INTERNATIONAL CONFERENCE ON ENGINEERING DESIGN
ICED 05 MELBOURNE, AUGUST 15-18, 2005

ENHANCEMENT IN COUPLING TOLERANCE ANALYSIS AND ELASTIC DEFORMATIONS ON THE EXAMPLE OF A SERIAL LINEAR SUPPORT SYSTEM

Ralph Lustig, Rüdiger Hochmuth and Harald Meerkamm

Abstract

The following paper presents the extension of a concept of linking elastic deformations and tolerance zones. This draft makes it possible to simulate the combined influence of elastic deformation and tolerance zones integral of former purely rigid supposed assemblies. Here we do not act on the component layer but on the assembly layer of a product classification. For this improvement in results the simulation requires a more exact modelling indeed. The relevant information and parameters have to be deposited in the model. A step towards this direction is given through the consideration of the real contact situation in the determination of elastic deformations. On the example of a serial linear support system the procedures are explained, demonstrated and the essential simulation results are represented.

Keywords: tolerance modelling, variational modelling, functional modelling, simulation

1 Introduction and Motivation

Tolerance and elasticity influences have comparable orders of magnitude and an alternating effect on the function of an assembly, for example in the joining process. Because of this fact the two parameters must be jointly simulated in the future. This is the bias that is to be observed for some time in industry and research [1, 2]. For some time procedures for the determination of both phenomena are among the state of the technology, for both Finite Element Analysis and Tolerance Analyses have a firm place in the product development process. Therefore it must be the aim to couple both influences in a suitable type and manner and thereby to enable a hybrid simulation. Basically the model characteristic can be distinguished in dependence of the respective system after CAD model geometry, tolerance model and FE-model. In CAD models all components are supposed as pure rigid and thus ideal, only tolerance statements can be attributed (fig. 1). In the tolerance analysis the component-/assembly-structure is supposed as stiff, tolerance data permit a variation of the boundary shape. FE-models suppose the boundary areas as ideal and define the material characteristics over the E-module. Our extension is the combination of both last mentioned aspects in one model. This can represent a field of use for sheet metal components, precision mechanics, mechanical engineering, medical technology. The extension based on flexible sheet metal components through the direct coupling of stochastic information and elasticity was shown in [3].

Table 1. System description

Type of Simulation	Component 1 Tolerance representation Stiffness representation	Component 2 Tolerance representation Stiffness representation
CAD-Situation	ideal ideal	ideal Ideal
Tolerance-Analysis	non-ideal absolutely stiff	non-ideal absolutely stiff
Finite-Element-Theory	ideal elastic	ideal elastic
Approach	non-ideal non-ideal	non-ideal non-ideal

2 Concept and Enhancement Possibilities

2.1 Basics

When connecting tolerances and elasticity in order to be able to determine the joint influence a suitable description language that fulfills several important factors must be found. Both phenomena have to be able to be expressed by it, a superposition should be subsequently possible and the integration in an already available, computer-aided-tool is required.

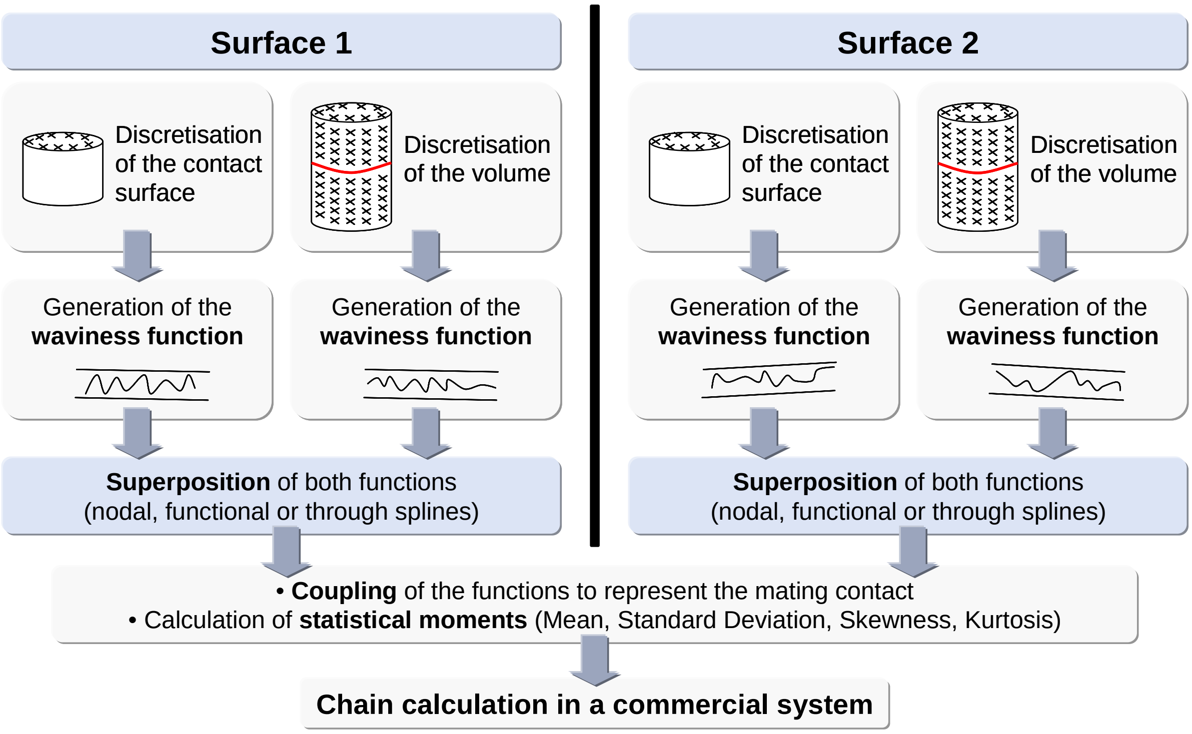

Figure 1. Concept for coupling tolerances and elastic deviations

The fundament of the concept is the description of both phenomena tolerances and elastic deformation on basis of mathematics (fig. 1) [4]. Through that each contact area can be characterized with a suitable surfaces function that consists of a certain amount of base points. Both elasticity and tolerance zones can be described and connected reasonable because they are of a same order of magnitude in the case of technically relevant assemblies. Each relevant contact area must therefore be discretized twice and therewith be characterized. Through superposition of the respective base points the discretized surfaces can be connected together stochastically thereafter, in order to be able to make a statement of the common influence. In order to simulate the complete scenario now, it is necessary that each pair of mating surfaces of different parts is varied stochastically at a time. Through that the actual, real contact is represented. For the integration into the simulation environment, a statistical distribution is calculated for each contact area, which can be characterized by the four statistical moments (mean, standard deviation, skewness, kurtosis) and can be processed further on.

2.2 Possibilities of contact representation

For the analysis of contact problems in FE-systems, the product developer has four different options. The first approach does not pursue an exact contact calculation in the actual sense because the meshing of assemblies with continuous meshes acts on the assumption that the involved components mate ideally together. Consequently three element types remain which can be uses reasonable for the described application [5]: Gap Elements, Slide Line Element, Contact Element.

Gap Elements

Whe using Gap Elements (fig. 2) interactions between the components are considered. Therefore the contact areas have to be connected by means of these element types at all places of opposing nodes. This technology makes great demands on the mesh generation but frequently the accuracy is afflicted with that fact. Thus it can occur that a mesh generation does not lead to the desired result or fails even completely. Moreover the assembly is considered globally in the use of Gap Elements, what means that the calculation result mainly represents the behaviour of the complete structure. But this is not determined in many problem definitions, because only some single components are provided with tolerances or other limits.

The application of this technology is not practicable problem-free. All contact places must be known and already be connected with Gap Elements before the start of the analysis. When concerning a large assembly which consists of lots of subassemblies with individual part movements, this procedure marks a large not trivial challenge. In addition this type of contact analysis allows only relatively small movements between the contact areas of the components, even in the case of geometric non-linearities. Through this a containment of the possible problem definitions arises even if the areas of contact are exactly predefined, because only those contacts can be calculated where the nodes of the single components adjust themselves. This again prefaces compatible FE-meshes.

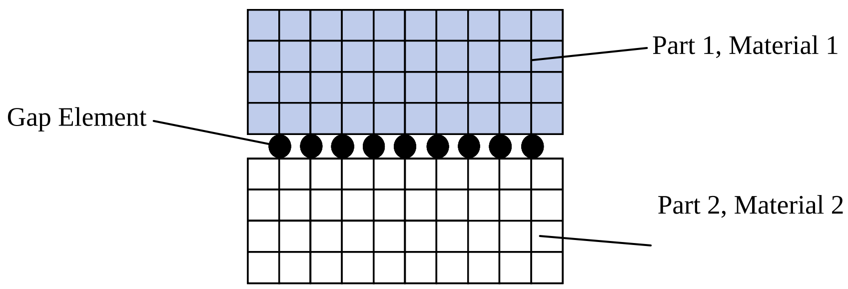

Figure 2. Schematic Representation of Gap Elements [5]

Slide-Line Elements

The essential difference between Gap and Slide-Line-Elements (fig. 3) lies therein that with this element type the analysis of large deformation and large relative component movements is possible. At the same time no exact knowledge of contact places are necessary in advance. Also no compatible meshes have to be used in the area of the boundary points. These element types are to be used essentially in the modelling of point-to-plane contacts, where the contact area of the component is clearly smaller than the component itself.

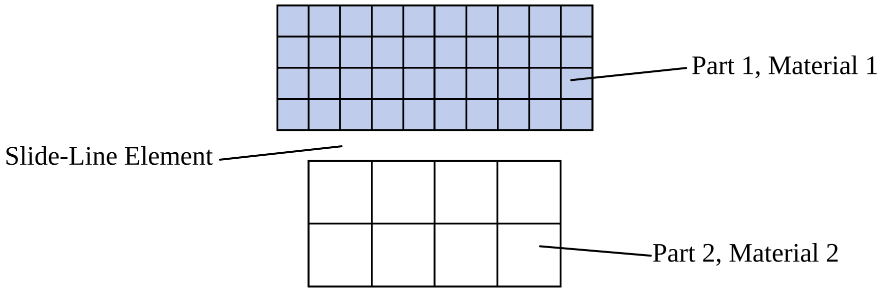

Figure 3. Schematic Representation of Slide-Line-Elements [5]

Contact Elements

The use of Contact Elements (fig. 4) represents the most common form of the contact analysis. They are suitable for many contact cases, both for large deformations, component movements, and in case of component penetrations before the start of the analysis. Also the initial boundary points do not have to be known.

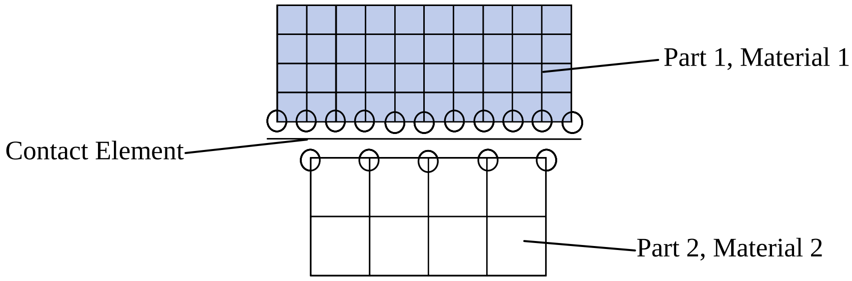

Figure 4. Schematic Representation of Contact Elements [5]

Although the use of Gap Element appears most unproblematic, finally Contact-Elements are used in the example. They are characterized by the fact that they offer the most parameters to the product developer in order to adapt the elements to real facts.

3 Modelling

The simulation object is the fundamental, simplified, serial construction of the tool/workpiece behaviour of a vertical turning centre (fig. 5).

Figure 5. Vertical turning center [6]

On a comparatively stiff chassis there are assembled guideways, on which a first sled can move in z-direction. On this first one, a second tool sled is mounted serially which can be moved along the x-axis. On that again the actual tool reception is located, the spindle box with the workpiece spindle.

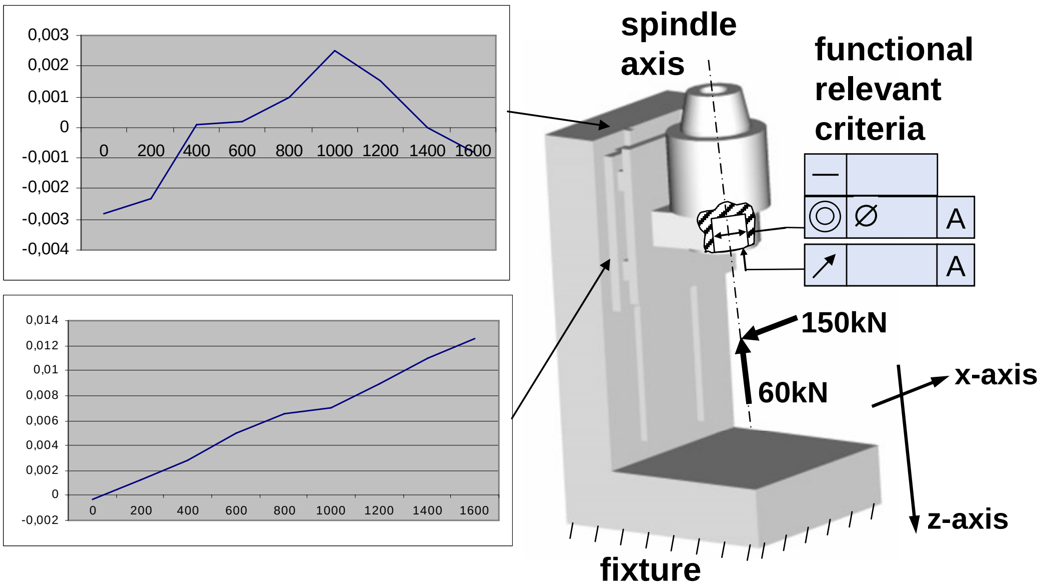

Figure 6. Modelling

The functional relevant question is the size of the coaxiality and straigthness of the spindle axis as well as the circular runout of the planar face of the spindle nose. The reference “A” is in this case a fixed axis of the chassis, which could be interpreted as a tooling axis. These criteria describe the resulting accuracy of manufactured parts with this kind of tooling machine. In predictive engineering it is important to have a focus on both aspects as tolerance chains and stiffness evaluation in order to reduce the iterative steps in design and/or experiments. Having a tool which integrates these analysis methods would improve the design quality and design duration.

The basic construction of the machine was modelled in the CAD system Pro/ENGINEER™ and was exported subsequently into the IGES-format (fig. 6). So it can be guaranteed that the necessary, pure geometric information remains and can be transferred into the simulation environment without any problems. The contact surface of main and support guidelines of x-and z-axis are discretized by a number of points, at which the coupling can be performed. The number of points can be varied. The points are modelled as hardpoints.

For coupling first the effects elasticity and tolerance zones have to be calculated and/or evaluated and subsequently transferred into a suitable description language. As a unifying language, the mathematical description of a technical surface by stastical moments is used. Furtheron kinematic variations of complex technical systems can also be simulated by a sequential calculation of the model.

3.1 Calculation of elastic deformations

For the calculation of elastic deformations, which result during manufacturing processes, applications specific boundary conditions have to be considered. In this way the complete base of the structure is fixed, the middle of the spindle is enforced with 60000N in axial direction and with 150000N in the operation axis. All relevant contact faces are meshed with so called contact elements. The automatic meshing algorithm generates 24839 elements and 5881 edges. For the exact coupling the sliding contact is modelled with hard points. With these the displacement in all three spacial dimensions can be established. The over-sized calculation of displacement is figured in figure 7. The result is a functional relevant displacement at the spindle face of 0,2 mm. Of course this value is above realistic conditions, but this example has been build up to encourage the functionality of the method and of the algorithm.

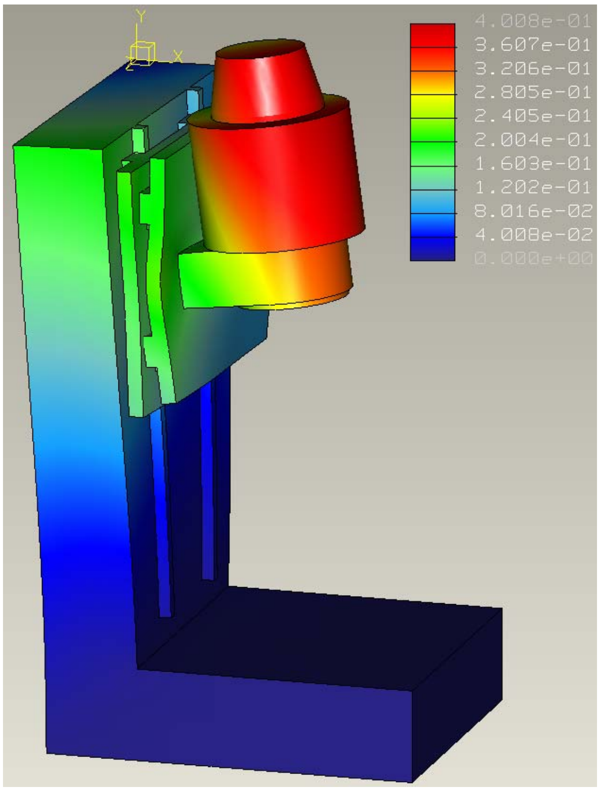

Figure 7. Displacements of the whole structure

Table 2 shows the results of elastic deformations at each control point. The control points are equal to them for tolerance analysis. This procedure is important to get corresponding results.

Table 2. Results for stiffness analysis

	Point	stiffness			tolerance zone			coupling	mean	std.dev.	skewness	courtosis
		DX	DY	DZ	x	y	z					
main left	APNT0	-0,161	0,009	0,074								
	APNT1	-0,140	0,008	0,068								
	APNT2	-0,119	0,008	0,061								
	APNT3	-0,097	0,008	0,054								
	APNT4	-0,076	0,008	0,045								
	APNT5	-0,055	0,007	0,035								
	APNT6	-0,036	0,006	0,024								
	APNT7	-0,020	0,005	0,015								
	APNT8	-0,008	0,003	0,007								
main right	APNT9	-0,081	0,017	0,075								
	APNT10	-0,064	0,018	0,069								
	APNT11	-0,048	0,017	0,063								
	APNT12	-0,032	0,016	0,056								
	APNT13	-0,019	0,014	0,048								
	APNT14	-0,009	0,011	0,037								
	APNT15	-0,003	0,008	0,026								
	APNT16	0,000	0,005	0,016								
	APNT17	0,001	0,002	0,008								
support left	APTN18	-0,153	0,012	0,077								
	APTN19	-0,135	0,012	0,072								
	APTN20	-0,113	0,011	0,066								
	APTN21	-0,092	0,010	0,059								
	APTN22	-0,073	0,009	0,051								
	APTN23	-0,050	0,009	0,037								
	APTN24	-0,033	0,007	0,026								
	APTN25	-0,018	0,005	0,016								
	APTN26	-0,007	0,004	0,007								
support right	APTN27	-0,072	0,020	0,078								
	APTN28	-0,055	0,019	0,073								
	APTN29	-0,040	0,020	0,067								
	APTN30	-0,025	0,019	0,061								
	APTN31	-0,010	0,020	0,055								
	APTN32	-0,004	0,012	0,039								
	APTN33	0,001	0,007	0,028								
	APTN34	0,002	0,004	0,017								
	APTN35	0,002	0,002	0,009								
main up	APTN36	-0,164	0,016	0,088								
	APTN37	-0,124	0,020	0,085								
	APTN38	-0,090	0,021	0,085								
	APTN39	-0,058	0,024	0,084								
main down	APTN45	-0,116	0,012	0,073								
	APTN46	-0,072	0,018	0,073								
	APTN47	-0,045	0,024	0,072								
	APTN48	-0,023	0,026	0,070								
support up	APTN40	-0,160	0,023	0,090								
	APTN41	-0,118	0,023	0,088								
	APTN42	-0,080	0,022	0,088								
	APTN43	-0,042	0,022	0,088								
support down	APTN49	-0,115	0,012	0,078								
	APTN50	-0,066	0,019	0,077								
	APTN51	-0,033	0,026	0,077								
	APTN52	-0,004	0,033	0,078								

3.2 Tolerance Analysis

The tolerance analysis results in deviations of each functional relevant surface. These deviations have been predefined (tab. 3) according the expected values. The calculation of the tolerance deviations was made upon the base of the deviations of the sliding elements as well as on a planeness of the base of the spindle support. 9874 iterations were calculated in this context. The control points are the same as mentioned above. The results are listed for the control points of the functional relevant faces.

Table 3. Results for tolerance analysis

	Point	stiffness			tolerance zone			coupling	mean	std.dev.	skewness	courtosis
		DX	DY	DZ	x	y	z					
main left	APNT0				-0,003	0,000	0,000					
	APNT1				-0,002	200,000	0,000					
	APNT2				0,000	400,000	0,000					
	APNT3				0,000	600,000	0,000					
	APNT4				0,001	800,000	0,000					
	APNT5				0,003	1000,000	0,000					
	APNT6				0,002	1200,000	0,000					
	APNT7				0,000	1400,000	0,000					
	APNT8				-0,001	1600,000	0,000					
main right	APNT9				0,006	0,000	0,000					
	APNT10				0,005	200,000	0,000					
	APNT11				0,003	400,000	0,000					
	APNT12				0,002	600,000	0,000					
	APNT13				0,002	800,000	0,000					
	APNT14				0,001	1000,000	0,000					
	APNT15				0,002	1200,000	0,000					
	APNT16				0,003	1400,000	0,000					
	APNT17				0,004	1600,000	0,000					
support left	APTN18				0,000	0,000	0,014					
	APTN19				0,000	200,000	0,014					
	APTN20				0,000	400,000	0,011					
	APTN21				0,000	600,000	0,009					
	APTN22				0,000	800,000	0,008					
	APTN23				0,000	1000,000	0,007					
	APTN24				0,000	1200,000	0,005					
	APTN25				0,000	1400,000	0,003					
	APTN26				0,000	1600,000	0,001					
support right	APTN27				0,000	0,000	0,014					
	APTN28				0,000	200,000	0,014					
	APTN29				0,000	400,000	0,011					
	APTN30				0,000	600,000	0,009					
	APTN31				0,000	800,000	0,008					
	APTN32				0,000	1000,000	0,007					
	APTN33				0,000	1200,000	0,005					
	APTN34				0,000	1400,000	0,003					
	APTN35				0,000	1600,000	0,001					
main up	APTN36				-0,003	0,000	0,000					
	APTN37				-0,002	0,000	200,000					
	APTN38				0,000	0,000	400,000					
	APTN39				0,001	0,000	600,000					
main down	APTN45				0,006	0,000	0,000					
	APTN46				0,006	0,000	200,000					
	APTN47				0,003	0,000	400,000					
	APTN48				0,002	0,000	600,000					
support up	APTN40				0,000	0,000	0,000					
	APTN41				0,000	0,001	200,000					
	APTN42				0,000	0,004	400,000					
	APTN43				0,000	0,005	600,000					
support down	APTN49				0,000	0,014	0,000					
	APTN50				0,000	0,014	200,000					
	APTN51				0,000	0,011	400,000					
	APTN52				0,000	0,009	600,000					

3.3 Simulation of the coupling of both aspects

The result of the tolerance simulation gives as a meaning of the deviations of each sliding element in tabellaric form as well as of the functional relevant support of the sliding system (fig. 6). The base is the Finite Element Calculation of the stiffness, the tolerance stack-up, the additional coupling and the derivation of all four statistical moments. These aspects are base of the common analysis of the assembly (tab. 4).

Table 4. Results for the coupling of both aspects

	point	stiffness			tolerance zone			coupling	mean	std.dev.	skewness	courtosis
		dx	dy	dz	x	y	z					
	0							-0,163				
	1							-0,143				
	2							-0,119				
	3							-0,097				
main left	4							-0,075	**-0,079**	**0,052**	**-0,205**	**1,696**
	5							-0,052				
	6							-0,034				
	7							-0,020				
	8							-0,009				
	0							-0,075				
	1							-0,059				
	2							-0,045				
	3							-0,030				
main right	4							-0,017	**-0,025**	**0,027**	**-0,545**	**1,918**
	5							-0,009				
	6							-0,002				
	7							0,003				
	8							0,005				
	0							-0,153				
	1							-0,135				
	2							-0,113				
	3							-0,092				
support left	4							-0,073	**-0,075**	**0,049**	**-0,153**	**1,670**
	5							-0,050				
	6							-0,033				
	7							-0,018				
	8							-0,007				
	0							-0,072				
	1							-0,055				
	2							-0,040				
	3							-0,025				
support right	4							-0,010	**-0,022**	**0,026**	**-0,716**	**2,074**
	5							-0,004				
	6							0,001				
	7							0,002				
	8							0,002				
	0							-0,167				
main up	1							-0,126	**-0,110**	**0,041**	**-0,131**	**1,661**
	2							-0,090				
	3							-0,058				
	0							-0,110				
main down	1							-0,067	**-0,060**	**0,033**	**-0,417**	**1,810**
	2							-0,043				
	3							-0,021				
	0							-0,160				
support up	1							-0,118	**-0,100**	**0,044**	**-0,042**	**1,657**
	2							-0,080				
	3							-0,042				
	0							-0,115				
support down	1							-0,066	**-0,055**	**0,041**	**-0,293**	**1,741**
	2							-0,033				
	3							-0,004				

The row “coupling” is the mathematical addition for all three components of both aspects, whereas the resulting value represents the perpendicular direction of the focused sliding element. Out of the nine or four values the mean-value, standard deviation, skewness and courtosis is derived. These values are now the base for the next step for modelling the effect-coupled system in a commercial tolerance analysis tool, which is fed with those values.

In coupling the results it is obvius that the run of the base plan of the spindle is behond 0,012 mm, whereas the coaxial deviation of the whole axis is in a range of 0,17 mm (fig. 8), which is of course a non-realistic value. The straightness of the whole axis can be fixed with a value of 0,004 mm.

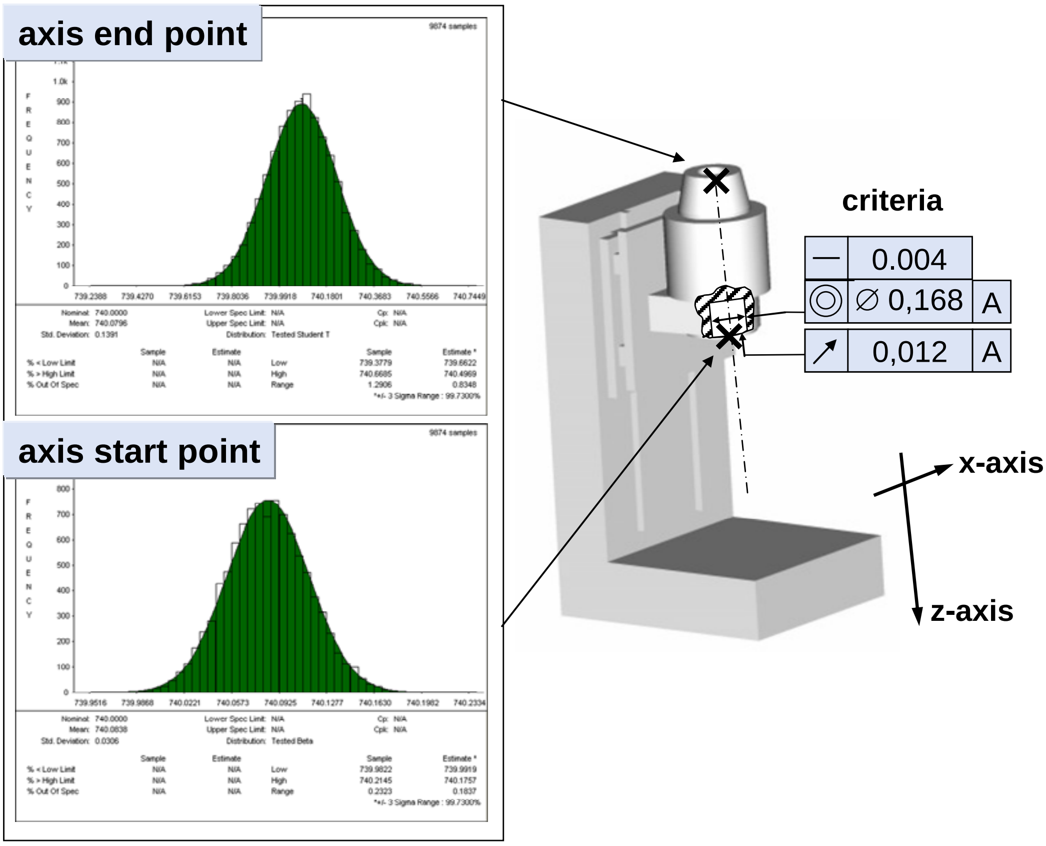

Figure 8. functional-relevant presentation of the results

This is the base for further optimization according both criteria. The presentation of results in a sensitivity window will give the hint due to the most influencing tolerances, elastic deformations and assembly operations.

4 Conclusion

The authors have shown that stiffness calculation and tolerance boundary conditions are able to be coupled. This can be realised in different ways, i.e. by using stochastic boundary conditions for Finite Element displacements [3, 7]. Here it is realised by the integral simulation of the parameter representation by four stochastical moments derived from tolerance criteria as well as from elastic aspects. They are coupled for both criteria, which results of the former undependent simulation models.

With the example of a vertical turning machine the method and the algorithm was presented exemplariliy. This example is added to real drilling machines but one has to take into account that the displacement values are not in a technical relevant, real range.

Further work will have to be done on the simulation of parallel structures, the sensitivity analysis of stiffness results besides the tolerance influence and in the coupling procedure, dynamic aspects, the knowledge based diagnosis of the results for an optimisation as part of the design process as well as the integral use of statistic process control data. An evaluation on realistic information will conclude.

References

[1] Glöggler C., Ströhle H., Mbang S., Haasis S., "Integriertes Toleranzmanagement im CAD-System", In: Konstruktion 06-2004, pp. 71-74.

[2] Von Praun S., "Toleranzanalyse nachgiebiger Baugruppen im Produktentstehungsprozess", Herbert Utz Verlag, München, 2002.

[3] Meerkamm, H., Lustig, R., Hochmuth, R., "Tolerance analysis of sheet metal assemblies with focus on non-rigid geometry", Proceedings of the SheMet 05, Erlangen, 2005.

[4] Lustig, R., "Linking Elastic Deformations and Fabrication Deviations in Computer-Aided-Analysis", Proceedings of the 14th International Conference on Engineering Design, Stockholm, 2003.

[5] N.N., NeNastran, http://www.nenastran.com, "Data Sheets for Gap/Slide-Line/Contact Surface Elements".

[6] N.N., Ansys White Paper, http://www.ansys.com, "Assembly Analysis".

[7] Hochmuth, R., "Methoden und Werkzeuge als Teil eines Assistenzsystems zur rechnergestützten Analyse und Optimierung robuster Produkte", Dissertation, Universität Erlangen-Nürnberg. Düsseldorf: VDI-Verlag, 2002.

Corresponding author:
Dipl.-Ing. Ralph Lustig
Dr.-Ing. Rüdiger Hochmuth
Friedrich-Alexander-University Erlangen-Nuremberg, Institute of Engineering Design
Martensstrasse 9
D - 91058 Erlangen
Germany
Tel: Int +9131 85 27987
Fax: Int +9131 85 27988
E-mail: lustig@mfk.uni-erlangen.de
URL: http://www.mfk.uni-erlangen.de