$,
- the compressive strength of concrete is $f_c = 63.8$ MPa,
- the tensile strength of concrete is $f_t = 3.71$ MPa.

An experimentally observed period of vibration for a test plate with the same dimensions as the present plate was about $T = 0.023$ s giving for the frequency $f = 43.3$ 1/s and $\omega = 2\pi f = 270$ 1/rad.

Stainless steel missile

The missile fuselage is made of stainless steel pipe with a diameter of $d = 0.25$ m and a wall thickness of 2 mm. The mass of a steel pipe with a length of 2 m assuming a density of 7850 kg/m$^3$ is about 24.66 kg. By adding a front cap and some steel parts at the rear end of the missile the total mass increases to the desired 50 kg. The impact velocity of the missile in the analysed test is 110 m/s.

The load function due to a stainless steel missile in the case of an impact velocity of $v_0 = 110$ m/s is depicted in Figure 1. Curve $F_{fvp}$ is obtained with the Riera method by assuming a folding viscoplastic crushing mechanism and adopting the material parameter values: $E = 210$ GPa, $\sigma_y = 460$ MPa, $\rho = 7850$ kg/m$^3$ and parameter values for the Cowper-Symonds 1-D visco-plastic model, $D = 1522$ 1/s and $q = 5.13$. 
Results obtained by a two degree of freedom model and finite element model

By a two degree of freedom model (TDOF), like the CEB model, the bending behaviour and possible punching of the target plate can be analysed.

The simply supported plate tests are analysed by a two degree of freedom (TDOF) model assuming Rayleigh damping with a damping factor $\zeta_1 = 0.07$ for the bending mode. The damping matrix for the TDOF system becomes

$$
C = \begin{bmatrix}
c_1^\alpha + c_1^\beta + c_2^\beta & -c_2^\beta \\
-c_2^\beta & c_2^\alpha + c_2^\beta
\end{bmatrix} = \begin{bmatrix}
\alpha m_1 + \beta (k_1 + k_2) & -\beta k_2 \\
-\beta k_2 & \alpha m_2 + \beta k_2
\end{bmatrix}. \tag{1}
$$

The finite element results are computed using the Bogner-Fox-Schmidt (BFS) element with 16 degrees of freedom. The BFS element is based on Kirchhoff plate theory. A resultant formulation is adopted in which the material nonlinearities are formulated in terms of the stress resultants and curvatures. Moment curvature relationships are shown in Figure 5. Curve with label l16m is used in the FE calculations.

The plate is assumed symmetric and simply supported. The finite element model consists of BFS plate elements. For a plate quadrant a mesh of 8 by 8 elements is adopted. Figure 6 shows the deflected shape of the plate at the time of maximum deflection.

Figure 7 shows the central deflection history of plate B1. D1 is the test result. Curve plk is obtained with the FE method using BFS elements. For the sake of comparison curve
Figure 2 Load function of stainless steel missile with an impact velocity of $v_0 = 110$ m/s obtained by folding viscoplastic model and with an axisymmetric folding model where the formation of folds is taken into account.

plr is calculated with finite elements based on Reissner-Mindlin plate theory. For both FE methods a 8 by 8 mesh is used. A Rayleigh damping with a damping ratio of 0.05 at frequencies of 40 and 200 Hz is assumed.

Deflection histories at locations D1, D5 and D4 calculated with BFS elements using a 8 by 8 mesh are shown in Figure 8.

Rebar strains at locations 3 and 5 are shown in Figure 9. B1-3 and B1-5 are test results, l16-3 and l16-5 refer to FE results obtained by the BFS element.

Force history of TDOF model is depicted in Figure 10, in which $r_1$ and $r_2$ are the spring forces, $g_1$ and $g_2$ are the damping forces and $f$ is the load history.
Figure 3  Final shape of stainless steel missile after an impact with a velocity of \(v_0 = 110\) m/s.

Figure 4  TDOF plate model with Rayleigh damping.
Figure 5  Moment-curvature relationship for test plate B1

Figure 6  Deflection at time $t = 0.0136$ s for test plate B1 obtained by a FEM model of 8 by 8 BFS elements for one quadrant.
Figure 7  Central deflection history of plate B1. D1 is test result. Curve plk is obtained with FE method using BFS elements based on Kirchhoff plate theory and plr is calculated with finite elements based on Reissner-Mindlin plate theory. 8 by 8 mesh is used in both cases.

Figure 8  Deflection histories of plate B1 at locations D1, D5 and D4 calculated with FE method using BFS elements and 8 by 8 mesh.
Figure 9  Rebar strains of plate B1 at locations 3 and 5. B1-3 and B1-5 are test results, l16-3 and l16-5 refer to FE results.

Figure 10  Force history of TDOF model of plate B1. $r_1$ and $r_2$ are spring forces, $g_1$ and $g_2$ are damping forces and $f$ is load.
1 Introduction

The goal of the IRIS 2010 benchmark was to evaluate state of the art simulation technologies in comparison with real test data for investigating missile impacts on concrete structures. In a first step of this project, carried out 2010, the participants used data of Meppen tests on concrete slabs to calibrate the material models and the simulation methods. Afterwards a blind simulation of a flexural and a punching missile impact test conducted at the VTT research centre in Finland were carried out. In the final assessment of all simulation results varieties in the material assumptions of the participating groups especially for the concrete were observed. Accordingly the simulation results were based on different input data respectively assumptions, leading to uncertainties in the evaluation process.

For reducing this effect additional tri-axial test data was provided for the concrete used for the flexural and the punching setup at the VTT centre. Respectively in 2012 a second step of the project started using this additional data. A further aspect of this second project step is sensitivity analysis with respect to modelling parameters.

Woelfel Beratende Ingenieure takes part in this second step. As now almost all relevant test data for the flexural and punching test are provided, the calibration step using the Meppen slab test is not considered. Independent therefrom it has to be mentioned, that no dynamic testing data neither of the concrete nor the reinforcement or the missile is available. As these information are very important for the high-speed dynamic simulation of a missile impact, literature values are taken into account for the here presented approach.

For modelling the setups the pre-processor HyperMesh [1], version 6.11, is applied. As FE-solver ABAQUS version 6.10-3 [2] with an explicit algorithm is used. There are several years of experience at Woelfel using this solver for the nonlinear crash analysis of passenger aircrafts on civil engineering structures. The identification of the material parameters is carried out using Matlab version 7.5 [3]. Spend work time was about 6 weeks for one person.

2 Material Models

Beside standard FE-simulation requirements as choice of element type, discretization, boundary definition and stability controlling effects, the most important step is the definition of the material behaviour. This covers always two steps. First part is the choice of the material model, which should qualitatively correlate with the observed real behaviour. Second part is the identification of the free parameters to describe the specific material test data. The following subchapters describe briefly the used material models and the results of the identification process.

2.1 Concrete

ABAQUS Explicit provides beside standard material models as the Drucker-Prager approach the Concrete Damage Plasticity model [4]. This model provides the general capability for modelling concrete or other brittle materials which could include reinforcements. It is designed for setups with monotonic and dynamic loading scenarios but has the limitation of low confining pressures which could be a problem for the simulation of the tri-axial tests with higher pressure values. In the following figure 1 the modelled behaviour is qualitatively shown for compression and tension.
The material definition is separated in tension and compression. For each of these material states the plasticity and damage behaviour must be described. The damage part specifies thereby the reduction of Young’s modulus which is shown in figure 1.

In the next step the material parameters are identified using Matlab. Out of the provided test data for uniaxial compression the maximum strength is assumed with 65 MPa at a strain of 0.5 %. For tension the maximum value is 4.5 MPa. As Young’s modulus the measured value of 30,000 MPa is assumed. Regarding these values it has to be mentioned, that compared to standard definitions as e.g. from the CEB-FIP Model Code 90 [5], the strain value is very high (normally about: 0.22 %) and the Young’s modulus is very low (for $f_{ck}$ of 65 MPa normally about 41,000 MPa).

Beside these key data the characteristics of the stress-strain curve are defined following the CEB-FIP Model Code 90. Regarding the damage a value of 30 % is assumed at maximum peak stress. Regarding the high-speed dynamic character of the missile test, the quasi-static material approach is enhanced with strain rate dependence. Based on the static behaviour the dynamic properties are computed for strain rates up to 200 s$^{-1}$ by following the equations out of the CEB-FIP Model Code 90. In figure 2 the stress-strain curve is shown for the different strain rates. The changes in the maximum compressive stresses are correlating with other literature values [6]
Finally the standard material approach is enhanced by a user subroutine for model erosion depending on principal strain values. This enhanced material definition is required especially for simulations with greater damage areas as e.g. the punching test.

2.2 Reinforcement

Beside the concrete the behaviour of the reinforcement has to be defined for the investigated plates. Within the VTT research report [7] for both plates (flexural and punching) a yield strength of 500 MPa was defined. Looking on the provided measurement data in this report a difference is recognized regarding maximum strength. Consequently two different materials are chosen for simulation of flexural (Bst 500-550) and punching test (Bst 420-500).

Similar to many of the other groups the Johnson-Cook plasticity approach is used for describing the reinforcement. This includes the plastic behaviour and the strain rate dependence as shown by the equation:

\[
\bar{\sigma} = \left[ A + B\left(\varepsilon^p\right)^n \right] \left[ 1 + C \ln\left( \frac{\varepsilon^p}{\varepsilon_0} \right) \right] \left( 1 - \dot{\varepsilon}^n \right)
\]

Formulation of Johnson Cook plasticity

As no test data about strain rate dependence is provided literature values are assumed [8]. Furthermore a standard damage model is included, which is important for the punching simulation. In figure 3 the comparison of input data out of [8] is compared with the identified material behaviour for Bst 500-550.

![Figure 3: Defined stress strain curve under tension for Bst 500-550 with different strain rates](image)
2.3 Missiles

Finally the properties of the both missiles have to be defined. Following the given data of the report these materials are used:

Flexural test:

- Stainless steel end cap and pipe: EN 1.4432
- Carbon steel pipe and plate: S355

Punching test

- Dome, pipe and endplate: S355
- Concrete filling: LC25/28

For all metals the Johnson Cook plasticity material model including damage is considered. Furthermore standard values for rate dependence are assumed. The key data as yield strength and failure strain are defined following literature values.

The lightweight concrete of the missile filling for the punching test is defined by the Concrete Damage Plasticity Model. The key parameters as the maximum compressive strength are specified corresponding to DIN 1045-1 [9].

3 Simulation and Results of Tri-axial Concrete Test

The first simulation carried out is the investigation of the concrete behaviour by the tri-axial test. Thereby the used material model and the identified material parameter should be calibrated in an iterative process for the uniaxial and a three dimensional loading case with confinement pressure. Another goal is to simulate the crack behaviour which appeared in the tri-axial test data and changes with respect to the confining pressure.

The simulation consists of two steps. In the first part the confining pressure is applied to all surfaces of the cylinder, i.e. bottom, top and lateral. In the second part the vertical loading is executed displacement controlled, by giving all nodes at the top and bottom a constant velocity. Corresponding to the real test setup the nodes at the bottom and the top are limited in their lateral displacement. This represents in the measurement the contact to the pistons.

Within the work a sensitivity analysis regarding element discretization is carried out. In a first model setup the test cylinder is modelled by hexahedron elements with reduced integration (C3D8R) and characteristic element length of 3 mm. The model consists of 20520 elements with about 66300 degrees of freedom. It should be mentioned for this detailed setup, that the model assumes a homogenous material behaviour over the complete specimen. This represents not the real properties as the cement and the aggregates have different properties. For detailed investigation of crack a micro-model including all details should be used.
Although these limitations it was possible to analysis the global crack behaviour, i.e. differences in the crack propagation due to changes in the confining pressure can be observed, see in figure 4 the damage due to compression for 0, 15 and 26 MPa.

Figure 4: Start of cracking for tri-axial test with different confining pressures: 0, 15, 26 MPa

A problem of this model setup is that it starts to become instable just after reaching the peak stress state. Reason therefore is that the damage control for compression is depending on strains and not displacements. Accordingly the dropping off part of the stress strain curve could not be computed. As the effect of instability disappears by reducing the number of elements over the height the element size is increased step by step. The results presented in the excel sheet are belonging to a model setup with a characteristic element length of 10 mm, i.e. 600 elements with 2100 degrees of freedom.

Comparing the simulation results for 0, 15 and 26 MPa with the real measurement data a good correlation for the stress-strain curves in vertical and lateral direction is observed, see excel sheet for tri-axial concrete test. Greater differences appear for the simulations with higher confining pressure. Although the maximum peak stress is similar to the measurement the simulations seems to less compliant, i.e. the final failure strain is much lower.

Coming back to the initial description of the used material model Concrete Damage Plasticity it must be asserted, that the limitation regarding higher confining pressure is shown by this validation process. Summarizing the comparison to the whole set of measurement results a good correlation is observed.

4 Simulation and Results of VTT Flexural Test

4.1 Missile Modelling

Facing the results of the first Benchmark step of 2010, a general course of action for the modelling of the missile but also for the plate is defined. Representing the real structure of the missile the four different parts: end cap, stainless steel pipe, carbon steel pipe and carbon steel plate are considered in the setup. As the thickness of all parts is small compared to the in plane dimensions, the modelling is carried out with shell elements with reduced integration (S3R and S4R). The element size varies over the length of the missile. While the end cap and the first part of the pipe are built up with an element length of 4 mm, the back part consists of shells with a size of 18 mm, see figure 5. The thickness is specified after the values of the VTT report [7]. In a first setup the density of the steel material is assumed with 7.85 t/m³. Comparing the masses of each part with the weight values given in the VTT report [7] a difference is observed.
Consequently the density is adjusted to fulfil the values of the report. Finally the missile has a mass of 50.5 kg. The materials are chosen following the specification of the report, i.e. the cap and the complete pipe are out of S355 and the end plate and end pipe are of EN 1.4432.

Figure 5: Missile model of flexural test: Complete model and detail view of end cap

Altogether the model consists of about ~73000 shell elements and ~72800 nodes with 6 degrees of freedom.

4.2 Plate Modelling

The setup of the plate can be separated in the concrete part, the reinforcement and the connection of both parts. The modelling of the plate is carried out by hexahedron elements with reduced integration (C3D8R) and an element size of 20 x 20 x 18.75 mm³, i.e. eight elements over the plate thickness of 150 mm. The complete model consists of 91500 elements with 105000 nodes and three degrees of freedom per node, see figure 6.

Figure 6: Plate model of flexural test

In a sensitivity analysis of the model setup the size of the plate elements is increased via 40 x 40 x 25 mm³ up to 80 x 80 x 37.5 mm³.

The bending reinforcement exists of linear beam elements (B31), see figure 7. The element length is in the outer part 20 mm and in the inner area 10 mm. The stirrups are over the complete plate modelled with an element length of 10 mm. The overlapping length illustrated in the sketch of the VTT report is considered for the bending reinforcement and the stirrups.
Figure 7: Reinforcement model of flexural test

The coupling of both parts is carried out by using the Embedded Element Approach of ABAQUS. This represents a kinematic coupling definition between the nodes of embedded elements, i.e. the reinforcement, and the host elements, i.e. the plate. The coupling is due to its kinematic character rigid, consequently the real slip behaviour between concrete and reinforcement is not considered.

4.3 Frame Modelling

A very important part for the complete analysis is the modelling of the frame. With the carried out investigation several setups are tested and a great sensitivity of the results is observed. A problem for the modelling is the fact that the frame is not completely defined. Out of the given data it is not clear if the plate bearing is only a roller support or if it also can take moment loads. Furthermore the compliance of the frame is not given in detail especially if any pre-stressing appears due to the lateral screw threads connecting the upper and the lower beams. In figure 8 the complete model setup of the frame and a detail of the connection to the plate are shown.

Figure 8: Frame model: complete setup and detail about connection to the plate

Thereby the backpipes are represented by beam elements. The connection to the plate bearings, implemented by solid elements (C3D8R), is carried out with kinematic couplings. Finally upper and lower bearings are connected to each other with beam elements enabling a pre-stressing of the plate.
4.4 Simulation and Results

For the simulation the explicit solver of ABAQUS is used. The connection between all model parts is defined by a general contact algorithm using a penalty approach. The friction coefficient for the contact between the different model parts is assumed with a value of 0.5. The computed time of the impact is simulation time with a varying stable time increment of 1 E-7 up to 5 E-7 s. The simulation itself consists of two steps. In the first part a small pre-stressing of the frame bearing is applied to take into account the effect of the lateral screw threads. In the second step the missile impacts the plate with its initial velocity of 110.5 m/s in normal direction of the plate. No imperfections in flight direction are taken into account. The simulation is carried out on a Linux system using 12 cpus in parallel. Altogether the base model has 1,150,000 degrees of freedom. The computational time for 10 ms is about 1 hour. Altogether 100 ms are simulated.

The following picture represents the main results of base model for the concrete cracking and the deformation of the missile. All other quantitative results are presented in the excel sheets.

Figure 9: Result of flexural test: concrete cracking due to tension at the backside of the plate

Figure 10: Result of flexural test: concrete cracking due to tension at the front side of the plate
Basically the results out of the test could be achieved using the described model setup for the plate and the missile. Comparing the maximum displacement the model is with a value of about 27 mm in the range of the measurement, see figure 12. A difference is observed for the swing back phase of the plate. While for the measurement almost elastic behaviour is observed, i.e. the final displacement is about 7.5 mm, the simulation shows also plastic behaviour i.e. the final value is about 22 mm.

Looking in detail on the curves, the simulation shows a smaller inclination in the swing back phase than the measurement. This effect could only be due to the presence of a greater damage. As almost no damage of the concrete due to compression occurs, this must be related to the tensional behaviour. So far the tensional behaviour is only defined by the maximum strength, which was given by the Brazilian test. As no detailed information about the cracking evolution
was given, the behaviour is defined following the standard CEB-FIP Model Code 90 [5]. To evaluate the influence of this material parameter additional simulations changing the tensional damage evolution should be carried out.

Further it is observed that the wave length of the oscillation after the impact is for the simulation smaller than for the measurement. This is normally a signal for higher stiffness respectively a lower damage of the plate, which disagrees with the observation before. The differences with respect to the wave length are especially a problem for the investigation of induced vibrations.

A possible reason that the simulation oscillates with a higher frequency is the fact, that the frame compliance is not considered within the model setup. Computing the eigenfrequencies of the used model setup with varying boundary conditions at the roller support (fixed or only support of translational dofs) values between 100 and 220 Hz are determined. Looking on similar computations of other groups which considered the frame in detail, the first eigenfrequency of the plate is only about 50 Hz. With respect to these differences of the initial state a remodeling of the setup must be carried out to improve the results in the swing back and oscillating phase.

Consequently a learned lesson is that the model must always be checked by an eigenfrequency analysis of the initial state. Validating the setup with results of an experimental modal analysis or former studies on similar structures is a good basement for the investigation of induced vibrations.

In a sensitivity analysis the element size for the concrete was varied from 20 x 20 x 18 mm³ up to 80 x 80 x 37.5 mm³. Regarding the maximum displacement no big influence due to the varying element size is observed, see figure 13.

![Figure 13: Variation of displacement with respect to different element sizes](image)

Looking on the damage behaviour a difference is determined, as with increased element size the resolution of the damage decreases, see figure 14. A possible reason for this is that increasing the element size lumps the damage and therefore shows smaller values.
Figure 14: Comparison of cracking resolution due to element size

Base model: 20 x 20 x 18.75 mm³

Variant I: 40 x 40 x 25 mm³

Variant I: 80 x 80 x 37.5 mm³
5 Simulation and Results of VTT Punching Test

5.1 Missile Modelling

Compared to the flexural test the missile of the punching setup is obviously different. Main difference is the concrete filling of the pipe. The model consists of four parts: the steel dome, the pipe and the plate and finally the concrete filling. While the dome and the filling are modelled with solid elements (C3D8R – hexahedron), the two outer parts are modelled with shell elements (S4R). Based on the fact, that the deformation of the missile is small, the used mesh density is coarse compared to the setup of the flexural test. The characteristic element length is in the frontal part about 10 mm and in the back about 20 mm. The materials are chosen following the VTT report, i.e. all steel parts are defined with S355 and the lightweight concrete LC25/28 is used for the filling. In figure 15 the setup of the missile is shown.

![Figure 15: Deformation of the missile due to the impact](image)

Altogether the missile model consists of about 12,400 elements and 11,700 nodes with three respectively 6 degrees of freedom. The final mass of the missile model is 47.3 kg.

5.2 Plate Modelling

The modelling of the plate is similar to the flexural test. The concrete is implemented with hexahedron elements (C3D8R) with an average length of about 20 mm, i.e. 12 elements are used over the thickness of 250 mm. Altogether the model consists of 132,000 elements with 146,000 nodes and about 420,000 degrees of freedom.

Evaluating the influence of the element size two model variants with 40 x 40 x 40 mm³ and 80 x 80 x 50 mm³ are generated.

The setup of the reinforcement is different to the flexural test as no stirrups are part of it. The modelling itself is similar, i.e. beam elements (B31) are used for the bending rebars. The element length varies from the outer plate area to the inner from 20 to 5 mm. The finer mesh setup in the impact area is made for detailed analysis of the punching cone. The cover of the concrete is 20 mm. The reinforcement consists of ~18,000 elements with ~18,000 nodes. In figure 16 the model setup of the plate is shown.
The coupling of both parts is as for the flexural setup carried out by using the Embedded Element Approach of ABAQUS, see chapter 4.2.

5.3 Frame Modelling

The frame modelling of the punching test is similar to the bending test. It should be mentioned, that the results of the punching simulation are not so sensitive to the modelling of the frame.

5.4 Simulation and Results

For the simulation the explicit solver of ABAQUS is used. The connection between all model parts is defined by a general contact algorithm using a penalty approach. It should be mentioned that element erosion has to be taken into account. Consequently the contact definition must be enhanced to consider new outer element surfaces which appear within the simulation.

The friction coefficient for the contact between the different model parts is assumed with a value of 0.5. The computed time of the impact is simulation time with a varying stable time increment of 1 E-7 up to 5 E-7 s. The simulation itself consists of two steps. In the first part a small pre-stressing of the frame bearing is applied to take into account the effect of the lateral screw threads. In the second step the missile impact the plate with its initial velocity of 135 m/s in normal direction of the plate. No imperfections in flight direction are taken into account. The simulation is carried out on a Linux system using 12 cpus in parallel. Altogether the model has about ~730,000 degrees of freedom. The computational time for 10 ms is about 1 hour 20 minutes. Altogether 50 ms are simulated.

The following figures represent the main results: the concrete cracking of the plate, the deformation of the missile and the residual velocity of the missile.
Figure 17: Result of punching test: concrete cracking due to tension at the front side of the plate

Figure 18: Result of punching test: concrete cracking due to tension at the backside of the plate

Figure 19: Result of punching test: missile deformation shown by the minimal principal strain
Comparing the results of the plate visually with the real test a good correlation is observed for the base simulation.

The deformations of the missile are as expected small. Higher strain values are only observed in the transition zone from dome to pipe, which correlates with the real test results.

Carrying out a similar sensitivity study for the element size as for the flexural plate (element length varies between 20x 20 x20 up to 80 x 80 x50 mm³) a great influence onto the residual velocity is observed, see figure 20. For the model variant II with the maximum element size the value decreases down to 18 m/s being only 30 % of the base model. The reason is related to the punching cone. Due to the bigger element size the missile must for variant II spall out a bigger amount of concrete, see figure 21. This requires a higher amount of energy leading finally to a lower residual velocity.

Figure 20: Residual missile velocity: Results for different model variants

Figure 21: Comparison of punching cone area between base model and variant II
Further a sensitivity study is carried out for the erosion criteria of the concrete elements varying the limit value from 5 up to 20 %. Looking on the residual velocity a big influence of the material definition is observed.

![Time - Velocity History of Missile for Punching Test](image)

**Figure 22: Influence of element deletion criteria on residual velocity**

In reality no erosion criteria exist, i.e. it is only a model assumption and does not represent any physical quantity. A possible solution for this problem could be a study with different wall designs (thickness, amount of reinforcement …) to calibrate this value in detail.

An alternative is a new simulation approach, where the volume elements are not deleted, but converted to smooth particles. This is still a model assumption but it has the benefit that it only influences the stiffness. The mass inertia is not affected by the element conversion. Within first test simulations, see figure 23, good results are achieved for the plate damage and the residual velocity. Compared to the approach using the erosion criteria the method seems to be less sensitive regarding the definition of the conversion criteria.

![Simulation converting volumetric elements into particles after reaching a limit strain](image)

**Figure 23: Simulation converting volumetric elements into particles after reaching a limit strain**
6 Conclusions and Lessons learned

Comparing the simulation results with the real test data qualitatively a good correlation is achieved. Looking into details differences are determined, which are caused by method limitations or model assumptions. This should be considered within future projects. Following the main aspects of each simulation part is described.

6.1 Material Behaviour of Concrete and Simulation of Tri-axial Tests

1. Regarding the material behaviour of the concrete it is important that detailed information is given by uniaxial and tri-axial loading tests (compression and tension). Using information only based on code definitions could lead to wrong material assumptions, as e.g. for Young’s modulus of the used concrete. Following the code definition a maximum compressive strength of 65 MPa would lead to a Young’s modulus of 40 GPa. Contrary thereto the value of 30 GPa was determined in the measurements!

2. An additional benefit for this kind of simulations would be the input out of dynamic tests with different strain rates. Current code definitions include only rough statements about the change of maximal strength with respect to the strain rate.

3. Within the project the limitation of the used material model in ABAQUS (‘Concrete Damaged Plasticity’) regarding high confinement pressures was determined. While a good correlation was observed between simulation and measurement for low confining pressure, differences are realized for high confining pressure. Possible improvements could be an enhancement of the material definition by user subroutines.

6.2 Simulation of Flexural Test

1. Within the comparison between simulation and measurement differences are especially observed in the swing back phase and the further plate oscillation. Compare the eigenfrequencies for the initial state from the presented model with setups including the frame in detail a big difference can be observed, i.e. that here presented approach shows much higher values. This could be a reason that the wave length of the oscillating phase is much lower than for the measurements. Out of that it could be learned that especially for investigation of induced vibration an enhanced model validation is required by an eigenfrequency analysis. As validation data results of similar structures or of an experimental modal analysis should be used.

2. Regarding the evaluation of the damage distribution an influence of the element size is observed. Increase the element size lumps the results, so that the maximal local damage values are reduced.

6.3 Simulation of Punching Test

1. Basically the simulation correlates with the measurement. Differences are especially observed with respect to the residual velocity. On reason therefore is the size of the elements. One lesson learned out of carried out sensitivity study is that the element length should be chosen to describe the outer shape of the punching cone accurately. Element size above will lead to lower residual velocities as the punching cone increases.

2. Finally a sensitivity analysis of the element erosion criteria was carried out. By the results it could be observed that this model assumption has a big influence on the simulation results. As it does not represent a real physical quantity an accurate definition of the
erosion criteria is very difficult and therefore causes problems for the simulation. A possible solution could be a study over several wall designs, finding in comparison to real measurements an erosion value which fits in general.

A further alternative could be a new model approach converting volume elements into smooth particles after reaching a defined strain value. Out of first test simulations it could be observed, that the simulation results are less sensitive to this conversion criteria.

6.4 General open issues

1. For the investigation of local damages as e.g. for the tri-axial test or the plate simulation it must always be taken into account, that the presented models are a macroscopic approach with homogeneous properties. In reality concrete is not homogeneous (aggregates with varying grain size, cement …) and therefore difficulties can occur by applying macroscopic models to predict the local cracking behaviour. This problem can’t be solved by decreasing the element size.

2. For local damage analysis the coupling between reinforcement and concrete is very important. The current models are using kinematic couplings, which is a simplified approach. Therefore engineers should consider this assumption within the assessment of simulation results.

3. A final statement for these simulations is that always the same model setups: material (stiffness, failure criteria) and elements should be used for a detailed structural analysis. Evaluating all types of simulation which have been used in the benchmark it may be concluded, that for one scenario one method is better than the other ones and vice versa. But looking on the application within daily work the engineer is not 100% sure which scenario he is facing with.

7 References