Masterarbeit

zum Erwerb des akademischen Grades eines
Diplom-Ingenieurs (Dipl.-Ing.)
der Studienrichtung Rohstoffgewinnung und Tunnelbau
Schwerpunktfach Geotechnik und Tunnelbau

eingereicht am
Lehrstuhl für Subsurface Engineering
an der Montanuniversität Leoben

Paul Gehwolf, BSc
Leoben, im Juli 2014

Thema

“Numerical Modelling of the Small Scale Rock Cutting Test”

Eidesstattliche Erklärung:

Hiermit erkläre ich an Eides statt, dass ich die vorliegende Arbeit selbstständig verfasst, keine anderen als die angeführten Quellen verwendet und die wörtlich oder inhaltlich entnommenen Stellen als solche kenntlich gemacht habe.

Affidavit

I declare in lieu of oath that I wrote this thesis and performed the associated research myself, using only literature cited in this volume.

Datum

Paul Gehwolf, BSc

Danksagung

Mein Dank gilt:

Herrn Univ.-Prof. Dipl.-Ing. Dr.mont. Robert Galler, Leiter des Lehrstuhls für Subsurface Engineering, für die Anerkennung meiner Diplomarbeit und die kooperative Zusammenarbeit sowie Herrn Dipl.-Ing. Erik Schuller für die Betreuung bei der Verfassung meiner Arbeit und für die Bereitstellung der von mir benötigten Unterlagen,

Herrn Univ.-Prof. Dipl.-Ing. Dr.mont. Thomas Antretter, Leiter des Instituts für Mechanik, Herrn Dipl.-Ing. Dr.mont. Gunter Gernot Gschwandtner, Herrn Dipl.-Ing. Dr.mont. Markus Mikl und Herrn Dipl.-Ing. Stefan Barwart für die Hilfestellungen bei etwaigen Fragen und für die vielen wertvollen Ratschläge im Bereich der Numerik,

Herrn Thomas Stoxreiter, BSc und Frau Nina Klug für die gute Zusammenarbeit, die unzähligen Diskussionen und Denkanstöße,

meinen Freunden und Studienkollegen, die mich während meines Studiums begleitet haben und für unvergessliche Erinnerungen gesorgt haben,

und ganz besonderer Dank gilt meinen Eltern Monika und Michael Gehwolf, die mir diese Ausbildung durch ihre finanzielle und menschliche Unterstützung ermöglich haben und bei jeder Entscheidung hinter mir gestanden sind.

Contents

1 Task

The aim of this paper is to build up a model (chapter 5) for the numerical simulation of the newly developed Small Scale Rock Cutting Test (chapter 2) with the finite-element software Abaqus (chapter 3). The calculated rolling force should be compared to forces of the experiments. The use of the Concrete Damaged Plasticity model (chapter 4) as constitutive law is designated.

Aufgabenstellung

Die Aufgabe dieser Diplomarbeit ist die Modellerstellung (Kapitel 5) für die Simulation des neu entwickelten Modelschneidversuchs (Kapitel 2) mit dem Finiten Element Programm Abaqus (Kapitel 3). Die errechnete Rollkraft soll mit den Versuchsergebnissen verglichen werden. Als Materialgesetz ist das Concrete Damaged Plasticity Model (Kapitel 4) vorgesehen.

2 Small Scale Rock Cutting Test (1)(2)

The Small Scale Rock Cutting Test is a 1:8 downscaled linear cutting test. One of the advantages is that only a drilling core with a diameter of ten centimetres is needed to perform the trial and no block like in a 1:1 cutting test. Due to the economic availability in an early project phase the advance rate of the TBM can be estimated soon.
The scaled test rig is suitable for hydraulic presses, which are common in every geotechnical laboratory. The rig mapped on Illustration 2-1 was build for the - at the Chair of Subsurface Engineering of the University Leoben existing - hydraulic press MTS 815.

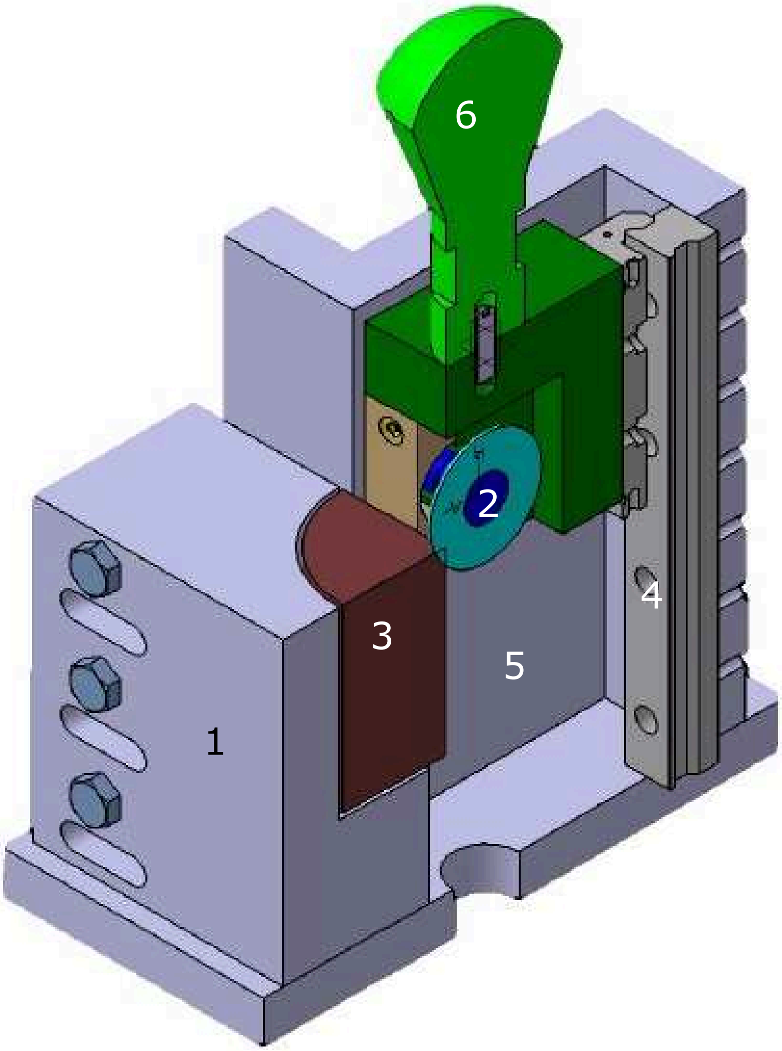


Illustration 2-1: Design drawing of the Small Scale Rock Cutting Test (1)

1	Sample holder	4	Guide rail
2	Disc cutter with bearing	5	Base body
3	Sample	6	Load introduction

Table 2-1: Components of the Small Scale Rock Cutting Test Rig

The halved drilling core with a length of 90 mm is glued whit a two component adhesive, based on epoxy, in the sample holder. Caused by oblong holes the sample could be positioned with an offset up to 15 mm perpendicular to the centreline. The results are two cutting traces with a maximum spacing of 30 mm. Each of the fife cuts of one cutting trace has an additional penetration depth of 1,5 mm. The maximum penetration depth is reached by 7,5 mm. The hydraulic cylinder pushes the disc cutter with a velocity of 1 mm/s.

For the evaluation the average value of the cutting force, measured via load cell, in the central 50 mm is used. More details to the test rig and results of this test are given in (1).

2.1 Cutting process (1)(3)

In Illustration 2-2 the phases of the cutting process with a disc cutter is shown. In phase 1 the disc cutter penetrates the rock and the material underneath is powdered. This zone is called crushed zone. Caused by the high normal force (F_N) and the big deformations in phase 2 tension cracks occur. After reaching the ultimate state or due to the superposition of the cracks of adjacent cutting traces (phase 3) chipping happens (phase 4).

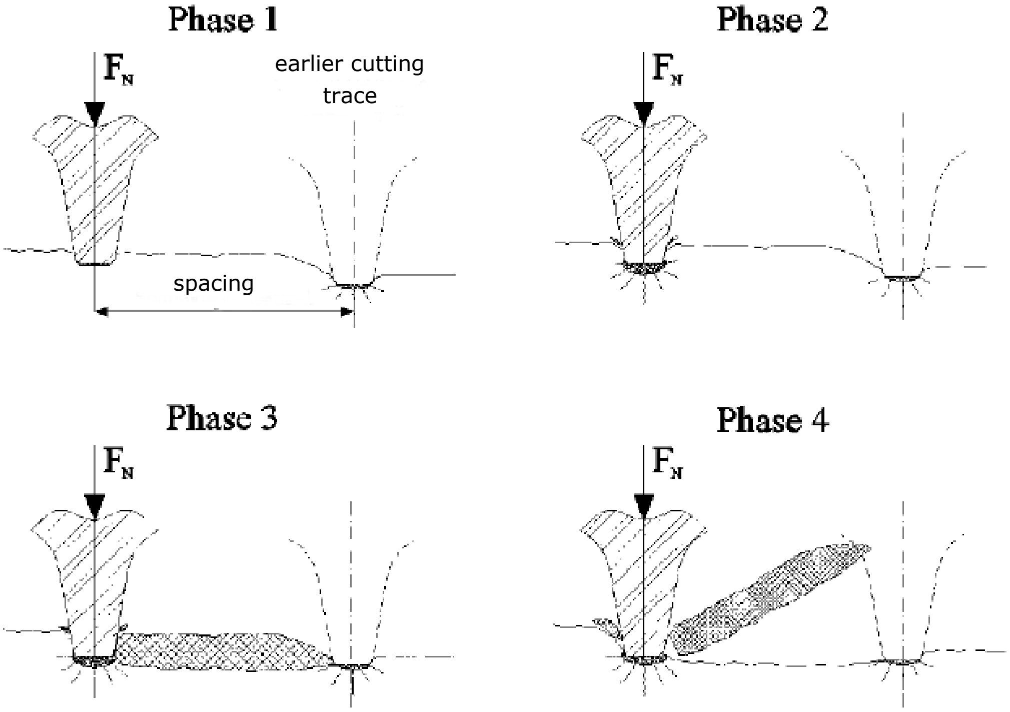


Illustration 2-2: Cutting process (3)

3 Abaqus 6.12/6.13 (4)(5)(6)

Abaqus is a modular based finite-element-software. The most important modules for mechanical analysis are:

- Abaqus/CAE (Complete Abaqus Environment)
- Abaqus/Viewer
- Abaqus/Standard
- Abaqus/Explicit

The input in a finite-element software takes place via pre-processor. In this program the real problem is converted in a simplified simulation model. The whole geometries, constitutive laws, boundary conditions, loads etc. are entered in this software, in this case Abaqus/CAE. Also the discretisation of the infinite problem in a finite number of elements takes place in this software. After completion of the feed, an inputfile is created. This file is hand on to a solver. For the calculation in this paper the used solvers are Abaqus/Standard and Abaqus/Explicit.

All results are visualized by a post-processor. Therefore, Abaqus/CAE or Abaqus/viewer is used.

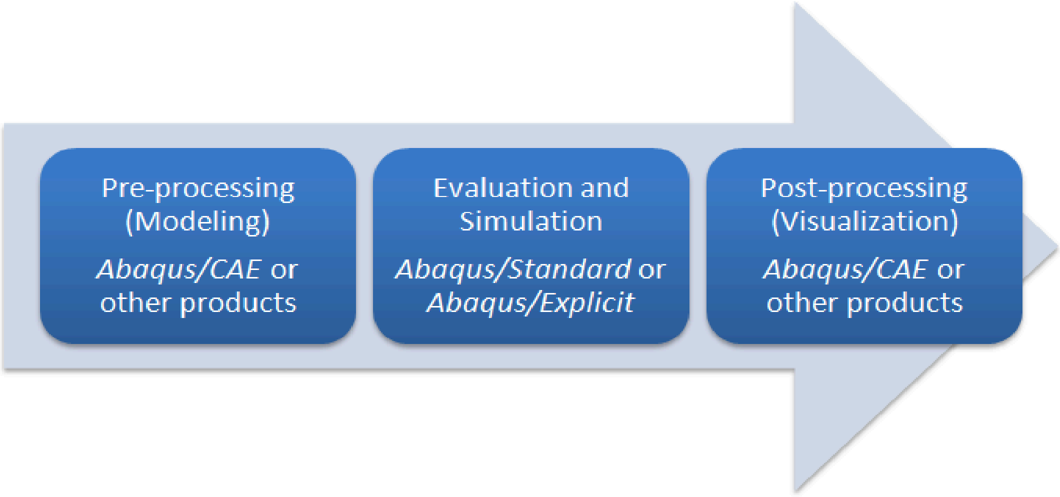


Illustration 3-1: Functional schematic of a FEM-Software (7)

3.1 *Implicit vs. Explicit*

Implicit solvers are getting a solution for the state of a system at a later time by considering the state of the system at a later time and the current state. For this type the equilibrium is reached in each increment and they are numerical more stable.

"Explicit methods calculate the state of the system at a later time from the state of the system at current time without the need to solve algebraic equations."(6) To achieve numerical stability the chosen time (critical time increment) between the increments has to be small enough. The so called critical time increment is influenced by the size of the smallest element (characteristic length) and by the material properties (density, Young's modulus). The finer the mesh and the resulting element size the smaller the critical time increment becomes.

4 Concrete Damaged Plasticity (4)(8)

The Concrete Damaged Plasticity constitutive law was developed for the modelling of concrete and other quasi-brittle materials. The prerequisite should be that the main failure mechanisms are tension-cracks and compressive crushing. A constitutive law for concrete should consider strain softening which occurs by micro-cracks and the hardening behaviour under compressive loading.

"The main characteristic of" plasticity "models is a plasticity yield surface that includes pressure sensitivity, path sensitivity, non-associative flow rule, and work or strain hardening."(8) This constitutive laws are unable to describe the behaviour due to micro-cracking and the consequently stiffness degradation. "On the other hand, the continuum damage theory has been also employed alone to model the material nonlinear behaviour such that the mechanical effects of the progressive micro-cracking and strain softening are represented by a set of internal variables which act on the elastic behaviour at the macroscopic level." (8)

Caused by the behaviour of concrete due to irreversible deformations, inelastic volumetric expansion in compression and crack opening/closing plasticity models or damage models solely are not sufficient.

The CDP-model is a combination of plasticity and damage. Isotropic damaged elasticity is combined with isotropic tensile and compression plasticity to describe the inelastic behaviour of concrete. The irreversible damage which appears during fracturing is characterised by a combination of non-associated multi-hardening plasticity and scalar damaged elasticity.

The degradation of the stiffness is controlled by the isotropic (scalar) damage variables for compression and tension with values are between zero (undamaged) and one (fully damaged). It has to be considered that a damage parameter of one is numerical impossible.

The yield surface is based on the developments of Lubliner et al. and the modifications of Lee and Fenves. (Illustration 4-1). The non-associative plasticity flow rule is an adjustment of the Drucker-Prager hyperbolic function.

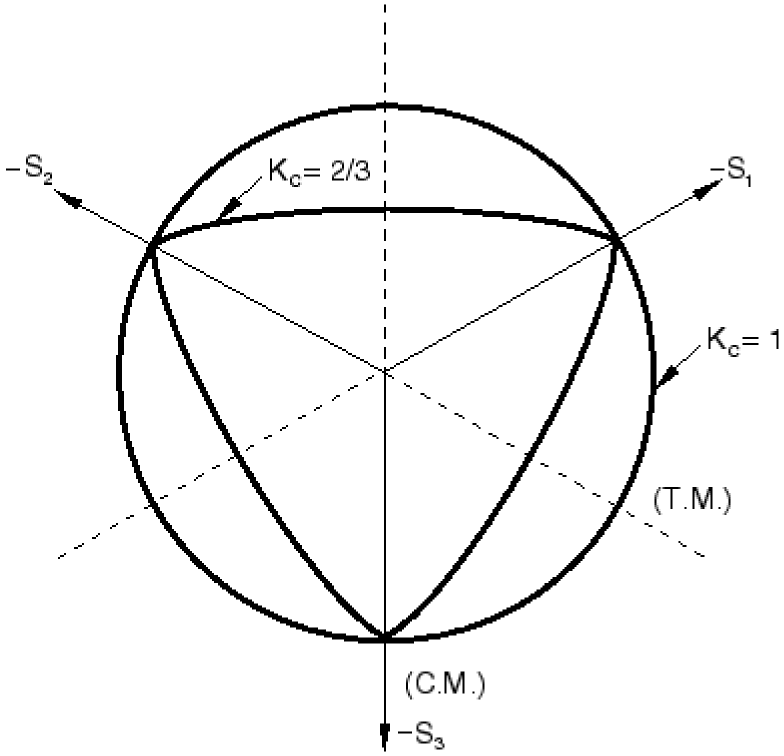


Illustration 4-1: Yield surface in deviatoric plane (4)

The required parameters for the Concrete Damaged Plasticity model are:

- Young's modulus (E_0)
- Poisson's ratio
- Dilation angle
- Eccentricity: ratio of σ_{t0}/σ_{c0} (uniaxial tension strength/uniaxial compressive strength)
- f_{b0}/f_{c0} or σ_{b0}/σ_{c0}: ratio of initial biaxial compressive yield stress/initial uniaxial compressive yield stress
- K_c: ratio of second stress invariant on tensile meridian/second stress invariant on compressive meridian for any given value of the hydrostatic pressure
- Viscosity Parameter
- Compressive Behaviour: yield stress, inelastic strain, damage parameter
- Tensile Behaviour: yield stress, cracking strain, damage parameter

The parameter designation is described in (9).

In Illustration 4-2 and Illustration 4-3 the stress-strain curves of concrete in compression and tension are shown. The behaviour of concrete in compression is linear elastic until the value of initial yield σ_{c0} is reached. After this hardening followed by softening occurs. In tension the behaviour is approximately linear elastic until the value of failure stress σ_{t0} is reached. After this value softening occurs.

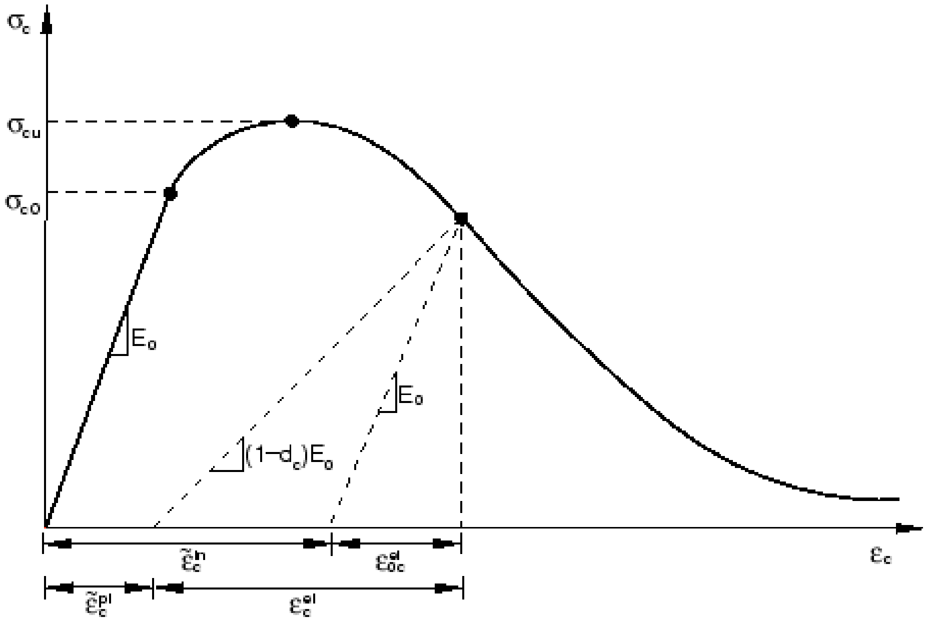


Illustration 4-2: Stress-strain curve of concrete for compression (4)

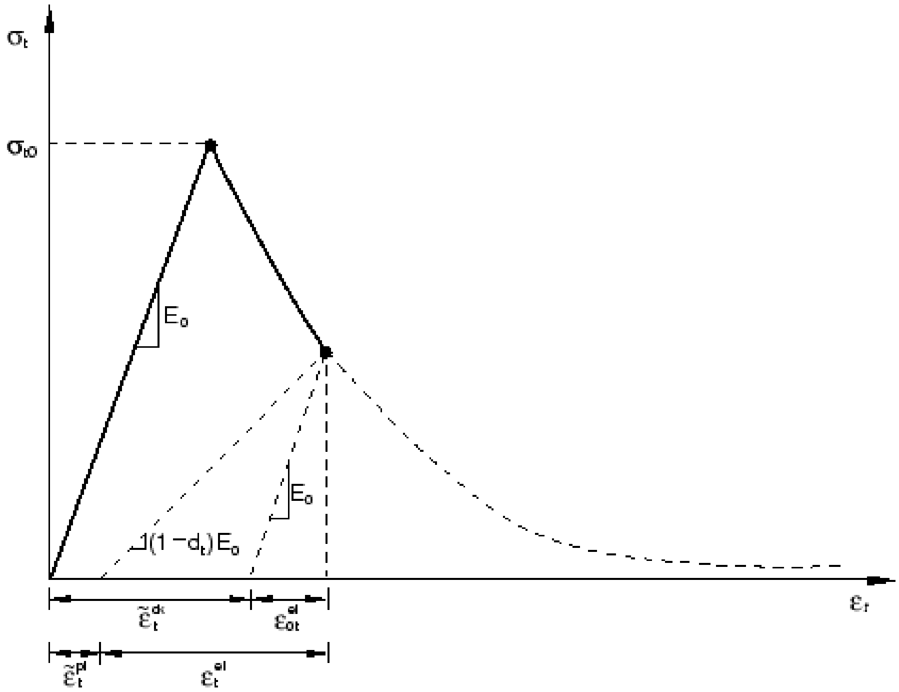


Illustration 4-3: Stress-strain curve of concrete for tension (4)

5 Model

Below the final model (Illustration 5-1) with only one single cut in the centre of the sample is described.

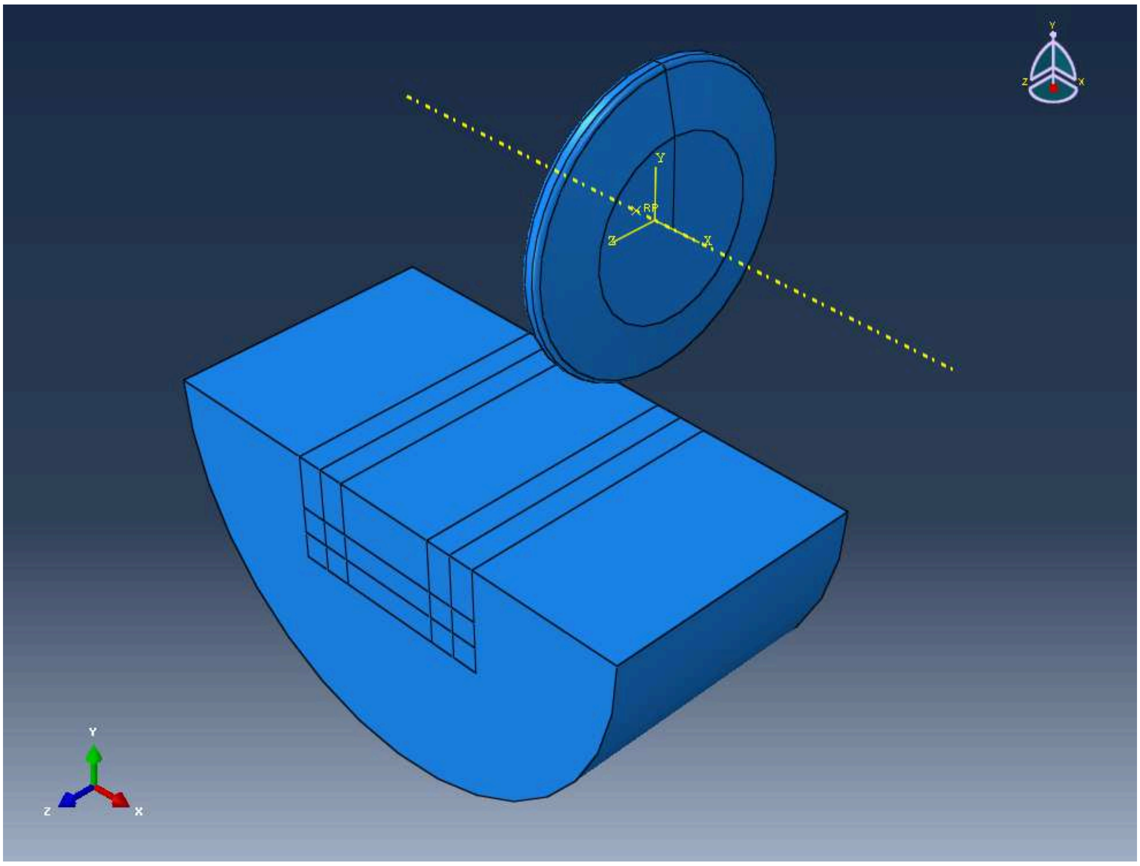


Illustration 5-1: Final model of the Small Scale Rock Cutting Test

5.1 Parts

To reduce the computing effort the sample holder and the adhesive (Illustration 2-1) are replaced by boundary conditions. Also the complete bearing and guide of the disk cutter are reduced to boundary conditions (Chapter 5.6).

5.1.1 Sample

The sample is presented by a halfcylinder with a diameter of 100 mm and a length of 90 mm. Only the first 50 mm of the sample are modelled to reduce the calculation time.

5.1.2 Disc cutter

Because the deformations of the disc cutter are negligible to the rock-deformations and predictions of the disc cutter wear are not necessary, this part is modelled analytical rigid. The dimensions are given in Illustration 5-2 and Illustration 5-3 shows the implementation in Abaqus/CAE.

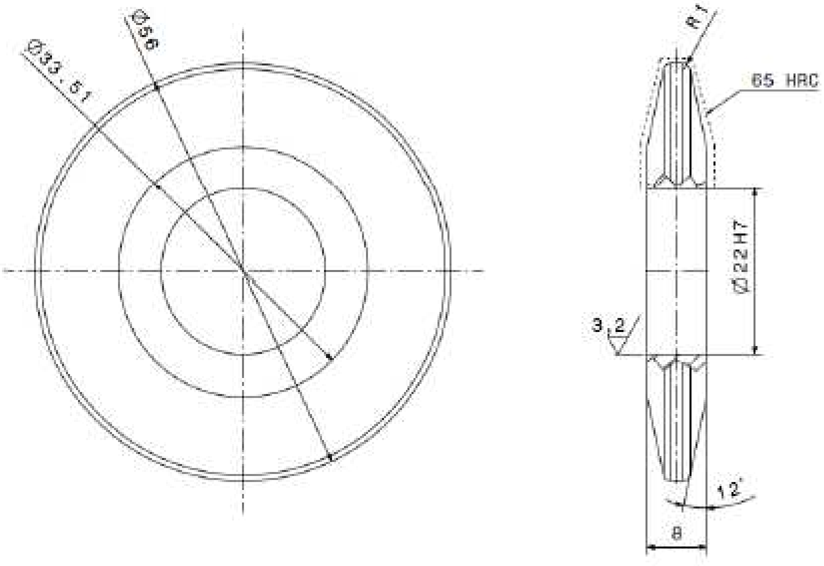


Illustration 5-2: Disc cutter geometries (all dimensions in mm)(1)

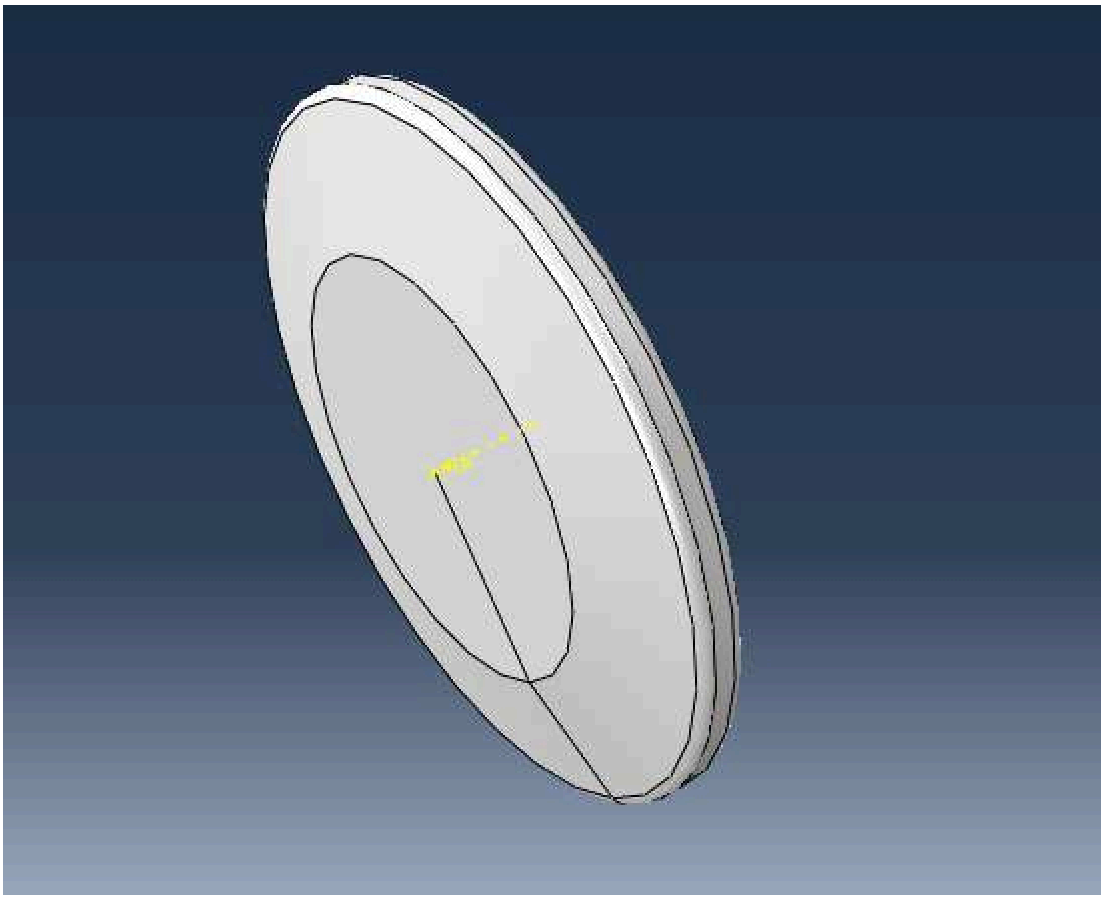


Illustration 5-3: Disc cutter implementation in Abaqus/CAE

5.2 Properties

5.2.1 Sample

The sample in this simulation is a Granite and its material parameters are given in the hereinafter tables. The parameter designation is described in (9).

5.2.1.1 Density

Density, g/cm³
2,70

Table 5-1: Input parameter density

5.2.1.2 Elasticity

Young's Modulus , Pa	Poisson's ratio, -
6,50E+10	0,117

Table 5-2: Input parameter elasticity

5.2.1.3 Concrete Damaged Plasticity:

Dilatation Angel, °	Eccentricity, -	fb0/fc0, -	K, -
35	0,04	1,11	0,667

Table 5-3: Input parameter CDP

5.2.1.4 Compression Behaviour

Yield Stress, Pa	Inelastic Strain, -	Damage Parameter, -
1,12E+08	0,000E+00	0,000E+00
1,38E+08	3,600E-05	0,000E+00
1,50E+08	8,200E-05	0,000E+00
1,45E+08	1,700E-04	3,333E-02
7,50E+07	1,546E-03	5,000E-01
7,40E+07	5,000E-03	5,070E-01

Table 5-4: Input parameter compression behaviour

5.2.1.5 Tension Behaviour

Yield Stress, Pa	Cracking Strain, -	Damage Parameter, -
6,000E+06	0,000E+00	0,000E+00
3,000E+06	9,660E-05	5,000E-01
2,900E+04	1,200E-04	5,170E-01

Table 5-5: Input parameter tension behaviour

5.2.2 Disc cutter

The properties of the disc cutter are assigned to the reference point (RP) (Illustration 5-3). Only the mass and the moment of inertia about the rotation axis have to be considered for an analytical rigid part.

Mass, kg	I, kgm²
0,16	6,27E-05

Table 5-6: Properties of the disc cutter

5.3 Assembly

The two parts (sample and disc cutter) of the model are composed in such a way, that a constant penetration depth of 1,5 mm is ensured.

5.4 Step

Caused by the use of an user subroutine (chapter 6), written for Abaqus/Explicit, a dynamic, explicit step with a duration of 0,07 seconds is chosen. To accelerate the simulation a mass scaling factor of 1,03 is entered.

5.5 Interaction

For the contact between the disc cutter, sample and parts of the fractured sample a general contact definition is implemented. The tangential behaviour is set to penalty (μ=0,4) and the normal to hard contact. If an element deletion routine is used an inner surface has to be created (Chapter 11.3).

5.6 Boundary Conditions

5.6.1 Sample

In Illustration 5-4 the boundary conditions of the sample are represented. The adhesive (red) is modelled as a boundary condition fixing the sample in all directions. The front and end face (yellow) are fixed against movement into the direction of the z-axis.

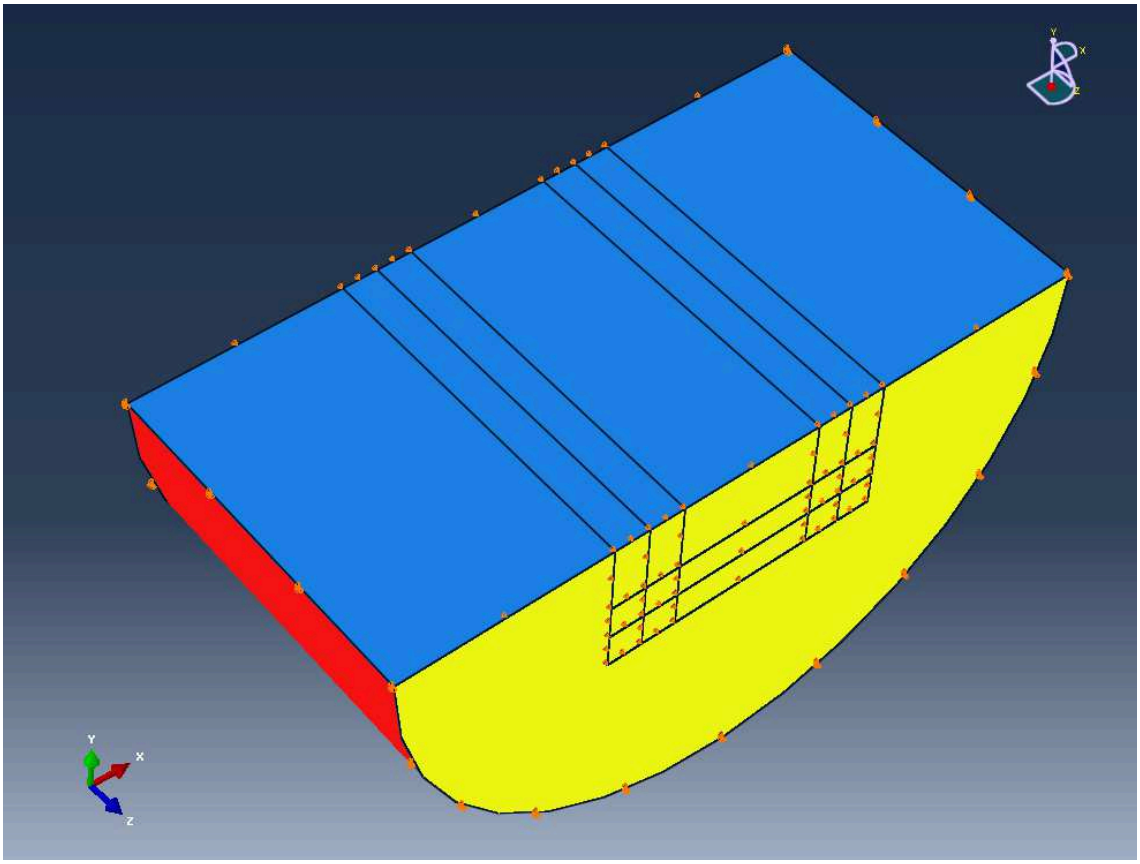


Illustration 5-4: BC of the sample

5.6.2 Disc cutter

The disc cutter is fixed against movement into the x and y axis and also against rotation about the y and z axis. To get an efficient computing time the velocity of the disc cutter in z direction is 1 m/s in the dynamic, explicit step. The effects that can occur caused by the high velocity are given in chapter 7.3. The rotation about the x axis results by the velocity in z direction and the interaction (friction) with the sample. All boundary conditions are assigned to the reference point.

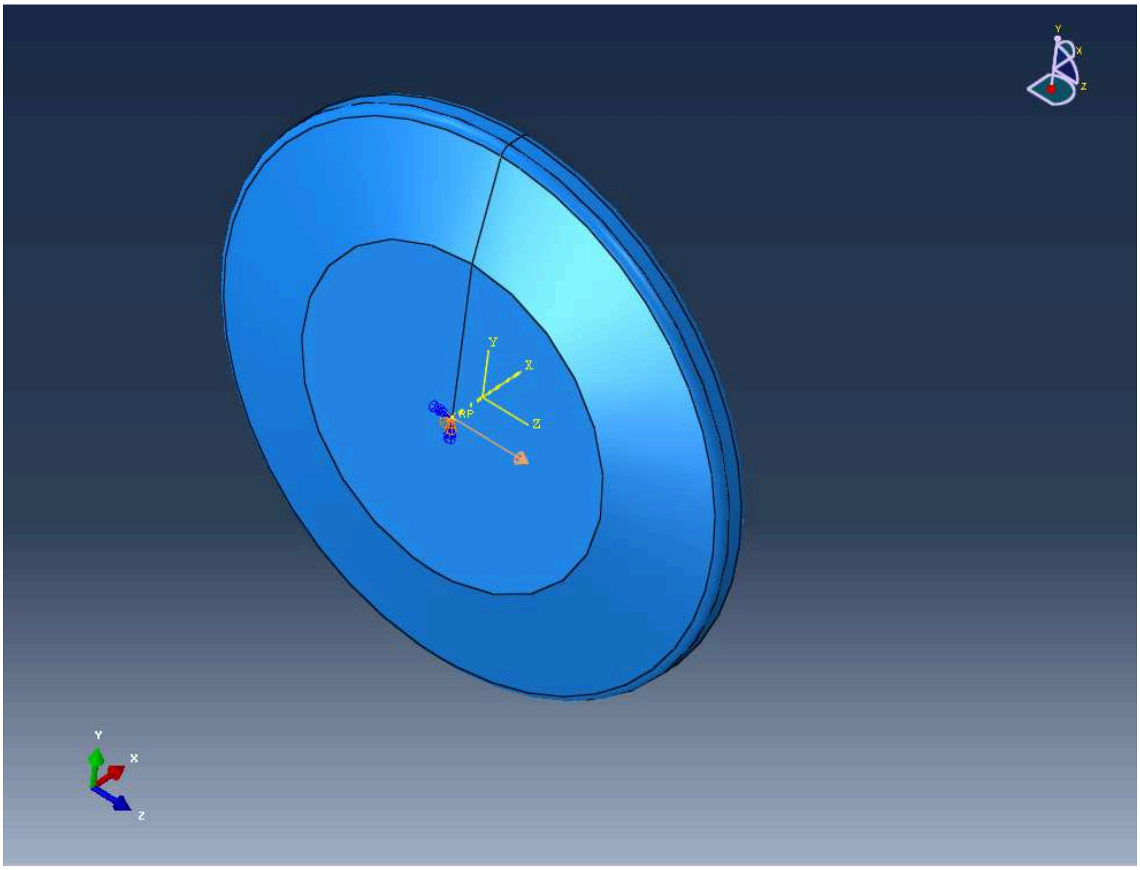


Illustration 5-5: BC of the disc cutter

5.7 Mesh

In Illustration 5-6 the partition of the sample is shown. The dimensions are given in Illustration 5-7. All regions except region 5 are meshed with C3D8R elements. The colours in Illustration 5-6 matter:

- Green: structured mesh
- Yellow: sweep mesh
- Red: free mesh

The region 1 in the middle of the sample is meshed by cubes with a length of 0,3 mm. The regions 2 are used to get a coarser mesh into the radial direction (0,3 mm to 2 mm) (Illustration 5-8). For a coarser mesh in the axial direction the regions 3 are applied (0,3 mm to 3 mm). The connections between the regions 2 and 3 are the regions 4 and 5. It is necessary to mesh the regions 5 with tetrahedron elements (C3D10M) otherwise a meshing is impossible. Region 6 represents the rest of the sample.

In Illustration 5-8 the final mesh of the sample is shown.

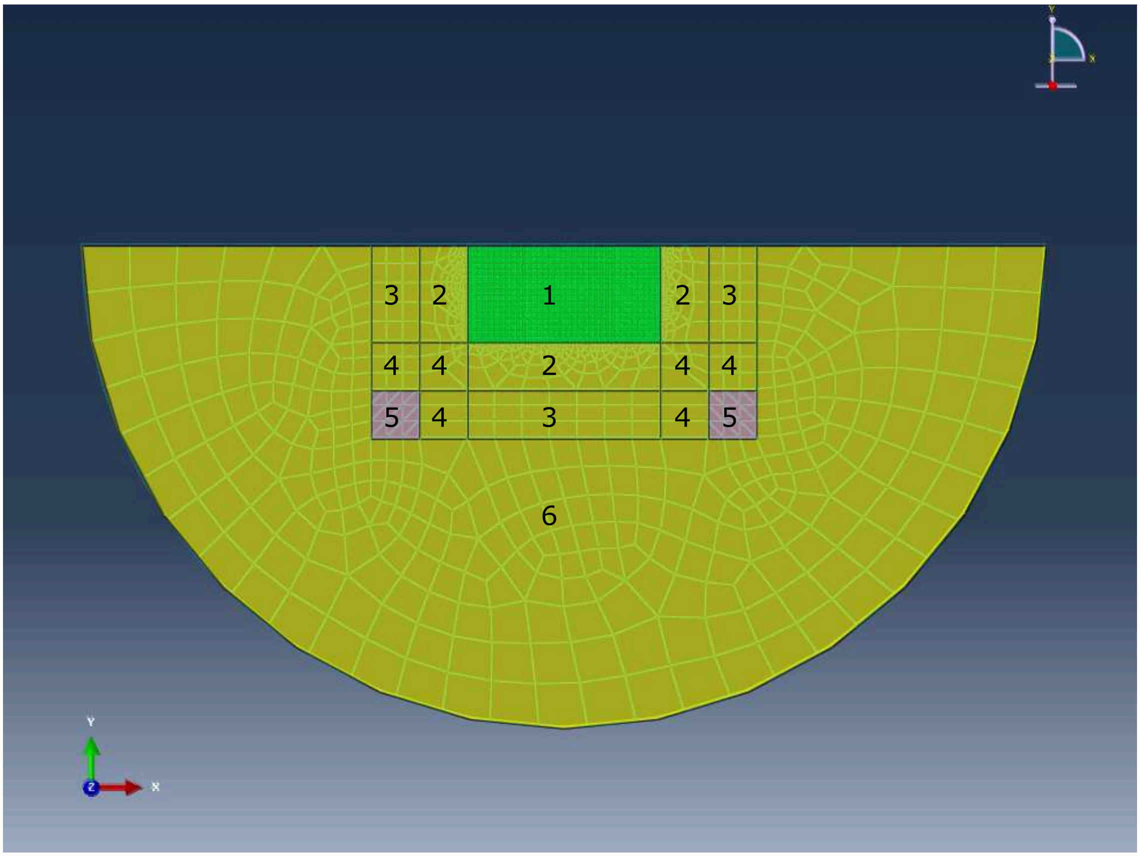


Illustration 5-6: Partition of the sample

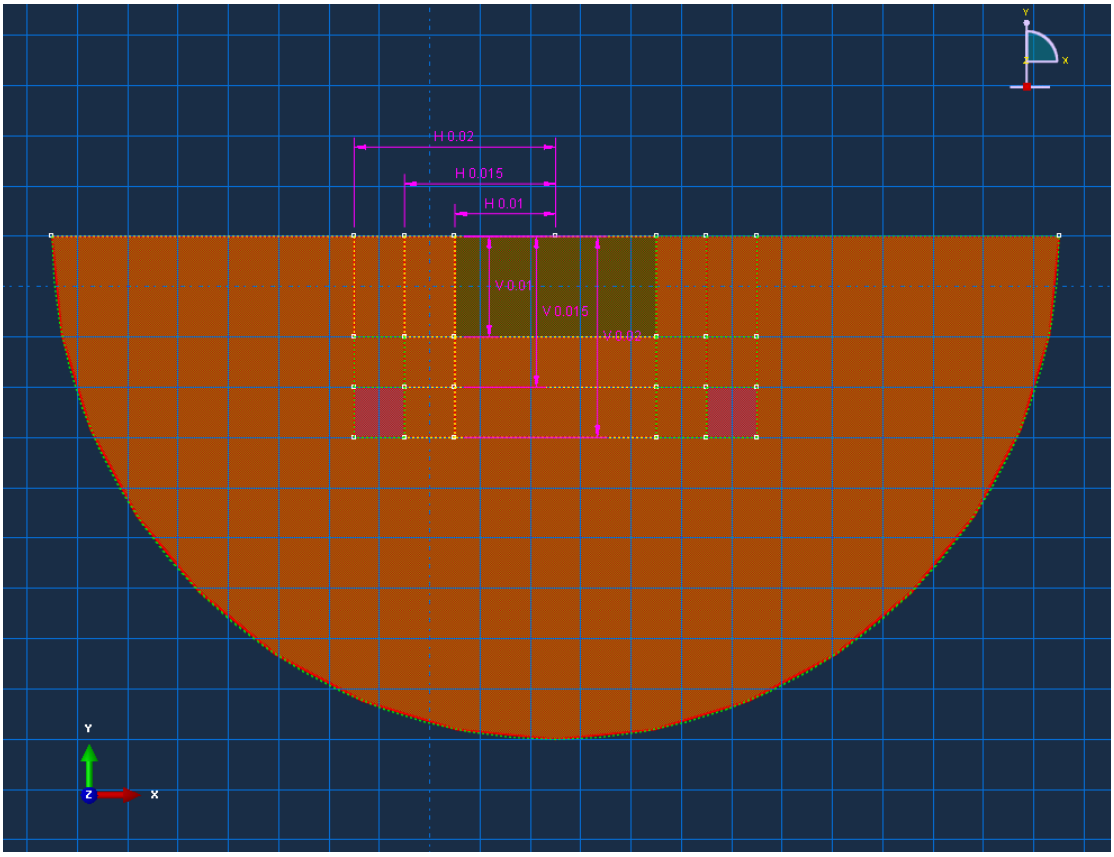


Illustration 5-7: Dimensions of the regions (all dimensions in m)

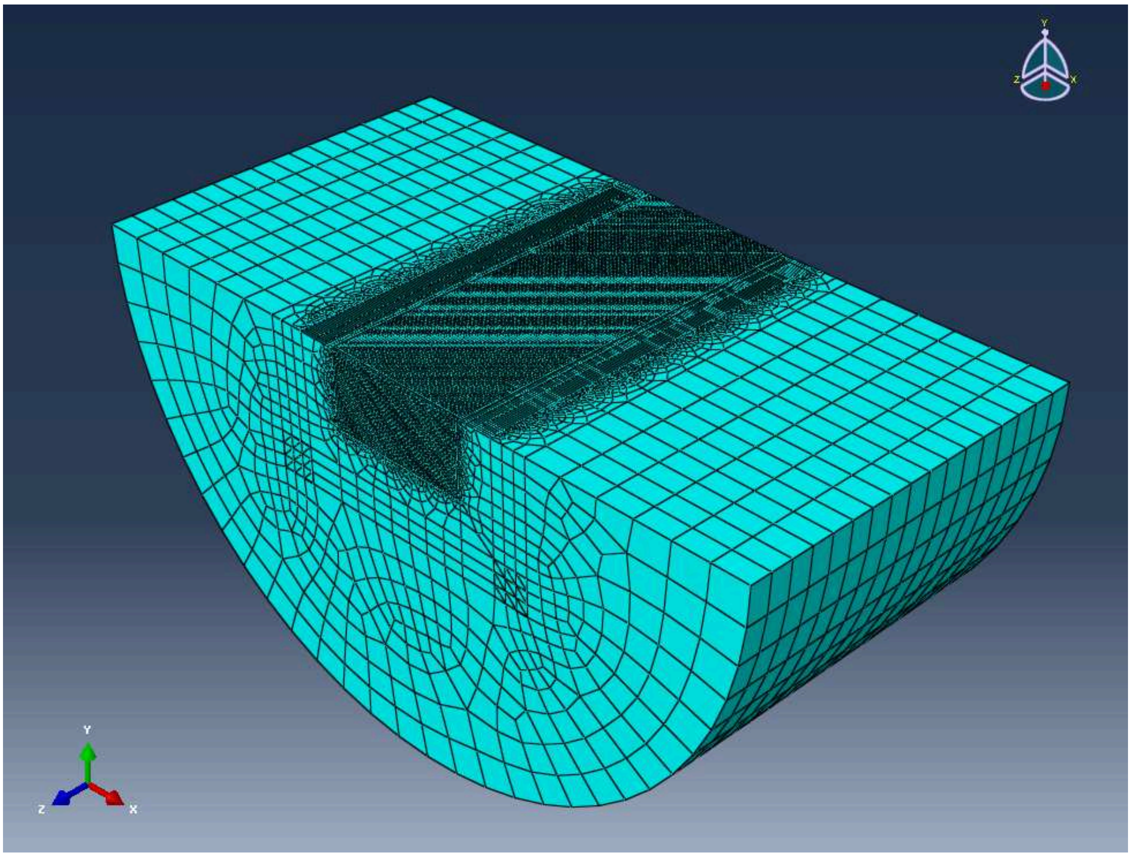


Illustration 5-8: Final mesh of the sample

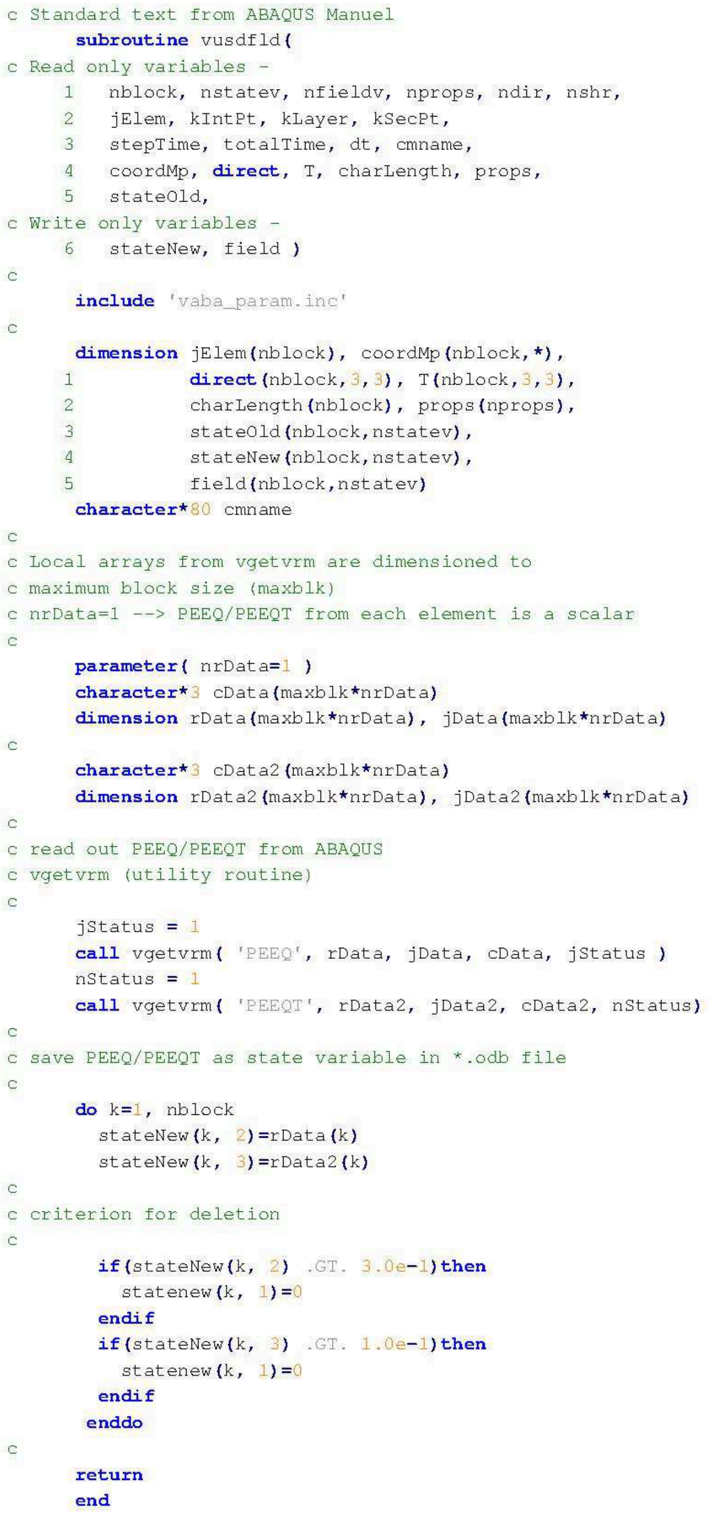
6 Subroutine (4)(10)

The Concrete Damaged Plasticity Model, which is implemented in Abaqus, is unable to delete the fully distorted elements. The critical time increment gets much smaller by this few elements and therefore the computing effort is rising up. Also the stability of the calculation can be influenced by these elements. For this purpose an user-subroutine has to be written. The coding language of this so called VUSDFLD is FORTRAN.

The chosen criterions for deletion are the compressive equivalent plastic strains (PEEQ) and the tensile equivalent plastic strains (PEEQT). The decision criterion for the deletion and the effects of the deletion are given in chapter 7.3.2. The written subroutine is illustrated below (Illustration 6-1).

For the first tests of this user subroutine uniaxial compression tests were used (Illustration 6-3 and Illustration 6-3). After successful completion this subroutine was used for the simulation of the Small Scale Rock Cutting Test.

The interaction between Abaqus and an user subroutine is given in chapter 11.1.



```
c Standard text from ABAQUS Manuel
      subroutine vusdfld(
c Read only variables -
     1   nblock, nstatev, nfieldv, nprops, ndir, nshr,
     2   jElem, kIntPt, kLayer, kSecPt,
     3   stepTime, totalTime, dt, cmname,
     4   coordMp, direct, T, charLength, props,
     5   stateOld,
c Write only variables -
     6   stateNew, field )
c
      include 'vaba_param.inc'
c
      dimension jElem(nblock), coordMp(nblock,*),
     1          direct(nblock,3,3), T(nblock,3,3),
     2          charLength(nblock), props(nprops),
     3          stateOld(nblock,nstatev),
     4          stateNew(nblock,nstatev),
     5          field(nblock,nstatev)
      character*80 cmname
c
c Local arrays from vgetvrm are dimensioned to
c maximum block size (maxblk)
c nrData=1 --> PEEQ/PEEQT from each element is a scalar
c
      parameter( nrData=1 )
      character*3 cData(maxblk*nrData)
      dimension rData(maxblk*nrData), jData(maxblk*nrData)
c
      character*3 cData2(maxblk*nrData)
      dimension rData2(maxblk*nrData), jData2(maxblk*nrData)
c
c read out PEEQ/PEEQT from ABAQUS
c vgetvrm (utility routine)
c
      jStatus = 1
      call vgetvrm( 'PEEQ', rData, jData, cData, jStatus )
      nStatus = 1
      call vgetvrm( 'PEEQT', rData2, jData2, cData2, nStatus)
c
c save PEEQ/PEEQT as state variable in *.odb file
c
      do k=1, nblock
        stateNew(k, 2)=rData(k)
        stateNew(k, 3)=rData2(k)
c
c criterion for deletion
c
        if(stateNew(k, 2) .GT. 3.0e-1)then
          statenew(k, 1)=0
        endif
        if(stateNew(k, 3) .GT. 1.0e-1)then
          statenew(k, 1)=0
        endif
       enddo
c
      return
      end
```

Illustration 6-1: Subroutine

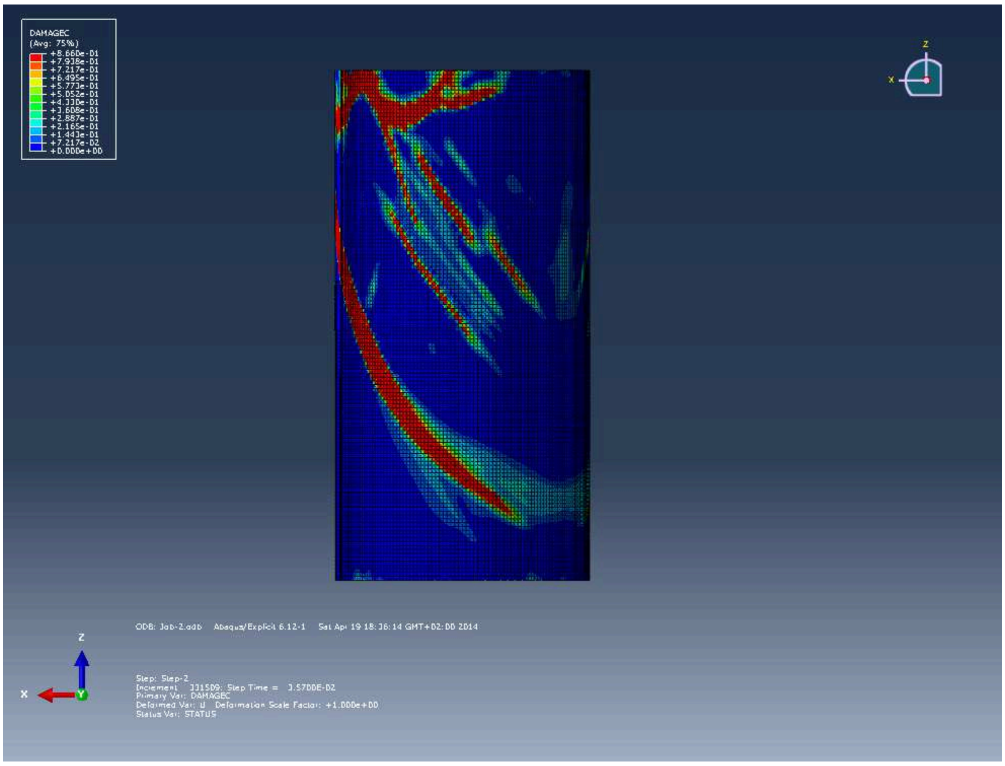


Illustration 6-2: Uniaxial compression test before deletion

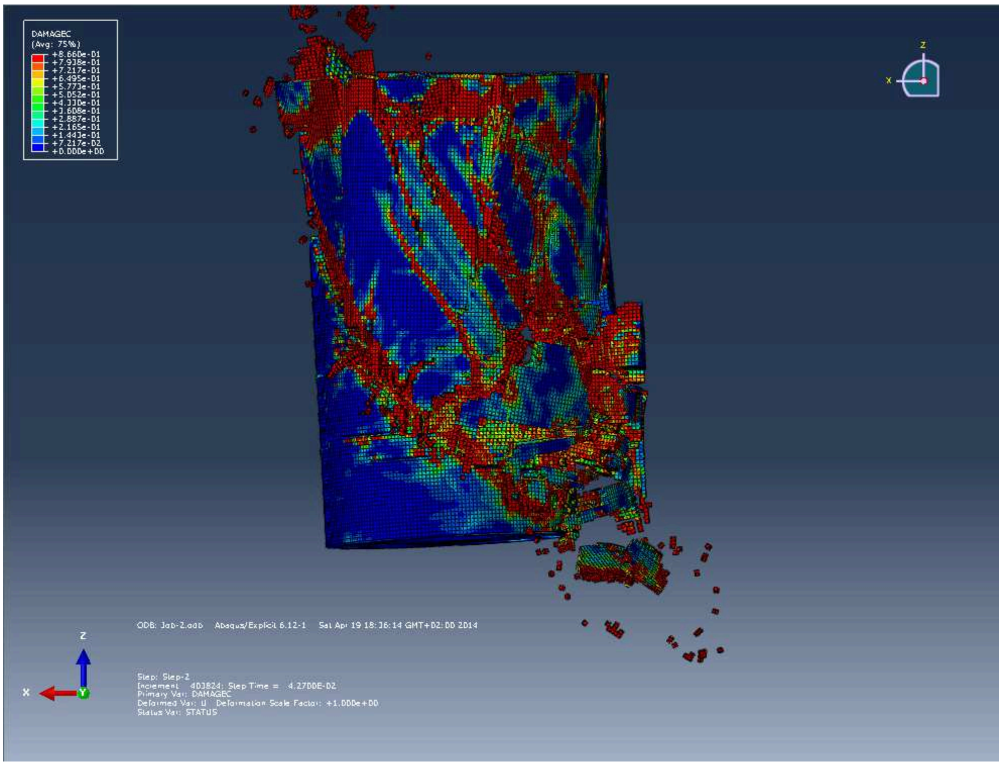


Illustration 6-3: Uniaxial compression test after deletion

7 Conclusion

7.1 Concrete Damaged Plasticity

One of the biggest challenges is the designation of the post failure parameters from the uniaxial compression and tension tests. A typical stress-strain curve as result of an uniaxial compression test is shown in Illustration 7-1. In contrary to reinforced concrete (Illustration 4-2), where the reinforcement takes the loads after cracking of the concrete in the post peak section, it can be seen, that after cracking no residual strength is existing. Also after the crack in the uniaxial tension test occurs the contact between the two parts of the sample is getting lost (Illustration 7-2) and no values of the post peak behaviour can be registered (Illustration 7-3). By reinforced concrete the reinforcement carries the tension loads after cracking of the concrete. The post failure parameters are important for the representation of the damaged material next to the disc cutter and its remaining strength. So the post failure parameters of the uniaxial compression and tension of the tests can't be used for the Small Scale Rock Cutting Test.

Too low residual strength leads to too low rolling and contact forces caused by the low resistance against deformation of the damaged elements. A too high residual strength has an impact on the impairment ahead. The material in front of the disc cutter is damaged too early and the disc cutter is "cutting" damaged material with lower strength and Young's modulus.

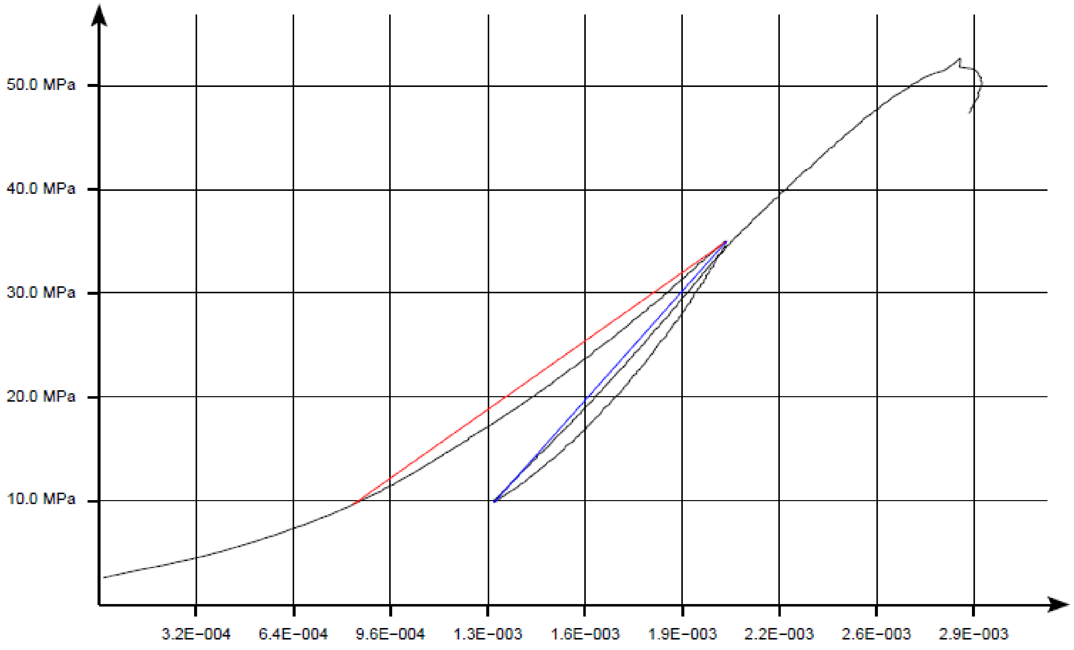


Illustration 7-1: Stress-strain curve of an uniaxial compression test

Illustration 7-2: Uniaxial tension test (11)

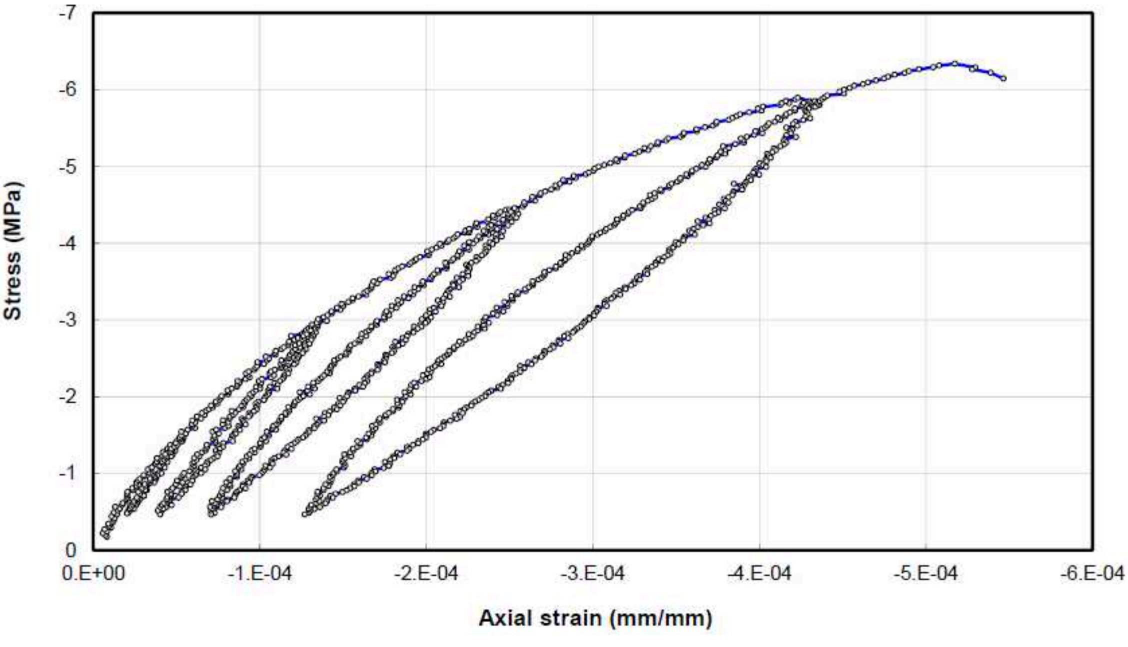


Illustration 7-3: Stress-strain curve of an uniaxial tension test (11)

The Concrete Damaged Plasticity model is a constitutive law with smeared cracking and scalar damage parameters. So the rock/element is impaired similar in all directions. In Illustration 7-4 is shown that a real crack would reduce the strength in y direction but the rock/element strength in x direction is nearly uninfluenced. To reduce the impact of the direction independent damage parameters a very fine mesh with an element length of 0,3 mm was created. In Illustration 7-5 a comparison of the real sample with cracks and the numerical mesh is shown. In Illustration 7-6 it becomes obvious that the crack wide is about one element length.

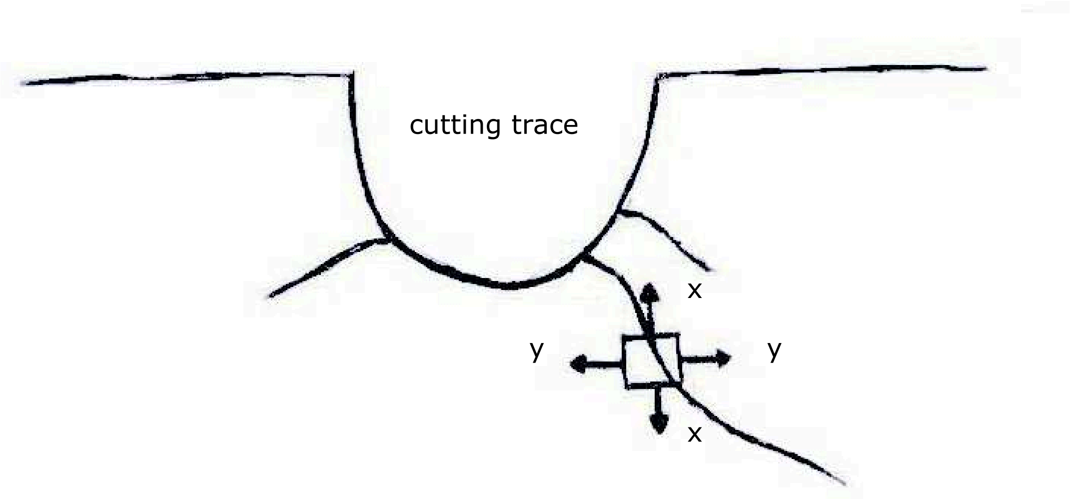


Illustration 7-4: Direction dependence of the material properties

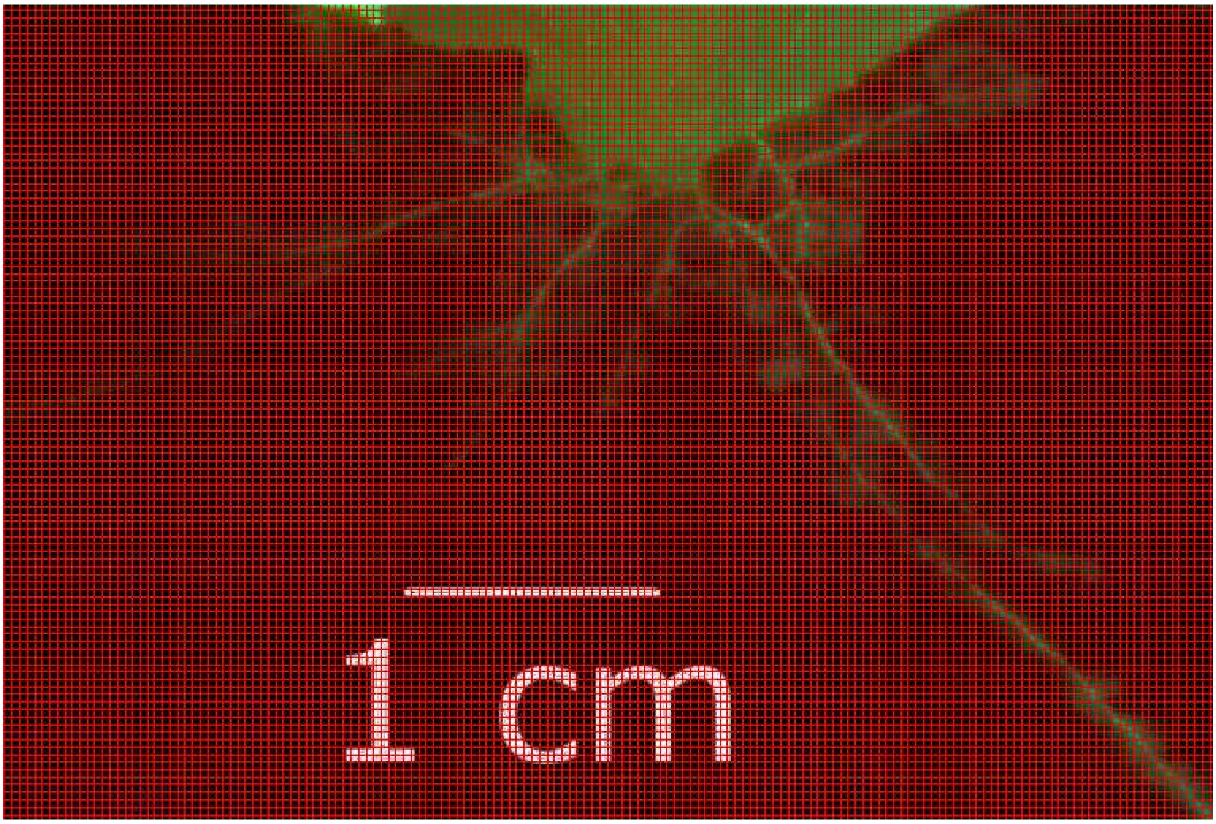


Illustration 7-5: Real sample with mesh of numerical simulation (2)

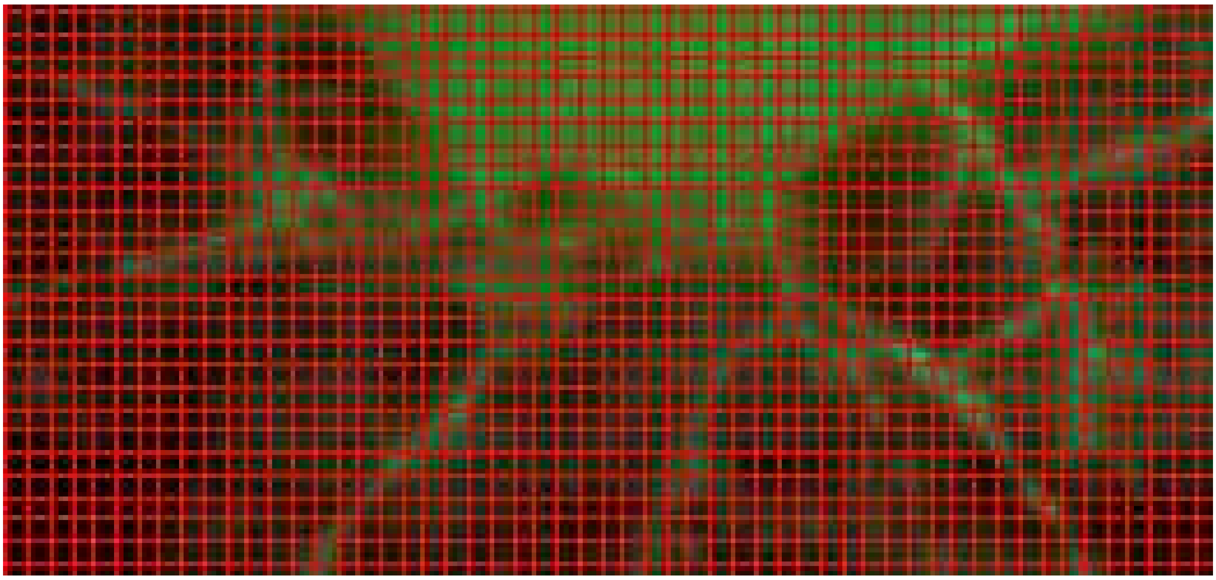


Illustration 7-6: Enlarged part of Illustration 7-5

In Illustration 7-7 the influence of the dilation angle on the results is depicted. The calculations were performed with an implicit model with a mesh size of 5 mm (Illustration 7-8). For this scope with a penetration of 1,5 mm, of course is this a too coarse mesh, but the major influence of this parameter can be seen. The bigger the dilation angle the higher are the rolling forces (Fr) and the normal forces (Fn).

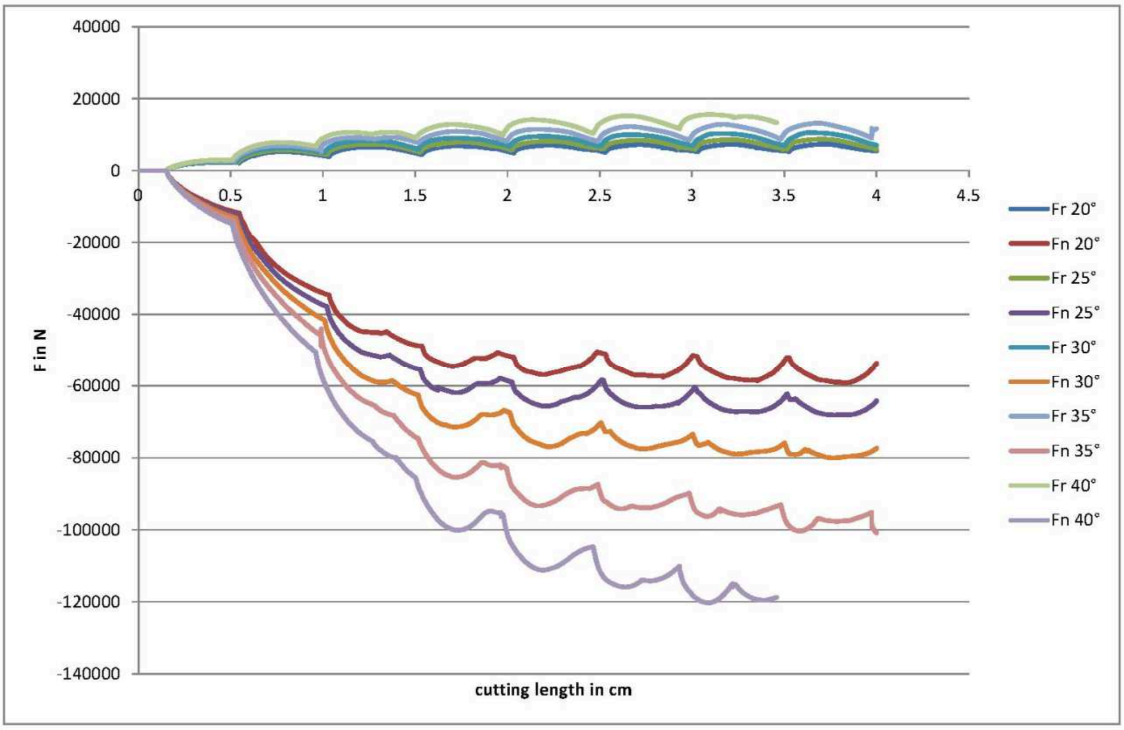


Illustration 7-7: Influence of the dilation angle

7.2 Implicit

The first models were built with dynamic, implicit steps. The mesh was very course with an average element length of 5mm and the disc was implemented as deformable part with the material properties of steel. The time for calculation was about four hours for one single, central cut. Through refining of the mesh to an element length of 3 mm the simulation took nearly ten hours. The next mesh size of 2 mm leads to 48 hours computing effort. To have a fine enough mesh under the disc cutter a 0,3 mm small mesh was chosen (Illustration 7-5). This made an efficient simulation with the implicit solver nearly impossible.

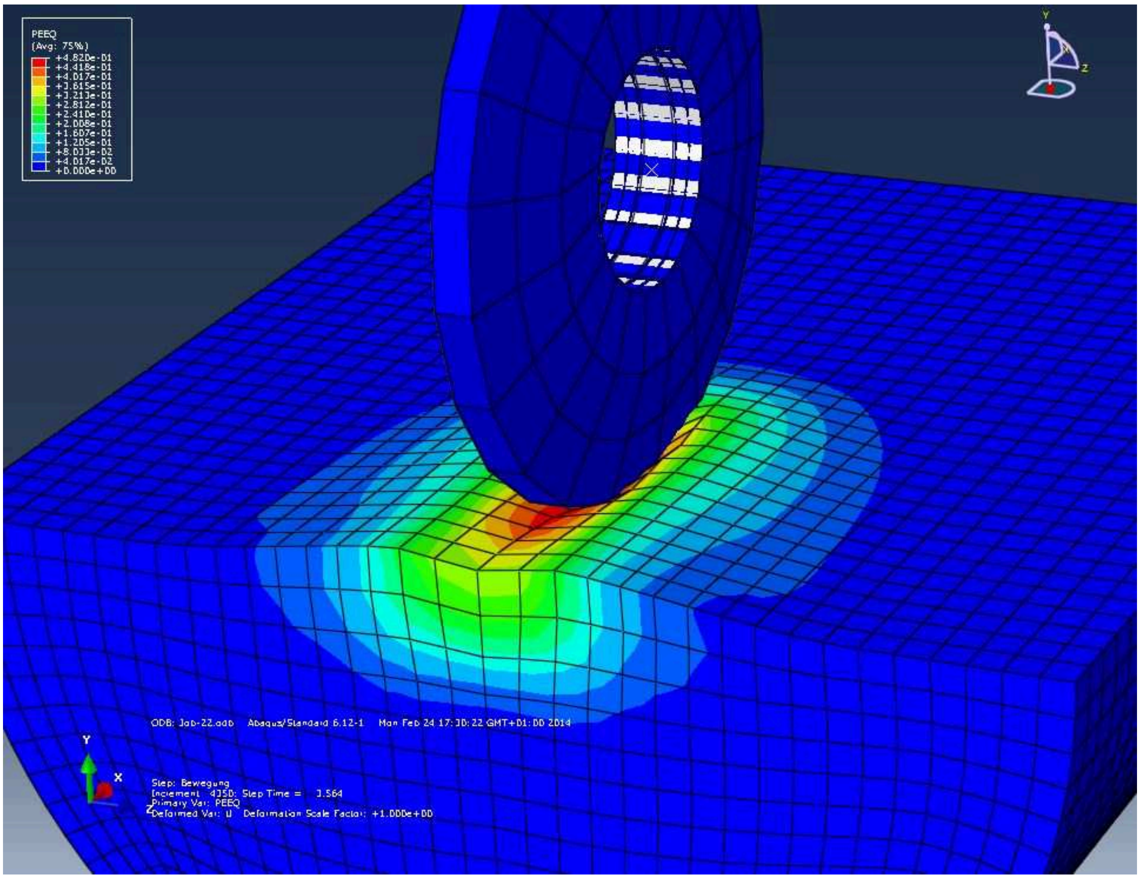


Illustration 7-8: First model with dynamic, implicit step

The biggest advantage of the implicit solvers is the independence of the loading velocity. In each increment the equilibrium of state has to be reached. The increment size is detected automatically to get convergence.

7.2.1 Extended Finite Element Method (XFEM) (4)

It was tried to envisage the Extended Finite Element Method to describe the crack pattern with implicit time integration. But this method is only implemented for static steps so it’s useless for a dynamic process like the Small Scale Rock Cutting Test. Also the fact that it is impossible to parallelize the calculation is a big disadvantage. Illustration 7-9 shows the trying of the simulation of an uniaxial compression test with XFEM and Illustration 7-10 the resulting fracture face.

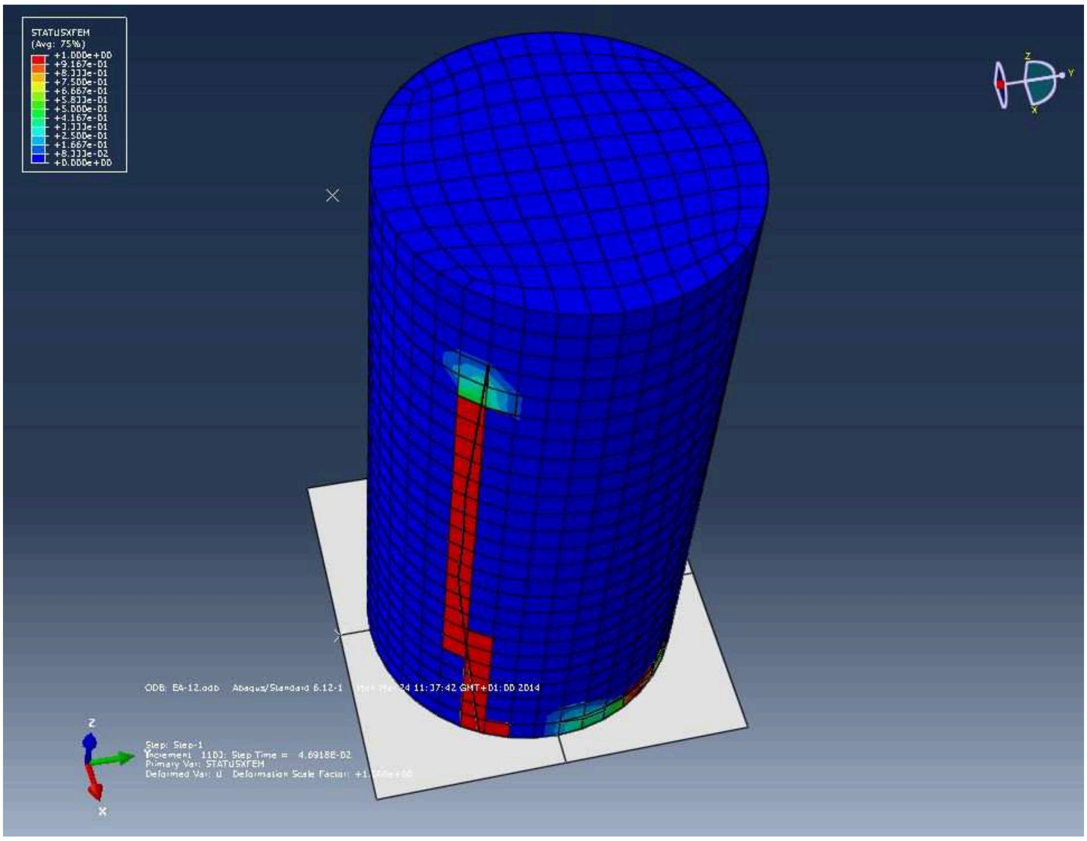


Illustration 7-9: Uniaxial compression test with XFEM

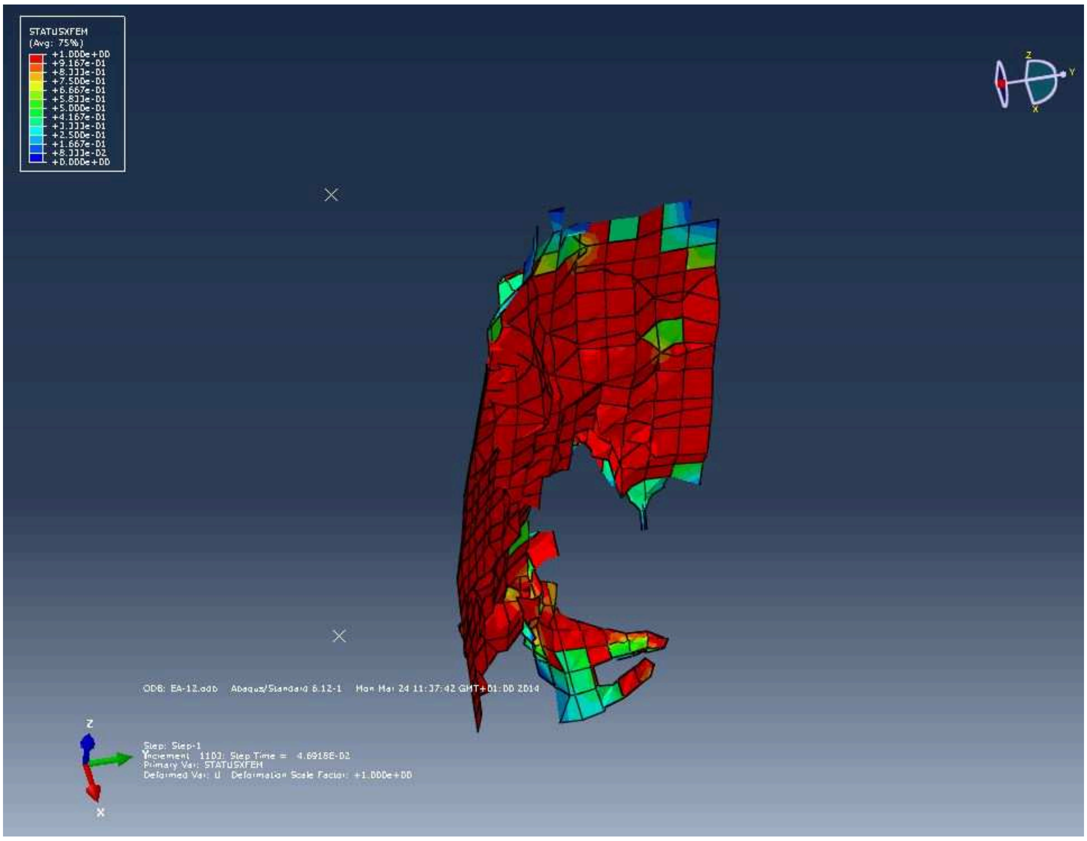


Illustration 7-10: Crack pattern with XFEM

7.3 Explicit

For explicit solvers the influences of the loading velocity and the mesh sensitivity, a common problem in the numerical analyse of problems, have to be considered (9). Caused by the way of solving the job too high (loading-) velocity can occur dynamic effects like waves respectively oscillations in the sample. These waves can be damped by infinite elements as non-reflecting ("quiet") boundary condition at the end face of the sample. How to create this special element type is explained in chapter 11.1.

Also the mass scaling factor has to be chosen carefully. The calculation is fastened by raising the critical time increment, but this factor can change the whole dynamic of the problem and lead to wrong results. Explicit solvers deliver results although they are completely wrong and the equilibrium conditions are violated.

7.3.1 Smoothed Particle Hydrodynamics (SPH) (4)

Depending on the solution technique this method is only available in Abaqus/Explicit. It was tried to convert the crushed material into SPH particles to simulate the produced powder underneath the disc cutter. Smoothed Particle Hydrodynamics is a mesh-free method without nodes and elements. Only a collection of particles which are interacting with each other represent the part.

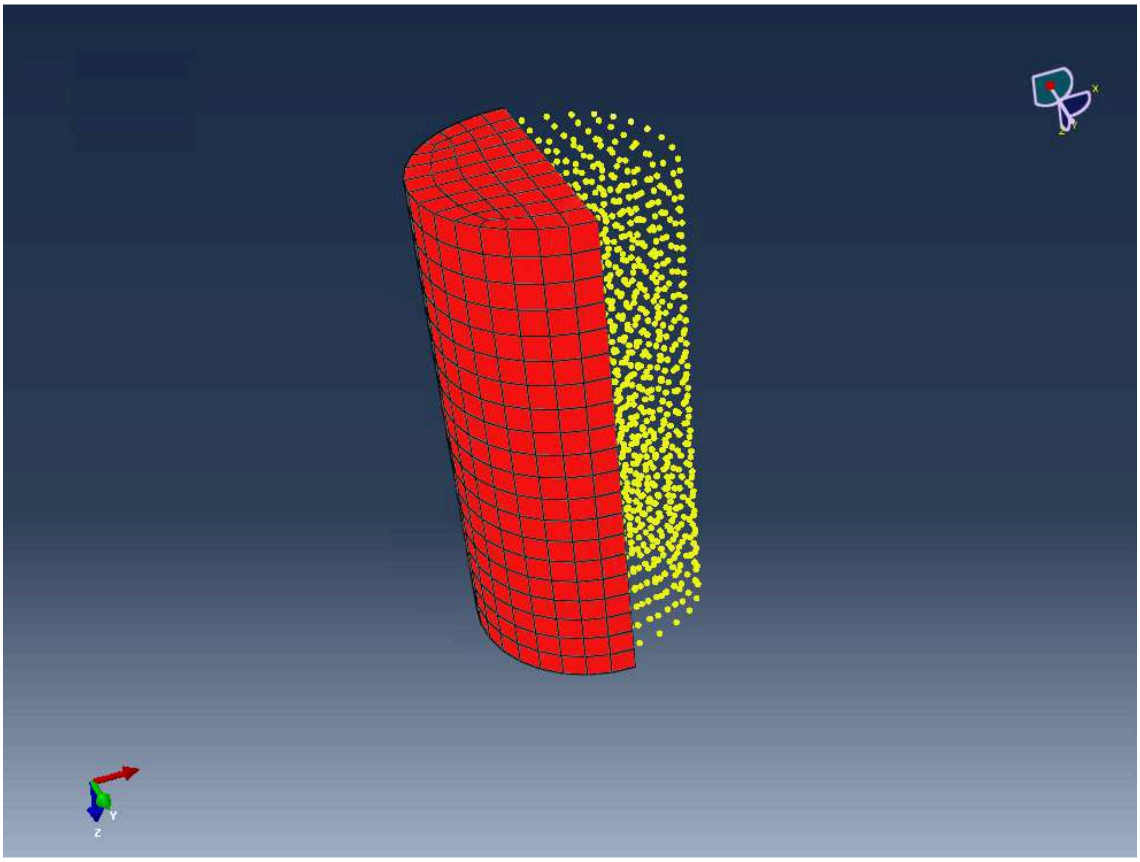


Illustration 7-11: Comparison between normal (red) mesh and SPH method (yellow)

The biggest disadvantage of this method is the fact that it is impossible to parallelize the simulation. This causes a much longer calculation time. Also due to the problems with the criterion for conversion this strategy was discarded. As implemented conversion criterion only time, maximum principal stress and maximum principal strain is available.

7.3.2 Subroutine (4)

The implementation of a subroutine with deletion is only suitable in Abaqus/Explicit. There are numerical instabilities and convergence problems if an element is deleted in Abaqus/Standard, caused by the violation of the state of equilibrium, and the writing of such a subroutine is very difficult and time-consuming.

As subroutine the VUSDFLD (user subroutine to redefine field variables at a material point) has been selected. This subroutine is an addition to the Concrete Damaged Plasticity model which has no possibility to delete the elements. The VUMAT (user subroutine to define material behaviour) subroutine is incongruous for this problem, because the CDP model is already implemented in Abaqus and it has any sense to implement this complete model into a subroutine.

As interface between Abaqus and subroutine the utility routine “vgetvrm” is chosen to read out the in Abaqus calculated values for the subroutine. With regard to the used utility routine, it has to be noted that for this problem only a few output variables that are supported can be taken into consideration. Namely:

- S: All stress components
- LE: All logarithmic strain components
- PE: All plastic strain components
- PEEQ: Equivalent plastic strain
- PEEQT: Equivalent plastic strain in uniaxial tension

Caused by the post failure behaviour of the CDP model with softening, deletion via stress components or equivalent stress (σ_{del}) is not useful. Each stress underneath the peak has no clear indication (Illustration 7-12).

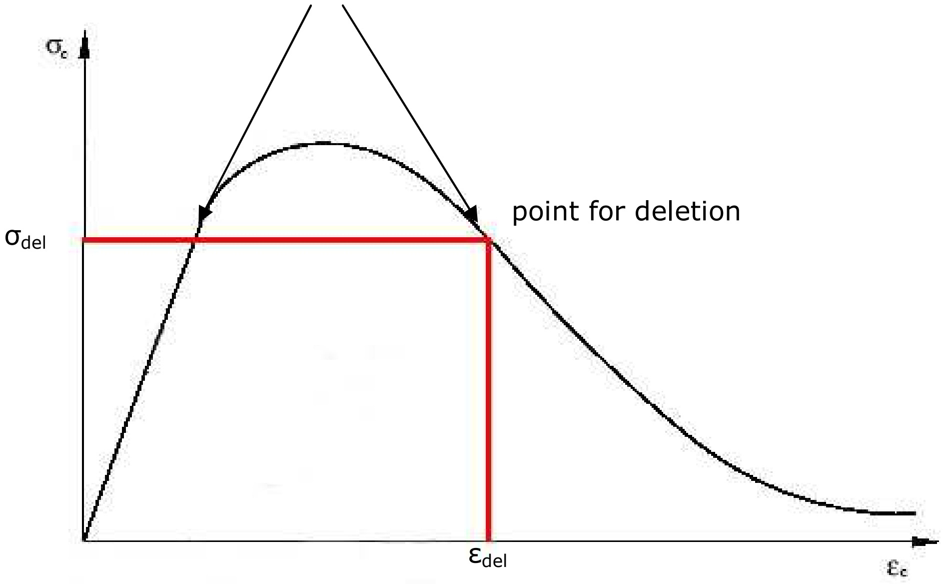


Illustration 7-12: Stress-strain curve (4)

Only strains are indicated clear (ε_{del}). To keep the subroutine as short as possible the PEEQ und PEEQT are selected as deletion criterion, because for the LE tensor and the PE tensor an equivalent strain has to be determined. Short because the whole do-loop of the subroutine (Illustration 6-1) is performed in each increment (about 5M up to 10M increments) for each element (about 700k).

For the selected deletion value it has to be considered that a too early deletion reduces the rolling force – loss of contact - and a too late deletion raises up the time needed for the calculation. No rock parameter can be used to estimate these values and also from the simulation of the uniaxial tests they can't be derived. For the simulation of uniaxial rock tests (Illustration 7-1) deletion of elements in the post peak section has no influence, because after cracking almost no residual strength is present.

For the use of a subroutine with element deletion an inner surface has to be defined. Otherwise after the first element row (with exterior surface) is deleted, the contact between disc cutter and sample will be loosed. The creating of an inner surface is illustrated in chapter 11.3.

7.4 Result

For the given material properties (chapter 5.2.1) the resulting forces are given in Illustration 7-13. In comparison to the real Small Scale Rock Cutting Test (mean of about 1,9 kN) the mean of the calculated rolling force with the Concrete Damaged Plasticity constitutive law and the entered post failure parameters is about 1 kN.

In Illustration 7-13 "plasticity" means that the damage parameters were set to zero for the whole stress-strain curve, which represents a plasticity constitutive law. The average of the rolling force with pure plasticity is about 1,6 kN which is much nearer to the real test, but no damage evaluation/crack propagation is possible for such a material law.

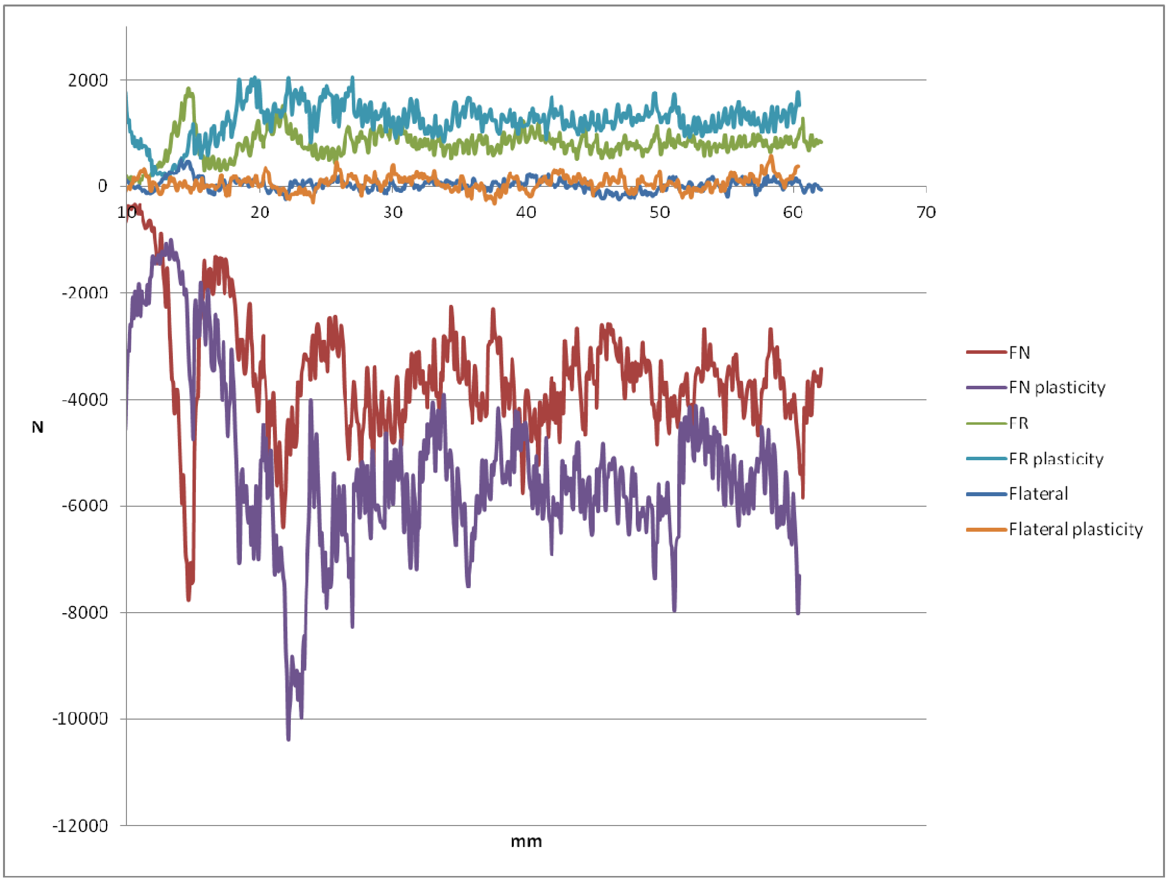


Illustration 7-13: Results of the simulation

As already mentioned a too low rolling force can be caused by a too low residual strength or a too early deletion of the distorted elements. For the next tests the deletion criterion should be raised.

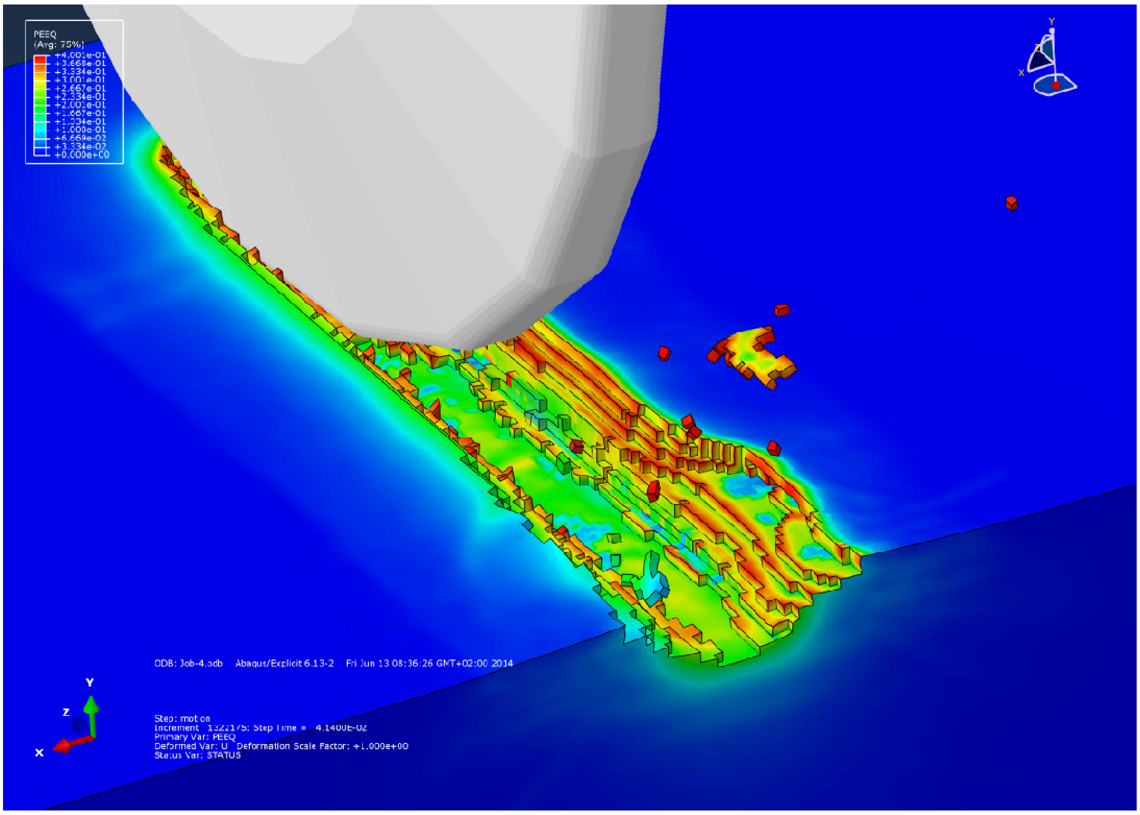


Illustration 7-14: Visualization of Small Scale Rock Cutting Test

7.5 Outlook

Finally it has to be mentioned that with very high computing effort – over 14 days with parallelization on 10 CPUs for one single cut of 50 mm - it is entirely possible to simulate the Small Scale Rock Cutting Test with Abaqus and the Concrete Damaged Plasticity model. The parameters in the post peak section and the deletion criterion should be regarded as numerical parameters with big influence on the result. The adjustment of these values takes place via trial-and-error and is very time-consuming. Calculating backwards from the results of the laboratory tests via parameter study, performed automatically, is currently not feasible caused by the computing time and the post failure behaviour is not a single value, it's a curve.

The next point is to depict the development of cracks and chipping. The crack pattern is associated with the post failure behaviour and the deletion criterion. A quick softening after cracking and low residual strength or a low deletion criterion lead to a relatively unaffected sample (Illustration 7-14 and Illustration 7-15). The question is whether the possibility of crack illustration with a constitutive law with smeared cracking is given.

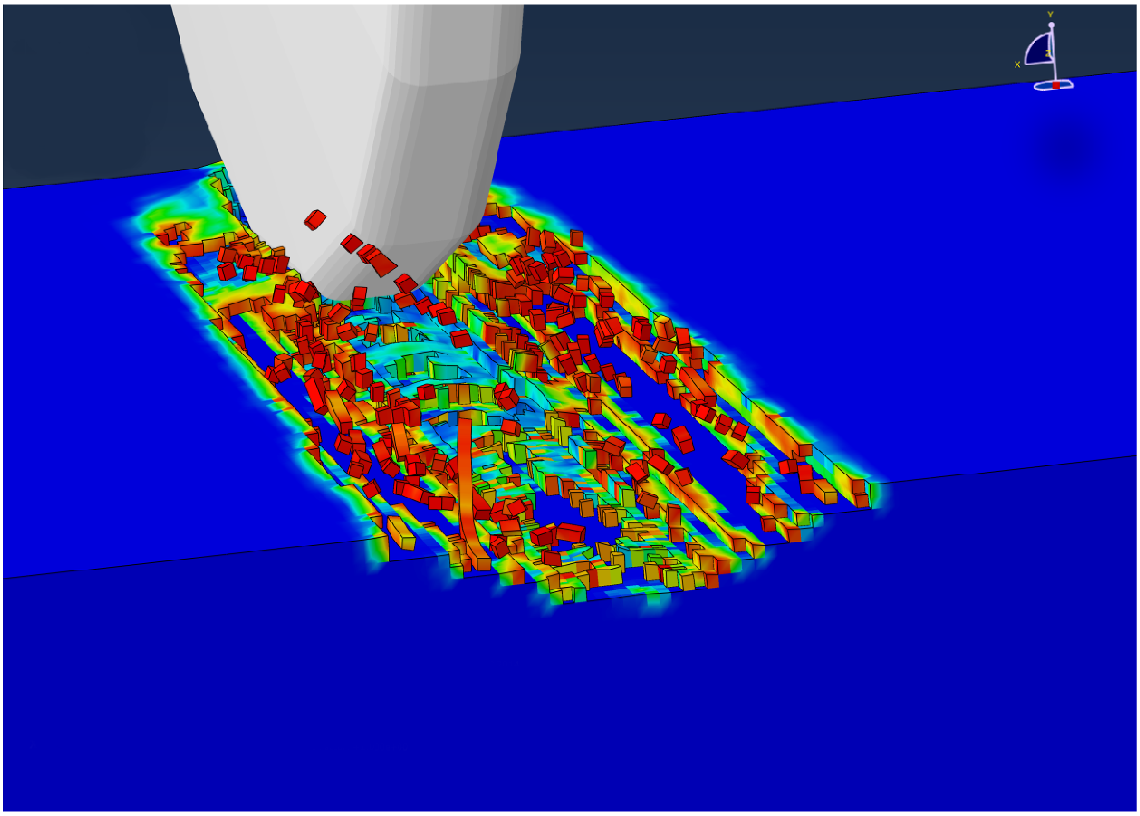


Illustration 7-15: Relatively unaffected sample caused by low deletion criterion

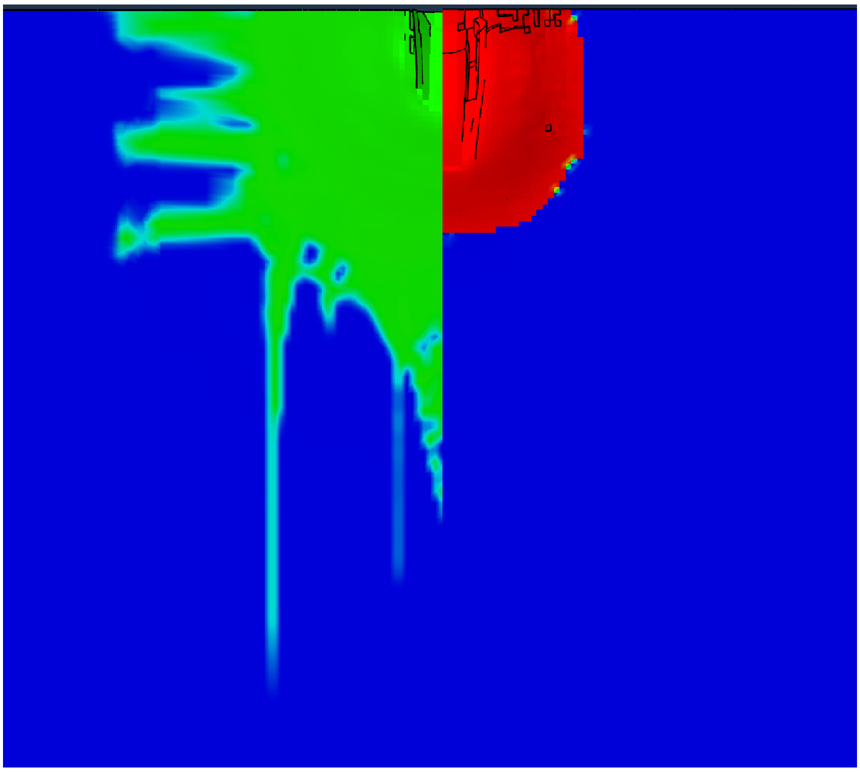


Illustration 7-16: Comparison of impairment (DamageC) by late (green) and early (red) deletion

It's impossible to make a statement of the velocity influence on the result, because a lower velocity raises the computing effort and with the present available computer hardware this computing effort isn’t feasible. Also a finer mesh for the whole sample or a finer mesh around the cutting trace (region 1 in Illustration 5-6) leads to impractical calculation time.

In case one of the next versions of Abaqus has the possibility to parallelize the Smoothed Particle Hydrodynamics method, the idea to simulate the powdered material with conversion into particles by an user subroutine shouldn’t be given up.

One big difficulty in the future will be to regard the inhomogeneity and the anisotropy of rock caused by mineralogy, stratification, cleavage, joints and so on.

Also the simulation of the two cutting traces with different spacing and repeated cut in the same cutting trace should be tested.

8 List of tables

9 List of illustrations

10 Bibliography

1. **Lorenz, Stefan.** *Entwicklung eines Modellversuchs zur Schneidbarkeitsermittlung von Hartgestein.* Leoben : s.n., 2013.

2. **Entacher, Martin.** *Measurement and interpretation of disc cutting forces in mechanized tunneling.* Leoben : s.n., 2013.

3. *Leistungs- und Verschleißprognosen im maschinellen Tunnelbau.* **Gehring, K.** Felsbau Nr.13, 1995 : s.n.

4. **Abaqus 6.12 Online Documentation (HTML); Dassault Systèmes.** [Online] 2012.

5. **Wittel, Falk K.** *Eine kurze Einführung in die Finite Elemente Methode.* Zürich : s.n., 2010.

6. **Bui, Timothy.** *Explicit and Implicit Methods In Solving Differnetial Equations.* Conneticut : s.n., 2009.

7. http://en.wikipedia.org/wiki/File:Abaqus_software_FEA_process.png. [Online] [Cited: 27 May 2014.]

8. **Omidi, O. and Lotfi, V.** Finite Element Analysis of Concrete Structures Using Plastic-Damage Model in 3-D Implementation. *International Journal of Civil Engineering, Vol. 8, No. 3.* 3. September 2010.

9. **Gehwolf, Paul, Stoxreiter, Thomas and Klug, Nina.** *Prediction of damage evolution from micro to makro scale - New simulation tools and design concepts, WP5: Mechanical behaviour and damage of brittle disordered materials.* Leoben : (unveröffentlicht), 2014.

10. **Elsner.** *Einführung in die FORTRAN Programmierung.* Osnabrück : s.n., 1997.

11. **Pittino, Gerhard.** Leoben : (unveröffentlicht), 2014.

12. **Gong, Q. M., Jiao, Y. Y. and Zhao, J.** Numerical modelling of the effets of joint spacing on rock fragmentation by TBM cutters. *Tunneling and Underground Space Technology 21.* 2006.

13. **Cho, Jung-Woo, et al.** Optimum spacing of TBM disc cutters: A numerical simulation using the three-dimensional dynamic fracturing method. *Tunneling and Underground Space Technology 25.* 2010.

14. **Cho, Jung-Woo, et al.** Evaluation of cutting efficiency during TBM disc cutter excavation within a Korean granitic rock using linear-cutting-machine testing and photogrammetric measurement. *Tunnelling and Underground Space Technology 35.* 2013.

11 Appendix

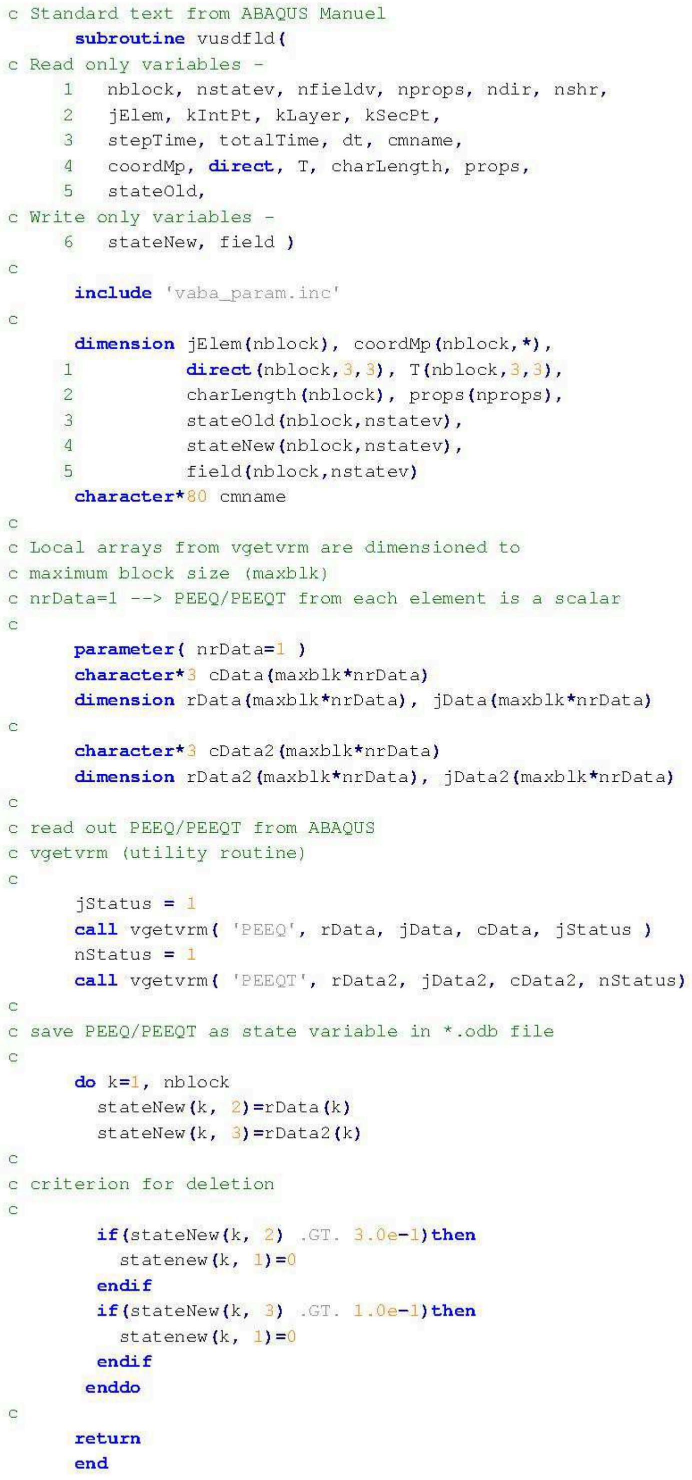
11.1 Notes to user subroutines and input files

In the following illustrations (Illustration 11-1 to Illustration 11-4) the italics red writing marks parameters and the italics green writing marks names that have to be written by the user in the input file. Comments that have not to be written in the input file are displayed blue.

As interface between the subroutine and Abaqus the solution-dependent state variables (SDV) act. The “stateNew” values of the subroutine (Illustration 6-1) are saved in each increment as SDV in the *.odb file. The keyword to define such variables in the input file is “Depvar” (Illustration 11-1). The variable “stateNew(k, 1)” in the subroutine is saved as SDV_1, called “Status”, “stateNew(k, 2)” as SDV_2 and so on. “delete=1” means, if the SDV 1 of an element k is set to zero, this element will be deleted. The deletion criterion can be specified in the subroutine. Replacing the “delete” by “convert” the element is converted into SPH particles instead of being deleted.

```
...
** MATERIALS
**
*Material, name=nameofmaterial
*Density
density,
*Depvar, delete=1 SDV that control the deletion
3 number of SDVs
1, Status, "Status"
2, PEEQ, "PEEQ"
3, PEEQT, "PEEQT"
*Elastic
 Young's modulus, Poisonratio
*User Defined Field
*Concrete Damaged Plasticity
...
```

Illustration 11-1: Material properties



```
c Standard text from ABAQUS Manuel
      subroutine vusdfld(
c Read only variables -
     1   nblock, nstatev, nfieldv, nprops, ndir, nshr,
     2   jElem, kIntPt, kLayer, kSecPt,
     3   stepTime, totalTime, dt, cmname,
     4   coordMp, direct, T, charLength, props,
     5   stateOld,
c Write only variables -
     6   stateNew, field )
c
      include 'vaba_param.inc'
c
      dimension jElem(nblock), coordMp(nblock,*),
     1          direct(nblock,3,3), T(nblock,3,3),
     2          charLength(nblock), props(nprops),
     3          stateOld(nblock,nstatev),
     4          stateNew(nblock,nstatev),
     5          field(nblock,nstatev)
      character*80 cmname
c
c Local arrays from vgetvrm are dimensioned to
c maximum block size (maxblk)
c nrData=1 --> PEEQ/PEEQT from each element is a scalar
c
      parameter( nrData=1 )
      character*3 cData(maxblk*nrData)
      dimension rData(maxblk*nrData), jData(maxblk*nrData)
c
      character*3 cData2(maxblk*nrData)
      dimension rData2(maxblk*nrData), jData2(maxblk*nrData)
c
c read out PEEQ/PEEQT from ABAQUS
c vgetvrm (utility routine)
c
      jStatus = 1
      call vgetvrm( 'PEEQ', rData, jData, cData, jStatus )
      nStatus = 1
      call vgetvrm( 'PEEQT', rData2, jData2, cData2, nStatus)
c
c save PEEQ/PEEQT as state variable in *.odb file
c
      do k=1, nblock
        stateNew(k, 2)=rData(k)
        stateNew(k, 3)=rData2(k)
c
c criterion for deletion
c
        if(stateNew(k, 2) .GT. 3.0e-1)then
          statenew(k, 1)=0
        endif
        if(stateNew(k, 3) .GT. 1.0e-1)then
          statenew(k, 1)=0
        endif
       enddo
c
      return
      end
```

Illustration 6-1: Subroutine

The software requirement to use Abaqus with subroutines, an installation guide and other subroutines are given in (9).

11.2 Infinite elements

The simplest way to model the infinite elements is to partition the sample a few millimetres before the end face, assign to this region acoustic elements and a sweep mesh. The direction of the sweep path is from the front face to the end face in axial direction. In sweep direction only one element row is allowed. Also each infinite element has to share only one face with one finite element. After writing the input file the acoustic elements can be converted into infinite elements by changing the AC3D8R elements to CIN3D8 elements.

```
...

*Element, type=~~AC3D8R~~ CIN3D8

 1,  74, 313, 616, 205,   1,  13, 145,  34

...
```

Illustration 11-2: infinite elements

11.3 Inner surface

An inner surface can be inserted by creating an element set in the mesh module. After creating the input file an inner surface can be introduced.

```
...

**

*Surface, type=ELEMENT, name=nameofinnersurface

,

nameofelementset, INTERIOR this paragraph has to be written in front of "end assemply"

end assembly

**

...
```

Illustration 11-3: Creation of inner surface

Also the contact assignment has to be changed (Illustration 11-4). "Nameofinnersurface" is the defined inner surface (slave) and the "mastersurface" is in this example the surface of the analytical rigid disc cutter (disccutter-1.disccuttersurface). "nameofinnersurface, " means a selfcontact of the sample.

```
...
** Interaction: nameofcontact
*Contact, op=NEW
*Contact Inclusions
nameofinnersurface, mastersurface nameofinnersurface = slave, mastersurface = master
nameofinnersurface, selfcontact of innersurface
*Contact Property Assignment
 , , propertiesofcontact
...
```

Illustration 11-4: Changing the contact parameters