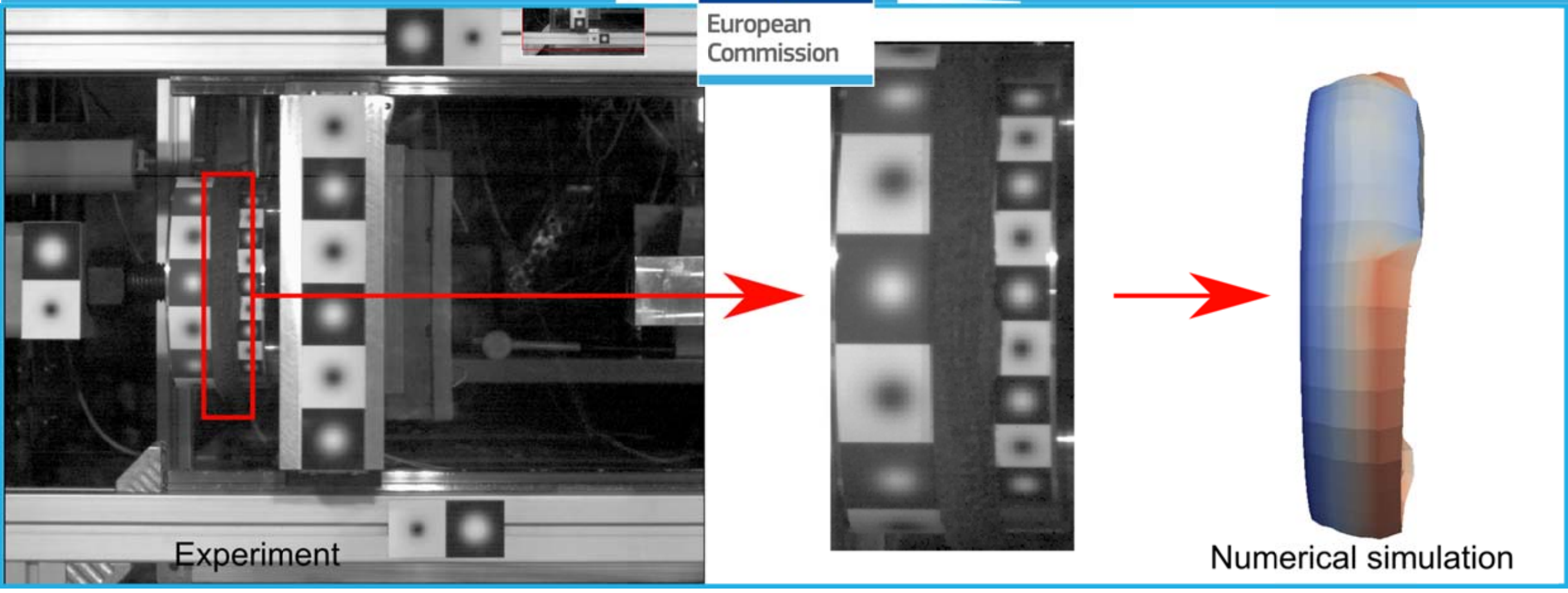

JRC TECHNICAL REPORTS

Numerical material modelling for the blast actuator

Administrative Arrangement Nº JRC 32253-2011 with DG-HOME
Activity A5 - Blast Simulation Technology Development

Martin Larcher
Georgios Valsamos
George Solomos

2013

Report EUR 26407 EN

European Commission
Joint Research Centre
Institute for the Protection and Security of the Citizen (IPSC)

Contact information
Martin Larcher
Address: Joint Research Centre, Via Enrico Fermi 2749, TP 480, 21027 Ispra (VA), Italy
E-mail: martin.larcher@jrc.ec.europa.eu
Tel.: +39 0332 78 9563

http://europlexus.jrc.ec.europa.eu/
http://www.jrc.ec.europa.eu/

This publication is a Reference Report by the Joint Research Centre of the European Commission.

JRC86348
EUR 26407 EN
ISSN 1831-9424
ISBN 978-92-79-35056-6
DOI 10.2788/52398

Luxembourg: Publications Office of the European Union, 2013

Printed in Italy

CONTENTS

1 Introduction

The European Laboratory for Structural Assessment is currently designing a new test facility that should model the loading of structures by air blast waves without using explosives. The idea is to use a fast actuator that accelerates a mass, which impacts the structure under investigation (Figure 1). Obviously, more such masses and actuators should be employed simultaneously for loading a structural element, e.g. a column, along its entire length. To obtain a good agreement with pressures from relevant air blast waves an elastic material should be placed between the impacting mass and the structure.

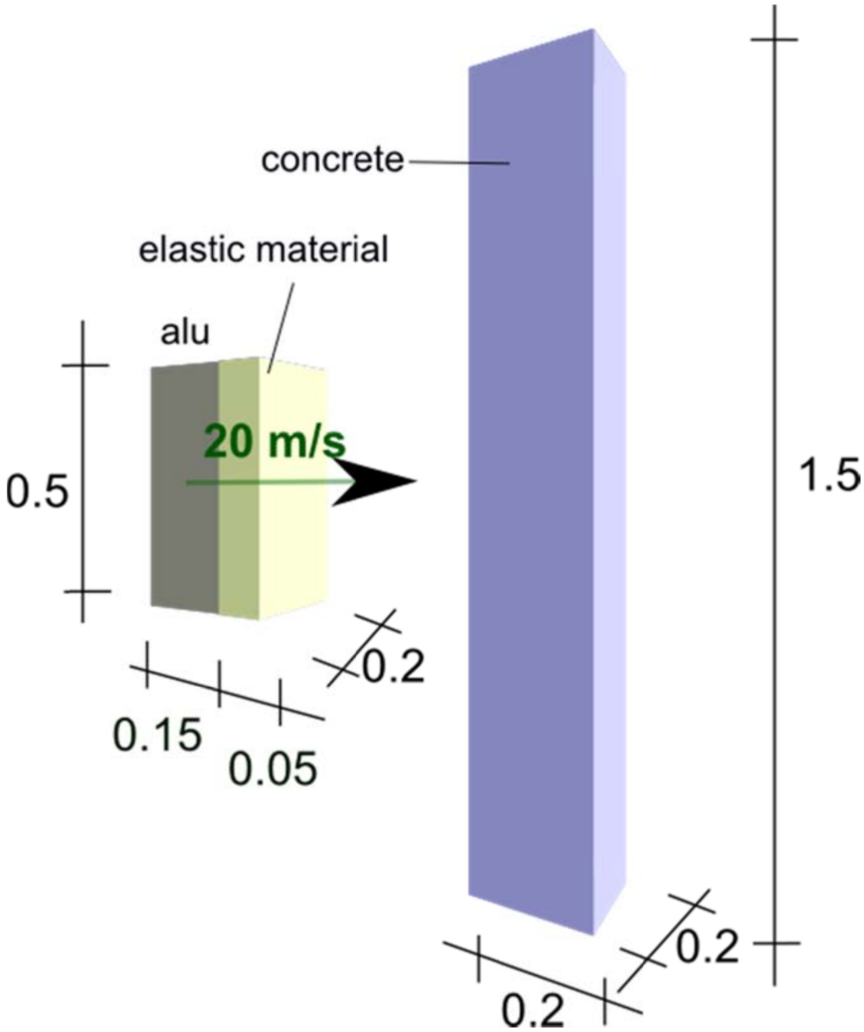

Figure 1: Sketch of principle of blast simulator testing of a concrete column.

Numerical simulations have been carried out [14] to investigate the influence of several materials on the size and the form of the impacting mass, and consequently on the magnitude[3] and shape of the resulting pressure wave and on the structural failure of the concrete column. These studies were done using the explicit finite element code EUROPLEXUS [3] co-developed by the JRC and the Commissariat à l'énergie atomique et aux énergies alternatives (CEA). This report describes the material modelling developments needed to perform these investigations. Two main materials have been considered: hyperelastic material and concrete.

2 Concrete models

2.1 Concrete behaviour

Concrete is a quasi-brittle material. In addition to the brittle failure, it shows under tension a softening after the initiation of the cracks. This softening results from the fact that the cracks can still transmit part of the forces. As illustrated in Figure 2, micro cracks are at the beginning randomly distributed but under tension loading these micro cracks are growing orthogonal to the loading direction (A). At a certain point these micro cracks are concentrated at a section (B). The macro crack starts to develop but it can still sustain a part of the tensile forces. After passing the peak stress the crack is opened by a crack opening Δu (C). This softening can be described by a fracture process zone together with discrete cracks (e.g. Larcher [7]). Another possibility for modelling this behaviour is through a combined damage-plastic model.

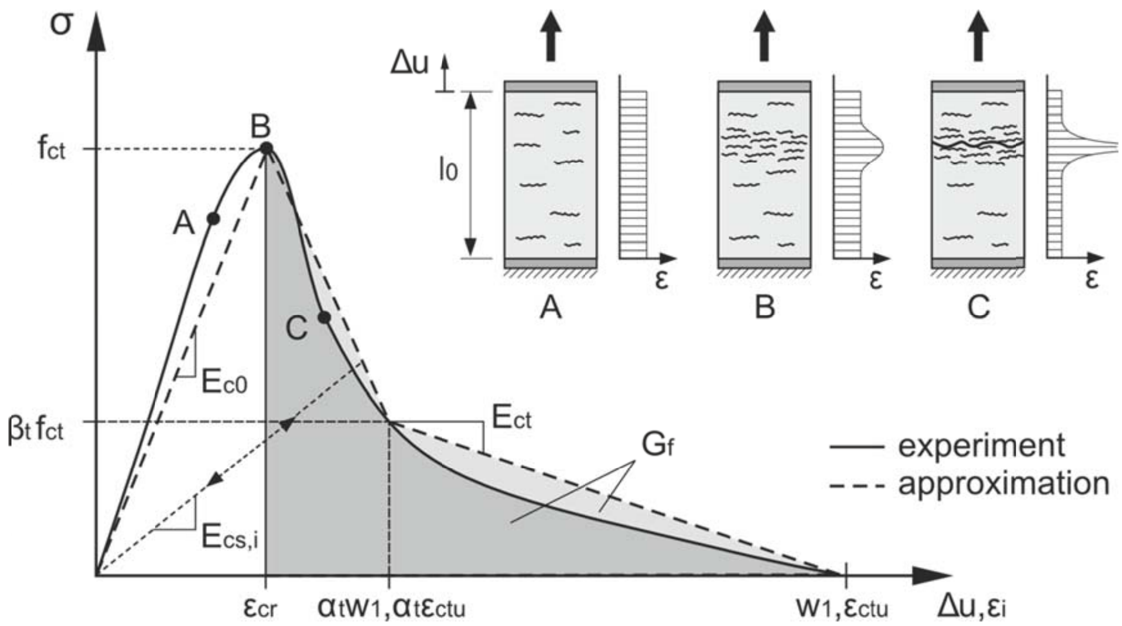

Figure 2: 1d tensile failure of concrete (from Akkermann [1] / Larcher [7])

Figure 3 shows some quasi static experimental results that were used in the following for comparison with numerical results.

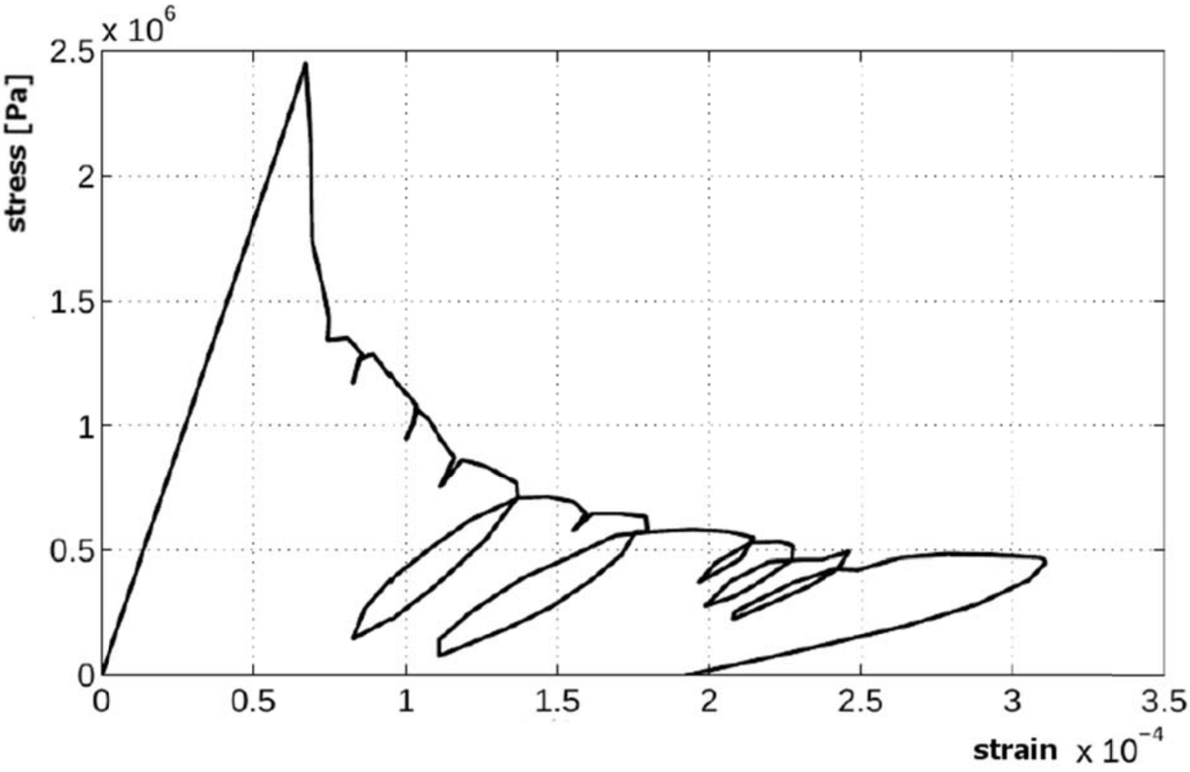

Figure 3: 1d tensile failure of concrete, experimental results (from Terrien [13])

The softening behaviour under one-dimensional compression is similar as the failure of concrete under one-dimensional compression is mainly influenced by the lateral tension. Therefore, the curve has a similar behaviour, as can be seen in Figure 4. To perform a pure one-dimensional compression test the friction between the sample and the load plate must be eliminated or reduced. If this is not done the concrete near the loading plates is in a tri-axial stress state and can sustain higher stresses. This leads to a classical hourglass failure form of standard compression tests for concrete. That effect can also be seen in Figure 4. A more slender specimen form results in a more pure one-dimensional compression in the middle of the specimen and with that to a reduction of the softening part.

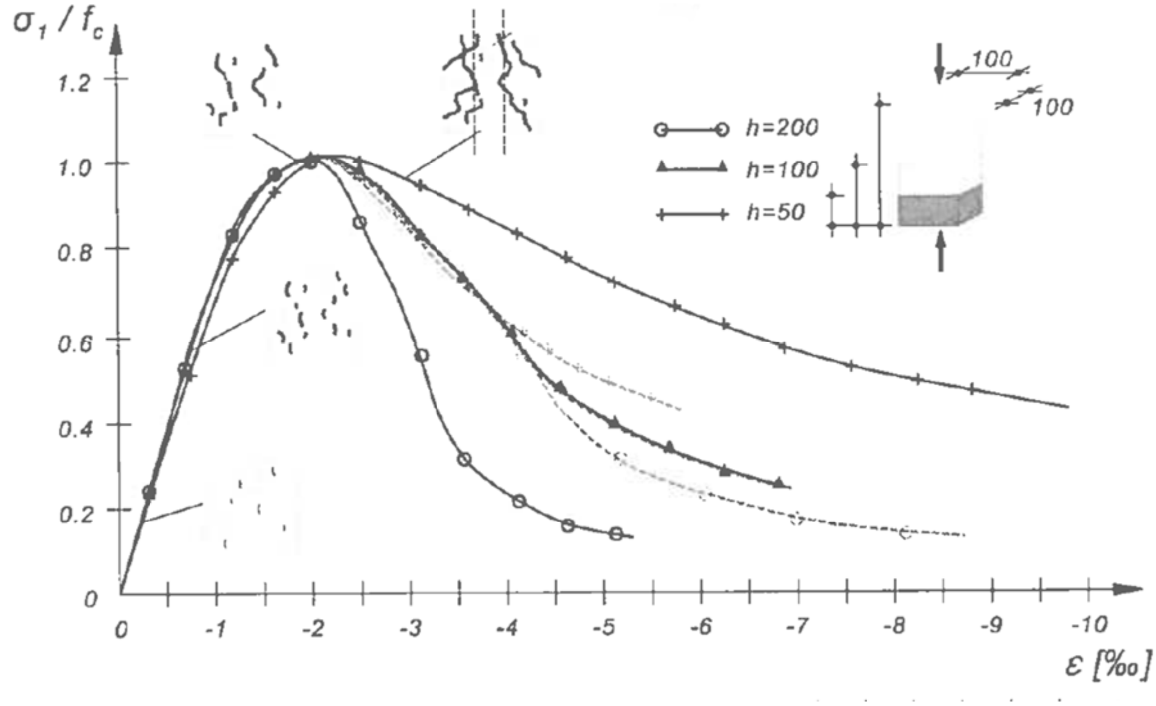

Figure 4: 1d compressive failure of concrete (from Akkermann [1], Van Mier [15])

The advantage of discrete crack models is that the location of the crack is built up in detail. Also the behaviour in the crack and around the crack is more physical. Nevertheless, it is complicated to introduce discrete cracks in a numerical model. The location of the crack must be stored in a proper way. This information must be taken into account for the element calculation. Either the elements must be remeshed in order to have the crack along the element edges or approaches dealing with cracks inside of elements must be used (e.g. X-FEM, EFG). All these methods are time consuming and are complex especially for 3D application. In such a case the crack tip becomes a crack tip line and the crack development must be performed along this line.

The idea of damage models or continuum models in general is that the crack is smeared over one or more elements. This may physically not be the best way but it is in general much easier to be implemented since the element formulation must not be changed. The damage results in the reduction of the stiffness after reaching the maximum strength. At a certain point the stiffness becomes very small. This can result in big displacements, which are also observed in experiments. These big displacements could lead in distorted elements. To avoid this, elements are often eroded in explicit simulations. Erosion may be a straightforward approach but it leads in a mass reduction.

In this work only damage models are considered since discrete crack models (X-FEM) are not yet available for 3D simulations.

2.2 Concrete material laws in EUROPLEXUS

In the case of the simulations that must be done for the fast actuator, the behaviour of the concrete after the maximum stress is important. It is foreseen in the experiment that the concrete undergoes failure. Pure plastic models cannot represent the reduction of the stress after reaching their yield stress; pure erosive models cannot consider the softening or must include it in other ways. In both cases the fracture energy, as the area below the stress-strain curve, is wrong, and the investigations concerning the appropriate concrete model must consider this.

The capability of concrete models implemented until now in EUROPLEXUS was limited. The following models are in principle available:

- BETO Concrete (NAHAS model, identified as old model)
- BL3S Reinforced concrete for discrete elements (straight forward but complicated to use)
- BLMT DYNAR LMT Concrete
- DADC Dynamic Anisotropic Damage Concrete
- DPDC Dynamic plastic damage concrete
- DPSF Drucker Prager with softening and viscoplastic regularization
- EOBT Anisotropic damage of concrete

Only DADC, DPDC and DPSF are recommended from the EUROPLEXUS developers to be used for concrete. DADC and DPDC are both plastic-damage models and should be able to represent the concrete behaviour. DADC uses an anisotropic damage model, which could be helpful for concrete. DPDC has so far no strain rate behaviour implemented. Since the loading rates are not very big this could be accepted. DPSF is a plastic model with softening and viscoplastic regularization.

To test the models a numerical one-element experiment has been built (for input files see appendix). This element is loaded in one direction under tensile and compressive loading. The loading is applied using a displacement curve. Different loading rates are investigated to determine the strain rate influence. Proper boundary conditions are chosen in order to avoid constraining the lateral deflection (Figure 5).

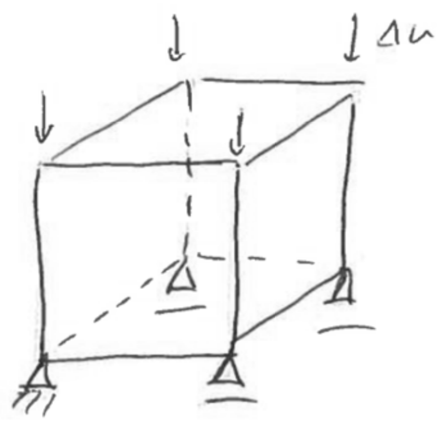

Figure 5: 1 element model for material tests

For all investigations the concrete material parameters are assumed to be given in Table 1

Table 1: Parameters for concrete

	Unit	Value
Density	kg/m^3	2400
Young's Modulus	N/m^2	4.2e10
Poisson's ratio	-	0.2
Compressive strength	N/m^2	30e6
Tensile strength	N/m^2	~3e6

2.2.1 DPSF

This material model uses a Drucker Prager failure surface with softening and viscoplastic regularization. The model does not use any damage but only plasticity. The viscoplastic regularisation can be helpful to stabilize static calculations. For dynamic calculations the viscos term can represent the strain rate effect.

Figure 6 shows that the strain rate has an influence on the tensile behaviour but it is inverse. The tensile strength is decreased for higher strain rates. The tensile strength itself is slightly overestimated. A kind of softening can be observed, but in comparison with the experimental results, the softening is too big.

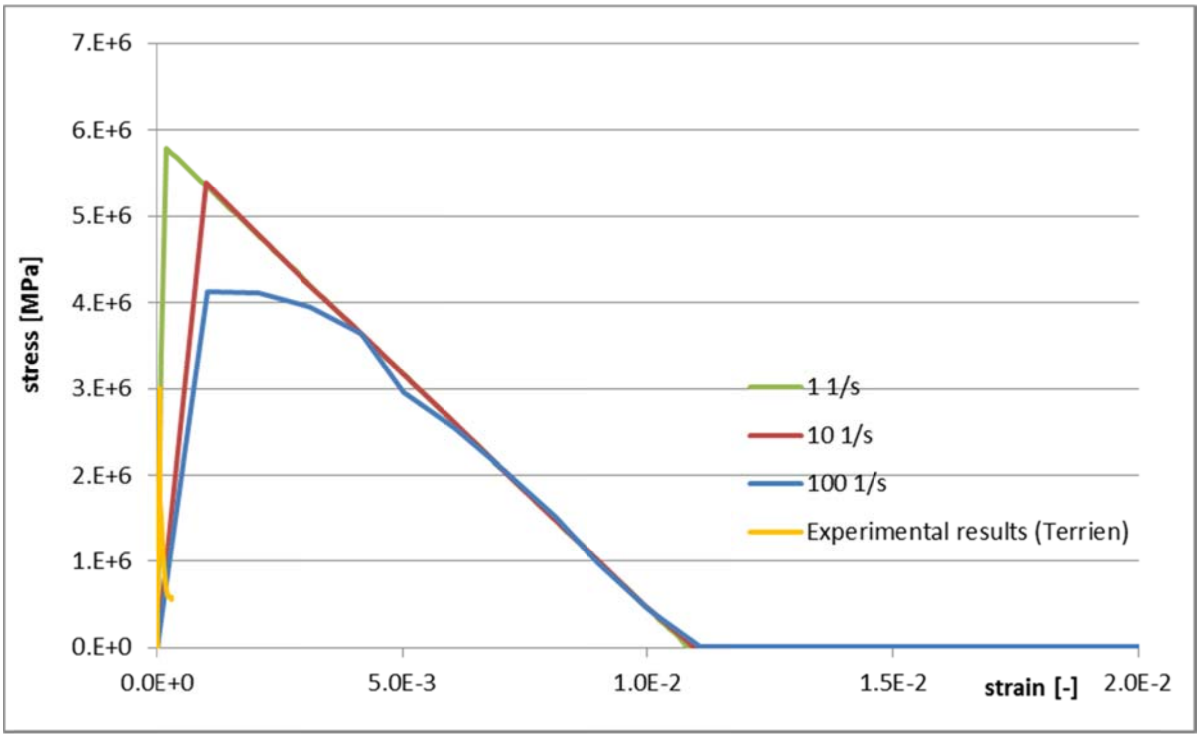

Figure 6: DPSF for tensile loading

The behaviour under compression cannot be represented using the DPSF material law. There is no failure strength, no softening and no influence of the strain rate observed (Figure 7). Since these characteristics of concrete are fundamental to represent its failure behaviour, DPSF should not be used for the investigations of the blast actuator.

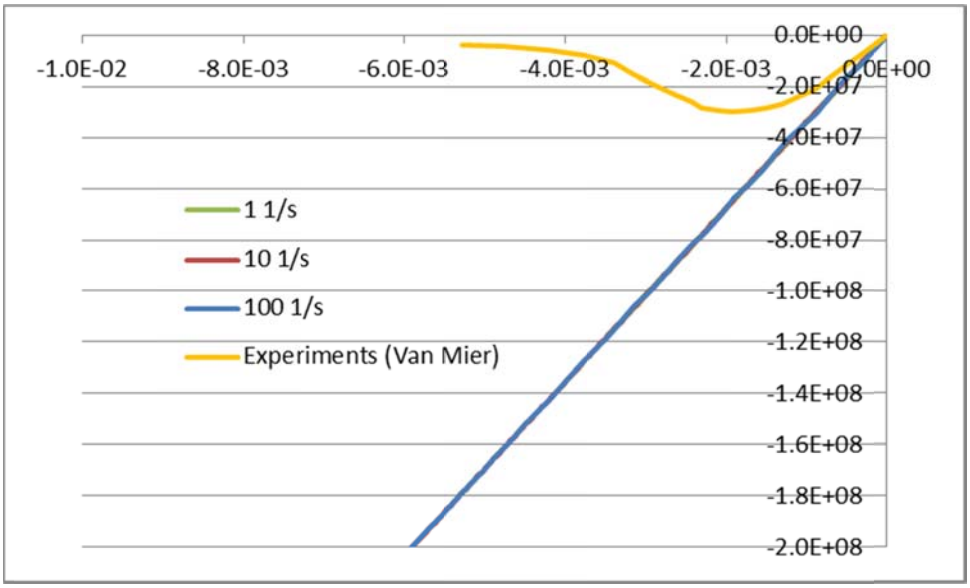

Figure 7: DPSF for compressive loading

The most critical parameter is ETA, which represents the relaxation time (viscoplastic parameter). The presented simulations use as ETA a value of 0.0, representing no relaxation. If higher values are taken (e.g. 0.002 as it is done in the benchmarks) the yield strain under tension becomes too big.

2.2.2 DPDC

The material law DPDC has been developed by CEA recently. A preliminary version of the material law is already available. Since the evolved source is a little bit out-of-date (damage model is not available) a recent development version from CEA is used. This version includes also the damage part. Not yet included in this version is the strain rate effect. Since the strain rates for the impact are not too big the material law should be usable.

The material versions 1 and 7 (VERS 1 and VERS 7) are used for all tests in order to show their difference. Version 1 includes only the plastic part, version 7 also the damage part. Figure 8 shows the one-dimensional tension and compression loading for the DPDC material without the damage part. It can be seen that the tensile and compressive strength are represented well. The curves for the strain rates of 1 1/s to 100 1/s are nearly identical, i.e. the strain rate effect is not considered. The softening part, after reaching the compressive strength, cannot be represented very well with the chosen elastoplastic material law. Therefore, it is not recommended to use this material option (VERS 1) for concrete simulations in which a failure of the concrete is foreseen.

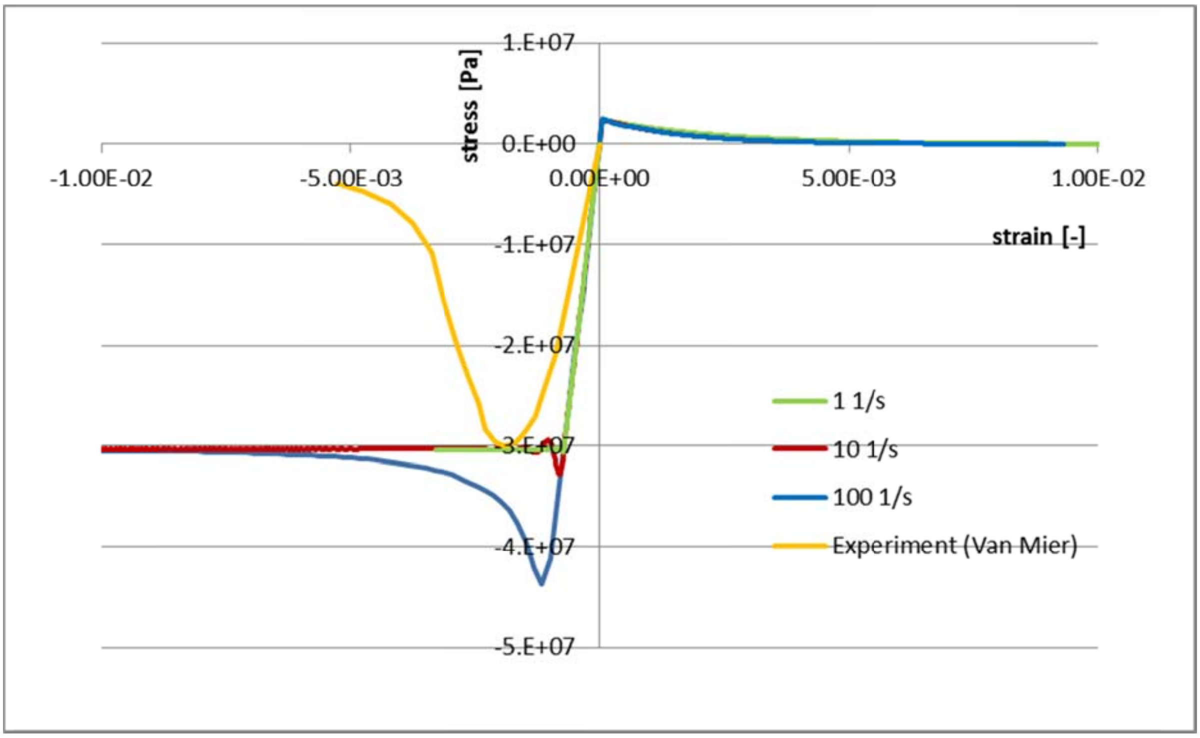

Figure 8: DPDC without damage option, VERS 1

As shown in Figure 9 (VERS 7), the damage part is needed in order to obtain a much better description of the softening. The softening is represented well in case of high strain rate but for lower strain rates the softening is too high. Also in that case the influence of the strain rate effect cannot be well represented.

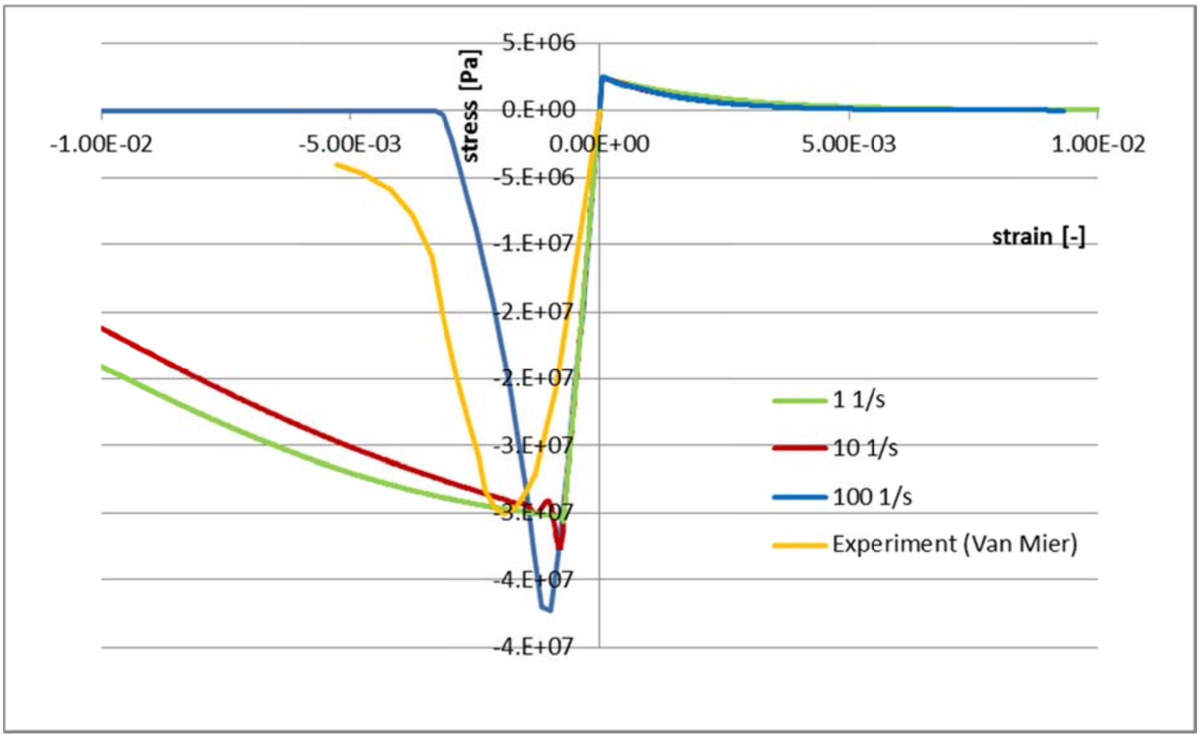

Figure 9: DPDC with damage option, VERS 7

The tensile behaviour of DPDC with option VERS 7 is shown in Figure 10. It can be seen that the maximum tensile stress is represented and also softening can be observed. Nevertheless, the material law cannot represent the strains. The softening under tension is too high. An influence of the strain rate on the softening is observable.

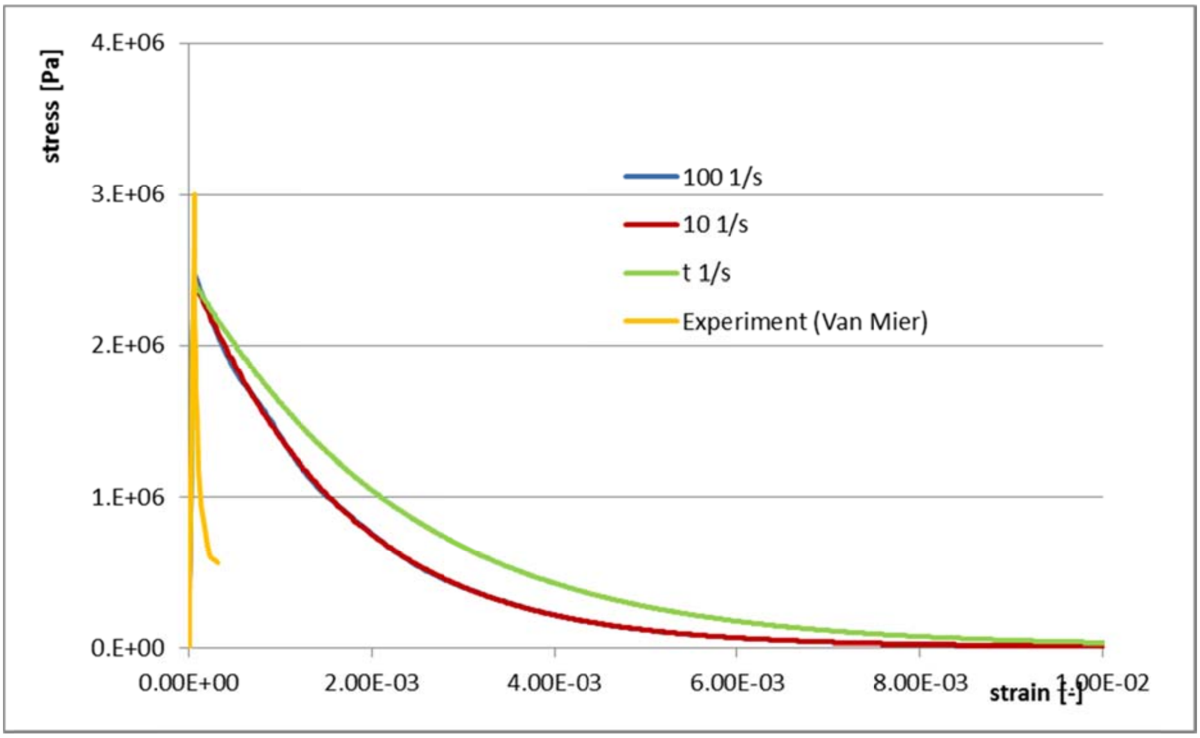

Figure 10: DPDC with damage option, VERS 7, tensile behaviour

2.2.3 DADC

The Dynamic Anisotropic Damage Concrete model (DADC) has been developed by EDF. It is a pure damage model including an anisotropic damage. That can have a certain advantage for concrete since the stiffness parallel to the crack is not significantly reduced. The anisotropic damage is included by using 6 damage variables.

The same tests, as for the previous materials, are done with that material. Since some parameters of that material law are not easy to obtain, the parameters from one benchmark exercise are taken (bm_str_dadc_multicomp.epx). The parameters for the elastic limit for compression and tension (SIGT and SIGC) are adapted to the values of the other investigations. For the parameter of the two-dimensional compression SGBC, the one-dimensional compressive limit is adopted multiplied by a factor of 1.1 (see classical 2D diagram by Kupfer [6]).

The parameter for BETA is reduced to 0.4 in order to allow a calculation. All other parameters are unchanged (ALPH 11E+6, BT 1, DC 1, DINF 50000, BV 1, DTFI 1E-8).

The parameters used for the investigations are shown in the example in the Appendix.

The simulations with one element show that the softening part cannot be represented very well for the compressive part. This may result from the value of the damage parameter BETA. This parameter allows to modify the post-peak behaviour in compression and bi-compression. It can be seen that this value influences the behaviour after reaching the maximum stress.

The strain rate effect is observable but slightly underestimated. The increase factor for concrete for strain rates about 100 1/s is in the rage of 1.5 while the numerical simulation results in 1.16.

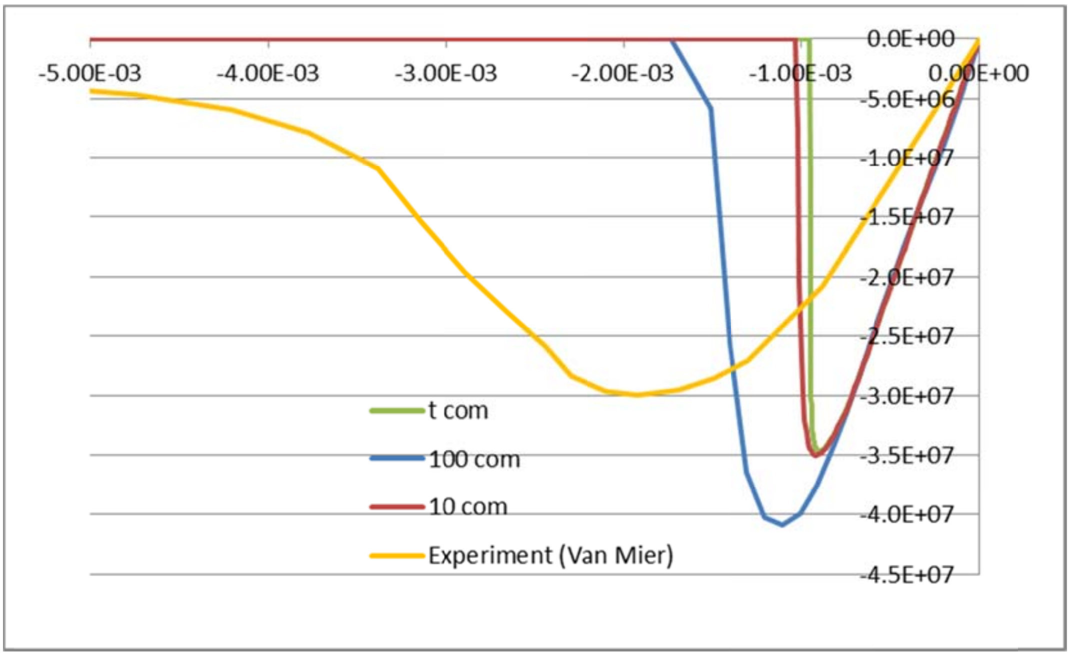

Figure 11: DADC under compression

The behaviour under tension shows a big influence of the strain rate effect (Figure 12). The order seems to be better in comparison to the other material laws. For a strain rate of 100 1/s under tension an increase factor of 6 is expected while the numerical simulation shows a value of about 11. For a strain rate of 10 1/s an increase factor of 2 is expected while the numerical simulation shows a value of about 2.6. The softening and the maximum tensile strength are overestimated. The failure occurs not as brittle as in the experiment. Nevertheless the tensile part is represented much better than in both other material laws.

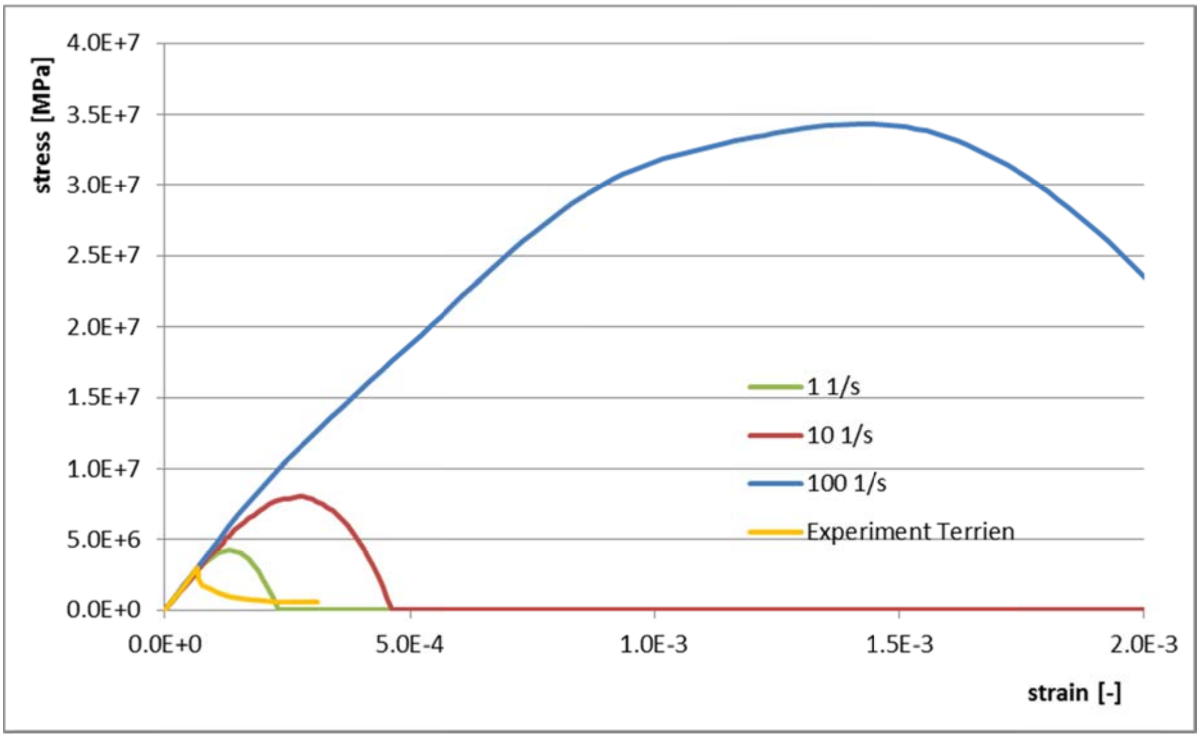

Figure 12: DADC under tension

Figure 13 shows the tensile and compressive behaviour in one diagram. Since some material parameters are not yet defined in a clear way the behaviour may be represented better by choosing appropriate values. There exists also a matlab routine from EDF that can be used to determine the parameters. This should be tested in further investigations.

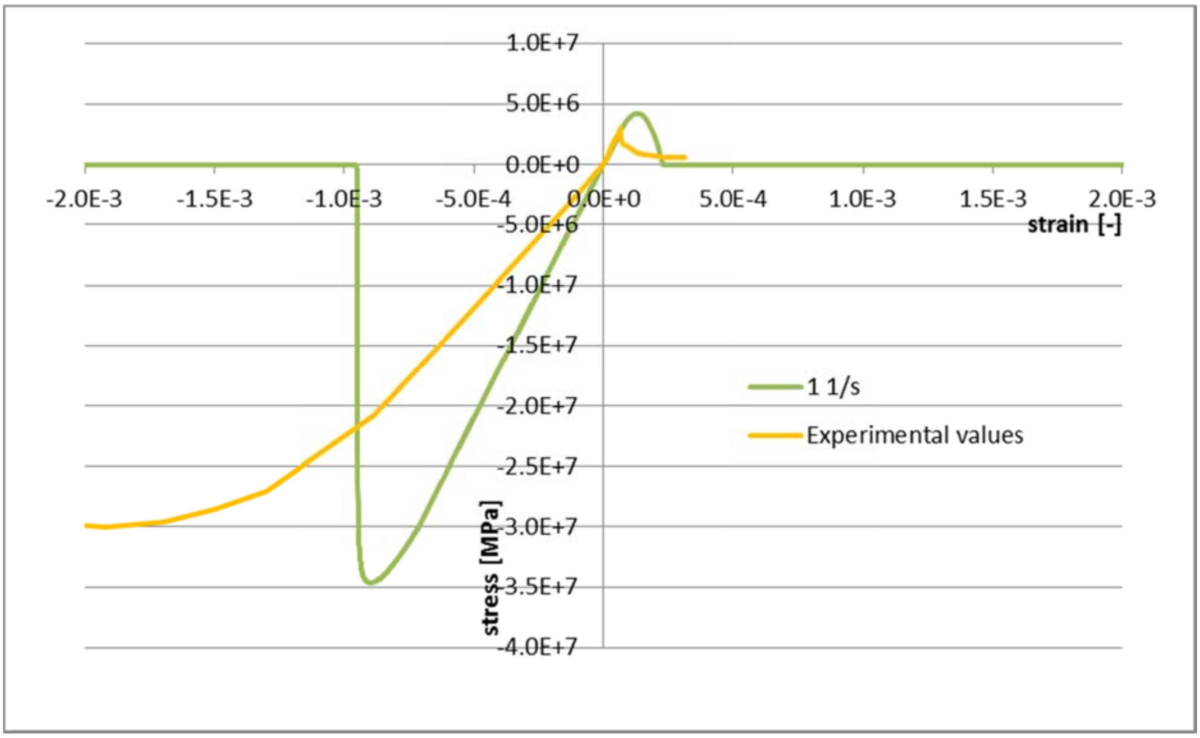

Figure 13: DADC under compression and tension for strain rate of 1 1/s

2.3 Reinforcement

There are several possibilities for considering reinforcement in concrete. A very simple one is to build it up by reducing the bars to a corresponding steel plate. This can be used very effectively in case of shell elements with different integration points through the thickness. The methodology is similar to the one used for laminated glass in a previous report [8]. The method cannot represent the behaviour of the separated bars and is limited to a full bond.

The reinforcement behaviour can be considered more in detail by using separate steel bars. These bars can be built up by using general bar elements. The bond between the bars and the concrete (i.e. in this case mainly solid elements) can be done by using the same nodes for the bar and the concrete assuming a full bond. Specific interface routines are also given for cases in which the nodes are not coincident (ARMA command). The method looks for the concrete nodes in the neighbourhood of the steel bar and connects them to the concrete. Until now only a full bond is possible for this procedure.

3 Hyperelastic materials

3.1 *Rubber*

Rubber shows a hyperelastic material behaviour. This is indicated by the nonlinear stress-strain curve and the fact that the Young's modulus is increased with increasing strain, as shown in Figure 14. Such behaviour cannot be described with a plastic material since such materials allow only a decreasing Young's modulus. In addition plastic material laws assume the existence of plastic strains. Hyperelasticity assumes a pure elastic behaviour i.e. the loading and unloading curves are identical and there are no residual deformations when the material is completely unloaded.

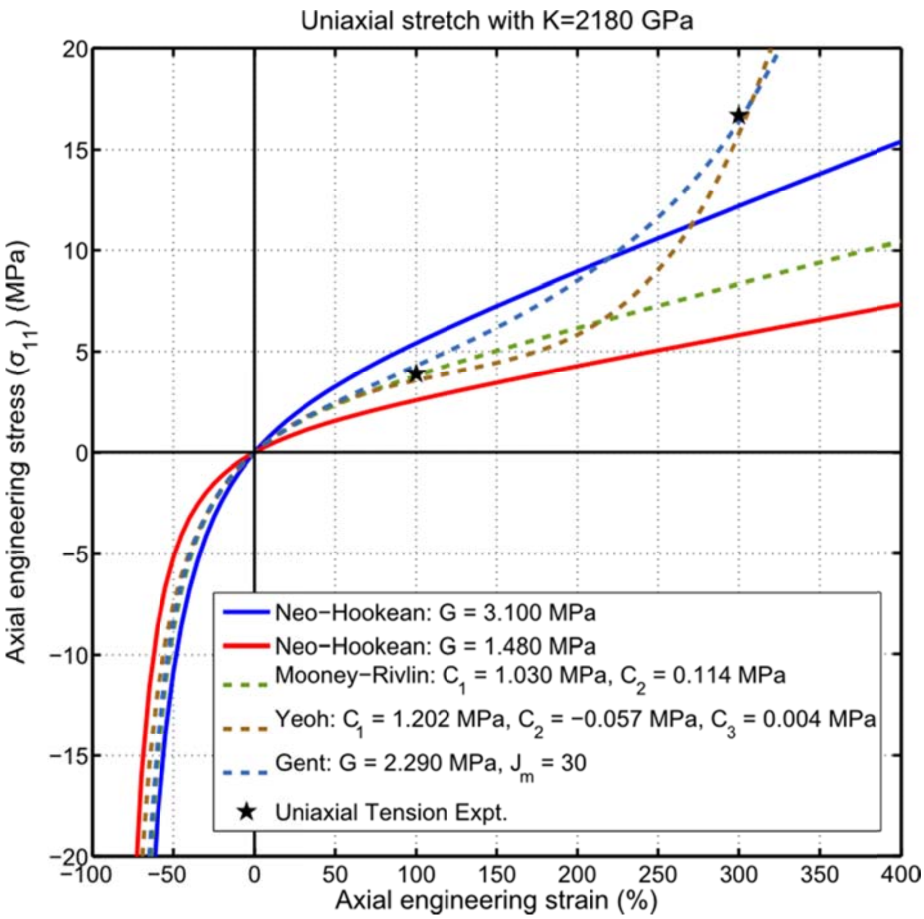

Figure 14: Hyperelastic material laws (Wikipedia)

3.1.1 Hyperelastic material laws

Hyperelastic material laws are defined by a strain energy density function. This is mainly a phenomenological approach. The parameters of the strain energy density functions are determined by using best fit approaches.

The energy functions use, for example, polynomial expressions for strain invariants or other strain expressions. This has the advantage that the parameters of these equations can be obtained by fitting them to experimental results. This is described for Mooney-Rivlin by Sun [11].

Several hyperelastic approaches are available. As shown in Figure 14 their capability to describe the material differs. Material laws with a higher number of parameters can often describe the material better but it must be considered that the behaviour outside the parameter fit could be wrong.

The most often used and cited laws are the Mooney–Rivlin, Ogden and Neo-Hookean ones.

The **Neo-Hookean** is one of the easiest descriptions:

$$W = C_1(I_1 - 3) \tag{1}$$

With the first invariant as

$$I_1 = \lambda_1{}^2 + \lambda_2{}^2 + \lambda_3{}^2 \tag{2}$$

there is only one parameter that can be used to fit the material law to an experimental curve. Therefore, this law can often not describe the material very well, as it can also be seen in Figure 14.

The **Mooney-Rivlin** material law is a two parameter model but its capability to describe the experimental points in Figure 14 is also limited. The energy density function is given as

$$W = C_1(\overline{I}_1 - 3) + C_2(\overline{I}_2 - 3) \tag{3}$$

with the following invariants

$$\begin{aligned} &\overline{I}_1 = J^{-2/3} I_1, \quad \overline{I}_2 = J^{-4/3} I_2, \quad J = \det(F) \\ &I_2 = \lambda_1^2\lambda_2^2 + \lambda_2^2\lambda_3^2 + \lambda_1^2\lambda_3^2 \end{aligned} \tag{4}$$

The Mooney-Rivlin law can be extended to the **generalized Rivlin model** (also called polynomial hyperelastic model) by using an energy equation with an unlimited number of parameters

$$W = \sum_{p,q=0}^{N} C_{pq}(\overline{I}_1 - 3)^p(\overline{I}_2 - 3)^q + \sum_{m=1}^{M} D_m (J-1)^{2m} \tag{5}$$

By using more parameters the model can be much better adapted to experimental data. The first summation of the equation concerns the deviatoric part; the second summation concerns the volumetric part of the energy.

The **Ogden** material law is not using invariants. Here, the principal stretches are used. It is important to mention that the stretches S are different from the strains and are defined by

$$S = \frac{l}{l_0} = \varepsilon + 1 \tag{6}$$

The energy function of the Ogden material allows also the possibility of an unlimited number of parameters.

$$W = \sum_{p=1}^{N} \frac{\mu_p}{\alpha_p} \left(\lambda_1^{\alpha_p} + \lambda_2^{\alpha_p} + \lambda_2^{\alpha_p} - 3 \right) + K \left(J - 1 - \ln J \right) \tag{7}$$

3.1.2 Bulk modulus

The bulk modulus of rubber materials is a very important topic. In general the bulk modulus can be determined by using the Young's modulus and the Poisson's ratio by

$$E = 3K(1 - 2\upsilon) \tag{8}$$

By using a typical rubber (polybutadiene) the value of E is about 10^5 N/m^2 (Tabor [12]). The bulk modulus is much higher. The reason for that is the different material behaviour under hydrostatic compression (van der Waals interactions between chains, Tabor [12]). The bulk modulus could be about $2 \cdot 10^9$ N/m^2. Using these values the Poisson's ratio becomes 0.499992. But the Poisson's ratio for rubber is meaningless since the Young's and the bulk modulus are driven by completely different micromechanical material behaviours. Finally, rubber can be considered as incompressible.

3.1.3 Hyperelastic material law in EUROPLEXUS

There is one hyperelastic material available in EUROPLEXUS. This material has four different main option by which it is possible to define the behaviour of the law. These are

- TYPE 1: Mooney-Rivlin material
- TYPE 2: Hart-Smith material
- TYPE 3: Ogden material
- TYPE 4: Ogden material, new implementation

All four materials depend on the number of parameters. While Mooney-Rivlin allows up to 14 parameters (it is a generalized Rivlin model), Hart-Smith allows 3 parameters, and Ogden (type 3) 12 parameters since also the volumetric term is written as a summation with additional parameters. The new Ogden implementation allows up to 8 since equation (7) is used.

As a new functionality, a parameter search is implemented in EUROPLEXUS in order to get from a given one-dimensional stress-strain relation parameters for a specific hyperelastic law. This is done by an optimized search over all parameters to find the best fit using a least square method. For each tested set of parameters and for each known stress-strain point the material routine is called to get the numerical stress for these inputs. The length of this parameter search depends on the number of parameters. To reduce the calculation time for large parameter models the search is done in steps. After finding the first minimum in a coarse parameter mesh, the mesh is refined around the first best fit. It is important to mention that the values for the exponents in the Ogden-formulation can also be negative. This must be considered by the parameter search algorithm.

The parameter identification is implemented for one-dimensional experiments without confinement. During the parameter identification also the lateral strains are needed for the one-dimensional loading in order to have a not confined configuration. The classical equations for lateral strain are only valid for small strains. Tests for hyperelastic materials are often conducted with very high strains (up to 95% in compression). Therefore, the following equation has to be used to get the lateral strains (definitions in Figure 15):

$$\left(1+\frac{\Delta L}{L}\right)^{-\upsilon}=1-\frac{\Delta L'}{L} \tag{9}$$

with the Poisson's ratio ν. This leads to a lateral strain $\varepsilon_y = \varepsilon_z$ by using the longitudinal strain ε_x

$$\varepsilon_y=\varepsilon_z=\left(1+\varepsilon_x\right)^{-\upsilon}-1 \tag{10}$$

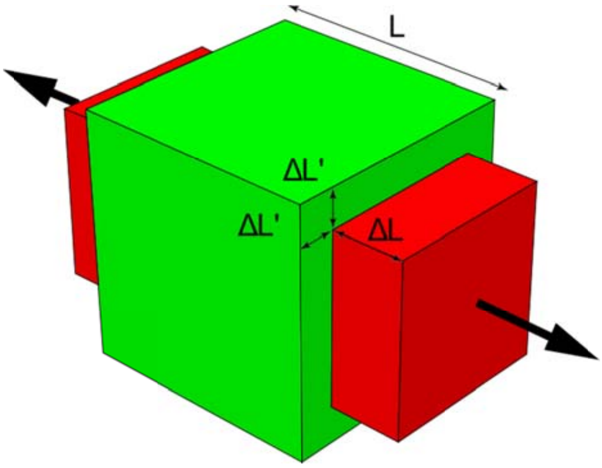

Figure 15: Definitions of L, ΔL and ΔL' (from Wikipedia)

Several preliminary tests were done also with the fast actuator model using hyperelastic material as a damping material between the aluminium impactor and the concrete target. Material type 1 shows a behaviour that seems to be realistic. Material 3 (Ogden) shows for the one-element experiment (under no loading and no boundary conditions) a shrinkage of the element. By investigating this behaviour in detail it is observable that under these conditions the internal calculation of the principal strains is not correct. The principal strains are calculated by using in a certain way the invariants. Also when the strains are all zero, the principal strains are not zero and this is physically not possible.

All hyperelastic material laws are implemented by SAMTECH by using similar Mecano routines that were written for the implicit time integration. A support by SAMTCH concerning these laws is not any more possible. The concerned Ogden material is not written very clear. The report about the implementation [4] is not giving more details and shows no examples using the Ogden implementation. Therefore, it was recommended to rewrite the Ogden material in order to offer a usable implementation of this law.

3.1.4 New Ogden implemenation

The original article from Ogden [9] deals only with incompressible materials and is mainly focused on implicit methods. To facilitate the implementation a pure explicit formulation that is also used for LS-DYNA, is used here (Du Bois [2] and Freidenberg [5]). The energy (which is not used for further calculations) consists for n parameters form

$$W = \sum_{i=1}^{3} \sum_{j=1}^{n} \frac{\mu_j}{\alpha_j} \left(\lambda_i^{*\alpha j} - 1 \right) + K(J - 1 - lnJ)$$

By taking the derivative of this expression the true stresses can be obtained

$$\sigma_i = \sum_{j=1}^{n} \frac{\mu_j}{J} \left(\lambda_i^{*\alpha j} - \sum_{k=1}^{3} \frac{\lambda_i^{*\alpha j}}{3} \right) + K \frac{J-1}{J}$$

For incompressible material the last equation can be reduced to

$$\sigma_i = \sum_{j=1}^{n} \mu_j \left(\lambda_i^{*\alpha j} - \sum_{k=1}^{3} \frac{\lambda_i^{*\alpha j}}{3} \right)$$

It is important to define here the principal stretch λ. As mentioned earlier, in comparison to the strain, the stretch is defined by the ratio between the current length and the initial length. Therefore, the stretch S can be calculated by using

$$S = \varepsilon + 1$$

The stretch is always positive and bigger than zero. The stretch for zero strain is one.

3.1.5 Parameter examples for rubber

Some material tests were done on a first sample of rubber that was already available at the laboratory. The results of a one-dimensional compression test are shown in Figure 16. The test was not confined but due to the friction between the loading plates and the rubber the displacements near the plates were limited. The difference between the two chosen loading rates is small. There is also not observed a very high hysteresis curve which means that the viscoelastic influence is small. Also when the maximum strains are not very high it can be observed that the behaviour is hyperelastic and it should be possible to describe this material by a hyperelastic material law.

The parameter identification functionality of EUROPLEXUS is used to determine the Ogden parameters for the new implementation.

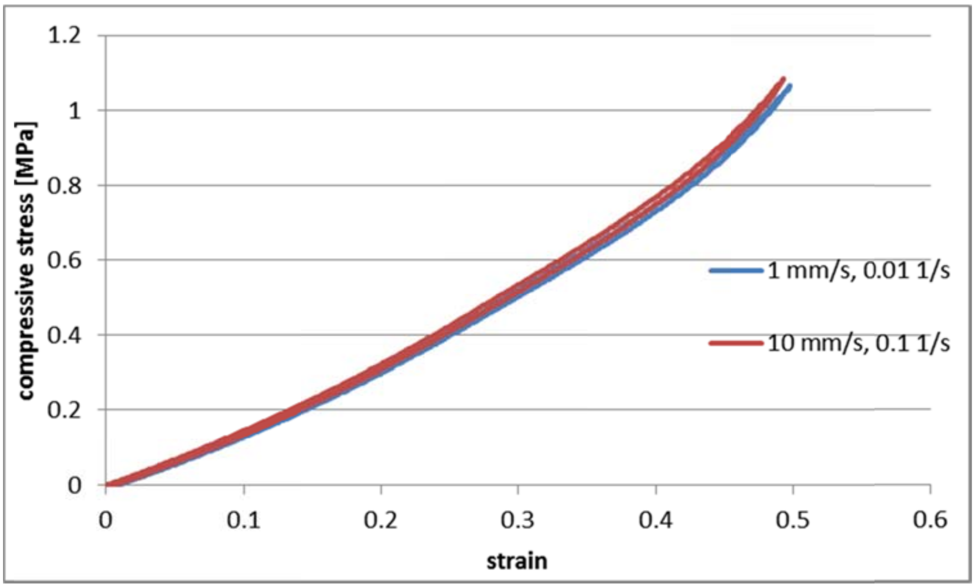

Figure 16: One-dimensional material test for a specific rubber

The experimental values of Figure 16 are scattered as shown in Figure 17 (Input). The stress-strain relations resulting from the determination of the best fit for TYPE 1 (Mooney-Rivlin) and TYPE 4 (Ogden new) are also shown in Figure 17 (EPX). It can be noticed that the influence of the material type is still small since the nonlinear behaviour of the material up the tested maximum strain of about 50% is small. But it can already be seen that the curvature of the experiment and the material TYPE 1 is different.

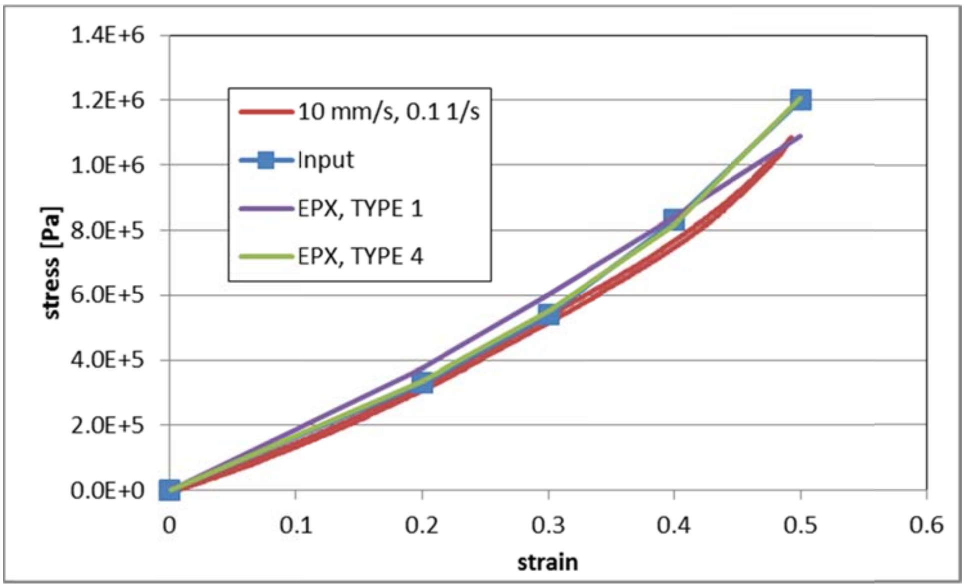

Figure 17: Parameter identification for a specific rubber: stress-strain relation

The input parameters and the parameters for the hyperelastic material law are given in Table 2. It must be noticed that the bulk modulus should be set to 0 for TYPE 1 (incompressible material) whereas for TYPE 4 a bulk modulus should be given. It is important to mention that the bulk modulus used for the determination of the parameters must also be used for the simulations. Should another bulk modulus be used, the parameter identification must be repeated.

Table 2: Parameter identification for a specific rubber: parameters

INPUTS	TYPE	1	4
	BULK	0	2e9
OUTPUT	CO1	0.28184E+5	-0.61233E+0
	CO2	0.38459E+6	0.28943E-1
	CO3		0.69221E+1
	CO5		0.57544E+5
	CO6		0.25119E+8
	CO7		0.47863E+5

The materials are also tested in one-element tests. The tests are not easy to perform since under dynamic conditions vibrations often occur. These oscillations could be damped using the quasistatic option. Nevertheless, the stress-strain curves are not as smooth as the one that could be performed using the output from the parameter identification. Since the Ogden material is implemented by total strains, the stresses can be calculated step by step without dynamic effects.

In addition the critical time step for one-dimensional benchmarks must be set small in order to avoid stability problems. This seems not to be a problem in real simulations.

3.1.6 Real-Scale test

In order to validate the numerical model against experimental data, the test of Figure 16 is modelled with EUROPLEXUS using the parameters shown in Table 2. The experiment was done with a cylindrical part of rubber with 100 mm thickness and an outer diameter of 130 mm. The experimental sample has in addition a cylindrical hole in the centre with a diameter of 35 mm.

The first numerical tests for modelling this sample including the hole were not successful. The calculation stopped after loading the specimen with a certain displacement by too large lateral displacements. Since this problem always occurred near the inner hole the model was changed into a full cylinder. Also this model showed lateral instability problems. The material law should therefore be validated more in detail.

3.2 Elastic foam

An elastic foam retains its form completely after a loading, i.e. it can be considered that its plastic or damage limit is very high and it has not to be considered. Here two types of elastic foams from polymers are investigated. Even if their density is quite different (Table 3), they show a similar stress-strain relation. Both foams are taken from the JRC repository. More detailed information about the foams is not available.

Table 3: Densities of the both foams used

Foam	Density [kg/m^3]
Black	134
Grey	34.4

For both foams material tests are conducted at the JRC to get appropriate material data. These compressive tests were done under a one-dimensional configuration with blocks of about 10 cm x 10 cm x 5 cm.

The experimental results are shown in Figure 18.

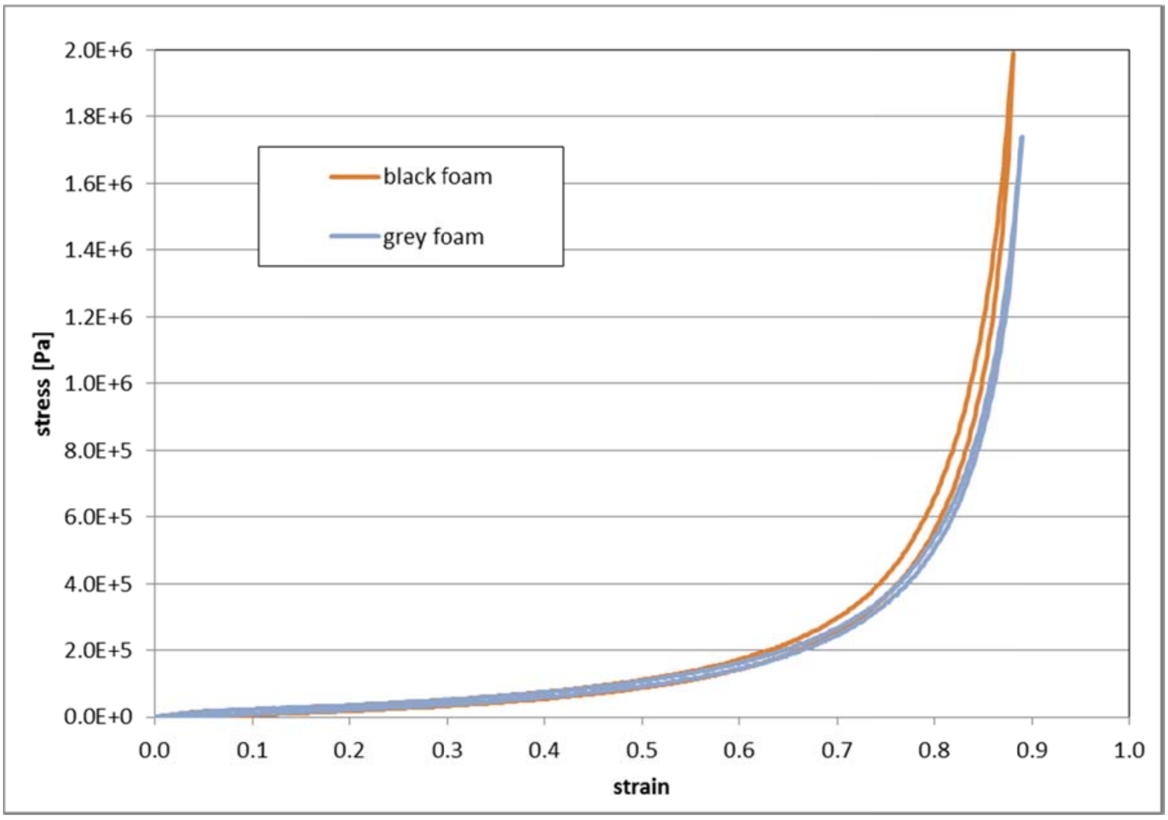

Figure 18: Stress-strain relation for compression experiment with the elastic foam

For the black foam also the lateral deflection is measured for one strain stage. At 80 % longitudinal strain the lateral strain could be measured to 0.1. Taking these results the Poisson's ratio can be estimated to 0.08. The average Young's modulus until that strain is about 565e3 Pa. The bulk modulus can determined with these parameters to 280e3 Pa. By taking these values the hydrostatic strain of the full compacted material e_D can be calculated to 0.87. The yield strain is 0.04, and the plastic Poisson's ratio is assumed to be 0.08.

Two different material models should be considered here: MAT_FOAM – a material created for aluminium foam and the hyperelastic material law.

3.2.1 Foam material law

The MAT_FOAM can be adapted in such a way that the elastic part and the plateau of the stress-strain relation can be accurately described. The material law [10] is based on a comparison between the hydrostatic strain of the full compacted material e_D and the recent one *e*. The following equation is used

$$\sigma = \sigma_p + \gamma \frac{e}{e_D} + \alpha_2 \ln\left(\frac{1}{1-\left(e/e_D\right)^{\beta}}\right) \tag{11}$$

It is obvious that in cases where $e > e_D$ the equation becomes undefined. This is seldom a problem for metallic foams since they are seldom compacted over this point. For the elastic foam used here that could happen.

The following material parameters for the MAT_FOAM material were identified for the weak foam (Table 4). The hydrostatic strain, where no pores are any more existent, is about $e_D = 0.872$. This parameter can be obtained from the initial density and the density without pores (fully compacted material). The plastic Poisson's ratio is estimated to be 0.08. The same value as the elastic one is used since the elastic one is calculated under strains of about 80%. With this parameter the value for ALFA can be calculated, while the yield strain is taken equal to 0.04.

The free values to adapt the experimental decay to the numerical model are ALF2 and BETA.

Table 4: Parameter identification for the weak foam: parameters for MAT_FOAM

Initial density	RO_F	kg/m3	134
Initial Young's modulus	YOUN	MPa	375000
Poisson's ratio	NU	-	0.1
Yield stress	SIGP	MPa	15000
Initial density of the material, not considering the voids	RO_0	kg/m3	1050
Shape of the yield surface	ALFA		1.870829
Initial hardening factor by reaching the plastic regime	GAMM		110000
Scale factor	ALF2		1000000
Scale factor	BETA		6

The resulting stress-strain curve is shown in Figure 19. It exhibits good agreement with the experimental values. Nevertheless the curve over the hydrostatic strain of about 0.9 could be indefinite due to the problems in equation (11).

3.2.2 Hyperelastic material law

The experimental stress-strain curve is also used to perform a parameter identification of the hyperelastic material laws. The resulted curves are also included in Figure 19. While the Ogden implementation gives quite good results, the Mooney-Rivlin law cannot represent the behaviour by using only two parameters. The results of the parameter identification are given in Table 5.

The identification of the bulk modulus (see beginning of this chapter) is vague. The influence of a changed bulk modulus is investigated for the Ogden material law by using values from 7.5e4 Pa to 7.5e5. The differences in the resulting stress-strain relation are very small. Nevertheless, the differences in the parameters are significant. It is therefore recommended to use the parameters only with the bulk modulus that was used to determine them. Otherwise the stress-strain relation could be wrong. In addition structural tests have shown that the smallest values used for the bulk modulus result in instable calculation. Therefore, a value of 7.44e5 Pa is used here for the bulk modulus.

Table 5: Parameter identification for the elastic foam: parameters for hyperelastic material laws

INPUTS	TYPE	1	4
	BULK	0	7.44E+05
OUTPUT	CO1	0.28184E-03	-0.49239E-02
	CO2	0.13964E+06	0.11001E+02
	CO3		0.11305E+02
	CO5		0.50119E+06
	CO6		0.43652E+04
	CO7		0.10000E+05

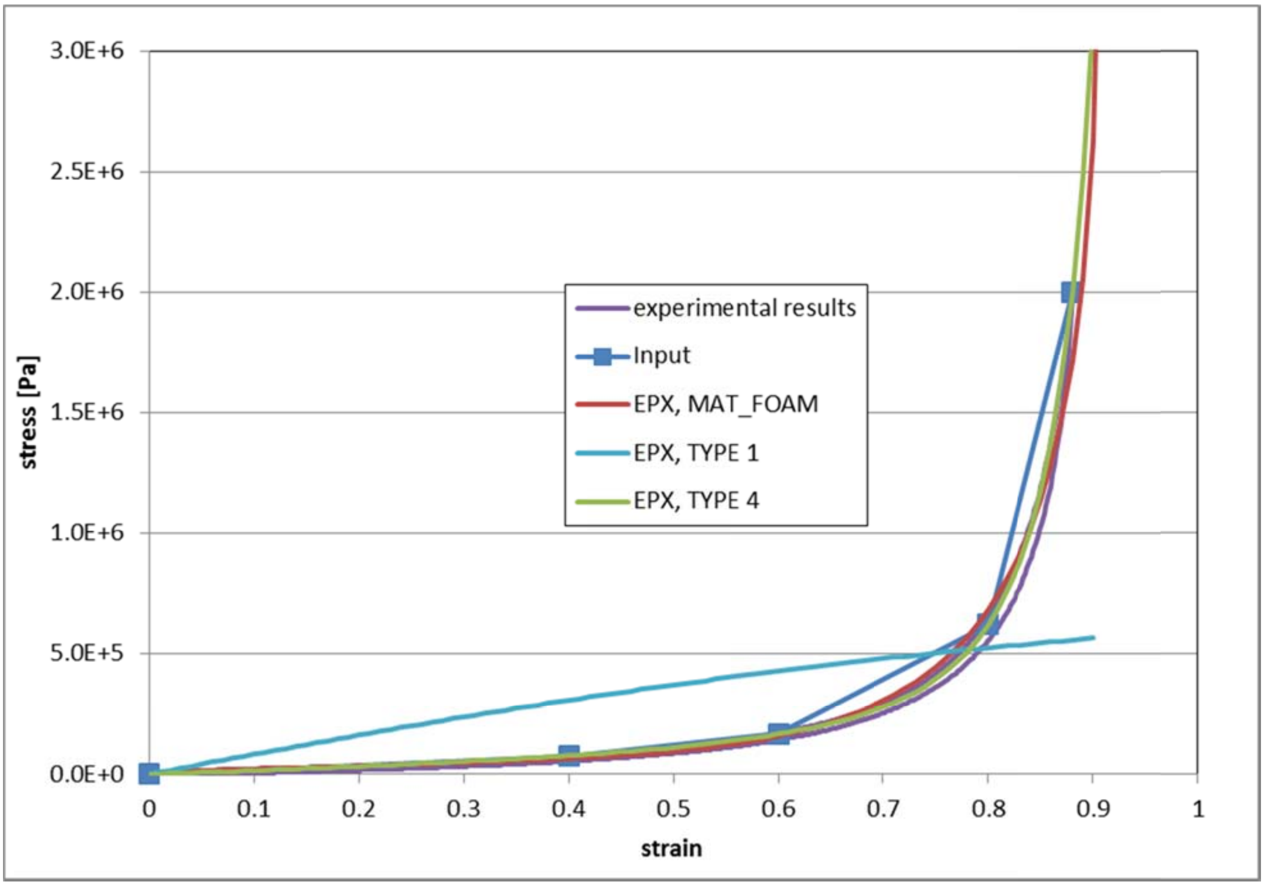

Figure 19: Stress-strain relation for compression experiment with the elastic foam

It is very important to mention that parameter identification could become critical if the behaviour is extrapolated (example for a stiffer foam in Figure 20). It can be seen that the behaviour of the three material laws is completely different. Therefore, it should always be controlled if a material law exceeds the limits for which the material parameters are determined.

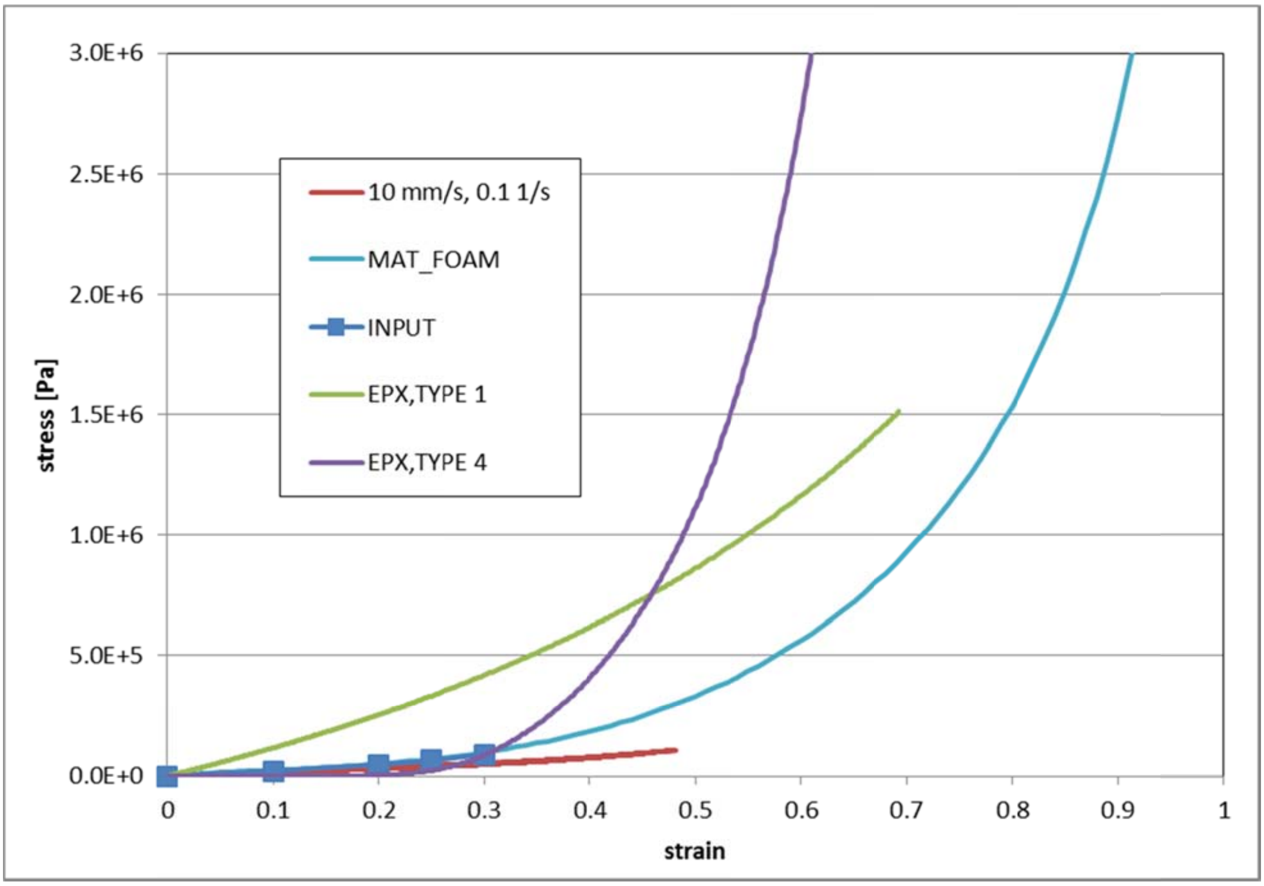

Figure 20: Extrapolation of the material law outside of the parameter room

4 Conclusions

The explicit finite element code EUROPLEXUS provides several possibilities to model concrete. Three of these material models were tested in this report, and not all of them were satisfactory. Since some of these models are still under development and since the influence of their material parameters is not completely clear, further investigations are recommended.

The new hyperelastic material implementation of the Ogden law produces good results. In combination with the parameter identification for hyperelastic materials, it is possible to build up a hyperelastic material law in an effective and precise way. Further structural tests with that material should validate this type of modelling.

5 References

[1] Jan Akkermann. *Rotationsverhalten von Stahlbeton-Rahmenecken*. Schriftenreihe des Instituts für Massivbau und Baustofftechnologie; Dissertation, Universität Karlsruhe, 2000.

[2] Paul A. Du Bois. A simplified approach to the simulation of rubber-like materials under dynamic loading. In *4th European LS-DYNA Users Conference*, 2003.

[3] Joint Research Centre and Commissariat à l'énergie atomique et aux énergies alternatives. Europlexus. http://www-epx.cea.fr/.

[4] Anthony Cheruet and Norbert Kill. Implantation d'un matériau hyperélastique dans europlexus. Technical Report SL/03/SAMTECH238/DRD, SAMTECH, Belgium, 2003.

[5] A. Freidenberg, C.W. Lee, B. Durant, V.F. Nesterenko, L.K. Stewart, and G.A. Hegemier. Characterization of the blast simulator elastomer material using a pseudo-elastic rubber model. *International Journal of Impact Engineering*, 60:58 – 66, 2013.

[6] Helmut Kupfer, Hubert Hilsdorf, and Hubert Rüsch. Behavior of concrete under biaxial stresses. *Journal of the American Concrete Institute*, 66:656–666, 1969.

[7] Martin Larcher. *Numerische Simulation des Betonverhaltens unter Stoßwellen mit Hilfe des Elementfreien Galerkin-Verfahrens*. Schriftenreihe des Instituts für Massivbau und Baustofftechnologie; Dissertation, Universität Karlsruhe, 2007.

[8] Martin Larcher. Simulation of several glass types loaded by air blast waves. Technical Report JRC48240, Joint Research Centre, Ispra, Italy, 2008.

[9] R.W. Ogden. Large deformation isotropic elasticity - on the correlation of theory and experiment for incompressible rubbrubber solids. *Proceedings of the Royal Society of London A Mathematical and Physical Science*, 326(1567):565–584, 1972.

[10] A. Reyes, O.S. Hopperstad, T. Berstad, A.G. Hanssen, and M. Langseth. Constitutive modeling of aluminum foam including fracture and statistical variation of density. *European Journal of Mechanics - A/Solids*, 22(6):815 – 835, 2003.

[11] D.Z. Sun, F. Andrieux, A. Ockewitz, H. Klamser, and J. Hogenmüller. Modelling of the failure behaviour of windscreens and component tests. In *5th European LS-DYNA Users Conference, 25-26 May*, 2005.

[12] D. Tabor. The bulk modulus of rubber. *POLYMER*, 35(13):2759–2763, 1994.

[13] M. Terrien. Emission acoustique et comportement mécanique post-critique d'un béton sollicité en traction. Technical Report 105, Bulletin de liaison du LCPC, 1980.

[14] Georgios Valsamos, Martin Larcher, and George Solomos. Numerical simulations in support of the blast actuator development. Technical Report JRC86464, Joint Research Centre, 2013.

[15] J.G.M. Van Mier. *Strain-Softening of Concrete under Multiaxial Loading Conditions*. PhD thesis, TU Eindhoven, 1984.

6 Appendix

6.1 Mesh for all test (1 element)

1el.dgibi

```
OPTI echo 1;
OPTI dime 3 elem cub8;
den=0.01;
DENS den;
sizx = 0.01;
sizy = 0.01;
sizz = 0.01;
p0 = 0 0 0;
x0 = (sizx) 0. 0.;
y0 = 0. (sizy) 0.;
z0 = 0. 0. (sizz);
*volume of the element
a1 = p0 droi x0 tran z0 volu tran y0 coul bleu;
elim a1;
TASS a1;
OPTI sauv form 'bm_str_hype_pcal.msh';
sauv form a1;
fin;
```

6.2 Material tests concrete

Only for the first test all strain rates and tension and compression are shown. For all other tests that was done similar.

dpsf.epx

```
$ BM_STRESS_STRAIN
bm_stress-strain tension strain rate 100
$
ECHO
CAST MESH
TRID LAGR
GEOM CUBE a1 TERM
MATE
 DPSF RO 2400 YOUN 34.e9 NU 0.21
     ALF1 2.40 C1 7.9928E6 BETA 2.40 ETA 0.E-3
     TRAA 2 2.40     0.0   2.40  5.E+2
     TRAC 3 7.9928e6 0.0    0.0 0.01        0.0
5.E+2
LECT a1 TERM
LIAI BLOQ 123 LECT 1 TERM
     BLOQ 2 LECT 2 3 4 TERM
CHAR 1 FACT 2 DEPL 2 1.0 LECT 5 6 7 8 TERM
       TABL 3 0 0 1.0 1.0 100 0.001
ECRI FICH ALIC TEMP tfreq 1e-5
               ELEM LECT 1 TERM
OPTI NOTE LOG 1
CALC TINI 0 TEND 5e-3
*-----------------------------------------------
SUIT
Post-treatment (time curves from alice file)
ECHO
*
RESU ALIC TEMP GARD
SORT GRAP
PERF 'ten-sr100.pun'
AXTE 1.0 'Time [s]'
COUR 1 'x' ECRO COMP 5 ELEM LECT 1 TERM !strain
COUR 2 'strain rate 100 ten' CONT COMP 2 ELEM
LECT 1 TERM !stress
TRAC 2 axes 1.0 'stress' XAXE 1 1.0 'strain'
yzer
LIST 2 axes 1.0 'stress' XAXE 1 1.0 'strain'
yzer
*-----------------------------------------------
SUIT
tension strain rate 10
ECHO
CAST MESH
TRID LAGR
GEOM CUBE a1 TERM
MATE
 DPSF RO 2400 YOUN 34.e9 NU 0.21
     ALF1 2.40 C1 7.9928E6 BETA 2.40 ETA 0.E-3
     TRAA 2 2.40     0.0   2.40  5.E+2
     TRAC 3 7.9928e6 0.0    0.0 0.01        0.0
5.E+2
LECT a1 TERM
LIAI BLOQ 123 LECT 1 TERM
     BLOQ 2 LECT 2 3 4 TERM
CHAR 1 FACT 2 DEPL 2 1.0 LECT 5 6 7 8 TERM
       TABL 3 0 0 1.0 0.1 100 0.001
ECRI FICH ALIC TEMP tfreq 1e-4
               ELEM LECT 1 TERM
OPTI NOTE LOG 1
CALC TINI 0 TEND 2e-2
*-----------------------------------------------
SUIT
Post-treatment (time curves from alice file)
ECHO
RESU ALIC TEMP GARD
SORT GRAP
PERF 'ten-sr10.pun'
AXTE 1.0 'Time [s]'
COUR 1 'x' ECRO COMP 5 ELEM LECT 1 TERM !strain
COUR 2 'strain rate 10 ten' CONT COMP 2 ELEM
LECT 1 TERM !stress
TRAC 2 axes 1.0 'stress' XAXE 1 1.0 'strain'
yzer
LIST 2 axes 1.0 'stress' XAXE 1 1.0 'strain'
yzer
*-----------------------------------------------
SUIT
tension strain rate 1
ECHO
CAST MESH
TRID LAGR
GEOM CUBE a1 TERM
MATE
 DPSF RO 2400 YOUN 34.e9 NU 0.21
     ALF1 2.40 C1 7.9928E6 BETA 2.40 ETA 0.E-3
     TRAA 2 2.40     0.0   2.40  5.E+2
     TRAC 3 7.9928e6 0.0    0.0 0.01        0.0
5.E+2
LECT a1 TERM
LIAI BLOQ 123 LECT 1 TERM
     BLOQ 2 LECT 2 3 4 TERM
CHAR 1 FACT 2 DEPL 2 1.0 LECT 5 6 7 8 TERM
       TABL 3 0 0 1 0.01 100 0.001
ECRI FICH ALIC TEMP tfreq 1e-4
               ELEM LECT 1 TERM
OPTI NOTE LOG 1
CALC TINI 0 TEND 2e-1
*-----------------------------------------------
SUIT
Post-treatment (time curves from alice file)
ECHO
RESU ALIC TEMP GARD
SORT GRAP
PERF 'ten-sr1.pun'
AXTE 1.0 'Time [s]'
COUR 1 'x' ECRO COMP 5 ELEM LECT 1 TERM !strain
COUR 2 'strain rate 1 ten' CONT COMP 2 ELEM
LECT 1 TERM !stress
LIST 2 axes 1.0 'stress' XAXE 1 1.0 'strain'
yzer
TRAC 2 axes 1.0 'stress' XAXE 1 1.0 'strain'
yzer
*LIST 1 axes 1.0 'stress' yzer
*LIST 2 axes 1.0 'stress' yzer
*-----------------------------------------------
```

```
SUIT
compression strain rate 100
ECHO
CAST MESH
TRID LAGR
GEOM CUBE a1 TERM
MATE
 DPSF RO 2400 YOUN 34.e9 NU 0.21
     ALF1 2.40 C1 7.9928E6 BETA 2.40 ETA 0.E-3
     TRAA 2 2.40     0.0   2.40  5.E+2
     TRAC 3 7.9928e6 0.0    0.0 0.01        0.0
5.E+2
LECT a1 TERM
LIAI BLOQ 123 LECT 1 TERM
     BLOQ 2 LECT 2 3 4 TERM
CHAR 1 FACT 2 DEPL 2 -1.0 LECT 5 6 7 8 TERM
       TABL 3 0 0 1.0 1.0 100 0.001
ECRI FICH ALIC TEMP tfreq 1e-6
               ELEM LECT 1 TERM
OPTI NOTE LOG 1
CALC TINI 0 TEND 2e-3
*-----------------------------------------------
SUIT
Post-treatment (time curves from alice file)
ECHO
RESU ALIC TEMP GARD
SORT GRAP
PERF 'com-sr100.pun'
AXTE 1.0 'Time [s]'
COUR 1 'x' ECRO COMP 5 ELEM LECT 1 TERM !strain
COUR 2 'strain rate 100 com' CONT COMP 2 ELEM
LECT 1 TERM !stress
LIST 2 axes 1.0 'stress' XAXE 1 1.0 'strain'
yzer
TRAC 2 axes 1.0 'stress' XAXE 1 1.0 'strain'
yzer
*-----------------------------------------------
SUIT
compression strain rate 10
ECHO
CAST MESH
TRID LAGR
GEOM CUBE a1 TERM
MATE
 DPSF RO 2400 YOUN 34.e9 NU 0.21
     ALF1 2.40 C1 7.9928E6 BETA 2.40 ETA 0.E-3
     TRAA 2 2.40     0.0   2.40  5.E+2
     TRAC 3 7.9928e6 0.0    0.0 0.01        0.0
5.E+2
LECT a1 TERM
LIAI BLOQ 123 LECT 1 TERM
     BLOQ 2 LECT 2 3 4 TERM
CHAR 1 FACT 2 DEPL 2 -1.0 LECT 5 6 7 8 TERM
       TABL 3 0 0 1.0 0.1 100 0.001
ECRI FICH ALIC TEMP tfreq 1e-6
               ELEM LECT 1 TERM
OPTI NOTE LOG 1
CALC TINI 0 TEND 2e-2
*-----------------------------------------------
SUIT
Post-treatment (time curves from alice file)
ECHO
RESU ALIC TEMP GARD
SORT GRAP
PERF 'com-sr10.pun'
AXTE 1.0 'Time [s]'
COUR 1 'x' ECRO COMP 5 ELEM LECT 1 TERM !strain
COUR 2 'strain rate 10 com' CONT COMP 2 ELEM
LECT 1 TERM !stress
LIST 2 axes 1.0 'stress' XAXE 1 1.0 'strain'
yzer
TRAC 2 axes 1.0 'stress' XAXE 1 1.0 'strain'
yzer
*-----------------------------------------------
SUIT
compression strain rate 1
ECHO
CAST MESH
TRID LAGR
GEOM CUBE a1 TERM
MATE
 DPSF RO 2400 YOUN 34.e9 NU 0.21
     ALF1 2.40 C1 7.9928E6 BETA 2.40 ETA 0.E-3
     TRAA 2 2.40     0.0   2.40  5.E+2
     TRAC 3 7.9928e6 0.0    0.0 0.01        0.0
5.E+2
LECT a1 TERM
LIAI BLOQ 123 LECT 1 TERM
     BLOQ 2 LECT 2 3 4 TERM
CHAR 1 FACT 2 DEPL 2 -1.0 LECT 5 6 7 8 TERM
       TABL 3 0 0 1.0 0.01 100 0.001
ECRI FICH ALIC TEMP tfreq 1e-6
               ELEM LECT 1 TERM
OPTI NOTE LOG 1
CALC TINI 0 TEND 2e-1
*-----------------------------------------------
SUIT
Post-treatment (time curves from alice file)
ECHO
RESU ALIC TEMP GARD
SORT GRAP
PERF 'com-sr1.pun'
AXTE 1.0 'Time [s]'
COUR 1 'x' ECRO COMP 5 ELEM LECT 1 TERM !strain
COUR 2 'strain rate 1 com' CONT COMP 2 ELEM
LECT 1 TERM !stress
LIST 2 axes 1.0 'stress' XAXE 1 1.0 'strain'
yzer
TRAC 2 axes 1.0 'stress' XAXE 1 1.0 'strain'
yzer
FIN
```

dpdc.epx

```
$ BM_STRESS_STRAIN
$ BM_STRESS_STRAIN
bm_stress-strain tension strain rate 100
$
ECHO
CAST MESH
TRID LAGR
GEOM CUBE a1 TERM
MATE
DPDC RO 2400 YOUN 4.2E+10 NU 0.2
     FC 30.E+6 DAGG 1.E-2  VERS 7
LECT a1 TERM
LIAI BLOQ 123 LECT 1 TERM
     BLOQ 2 LECT 2 3 4 TERM
CHAR 1 FACT 2 DEPL 2 1.0 LECT 5 6 7 8 TERM
       TABL 3 0 0 1.0 1.0 100 0.001
ECRI FICH ALIC TEMP tfreq 5e-8
               ELEM LECT 1 TERM
OPTI NOTE LOG 1 CSTA 0.1
CALC TINI 0 TEND 5e-4
*-----------------------------------------------
SUIT
Post-treatment (time curves from alice file)
ECHO
*
RESU ALIC TEMP GARD
SORT GRAP
PERF 'ten-sr100.pun'
AXTE 1.0 'Time [s]'
COUR 1 'x' EPST COMP 2 ELEM LECT 1 TERM
COUR 2 'strain rate 100 ten' CONT COMP 2 ELEM
LECT 1 TERM !stress
TRAC 2 axes 1.0 'stress' XAXE 1 1.0 'strain'
yzer
LIST 2 axes 1.0 'stress' XAXE 1 1.0 'strain'
yzer
*Continuation for all other strain rates and
compression
```

dadc.epx

```
$ BM_STRESS_STRAIN
bm_stress-strain tension strain rate 100
$
ECHO
```

```
CAST MESH
TRID LAGR
GEOM CUBE a1 TERM
MATE
DADC     RO 2400   YOUN 4.2E+10   NU 0.2   SIGT
3.0E+6  SIGC 30.0E+6
       SGBC 33E+06 ALPH 16.5E+6  BETA 0.67  BT
1  DC 1 DINF 50000
       BV 1 DTFI 1E-8
LECT a1 TERM
LIAI BLOQ 123 LECT 1 TERM
     BLOQ 2 LECT 2 3 4 TERM
CHAR 1 FACT 2 DEPL 2 1.0 LECT 5 6 7 8 TERM
       TABL 3 0 0 1.0 1.0 100 0.001
ECRI FICH ALIC TEMP tfreq 5e-8
               ELEM LECT 1 TERM
OPTI NOTE LOG 1 CSTA 0.1
CALC TINI 0 TEND 5e-4
*-----------------------------------------------
SUIT
Post-treatment (time curves from alice file)
ECHO
*
RESU ALIC TEMP GARD
SORT GRAP
PERF 'ten-sr100.pun'
AXTE 1.0 'Time [s]'
COUR 1 'x' EPST COMP 2 ELEM LECT 1 TERM
COUR 2 'strain rate 100 ten' CONT COMP 2 ELEM
LECT 1 TERM !stress
TRAC 2 axes 1.0 'stress' XAXE 1 1.0 'strain'
yzer
LIST 2 axes 1.0 'stress' XAXE 1 1.0 'strain'
yzer
*Continuation for all other strain rates and
compression
```

6.3 Parameter calculation

m_material_hype.ff

The shown source code is part of the module m_material_hype.

```
      SUBROUTINE CALC_PARAMETERS_CURVE
*
* read of the inputs for the calculation of the
parameters
*
      USE M_REDLEC
      USE M_ALLOCATION
*
      INCLUDE 'NONE.INC'
      INCLUDE 'REDCOM.INC'
*
      INTEGER :: I
*
      CALL ALLOCA(STRAIN,NB_POINTS,3)
      CALL ALLOCA(STRESS,NB_POINTS)
      CALL    ALLOCA(INFLUENCE_STRAIN,NB_POINTS)
!LENGTH OF INFLUENCE OF EACH STRAIN
      STRAIN(1,1)=DPREC
      CALL RED_LEC(2)
      STRESS(1)=DPREC
      DO I=2,NB_POINTS
        CALL RED_LEC(2)
        STRAIN(I,1)=DPREC
        CALL RED_LEC(2)
        STRESS(I)=DPREC
      ENDDO
      IF(NB_POINTS<2)THEN
        CALL ERRMSS('CALC_PARAMETERS_CURVE',
     >      'AT LEAST TWO POINTS ARE NEEDED TO
DETERMINE THE PARAMETERS')
        STOP 'CALC_PARAMETERS_CURVE'
      ENDIF
*      check if points are ordered
      INFLUENCE_STRAIN(1)=ABS((STRAIN(2,1)-
STRAIN(1,1))*0.5)

INFLUENCE_STRAIN(NB_POINTS)=ABS((STRAIN(NB_POIN
TS,1)-
     >
STRAIN(NB_POINTS-1,1))*0.5)
      IF(NB_POINTS>2)THEN
        DO I=2,NB_POINTS-1

INFLUENCE_STRAIN(I)=ABS((STRAIN(I+1,1)-
STRAIN(I-1,1))*0.5)
        ENDDO
      ENDIF
      END SUBROUTINE CALC_PARAMETERS_CURVE
*============================================
SUBROUTINE CALC_PARAMETERS(MAT,ORDER)
*
* read of the inputs for the calculation of the
parameters
*
*         type = 1    material type 95 = mooney
rivlin
*         type = 2    material type 95 = hart
smith
*        type = 3   material type 95 = ogden
*        type = 4   material type 95 = ogden new
formulation
*        order (only for ogden new: number of my
values)
*
      USE M_REDLEC
      USE M_ALLOCATION
*
      INCLUDE 'NONE.INC'
      INCLUDE 'REDCOM.INC'
      INCLUDE 'CUNIT.INC'
      INCLUDE 'POUBTX.INC'
*
      TYPE (MATERIAL), INTENT(INOUT) :: MAT
      INTEGER, INTENT(IN) :: ORDER
*
      INTEGER                                   ::
I,K,IPG,MATTYP,N_STEP(8),PAR_ACTIVE(12),
     &
POINT_YOU,N_STEP_MAX,NEG_VALUES(8),NEG
      REAL(8)                                   ::
EPST(6),SIG(6),DSIG(6),ECR(7),ENI,DEPS(6),VOL,C
SON,
     >
XLONG,D_START(8),D_STEP(8),D_END(8),
     >           BULK,YOU,NU,MIN_STRAIN
*
      XLONG=1
      CSON=1
      IPG=1
      VOL=1
      ENI=0
      EPST(2:6)=0
      PAR_ACTIVE(:)=0
      MATTYP=MAT%MATENT(1)
      NEG_VALUES=0        !0  SIGNIFICATES  NO
CONSIDERATION, 1 SPLIT THE PARAMETER FIELD IN A
POSITIVE AND A NEGATIVE PART, 2 THIS VALUE IS
NEGATIVE
* determination of the transversal strain
      BULK=MAT%MATVAL(2)
      IF(BULK==0.0) THEN
        NU = 0.5
      ELSE
        POINT_YOU=0
        MIN_STRAIN=1E20
        DO I=1,NB_POINTS

IF((STRAIN(I,1).NE.0.0).AND.(MIN_STRAIN>ABS(STR
AIN(I,1))))THEN
            POINT_YOU=I
            MIN_STRAIN=ABS(STRAIN(I,1))
          ENDIF
```

```
        ENDDO
        IF(POINT_YOU.NE.0)THEN
          YOU = STRESS(1)/STRAIN(1,1)
          NU = (3.0*BULK-YOU)/(6.0*BULK)
        ELSE
          CALL ERRMSS('CALC_PARAMETERS',
     &            'AT LEAST ONE STRAIN SHOULD BE
NOT ZERO')
          STOP 'CALC_PARAMETERS'
        ENDIF
        WRITE(TAPOUT,*)'FROM    THE    SMALLEST
STRAIN (', MIN_STRAIN,') :'
        WRITE(TAPOUT,*)'YOUNGS
MODULUS',YOU,'NU',NU
      ENDIF
      DO I=1,NB_POINTS
          STRAIN(I,2)=(1.0+STRAIN(I,1))**(-NU)-
1.0
          STRAIN(I,3)=STRAIN(I,2)
      ENDDO
*
      D_START(:)=-3.0   !START AT 10^D_START
      D_STEP(:)=0.5
      D_END(:)=9.0
      SELECT CASE(MATTYP)
      CASE(1)   !MOONEY-RIVLIN (2 PARAMETERS)
          PAR_ACTIVE(1:2)=1
      CASE(3)   !ODGEN (6 PARAMETERS!!!)
          MAT%MATVAL(11:14)=1.0
          PAR_ACTIVE(1:3)=1
          PAR_ACTIVE(5:7)=1
          PAR_ACTIVE(9:11)=1
      CASE(4)   !ODGEN NEW FORMULATION (ORDER*2
PARAMETERS!!!)
          MAT%MATVAL(11:14)=0.0
          PAR_ACTIVE(1:ORDER)=1
          PAR_ACTIVE(5:5+ORDER-1)=1
          D_START(1:ORDER)=-4.5      !START  AT
10^D_START  BUT  NEG_VALUES  REDUCES  IT  TO  THE
HALF
          D_STEP(1:ORDER)=0.5
          D_END(1:ORDER)=2.0
          D_START(5:4+ORDER)=4.0      !START  AT
10^D_START
          D_STEP(5:4+ORDER)=0.5
          D_END(5:4+ORDER)=9.0
          NEG_VALUES(1:ORDER)=1
      CASE DEFAULT
          CALL  ERRMSS('CALC_PARAMETERS',  'NOT
INCLUDED FUNCTION')
          STOP 'CALC_PARAMETERS'
      END SELECT
      N_STEP(:)=(D_END(:)-
D_START(:))/D_STEP(:)+1
      N_STEP_MAX=0
      DO I=1,8
        IF(N_STEP_MAX<N_STEP(I))
N_STEP_MAX=N_STEP(I)
      ENDDO
      DO K=1,3
        WRITE(*,*)  'PARAMETER  LIMITS  FOR  THE
NEXT RUN'
        WRITE(*,*)  'PARAMETER            FROM
TO        STEP'
        WRITE(TAPOUT,*)  'PARAMETER  LIMITS  FOR
THE NEXT RUN'
        WRITE(TAPOUT,*)  'PARAMETER         FROM
TO        STEP'
        DO I=1,8
          NEG=1
          IF(NEG_VALUES(I)==2)NEG=-1
          IF(PAR_ACTIVE(I)==1) THEN
             WRITE(*,1003)                    I,
NEG*10**D_START(I),NEG*10**D_END(I),
     &                      10**D_STEP(I)
             WRITE(TAPOUT,1003)               I,
NEG*10**D_START(I),NEG*10**D_END(I),
     &                      10**D_STEP(I)
          ENDIF
        ENDDO
```

```
        CALL
CALC_PARAMETERS_INC(MAT,D_START,D_STEP,D_END,K,
     &
PAR_ACTIVE,N_STEP_MAX,NEG_VALUES)
        WRITE(*,*) 'RUN ',K
        WRITE(TAPOUT,*)  'PARAMETERS  AT  THE  END
OF RUN ',K
        DO I=1,12
          IF(PAR_ACTIVE(I)==1)THEN
            SELECT CASE(I)
            CASE (1:9)
              WRITE(TAPOUT,1001)
I,MAT%MATVAL(I+2)
            CASE (10:99)
              WRITE(TAPOUT,1002)
I,MAT%MATVAL(I+2)
            END SELECT
          ENDIF
        ENDDO
        DO  I=1,8      !CALCULATION  OF  THE  NEW
PARAMETER LIMITS
          IF(PAR_ACTIVE(I)==1)THEN
            D_START(I)=D_START(I)-2*D_STEP(I)
            D_END(I)=D_START(I)+4*D_STEP(I)
            D_STEP(I)=(D_END(I)-
D_START(I))/(N_STEP(I)-1)
          ENDIF
        ENDDO
      ENDDO
      WRITE(TAPOUT,*)'             GIV   STRAIN
INFLUENCE     GIV STRESS',
     >          '      SIG            ENI'
      WRITE(*,*)'     GIV STRAIN       INFLUENCE
GIV STRESS',
     >          '      SIG            ENI'
      DO I=1,NB_POINTS
         DO K=1,3
           EPST(K)=STRAIN(I,K)
         ENDDO
         CALL
M3_HYPE(MAT,EPST,SIG,DSIG,ECR,ENI,DEPS,VOL,CSON
,XLONG,
     >                   IPG)

WRITE(TAPOUT,1000)STRAIN(I,1),INFLUENCE_STRAIN(
I),STRESS(I),
     >              SIG(1),ENI
*                    write(tapout,*)'strain',
strain(i,1:3),'stress', sig(1:6)

WRITE(*,1000)STRAIN(I,1),INFLUENCE_STRAIN(I),ST
RESS(I),
     >              SIG(1),ENI
      ENDDO

      WRITE(*,*)
      WRITE(*,*)'THE  FOLLOWING  PARAMETERS  ARE
RECOMMENDED ',
     >    'FOR THE STRESS-STRAIN CURVE GIVEN:'
      WRITE(TAPOUT,*)
      WRITE(TAPOUT,*)'THE  FOLLOWING  PARAMETERS
ARE RECOMMENDED ',
     >    'FOR THE STRESS-STRAIN CURVE GIVEN:'
      WRITE(*,*)'BULK ',BULK
      WRITE(TAPOUT,*)'BULK ',BULK
      DO I=1,12
        IF(PAR_ACTIVE(I)==1)THEN
          SELECT CASE(I)
          CASE (1:9)
            WRITE(TAPOUT,1001)
I,MAT%MATVAL(I+2)
            WRITE(*,1001) I,MAT%MATVAL(I+2)
          CASE (10:99)
            WRITE(TAPOUT,1002)
I,MAT%MATVAL(I+2)
            WRITE(*,1002) I,MAT%MATVAL(I+2)
          END SELECT
        ENDIF
      ENDDO
* calculation of a stress-strain-curve
```

```
      WRITE(TAPOUT,*)'STRESS-STRAIN   CURVE   FOR
THESE PARAMETERS'
      DO I=-90,90
         EPST(1)=I/100.0
         EPST(2)=(1.0+EPST(1))**(-NU)-1.0
         EPST(3)=EPST(2)
         CALL
M3_HYPE(MAT,EPST,SIG,DSIG,ECR,ENI,DEPS,VOL,CSON
,XLONG,
     >                  IPG)
         WRITE(TAPOUT,*)EPST(1),SIG(1)
      ENDDO
*
      WRITE(*,*)'END  OF  DETERMINATION  OF  THE
HYPE-PARAMETERS'
      WRITE(TAPOUT,*)'END  OF  DETERMINATION  OF
THE HYPE-PARAMETERS'
      WRITE(BLABLA,*) 'VALIDATION: OK'
      WRITE (0, *) BLABLA(1:72)
*
CIF FRANCAIS
      STOP 'ARRET NORMAL'
CELSE
      STOP 'NORMAL END'
CENDIF
 1000 FORMAT(5(3X,E12.5))
 1001 FORMAT('CO',I1.1,' ',E12.5)
 1002 FORMAT('CO',I2.2,' ',E12.5)
 1003 FORMAT(I2,'      ',3E12.5)
      END SUBROUTINE CALC_PARAMETERS
*===============================================
SUBROUTINE
CALC_PARAMETERS_INC(MAT,D_START,D_STEP,D_END,
     >
RUN,PAR_ACTIVE,N_STEP_MAX,NEG_VALUES)
*
*         type = 1    material type 95 = mooney
rivlin
*       type = 3   material type 95 = ogden
*       type = 4   material type 95 = ogden new
formulation
*
      USE M_REDLEC
      USE M_ALLOCATION
*
      INCLUDE 'NONE.INC'
      INCLUDE 'REDCOM.INC'
      INCLUDE 'CUNIT.INC'
*
      TYPE (MATERIAL), INTENT(INOUT) :: MAT
      REAL(8), INTENT(IN):: D_STEP(8),D_END(8)
      REAL(8), INTENT(INOUT):: D_START(8)
      INTEGER,                      INTENT(IN)::
RUN,PAR_ACTIVE(12),N_STEP_MAX
      INTEGER, INTENT(INOUT):: NEG_VALUES(8)
*
      INTEGER :: I,K,IPG,MATTYP,I_END(8),
     >
C_EI1,C_EI2,C_EI3,C_EI4,C_EI5,C_EI6,C_EI7,C_EI8
      REAL(8)
EPST(6),SIG(6),DSIG(6),ECR(7),ENI,DEPS(6),VOL,C
SON,
     >
XLONG,DIFF,DIFF_MIN,C(8),PAR_TEST(8,N_STEP_MAX)
*
      XLONG=1
      CSON=1
      IPG=1
      VOL=1
      ENI=0
      EPST(:)=0
      DIFF_MIN=1E20
      PAR_TEST(:,:)=0.0
      MATTYP=MAT%MATENT(1)
      I_END(:)=1
      DO I=1,8
        IF(PAR_ACTIVE(I)==1)I_END(I)=(D_END(I)-
D_START(I))/D_STEP(I)+1
      ENDDO
      DO I=1,8
        IF(PAR_ACTIVE(I)==1)THEN
          DO K=1,I_END(I)
            PAR_TEST(I,K)=10**((K-
1)*D_STEP(I)+D_START(I))
          ENDDO
          IF(NEG_VALUES(I)==1)THEN
            IF(MOD(I_END(I),2)==1) THEN
              CALL
ATTMSS("CALC_PARAMETERS_INC",
     &              "NUMBER OF ELEMENTS MUST BE
ODD")
            ENDIF
            DO K=1,I_END(I)/2
              PAR_TEST(I,K)=-
PAR_TEST(I,I_END(I)-K+1)    !NEGATIVE VALUES ARE
PRODUCED IN HALF OF THE CASES
            ENDDO
          ENDIF
          IF(NEG_VALUES(I)==2)THEN
             DO K=1,I_END(I)
              PAR_TEST(I,K)=-PAR_TEST(I,K)
!NEGATIVE VALUES
             ENDDO
          ENDIF
        ENDIF
      ENDDO
      DO C_EI1=1,I_END(1)     !ALPHA1
      DO C_EI2=1,I_END(2)     !ALPHA2
      DO C_EI3=1,I_END(3)     !ALPHA3
      DO C_EI4=1,I_END(4)     !ALPHA4
      DO C_EI5=1,I_END(5)     !MU1
      DO C_EI6=1,I_END(6)     !MU2
      DO C_EI7=1,I_END(7)     !MU3
      DO C_EI8=1,I_END(8)     !MU4
              MAT%MATVAL(3)=PAR_TEST(1,C_EI1)
              MAT%MATVAL(4)=PAR_TEST(2,C_EI2)
              MAT%MATVAL(5)=PAR_TEST(3,C_EI3)
              MAT%MATVAL(6)=PAR_TEST(4,C_EI4)
              MAT%MATVAL(7)=PAR_TEST(5,C_EI5)
              MAT%MATVAL(8)=PAR_TEST(6,C_EI6)
              MAT%MATVAL(9)=PAR_TEST(7,C_EI7)
              MAT%MATVAL(10)=PAR_TEST(8,C_EI8)
              DIFF=0.0
*
              DO I=1,NB_POINTS
                DO K=1,3
                  EPST(K)=STRAIN(I,K)
                ENDDO
                CALL
M3_HYPE(MAT,EPST,SIG,DSIG,ECR,ENI,DEPS,VOL,CSON
,
     >                  XLONG,IPG)
                DIFF=DIFF+INFLUENCE_STRAIN(I)*
     &            (SIG(1)-STRESS(I))**2
              ENDDO
              IF(DIFF<DIFF_MIN)THEN
                DIFF_MIN=DIFF
                DO I=1,8
                  C(I)=MAT%MATVAL(I+2)
                ENDDO
                WRITE(*,*)
                WRITE(*,1002) RUN
                WRITE(*,1004) SQRT(DIFF)
                DO I=1,8
                  IF(PAR_ACTIVE(I)==1)
WRITE(*,1001)I,C(I)
                ENDDO
                WRITE(*,*)  '            STRAIN
GIV STRESS       STRESS'
                DO I=1,NB_POINTS
                  DO K=1,3
                    EPST(K)=STRAIN(I,K)
                  ENDDO
                  CALL
M3_HYPE(MAT,EPST,SIG,DSIG,ECR,ENI,DEPS,VOL,CSON
,
     >                  XLONG,IPG)

WRITE(*,1003)STRAIN(I,1),STRESS(I),SIG(1)
                ENDDO
```

```
            ENDIF
          ENDDO
          ENDDO
          ENDDO
          ENDDO
          ENDDO
          ENDDO
          ENDDO
          ENDDO
      DO I=1,8
        MAT%MATVAL(I+2)=C(I)
      ENDDO
      NEG_VALUES(:)=0
      DO I=1,8
*                 if(d_start(i)==log10(c(i)))call
attmss('m_material_hype',
*       &    'parameter limit of the next step
like the one before.')
        IF(PAR_ACTIVE(I)==1)THEN
          IF(C(I)>0)THEN
            D_START(I)=LOG10(C(I))
          ELSE
            D_START(I)=LOG10(-C(I))
            NEG_VALUES(I)=2
          ENDIF
        ENDIF
      ENDDO
*
 1001 FORMAT(10X,'CO',I1,' ',E12.5)
 1002 FORMAT(' NEW BEST FIT (RUN ',I2,')')
 1003 FORMAT(3(3X,E12.5))
 1004 FORMAT(' DIFF IS NOW',E12.5,' WITH THESE
PARAMETERS:')
      END SUBROUTINE CALC_PARAMETERS_INC
```

hype_pcal.epx

The first data set is active (rubber) all other inputs are commented out but could be activated.

```
$ PARAMETER IDENTIFICATION
Identification of the parameters for
hyperelastic material
ECHO
CAST MESH
TRID LAGR
GEOM CUBE a1 TERM
MATE
 HYPE TYPE 4
*rubber with hype form
 BULK  2e9
 PCAL 3
 TRAC 4
-0.2   -330000
-0.3   -540000
-0.4   -830000
-0.5   -1200000
* foam with foam form
*BULK  7.44E+05
* PCAL 3
* TRAC 4
*-0.4 -0.075e6
*-0.6 -0.168e6
*-0.8 -0.6227e6
*-0.88 -2.0e6
* foam with hype form
* BULK  7.44E+5
*  PCAL 3
*  TRAC 4
*-1.00E-01     -1.90E+04
*-2.00E-01     -4.62E+04
*-2.50E-01     -6.50E+04
*-3.00E-01     -9.00E+04
LECT a1 TERM
OPTI NOTE LOG 1
CALC TINI 0 TEND 5e-3
FIN
```

ogden_new_rubber.epx

```
$ BM_STRESS_STRAIN
compression strain rate 100
ECHO
CAST MESH
TRID LAGR
GEOM CUBE a1 TERM
MATE
 HYPE TYPE 4
 RO  1.150000000000000E+03
*TYPE 1
*CO1    0.28184E+05
*CO2    0.38459E+06
 BULK    2000000000.00000
CO1 -0.61233E+00
CO2  0.28943E-01
CO3  0.69221E+01
CO5  0.57544E+05
CO6  0.25119E+08
CO7  0.47863E+05
LECT a1 TERM
LIAI BLOQ 123 LECT 1 TERM
     BLOQ 2 LECT 2 3 4 TERM
CHAR 1 FACT 2 DEPL 2 -1.0 LECT 5 6 7 8 TERM
       TABL 3 0 0 1.0 1.0 100 0.001
ECRI FICH ALIC TEMP tfreq 1e-6
               ELEM LECT 1 TERM
OPTI QUAS STAT 100000 0.1
     NOTE LOG 1 CSTA 0.05
CALC TINI 0 TEND 5.8e-3
```

ogden_new_foam.epx

```
$ BM_STRESS_STRAIN
compression strain rate 10
ECHO
CAST MESH
*CONV WIN
TRID LAGR
GEOM CUBE a1 TERM
MATE
 HYPE  TYPE 4
 RO 134
BULK   7.44E+05
CO1    -0.49239E-02
CO2    0.11001E+02
CO3           0.11305E+02
CO5           0.50119E+06
CO6           0.43652E+04
CO7           0.10000E+05
LECT a1 TERM
LIAI BLOQ 123 LECT 1 TERM
     BLOQ 2 LECT 2 3 4 TERM
CHAR 1 FACT 2 DEPL 2 -1.0 LECT 5 6 7 8 TERM
       TABL 3 0 0 1e-1 1e-2 100 1e-4
ECRI   FICH ALIC TEMP tfreq 1e-6
               ELEM LECT 1 TERM
OPTI QUAS STAT 500 0.2
     NOTE LOG 1 CSTA 0.0005
CALC TINI 0 TEND 2e-2
```

matfoam.epx

```
$ BM_STRESS_STRAIN
bm_stress-strain tension strain rate 100
ECHO
CAST MESH
TRID LAGR
GEOM CUBE a1 TERM
MATE
FOAM  RO_F  43.3  YOUN  0.34e6  NU  0.1  SIGP  17e3
RO_0 72.22
ALFA 1.809 GAMM 80000 ALF2 500000 BETA 5.5
*FOAM  RO_F  43.3  YOUN  0.34e6  NU  0.1  SIGP  17e3
RO_0 866.66
*ALFA 1.809 GAMM 150000 ALF2 2000000 BETA 2.9
LECT a1 TERM
```

```
LIAI BLOQ 123 LECT 1 TERM
     BLOQ 2 LECT 2 3 4 TERM
CHAR 1 FACT 2 DEPL 2 1.0 LECT 5 6 7 8 TERM
       TABL 3 0 0 1.0 1.0 100 0.001
ECRI FICH ALIC TEMP tfreq 5e-8
               ELEM LECT 1 TERM
OPTI NOTE LOG 1 CSTA 0.2
CALC TINI 0 TEND 5e-2
*----------------------------------------------
SUIT
Post-treatment (time curves from alice file)
ECHO
RESU ALIC TEMP GARD
SORT GRAP
PERF 'ten-sr100.pun'
AXTE 1.0 'Time [s]'
COUR 1 'x' EPST COMP 2 ELEM LECT 1 TERM
COUR 2 'strain rate 100 ten' CONT COMP 2 ELEM
LECT 1 TERM !stress
TRAC 2 axes 1.0 'stress' XAXE 1 1.0 'strain'
yzer
LIST 2 axes 1.0 'stress' XAXE 1 1.0 'strain'
yzer
```

European Commission
EUR 26407 – Joint Research Centre – Institute for the Protection and Security of the Citizen (IPSC)

Title: Numerical material modelling for the blast actuator

Authors: Martin Larcher, Georgios Valsamos, George Solomos

Luxembourg: Publications Office of the European Union

2013 – 37 pp. – 21.0 x 29.7 cm

EUR – Scientific and Technical Research series – ISSN 1831-9424

Abstract

The European Laboratory for Structural Assessment is currently designing a new test facility that should model the loading of structures by air blast waves without using explosives. The idea is to use a fast actuator that accelerates a mass, which impacts the structure under investigation. Obviously, more such masses and actuators should be employed simultaneously for loading a structural element, e.g. a column, along its entire length. To obtain a good agreement with pressures from relevant air blast waves an elastic material should be placed between the impacting mass and the structure.

Numerical simulations have been carried out to investigate the influence of several materials on the size and the form of the impacting mass, and consequently on the magnitude and shape of the resulting pressure wave and on the structural failure of the concrete column. These studies were done using the explicit finite element code EUROPLEXUS co-developed by the JRC and the Commissariat à l'énergie atomique et aux énergies alternatives (CEA). This report describes the material modelling developments needed to perform these investigations. Two main materials have been considered: hyperelastic material and concrete.

LB-NA-26407-EN-N

As the Commission's in-house science service, the Joint Research Centre's mission is to provide EU policies with independent, evidence-based scientific and technical support throughout the whole policy cycle.

Working in close cooperation with policy Directorates-General, the JRC addresses key societal challenges while stimulating innovation through developing new standards, methods and tools, and sharing and transferring its know-how to the Member States and international community.

Key policy areas include: environment and climate change; energy and transport; agriculture and food security; health and consumer protection; information society and digital agenda; safety and security including nuclear; all supported through a cross-cutting and multi-disciplinary approach.

Publications Office

ISBN 978-92-79-35056-6