Available online at www.sciencedirect.com

SciVerse ScienceDirect

Procedia CIRP 2 (2012) 92 – 97

www.elsevier.com/locate/procedia

1st CIRP Global Web Conference: Interdisciplinary Research in Production Engineering

Multi-Scale Positioning Control Model of a Novel Fluid Dynamic Drive by Coupling Process and Adapted CFD Models

H.-C. Möhring[a], H. Kayapinar[a,]*, B. Denkena[a]

[a]*Leibniz Universität Hannover, Institute of Production Engineering and Machine Tools, 30823 Garbsen, Germany*

* Corresponding author. Tel.: +49-511-762-18068; fax: +49-511-762-5115. E-mail address: kayapinar@ifw.uni-hannover.de

Abstract

In this paper multi-scale modeling of a novel fluid dynamic planar positioning system is described and compared with a simplified plant model. The multi-scale model is realized by coupling a mechatronic simulation model implemented in Matlab/Simulink and a transient 2D-CFD model realized with the Finite Element-software Ansys using the Flotran solver. The complex behavior of the fluidic system between two control tasks could be observed. The permission for large movements of the slide is solved using an appropriate remeshing concept.

Keywords: Machine; Drive; Modeling

1. Introduction

Ablative and machining processes are the most flexible concerning material selection and geometric complexity of small workpieces with edge dimensions below 10 mm [1]. They play a significant role for the generation of microstructured surfaces and precision parts [2]. Considering energy consumption, space requirements and kinematics of commercial machines, a disproportion exists between machine characteristics and the small workpiece dimensions.

The priority program (SPP) 1476 *Small machine tools for small workpieces* (funded by the German Research Foundation, DFG) aims at finding novel machine concepts to solve these disproportions. At the Institute of Production Engineering and Machine Tools (IFW) a novel fluidic planar drive principle has been invented which allows a very compact assembly and integrates drive and guiding functionality. It mainly consists of only a frame and a slide (Fig.1). Thrust forces on the slide are generated by appropriately commutated fluid jets. Pressurized air streams out of the particular openings (*u, v, w*) and impinges the periodically arranged drive profiles. The positioning accuracy strongly depends on the measurement resolution and the weight of the slide. A stationary deviation of 72 nm could be measured with a first linear prototype (Fig. 2).

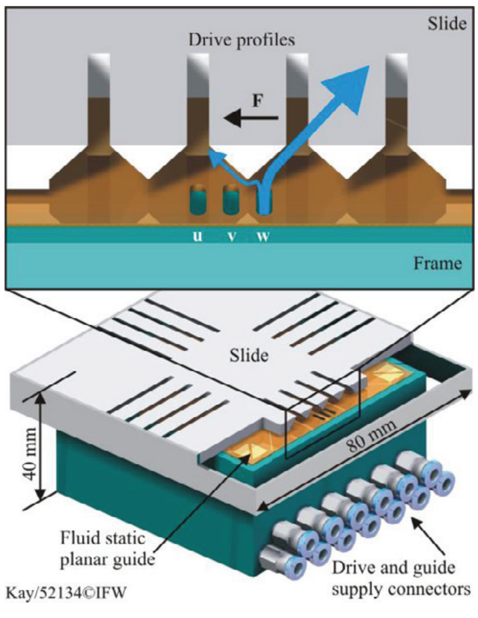

Fig. 1. Fluidic planar positioning stage for small machine tools

http://dx.doi.org/10.1016/j.procir.2012.05.047

$a_{x,n}$	Acceleration in x-direction at time $n \cdot T_{CFD}$
$a_{x,set}$	Calculated acceleration in x-direction
F_{ref}	Reference thrust force
k	Index-abbreviation for $k \cdot T_S$
K_d, K_p	Control parameters
m	Mass of the slide
n	Index-abbreviation for $n \cdot T_{CFD}$
$PD(z)$	Discretized PD-Control
p_{ref}	Reference pressure
p_{set}	Pressure value to be set on a nozzle
p_U,p_V,p_W	Pressure values at respective nozzles
s	Location of the profiles in x-direction
t	Simulation time
T_{CFD}	Time step of the transient CFD analysis
T_S	Sample time of the control task
$u_{x,n}$,$u_{y,n}$	Node displacement in x-/y-direction at $n \cdot T_{CFD}$
$v_{x,n}$,$v_{y,n}$	Fluid velocity in x-/y-direction at $n \cdot T_{CFD}$

During control design, the system has been modeled as a double integrator. The dynamics of the pneumatics (valves and connections) has been neglected as well as air resistance. The differences of the simulated and the measured positioning dynamics necessitated a more detailed model in order to better understand and improve the system.

Application of the Finite Element Method (FEM) is widespread especially in structural problems. Its particular application for plant modeling in closed-loop control analysis has been shown e.g. in [3] or [4]. In [4] the author implements the control law for active vibration control of a beam using the Ansys Parametric Design Language (APDL). The control task period equals the transient simulation time step. Further research works propose the coupling of general purpose 1D-process simulation and Computational Fluid Dynamics (CFD) simulation in order to simulate the control behavior of fluidic systems [5, 6, 7]. However, due to numerical problems occurring at high mesh deformations, the motions of these models are limited.

In this work a CFD model is proposed that combines the Arbitrary Lagrangian-Eulerian (ALE) formulation for small movements in small time steps with remeshing at larger time periods. This makes unlimited motion of a rigid body in a fluid domain possible. By coupling the multi-dimensional CFD model characterized by high time resolution with a 1D-control model with low time resolution, a spatial and temporal multi-scale model is obtained allowing for analyzing the overall transient behavior of the fluidic drive system. According to [8] this approach can be classified as a correlative multi-scale method since the systems behavior on the higher scale (control model) is derived by analyzing the mechanisms on the lower scale (CFD model). The CFD model is implemented in Ansys and solved with the Flotran solver. The control model is implemented in Matlab/Simulink. It takes the master role by calling the CFD model and handling the data transfer by means of a user implemented S-function block.

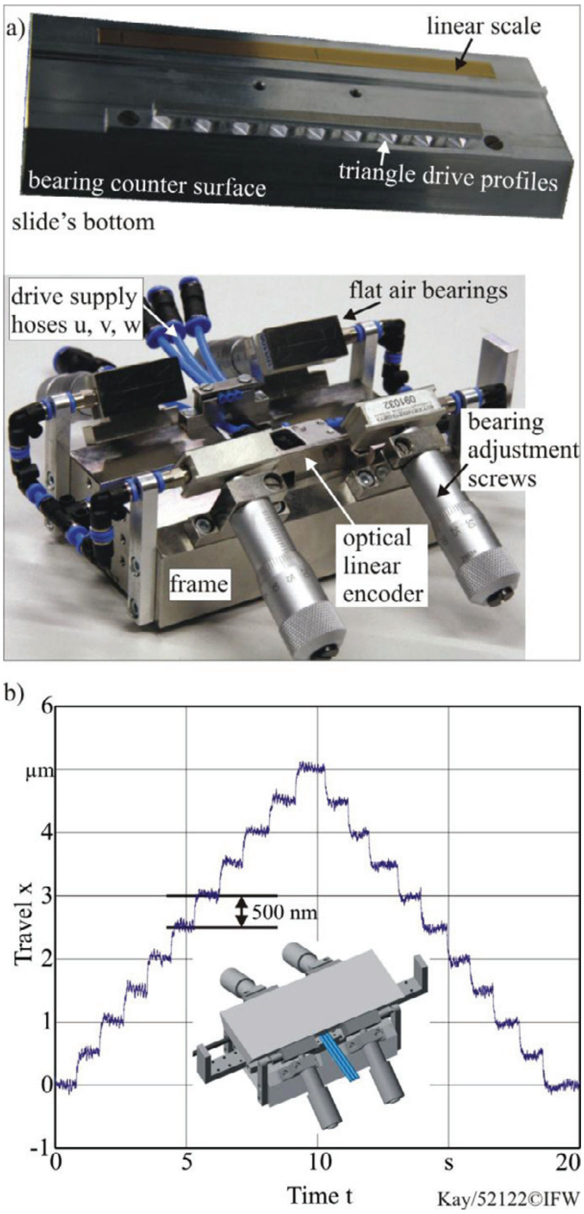

Fig. 2. (a) Linear prototype; (b) System response to 500 nm steps

2. Modeling of the Control Loop

The multi-scale modeling can uncover erroneous assumptions of a simplified model made during control design. Here, the calculated step response of a lumped model is compared with the detailed multi-scale model.

The simplified model supposes dynamic linearity of the plant. Possible static nonlinearities (e.g. of the valves or the pressure-force dependency) are supposed to be compensable by inverse modeling. Damping forces resulting from the fluid as well as time delays between the pressure set and the full force development are neglected. Finally, the resulting plant model equates a normalized double integrator (Fig. 3). A discretized PD-control has been chosen for the closed-loop positioning control since it fulfills the minimum demand of stability.

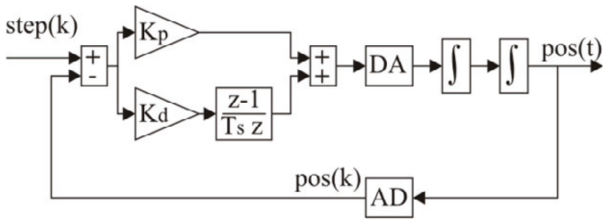

Fig. 3. Control loop simulation model (simplified linear plant)

In the scheme of the multi-scale model (Fig. 4) the linear plant model is replaced by a transient CFD model. The control parameters and sample time T_s are maintained. Fig. 5 shows a detailed coupling scheme.

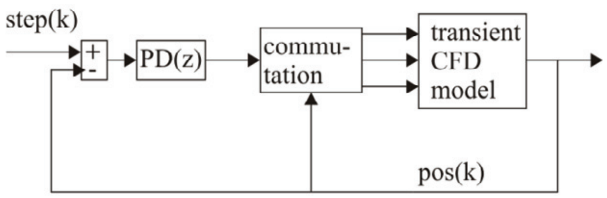

Fig. 4. Control loop simulation (coupled transient CFD plant model)

The workflow of the coupled analysis is controlled by the closed-loop position control model. After reading the preset initial motion state the control task is executed at a fixed sample time $T_S = 5\ ms$. In order to perform the computed acceleration a commutation has to be applied, that translates the acceleration to the three pressure values (p_U, p_V, p_W) depending on the actual position. The pressure values are written in a file that is read by the CFD analysis. The periodic call of the Ansys application in batch mode is implemented by a Matlab-S-Function using the CFD macro name as call parameter. The FEM-model is built based on the node displacements of the last run and the last results containing the velocity, pressure, turbulence, kinetic energy and dissipation fields. The sets of results are interpolated to the new mesh as initial conditions for the next solver step. Only the pressures at the nozzle inlets are overwritten with the new pressure values calculated at the control step before.

In one control task period T_S, the fluid flow is calculated with a much higher time resolution T_{CFD}. Thus, the quasi-continuous flow evaluation between two control samples enables a more realistic analysis.

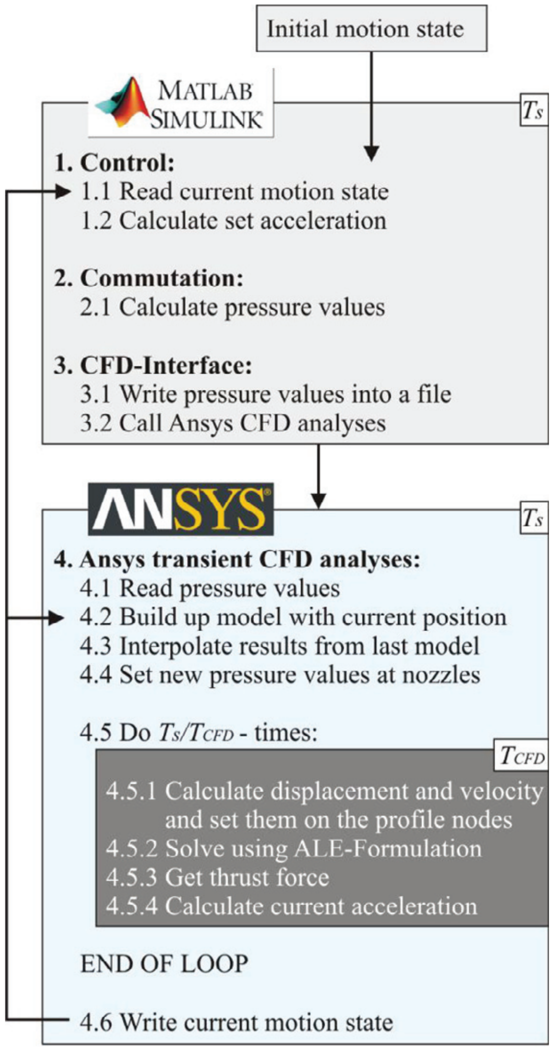

Fig. 5. Workflow of the coupled Control - CFD - simulation

The motion of the profiles (displacement of the respective finite-element nodes) within T_S is determined stepwise according to Newton's second law:

$$
\begin{aligned}
&u_{x,n}=0.5\cdot a_{x,n-1}\cdot T_{CFD}^{2}+v_{x,n-1}\cdot T_{CFD}+u_{x,n-1}\\
&v_{x,n}=a_{x,n-1}\cdot T_{CFD}+v_{x,n-1}\\
&\textit{Solve using ALE formulation}\\
&a_{x,n}=F_{x,n}/m
\end{aligned}
\tag{1}
$$

For later analysis, the motion states are written in a file at each CFD time step T_{CFD}. The motion feedback data for the next control task is written once after the last loop (every T_s time step).

2.1. Commutation

For appropriate commutation, the thrust force has been calculated by static CFD analyses. The position of the drive profiles has been varied pressurizing a single nozzle with 50 mbar and 25 mbar. Using the force characteristics (Fig. 6), a logic has been implemented that evaluates the desired force direction and the actual position in order to select the appropriate nozzle.

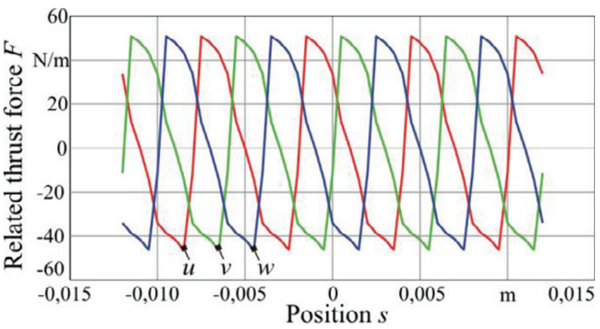

Fig. 6. Force characteristics (50 mbar) at the nozzles *u, v* and *w*

The pressure p_{set} is approximated by:

$$p_{set} \approx \sqrt{\frac{m \cdot a_{set}}{F_{ref}}} p_{ref} \tag{2}$$

This has been obtained by a coarse regression analysis using the force characteristics with a second reference pressure of 25 mbar.

2.2. CFD-FE model

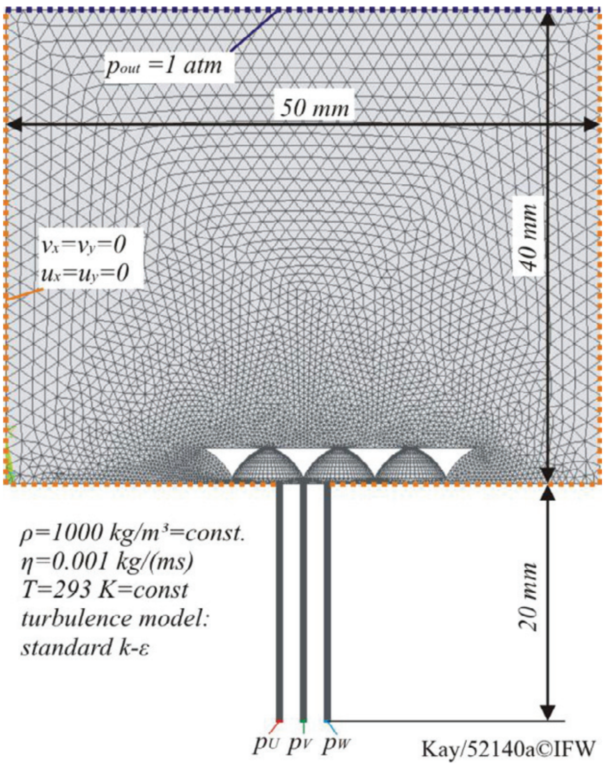

Fig. 7. FE-model of the control area and boundary conditions

The FE-model uses 2D-Flotran CFD elements (FLUID141). The fluid domain of the model consists of a rectangular control area and three nozzles. The drive profiles are subtracted from the fluid domain since they have to be treated as solid bodies. The control volume has to be chosen in such a way that the flow is fully developed at the boundaries. Design studies have shown that it can be kept compact if it is partially enclosed with wall boundaries since swirls are resolved and guided at near wall regions (Fig. 7). The fluid properties in a first step correspond to incompressible water at isothermal conditions at 20°C. The flow becomes turbulent exceeding a velocity of about 2 m/s. The standard κ-ε-model has been chosen to take turbulence into account.

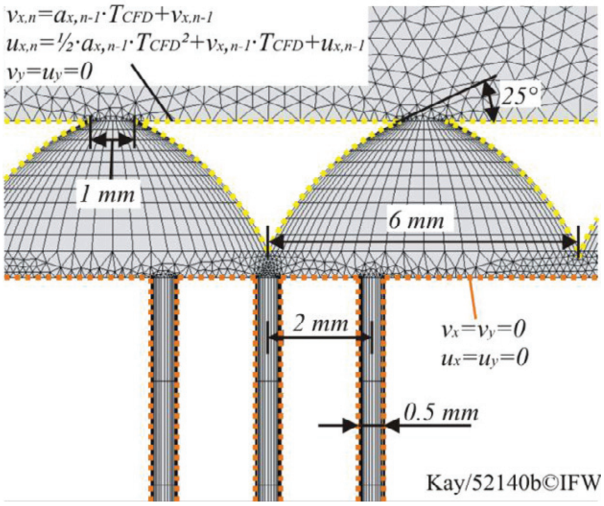

Fig. 8. FE-model and boundary conditions on the profiles

Ideally the simulation time step has to be chosen small enough to prevent the passage of a small fluid parcel through more than one element length in a single time step. For the presented model a time step of about 0.5 µs would have been appropriate. In order to reduce the overall simulation time, time resolution studies have been carried out. It has been found out that a choice of 100 µs is a reasonable compromise. The boundary conditions on the profiles correspond to Newton's second law. In order to obtain "no-slip"-conditions on the profile surfaces, it is very important that the velocity v_x equals the temporal derivative of the displacement u_x (Fig. 8). When moving the profiles the Arbitrary Langrangian Eulerian formulation uses the displacement boundary conditions between two time steps to update the FE-mesh by an elasticity based morphing algorithm.

2.3. Remeshing

If the movement of the profiles is too large, the elastic morphing algorithm produces extremely squeezed poor quality elements. For such cases, the Flotran solver provides an automatic remeshing option, which remeshes the fluid domain if the quality of the worst element falls below a user defined quality requirement at a user defined time step. But there are some restrictions of this option for the presented application.

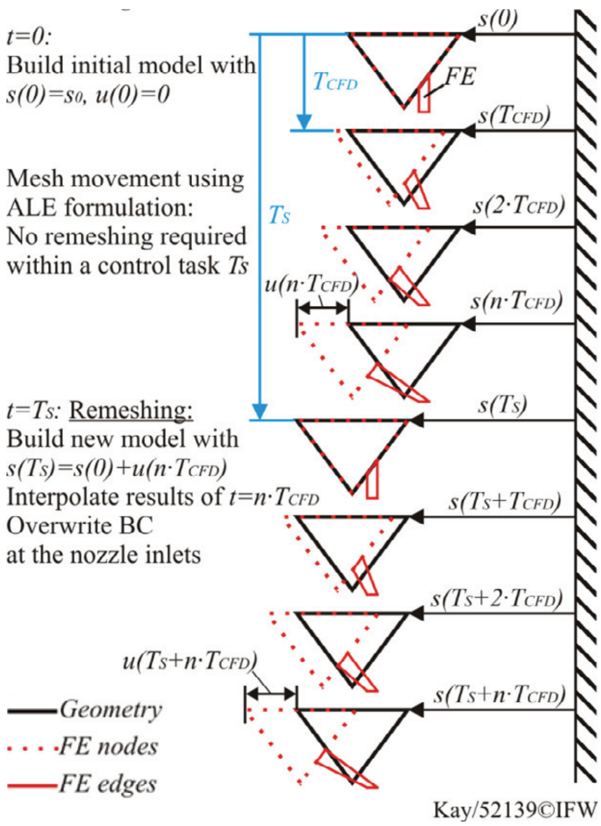

Fig. 9. Remeshing procedure

The analysis is called periodically by the control simulation. At the end of each CFD analysis the memory of the morphing and remeshing option is lost. Furthermore the automatic remeshing option only supports triangular (2D) or tetrahedral (3D) elements which would make modeling of a reasonable mesh with small elements at near wall regions and large elements at homogenous areas at the same time complicated.

In order to overcome these restrictions, a remeshing procedure has been implemented in APDL that builds up a new model every time step T_s and uses the morphing algorithm within one control task (Fig. 9). For this purpose the new geometry of the profiles is located at the node displacements of the last solved iteration step of the last control task. In order to save and not to overwrite result files from previous control tasks, the last results file, where the pressure and velocities etc. for each node are stored in, is copied first to a file with unique name. Afterwards the results (DOFs) of the last solution are interpolated to the new mesh as initial conditions for the next solver step. The inlet boundary conditions (respective pressures at the nozzles) are overwritten with the new ones. Due to the remeshing at every control task high velocities of the profiles are not allowed since this would lead to poor element qualities within one control task. In future works the remeshing will be activated if necessary.

3. Simulation Results

The coupled CFD simulation has been performed for a simulation time of *1 s* at a control task time of T_s *= 5 ms* and a time step size of the CFD solver of $T_{CFD} = 100\ \mu s$. The number of global iterations per time step has been chosen to 20. This resulted in 200,000 iterations. The number of elements varied as the model has been remeshed at each control task period but the average can be given as 8740. The parameters of the discretized PD-Control have been chosen to $K_p = 100$ and $K_d = 10$. The simulation took 6 hours and 38 minutes using four cores of Intel Core i7 at 2.79 GHz. The fluid velocity fields at $t = 16\ ms$ and at $t = 381\ ms$ can be seen in Fig. 10 and Fig. 11 respectively.

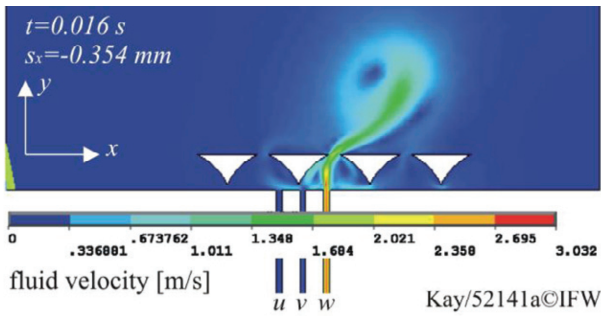

Fig. 10. Velocity field after 16 ms

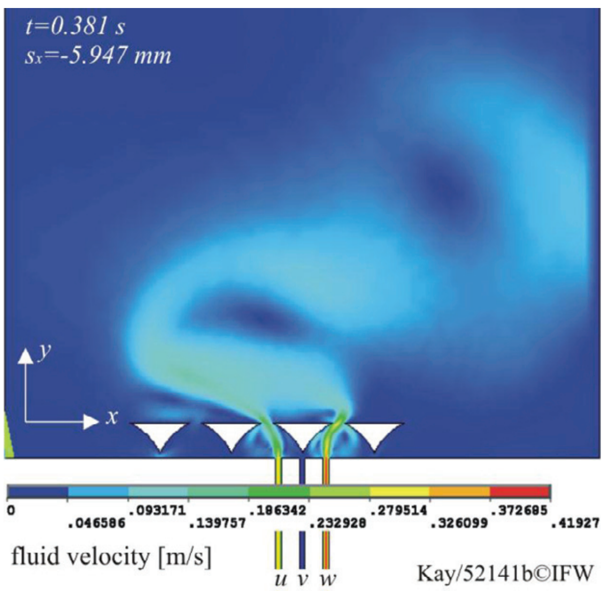

Fig. 11. Velocity field after 381 ms

The development of a swirl (Fig. 10) can still be seen in Fig. 11 on the right. The swirl on the left in Fig. 11 occurred when nozzle *w* tried to decelerate the slide by impinging the left flank of the third profile at $t = 60\ ms$. After reaching the set-point position of $s = -6\ mm$ the flows at the nozzles *u* und *w* are alternating at very low velocities until the end of the simulation.

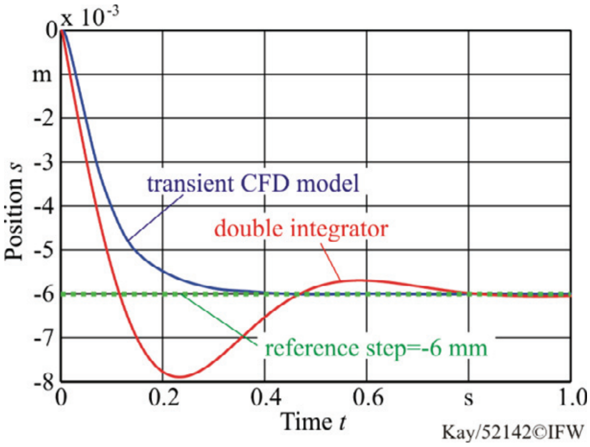

Fig. 12. Comparison of step responses

The step responses of the coupled and the simplified model for a reference step of -6 mm are compared in Fig. 12. A larger time delay and damping can be noticed in the reaction of the transient CFD-model. The reason for the larger time delay in the step response could be found in time delays between a pressure set command and the achieved thrust force (Fig. 13) since the set command of the control at the beginning of the simulation is the same for both models.

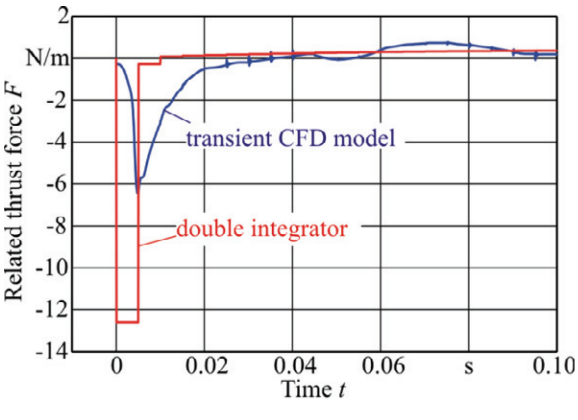

Fig. 13. Comparing set forces

A further reason could be a wrong calculation of the set pressure in formula 2 since only a second reference point had been used for the regression analysis between force and pressure. A fluidic damping force that has been neglected in the simplified model could be a reason for the larger time delay as well as for the missing overshoot of the step response.

4. Summary and Outlook

In this paper multi-scale modeling of a novel fluidic positioning device is described and compared with a simplified model. The multi-scale model is realized by coupling a process simulation and a transient CFD model. Choosing a higher temporal resolution in the CFD model the complex behavior of the system between two control tasks could be observed. Large movements of the drive could be allowed using a remeshing concept. Differences between the two models are caused by time delays between the pressure command and the generated thrust force, incorrect modeling of the static pressure-force dependency and fluidic damping forces.

In future works, the assumptions will be verified and the simplified model will be extended in order to obtain a sufficient low order model for the control design.

Acknowledgements

The authors thank the German Research Foundation (DFG) for funding this work within the SPP1476.

References

[1] Wulfsberg JP, Grimske S, Kohrs P, Kong N. *Kleine Werkzeugmaschinen für kleine Werkstücke*. wt Werkstatttechnik online, 11/12-2010: 886-891.
[2] Brinksmeier E et al. *Machining precision parts and microstructures*. Proceedings of the 10th ICPE; 2001: p.1-9
[3] Roeck S, Pritschow G. Real-time capable Finite Element Models with closed-loop control: a method for Hardware-in-the-Loop simulation of flexible systems. *Production Engineering*, Vol 1, Issue 1, Springer Berlin / Heidelberg; 2007, p. 37-43
[4] Takacs G. Experimental Verification of the Closed-Loop Response of an Active Vibration Control System Modeled in ANSYS. *Modelling of Mechanical and Mechatronic systems (MaMS) 2011*, Herlany, Conference Proc.; 2011: p. 478-486
[5] Xie H, Zhang D, Shen L. Collaborative simulation based on MATLAB/Simulink and FLUENT. *Journal of System Simulation*, 2007, 19(8): 1824-1827
[6] Yang C, Gou X-I, Tang S-L. Multi-scale Modeling of Energy System via the Means of Coupling Process and CFD Models. *International Conference on Power Engineering-2007*, Hangzhou, China , Conference Proceedings; 2007: p. 1380-1384
[7] Zhang X, Wang S, You W. 1D and CFD Co-simulation Approach Basing on General Purpose Simulation Software. *International Conference on Computational Intelligence and Software Engineering (CiSE)*, Wuhan, China, Conference Proceedings; 2009: p. 1233-1237
[8] Ge W, Wang L, Wang W et al. Multi-Scale Modeling. *Chemical Engineering and Processing Technology*. Vol. V. Chap. 7. Editors: Pohorecki R, Bridgewater J, Molzahn M, Gani R.