

See discussions, stats, and author profiles for this publication at: <https://www.researchgate.net/publication/328956103>

# How to get meaningful and correct results from your finite element model

Preprint · November 2018

---

CITATIONS

0

---

READS

59,776

1 author:



[Martin Baeker](#)

Technische Universität Braunschweig

223 PUBLICATIONS 2,793 CITATIONS

SEE PROFILE

# How to get meaningful and correct results from your finite element model

Martin Bäker

Institut für Werkstoffe, Technische Universität Braunschweig,  
Langer Kamp 8, D-38106 Braunschweig, martin.baeker@tu-bs.de

November 15, 2018

## Abstract

This document gives guidelines to set up, run, and postprocess correct simulations with the finite element method. It is not an introduction to the method itself, but rather a list of things to check and possible mistakes to watch out for when doing a finite element simulation.

The finite element method (FEM) is probably the most-used simulation technique in engineering. Modern finite-element software makes doing FE simulations easy – too easy, perhaps. Since you have a nice graphical user interface that guides you through the process of creating, solving, and postprocessing a finite element model, it may seem as if there is no need to know much about the inner workings of a finite element program or the underlying theory. However, creating a model without understanding finite elements is similar to flying an airplane without a pilot’s license. You may even land somewhere without crashing, but probably not where you intended to.

This document is not a finite element introduction, see, for example, [3, 7, 10] for that. It is a guideline to give you some ideas how to correctly set up, solve and postprocess a finite element model. The techniques described here were developed working with the program Abaqus [9]; however, most of them should be easily transferable to other codes. I have not explained the theoretical basis for most of them; if you do not understand why a particular consideration is important, I recommend to study finite element theory to find out.

# 1 Setting up the model

## 1.1 General considerations

These considerations are not restricted to finite element models, but are useful for any complex simulation method.

- 1.1-1. Even if you just need some number for your design – the main goal of an FEA is to *understand* the system. Always design your simulations so that you can at least qualitatively understand the results. *Never* believe the result of a simulation without thinking about its plausibility.
- 1.1-2. Define the goal of the simulation as precisely as possible. Which question is to be answered? Which quantities are to be calculated? Which conclusions are you going to draw from the simulation? Probably the most common error made in FE simulations is setting up a simulation without having a clear goal in mind. Be as specific as possible. Never set up a model “to see what happens” or “to see how stresses are distributed”.
- 1.1-3. Formulate your expectations for the simulation result beforehand and make an educated guess of what the results should be. If possible, estimate at least some quantities of your simulation using simplified assumptions. This will make it easier to spot problems later on and to improve your understanding of the system you are studying.
- 1.1-4. Based on the answer to the previous items, consider which effects you actually have to simulate. Keep the model as simple as possible. For example, if you only need to know whether a yield stress is exceeded somewhere in a metallic component, it is much easier to perform an elastic calculation and check the von Mises stress in the postprocessor (be wary of extrapolations, see 3.2-1) than to include plasticity in the model.
- 1.1-5. What is the required precision of your calculation? Do you need an estimate or a precise number? (See also 1.4-1 below.)
- 1.1-6. If your model is complex, create it in several steps. Start with simple materials, assume frictionless behaviour etc. Add complications step by step. Setting up the model in steps has two advantages: (i) if errors occur, it is much easier to find out what caused them; (ii) understanding the behaviour of the system is easier this way because you understand which addition caused which change in the model behaviour. Note, however, that checks you made in an early stage (for example on the mesh density) may have to be repeated later.

- 1.1-7. Be careful with units. Many FEM programs (like ABAQUS) are inherently unit-free – they assume that all numbers you give can be converted without additional conversion factors. You cannot define your model geometry in millimeter, but use SI units without prefixes everywhere else. Be especially careful in thermomechanical simulations due to the large number of different physical quantities needed there. And of course, be also careful if you use antiquated units like inch, slug, or BTU.

## 1.2 Basic model definition

- 1.2-1. Choose the correct type of simulation (static, quasi-static, dynamic, coupled etc.). Dynamic simulations require the presence of inertial forces (elastic waves, changes in kinetic energies). If inertial forces are irrelevant, you should use static simulations.
- 1.2-2. As a rule of thumb, a simulation is static or quasi-static if the excitation frequency is less than  $1/5$  of the lowest natural frequency of the structure [2].
- 1.2-3. In a dynamic analysis, damping may be required to avoid unrealistic multiple reflections of elastic waves that may affect the results [2].
- 1.2-4. Explicit methods are inherently dynamic. In some cases, explicit methods may be used successfully for quasi-static problems to avoid convergence problems (see 2.1-9 below). If you use mass scaling in your explicit quasi-static analysis, carefully check that the scaling parameter does not affect your solution. Vary the scaling factor (the nominal density) to ensure that the kinetic energy in the model remains small [12].
- 1.2-5. In a static or quasi-static analysis, make sure that all parts of the model are constrained so that no rigid-body movement is possible. (In a contact problem, special stabilization techniques may be available to ensure correct behaviour before contact is established.)
- 1.2-6. If you are studying a coupled problem (for example thermo-mechanical) think about the correct form of coupling. If stresses and strains are affected by temperature but not the other way round, it may be more efficient to first calculate the thermal problem and then use the result to calculate thermal stresses. A full coupling of the thermal and mechanical problem is only needed if temperature affects stresses/strains (e.g., due to thermal expansion or temperature-dependent material problems) and if stresses and strains also affect the thermal problem (e.g., due to plastic heat generation or the change in shape affecting heat conduction).

- 1.2-7. Every FE program uses discrete time steps (except for a static, linear analysis, where no time incrementation is needed). This may affect the simulation. If, for example, the temperature changes during a time increment, the material behaviour may strongly differ between the beginning and the end of the increment (this often occurs in creep problems where the properties change drastically with temperature). Try different maximal time increments and make sure that time increments are sufficiently small so that these effects are small.
- 1.2-8. Critically check whether non-linear geometry is required. As a rule of thumb, this is almost always the case if strains exceed 5%. If loads are rotating with the structure (think of a fishing rod that is loaded in bending initially, but in tension after it has started to deform), the geometry is usually non-linear. If in doubt, critically compare a geometrically linear and non-linear simulation.

### **1.3 Symmetries, boundary conditions and loads**

- 1.3-1. Exploit symmetries of the model. In a plane 2D-model, think about whether plane stress, plane strain or generalized plane strain is the appropriate symmetry. (If thermal stresses are relevant, plane strain is almost always wrong because thermal expansion in the 3-direction is suppressed, causing large thermal stresses. Note that these 33-stresses may affect other stress components as well, for example, due to von Mises plasticity.) Keep in mind that the loads and the deformations must conform to the same symmetry.
- 1.3-2. Check boundary conditions and constraints. After calculating the model, take the time to ensure that nodes were constrained in the desired way in the postprocessor.
- 1.3-3. Point loads at single nodes may cause unrealistic stresses in the adjacent elements. Be especially careful if the material or the geometry is non-linear. If in doubt, distribute the load over several elements (using a local mesh refinement if necessary).
- 1.3-4. If loads are changing direction during the calculation, non-linear geometry is usually required, see 1.2-8.
- 1.3-5. The discrete time-stepping of the solution process may also be important in loading a structure. If, for example, you abruptly change the heat flux at a certain point in time, discrete time stepping may not capture the exact point at which the change occurs, see fig. 1. (Your software may use some averaging procedure to alleviate this.) Define load steps or use other methods to ensure that the time of the abrupt change actually corresponds

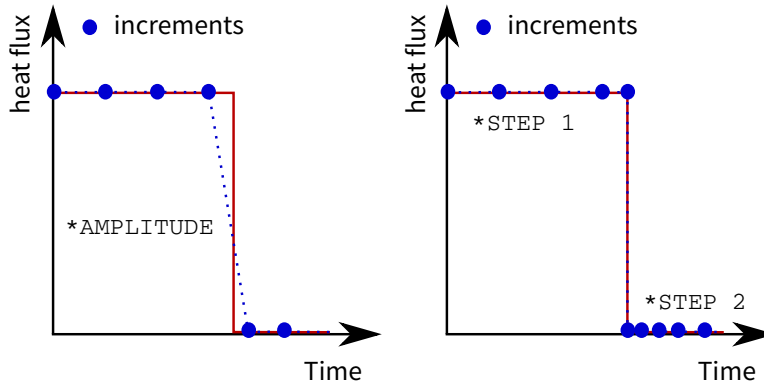


Figure 1: Discrete time steps may affect the load of a structure. If, for example, a heat flux abruptly changes, the change may actually only become relevant at a later moment (left). Use a new load step (or another method) to ensure that you capture the correct moment (right).

to a time step in the simulation. This may also improve convergence because it allows to control the increments at the moment of abrupt change, see also 2.1-4

## 1.4 Input data

- 1.4-1. A simulation cannot be more precise than its input data allow. This is especially true for the material behaviour. Critically consider how precise your material data really are. How large are the uncertainties? If in doubt, vary material parameters to see how results are affected by the uncertainties.
- 1.4-2. Be careful when combining material data from different sources and make sure that they are referring to identical materials. In metals, don't forget to check the influence of heat treatment; in ceramics, powder size or the processing route may affect the properties; in polymers, the chain length or the content of plasticizers is important [13]. Carefully document your sources for material data and check for inconsistencies.
- 1.4-3. Be careful when extrapolating material data. If data have been described using simple relations (for example a Ramberg-Osgood law for plasticity), the real behaviour may strongly deviate from this.
- 1.4-4. Keep in mind that your finite element software usually cannot extrapolate material data beyond the values given. If plastic strains exceed the maximum value specified, usually no further hardening of the material will be considered. The same holds, for example, for thermal expansion coefficients which usually increase with temperature. Using different ranges in differ-

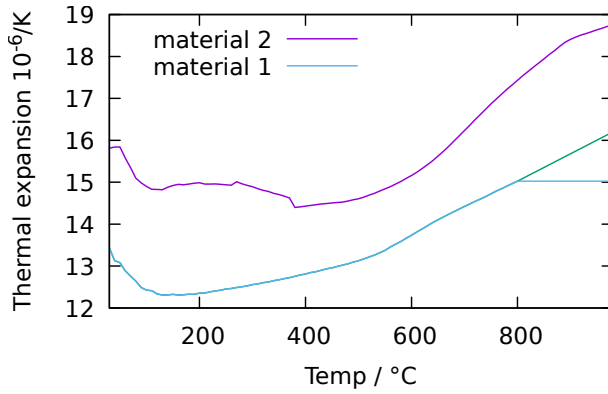


Figure 2: Coefficient of thermal expansion for two materials. Experimental data for material 1 end at a temperature of 800 °C, data for material 2 extend to a higher temperature. Most finite element programs assume a constant value beyond the final data point (blue line). If data are not extrapolated, the mismatch of the expansion coefficient between the materials is overestimated, causing large thermal stresses.

ent materials may thus cause spurious thermal stresses. Fig. 2 shows an example

- 1.4-5. If material data are given as equations, be aware that parameters may not be unique. Frequently, data can be fitted using different parameters. As an illustration, plot the simple hardening law  $A+B\epsilon^n$  with values (130, 100, 0.5) and (100, 130, 0.3) for  $(A, B, n)$ , see fig. 3. Your simulation results may be indifferent to some changes in the parameters because of this.
- 1.4-6. If it is not possible to determine material behaviour precisely, finite element simulations may still help to understand how the material behaviour affects the system. Vary parameters in plausible regions and study the answer of

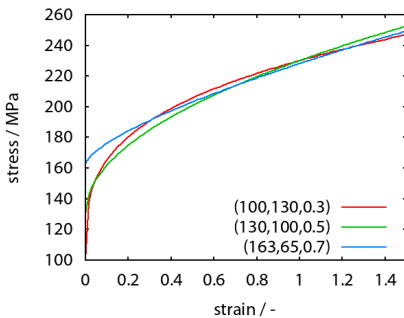


Figure 3: Different values of the parameters  $A, B$ , and  $n$  in a flow stress law  $\sigma = A + B\epsilon^n$  result in very similar curves.

the system.

- 1.4-7. Also check the precision of external loads. If loads are not known precisely, use a conservative estimate.
- 1.4-8. Thermal loads may be especially problematic because heat transfer coefficients or surface temperatures may be difficult to measure. Use the same considerations as for materials.
- 1.4-9. If you vary parameters (for example the geometry of your component or the material), make sure that you correctly consider how external loads are changed by this. If, for example, you specify an external load as a pressure, increasing the surface also increases the load. If you change the thermal conductivity of your material, the total heat flux through the structure will change; you may have to specify the thermal load accordingly.
- 1.4-10. Frictional behaviour and friction coefficients are also frequently unknown. Critically check the parameters you use and also check whether the friction law you are using is correct – not all friction is Coulombian.
- 1.4-11. If a small number of parameters are unknown, you can try to vary them until your simulation matches experimental data, possibly using a numerical optimization method. (This is the so-called inverse parameter identification [6].) Be aware that the experimental data used this way cannot be used to validate your model (see section 3.3).

## 1.5 Choice of the element type

*Warning:* Choosing the element type is often the crucial step in creating a finite element model. Never accept the default choice of your program without thinking about it.<sup>1</sup> Carefully check which types are available and make sure you understand how a finite element simulation is affected by the choice of element type. You should understand the concepts of element order and integration points (also known as Gauß points) and know the most common errors caused by an incorrectly chosen element type (shear locking, volumetric locking, hourglassing [1,3]).

The following points give some guidelines for the correct choice :

- 1.5-1. If your problem is linear-elastic, use second-order elements. Reduced integration may save computing time without strongly affecting the results.
- 1.5-2. Do not use fully-integrated first order elements if bending occurs in your structure (shear locking). Incompatible mode elements may circumvent this problem, but their performance strongly depends on the element shape [7].

---

<sup>1</sup>The only acceptable exception may be a simple linear-elastic simulation if your program uses second-order elements. But if all you do is linear elasticity, this article is probably not for you.



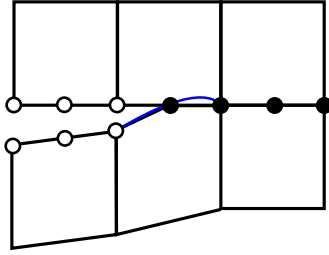


Figure 4: Contact between second-order elements. If an edge and one corner node of an element are in contact with a surface, the quadratic interpolation of the edge shape causes a penetration of the elements. Adapted from [4].

- 1.5-3. If you use first-order elements with reduced integration, check for hourglassing. Keep in mind that hourglassing may occur only in the interior of a three-dimensional structure where seeing it is not easy. Exaggerating the displacements may help in visualizing hourglassing. Most programs use numerical techniques to suppress hourglass modes; however, these may also affect results due to artificial damping. Therefore, also check the energy dissipated by this artificial damping and make sure that it is small compared to other energies in the model.
- 1.5-4. In contact problems, first-order elements may improve convergence because if one corner and one edge node are in contact, the second-order interpolation of the element edge causes overlaps, see fig. 4. This may especially cause problems in a crack-propagation simulation with a node-release scheme [4, 11].
- 1.5-5. Discontinuities in stresses or strains may be captured better with first-order elements in some circumstances.
- 1.5-6. If elements distort strongly, first-order elements may be better than second-order elements.
- 1.5-7. Avoid triangular or tetrahedral first-order elements since they are much too stiff, especially in bending. If you have to use these elements (which may be necessary in a large model with complex geometry), use a very fine mesh and carefully check for mesh convergence. Think about whether partitioning your model and meshing with quadrilateral/hexahedral elements (at least in critical regions) may be worth the effort. Fig. 5 shows an example where a very complex geometry has to be meshed with tetrahedral elements. Although the mesh looks reasonably fine, the system answer with linear elements is much too stiff.
- 1.5-8. If material behaviour is incompressible or almost incompressible, use hybrid elements to avoid volumetric locking. They may also be useful if plastic

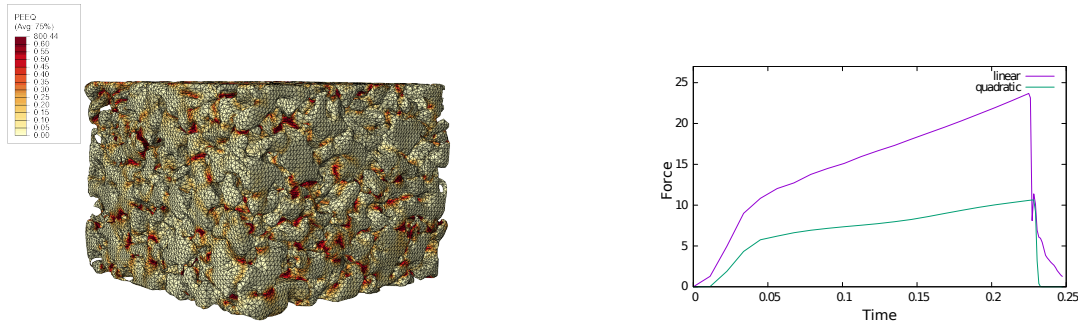


Figure 5: Simulation of the compression of a metallic foam with a tetrahedral mesh. As the force-time curve shows, the result strongly differs between linear and quadratic elements although the mesh looks rather fine and comprises more than 700000 elements. Note that the simulation is displacement-controlled so evaluating forces reveals whether the model is too stiff, see 1.6-5.

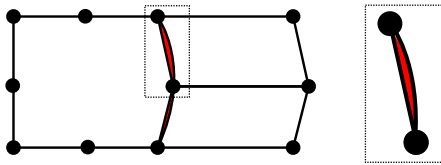


Figure 6: Mixing of elements with different order (left: one second-order element, right: two first-order elements) is not allowed as it would lead to unphysical deformation (gaps or overlaps): The quadratic element uses a second-order function to calculate the elements edge, the two linear elements assume a piecewise linear shape.

deformation is large because (metal) plasticity is also volume conserving.

- 1.5-9. Do not mix elements with different order. This can cause overlaps or gaps forming at the interface (possibly not shown by your postprocessor) even if there are no hanging nodes (see fig. 6). If you have to use different order of elements in different regions of your model, tie the interface between the regions using a surface constraint. Be aware that this interface may cause a discontinuity in the stresses and strains due to different stiffness of the element types.
- 1.5-10. In principle, it is permissible to mix reduced and fully integrated elements of the same order. However, since they differ in stiffness, spurious stress or strain discontinuities may result.
- 1.5-11. If you use shell or beam elements or similar, make sure to use the correct formulation. Shells and membranes look similar, but behave differently. Make sure that you use the correct mathematical formulation; there are

a large number of different types of shell or beam elements with different behaviour.

## 1.6 Generating a mesh

- 1.6-1. If possible, use quadrilateral/hexahedral elements. Meshing 3D-structures this way may be laborious, but it is often worth the effort (see also 1.5-7).
- 1.6-2. A fine mesh is needed where gradients in stress and strain are large.
- 1.6-3. A preliminary simulation with a coarse mesh may help to identify the regions where a greater mesh density is required.
- 1.6-4. Keep in mind that the required mesh density depends on the quantities you want to extract and on the required precision. For example, displacements are often calculated more precisely than strains (or stresses) because strains involve derivatives, i.e. the differences in displacements between nodes.
- 1.6-5. A mesh convergence study can be used to check whether the model behaves too stiff (as is often the case for fully integrated first-order elements, see fig. 5) or too soft (which happens with reduced-integration elements). Be careful in evaluating this study: If your model is load-controlled, evaluate displacements or strains to check for convergence, if it is strain-controlled, evaluate forces or stresses. (Stiffness relates forces to displacements, so to check for stiffness you need to check both.) If you use, for example, displacement control, displacements are not sensitive to the actual stiffness of your model since you prescribe the displacement.
- 1.6-6. Check shape and size of the elements. Inner angles should not deviate too much from those of a regularly shaped element. Use the tools provided by your software to highlight critical elements. Keep in mind that critical regions may be situated inside a 3D-component and may not be directly visible. Avoid badly-shaped elements especially in region where high gradients occur and in regions of interest.
- 1.6-7. If you use local mesh refinement, the transition between regions of different element sizes should be smooth. As a rule of thumb, adjacent elements should not differ by more than a factor of 2–3 in their area (or volume). If the transition is too abrupt, spurious stresses may occur in this region because a region that is meshed finer is usually less stiff. Furthermore, the fine mesh may be constrained by the coarser mesh. (As an extreme case, consider a finely meshed quadratic region that is bounded by only four first-order elements – in this case, the region as a whole can only deform as a parallelogram, no matter how fine the interior mesh is.)

- 1.6-8. Be aware that local mesh refinement may strongly affect the simulation time in an explicit simulation because the stable time increment is determined by the size of the smallest element in the structure. A single small or badly shaped element can drastically increase the simulation time.
- 1.6-9. If elements are distorting strongly, remeshing may improve the shape of the elements and the solution quality. For this, solution variables have to be interpolated from the old to the new mesh. This interpolation may dampen strong gradients or local extrema. Make sure that this effect is sufficiently small by comparing the solution before and after the remeshing in a contour plot and at the integration points.
- 1.6-10. Another way of dealing with strong mesh distortions is to start with a mesh that is initially distorted and becomes more regular during deformation. This method usually requires some experimentation, but it may yield good solutions without the additional effort of remeshing.

## 1.7 Defining contact problems

- 1.7-1. Correctly choose master and slave surfaces in a master-slave algorithm. In general, the stiffer (and more coarsely meshed) surface should be the master.
- 1.7-2. Problems may occur if single nodes get in contact and if surfaces with corners are sliding against each other. Smoothing the surfaces may be helpful.
- 1.7-3. Nodes of the master surface may penetrate the slave surface; again, smoothing the surfaces may reduce this, see fig. 7.
- 1.7-4. Some discretization error is usually unavoidable if curved surfaces are in contact. With a pure master-slave algorithm, penetration and material overlap are the most common problem; with a symmetric choice (both surfaces are used as master and as slave), gaps may open between the surfaces, see fig. 8. Check for discretization errors in the postprocessor.
- 1.7-5. Discretization errors may also affect the contact force. Consider, for example, the Hertzian contact problem of two cylinders contacting each other. If the mesh is coarse, there will be a notable change in the contact force whenever the next node comes into contact. Spurious oscillations of the force may be caused by this.
- 1.7-6. Make sure that rigid-body motion of contact partners before the contact is established is removed either by adding appropriate constraints or by using a stabilization procedure.

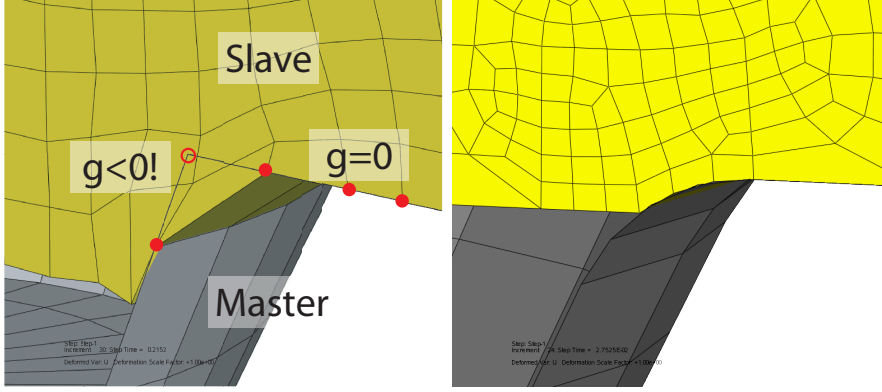


Figure 7: Left: A node on a sharp corner on the master surface may penetrate the slave surface (distance of node from surface  $g < 0$ ). Right: Smoothing the corner and refining the slave mesh reduces (but does not completely eliminate) the penetration.

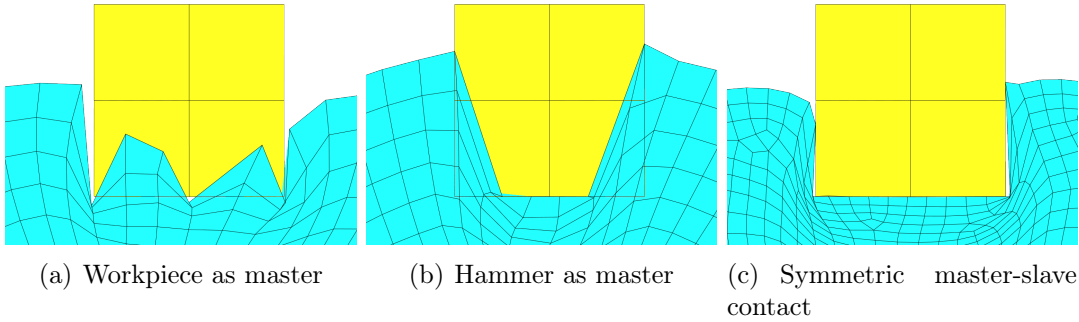


Figure 8: Master-slave algorithm for the example of a hammer hitting a workpiece. Using a master-slave algorithm results in penetrations of master nodes into the slave surface; a symmetric choice avoids this, but causes gaps to form.

- 1.7-7. Second-order elements may cause problems in contact (see 1.5-4 and fig. 4) [4, 11]; if they do, try switching to first-order elements.

## 1.8 Other considerations

- 1.8-1. If you are inexperienced in using finite elements, start with simple models. Do not try to directly set up a complex model from scratch and make sure that you understand what your program does and what different options are good for. It is almost impossible to find errors in a large and complex model if you do not have long experience and if you do not know what results you expect beforehand.
- 1.8-2. Many parameters that are not specified by the user are set to default values in finite element programs. You should check whether these defaults are correct; especially for those parameters that directly affect the solution (like element types, material definitions etc.). If you do not know what a parameter does and whether the default is appropriate, consult the manual. For parameters that only affect the efficiency of the solution (for example, which solution scheme is used to solve matrix equations), understanding the parameters is less important because a wrongly chosen parameter will not affect the final solution, but only the CPU time or whether a solution is found at all.
- 1.8-3. Modern finite element software is equipped with a plethora of complex special techniques (XFEM, element deletion, node separation, adaptive error-controlled mesh-refinement, mixed Eulerian-Lagrangian methods, particle based methods, fluid-structure interaction, multi-physics, user-defined sub-routines etc.). If you plan to use these techniques, make sure that you understand them and test them using simple models. If possible, build up a basic model without these features first and then add the complex behaviour. Keep in mind that the impressive simulations you see in presentations were created by experts and may have been carefully selected and may not be typical for the performance.

## 2 Solving the model

Even if your model is solved without any convergence problems, nevertheless look at the log file written by the solver to check for warning messages. They may be harmless, but they may indicate some problem in defining your model.

Convergence problems are usually reported by the program with warning or error messages. You can also see that your model has not converged if the final time in the time step is not the end time you specified in the model definition.

There are two reasons for convergence problems: On the one hand, the solution algorithm may fail to find a solution albeit a solution of the problem does exist. In this case, modifying the solution algorithm may solve the problem (see section 2.2). On the other hand, the problem definition may be faulty so that the problem is unstable and does not have a solution (section 2.3).

If you are new to finite element simulations, you may be tempted to think that these errors are simply caused by specifying an incorrect option or forgetting something in the model definition. Errors of this type exist as well, but they are usually detected before calculation of your model begins (and are not discussed here). Instead, treat the non-convergence of your simulation in the same way as any other scientific problem. Formulate hypotheses why the simulation fails to converge. Modify your model to prove<sup>2</sup> or disprove these hypotheses to find the cause of the problems.

## 2.1 General considerations

- 2.1-1. In an implicit simulation, the size of the time increments is usually automatically controlled by the program. If convergence is difficult, the time increments are reduced.<sup>3</sup> Usually, the program stops if the time increment is too small or if the convergence problems persist even after several cut-backs of the time increment. (In Abaqus, you get the error messages **Time increment smaller than minimum** or **Too many attempts**, respectively.) These messages themselves thus do not tell you anything about the reason for the convergence problems.

To find the cause of the convergence problems, look at the solver log file in the increment(s) before the final error message. You will probably see warnings that tell you what kind of convergence problem was responsible (for example, the residual force is too large, the contact algorithm did not converge, the temperature increments were too large). If available, also look at the unconverged solution and compare it to the last, converged timestep. Frequently, large changes in some quantity may indicate the location of the problem.

- 2.1-2. Use the postprocessor to identify the node with the largest residual force and the largest change in displacement in the final increment. Often (but not always) this tells you where the problem in the model occurs. (Apply the same logic in a thermal simulation looking at the temperature changes and heat fluxes.)

---

<sup>2</sup>Of course natural science is not dealing with “proofs”, but this is not the place to think about the philosophy of science. Replace “prove” with “strengthen” or “find evidence for” if you like.

<sup>3</sup>The rationale behind this is that the solution from the previous increment is a better initial guess for the next increment if the change in the load is reduced.

- 2.1-3. If the first increment does not converge, set the size of the first time increment to a very small value. If the problem persists, the model itself may be unstable (missing boundary conditions, initial overlap of contacting surfaces). To find the cause of the problem, you can remove all external loads step by step or add further boundary conditions to make sure that the model is properly constrained (if you pin two nodes for each component, rigid body movements should be suppressed – if the model converges in this case, you probably did not have sufficient boundary conditions in your original model). Alternatively or additionally, you may add numerical stabilization to the problem definition. (In numerical stabilization, artificial friction is added to the movement of nodes so that stabilizing forces are generated if nodes start to move rapidly.) However, make sure that the stabilization does not affect your results too strongly.

Also check for abrupt jumps in some boundary conditions, for example a finite displacement that is defined at the beginning of a step or a sudden jump in temperature or load. If you apply a load instantaneously, cutting back the time increments does not help the solution process. If this occurs, ramp your load instead.

- 2.1-4. Avoid rapid changes in an amplitude within a calculation step (see also 1.2-7 and 1.3-5). For example, if you hold a heat flux (or temperature or stress) for a long time and then abruptly reduce it within the same calculation step, the time increment will suddenly jump to a point where the temperature is strongly reduced. This abrupt change may cause convergence problems. Define a second step and choose small increments at the beginning of the second step where large changes in the model can be expected.
- 2.1-5. Try the methods described in section 2.2 to see whether the problem can be resolved by changing the solution algorithm.
- 2.1-6. Sometimes, it is the calculation of the material law at an integration point that does not converge (to calculate stresses from strains at integration point inside the solver, another Newton algorithm is used at each integration point [3]). If this is the case, the material definition may be incorrect or problematic (for example, due to incorrectly specified material parameters or because there is extreme softening at a point).
- 2.1-7. Simplify your model step by step to find the reason of the convergence problems. Use simpler material laws (simple plasticity instead of damage, elasticity instead of plasticity), switch off non-linear geometry, remove external loads etc. If the problem persists, try to create a minimum example – the smallest example you can find that shows the same problem. This has several advantages: the minimum example is easier to analyse, needs less



computing time so that trying things is faster, and it can also be shown to others if you are looking for help (see section 4).

- 2.1-8. If your simulation is static, switching to an implicit dynamic simulation may help because the inertial forces act as natural stabilizers. If possible, use a quasi-static option.
- 2.1-9. Explicit simulations usually have less convergence problems. A frequently-heard advice to solve convergence problems is to switch from implicit to explicit models. I strongly recommend to only switch from implicit static to explicit quasi-static for convergence reasons if you understand the reasons of the convergence problems and cannot overcome them with the techniques described here. You should also keep in mind that explicit programs may offer a different functionality (for example, different element types). If your problem is static, you can only use a quasi-static explicit analysis which may also have problems (see 1.2-4). Be aware that in an explicit simulations, elastic waves may occur that may change the stress patterns.

## 2.2 Modifying the solution algorithm

If your solution algorithm does not converge for numerical reasons, these modifications may help. They are useless if there is a true model instability, see section 2.3.

- 2.2-1. Finite element programs use default values to control the Newton iterations. If no convergence is reached after a fixed number of iterations, the time step is cut back. In strongly non-linear problems, these default values may be too tight. For example, Abaqus cuts back on the time increment if the Newton algorithm does not converge after 4 iterations; setting this number to a larger value is often sufficient to reach convergence (for example, by adding `*Controls, analysis=discontinuous` to the input file).
- 2.2-2. If the Newton algorithm does not converge, the time increment is cut back. If it becomes smaller than a pre-defined minimum value, the simulation stopswith an error message. This minimum size of the time increment can be adjusted. Furthermore, if a sudden loss in stability (or change in load) occurs so that time increments need to be changed by several orders of magnitude, the number of cutbacks also needs to be adapted (see next point). In this case, another option is to define a new time step (see 2.1-4) that starts at this critical point and that has a small initial increment.
- 2.2-3. The allowed number of cutbacks increment can also be adapted (in Abaqus, use `*CONTROLS, parameters=time incrementation`). This may be helpful if the simulation proceeds at first with large increments before some

difficulty is reached – allowing for a larger number of cutbacks enables the program to use large timesteps at the beginning. Alternatively, you can reduce the maximum time increment (so that the size of the necessary cutback is reduced) or you can split your simulation step in two with different time incrementation settings in the step where the problem occurs (see 2.1-4).

- 2.2-4. Be aware that the previous two points will work sometimes, but not always. There is usually no sense in allowing a smallest time increment that is ten or twenty orders of magnitude smaller than the step size or to allow for dozens of cutbacks, this only increases the CPU time.
- 2.2-5. Depending on your finite element software, there may be many more options to tune the solution process. In Abaqus, for example, the initial guess for the solution of a time increment is calculated by extrapolation from the previous steps. Usually this improves convergence, but it may cause problems if something in the model changes abruptly. In this case, you can switch the extrapolation off (`*STEP, extrapolation=no`). You can also add a line search algorithm that scales the calculated displacements to find a better solution (`*CONTROLS, parameters=line search`). Consult the manual for options to improve convergence.
- 2.2-6. While changing the iteration control (as explained in the previous points) is often needed to achieve convergence, the solution controls that are used to determine whether a solution has converged should only be changed if absolutely necessary. Only do so (in Abaqus, use `*CONTROLS, parameters=field`) if you know exactly what you are doing. One example where changing the controls may be necessary is when the stress is strongly concentrated in a small part of a very large structure [5]. In this case, an average nodal force that is used to determine convergence may impose too strong a constraint on the convergence of the solution, so that convergence should be based on local forces in the region of stress concentration. Be aware that since forces, not stresses, are used in determining the convergence, changing the mesh density requires changing the solution controls.

Make sure that the accepted solution is indeed a solution and that your controls are sufficiently strict. Vary the controls to ensure that their value does not affect the solution.
- 2.2-7. Contact problems sometimes do not converge due to problems in establishing which nodes are in contact (sometimes called “zig-zagging” [14]). This often happens if the first contact is made by a single node. Smoothing the contact surfaces may help.
- 2.2-8. If available and possible, use general contact definitions where the contact surfaces are determined automatically.

- 2.2-9. If standard contact algorithms do not converge, soft contact formulations (which implement a soft transition between “no contact” and “full contact”) may improve convergence; however, they may allow for some penetration of the surfaces and thus affect the results.

## 2.3 Finding model instabilities

A model is unstable if there actually is no solution to the mechanical problem.

- 2.3-1. Instabilities are frequently due to a loss in load bearing capacity of the structure. There are several reasons for that:
- The material definition may be incorrect. If, for example, a plastic material is defined without hardening, the load cannot increase after the component has fully plastified. Simple typos or incorrectly used units may also cause a loss in material strength.
  - Thermal softening (the reduction of strength with increasing temperature) may cause an instability in a thermo-mechanical problem.
  - Non-linear geometry may cause an instability because the cross section of a load-bearing component reduces during deformation.
  - A change in contact area, a change from sticking to sliding in a simulation with friction or a complete loss of contact between two bodies may also cause instabilities because the structure may not be able to bear an increase in the load.
- 2.3-2. Local instabilities may cause highly distorted meshes that prevent convergence. It may be helpful to define the mesh in such a way that elements become more regular during deformation (see also 1.6-10).
- 2.3-3. If your model is load-controlled (a force is applied), switch to a displacement-controlled loading. This avoids instabilities due to loss in load-bearing capacity.
- 2.3-4. Artificial damping (stabilization) may be added to stabilize an unstable model. However, check carefully that the solution is not unduly affected by this. Adding artificial damping may also help to determine the cause of the instability. If your model converges with damping, you know that an instability is present.

## 2.4 Problems in explicit simulations

As already stated in 2.1-9, explicit simulations have less convergence problems than implicit simulations. However, sometimes even an explicit simulation may run into trouble.

- 2.4-1. During simulation, elements may distort excessively. This may happen for example if a concentrated load acts on a node or if the displacement of a node becomes very large due to a loss in stability (for example in a damage model). In this case, the element shape might become invalid (crossing over of element edges, negative volumes at integration points etc.). If this happens, changing the mesh might help – elements that have a low quality (large aspect ratio, small initial volume) are especially prone to this type of problem. Note that second-order elements are often more sensitive to this problem than first-order elements.
- 2.4-2. The stable time increment in an explicit simulation is given by the time a sound wave needs to travel through the smallest element. If elements distort strongly, they may become very thin in one direction so that the stable time increment becomes unreasonably small. In this case, changing the mesh might help.

### 3 Postprocessing

There are two aspects to checking that a model is correct: Verification is the process of showing that the model was correctly specified and actually does what it was created to do (loads, boundary conditions, material behaviour etc. are correct). Validation means to check the model by making an independent prediction (i. e., a prediction that was not used in specifying or calibrating the model) and checking this prediction in some other way (for example, experimentally).<sup>4</sup>

**General advice** If you modify your model significantly (because you build up a complicated model in steps, have to correct errors or add more complex material behaviour to get agreement with experimental results etc.), you should again check the model. It is not clear that the mesh density that was sufficient for your initial model is still sufficient for the modified model. The same is true for other considerations (like the choice of element type etc.)

#### 3.1 Checking the plausibility and verifying the model

- 3.1-1. Check the plausibility of your results. If your simulation deviates from your intuition, continue checking until you are sure that you understand why your intuition (or the simulation) was incorrect. *Never* believe a result of a simulation that you do not understand and that should be different according to your intuition. Either the model or your understanding of the physical problem is incorrect – in both cases, it is important to understand all effects.

---

<sup>4</sup>Note that the terms “verification” and “validation” are used differently in different fields.

- 3.1-2. Check your explanations for the solution, possibly with additional simulations. For example, if you assume that thermal expansion is the cause of a local stress maximum, re-run the simulation with a different or vanishing coefficient of thermal expansion. Predict the results of such a simulation and check whether your prediction was correct.
- 3.1-3. Check all important solution variables. Even if you are only interested in, for example, displacements of a certain point, check stresses and strains throughout the model.
- 3.1-4. In 3D-simulations, do not only look at contour plots of the component's surface; also check the results inside the component by cutting through it.
- 3.1-5. Make sure you understand which properties are vectors or tensors. Which component of stresses or strains are relevant depends on your model, the material, and the question you are trying to answer. Default settings of the postprocessor are not always appropriate, for example, Abaqus plots the von-Mises-stress as default stress variable, which is not very helpful for ceramic materials.
- 3.1-6. Check the boundary conditions again. Are all nodes constrained in the desired manner? Exaggerating the deformation (use `Common plot options` in Abaqus) or picking nodes with the mouse may be helpful to check this precisely.
- 3.1-7. Check the mesh density (see 1.6-5). If possible, calculate the model with different mesh densities (possibly for a simplified problem) and make sure that the mesh you finally use is sufficiently fine. When comparing different meshes, the variation in the mesh density should be sufficiently large to make sure that you can actually see an effect.
- 3.1-8. Check the mesh quality again, paying special attention on regions where gradients are large. Check that the conditions explained in section 1.6 (element shapes and sizes, no strong discontinuities in the element sizes) are fulfilled and that discontinuities in the stresses are not due to a change in the numerical stiffness (due to a change in the integration scheme or element size).
- 3.1-9. Check that stresses are continuous between elements. At interfaces between different materials, check that normal stresses and tangential strains are continuous.
- 3.1-10. Check that the normal stress at any free surface is zero.
- 3.1-11. Check the mesh density at contact surfaces: can the actual movement and deformation of the surfaces be represented by the mesh? For example, if a

mesh is too coarse, nodes may be captured in a corner or a surface may not be able to deform correctly.

- 3.1-12. Keep in mind that discretization errors at contact surfaces also influence stresses and strains. If you use non-standard contact definitions (2.2-9), try to evaluate how these influence the stresses (for example by comparing actual node positions with what you would expect for hard contact).
- 3.1-13. Watch out for divergencies. The stress at a sharp notch or cack tip is theoretically infinite – the value shown by your program is then solely determined by the mesh density and, if you use a contour plot, by the extrapolation used by the postprocessor (see 3.2-1).
- 3.1-14. In dynamic simulations, elastic waves propagate through the structure. They may dominate the stress field. Watch out for reflections of elastic waves and keep in mind that, in reality, these waves are dampened.
- 3.1-15. If you assumed linear geometry, check whether strains and deformations are sufficiently small to justify this assumption, see 1.2-8.

## 3.2 Implementation issues

- 3.2-1. Quantities like stresses or strains are only defined at integration points. Do not rely on extreme values from a contour plot – these values are extrapolated. It strongly depends on the problem whether these extrapolated values are accurate or not. For example, in an elastic material, the extrapolation is usually reasonable, in an ideally-plastic material, extrapolated von Mises stresses may exceed the actual yield stress by a factor of 2 or more. Furthermore, the contour lines themselves may show incorrect maxima or minima, see fig. 9 for an example.
- 3.2-2. It is often helpful to use “quilt” plots where each element is shown in a single color averaged from the integration point values (see also fig. 9).
- 3.2-3. The frequently used rainbow color spectrum has been shown to be misleading and should not be used [8]. Gradients may be difficult to interpret because human color vision has a different sensitivity in different parts of the spectrum. Furthermore, many people have a color vision deficiency and are unable to discern reds, greens and yellows. For variables that run from zero to a maximum value (temperature, von-Mises stress), use a sequential spectrum (for example, from black to red to yellow), for variables that can be positive and negative, use a diverging spectrum with a neutral color at zero, see fig. 10.

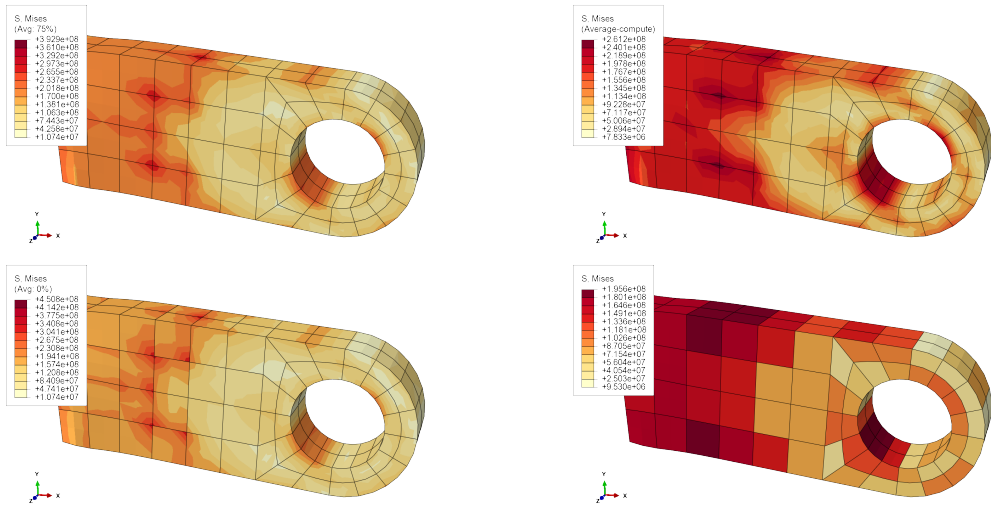
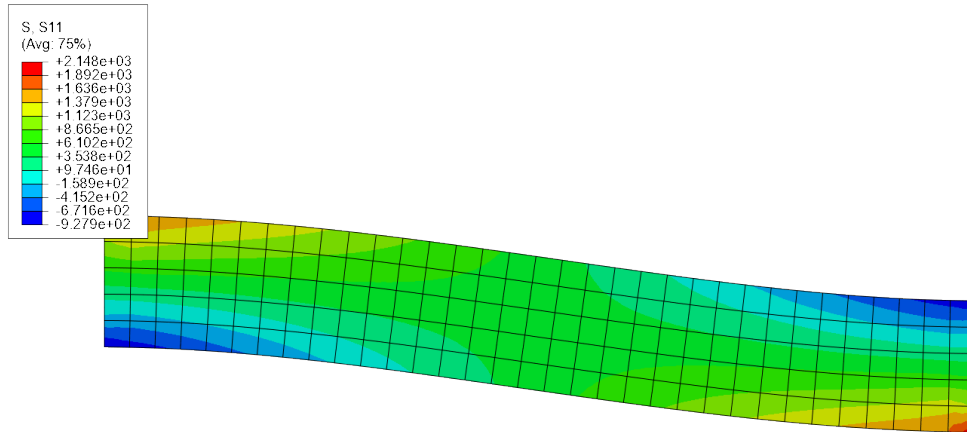
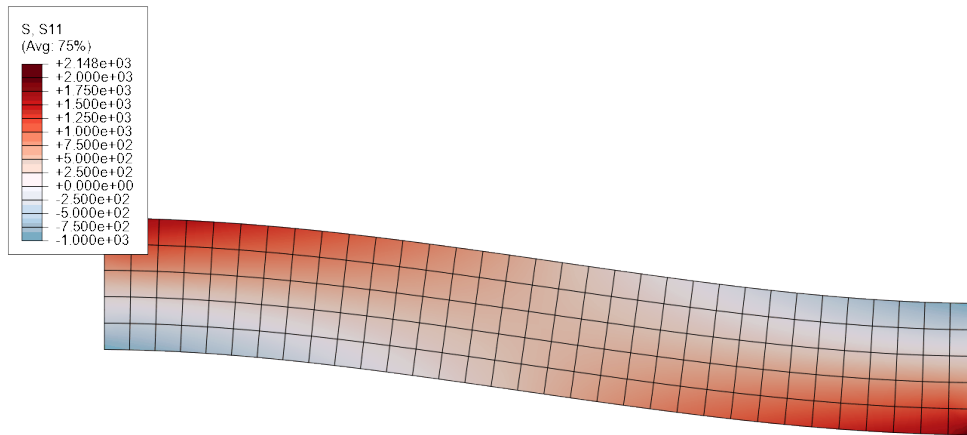


Figure 9: Von Mises stress in a simple lug constrained on the left and loaded in the hole on the right. The material is ideally plastic with a yield stress of 180 MPa so that the von Mises stress at the integration points never exceeds this value. All four figures show the same simulation result. Due to extrapolation from the integration points, the maximum value in the contour plot is much too large, except for the quilt plot on the lower right. The exact maximum value depends on how values are extrapolated and averaged at the surface and between elements.



Y ODB: biegebalken1.odb Abaqus/Standard 3DEXPERIENCE R2016x Mon Sep 12 10:33:20 CEST 2016  
 X Step: Step-1  
 Increment: 1: Step Time = 1.000  
 Primary Var.: S, S11  
 Deformed Var.: U Deformation Scale Factor: +3.250e+00



Y ODB: biegebalken1.odb Abaqus/Standard 3DEXPERIENCE R2016x Mon Sep 12 10:33:20 CEST 2016  
 X Step: Step-1  
 Increment: 1: Step Time = 1.000  
 Primary Var.: S, S11  
 Deformed Var.: U Deformation Scale Factor: +3.250e+00

Figure 10: Plot of the stress component  $\sigma_{11}$  in a bar under tension and bending. In a rainbow plot, it is difficult to see whether the gradient is homogeneous or not and to find the line of zero stress. A divergent spectrum with a neutral color at zero makes understanding the stress field easier. Furthermore, it is easier to interpret for people with color vision deficiency. The plot was done using the Abaqus plugin SpectrumBaker.



- 3.2-4. Discrete time-stepping (see 1.2-7) may also influence the post-processing of results. If you plot the stress-strain curve of a material point by connecting values measured at the discrete simulation times, the resulting curve will not coincide perfectly with the true stress-strain although the data points themselves are correct.
- 3.2-5. Complex simulation techniques (like XFEM, element deletion etc., see 1.8-3) frequently use internal parameters to control the simulation that may affect the solution process. Do not rely on default values for these parameters and check that the values do not affect the solution inappropriately.
- 3.2-6. If you use element deletion, be aware that removing elements from the simulation is basically an unphysical process since material is removed. This may affect the energy balance or stress fields near the removed elements. For example, in models of machining processes, removing elements at the tool tip to separate the material strongly influences the residual stress field.

### 3.3 Validation

- 3.3-1. If possible, use your model to make an independent prediction that can be tested.
- 3.3-2. If you used experimental data to adapt unknown parameters (see 1.4), correctly reproducing these data with the model does not validate it, but only verifies it.
- 3.3-3. The previous point also holds if you made a prediction and afterwards had to change your model to get agreement with an experiment. After this model change, the experiment cannot be considered an independent verification.

## 4 Getting help

If you cannot solve your problem, you can try to get help from the support of your software (provided you are entitled to support) or also from the internet (for example on researchgate or imechanica). To get helpful answers, please observe the following points:

- 4-1. Check that you have read relevant pages in the manual and that your question is not answered there.
- 4-2. Describe your problem as precisely as possible. Which error did occur? What was the exact error message and which warnings did occur? Show pictures of the model and describe the model (which element type, which material, what kind of problem – static, dynamic, explicit, implicit etc.).

- 4-3. If possible, provide a copy of your model or, even better, provide a minimum example that shows the problem (see 2.1-7).
- 4-4. If you get answers to your request, give feedback whether this has solved your problem, especially if you are in an internet forum or similar. People are sacrificing their time to help you and will be interested to see whether their advice was actually helpful and what the solution to the problem was. Providing feedback will also help others who find your post because they are facing similar problems.

## Acknowledgement

Thanks to Philipp Seiler for many discussions and for reading a draft version of this manuscript, and to Axel Reichert for sharing his experience on getting models to converge.

## References

- [1] F Armero. On the locking and stability of finite elements in finite deformation plane strain problems. *Computers & Structures*, 75(3):261–290, 2000.
- [2] CAE associates. Practical FEA simulations. [https://caeai.com/blog/practical-fea-simulations?utm\\_source=feedblitz&utm\\_medium=FeedBlitzRss&utm\\_campaign=caeai](https://caeai.com/blog/practical-fea-simulations?utm_source=feedblitz&utm_medium=FeedBlitzRss&utm_campaign=caeai). Accessed 31.5.2017.
- [3] Martin Bäker. *Numerische Methoden in der Materialwissenschaft*. Fachbereich Maschinenbau der TU Braunschweig, 2002.
- [4] Martin Bäker, Stefanie Reese, and Vadim V. Silberschmidt. Simulation of crack propagation under mixed-mode loading. In Siegfried Schmauder, Chuin-Shan Chen, Krishan K. Chawla, Nikhilesh Chawla, Weiqiu Chen, and Yutaka Kagawa, editors, *Handbook of Mechanics of Materials*. Springer Singapore, Singapore, 2018.
- [5] Martin Bäker, Joachim Rösler, and Carsten Siemers. A finite element model of high speed metal cutting with adiabatic shearing. *Computers & Structures*, 80(5):495–513, 2002.
- [6] Martin Bäker and Aviral Shrot. Inverse parameter identification with finite element simulations using knowledge-based descriptors. *Computational Materials Science*, 69:128–136, 2013.
- [7] Klaus-Jürgen Bathe. *Finite element procedures*. Klaus-Jurgen Bathe, 2006.

- [8] David Borland and Russell M Taylor II. Rainbow color map (still) considered harmful. *IEEE computer graphics and applications*, (2):14–17, 2007.
- [9] Dassault Systems. *Abaqus Manual*, 2017.
- [10] Guido Dhondt. *The Finite Element Method for Three-Dimensional Thermomechanical Applications*. Wiley, 2004.
- [11] Ronald Krueger. Virtual crack closure technique: History, approach, and applications. *Applied Mechanics Reviews*, 57(2):109, 2004.
- [12] AM Prior. Applications of implicit and explicit finite element techniques to metal forming. *Journal of Materials Processing Technology*, 45(1):649–656, 1994.
- [13] Joachim Rösler, Harald Harders, and Martin Bäker. *Mechanical behaviour of engineering materials: metals, ceramics, polymers, and composites*. Springer Science & Business Media, 2007.
- [14] Peter Wriggers and Tod A Laursen. *Computational contact mechanics*, volume 30167. Springer, 2006.