Chapter 5
Finite Element Method (FEM)

Machining processes have been modelled numerically with the finite element method (FEM) for some years, leading already to highly promising results in the modelling of cutting processes. The use of numerical models for simulating cutting processes makes it possible to illustrate complex tools while simultaneously taking plasto-mechanical and thermal processes into consideration. Besides FEM, analytic and empirical process models can also be consulted that have the advantage of providing a quick representation of the process. Empirical models are of limited use: they are generally only calibrated to be valid for a limited process range. Due to the simplification, analytical models are only partially suitable for describing complex processes such as can be described by FEM. FEM is a numerical method for finding approximate solutions to continuous field problems. Originally, it was developed to solve stress problems in structural mechanics, but its use was soon expanded to the large field of continuum mechanics [Bett03].

5.1 Basic Concepts of FEM

It is helpful for the user to be acquainted with the basic concepts of FEM in order to avoid errors and to be able to evaluate the results of calculations. The user should be aware of the assumptions made by the software and their possible effects on the calculation result. The consideration of a process with FEM is also called finite element analysis (FEA). The following steps are taken in every FEA [Hueb82, Zien00, Redd93, Roll93]:

1. discretization of the continuum,
2. selection of interpolation functions,
3. determination of the element properties,
4. assembly of the element equations and
5. solution of the equation system.

As the continuum is discretized, solution domains (e.g. the workpiece) are subdivided into a finite number of subdomains – the finite elements. The type, number, size and distribution of the elements is also determined. Continuum elements make

F. Klocke, *Manufacturing Processes 1*, RWTH edition,
DOI 10.1007/978-3-642-11979-8_5, © Springer-Verlag Berlin Heidelberg 2011

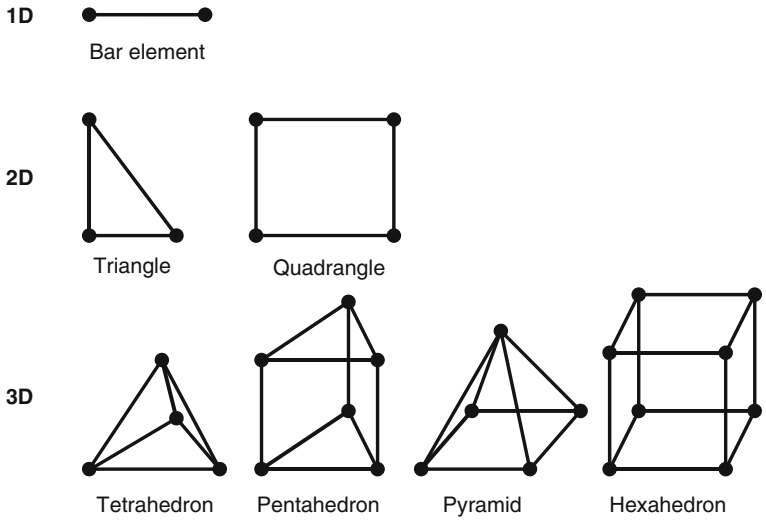

Fig. 5.1 Types of elements for the discretization of continuum problems, acc. to STEINBRUCH [Stei98]

it possible to grasp all normal and shear stresses. The complete definition of an element type includes the element form, the number of nodes, the type of node variables and the interpolation functions. Figure 5.1 shows exemplary some element types that are used for discretization.

The selection of interpolation functions, which are often designated as shape, form or basic functions, is made in practice simultaneously with the selection of the element type. Interpolation functions serve to approximate the profile of state variables within an element. Element nodes function as support points for the interpolation. In the case of linear elements, these are the corner points of the element. Higher order elements have a set number of additional nodes on the element edges or inside the elements; due to their higher number of nodes, they provide a more exact solution. Because of their differentiability or integratability, polynomials are frequently used as interpolation functions. The order of the polynomial depends on the number of element nodes, the number of unknowns of each node and the continuity conditions at the nodes. Since the interpolation functions represent the properties of the state variables within the element, the values of the state variables at the nodes represent the unknowns of the discretized problem.

After choosing element types and interpolation functions, the element equations (element matrices) are determined. These equations describe the relations between the primary unknowns (e.g. speed, displacement, temperature) and the secondary unknowns (e.g. stresses). To determine the unknowns, several approaches can be considered. One approach for example is using the principle of virtual work (energy).

One of the fundamental differences between FEM and other numerical methods of approximation is that the solution is first formulated for each individual element.

In order to approximate the properties of the total system comprising the sum of all the elements, the element matrices are combined (assembled) into the global matrix of the problem. The boundary conditions (clamps, external forces, etc.) are also defined. The assembly of element equations leads to system equations that can be solved with the help of the right methods. The numerical integration methods used to solve the element matrices require the evaluation of the integrals at certain points within an element, called integration points. The number of required integration points can be reduced while maintaining the same accuracy by careful selection of their positions. The Gauss quadrature is a very common method for numerical integration. The positions of the integration points within an element are exactly set and represent the positions at which stresses and strains are calculated [Koba89, Roll93, Zien00].

5.2 Lagrangian and Eulerian Considerations of the Continuum

The continuum can be discretized from different standpoints, where the LAGRANGian and EULERian approach are the most common in FEM [Bath96].

In the case of the LAGRANGian approach, the nodes of an element move with the material. An observer travelling on a node would see state variable changes of a particular particle throughout the entire forming process. One disadvantage of the Lagrangian method is the distortion of the mesh brought about by large plastic deformations, which sometimes requires remeshing. The now necessary interpolation of the state variables from the distorted to the newly generated mesh leads, depending on the number of remeshing cycles, to an undesirable, more or less distinct smoothing of the state variables.

The Eulerian approach considers the motion of the continuum through a fixed mesh. An observer on a node of such a mesh would see the states of all particles that pass his fixed observation point. This method is especially suited to the investigation of stationary processes and is frequently employed in flow simulations. The "arbitrary LAGRANGian EULERian" method (ALE) is becoming more and more accepted, which is a combination of the above approaches and permits the mesh a motion independent of the material as long as the form of the domains under consideration remains the same [Koba89, Wu03].

5.3 Explicit and Implicit Methods of Solution

Many FE programs utilized to calculate large plastic deformations make use of "implicit" methods. For highly dynamic applications on the other hand, such as crash simulation, explicit time integration is prevalent in FE programs.

Explicit methods consider the process under investigation as a dynamic problem subdivided into time steps. The desired quantities at time $t + \Delta t$ are determined solely from the values available at time t. This is done usually with the help of difference formulae. However, this method is only stable if the time step Δt is smaller

than the time it takes for an elastic wave to travel a route corresponding to the shortest element edge. In this way, the possible length of the time step is a function of the sonic velocity c existing in the material. For solids:

$$c = \sqrt{\frac{E}{\rho}} \tag{5.1}$$

The maximum possible length of the time step thus depends on the density ρ and the elastic modulus E of the material. Since the length of the time step can be in the range of microseconds, a very large amount of computing steps is sometimes necessary. "Mass scaling", i.e. artificial increase of the material's density or artificial reduction of the process time, represents an attempt to increase the possible length of the time step. Mass effects caused by such interventions have to be compensated by appropriate countermeasures [Roll93, Chun98].

When using implicit methods, there is no such limitation. Implicit solvers look for the solution for every time $t + \Delta t$ under consideration of the values of the desired quantities both at time t as well as at time $t + \Delta t$ [Hueb82]. The solution of such a non-linear system of equations demands special iteration methods (e.g. NEWTON-RAPHSON) [Roll93, Zien00]. The advantage of the length of the time step being up to 1000 times larger compared to the explicit method is therefore accompanied by the computing time required for the iterative equation solution.

5.4 Combined Thermal and Mechanical FEA

In cutting processes, heat is generated both from inelastic deformation as well as from the work of friction on the rake and flank faces. In order to take thermal processes into account, mechanical and thermal calculations must be combined. In the case of simultaneous combination, this is accomplished by establishing a completely combined equation system. Non-simultaneous combination proceeds from a purely mechanical formulation in which temperature merely serves to help ascertain temperature-dependent material characteristic values (e.g. the flow curve). The mechanical calculation determines the heat of friction, heat from plastic deformation and heat exchange with other objects or the environment. This data serve as input quantities for the thermal calculation. The combination can take place at every iteration (iterative combination), or at every time increment (incremental combination) [Kopp99].

5.5 Nonlinearities

Linear analysis is the analysis of a problem that exhibits a linear relationship between the applied load and the response of the system. Linear analysis is a simplification because every real physical system is nonlinear. These nonlinearities can

however be neglected in many cases. If we speak of nonlinear problems in FEM, these nonlinearities must be taken into consideration.

Basically, there are three different types of nonlinearities:

- material nonlinearities,
- geometric nonlinearities and
- nonlinearities in the boundary conditions.

Material nonlinearities result, for example, from a nonlinear relation between stress and displacement, as occurs in the case of metallic materials after leaving the Hookian range. Further causes of material nonlinearities include material behaviours contingent on forming speed and/or temperature as well as material failure. Geometric nonlinearities are caused by change in geometry during the calculation. As soon as the displacements are large enough to influence the behaviour of the system, there is a nonlinearity. Nonlinearities in the boundary conditions arise, for example, when there is a change in external loads or new contact, or in the case of loss of contact between two objects (e.g. tool and workpiece). In the context of a typical machining simulation, usually all three types of nonlinearity appear.

5.6 Material Laws

The material laws applied in FEM can basically be classified in two main groups: those concerning elastic material properties by considering the material elastically and in case of further deformation plastically, and those concerning the material as rigid until the plastic limit is reached (Fig. 5.2). The use of a rigid/plastic material law speeds up the calculation and provides satisfactory simulation results for many applications, in which the amount of plastic deformation is significantly larger than the amount of elastic deformation [Roll93]. Elastic/plastic material models are of importance when elastic effects are not negligible (e.g. the calculation of residual stresses remaining in the component).

Material laws for the simulation of chip-removing machining tasks must satisfy the particular requirements of cutting. To this belongs the description of material flow stress as a function of deformation ε, the rate of deformation $\mathrm{d}\varepsilon/\mathrm{d}t$ and temperature T. One can see from Table 5.1 how extreme the conditions are in machining in comparison to other manufacturing processes. Large deformations (up to $\varepsilon = 5$) and very high deformation rates ($\mathrm{d}\varepsilon/\mathrm{d}t < 10^6 1/\mathrm{s}$) combined with high temperatures ($T < 1500°\mathrm{C}$) are common in cutting processes.

To determine the flow curves that are valid in these extreme conditions, special test methods are used. Among these is, for example, the use of the Split-Hopkinson pressure bar. This testing apparatus specially developed for high speed deformation can reach deformation speeds of up to $\mathrm{d}\varepsilon/\mathrm{d}t = 10^4 1/\mathrm{s}$.

The arrangement of the Split-Hopkinson pressure bar consists of two cylindrical bars of equal diameter arranged in a line, the input and output bar. Between them is found a sample of a smaller cross-section than that of the bars. An accelerated mass strikes the input bar. This sudden stress induces an elastic compression wave

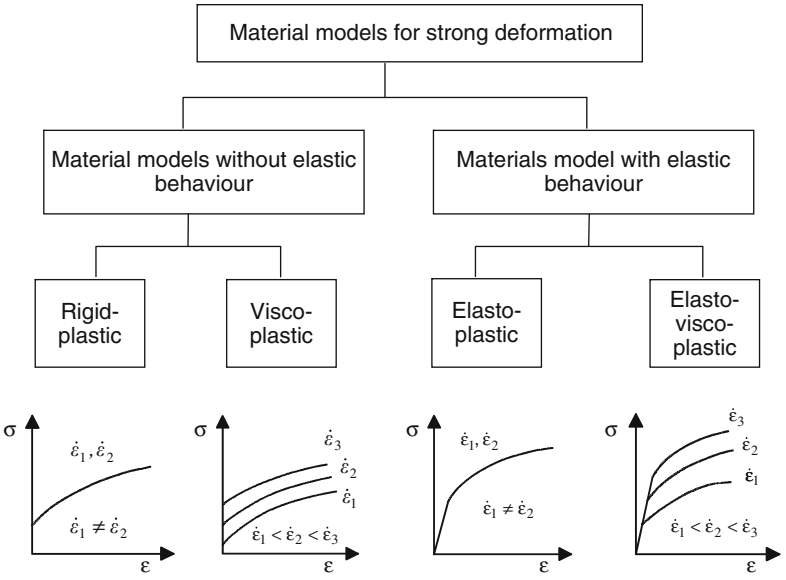

Fig. 5.2 Classification of the material models for high plastic deformation, acc. to ROLL [Roll93]

Table 5.1 Comparison of deformation, rate of deformation and temperature for various manufacuring processes, acc. to JASPERS [Jasp99]

Manufacturing process	Deformation	Deformation rate/s	$T_{\text{homologous}}$[a]
Extrusion	2–5	10^{-1}–10^{-2}	0.16–0.7
Forging/rolling	0.1–0.5	10–10^3	0.16–0.7
Sheet forming	0.1–0.5	10–10^2	0.16–0.7
Machining	1–5	10^3–10^6	0.16–0.9

[a] $T_{\text{homologous}} = T/T_{\text{melting}}$

that passes through the input bar and is measured at the first strain gauge. When it arrives at the contact area between the input bar and the sample, the wave is divided due to the change in cross-section in the following way: part of the elastic wave is reflected and the rest goes through the sample and plastically deforms it (because of the smaller cross-section of the sample). The remaining part of the wave which proceeds to the output bar is measured by the second strain gauge. With the time-strain curve determined by the strain gauges, both the compression speed and the flow curve can be determined [Abou05].

In order to reduce the number of experiments to a minimum and to make flow curve extrapolation possible, a constitutive material law is required that can be implemented in a FE program. This law must be capable of describing mechanical material properties under tensile, compressive or torsion load for a broad range

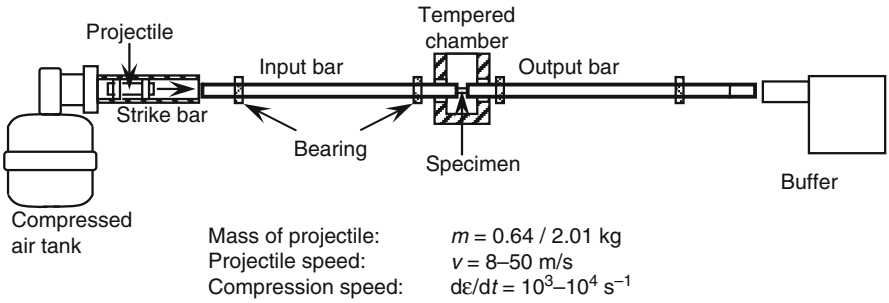

Fig. 5.3 Split-Hopkinson pressure bar, acc. to ABOURIDOUANE [Abou05]

of deformation, stress velocity and temperature. Several models have been developed for simulation, which take into consideration the influence of deformation (strain hardening), deformation speed (strain rate hardening) and temperature (thermal softening). Several of these are built on microstructure-mechanical foundations, yet most of the models used in cutting simulation are based on empirical methods.

For empirical material laws, usually constitutive equations are applied to describe flow curves. These equations make associations between the current values of stress, deformation, formation speed and temperature (σ, ε, $d\varepsilon/dt$, T as variables). The constants contained in these equations can be viewed as material parameters and adjusted to the experimental results with nonlinear regression or the method of least squares. SHIRAKASHI and USUI have suggested the following empirical relation,

$$\sigma = \mathrm{A} \cdot \varepsilon^n \cdot \dot{\varepsilon}^m \cdot [-\lambda\,(T - T_0)] \tag{5.2}$$

Which was successfully applied to describe dynamic viscoplastic material behaviour [Shir70]. Here, *A, n, m* and λ are material parameters.

MOLINARI and CLIFTON introduced a similar description,

$$\sigma = K(B + \varepsilon)^n \cdot \dot{\varepsilon}^m \cdot T^{-\upsilon} \tag{5.3}$$

which describes the strain hardening of the material (acc. to SWIFT) and the dependence of stress on temperature in another way [Moli83].

The material model frequently implemented in FE programs

$$\sigma = \left(A + B\varepsilon^n\right) \cdot (1 + C \cdot \ln(\dot{\varepsilon}/\dot{\varepsilon}_0)) \cdot \left(1 - \left[\frac{T - T_\mathrm{r}}{T_\mathrm{m} - T_\mathrm{r}}\right]^m\right) \tag{5.4}$$

originates from JOHNSON and COOK [John83]. In it, *A, B, n, C, m* are material constants, $(d\varepsilon/dt)_0$ a reference velocity and T_r, T_m room and absolute melting temperature. Material strain hardening is described in this model according to LUDWIK, velocity dependence logarithmically and the influence of temperature by means of a power function. However, an analytically closed formulation for the adiabatic flow curve is not possible with this temperature function.

JOHNSON and COOK also suggested a modified, simple equation with an exponential temperature dependence,

$$\sigma = \left(A + B\varepsilon^{\mathrm{n}}\right) \cdot (1 + C \cdot \ln(\dot{\varepsilon}/\dot{\varepsilon}_0)) \cdot e^{-\lambda(T-T_{\mathrm{r}})} \tag{5.5}$$

with which the temperature T by ε and $\mathrm{d}\varepsilon/\mathrm{d}t$ can be explicitly expressed. Here, λ is also a material constant, which takes into consideration the influence of temperature.

Based on physical foundations, ZERILLI and ARMSTRONG developed a semi-empirical model to describe material behaviour, which contains two equations [Zeri87]; the first type is meant for materials with fcc lattices:

$$\sigma = \Delta\sigma'_{\mathrm{G}} + C_2\varepsilon^{1/2} \cdot \exp\left[-C_3T + C_4T\ln(\dot{\varepsilon})\right] + k \cdot l^{-1/2} \tag{5.6}$$

and the second for materials with bcc lattice:

$$\sigma = \Delta\sigma'_{\mathrm{G}} + C_1\exp\left[-C_3T + C_4T\ln(\dot{\varepsilon})\right] + C_5\varepsilon^{n} + k \cdot l^{-1/2} \tag{5.7}$$

The ZERILLI-ARMSTRONG equation consists of additive components that contain an athermal component ($\Delta\sigma'_{\mathrm{G}}$: influence of dissolved materials and the initial dislocations density of inclusions), a thermal and velocity-affected component, the LUDWIK expression and a HALL-PETCH relation. Besides σ, ε, $\mathrm{d}\varepsilon/\mathrm{d}t$, and T, the grain size (l: average grain diameter) of the material to be modelled is also applied as an additional parameter. In 1995, a generalized, combined formulation was proposed by ZERILLI and ARMSTRONG [Zeri95]:

$$\sigma = C_0 + C_1 \cdot \exp\left[-C_2T + C_3T\ln(\dot{\varepsilon})\right] + C_4\varepsilon^{1/2} \cdot \exp\left[-C_5T + C_6T\ln(\dot{\varepsilon})\right] \tag{5.8}$$

5.7 Software

A number of FE programs have become commercially available that are tailored to simulating the cutting process. These programs are adapted to the requirements of machining technology, thus making it easier for the user to build and carry out simulations. The simplified operation of such specialized program systems usually entails a limitation how much the model can be influenced. On the other hand, "general purpose" systems are indeed highly flexible and can be used for a wide variety of applications, but they demand a large amount of experience for setting up the model as well as a larger amount of time. Depending on the spectrum of applications, the use of different FEM programs within one company is therefore quite common.

The following programs are used frequently in cutting simulations: SFTC/DEFORM™, THIRD WAVE/ADVANTEDGE™ and "general purpose" systems like ABAQUS or MSC/Marc [Denk04a].

The FE program DEFORM™ was originally developed for simulating forming processes. ADVANTEDGE™ is a program designed specially for machining, which

facilitates setting up and carrying out cutting simulations but is limited with respect to the influence of the user due to its specialization. Both programs are based on the implicit Lagrangian formulation and include an automatic remeshing routine followed by data interpolation from the old to the new mesh in order to avoid highly distorted mesh topologies. The DEFORM™ software also includes the Euler approach for calculating the quasi-stationary process condition.

Commercial FE programs make it possible to import object models from CAD volume software over standardized interfaces. Generally, the STL standard interface (*S*tandard *T*riangulation *L*anguage) is used as the exchange format for three-dimensional data models; often the interface generated for exchanging product data STEP (*ST*andard for the *E*xchange of *P*roduct model data) is used as well.

5.8 Hardware

With the rapid development of the computer industry, computational performance is being steadily improved. While a few years ago costly computer equipment was still required to carry out FEM calculations, nowadays already the majority of personal computers (PC) used at workstations fulfil the necessary requirements to carry out simple calculations with regard to memory and computing power. As a result of the low cost of hardware in the PC sector, the large majority of all commercial FE packages are also available for free or proprietary operating systems used in such computers. In order to handle very complicated problems with a large amount of elements, such hardware can also be operated in "clusters". Here, several computers are networked in such a way that an expansive problem can be calculated on several computers simultaneously. To do this, this function must be supported by the FE software. The computing time for a single problem does not however become reduced in direct proportion with the number of nodes (here: computers) in a cluster. The effective increase in speed depends to a larger extent on the hardware and software being used. The behaviour of a software in relation to the available computing power is called its scalability. A software scales well if the computing time required for a task is approximately halved when the available computing power is doubled.

5.9 Phases of a Finite Element Analysis (FEA)

A typical finite element analysis takes place in three phases from the standpoint of the user:

- data preparation with the preprocessor,
- calculation and
- evaluation of the results with the postprocessor.

With the help of the preprocessor, the user makes all the information necessary for modelling the problem available to the software. To do this, the problem must be abstract enough that the software can depict it. The simplifications made and the

quality of the data that the user inputs into the system is of decisive influence on the quality of the simulation results. As a rule, preprocessing has the following steps:

- defining the geometry,
- meshing,
- inputting the material data and
- defining the boundary conditions

Modern programs include import filters for common CAD formats. The geometries of the objects involved in the simulation can thus be taken from 2D or 3D CAD data sets of the design or tool construction in electronic form.

After defining the geometries of all the objects, they are meshed (discretized). For generating the mesh, most software manufacturers make tools available that make it possible to mesh geometries quickly. If the limitations of these tools are reached, separately offered commercial meshers can be used. If several element types are available, it is up to the user to make a selection between them. However, specialized software packages often specify one element type. The element type and the element density of the mesh have a significant effect on the quality of the simulation result. In principle, the use of a larger number of elements leads to more precise results. To save computing time, adaptive meshes with locally varying densities are frequently used in order to improve the resolution of local gradients in the state variables. Some programs offer the option of an automatic adaptive remeshing in which the element density is automatically adjusted to existing gradients.

After discretizing, we input the mechanical and thermo-physical material data. Following this, the boundary conditions (external loads, contact conditions, friction between different objects, speeds) are entered.

After calculation, the results are evaluated in the "postprocessor". Depending on the software, a variety of graphic representations and processing options are possible. Result evaluation also includes critical assessment by the user. It is essential to compare the results with those from experience, rough calculations or experimental results. Potential sources of error in FE analyses include:

- discretization errors from geometry interpolation when meshing and interpolation of the state variables,
- faulty input data (e.g. material data, process data, friction conditions),
- numerical errors (e.g. in numerical integration) and
- rounding errors due to the limited precision of the floating point representation in the computer.

5.10 The Use of FEM in Cutting Technology

For the most part, two-dimensional models are used to simulate the chip formation process. The FE model treats the free, orthogonal cross section because it is considerable less complex than, for example, external cylindrical turning (diagonal, bound cross section). The corner radius of the insert does not engage with the workpiece

and can therefore be ignored in the FE model. As long as the tool lead angle κ_r is equal to 90°, the tool inclination angle λ_s is equal to 0° and the depth of cut is many times larger than the undeformed chip thickness, it is permissible to assume a plane strain state of deformation, and the simulation of the cutting process is possible with a two-dimensional FE model. This cuts down on the time required for the simulation considerably.

5.10.1 Continuous Chip Simulation

Most cutting simulations use the Lagrangian method. In the Lagrangian formulation, the FE mesh follows the material, i.e. the nodes of an element move with the material, so that in the case of material deformation the elements are prestressed/distorted. Especially in cutting processes, particularly large deformations are seen in front of the cutting edge, the result of which is that the elements become highly distorted. In addition, considerable deformation and stress gradients appear in the area of the primary and secondary shear zones. For this reason, a finely structured mesh is necessary for a sufficiently precise representation in the FE model. Besides the difficulty of representing large deformations and deformation gradients in the model, there is also the problem of separating the chip from the workpiece. Definite crack formation in front of the cutting edge is still contentious and has not yet been demonstrated particularly in the case of cutting materials with ductile behaviour. In fact, in the case of ductile material, the material is extremely deformed in front of the cutting edge without forming an observable crack such as we see in forming processes. Crack developments have only been observed in the case of cutting brittle materials [Reul00, Schw36].

Basically, three different simulation methods exist for large deformations, deformations gradients and chip separation:

The separation can be realized

- on the basis of a geometric separation criterion, e.g. the criterion of the distance at which the separation begins as soon as the tool cutting edge has fallen short of a critical distance to the workpiece nodes lying ahead,
- on the basis of a physical separation criterion, e.g. exceeding a defined maximum effective strain or a previously set maximum stress, or
- by dispensing with a separation criterion (Fig. 5.4).

Besides providing a separation criterion in the first two approaches, it is necessary to define a separation line, along which the nodes are separated when the separation criterion is reached. This method lends itself to the use of a geometric separation criterion, i.e. as soon as the distance between the nodes and the cutting edge is below a critical distance – the length of one element edge as a rule – separation of the mesh occurs. The separation of the chip from the workpiece can also be realized by means of erasing the elements before the cutting edge of the tool. Element erasure is undertaken as a function of equivalent plastic strain or

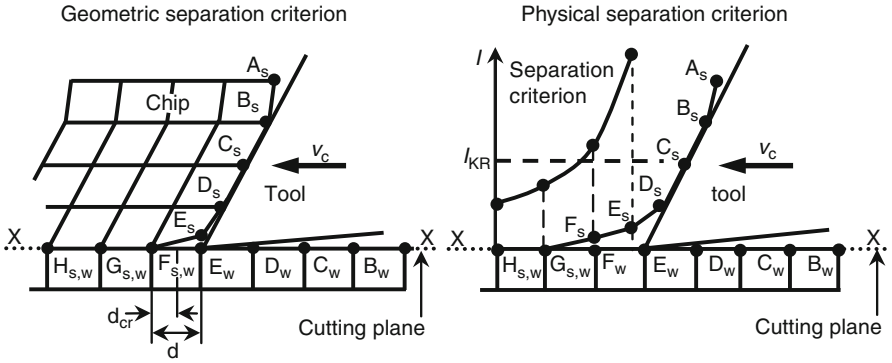

Fig. 5.4 Chip formation along a previously defined split line based on a geometrical separation criterion (on the *left*) and a physical separation criterion (on the *right*), acc. to VAZ [Vaz00]

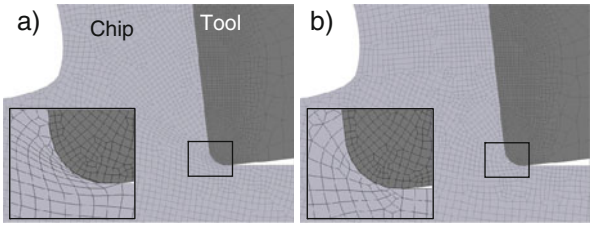

Fig. 5.5 (**a**) Distorted mesh topology before remeshing (**b**) New, undistorted mesh topology after remeshing

previous material damage [Sche06, Oute06]. One disadvantage of the element erasure method is that material is removed from the model – it must be guaranteed that this has no effect on the result of the simulation. When an automatic remeshing routine with subsequent data interpolation from the old to the new mesh is utilized and a purely ductile material is assumed, it is possible to simulate the cutting process without a separation criterion. The remeshing routine is invoked as soon as the elements are critically distorted (Fig. 5.5).

A criterion which can be used for automatically remeshing the model and interpolating the data during a calculation should fulfill the following conditions according to HABRAKEN and CESCOTTO [Habr90]:

- The criterion represents the quality of the mesh as faithfully as possible.
- The value of the criterion increases with increasing mesh deformity.
- If there is a remeshing, the value of the criterion is reduced.

In the actual remeshing, usually the outer edge of the old mesh is used as the starting point for the new mesh. The new mesh must now approximate the given edge as exactly as possible and balance any tool penetrations [West00]. After the successful remeshing, the data must finally be transferred from the old to the new discretization. The goal of a good data interpolation algorithm is to transfer the solutions of an FE

calculation at the integration points (e.g. temperature, stresses etc.) and node points (e.g. velocities, displacements etc.) of the old discretization as faithfully as possible to the new discretization. In the execution of the data transfer by means of extrapolation and interpolation, faults can arise that lead to a reduction of the size of calculated gradients. To avoid this effect, a fine discretization in the primary and secondary shear zone is necessary. As a rule, the mesher of a FE program offers the possibility of graduating the fineness of the mesh, whereby one should take heed that the transition of sections of varying discretization is defined as continuously as possible.

The information for the density distribution of elements is given with the so-called weight factors in the model. Besides the Lagrangian solution method, the Euler model is used to simulate static cutting processes. In contrast to the Lagrangian formulation, the material moves through a fixed mesh. The advantage of this is that separation criteria are unnecessary. By removing the material from the mesh, large deformations in front of the cutting edge no longer lead to highly distorted meshes, and no time-consuming remeshing is necessary. In order to represent a chip geometry that is as realistic as possible, an iterative adjustment of the free chip edges and chip surfaces is made in the calculation process based on an initial meshing of the chip root. Here, the stationary state is modelled: shear bands, entering and leaving processes as well as non-stationary chip formation processes, such as exist in milling, cannot be described by means of the Euler method [Leop01]. The fundamental advantage of the Euler method in comparison to the Lagrangian method is, besides dispensing with chip separation criteria, the shorter computing time for calculating a quasi-stationary process state.

5.10.2 Segmented Chip Simulation

Segmented chips are either produced by cracks and pores, adiabatic shear band formation or by a combination of both mechanisms. Segmented chips can be simulated in two ways:

- simulation of the segmented chip by deformation localization based on modified material characteristic values,
- simulation of the segmented chip by crack initiation based on fracture and crack hypotheses and
- a combination of both approaches

Deformation localization can be simulated either by a corresponding modification of the flow curve, which causes a softening of the material starting from a plastic limit strain, or by an artificial reduction of the specific thermal capacity and thermal conductivity. Generally, shear localization begins when thermal softening exceeds mechanical strain hardening. This softening leads to a concentration of plastic deformation, which results in a further increase in temperature and with it further concentration of plastic deformation. The process accelerates itself, leading to the formation of an adiabatic shear band [Abou05].

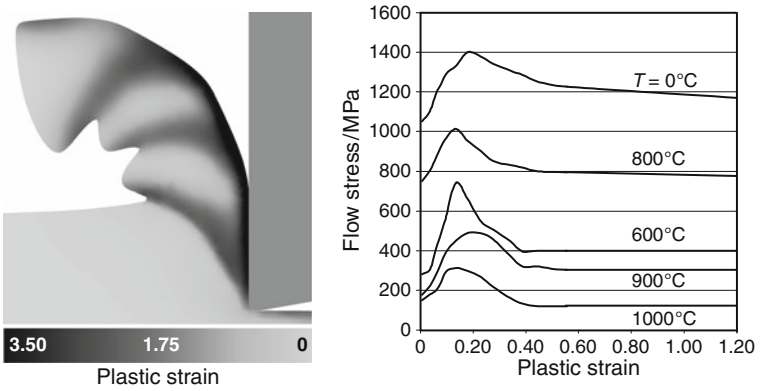

Fig. 5.6 Simulation of the segmented chip formation with $v_c = 25\,\text{m/ min}$ and $f = 0.20\,\text{mm}$ with appropriate flow curves which contain an artificial material hardening for high deformation

Figure 5.6 shows the simulation of a segmented chip while cutting titanium alloy Ti6-4 with uncoated cemented carbide cutting edges [Mess07]. The flow curves in Fig 5.6 were measured up to a strain of $\varepsilon = 0.25$ with help of high speed deformation experiments at a deformation velocity of 3000 1/s [Klim00]. For larger deformations, a material softening was assumed such as has been determined in the case of Ti6-4 for lower deformation speeds [Doeg86]. Between the chip segments, a localization of the deformations can be recognized, which leads to a softening of the material in accordance with the provided flow curve; the formation of shear bands can result from this.

If the segmented chip is simulated by crack initiation, such as occurs when cutting hardened steels, suitable fracture or crack hypotheses must be integrated into the FE model. In this case a distinction is drawn between macromechanical and micromechanical fracture hypotheses. Macromechanical fracture hypotheses describe the part of the form-changing energy introduced that has dissipated up to a crack, which serves as an indicator for the probability of failure of a material. Micromechanical failure hypotheses come from the consideration that a ductile fracture arises as a result of the formation, growth and consolidation of micropores. Considered microscopically, an inhomogeneous plastic deformation leads to the formation of microcracks, which form cavities under external loads [Brod01]. Macromechanical hypotheses are classified as ones that depend on the forming path or ones that are independent of it [Zitz95]. Since the capability to change form depends on the deformation history, hypotheses that only consider a momentary state of local process quantities (i.e. forming path- independent) are of only limited use for predicting the time of damage. On the other hand, hypotheses that take deformation history into account provide a damage value C, which is dependent on stress σ, expansion ε and material-specific parameters a (Eq. (5.9)). This damage value is added up over the forming path until a critical strain ε_c is reached. The critical damage value C_{crit} of the crack formation is a characteristic parameter of the

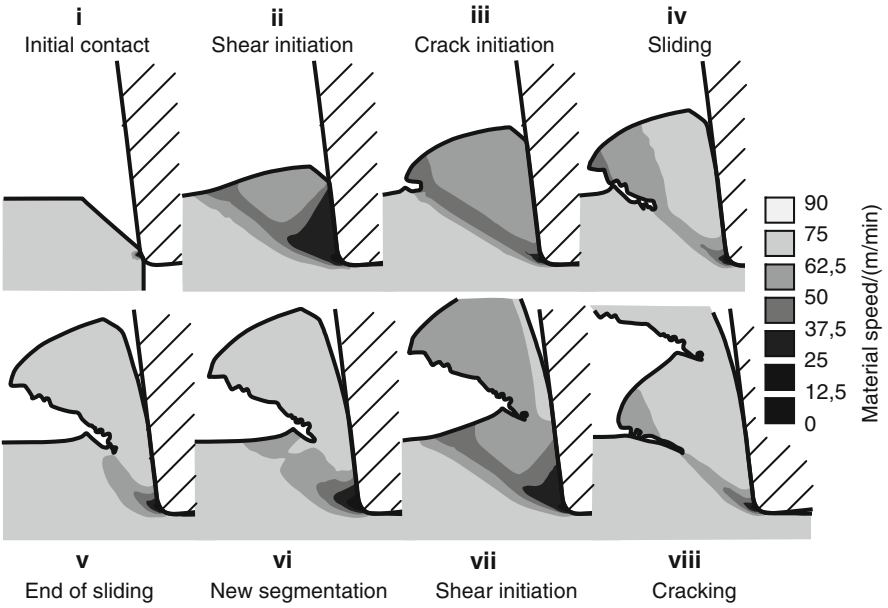

Fig. 5.7 Simulation of the segmented chip formation with $v_c = 70\,\mathrm{m/min}$, $f = 0.25\,\mathrm{mm}$ and $\gamma = -6°$ (ADI-900)

material [Kloc07]. As soon as an element in the FE simulation exceeds the critical damage value, it is erased.

$$C_{\mathrm{crit}} = \int_0^{\varepsilon_k} f(\sigma, \varepsilon, a)\mathrm{d}\varepsilon \tag{5.9}$$

Figure 5.7 provides an example of a simulation of segmented chip formation based on the integration of a fracture hypothesis when cutting ADI-900. BROZZO's criterion was used as a fracture hypothesis, which relates the tensile stress to the largest shear stress [Broz72]. Because experimental investigations have shown that the formation of segmented chips when machining ADI-900 is based on two different mechanisms, CHUZHOY's material model additionally assumes material damage as a function of the state values deformation and deformation velocity in order to take into account the accelerated sliding of the chip segments due to high shear stresses and the heterogeneous graphite sphere structure.

In the case of the segmented chip formation under consideration, after first contact and shear initiation a crack develops on the free surface which runs along the shear zone. The stagnation area disappears and the chip segment begins to slide jerkily from the cutting edge. Due to the crack which has developed, the supporting cross-section between the chip segment and the base material becomes smaller. The shear angle gets larger so that the chip segment is further pushed out of the chip root area. Starting from a critical shear angle, the original shear zone collapses and splits

up: one part wanders with the chip segment across the rake face; the other part reforms after repeated compression in from of the cutting edge, and the segmentation process repeats itself [Kloc07].

5.10.3 Simulation of the Cutting Process

5.10.3.1 Simulation of the Turning Process

Computing power is becoming increasingly cheaper and faster with the rapid development of the computer industry. This makes it possible to simulate cutting processes that do not allow for the assumption of a plane strain state three-dimensionally. Figure 5.8 gives an example of chip formation simulation for the external cylindrical turning of a normally annealed heat-treated steel C45E+N. In order to reduce computing time, only one section of the workpiece was treated in the model. The values for friction and heat transfer were taken from the model for machining C45E+N verified by HOPPE [Hopp03]. The mechanical material properties of the normally annealed heat-treated steel C45E+N were determined by EL-MAGD with the help of tensile and compression tests [Elma06]. The tool used was a Al_2O_3/TiN-coated insert of the form CNMG120408.

When wear models are integrated into the simulation, not only stresses, temperatures and chip forms can be predicted, but also possible tool wear. USUI discovered in experimental investigations that wear velocity dW/dt can be calculated with the

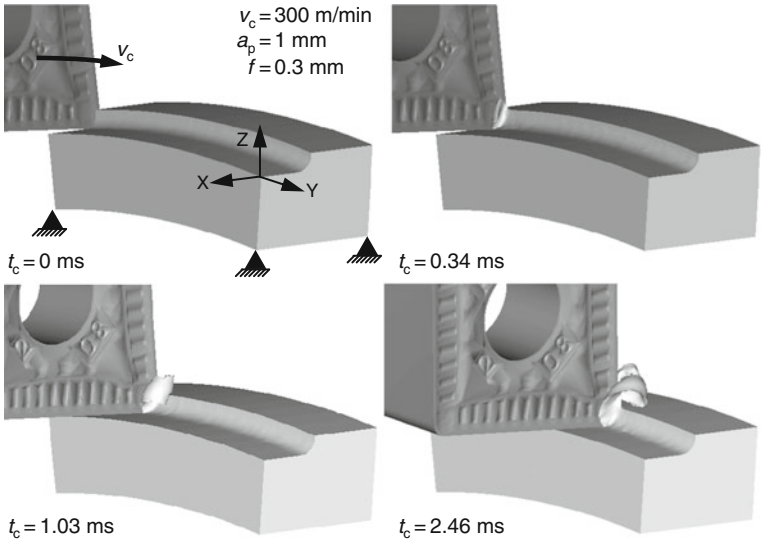

Fig. 5.8 Simulated first cut in external turning of C45E+N

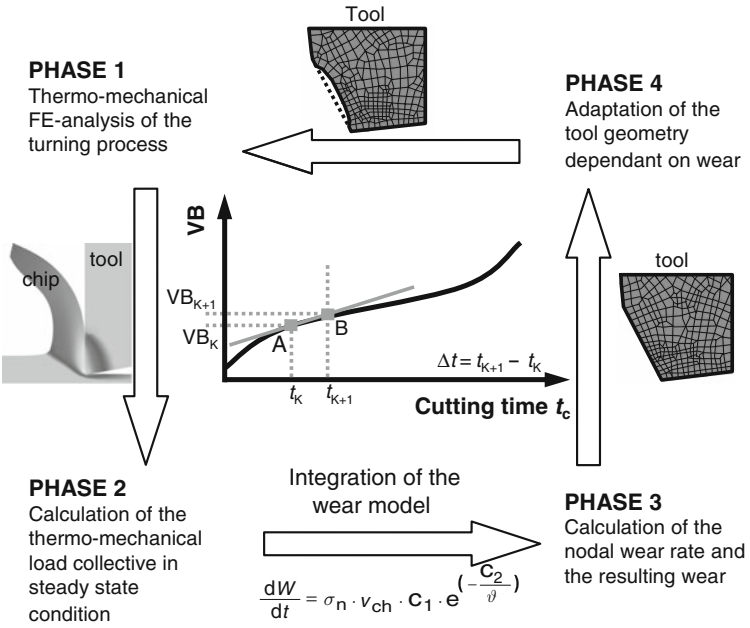

Fig. 5.9 Simulation of tool wear

following equation when the normal stress σ_n, sliding velocity v_{ch} and surface temperature of the tool ϑ are known [Usui78]:

$$\frac{dW}{dt} = \sigma_n \cdot v_{ch} \cdot C_1 \cdot \exp\left(-\frac{C_2}{\vartheta}\right) \tag{5.10}$$

The material constants C_1 and C_2 are dependent on the material/cutting tool material combination. To determine them, ALTAN has suggested a numerically combined calibration method in which the specific material constants can be determined based on experimental results for wear velocity and numerically calculated values for σ_n, v_{ch} and ϑ [Alta02]. Figure 5.9 shows the basic sequence of a wear simulation. After the collected thermo-mechanical load for the stationary state has been calculated in the first and second phases, the current wear rate dW/dt is calculated in the third phase based on the wear model provided in the user subroutine. This is followed by the modelling of the worn tool in the fourth phase, whereby the nodes of the tool contour are shifted as a function of the calculated wear volume ΔW. This 4-phase wear calculation cycle is continued until a user-defined tool life criterion is reached, and the simulation is stopped.

Figure 5.10 provides an example of wear prediction of uncoated cemented carbide inserts. Until the wear criterion of $VB_T = 200\ \mu m$ is reached, a total of seven

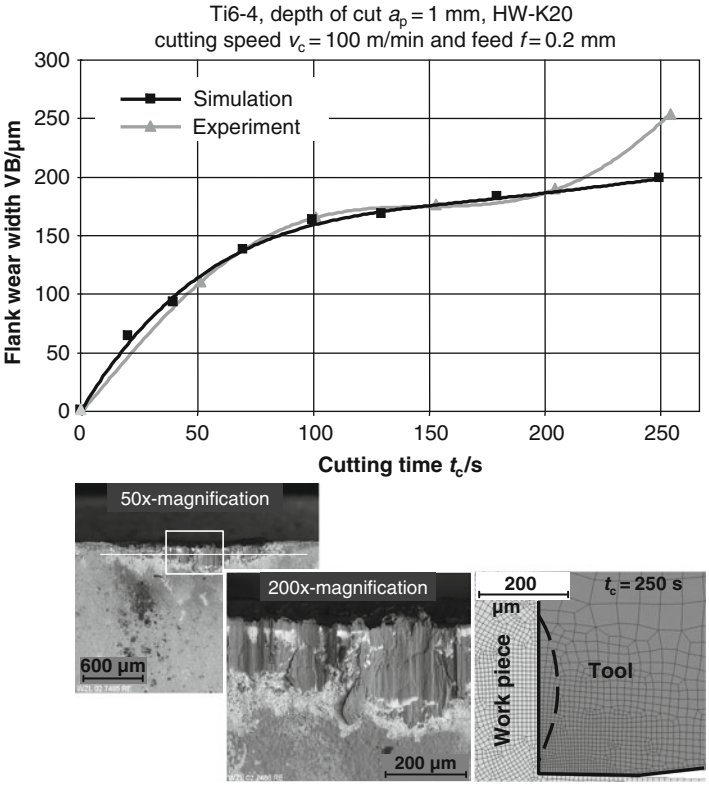

Fig. 5.10 Simulation of the flank face wear and comparison with the experimental results

4-phase cycles are calculated in the wear simulation. As the comparison of the wear curves of the simulation and the experiment shows, the simulation deviates only in the last calculation cycle. The SEM image of the cutting edge, which was taken at cutting time $t = 250$ s in the experiment, makes it clear that width of flank wear width VB increases erratically in the experiment as a result of the displacement of the cutting edge. As this point, the crater broke through the cutting edge and thus led to an acceleration of flank face wear. Since wear in the area of the rake face was ignored because of lacking material constants for crater wear in the FE simulation, simulating cutting edge displacement in the direction of the flank face was impossible.

5.10.3.2 Simulation of the Milling Process

The following example shows how cutting simulation has been applied in the industrial sector (Fig. 5.11):

Based on a numerical chip form simulation, the tool geometry of an inserted-tooth cutter was optimized with respect to chip flow in the product development

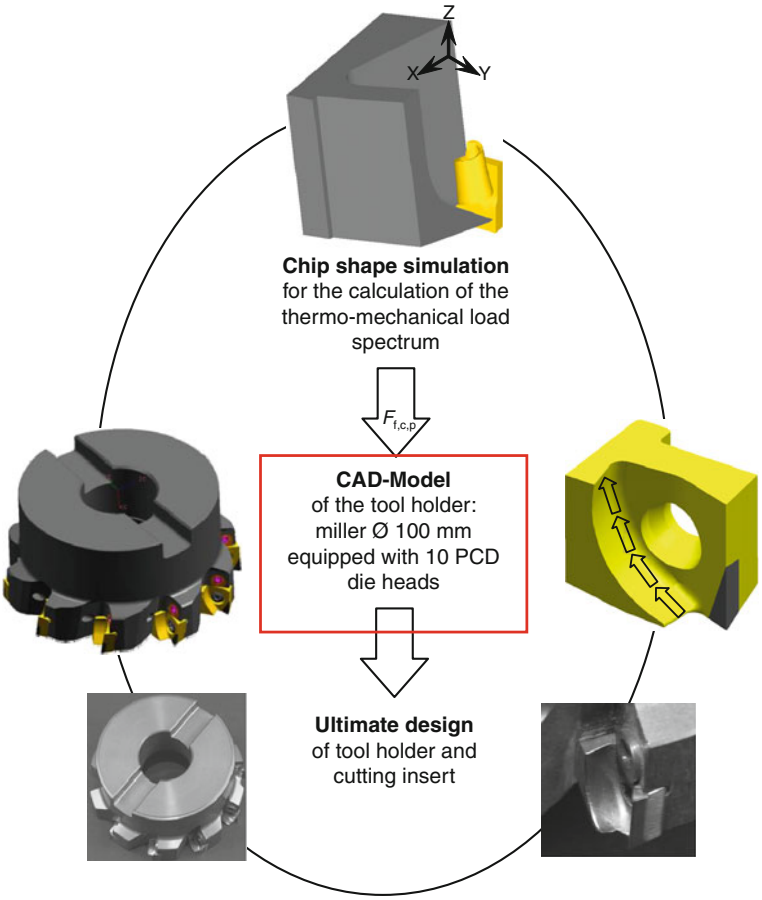

Fig. 5.11 Chip form simulation for the development of the tool body and the tool holder (Source: Kennametal)

phase. In the typical product development process for indexable inserts for metal machining, prototypes are manufactured and tested in several iteration stages in order to identify the ideal design.

The development process is time-consuming, extremely costly and usually lasts up to 8 weeks. By integrating FE simulations of chip formation into the development process, the number of iteration cycles in the design process can be significantly reduced. Furthermore, the design of the tool holder can be improved with the help of FE simulation; however, this type of simulation requires very exact input data regarding force and directions of force, which in the past could only be acquired with costly cutting experiments. Chip form simulation can be of assistance here. By simulating the chip flow, which essentially is determined by the lead angle of the cutting edge to the cutting direction as well as the chip former geometry, which is fixed in the tool body, predictions can be made not only about chip form but also

about forces and temperatures. These can then be utilized to help design the tool holder.

By incorporating chip formation simulation into the tool prototype development process, in the above example of an inserted-tooth cutter, we can find an optimal combination of adjusted, positive axial and radial lead angles as well as an optimized cutting edge bevel. This in turn leads to a considerably better surface quality after cutting. The chip is removed from the surface being produced, resulting in a very smooth surface. The two-tier cutting edge bevel results in lower cutting forces, which can minimize burr formation.

5.10.3.3 Simulation of the Drilling Process

Drilling is the most common means used in tool production. From the standpoint of cost and productivity, modelling and optimizing drilling processes is thus an of

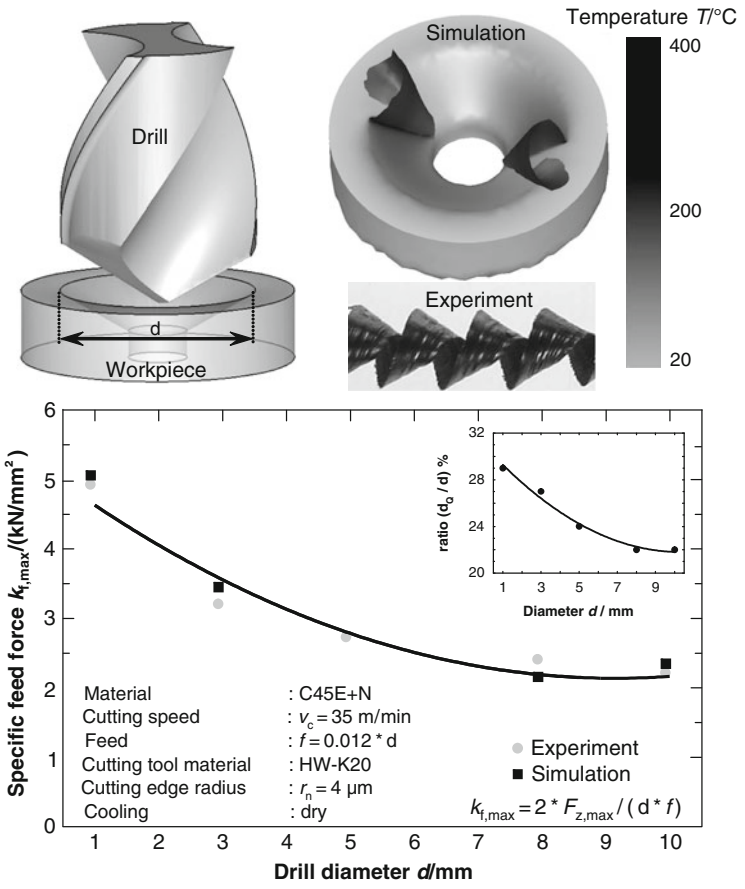

Fig. 5.12 FE-model of the drilling process compared with the experiment

enormous importance in the manufacturing industry. The drilling process is particularly challenging for 3D-FE simulation, demanding a lot of computing power from the hardware and efficient simulation tools due to the numerous influencing parameters involved (e.g. the complex geometry of the drill, cutting edge rounding, different contact friction processes, difficulty definable heat transfer and thermo-mechanical material properties) [Kloc06].

Due to its complexity, the process can not be represented by 2D simulation as can the orthogonal cross section in turning. A plane strain deformation state cannot be assumed in the case of drilling, since there are different cutting speeds along the drill radius and usually very complex, curved cutting edge geometries, causing a transverse material flow. For this reason, only a three-dimensional approach is purposeful. However, this increases the costs of both implementation and computation exponentially, whereby the computing time with today's computer technology is increased by the third power of the model size. Nevertheless, the tool and workpiece must be discretized in detail as volume bodies into finite elements. In order to reach a satisfactory level of accuracy in the simulation results, the most crucial area of the major cutting edge should be meshed especially finely [Kloc06]. Figure 5.12 gives an example of an FE model of the drilling process. In order to shorten computing time, the drilling process is considered from the point at which the entire major cutting edge is first engaged. The cylindrical workpiece model is adjusted to the material of the first cut process by notching a conic section on the cylinder. To verify the drilling model, both the chip form and the calculated feed force are compared with experimental results. The high level of agreement between them shows that even for such complex processes as drilling, numerical simulation of diverse target figures is possible.