CHALLENGES IN SIMULATING THE IMPACT OF MISSILES ON REINFORCED CONCRETE STRUCTURES

A. Faecke[1], F.H. Schlüter[1], A. Fila[2], R. Danisch[2]
[1]SMP Ingenieure im Bauwesen GmbH, Karlsruhe (Germany)
[2]AREVA NP GmbH, Offenbach (Germany)
E-mail of corresponding author: a.faecke@smp-ing.de

ABSTRACT

One increasingly important design requirement for reinforced concrete structures of nuclear power plants is to protect the vital equipment from potential impact loads caused by missiles or drop loads. In order to continuously improve the design of such load cases various experimental and numerical investigations are carried out which increase the understanding of the response of reinforced concrete structures under high-rate impact events. In this context a benchmark for validation of numerical simulation methods was performed in the framework of the IRIS 2010 Project, sponsored by the OECD/NEA IAGE WG. The task was to predict the experimental results in terms of load transfer from missiles on concrete slabs as well as their deformations and failure mechanism (e.g. bending, punching). The new tests have been performed at VTT in Finland and the results were not known to the participants who performed the simulations. Additional tests from Germany ('Meppen Tests' carried out in the 1980s in Germany) were provided for a calibration of the numerical models. AREVA NP GmbH, Germany and SMP Ingenieure im Bauwesen GmbH participated as a team in this project and performed simulations for the two test configurations. The finite element approach was used to model the missiles and the reinforced concrete slabs. Nonlinear material behavior with strain rate effects and contact effects were considered in the simulations. Especially the prediction of missile performance is highly dependent on strain rate effects. Because failure of the slabs was possible, material softening had to be considered which makes it sophisticated to gain unambiguous solutions. The work has revealed that the choice for parameters like fracture energy, tensile strength at high strain rates and mesh size for concrete as well as strength and load deformation behavior at high strain rates of steel are critical. The choice of these parameters has a distinct influence on results, but information in literature is sparse or diverges greatly. This report presents some details of this challenge. The numerical simulations, the selected material laws and the validation of simulation results are described. Finally the submitted results are outlined in comparison to the later disclosed experimental results.

INTRODUCTION

Mainly reinforced concrete structures are used for the protection of the vital equipment at nuclear power plants from potential impact loads caused by missiles or drop loads. This is because massive structures that can dissipate much energy are necessary and that can best and most economically be realized by using RC structures. These structures react highly nonlinear at impact loading due to crushing and cracking of concrete and yielding of reinforcing steel and their interaction. In order to continuously improve the design of RC structures for impact small and large scale tests are conducted at different facilities and numerical investigations are carried out which increase the understanding of their response under high-rate impact events.

In the framework of the IRIS 2010 Project two different types of tests on 2.1 m x 2.1 m RC slabs with impact loading have been performed at VTT in Finland. The impact loading was once a soft impact with a thin walled hollow steel missile and once a hard impact with a concrete filled missile. At the same time numerical simulations have been performed from many participants in order to predict the test results. Finally the experimental test results were disclosed and compared to the numerical predictions. AREVA NP GmbH, Germany and SMP Ingenieure im Bauwesen GmbH participated as a team and performed simulations for the two test configurations.

The simulations have been performed with the finite element code ABAQUS [1] and the explicit solver. Different constitutive equations from the ABAQUS library for concrete, missile steel and reinforcing steel have been used. Because many of the material parameters could not be defined by the provided data or by literature sensitivity studies have been done prior to the final simulations. The studies have shown that in particular rate effects on missile steel, concrete tensile strength and fracture energy have a significant influence on the results. A calibration of the models was necessary and that was done by using the 'Meppen Test' results that were performed in the 80s. The influence of the parameters and the finally applied parameters that were based on the calibration are presented in this paper.

SLAB WITH BENDING FAILURE

Test setup and numerical model

A 2.1 m x 2.1 m x 0.15 m reinforced concrete slab was loaded with a missile that had an initial velocity of 110 m/s. The slab was horizontally slightly reinforced but there was no transverse reinforcement. The concrete had an approximate compressive strength of f_{cm} = 55.2 MPa and a tensile strength of f_{ctm} = 3.71 MPa. The slab is supported along a 2 m side square on the front and rear side. The support is realized by a steel frame that is very stiff and that experiences negligible deformations compared to the slab deformations. The missile consists basically of a pipe with an end cap. It has a mass of 50 kg, a length of ~2 m, a diameter of 254 mm and a wall thickness of 2 mm. The steel in the deformation zone is EN 1.4432.

The experimental test was split into two separate simulations. In the first simulation only the missile was modeled and the impact was realized on a completely rigid target. In this simulation the history of contact forces was evaluated. In the second simulation the slab with the load bearing frame was modeled. In this simulation the load history from the first simulation was applied on the slab. This splitting of the test in two separate simulations is possible because the deformation of the missile is large in comparison to the target ('soft impact'). For both simulations 3D-models were used. The test configuration including slab and missile has two axis of symmetry so that simplifying assumptions could be applied. However both planes of symmetry cut the impact zone where high strains and high contact pressures have to be expected. If artificial boundaries at symmetry axis are implemented in order to reduce calculation time wave reflections lead to unrealistic damage accumulations in these regions. Hence no simplification due to symmetry was applied.

Numerical model of the missile simulation

Fig. 1 shows the model of the missile with the rigid target. The shape of folding of the missile in the crushing zone is mainly dependent on the diameter of the missile and the wall thickness. The maximum element length is again dependent on the shape of folding. Sensitivity studies of the finite element mesh have been performed and it was found that a maximum length to thickness ratio of 2.0 leads to unique results. With this mesh refinement a smooth curvature of the folded sheet is achieved (Fig. 3). With a wall thickness of 2 mm the element length in the potential crushing zone was selected with 4 mm.

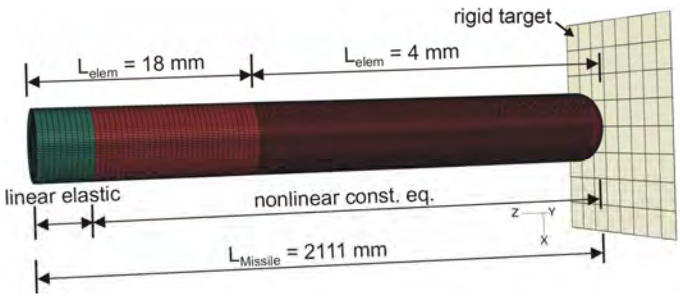

Fig.1: Finite element model of the first simulation with the missile and the rigid target

In the potential crushing zone plus an arbitrarily chosen length where possible nonlinear deformations may occur was modeled with nonlinear constitutive equations. Only for the rear part of the missile a linear elastic material model was applied. For the nonlinear part the Johnson Cook plasticity model [9] that is implemented in the ABAQUS material library was used. In this material law the yield stress is defined

$$\sigma = \left[A + B \cdot \varepsilon_{pl}{}^{n}\right]\left[1 + C \ln\left(\frac{\dot{\varepsilon}_{pl}}{\dot{\varepsilon}_{0}}\right)\right]\left[1 - \hat{\theta}^{m}\right] \tag{1}$$

In equation (1) the third term for temperature effects was omitted because it was assumed that temperature effects have a minor influence. A material softening or failure was not considered. The parameters A, B and the exponent n describe the uniaxial stress strain behavior at first loading. They were determined by curve fitting to provided test data to A = 310, B = 550, n = 0.4. The parameter C and $\dot{\varepsilon}_0$ describe the rate effects of the metal. The parameter $\dot{\varepsilon}_0$ defines the minimum strain rate where rate effects start. The parameter C defines the intensity of rate effects. For rate effects there were no test data provided and information in literature on these parameters are highly diverging. In particular the strain rate where rate effects start strongly vary: for example between 5.0e-5 s^{-1} acc. to Eibl et al. [5] and 1.0 s^{-1} acc. to Johnson et al. [9], Gamarino et al. [7]. The sensitivity study has shown that the influence on the missile result is significant (cp. Fig. 6). Because there was no better information the parameters for steel EN 1.4432 were calibrated to provided data of the 'Meppen Tests' for reinforcing steel BSt 420 and acc. to [5]. The resulting values are $\dot{\varepsilon}_0$ = 5.0e-5 s^{-1} and C = 0.0288. Although these are completely different steel grades the simulation results were good.

Numerical model of the slab simulation

Fig. 2 shows the finite element model of the second simulation with the RC slab and the load bearing steel frame. The steel frame is very stiff and experiences only negligible deformations however it was modeled in order to consider the restraint of the slab due to torque and uplift in the slab corners. It consists of simple beam elements with linear elastic material behavior. The load transfer between slab and frame is realized with contact surfaces.

The concrete is modeled with continuum elements and the rebars with truss elements. The length of the continuum elements is 0.03 m leading to five elements along the height of the slab. It was assumed sufficient to detect major cracks with that mesh refinement. The truss elements are 0.03 m long in order to be consistent to the concrete mesh. The rebars are attached (embedded) to the concrete elements. Hence bond slip relation between concrete and rebars was neglected. A multi-linear elastic plastic constitutive law using the provided uniaxial stress strain curve was applied for the rebars. For concrete the 'concrete damaged plasticity' model in ABAQUS was used. This constitutive law can consider crushing in compression, cracking in tension for multiaxial stress regime.

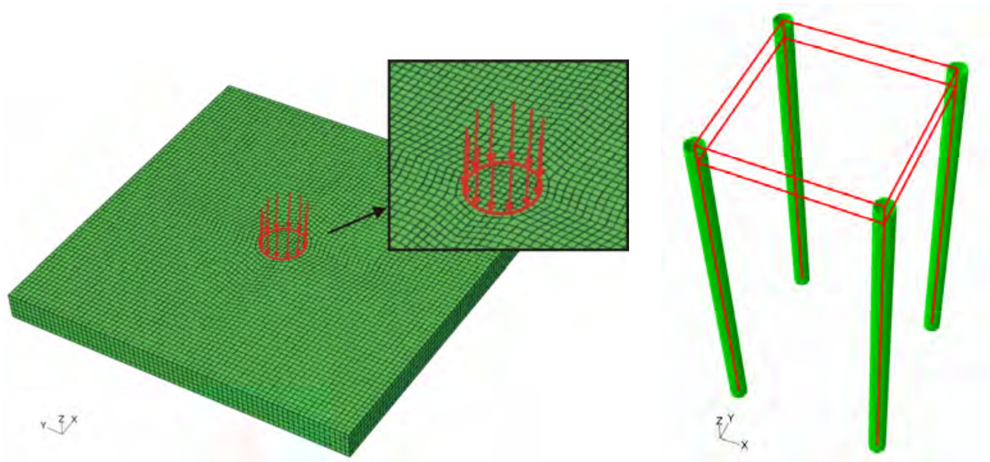

Fig.2: FE model of the simulation with the RC slab and ring-load (left)and frame were slab is supported (right)

In compression a uniaxial stress strain relation has to be provided for the 'concrete damaged plasticity' model in ABAQUS. This curve was developed using equation 63 of DIN 1045 [4] (Stress-strain curve for non-linear methods of analysis and for strain calculations) and using the mean cylinder compressive strength f_{cm}. Because the compressive strength was not reached in the simulations rate effects have a minor influence and they were neglected. The Poisson's ratio of concrete ν_c is in the range of 0.14 to 0.26 for compression stress up to 60% and tensile stress up to 80% of the corresponding strength acc. to [3]. A good approximation is 0.2 which was used in the numerical simulations.

In tension a uniaxial stress strain relation has to be provided for the ABAQUS material model, too. It is linear elastic until the tensile strength. Here the given mean axial tensile strength of concrete f_{ctm} was taken. However the sensitivity study has shown that the tensile strength has a major influence on the test result. Hence rate effects were indirectly considered by increasing the tensile strength. This can be done acc. to Model Code 90 [2]:

$$f_{ct,imp} / f_{ctm} = \left(\dot{\sigma}_{ct} / \dot{\sigma}_{ct0}\right)^{\delta_i} \quad \text{with } \delta_i = \frac{1}{10 + 6\, f_{cm} / f_{ctm}} \tag{2}$$

With the mean strengths f_{cm} = 55.2 MPa and f_{ctm} = 3.71 MPa, the exponent δ_i = 0.01, an observed stress rate in the simulations of $\dot{\sigma}_{ct}$ = 10 000 MPa/s, and the constant $\dot{\sigma}_{ct0}$ = 0.1 MPa acc. to [2] the tensile strength increases by a factor of 1.12. The approach for tensile strength increase due to rate effects has been simplified in the Model Code 2010 [3]. Now the increase only depends on the stress rate:

$$f_{ct,imp} / f_{ctm} = \left(\dot{\sigma}_{ct} / \dot{\sigma}_{ct0}\right)^{0.018} \tag{3}$$

with the observed stress rate of $\dot{\sigma}_{ct}$ = 10 000 MPa/s and the constant $\dot{\sigma}_{ct0}$ = 0.03 MPa acc. to [3] the tensile strength f_{ctm} increases by a factor of 1.26. However the preliminary calibrating simulations have shown that results are better with an even higher increase of 1.5 for the tensile strength. Hence in all calculations a factor $f_{ct,imp} / f_{ctm}$ = 1.5 has been used.

Beyond the tensile strength micro-cracks develop that lead macroscopically to a softening stress-strain response. In order to avoid mesh sensitivity the concept of Hillerborg et al. [8] that defines the energy required to open a unit area of crack as a material parameter G_F is applied in ABAQUS. This point is discussed in detail in [6]. The fracture energy G_F is the energy that is required to completely open a tensile crack of unit area and represents the area below the stress displacement curve. This curve is converted to the stress strain curve that is necessary for the finite element method as a function of the element size. According to ModelCode 90 [2] the fracture energy G_F [N/m] of concrete can be estimated:

$$G_F = G_{F0} \cdot \left(f_{cm} / f_{cm0} \right)^{0.7} \tag{4}$$

The value G_{F0} is given as a function of the aggregate size. For a medium aggregate size of 16 mm G_{F0} is 0.030 Nmm/mm² and the constant f_{cm0} is 10 MPa. With f_{cm} = 55.2 MPa eq. 5 leads to a fracture energy of G_F = 100 N/m. According to ModelCode 2010 [3] the fracture energy G_F [N/m] of concrete can be estimated with

$$G_F = 73 \cdot f_{cm}^{0.18} \tag{5}$$

Hence the new model code leads to a higher fracture energy of G_F = 150 N/m. In both Model Codes there is no information on strain rate effects on fracture energy. It is stated that "the information available regarding the effect of stress or strain rate on the fracture energy is too incomplete to be included in this Model Code". However the preliminary calibration simulations have shown that a much better fitting to test results is gained, when the fracture energy is increased. A multiplication of the fracture energy by a large factor of 3.8 showed the best test results. This factor was used in all simulations.

Simulation results of missile impact

Fig. 3 shows the result of the first simulation with the missile and the rigid target. The missile has shortened from originally 2.0 m to 1.19 m in the simulation and to 1.14 m in the experiment. The length of the uncrushed region was 0.99 m in the simulation and 0.95 m in the experiment. The shape of the crushed or folded region is very similar in the simulation and in the experiment, too. Hence the response of the missile can well be predicted by the finite element simulation. The history of contact forces between the missile and the rigid target was applied as load function in the second simulation with the slab and the frame.

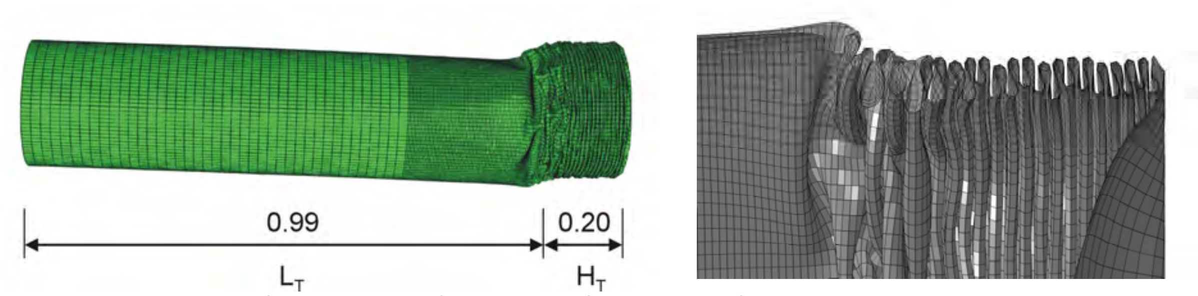

Fig.3: Deformed missile after impact (left) and detail of crashed zone (right bottom)

Simulation results of reinforced concrete slab

Fig. 4 shows the history of deflection at the centre of the slab from the simulation in comparison to the experiment. It is obvious that the model response is slightly too stiff. While the maximum deflection is 200 mm in the simulation, it is 280 mm in the experiment and in the simulation it occurs at 10 ms while it appears in the experiment at 12 ms. However in consideration of the highly nonlinear model with material softening and in comparison to all other participants with predictions between 4 and 100 mm the selected modeling can well predict the order of magnitude.

Fig. 5 shows the maximum strain in the front and rear reinforcement of the simulation. At the front rebars there was no plastification in the experiment like in the numerical simulation. The maximum rebar strain at the rear side in the VTT experiment was acc. to the published test data 3.75% while the maximum strain in the simulation was 3.8%. At the experiment and in the simulation there was almost no significant visible concrete damage (not shown). Hence it was possible to well predict the mechanical behavior of the slab in terms of potential rebar and concrete damage/failure, too.

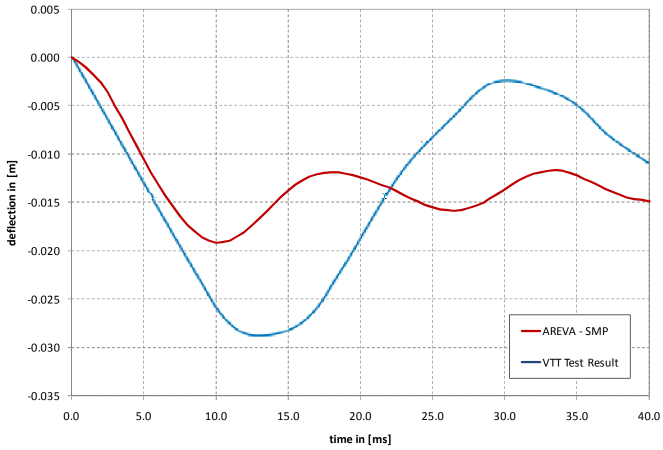

Fig.4: Deflection history at the center of the slab in comparison to VTT test result

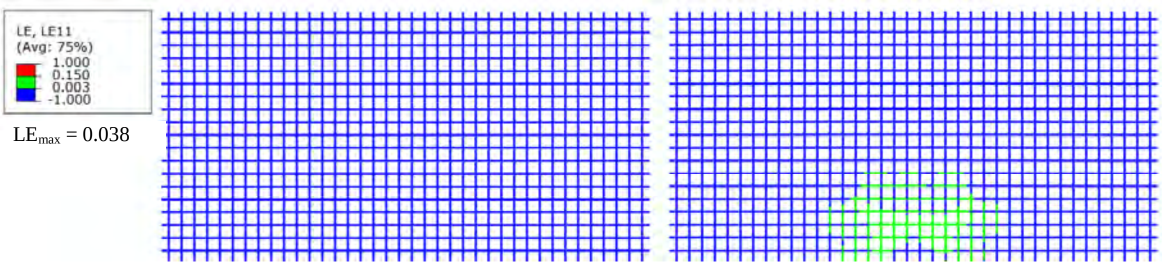

Fig.5: Reinforcement of half slab at front (left) and rear side (right) at 10ms; blue is elastic, green plastic, red failure

Parameter variations

In the diagrams in fig. 6 and 7 are some of the sensitivity studies shown that have been performed during the benchmark project. Acc to eq. 1 the strain rate effect is defined in the model with the two parameters $\dot{\varepsilon}_0$ and C. As mentioned above the parameter $\dot{\varepsilon}_0$ describes the strain rate where effects start to occur. In literature one can find values between 1.0e-5 s^{-1} and 1.0 s^{-1} for steel. For a better understanding only the parameter $\dot{\varepsilon}_0$ was varied between these two values while the parameter C was kept constant. Of course the two parameters C and $\dot{\varepsilon}_0$ should be changed in combination however the studies have shown that $\dot{\varepsilon}_0$ plays a major part because when that strain rate is not reached in the simulation no rate effects occur at all. The shortening of the missile as a function of the parameter $\dot{\varepsilon}_0$ is presented in fig. 6 (left). It can be seen that this parameter greatly influences the simulation result: the shortening varies between 0.80 m and 0.93 m.

The diagram in fig. 6 (right) depicts the maximum deflection of the slab versus the ratio $G_{F,imp}/G_F$. $G_{F,imp}$ is the fracture energy that is increased due to rate effects. In this example the factor was varied between 1.0 and 4.5. The significant influence of that factor can clearly be seen: while the deflection with no increase of the fracture energy is more than 40 mm the deflection is about two times smaller for a fracture energy that was increased by a factor of 4.5. In fig. 7 (left) the influence of the tensile strength at impact loading $f_{ct,imp}$ on the maximum deflection of the slab is shown. Again the influence is significant: while the deflection with no increase of the tensile strength is 33 mm the deflection is only 17 mm for a tensile strength that was increased by a factor of 1.6.

When using the explicit solver in time integration simulations the time increment must be smaller than the stability limit of the central-difference operator in order to reach stable results. The minimum time increment $t_{inc,min}$ is a function of the smallest transit time of a dilatational wave across any of the elements in the mesh [1]. Many simulations that have been performed by the authors have shown that the time increment has an influence on simulations when softening is apparent in the model. Fig. 7 (right) shows the maximum deflection of the slab as a function of the selected time increment. While the time increment has almost no influence on the missile simulation it is significant for the slab simulation. The maximum deflection varies by approximately 20% when a time increment is chosen that is ten times smaller than the minimum time increment. This is typical in simulations where material softening is present.

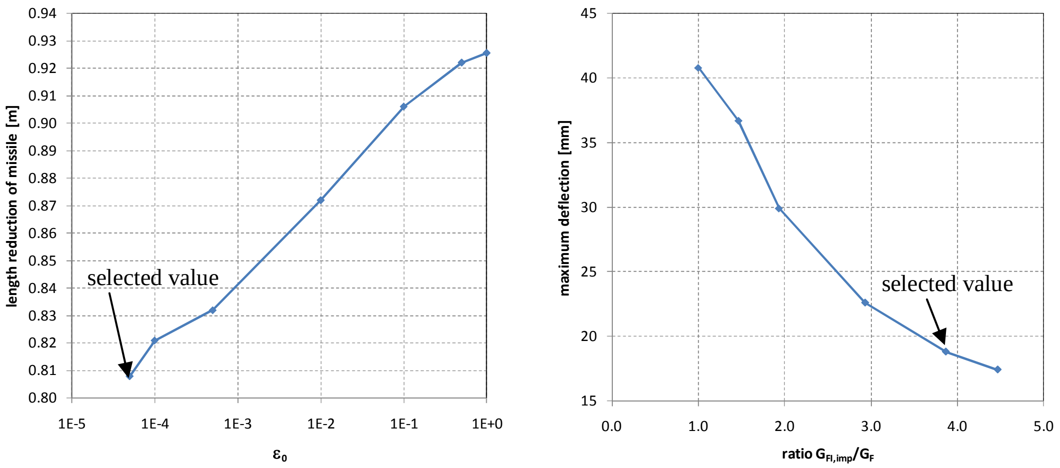

Fig.6: Influence of parameter $\dot{\varepsilon}_0$ (left) on the missile shortening and fracture energy (right) on the slab deflection

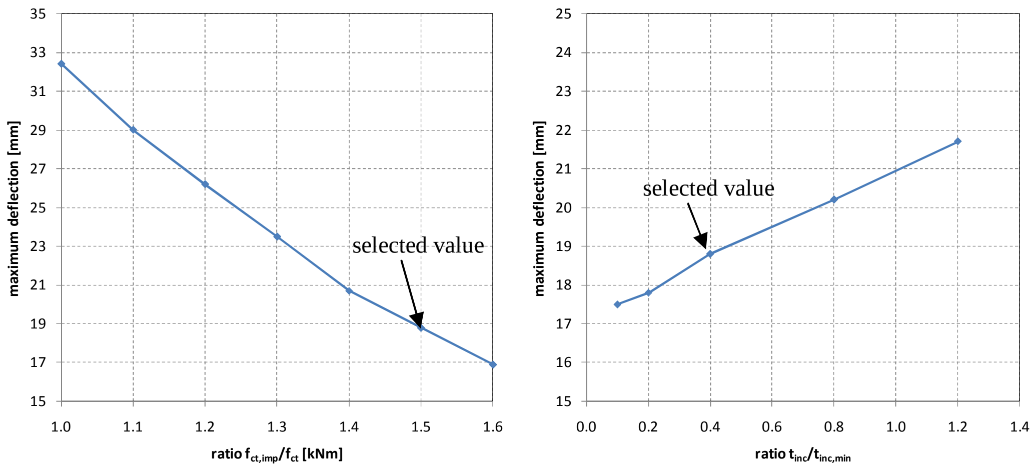

Fig.7: Influence of tensile strength (left) and time increment (right) on the slab deflection

NUMERICAL SIMULATION OF SLAB WITH PUNCHING FAILURE

Test setup and numerical model

A 2.1 m x 2.1 m x 0.25 m reinforced concrete slab was loaded with a missile that had an initial velocity of 120 m/s. The slab had longitudinal and transversal reinforcement. The concrete had an approximate compressive strength of f_{cm} = 60.0 MPa and a tensile strength of f_{ctm} = 4.04 MPa. The slab was again supported along a 2 m side square on the front and rear side. The missile consisted of a pipe with a massive end cap and had a lightweight concrete filling. It had a mass of 47 kg, a length of 0.64 m, and a diameter of 168 mm. A combined 3D model acc. to fig. 8 with the missile, the slab, and the frame was necessary because the deformation of the missile was small in comparison to the target ('hard impact'). The missile had an initial velocity and the forces are transferred to the slab by contact surfaces. Again no simplification due to symmetry was applied.

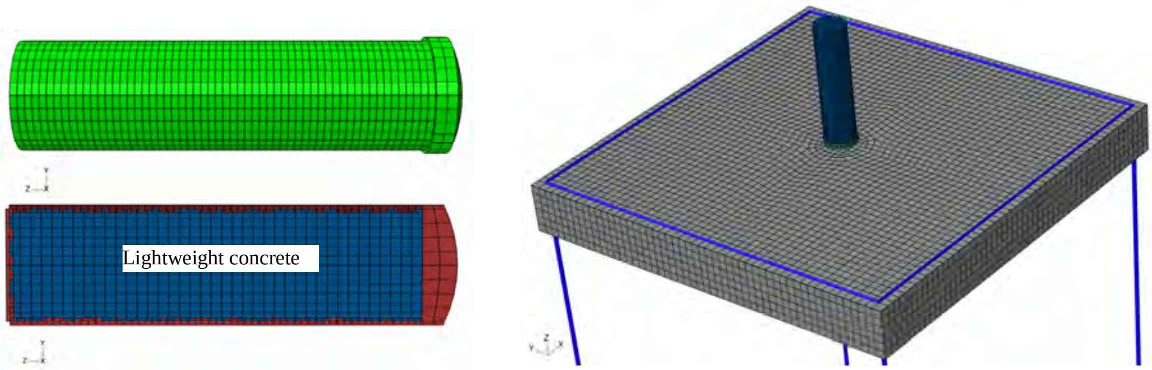

Fig. 8: Model of the hard impact simulation with the missile (left top), section (left bottom), and total model (right)

Simulation results of reinforced concrete slab

In the experiment the slab was completely and very quickly perforated by the missile. The slab only had a punching hole that was slightly larger than the diameter of the missile. Apart of the punching hole the slab deformations and the reinforcement strains were extremely small. In the simulation there was no erosion (element deletion) in the material model considered because the 'damaged plasticity model' in ABAQUS does not support that option in the current version. Hence the missile could not perforate the slab and because elements have a specific residual resistance at even large strains for numerical reasons the deformations of the slab were highly overestimated. Fig. 9 shows the deformed model at the end of the impact simulation. With this model it was not possible to predict the residual velocity of the missile, too. However this deficiency was known to the authors and a further interpretation of the results was done as follows: Fig. 10 shows the reinforcing strain at the front and rear side of the model after only 2 ms. On both sides the strains are larger than 15% which is the failure strain of the reinforcement. Therefore the inner concrete cone is completely disconnected from the slab and forces can only be transmitted by shear forces inside the concrete. Because the residual velocity of the missile at this time step was approximately 32 m/s a complete perforation of the slab was predicted in the benchmark project.

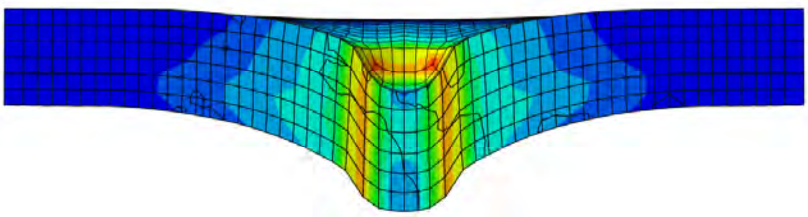

Fig.9: Section of the slab at the end of the simulation;

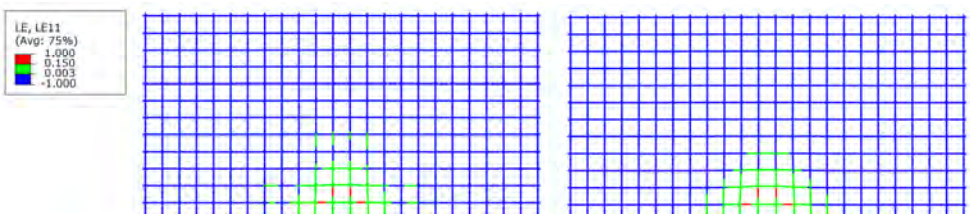

Fig.10: Reinforcement at front (left) and rear side (right) after 2ms; blue is elastic, green plastic, and red failure

CONCLUSION

Two impact tests, a soft impact and a hard impact, on reinforced concrete slabs have been performed at VTT in Finland. We were one of the participants who have predicted the test results by numerical simulations using the finite element method. The test results were not known to the participants at the time the simulations were performed. In our numerical model different constitutive equations have been used for concrete, reinforcement steel, and missile steel. Because many of the necessary material parameters could not precisely be defined by provided material data or by literature sensitivity studies have been performed prior to the final simulations. Especially rate effects on missile steel, concrete tensile strength and fracture energy have a significant influence on the simulation results in these tests as the presented parameter studies show in this paper. Hence a calibration of the numerical models was necessary and that was done by using the 'Meppen Test' results that were performed in the 80s. The parameters that were finally used in the simulations and that are based on the calibration are presented in this paper.

Although the materials and the size of the missiles and slabs are distinctly different in the 'Meppen Tests' compared to the VTT tests our performed numerical simulations can well predict the mechanical behavior of the missiles and the slabs. The deformational behavior and the residual length of the missile at the soft impact test could almost precisely be predicted by the simulations. The response in terms of slab deformations, elastic and plastic strains in reinforcement could well be predicted, too. In the second test with a hard impact the slab was completely perforated by the missile in the experiment. Because erosion (element deletion) was not considered in the model the missile could not perforate the slab in the simulation. Hence the deformational behavior of the slab was overestimated and the residual velocity of the missile could not be predicted. However this model deficiency was known to the authors and by interpretation of the numerical results it was possible to reliably predict the failure mechanism in terms of a complete perforation mainly based on the strains in reinforcement.

In order to enhance the results of numerical simulations for impacts and to decrease the necessary calibrations it is in particular necessary to further investigate the influence of rate effects on steel in general and on concrete fracture energy. Furthermore the well known influence of the mesh refinement and the in this paper shown influence of the time step when material softening is present have to be further investigated.

REFERENCES

[1] ABAQUS Software, Dassault Systems Simulia Corp., Providence, USA, 2010.

[2] CEB-FIP Model Code 1990: Design Code, Comité Euro-International du Béton, Thomas Telford Services Ltd. 1993.

[3] Model Code 2010 – First complete draft, Volume 1, fib Bulletin No. 55, CEB-FIP 2010

[4] DIN 1045-07, Plain, reinforced and prestressed concrete structures, part 1: design and construction, DIN Deutsches Institut für Normung e.V., BerlinJuly 2001

[5] Eibl, J., Häußler-Combe, U., "Sonderdruck aus dem Beton Kalender – Baudynamik", Ernst & Sohn Verlag 1997.

[6] Gamarino, M., Avalle, M., Vadori, R., Modelli della sensibilità allo strain-rate identificatione dei parametri caratteristice di lamiere di acciaio, AIAS (Associazione Italiana per l'Analisi delle Sollecitazioni) , XXXII Convegno nazionale AIAS, 2003.

[7] Hillerborg, A., Modeer, M., Petersson, P.E., Analysis of crack formation and crack growth in concrete by means of fracture mechanics and finite element, Cement and Concrete Research, Vol. 6, 1976, pp. 773-782.

[8] Johnson, G.R., Cook, W.H., "Fracture characteristics of three metals subjected to various strains, strain rated, temperatures and pressures", *Engineering Fracture Mechanics* Vol. 21, 1985, pp. 31-48.