SIMULATION OF THE NOTCHING EFFECT OF PORES IN 3D PRINTED COMPONENTS

Karl-Michael Nigge, Johannes Fieres, Christof Reinhart, Philipp Schumann[1]
Volume Graphics GmbH
Speyerer Straße 4-6
Heidelberg, Germany, D-69115
([1] Concept Laser GmbH, Lichtenfels, Germany)

ABSTRACT

Classical FEM simulations may not always be well suited for micromechanical simulations of 3D printed components with defects because they require the generation of geometry conforming meshes which must be fine enough to capture all relevant geometric details and coarse enough to keep the computational effort at a practical level on the other hand. Recently, immersed-boundary finite element methods have been used to overcome this meshing problem. In order to validate this simulation approach, a comparison between experimental and simulated results of tensile tests was conducted for two types of 3D printed components, showing a good agreement. The approach was also validated successfully against a classical FEM simulation for a solid cube and a cubic lattice. The simulation approach can be used to assess the effects of defects such as porosity in additively manufactured components.

1. INTRODUCTION

Lightweight design may lead to complex shapes of components resulting from bionic optimization which are increasingly being produced by 3D printing. Their mechanical properties may be particularly sensitive to defects such as porosity caused by the printing process. While porosity may be reduced or shifted to less critical zones, it may not be cost efficient to completely avoid it. Much rather, it would be desirable to include a quantification of the detrimental effect or otherwise of pores in production parameter optimizations.

As a consequence, there is an increased need for micromechanics simulations to determine the effective mechanical properties of complex materials and to assess the mechanical strength of components with optimized shapes and internal defects.

Simulating such complex structures with classical finite elements methods (FEM) is often difficult. FEM requires the creation of a geometry-conforming mesh. For highly structured materials this constitutes a major challenge. First, the scale of the relevant (micro-)structures is typically orders of magnitude smaller than the scale of the entire object of interest. Thus, very fine meshes with a huge number of mesh cells are needed, increasing the computational effort. Second, the structures typically exhibit organic or irregular geometries which make it hard to create numerically well-conditioned meshes. In fact, the meshing process is often the limiting factor in the simulation process, sometimes even rendering any simulation attempt impractical.

2. SIMULATION METHOD

2.1 Immersed Boundary Method

Recently, immersed-boundary finite element methods have been proposed to overcome the meshing problem (see, e.g., [1]). Such methods do not require the generation of a boundary-conforming mesh and are thus well suited for simulating complex geometries. The simulated domain is immersed (embedded) into an extended, much simpler, domain which can be discretized in a trivial way, e.g., with straight hexahedral elements. Using a custom integration scheme, the original object shape is still simulated accurately (see Figure. 1).

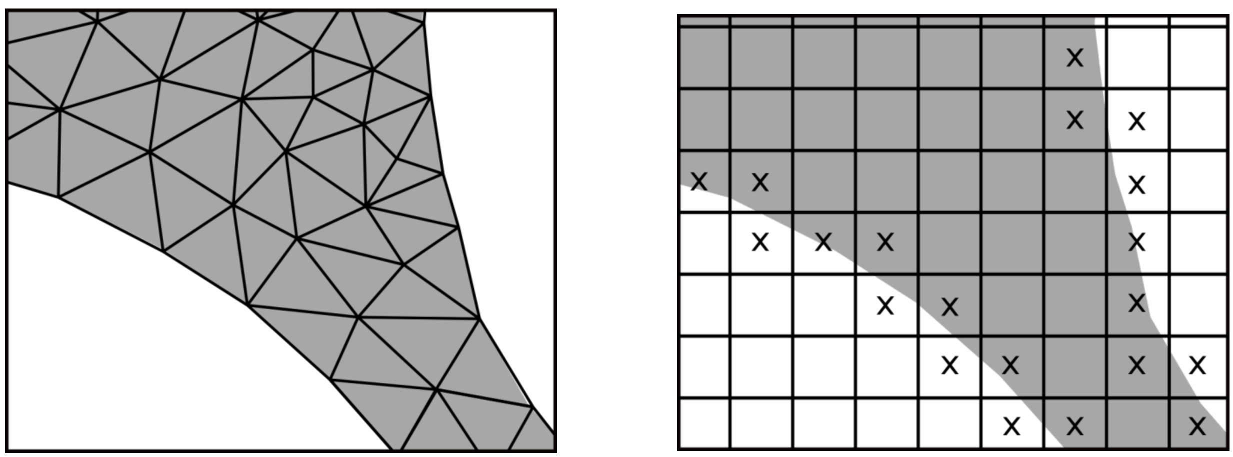

Figure 1. The drawing shows a detail of a larger simulated object (shaded area). Left: Classical FEM approach. The object is represented by a geometry-conforming mesh (black lines). Right: Immersed boundary method. The domain is extended beyond the object borders. The extended domain is discretized by a trivial quadrilateral mesh (black lines). The mesh cells cutting through the object boundary (marked with crosses) are treated specifically during integration, only considering the part interior to the simulated object.

2.2 Accurate Representation of Complex Microstructures with CT scans

In order to get hold of a realistic geometric model of the micro-structures in the first place, either modeled geometry descriptions or empirical 3D imaging can be used. For materials manufactured in processes with pronounced random elements (like foams or compounds), realistic geometry models are not easily established. Process variations in the manufacturing of designed structures like 3D-printed lattices or entire components will exhibit shape deviations and discontinuities like porosity which are difficult to track on a theoretical level. In these cases, realistic models can be obtained from physically existing parts. Here, computed tomography (CT) plays an increasing role as a volumetric imaging technique. Using well established image processing methods, the external and internal surfaces of such complex structures can be determined with sub-voxel accuracy from the CT scans [2] (Figure 2).

2.3 Implementation

The immersed boundary simulation presented here is implemented in the Structural Mechanics Simulation module of VGSTUDIO MAX by Volume Graphics. It allows to simulate stress and displacement fields resulting from static loads on materials or components with arbitrarily complex structures as represented by CT scans. It works evenly well with explicit models like surface meshes or CAD data. Stress distributions in nominal components (represented by CAD models) and in actual specimen with shape deviations and defects (represented by CT scans) can be visually and numerically compared, allowing for a quantification of the stress differences introduced by defects such as porosity.

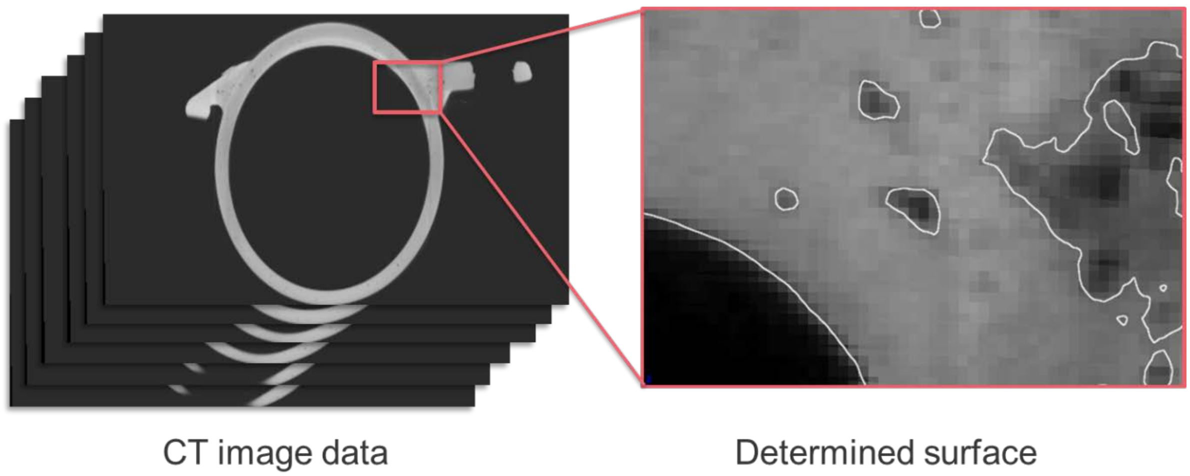

Figure 2: Example of a CT scan of a component with pores. Left: Stack of 2D slice views. Right: External and internal surfaces determined with sub-voxel accuracy

2.4 Stress Concentration Around Pores

Figure 3 shows an exemplary simulated stress distribution in a 3D printed, bionically optimized aeronautic bracket, clearly showing a pronounced stress concentration around a pore close to the surface.

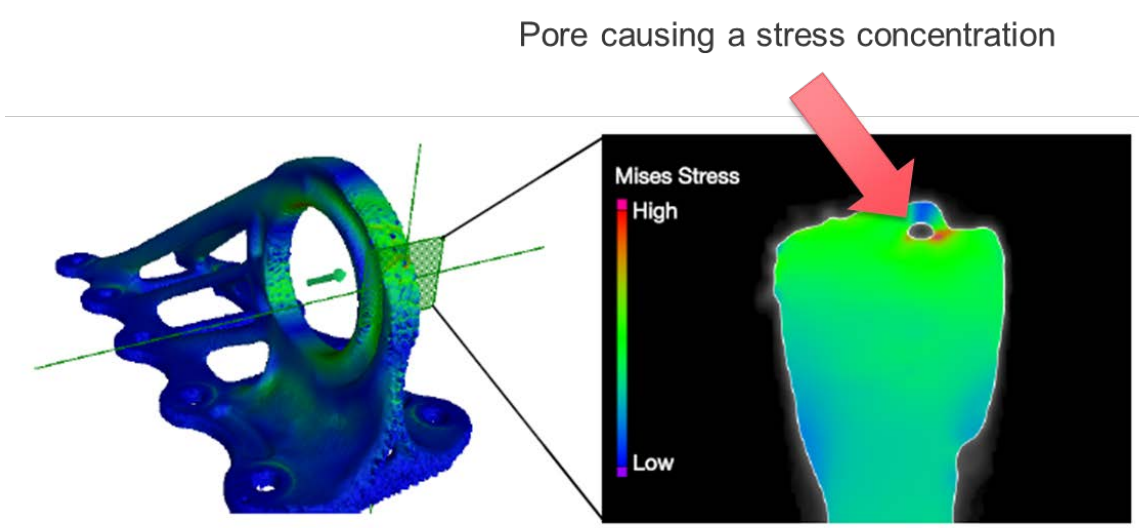

Figure 3: Simulated stress distribution in a 3D printed component under a static load, showing a pronounced stress concentration around a pore close to the surface

3. VALIDATION AGAINST EXPERIMENTS

As a validation of this simulation approach, a comparison between simulated and experimental results of tensile tests was conducted for two types of additively manufactured AlSi10Mg components, a tension rod and a bionically optimized aeronautic structural bracket. 18 samples of each were 3D printed with 6 different intentionally introduced porosity patterns (volume fractions, shapes and spatial distributions). The details of the sample design and manufacturing process, the CT scanning and the tensile test procedure are described in [3].

3.1 Tensile Strength

The von Mises stresses across the entire component are obtained as a result of the simulations, from which “hot spots” of the stress field, i.e. the N locations with the highest stress values HS_n (n=1...N) can be derived. The inverse of their average $<HS>_N$ were used as an indicator for the simulated tensile strength of the components. A comparison of $1/<HS>_N$ with the experimentally observed tensile strengths showed a very good correlation of 0.96 for the tensile rods and a good correlation of 0.86 for the aeronautic brackets (the latter when ignoring 3 samples with a notch-type porosity pattern which strongly differed from the other 15 samples) (Figure 4):

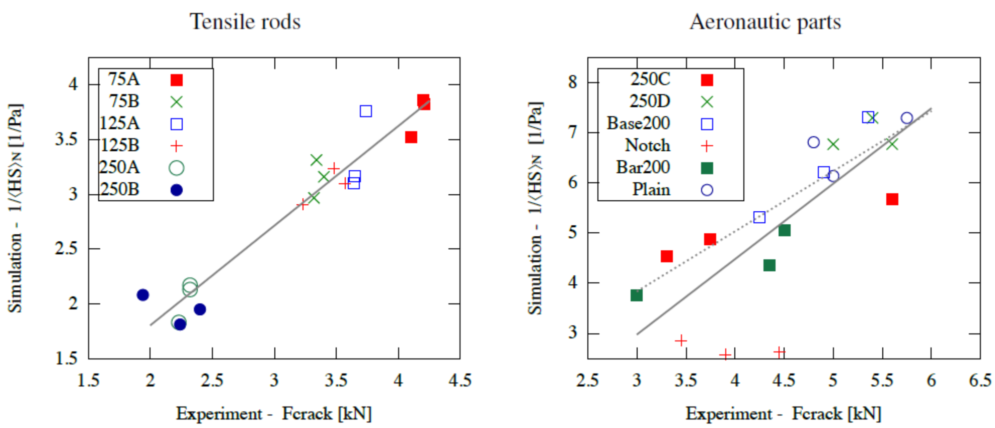

Figure 4. Measured force at first crack (F_{crack}) versus prediction from simulation ($1/<HS>_N$). Each data point represents one tested specimen. Left—tensile rods; right—aeronautic parts. The straight line is a least-squares fit to the data. Dotted line: least-squares fit without the notch-type porosity pattern [3]

3.2 Crack Locations

In 12 out of 18 specimen, a crack did actually occur at the first or second most-likely crack location (stress hotspot) as predicted by the simulation. In 3 specimen, cracks did occur within the top-ten most-likely predicted locations. Only for the remaining 3 specimen, there was no coincidence between any predicted and real crack location (Figure 5)

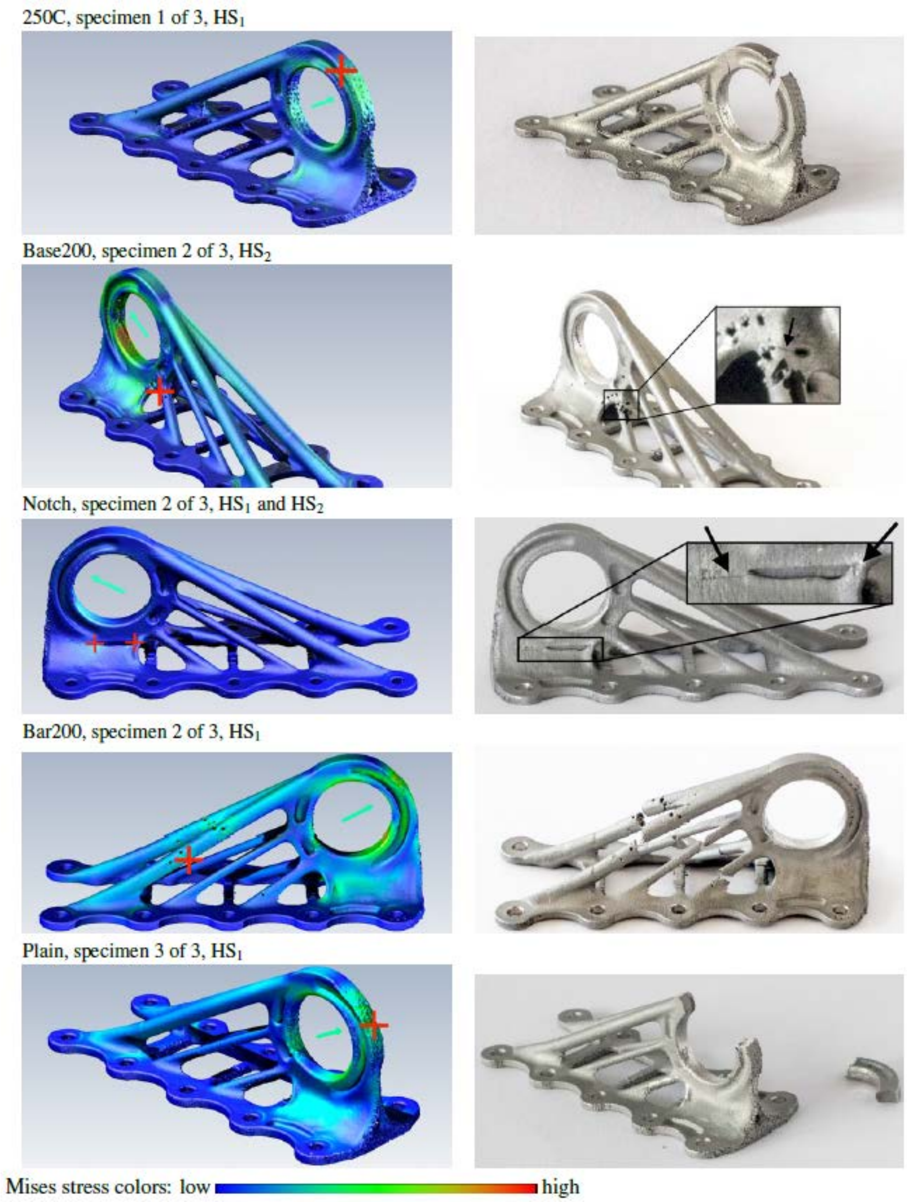

Figure 5. Predicted versus real locations of fracture for five examples. Left column—Digital reconstruction from CT-scan of intact part. Simulated von Mises stress shown as color overlay. Predicted fracture location indicated by red cross. Rows 1, 4, and 5: cross=global von Mises maximum (HS_1); row 2: cross=second-largest local von Mises maximum (HS_2); row 3: crosses=global and second-largest von Mises maxima (HS_1, HS_2). Right column—photographs of the parts after tensile tests. [3]

4. VALIDATION AGAINST CLASSICAL FEM SIMULATION

In [5], the results of the structural mechanics simulation in VGSTUDIO MAX are compared with those of a classical FEM simulation for a solid cube and a cubic lattice made from Ti6Al4V. In this case, the VGSTUDIO MAX simulation was carried out on an .stl mesh of the structures, and Autodesk Fusion 360 with a Nastran solver was used for the FEM simulation. For the solid cube, the maximum displacement and the effective Young's modulus calculated with the VGSTUDIO MAX simulation differs by only +0.7% from the results of the Nastran solver. In the case of the lattice structure, the maximum local von Mises stress and the effective Young's modulus are 2.5% higher than those obtained with the FEM solver. Given the inevitable variance of the setup of any simulation model and its influence on the results, this indicates a good agreement between the results of the VGSTUDIO MAX simulation and those of a classical FEM solver.

5. CONCLUSIONS

The simulation approach presented here can be used in both R&D and quality assurance of 3D printed components to determine the influence of defects or shape deviations on the mechanical stability. This can be done by simulating the internal stress distributions for both a CAD model of the ideal component and CT scans of prototypes or manufactured parts and comparing their respective hotspots. In such comparisons, the tolerancing criterion for the actual components is that defects or shape deviations must not lead to local stress peaks which are significantly higher than those found in the ideal component. The combination of widely used CT based metrology and defect analysis with this simulation approach for the assessment of potential effects of defects opens up the perspective of using computed tomography as a comprehensive method for the quality assurance of 3D printed parts.

6. REFERENCES

This paper is a modified and extended version of [3].

1. Schillinger, D., Ruess, M. "The Finite Cell Method: A Review in the Context of Higher-Order Structural Analysis of CAD and Image-Based Geometric Models." *Arch. Comp. Meth.Engin.* 22(3) (2015), pp. 391–455.

2. Heinzl, C., Amirkhanov, A., Kastner, J. "Processing, Analysis and Visualization of CT Data." *Industrial X-Ray Computed Tomography* Chapter 4 (pp. 99-142). Eds. Carmignato, S., Dewulf, W., Leach, R. Cham: Springer International Publishing AG, 2018

3. Fieres, J., Schumann, P., Reinhart, C. "Predicting failure in additively manufactured parts using X-ray computed tomography and simulation." *Procedia Engineering* 2013 (2018) 69-78

4. Nigge, K.-M., Fieres, J., Schumann, P., Reinhart, C. "Virtual Stress Tests Directly on CT Scans." *15th Asia Pacific Conference for Non-Destructive Testing (APCNDT)*. Singapore, 13-17 November 2017, Contribution ID 51

5. Du Plessis, A., Yadroitsava I., Yadroitsev, I., Le Roux, S.G. "Selection of lattice design for medical implants by additive manufacturing." *ASME J. Mech. Design* (2018), submitted.

7. ACKNOWLEDGEMENTS

We thank Airbus Emerging Technologies & Concepts for the permission to use the bionically optimized bracket model. We thank diondo X-ray Systems and Services, Hattingen, Germany, for performing the CT scanning.