FE-simulation of the Presta joining process for assembled camshafts – modelling of the joining process

R Scherzer, C B Silbermann, R Landgraf and J Ihlemann
Professorship of Solid Mechanics, Chemnitz University of Technology, Reichenhainer Straße 70, 09126 Chemnitz, Germany
robert.scherzer@mb.tu-chemnitz.de

Abstract. The Presta joining process is based on the separate manufacturing of internal profiled cams and shafts with previously widened diameters at the intended cam seat through rolling. With regard to further optimization and enhancement, the two substeps are modelled and simulated using FEM. The results of the rolling process simulation are employed as the starting point for the simulation of the joining step, which is presented in this contribution. First, the shaft is widened with a profile oriented orthogonal to the shaft axis at the intended cam seat. Subsequently, the shaft is joined with a cam that has an internal profile oriented parallel to the shaft axis. Due to plastic deformations these perpendicular profiles form a tight fit. Because of these complex plastic deformations, that are related to large local forming degrees, the use of precise material models is essential. Moreover, it is necessary to maintain the deformation history of the rolling simulation in the joining step with regard to effects like kinematic or isotropic hardening. For that purpose, the applied material model’s invariance under change of the reference configuration is used by transferring its internal state variables from one simulation step to another. Finally, the results of the simulation are compared to measurements of series production.

1. Introduction

Over the last decades, the use of numerical methods, especially the Finite Element Method (FEM), increased rapidly. The computer simulations basing on these methods allow to save costs by reducing the number of expensive prototypes in the development of new designs and technologies. However, to get reliable simulation results, it is absolutely necessary to understand the relevant processes and the behaviour of the employed materials. Moreover, it is required to apply realistic boundary conditions to the models as well as to simplify them as much as necessary to reduce computing time. Hence, the Presta joining process will first be explained in detail to make the boundary conditions of the simulation model introduced later in this work comprehensible.

The Presta joining process is a manufacturing method to build shaft-hub connections and is used primarily as the world’s leading process in the series production of assembled camshafts. It allows up to 30% reduced weight compared to conventional cast or forged camshafts. Consequently, there is a significant inertia reduction of the rotating camshafts [1, 2].

The manufacturing of Presta camshafts is divided into two substeps. First, the shaft is locally widened by rolling at the position of the intended cam seat. To achieve this widening the rollers have an approximately sinusoidal cross-sectional surface. Hence, due to the rolling of the shaft a circumferentially oriented profile is formed (figure 1a).
Subsequent to the rolling step, the shaft and the cam are joined, which results in a combination of press fit and form fit. This is achieved by the use of cams with an inner profile oriented parallel to the shaft axis. Consequently, the perpendicular profiles of the shaft and the cam generate a tight fit due to plastic deformations (figure 1b). The inner diameter of the cam is smaller than the outer diameter of the widened cam seat. Thus, during the joining process the rolled profile is squeezed by the cam (figure 2).

The simulation of the rolling step of the Presta process has already been presented and discussed before [3]. Within this rolling model two symmetries were used. First, the cam seat has been modelled symmetrically in axial direction such that the model size could be halved. Secondly, a cyclic symmetry due to the arrangement of rollers around the shaft was employed, which reduced the model size to a section of 120 degrees (figure 3).
The simulation model of the second substep of the Presta joining process simulation is shown in Section 2. At this point, the final state of the rolling simulation represents the initial state of the joining simulation. Thus, despite having new symmetries, load boundary conditions and a new FE mesh, the history of deformations remains in the model. This transfer of the deformation history implies a change of the reference configuration in the employed material model.

In both substeps of the simulation of the Presta joining process, a phenomenological model of finite strain viscoplasticity introduced by Shutov and Kreißig is applied [3, 4]. First, this material formulation is capable of representing the realistic material behaviour even for large deformations. Secondly, this model exactly meets the requirements of multi-stage deformations like the Presta joining process because of its invariance after a change of the reference configuration [5]. Section 3 shows the transfer of the deformation history.

Finally, in Section 4 the results of the forming simulation will be discussed and compared to the experience of the series production.

2. Three-dimensional simulation model of the joining step

2.1 Geometric model

In the second step of the Presta joining process, the cam is forced onto the previously rolled cam seat of the shaft. In analogy to the rolling step the joining process has a very small deformation zone compared to the size of the single parts used in there. Hence, to reduce the model size it is beneficial to utilise symmetries in the assembly of this joining step.

The simulation model of the rolling process included the arrangement of rollers around the shaft to reduce the model to a section of 120 degrees. Moreover, it has been modelled as cyclically symmetric. With regard to its resulting deformation, the joining process model is not axisymmetric (figure 1b). However, the resulting profile of the rolling step has been identified as approximately axisymmetric [3]. Hence, to use the rolled profile as the initial profile of the shaft for the joining model, it is necessary to expand the profile of the shaft in the axial direction. Here, it is sufficient to use one cutting plane to represent the complete shaft. Subsequently, this plane can be expanded to a three-dimensional model.

The size of the shaft sections arc in the joining step depends on the simplifications used in the model. Here, two possible options may be applied: The first one is to keep the original outer contour that controls the kinematics of the valve train in combustion engines (figure 4a). In this model, only one symmetry plane can be employed, which leads to a model that includes a section of 180 degree with regard to the shaft and, consequently, to high computational effort. Another option is to model the outer contour of the shaft as a circular shape (figure 4b).

Figure 4. The cam with its inner profile shown in (a) with the original outer contour and in (b) with the simplified circular shape. The dashed lines show the respectively minimal section needed for the simulation model.
The advantage of this approach is that a cyclic symmetry depending on the number of teeth of the inner profile can be applied. The model discussed in this work contains 36 teeth and, consequently, the assembly of shaft and cam expands over a section of 10 degrees (figure 5a).

As a result there are three boundary conditions in the joining simulation. First, cyclic symmetry has to be defined at the side surfaces of the model. Furthermore, to force the cam onto the rolled profile there has to be a displacement boundary condition at the cam. In addition, axial displacement of the shaft is set to zero to prevent rigid body motions (figure 5b).

2.2 FE-mesh of the shaft

Due to the change of geometric models between the two simulation steps, a new mesh is required for the simulation of the joining step. Thus, the new mesh can be adjusted to the requirements of the expected deformations in this second model. This in particular applies to the tangential mesh density. The resulting profile of the rolling step is nearly axisymmetric with small circumferential gradients in the displacements and stresses of the rolled profile [3]. Consequently, its tangential mesh is coarse compared to the cross-sectional mesh.

Due to the inner profile of the cam, the deformation of the rolled profile after the joining process cannot be expected as axisymmetric anymore. The teeth’s edges cause high circumferential gradients in displacements and in stresses. Thus, in this region a finer mesh is necessary. Furthermore, the interpolation error between the old and the new mesh can be minimized by reusing the cross-sectional mesh of the rolling step (figure 6).

Likewise, it is possible to generate a new mesh to the shaft and to interpolate the internal state variables at the new integration points. However, a new cross-sectional mesh leads to higher
interpolation errors regarding internal state variables. For this purpose, the following section shows the general procedure of transferring the deformation history between the several simulation models.

3. Transfer of deformation history

As already mentioned before, the Presta joining process is a multi-stage manufacturing method consisting of rolling and joining. Within the simulation of both steps, different geometric models with different boundary conditions are required. For the change between the FE-models, the deformed cross-sectional geometry of the rolling process is adopted within the joining step. However, the switch from a 120 degree section to 10 degree results in a new mesh for the second model. Consequently, the internal state variables of the material model would be set to zero by default and there is no deformation history stored at the integration points of the FE-model of the joining step. However, for the applied phenomenological model of finite strain viscoplasticity, a suitable approach to transfer the internal state variables to a new model has been introduced by Shutov et al. [5].

The transfer of internal state variables between the two simulation models has been implemented as follows. First, the integration point coordinates of the joining step simulation model are exported to a data file. Next, these coordinates are rotationally transformed into one plane. Subsequently, the resulting point coordinates are employed to define a path within the final FE-mesh of the rolling step (figure 7).

![Figure 7](image)

Figure 7. (a) The position of the path’s plane in the FE-mesh of the rolling step and (b) the data points of the path connected with lines.

At the data points of the path, the required internal state variables are interpolated by the employed FE-software and passed to a second data file. Hence, in this second file every integration point of the joining step has an associated set of internal state variables representing the deformation history of the rolling step. Finally, at the start of the joining simulation, the state variables are imported and initialized at every integration point.

The applied material model includes the scalar inelastic arc length that can be seen as the degree of deformation since it increases monotonically with plastic deformation [6]. In the rolling step this inelastic arc length reaches its maximum in the grooves of the rolled profile (figure 8).

![Figure 8](image)

Figure 8. Contour plots of the inelastic arc length at the end of the rolling simulation with its maxima in the grooves of the rolled profile. In addition, the coarse mesh in tangential direction is shown.
A comparison between the plots of the inelastic arc length at the end of the rolling step and at the beginning of the joining step shows almost no differences (figure 9). First, this is due to the identical cross-sectional mesh in both simulations. Secondly, this can be attributed to very small gradients of the physical values in tangential direction.

Figure 9. Contour plots of the inelastic arc length at the beginning of the joining simulation with the almost exact reproduction of the contour at the end of the rolling step. Here, the tangential mesh density with its refinement for the specific requirements of the joining simulation is shown.

4. Simulation results and discussion

Once the internal state variables are initialized in the simulation of the joining step, the transfer of the deformation history of the rolling process is completed. Hence, the effects of local isotropic and kinematic hardening due to the deformations of the rolling are still included in the model (figure 10).

Figure 10. Contour plot of the inelastic arc length after the joining simulation with the cam hidden to get a clear view of the resulting profile.

Furthermore, the final inelastic arc length shows that the rolling of the shaft causes more plastic deformation than the joining step. The degree of deformation increases significantly only at regions, where the inner edges of the cam cut into the rolled profile.

To compare the results of the joining simulation with the series production of Presta camshafts, it is appropriate to examine the joining force. The joining force can be measured easily in the production process. Mathematically, it results from the cam displacement boundary condition as the reaction force. The comparison between experimental and computed data shows suitable agreement (figure 11).
Both curves of the joining force show similar absolute values. Moreover, nearly constant distances between local maxima and their adjacent minima can be observed. This can be explained by the fact that the rolling profile in the simulation is perfectly symmetric. Manufactured rolling profiles are typically a little bit lopsided and, consequently, some of the rolled grooves are deeper and some of them are rather flat. Hence, the required force to push the cam over the bulges varies with their height.

The general accordance between the measured and computed joining forces can be seen as an indicator for a realistic simulation of the joining process. Nonetheless, more experiments are needed to prove the model’s ability to provide reliable predictions for different materials or geometries. For example, to compare the results of the FE-model with manufacturing a possible test would be to cut through a completed camshaft and to measure the deformed profile (cf. figure 2).

5. Conclusions
This work presents a simulation model for the joining step of the Presta process. The model refers to the simulation model of the rolling process introduced by Scherzer et al. [3]. Hence, like in the rolling simulation the parameters and geometries used in the model are taken from the series production of Presta camshafts.

First the geometry, the boundary conditions and the employed symmetries of the model have been explained in detail. Here, the model could be reduced to a section of 10 degree due to the cyclic symmetry based on the simplification to a circular outer profile of the cam. Furthermore, the mesh of the joining model has reused the existing cross-sectional mesh of the rolling simulation to minimize interpolation errors.

The simulation models representing the Presta joining process have been realized using different geometries. Such a change of models is generally associated with a loss of deformation history. However, in multi-stage forming processes like the Presta process the deformation in the first substep is significant. Therefore, in this work an approach to transfer the deformation history has been applied using the invariance under change of the reference configuration of the applied phenomenological model of finite strain viscoplasticity. This is done by transferring the internal state variables of the rolling simulation to the joining model and thus to the new reference configuration.

Ultimately, this work gives an overview of the results of the joining simulation. First, the quality of the transfer of internal state variables is proved by a comparison between the inelastic arc length at the end of the first step and at the beginning of the second step. Based on this, the further increase of the degree of deformation due to the joining process is shown. This reveals the major plastic deformation occurring in the rolling step and shows the importance of maintaining the deformation history in the
simulation of the Presta joining process. Most importantly, the comparison between the joining force curves of the series production and of the simulation exhibits suitable accordance.

Perspectively, the two simulation models of the rolling step and the joining step will be extended to a third substep: loading the connection. Here, the Presta camshaft will be loaded similar to the torque in combustion engines. Then, it will be tested how much load the connection endures (figure 12).

Figure 12. Loading of the camshaft representing the load of the valve train.

Acknowledgements
The research shown in this work originates from the collaborative research center SFB 692 “High-strength aluminum based lightweight materials for safety components”, which is supported by German Research Foundation (DFG). Furthermore, this research was carried out in close cooperation with ThyssenKrupp Presta GmbH in Chemnitz.

References
[1] Lengwiler A 2011 Fehlerfortpflanzung, Simulation und Optimierung von Prozessketten anhand der gebauten Nockenwelle (Eidgenössische Technische Hochschule ETH Zürich)
[2] Meusburger P 2006 Lightweight design in engine construction by use of assembled camshafts MTZ worldwide 67.3 p 10–2
[3] Scherzer R, Silbermann C B and Ihlemann J 2016 FE-simulation of the Presta joining process for assembled camshafts – local widening of shafts through rolling IOP Conf. Series: Materials Science and Engineering 118 p 012039
[4] Shutov A V and Kreißig R 2008 Finite strain viscoplasticity with nonlinear kinematic hardening: Phenomenological modeling and time integration Comp. Methods Appl. Mech. Eng. 197.21 p 2015–29
[5] Shutov A V, Pfeiffer S and Ihlemann J 2012 On the simulation of multi-stage forming processes: invariance under change of the reference configuration Materialwiss. Werkstofftech. 43.7 p 617-25
[6] Shutov A V, Kuprin C, Ihlemann J, Wagner M F X and Silbermann C 2010 Experimental investigation and numerical simulation of the incremental deformation of a 42CrMo4 steel Materialwiss. Werkstofftech. 41.9 p 765–75