Transactions, SMiRT-26
Berlin/Potsdam, Germany, July 10-15, 2022
Special session

USER MODIFIED ABAQUS CDP MATERIAL MODEL DEVELOPMENT IN IMPACT

Kim Calonius[1], Alexis Fedoroff[2]

[1] Senior Scientist, VTT Technical Research Centre of Finland Ltd., Finland (kim.calonius@vtt.fi)
[2] Senior Scientist, VTT Technical Research Centre of Finland Ltd., Finland

ABSTRACT

One of the most difficult tasks in VTT IMPACT project has proven to be the simulation of hard missile (or projectile) impact benchmark tests, especially if one considers accurate prediction of the residual velocity as the most important measure of success. The difficulty comes inherently from the shortages of classical FEM in describing macroscopic cracking and fragmentation of solid bodies. Hence the accuracy of hard missile impact simulations depends on how accurately not only the material constitutive relations are chosen, but also how well multibody dynamics can be implemented, including contact algorithms, timely tearing and breaking of ductile materials and physically plausible fracture and fragmentation of quasi-brittle materials.

The concrete model under consideration is the Concrete Damaged Plasticity (CDP) model available in the Abaqus Finite Element (FE) software with appropriate modifications applied via a VUSDFLD user subroutine. The IRIS 2010 punching tests is considered as the validation benchmark for this study. The main aim is not yet to simulate the test as accurately as possible, but to find out the effect of all the possible model and analysis parameters. The model is found to be relatively stable and insensitive to many of those parameters, but great care should be taken in defining the material parameters and time incrementation for the simulation.

VTT USER MODIFIED ABAQUS CDP MODEL

The actual concrete model under consideration is the Concrete Damaged Plasticity (CDP) model available in the Abaqus FE software (2019). Appropriate modifications to the model have been applied via the VUSDFLD user subroutine. The modifications to the constitutive behavior consist in additional confinement dependency of isotropic hardening/softening behavior in uniaxial compression and strain rate dependency of isotropic softening behavior in uniaxial tension. The extension to the classic FEM formulation consists in the introduction of an element removal algorithm based on a condition on shear strain measure together with a condition on triaxial confinement (Fedoroff et al., 2017). The exact description of the user modified model can be found in articles by Fedoroff, et al. (2019) and by Fedoroff & Calonius (2020). The objective is first to calibrate the physical model parameters using material test data provided from various compressive, tensile and triaxial concrete tests, (Calonius, et al., 2019), and subsequently to carry out a sensitivity study of internal model parameters that cannot be calibrated from material tests. Various experimental benchmark test responses have previously been used as reference.

IRIS 2010 PUNCHING TEST

In the work presented here, the performance of the user modified CDP model is tested against the results of IRIS 2010 benchmark tests P1, P2 and P3. That exercise consists of three almost identical punching tests, where a hard missile is shot onto a 25 cm thick reinforced concrete slab. The missile is a stainless steel tube filled with non-reinforced light-weight concrete and it is shot against the target slab using pressurized air. The target slab is held in place by a steel frame, which can be assumed stiff compared to the stiffness of the concrete slab. The free span of the target slab is 2m by 2m, and the steel frame connection to the slab can be assumed to be a hinged one. Detailed test description can be found in (Vepsä, Saarenheimo, Tarallo, Rambach, & Orbovic, 2012).

The main result in those tests is the final residual velocity of the missile after it has perforated the target slab. The initial impact velocity and the final residual velocity of the missile is listed in Table 1. The initial velocity of 135 m/s is used in all the FE simulations presented below.

Table 1: Results of IRIS 2010 punching case tests.

Test Results [m/s]	IRIS P1	IRIS P2	IRIS P3	average	st. deviation
initial velocity	135.9	134.9	136.5	135.8	0.81
residual velocity	33.8	45.3	35.8	38.3	6.14

Finite Element Model

A quarter FE model employing the symmetry in both in-plane directions (X and Z in Figure 1) of the slab is used for the simulations. The nonlinear dynamic simulations are carried out using Abaqus/Explicit with explicit central difference time integration. The concrete slab is bounded by steel channels in order to withstand the contact pressure from the steel rods on both sides of the slab. These steel rods are attached to the frame that holds the slab in place. In these simulation models, the steel frame is not modeled at all. The steel rods, which are assumed to simulate hinged boundary conditions, are modeled as rigid bodies and they are rigidly fixed to the frame of reference.

The concrete of the slab is modeled using eight-node reduced integration elements with Abaqus default distortion and hourglass control (C3D8R in Abaqus nomenclature). The characteristic element length is 10 mm, which means that there are 25 elements over the thickness of the slab. The steel reinforcement is modelled as stringers using two-node linear beam elements (B31 in Abaqus nomenclature) that are tied node-to-node to the C3D8R elements. They are located in the same level like as rebar web (as seen in Figure 2), whereas in reality they are placed in different levels, the horizontal rebars being closer to the slab surface. The concrete cover is 20 mm.

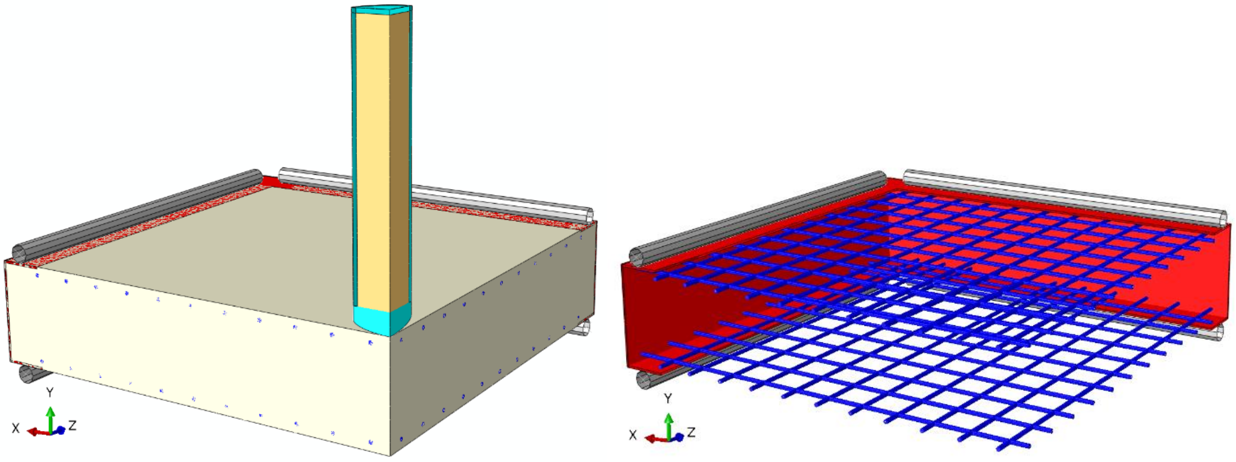


Figure 1. FE model (missile and concrete are removed in the right-hand side image).

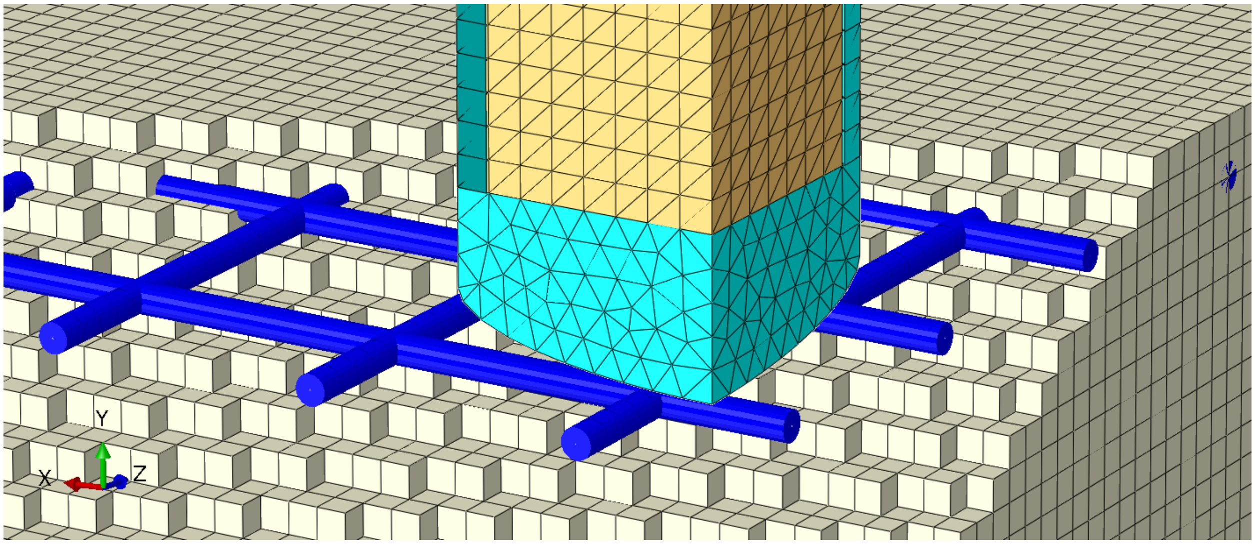


Figure 2. FE model detail showing the element meshes (with some concrete elements being removed).

SENSITIVITY STUDIES

Sensitivity to time increment size

The explicit procedure integrates through time by using many small time increments. The central difference operator is conditionally stable, and the stability limit for the operator (with no damping) is given in terms of the highest eigenvalue in the system as

$$\Delta t \leq \frac{2}{\omega_{max}} \tag{1}$$

In this case, this stable time increment calculated by Abaqus is 3.14e-7 s, which is for the missile. This can also be defined as the time for the elastic stress wave to pass the characteristic element size. If the concrete element size of 10 mm is chosen as the characteristic element size, we get 2.49e-6 s for the slab - a much larger stable increment. As there is some bulk viscosity applied, the stable increment is actually slightly smaller than that. Furthermore, as the concrete elements get distorted during the analysis, at some phases, they get more critical than any missile element. The time integration is fully automatic unless the user decides to define a constant or minimum and/or maximum increment size. In this sensitivity study, mainly different maximum increment sizes from 1e-6 s to 1e-10 s are defined. When 1e-6 s maximum is used, the average increment size for the whole analysis duration (10 ms) is 2.85e-7 s. In other cases, Abaqus uses the defined maximum size. When a constant increment size of 1e-6 s is chosen, the analysis terminates due to numerical problems (the ratio of deformation speed exceeds the stress wave speed). When a constant increment size of 2e-10 s is chosen, the analysis terminates due to the same reason. However, the underlying fundamental reasons are different. In the first case, the reason is truncation error, in the second case round-off error - even though all these analyses are run in double precision.

Figure 3 shows in the same graph the main result, which is the residual velocity of the missile, and two factors which give some indication on the stability of the analysis. Even though an analysis is completed successfully, the material constitutive routines may fail to converge in many increments. In some increments, the number of points that fail to converge may be relatively high. The figure shows that the most stable increment size is approximately 1e-8 s. This has been used in all the other sensitivity studies for this impact case. Residual velocity means the velocity of the missile after it has perforated the slab. The average residual velocity in the IRIS punching cases was 38.3 m/s. Time increment size of 1e-9 s yields a closest simulation result to that test result. However, as no analytic solution exists for this case and the FE model is very complicated, the most stable FE solution is sought for. Different model parameters need to be calibrated for the model to better match with existing test results.

Figure 4 shows energy components as function of time in simulations with different user-defined maximum time increment sizes (1e-6 s, 1e-8 s and 4e-10 s). Artificial strain energy is associated with constraints used to remove singular modes (such as hourglass control). Total strain energy is the sum of all the energy dissipated by different types of deformation (including artificial strain energy and energy dissipated by distortion control). One can see that with 4e-10 s total strain energy exceeds total energy of the system, which is unphysical. The energy components with 1e-6 s and 1e-8 s are very close to each other, as well as the simulated behaviour of the slab and the missile, which suggests that in this case maximum time increment of 1e-6 s can be chosen and the stable increments calculated by Abaqus practically lead to the best possible result with the model.

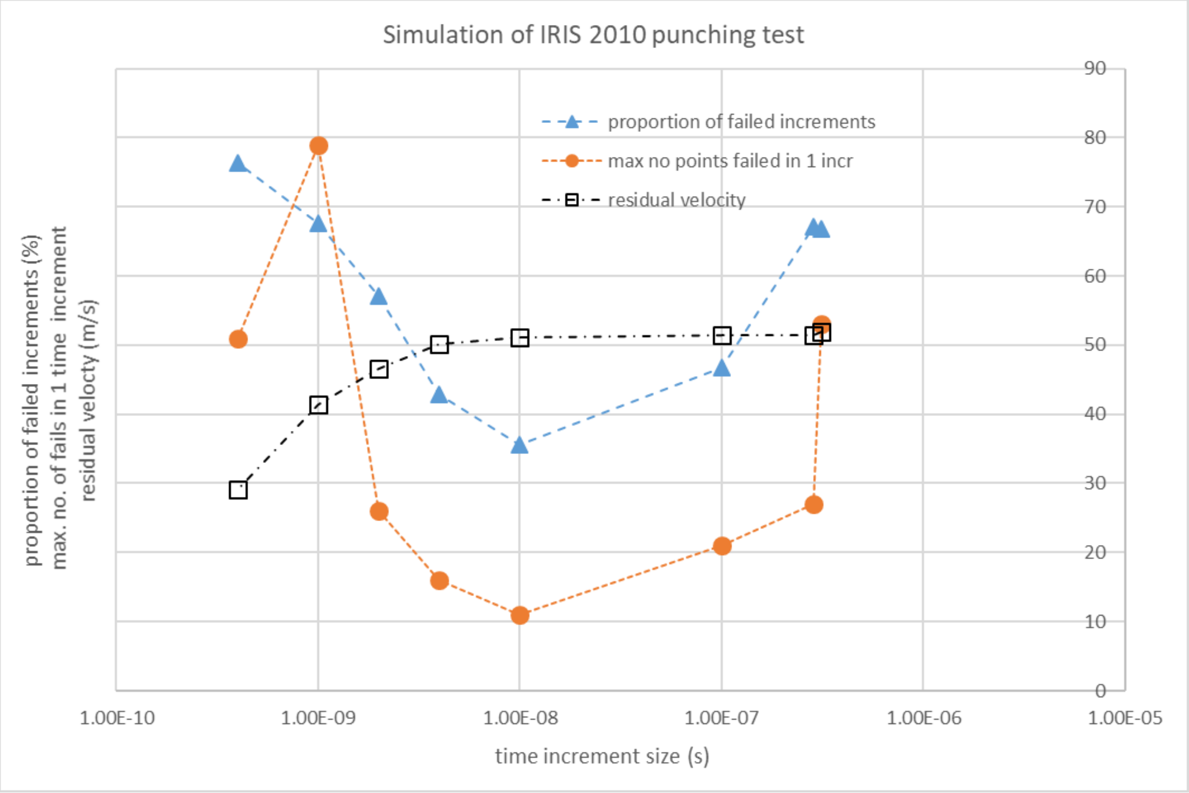


Figure 3. Proportion of failed increments (%), maximum number of failed points in one increment and residual velocity (m/s) as functions of time increment size (s; logarithmic). Automatic time incrementation (except the rightmost values).

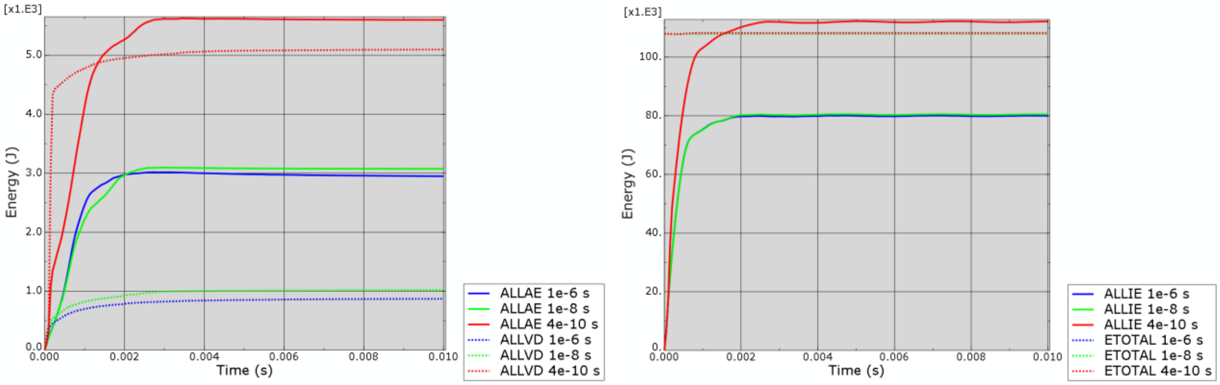


Figure 4. Energy components as function of time in simulations with different user-defined maximum time increment sizes. On the left: Artificial strain energy (ALLAE) and viscous dissipation (ALLVD). On the right: Total strain energy (ALLIE) and total energy (ETOTAL).

The effect of strain rate dependency for concrete (only in tension) with different time increment sizes is also studied. As expected, when the strain rate dependency is removed, the residual velocity is higher, and in this case, further away from the test results. This effect gets stronger as the time increment size is decreased. The damage mode of the slab, however, resembles more to the one observed in tests. This indicates that the dynamic increase factor (DIF) in tension should be decreased, but the punching capacity of the slab model should be increased by some other means.

Sensitivity to mesh size

In order to check the sensitivity to mesh size, the slab was also modelled with elements of size 5 mm and 25 mm. Based on the sensitivity study for the time increment size, 1e-8 s was chosen as the maximum time increment size. Figure 5 shows the missile velocity as a function of time for different mesh sizes, which are shown in the legend. Some material parameters are dependent on the element size. However, in this case, that does not play any significant role. In cases denoted by (*) in the legend, the material parameters have been generated for 10 mm elements even though the element size is either 5 mm or 25 mm. Interestingly, both 5 mm and 25 mm models yield lower residual velocity than the 10 mm model. However, the model is not very sensitive to the mesh size. The residual velocity varies only between 46 m/s and 50 m/s. The CPU time is approximately linearly dependent on the number of elements in the model.

Table 2 shows the deformed model shape and SDV4 as a contour plot for three different mesh sizes at three different time instants. Exactly the same material parameter values have been used these three simulations. SDV4 is solution-dependent state variable which is a measure of pure shear strain as a quotient of pure shear strain and cut-off threshold (one of the internal variables). If the quotient is larger than 2, the element is flagged for removal. The damage mode is similar in each case. Figure 6 shows the vertical and horizontal mid-sections of the slab after IRIS 2010 P1 test. The FE simulations have not produced all the damage modes observed in the test. There is an initial formation of a flat shear cone and the back surface rebars finally tear out the concrete cover from a large area. By calibrating material and model parameters, this FE model is able to better produce those modes, but currently the residual velocity has been too high in those cases.

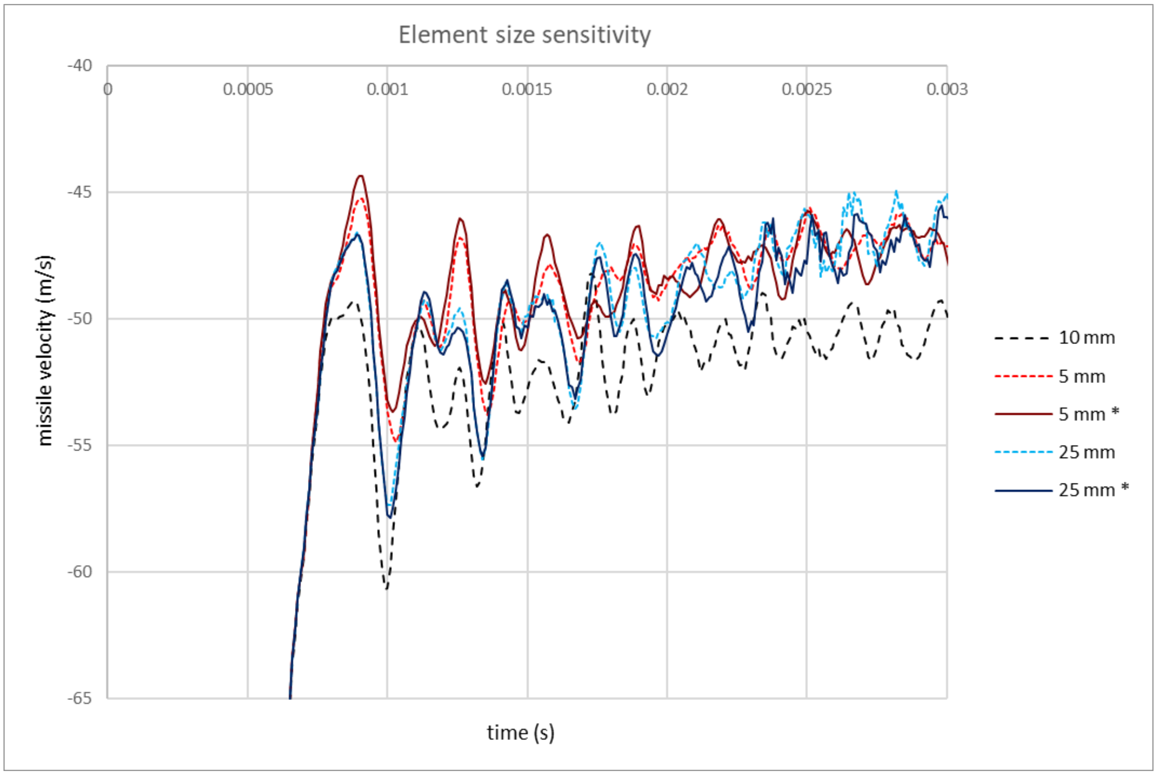


Figure 5. Effect of element size of the concrete slab to the projectile velocity (m/s). The material parameters have been generated for element size used in the model, except for cases denoted by (*), where they have been generated for 10 mm elements.

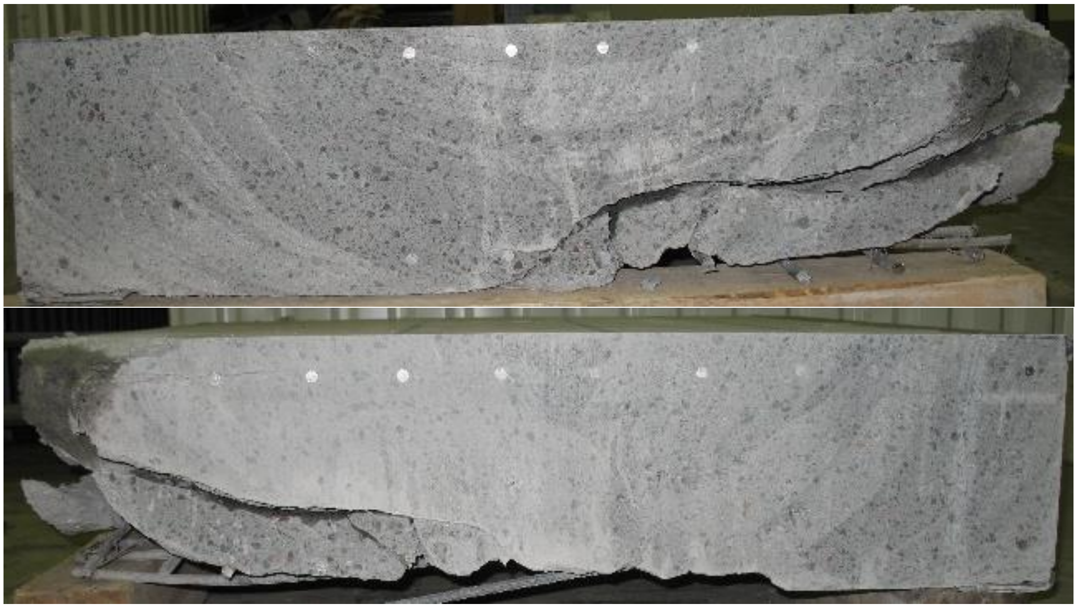
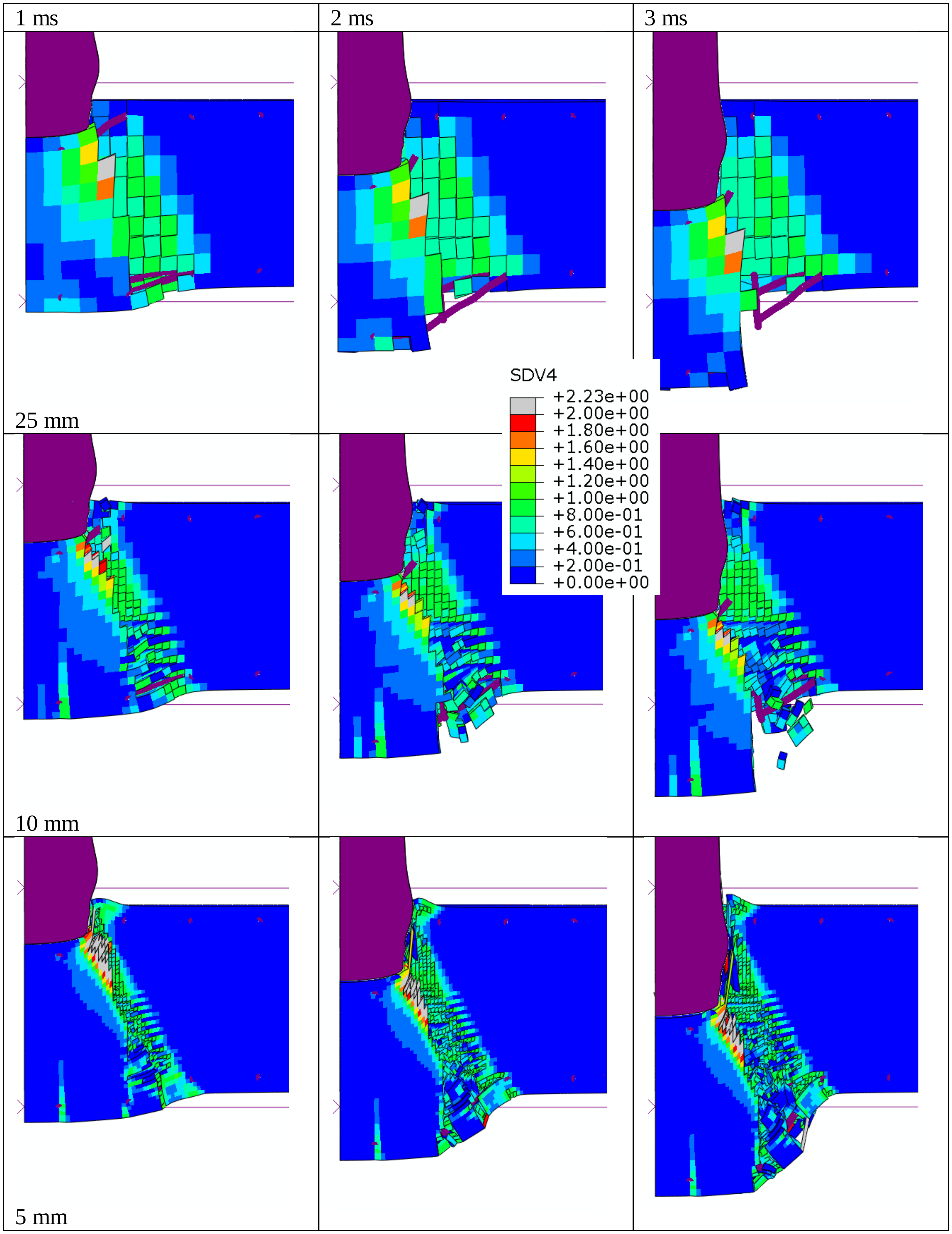


Figure 6. Vertical and horizontal mid-sections of the slab after IRIS 2010 P1 test.

Table 2: SDV4 at time instants 1 ms, 2 ms and 3 ms after impact for different element sizes; 25 mm, 10 mm and 5 mm.



Sensitivity to internal variables of user subroutine

There are three internal model parameters that this study is focusing on. The first is the choice of weight coefficients when computing the actual confinement stress as a weighted average of the current time frame value of confinement stress and the maximum value of confinement stress. The second is the threshold to choose with respect to the shear strain measure in the element removal algorithm. The third is the threshold to choose with respect to the confinement stress measure in the element removal algorithm. It has been shown in previous studies, (Fedoroff, et al., 2019), that these parameters have influence on the simulation response. A parameter value grid of 2x3x3 has been chosen in this study to carry out the sensitivity analysis. The response values that are monitored and compared to the experimental results are the residual velocity as well as the overall failure mode of the slab. The alternative parameter values are denoted by A and B for the first parameter and by A, B and C for the other two parameters. The baseline case is AAA, where all the parameters have the first alternative value that has mainly been used in all the other sensitivity studies presented here and in recent simulations of different tests.

Figure 7 shows the missile velocity as a function of time in all the cases for the first 3 ms. Tables 3 and 4 show the final residual velocities. The relative error (%) with respect to the experimental value is shown in parentheses. The parameters start to affect the missile behaviour only after approx. 0.7 ms, because they control the element removal, which is describing the concrete fracture that is known not to take place in the very early phase of the impact. All the simulations overestimate the residual velocity. Variable combinations ACB and ACC give results that are closest to the test results. The highest and lowest residual velocities are given by cases BAB and ACB, respectively. The confinement weight has the largest effect.

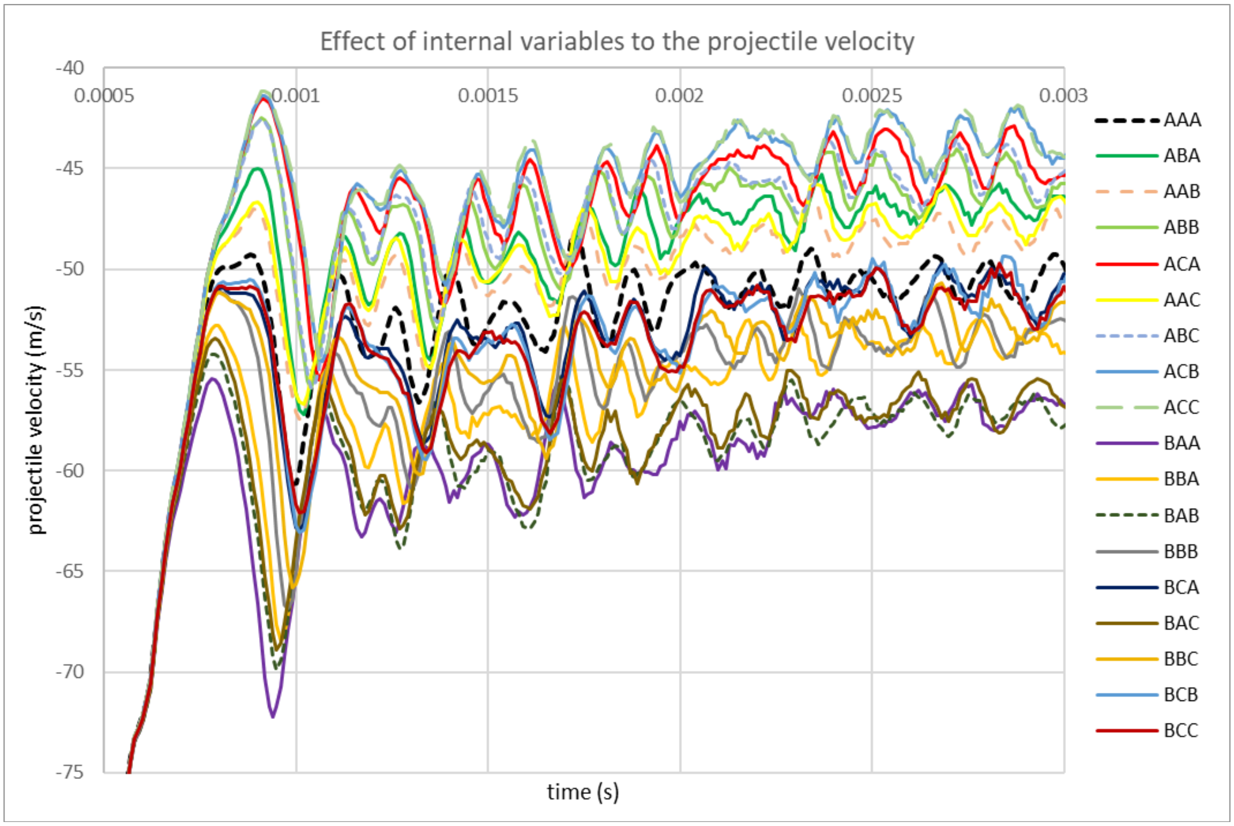


Figure 7. Effect of internal variables to the projectile velocity (m/s).

Table 3: Simulation results for confinement weight 0.4 / 0.6 (parameter 1 = A).

Final residual velocities [m/s] (relative error [%])			
confinement weight 0.4 / 0.6 (A)			
Confinement threshold coeff. **Strain threshold coeff.**	0.75 (A)	0.6 (B)	0.55 (C)
1.0 (A)	50.13 (31)	47.96 (25)	47.15 (23)
1.5 (B)	46.64 (22)	45.58 (19)	45.09 (18)
2.0 (C)	44.62 (17)	43.25 (13)	43.38 (13)

Table 4: Simulation results for confinement weight 0.6 / 0.4 (parameter 1 = B).

Final residual velocities [m/s] (relative error [%])			
confinement weight 0.6 / 0.4 (B)			
Confinement threshold coeff. **Strain threshold coeff.**	0.75 (A)	0.6 (B)	0.55 (C)
1.0 (A)	56.48 (47)	56.87 (48)	56.22 (47)
1.5 (B)	53.26 (39)	53.15 (39)	52.26 (36)
2.0 (C)	50.67 (32)	50.79 (33)	51.19 (34)

Sensitivity to other model or analysis parameters

The number of cores used for the computation has only a minimal effect on the results, but using more cores clearly decreases the CPU time. The amount of data Abaqus writes during the analysis has only a very minimal, in any, effect on the results. It has also been checked, whether some unforeseen small changes in the computational environment might affect the outcome of the simulation. This has been done by running twice exactly the same simulation in the same server and in the same way in every respect twice two weeks apart from each other. There were no differences in the results. The different hourglass control approaches yielded very similar results.

The basic case has also been computed with a complete model (as a comparison to the quarter model), where also reinforcement is modelled in more detail. The results of these two models are very similar. Thus, a quarter model is adequate for this type of direct impact with distinct punching behaviour.

CONCLUSION

Validation of a complex material model, including extensions to classical FEM formulation, requires validation against numerous different benchmark cases. The IRIS 2010 punching tests can be considered as the baseline for validation benchmarks. Other benchmarks have also been used although not presented here. One possibility for the validation process is to study the sensitivity of the simulation response with respect to internal model parameter variation. If one can find, as a result, a set of parameter values that are "universal" with respect to all benchmark cases, then the validation process can be considered as successful.

Altogether some 50 simulations have been conducted for the punching case. All the simulations with different internal parameter values of VUSDFLD user subroutine overestimate the residual velocity. The effect of internal parameters and their variation investigated here to the behaviour of the slab and consequently to the missile is not large.

The case is not very sensitive to the mesh size, which indicates that the material model is suitable for different types of cases. Concerning the material models, only the tensile softening behaviour of concrete is mesh size dependent. More complicated cases (inclined punching, shear reinforcement etc.) would probably be more sensitive to the mesh size.

As expected, the analysis is sensitive to the time increment size. Increments from 3e-7 s to 1e-8 s yield close results to each other, but smaller increments yield lower residual velocities (slab has more capacity against punching). Round-off errors start to dominate and the analysis will terminate with really low increments. CPU time is linearly dependent on the time increment size. The most stable time increment size in this case is approx. 1e-8 s. The stable time increment defined by Abaqus is much larger (approx. 3e-7 s). It gives nearly the same result, and in this case - with hindsight - it would have been an adequate choice. The sensitivity to the time increment size and to the strain rate dependency are dependent on each other. The smaller the time increment size is the higher the strain rates are in the material - even unrealistically high - and thus also the stronger the effect of the strain rate dependency on the result.

On a general level, the model is stable and not highly sensitive to any model or analysis parameters. In the future, the sensitivity study for the internal parameters should be performed further in a way that the best solution is the average solution and closer to the experimental test results than in the current study. Material test results should be used even more and the model parameter values should be defined in more detail. Dynamic increase factors should be studied more and be based on the material tests done to the actual material used in the impact tests. They should be considered as internal variables for sensitivity study. Quadratic bulk viscosity is also dependent on the strain rate and should be studied in more detail.

REFERENCES

Abaqus Manuals. (2019). SIMULIA. Dassault Systèmes.

Calonius, K., Fedoroff, A, Kolari, K, Kouhia, R and Vilppo, J., (2019). “Calibration of Abaqus CDP model parameters”. *Proceedings of the 32nd Nordic Seminar on Computational Mechanics*, Oulu.

Fedoroff, A. and Calonius, K., (2020). “Using the abaqus CDP model in impact simulations.” *Rakenteiden mekaniikka (Journal of Structural Mechanics)*, pp. 53(3), 180-207.

Fedoroff, A., Calonius, K. and Kuutti, J. (2019). Behavior of the abaqus CDP model in simple stress states. *Rakenteiden Mekaniikka (Journal of Structural Mechanics)*, pp. 52(2), 87-113.

Fedoroff, A., Kuutti, J. and Saarenheimo, A. (2017). “A physically motivated element deletion criterion for the concrete damage plasticity model.” *Busan, SMiRT-24 transactions.*

Vepsä, A.;Saarenheimo, A.;Tarallo, F.;Rambach, J.-M.;& Orbovic, N. (2012). “Impact tests for IRIS 2010 Benchmark Exercise.” *Journal of Disaster Research*, 7(5), 619-628.