Available online at www.sciencedirect.com

ScienceDirect

Procedia CIRP 31 (2015) 41 – 46

www.elsevier.com/locate/procedia

15th CIRP Conference on Modelling of Machining Operations

Finite Element Simulation for Quality Dependent Lifetime Analysis of Micro Gears

B. Haefner[a]*, M. Quiring[a], J. Gullasch[a], G. Glaser[a], T. Dmytruk[a], G. Lanza[a]

[a]*wbk Institute of Production Science, Karlsruhe Institute of Technology (KIT), Germany*

* Corresponding author. Tel.: +49-721-608-44296; fax: +49-721-608-45005. *E-mail address:* benjamin.haefner@kit.edu

Abstract

Nowadays, micro motors are used in combination with micro transmissions in manifold industrial applications such as dental drills or the equipment for minimally invasive surgery in the medical industry, hexapod micro positioning systems for wafer processing in the field of industrial automation or adjustable automotive components such as fixings of LCD monitors. Micro transmissions consist of micro gears, which are critical to their functionality. Micro gears are typically defined as gears with a module which is lower than 200 µm. To ensure proper operation of the micro gears for their expected purpose, a reliable prediction of their lifetime is crucial. Lifetime evaluation is particularly important for micro gears, as the influence of their geometric shape deviations on their load rating is significantly higher in comparison to gears with larger modules. This is a consequence of the larger shape deviations of micro gears in relation to their part size due to their manufacturing processes. The lifetime of micro gears can be evaluated by an experimental approach. Within this a pair of micro gears is systematically worn under realistic, clearly defined conditions, until a defect of one of the micro gears can be detected. This can be conducted by means of a highly precise experimental setup. In this article, a methodology to calculate the characteristic loads at the tooth flanks of the pair of micro gears during the experiments based on finite element analysis is introduced. For this purpose, CAD models of the real gear geometry of the specimen are deducted by means of high precision 3D measurements and spline interpolation. On the basis of these data, the lifetime of the micro gears dependent on their shape deviations can be predicted by means of a model based on reliability statistics.

Peer-review under responsibility of the International Scientific Committee of the “15th Conference on Modelling of Machining Operations

Keywords: Gear; Finite element method (FEM); Micro structure; Coordinate measuring machine (CMM)

1. Introduction

Micro transmissions are used in combination with micro motors in manifold industrial applications such as active prostheses, dental drills or movable automotive components such as LCD monitors. Micro gears are critical components to the desired functionality of the transmissions. They are defined as gears with a module lower than 200 µm [1].

Micro gears in transmissions are typically steel spur gears processed by hobbing. A hobbing machine consists of two skew spindles, one with the cutting tool and the other with the workpiece, which rotate at a proportional ratio [2]. In the industrial practice, hobbing is the most accurate process for micro gears due to its machining precision. Nevertheless, the micro gears show large shape deviations compared to their part size. Respective shape deviations of the flanks of 5-10 µm can significantly influence the rolling action of the gears.

To guarantee a reliable operation of the micro gears in the transmission, though, a valid method for their lifetime prognosis is crucial. However, as shown by Braykoff [3], existing approaches developed for the reliability analysis of larger gears are imprecise for the specific case of micro gears. The calculation of the load capacity according to ISO 6336 [4], in particular, does not account for the extraordinary magnitude of the shape deviations of micro gears. A valid model is very important to specify functionally meaningful tolerances for the hobbing process of micro gears.

In chapter 2 of this article, a research approach is described which models the influence of the manufacturing quality, e.g. of hobbing, on the lifetime of the gears. A crucial element of the research approach is the development of a suitable finite elements (FE) model of the gear operation, which is outlined in chapter 3. Chapter 4 and 5 comment on important details of the FE model. Finally, chapter 5 concludes with a summary.

Peer-review under responsibility of the International Scientific Committee of the “15th Conference on Modelling of Machining Operations
doi:10.1016/j.procir.2015.03.047

2. Lifetime prediction of micro gears dependent on their manufacturing quality

The presented methodology for the lifetime prediction of micro gears dependent on their manufacturing quality is based on an experimental approach (Fig. 1) [5]. Samples of pairs of micro gears are systematically worn under realistic, clearly defined conditions, until a defect of the gear pair can be detected. These experiments are conducted by means of a highly precise experimental setup. At the beginning as well as at certain intermediate times during the experiments, the geometry of the micro gears is measured by means of a micro coordinate measuring machine (CMM), which has a very low measurement uncertainty of approx. 250 nm. The measured data points serve as an input for a finite element analysis of the gear pair. As a result of the finite element analysis the characteristic loads of the micro gears can be determined.

Based on the determined loads by the finite element analysis and the failure times of the micro gears in the lifetime experiments, a load dependent reliability model can be developed for the micro gears. By means of this, the lifetime of a pair of micro gears of a specific gear type can be predicted based on the measured shape deviations of the micro gears.

The model is exemplarily implemented for a pair of typical hobbed, steel gears with integrated shafts with z=17 teeth, a module of m=175µm and a width of b=2mm for both gears. The gear pair is rolled at a rotational speed of n=500/min and a torque of M=50mNm in the experimental setup.

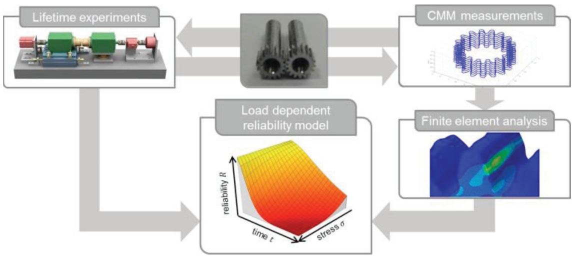

Fig. 1. Overview of the research approach.

3. Finite Elements Modelling of Micro Gear Operation

3.1. Model Overview

As stated in chapter 2, the purpose of the finite elements model in this research approach is to calculate the characteristic mechanical loads of the micro gear operation which determine the occurrence of a defect. This research approach only considers tooth root break as cause of failure, which is the crucial failure mode of the examined micro gears. The measured real gear geometries as well as their conditions in the experimental setup (gear position, rotational speed, torque) are inputs to the simulation. As output the stress is evaluated at different positions of each tooth root according to the von-Mises criterion.

The simulation is realized as 3D FE model in the software Abaqus 6.12. In the FE model a pair of micro gears with measured gear geometries is rotated by 360° and the respective tooth root stresses are evaluated. Fig. 2 illustrates the operation of an exemplary pair of micro gears [6].

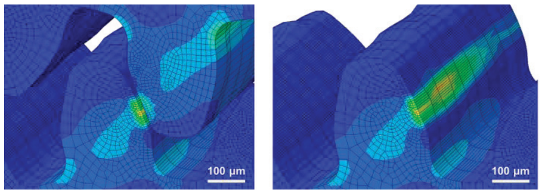

Fig. 2. FEM simulation of micro gears based on their measured geometry [6].

3.2. Specific Requirements for the Finite Elements Model

There are some specific requirements for the design of the FE simulation model in this research approach, as it is to be applied for the characterization of each pair of gear specimen in the experiments:

- Geometry: Representation of the gear geometries in the FE model as realistic as possible (lateral distances of measured data points 10 µm)
- Evaluation quality: Calculation of the tooth root stresses as realistic as possible
- Calculation effort: Acceptable calculation time and data size for available resources (approx. 4 simulations of 50 gear pairs each)
- Automation: Automatic generation, processing and evaluation of the FE model to reduce user effort
- Robustness: Stability of complex FE evaluation (which is challenging due to CAD models with measured input data)

Due to these contradictory requirements the design of the FE model is very complex. For an optimal realization of the model in Abaqus, there are manifold crucial simulation parameters, structured into 9 categories in the software, which have to be chosen (Fig. 3). In the following chapter some of the most important characteristics of the simulation design are described.

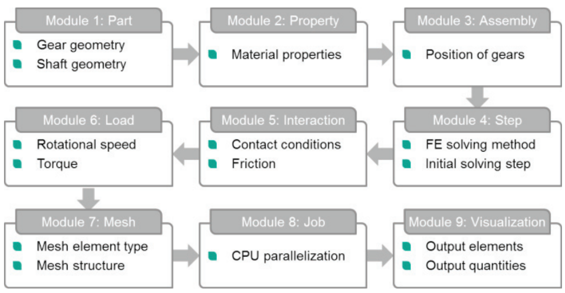

Fig. 3. Overview of the design of the FE model.

4. Selected Topics regarding the Design of the Finite Elements Model

4.1. Modelling of the Gear Geometry

For a realistic representation of the gear geometries in the FE simulation, 3D data points of the surfaces of each micro gears are measured at the tooth flanks, tooth roots and tooth tips with a high precision CMM. In order to use those for the FEM simulation in Abaqus, the data have to be transformed to a standard CAD model consisting of Non Uniform Rational B-Splines (NURBS) [7]. The data transformation is realized in a 2-step procedure by means of the software Mathworks MATLAB and the CAD software Siemens NX 9.

A MATLAB script has been developed to convert the measured data points to an areal NURBS model. Within this procedure, first the measured data points are transformed to a 2-dimensional B-Spline surface. The B-Spline surface is then converted to a NURBS surface according to [8] and the plain transverse surfaces at both sides of the gear are generated. Next, in order to support the FE pre-processing in Abaqus, the NURBS surface is partitioned into sub-surfaces based on [9]. Finally, the NURBS surface is exported to the universal CAD data format IGES according to [10].

In the second step in Siemens NX, the areal CAD model is converted to a volumetric CAD model, which can be imported to Abaqus.

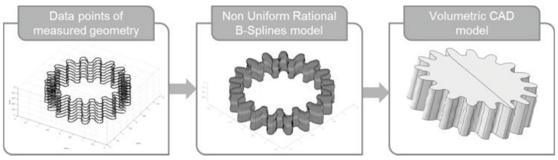

Fig. 4. CAD model of micro gears based on their measured geometry.

4.2. Modelling of the Shaft Geometry

Besides the gears, the shafts also have to be modelled, as their bending is crucial for a realistic tooth contact simulation. However, if the shafts are completely meshed, even with a coarse density, the computational effort is high due to the large volume of the shafts. A reasonable alternative is provided by the concept of connectors in Abaqus, which model kinematic relations by Lagrange multipliers [11].

Implementing the connector concept, the shaft geometry is modelled as an Euler Bernoulli beam with the material parameters of the shaft. The beam is fixed at the end of the shaft and has two force application points z_{S1} and z_{S2} (Fig. 5). At these points linear elastic springs with the stiffness C_1 and C_2 connect the shaft model with the meshed gear geometry. The force application points of the springs are located such that the Euler Bernoulli beam is optimally represented at the bottom and the top of the gear.

For comparison the shaft were simulated in Abaqus both fully meshed as well as with the connector model. The results show that the differences of the displacements between the models are less than 0.5 µm at all points at the gears. The simulation time decreased to 58% in the connector model.

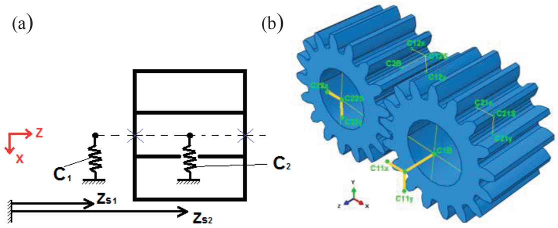

Fig. 5. (a) Euler Bernoulli model and (b) connector representation in Abaqus of the shaft geometry.

4.3. FE Solving Method

A FE model can be realized either as static or dynamic simulation. The main difference is that in a static simulation inertia effects (e.g. natural vibration, imbalance) are neglected. Movements and deformations over a series of time steps can be modelled, though [11]. Yet, static simulations require significantly less computation time. Thus, if inertia effects only have a minor influence on the quality of the simulation result, a static simulation is advantageous. According to [12] a static model is justified, if

$$\frac{p}{\varpi_{min}} < 0.25 \tag{1}$$

where p is the frequency of excitation and ϖ_{min} is the minimum natural angular frequency of the system. In the case of the gear pair, the frequency of excitation p is the angular frequency of the teeth contacts Ω_z [13]:

$$p = \Omega_z = 2\,\pi\,n\,z \tag{2}$$

With z=17 teeth and a rotational speed of n=500/min for the exemplary gear pair, formula (2) yields p=890.1/s. Thus, according to formula (1) $\varpi_{\min}$ has to exceed the natural angular frequency ϖ:

$$\varpi_{min} > \varpi = \frac{p}{0.25} = \frac{890.1\,/\,s}{0.25} = 3.6 \cdot 10^3/s \tag{3}$$

As relevant inertia effects (excited by the tooth contact) the natural angular frequency of the gears ϖ_{gear} as well as of the natural angular frequency of the shafts ϖ_{shaft} are analyzed.

ϖ_{gear} is calculated for the exemplary gear pair by means of the software STplus 6.0.0, which is part of the software suite FVA Workbench 3.10.0 according to [4] and yields:

$$\varpi_{gear} = 4.2 \cdot 10^5\,/s \gg \varpi = 3.6 \cdot 10^3/s \tag{4}$$

Clearly, the natural angular frequency of the gear exceeds the critical value by far.

For the calculation of ϖ_{shaft} the shafts are modelled with the gears as overhung rotors (Fig. 6) [14]. The natural angular frequencies ω of an overhung rotor are the roots of the term [13]:

$$(k_{11} - m\omega^2)(k_{22} + \omega n J_p - J_a\omega^2) - k_{12}^2 = 0 \quad (5)$$

n is the rotational speed, m is the mass of a gear and J_p and J_a are the moments of inertia around the rotating axis respectively in lateral direction. $k_{11} = 12EI/l^3$, $k_{12} = 6EI/l^2$ and $k_{22} = 4EI/l$ are the elements of the stiffness matrix of the model, where EI is the flexural rigidity and l the length of the shafts [15] (cf. [14] for more details regarding the overhung rotor model). The values of the input parameters for the exemplary gear pair are stated in table 1.

Table 1. Input values of the overhung rotor model for exemplary gear pair.

Input	Value	Input	Value	Input	Value
n	$500\,\frac{1}{min}$	J_p	$122\,\frac{kg}{m^2}$	EI	$1{,}7\,Nm^2$
m	$1{,}0 \cdot 10^{-8} kg$	J_a	$97\,\frac{kg}{m^2}$	l	$3\,mm$

Solving equation (5) with the values in table 1 gives the natural angular frequencies of the shaft (algebraic sign indicating the rotational direction of the vibrations):

$$\varpi_{shaft|1,2} = (\pm)6.7 \cdot 10^5\,/s \gg \varpi = 3.6 \cdot 10^3/s \quad (6)$$

$$\varpi_{shaft|3,4} = (\pm)2.9 \cdot 10^6\,/s \gg \varpi = 3.6 \cdot 10^3/s \quad (7)$$

As the natural angular frequencies of the shaft as well as of the gear are much larger than the critical value calculated by equation (3), it can be concluded that a static simulation is sufficient for the FE model of the micro gears.

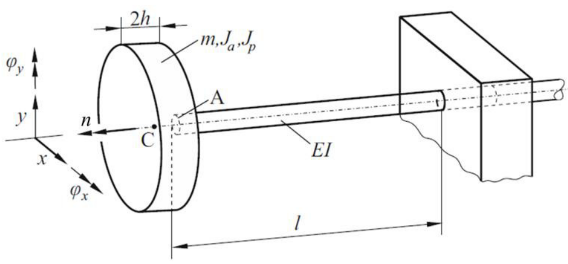

Fig. 6. Model of overhung rotor for the determination of the natural angular frequency of the shaft [14].

4.4. Mesh Elements

The definition of the mesh element is of crucial importance for the quality of a FE simulation. The main parameters are the selection of the element types of the mesh and their characteristics.

As continuum element types, in general, prismatic, tetrahedral or hexahedral elements can be considered. According to [11] hexahedral elements show the best convergence with limited calculation effort, which could be verified for the considered micro gears by FE simulations based on ideal gear geometries.

The hexahedral elements can be further characterized by the amount of notes per element as well as their integration properties. For an optimal selection of the suitable element characteristics these have to be considered in combination. Generally, there are two possibilities for the amount of notes of each element:

- Linear elements only have notes at each corner of an element
- Quadratic elements additionally have notes at each surface of the elements

The integration properties of the elements can be defined in three different ways in Abaqus 6.12 [11,16]:

- Elements with full integration
- Elements with reduced integration
- Elements with incompatible modes

The definitions of these are elaborated in the following. The interaction between the notes of an element is determined by its stiffness matrix, which is numerically calculated during the FE solving process by Gaussian quadrature [17]:

$$\int_I f(x)d\Omega = \sum_{i=1}^{m} f(x_i)\,\lambda_i \quad (8)$$

The domain of integration I is approximated by a weighted sum of the polynomials $f(x_i)$ at m integration points x_i with the weights λ_i. If the polynomials have a degree of $2n-1$, which yields an exact result, the finite elements are considered as fully integrated with respect to their integration properties [18].

For bending strains, like in the case of the teeth of gears, linear elements with full integration, however, typically show the phenomenon of shear locking, which leads to an overestimation of the stiffness of the elements and inaccurate simulation results [11,16]. Elements with reduced integration and elements with incompatible modes address this problem in different ways. In the case of elements with reduced integration the degree of the polynomials is decreased [16]. For linear elements with incompatible modes the displacement gradient $\partial u/\partial x$ of each elements is augmented with an additional, virtual displacement gradient field $\breve{G}$, [11]:

$$G = \frac{\partial u}{\partial u} + \breve{G} \quad (9)$$

Based on this, three potential element types in Abaqus 6.12 were preselected for further analysis:

- Linear elements with reduced integration (type C3D8R)
- Linear elements with incompatible modes (type C3D8I)
- Quadratic elements with reduced integration (C3D20R)

Linear elements with full integration were not considered because of the disadvantage of shear locking, which can be compensated by reduced integration or incompatible modes. The advantage of reduced integration is a decrease of the simulation time. However, in this mode the stiffness of the elements can be underestimated due to the phenomenon of hourglassing despite of respective compensation algorithms in Abaqus [11,16]. Quadratic elements typically lead to better simulation results than linear ones due to their additional notes, but have a significantly higher calculation effort for a specified mesh density [11,16]. Yet, the simulation time of quadratic elements can be decreased by means of reduced integration. In this case, there is no significant degradation of the simulation quality due to hourglassing [16].

The preselected alternatives were compared by FE simulations with ideal gear geometries in Abaqus. For comparison the gears were also simulated by means of the gear simulation software STIRAK 4.0, which is part of the software suite FVA Workbench 3.10.0. STIRAK is a FE based, gear-specific software for teeth contact analyses with loads [19]. While the FE simulation of measured gear geometries is not possible in the current software version, the simulation of nominal gear geometries has been demonstrated with fairly good accuracy [20]. The FE simulation in STIRAK is based on hexahedral, quadratic elements with 20 notes, which are comparable to the element type C3D20 with full integration in Abaqus.

Fig. 7 states the results of the simulations for the period of one teeth contact. To enable the comparison with STIRAK the major principal stress is evaluated as tooth root stress in this study. The results show that the C3D8R elements systematically underestimate the stress compared to the other simulations. The C3D8I and C3D20R elements lead to very similar results. The average absolute difference of the tooth root stress is less than 1.8%. For angles, at which there is only a single contact between the gear pair (angle 12°-22°), the results of the C3D8I and C3D20R elements are also close to the STIRAK simulation (average absolute difference 2.2% resp. 2.7%). At the other angles two teeth of one of the gears is in contact with one tooth of the other gear. For these angles there is a significant difference between the STIRAK simulation and the C3D8I and C3D20R elements in Abaqus. It is very likely that in this case the Abaqus results are more realistic than the STIRAK results, as in STIRAK at changing teeth contacts the tooth deformation is neglected due to its specific time-saving simulation algorithm [19].

In conclusion, linear elements with incompatible modes and quadratic elements with reduced integration are capable alternatives for the gear simulation in Abaqus. Yet, with very fine mesh density, which is necessary for the simulation of the gears based on their measured geometries, linear elements with incompatible modes take less computation time [11]. The aforementioned simulation, for instance, required only 17% of the simulation time for quadratic elements with reduced integration. Thus, linear elements with incompatible modes (Abaqus type C3D8I) are selected for the FE model.

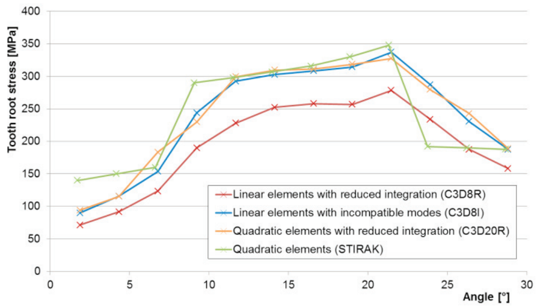

Fig. 7. Comparison of potential element types of the simulation model.

4.5. Mesh Structure

During the simulation of a gear pair, at each time only one or two teeth are in contact. Thus, the FE simulation is divided into partial simulations for each tooth contact of the angle φ = 360°/(no. teeth). The output values are finally combined to a common result. At each partial simulation only the two involved teeth of each gear are modelled by a very fine mesh (Fig. 8). The other teeth and the inner area of the gear, where the connectors of the shaft model are allocated, only have a coarse mesh. One tooth at each side of the finely meshed teeth has an intermediate mesh density to prevent strongly distorted elements in these regions. In tooth trace direction also a fine mesh is implemented. The selection of the specific mesh densities in the defined regions is subject to further analysis.

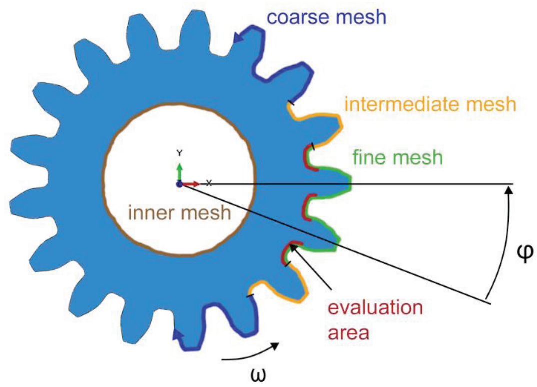

Fig. 8. Regions of different mesh densities in the FE model.

5. Automation of the Finite Elements Model

The FE model is to be applied for each specimen of gear pairs in the lifetime experiments. About 200 simulations are necessary in total, if approx. 4 simulations at different times during the lifetime experiments are assumed for each of the 50 gear pairs.

The manual effort of the developed FE model is more than 1 hour for the pre-processing and 30 min for the post-processing. Thus, due to the complexity of the model there is a high risk of human errors, e.g. at the setup of the manifold input parameters, the correct allocation of the faces for the measured gear geometries as well as the definition of the different mesh regions for each of the partial simulations or the combination of the evaluation results of the partial simulations to a common output file.

Besides the conventional Graphical User Interface (GUI), in Abaqus all user commands can be defined in an input script alternatively, which is transmitted by a Python interpreter to the Abaqus calculation kernel (Fig. 9) [11]. Thus, for the FE model of this research approach all necessary user command for the pre-processing, the simulation and the post-processing were implemented in Python scripts. If the scripts are run in Abaqus, the CAD models of the measured gear geometries are loaded and all processes are started automatically. Standardized output files of the tooth root stress are generated in xlsx format at each simulation step.

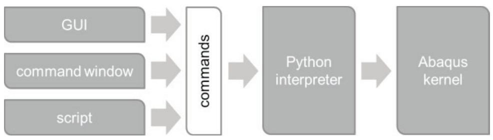

Fig. 9. Automation of the FE simulation by means of Python scripts [11].

6. Conclusion

The specification of functionally meaningful tolerances for the machining processes of micro gears such as hobbing is very important. Therefor, a suitable model to predict their lifetime dependent on the relatively large shape deviations caused by the machining is crucial. In this article, a respective research approach is outlined. In this methodology, specimen of gear pairs are rolled in a highly precise experimental setup and geometrically measured concurrently, until a failure occurs. Based on the measured geometries, the tooth root stress of the gear pair is calculated by means of finite elements simulations. Thus, a lifetime prediction model can be derived, by the statistical correlation of the failure times and the loads of the tooth root stress.

Moreover, in this article the FE model was elaborated, which is realized by means of Abaqus. The model requires both a high accuracy of the simulation results and computational efficiency. Thus several special characteristics are implemented in the FE model. The gear geometries in the simulation consist of CAD models based on measured data points. In contrast, the shafts are modelled as Euler Bernoulli beams by means of connectors to increase the computational efficiency. For the same reason a static simulation is chosen. It was shown that this is sufficient for the simulation of the micro gears. Furthermore, it was derived that hexahedral, linear elements with incompatible modes are the most suitable mesh element type for this simulation purpose. The mesh is structured into regions with fine density at the teeth in contact and a coarse density at the other parts of the gear. For a full rotation of the gear, the simulation is split into partial simulation of each tooth contact.

For the automation of the necessary user commands of the pre-processing, the simulation and the post-processing, Python scripts are implemented.

Future research will focus on a further refinement of the FE simulation as well its practical application for the load calculation in the lifetime experiments of the micro gears.

References

[1] NN. VDI 2731: Microgears - basic principles. Berlin: Beuth; 2009.
[2] Davim JP. Machining: fundamentals and recent advances. London: Springer; 2008.
[3] Braykoff C. Tragfähigkeit kleinmoduliger Zahnräder. Dissertation. University of Munich; 2007.
[4] NN. ISO 6336: Calculation of load capacity of spur and helical gears. Berlin: Beuth; 2006.
[5] Lanza G, Haefner B, Gibmeier J. Methodology for the lifetime prediction of micro gears dependent on present shape deviations and the material structure. Proc. European Saftey and Reliability Conference. Amsterdam: 2013. p 2663-2669.
[6] Haefner B, Quiring M, Hirtler M, Lanza G. Finite Element Analysis of Micro Gears for the Prediction of their Lifetime. Proc. 14th International Conference of the European Society for Precision Engineering & Nanotechnology. Dubrovnik: 2014. P. 277-280.
[7] Mukundan R.. Advanced Methods in Computer Graphics. London: Springer; 2012.
[8] Spink DM. NURBS Toolbox. http://www.mathworks.com/matlabcentral/fileexchange/26390-nurbs-toolbox-by-d-m-spink. 2010. Online access: March 28th, 2014.
[9] NN. B-spline Curves: Subdividing a B-spline Curve. http://www.cs.mtu.edu/~shene/COURSES/cs3621/NOTES/spline/Bspline/subdivision.html. 1998. Online access: March 20th, 2014.
[10] NN. igesout http://www.mathworks.com/matlabcentral/ fileexchange/14470-igesout. 2009. Online access: March 14th, 2014.
[11] Dassault Systèmes. Abaqus 6.12 Online Documentation. http://www.maths.cam.ac.uk/computing/software/abaqus_docs/docs/v6.12. 2012. Online access: March 20th, 2014.
[12] Bathe K. Finite element procedures in engineering analysis. Englewood Cliffs, NJ: Prentice Hall; 1996.
[13] Schlecht B. Maschinenlemente 2: Getriebe, Verzahnungen, Lagerungen. 2nd ed. Munich: Pearson; 2010.
[14] Brommundt E, Sachau D. Schwingungslehre mit Maschinendynamik. 2nd ed. Wiesbaden: Springer; 2008.
[15] Grote K, Feldhusen J. Dubbel. 24th ed. Berlin Heidelberg: Springer; 2014.
[16] Sun EQ. Shear Locking and Hourglassing in MSC Nastran, ABAQUS and ANSYS. In: Proc. MSC.Software Corporation's 2006 Americas Virtual Product Development Conference: Evolution to Enterprise Simulation. Huntington Beach, CA; 2006.
[17] Engeln-Müllges, G, Niederdrenk K, Wodicka R. Numerik-Algorithmen. 10th ed. Berlin, Heidelberg: Springer; 2011.
[18] Fish J, Belytschko T. A First Course In Finite Elements. Chichester: Wiley; 2008.
[19] Brecher C. FE-Stirnradkette v4.0. User manual. Aachen: 2010.
[20] Brecher C, Gorgels C, Gacka A. Anwendung der FE-basierten Zahnkontaktanalyse zur Berechnung von Lastverteilungsfaktoren für einen zertifizierten Tragfähigkeitsnachweis von Stirnradverzahnungen. Final report FVA 942; 2010.