Creep behaviour of Ti-6Al-4V produced by SLM

Amandine Cardon[1,2, a)] Charles Mareau[2, b)] Yessine Ayed[2, c)] Sjoerd Van Der Veen[1,d)] Philippe Dal Santo[2, e)]

[1]*Airbus Operations SAS, 316 route de Bayonne, Toulouse, France*
[2]*Laboratoire Angevin de Mécanique, Procédés et innovAtion, 2 Boulevard du Ronceray, Angers, France*

[a)]*amandine.cardon@airbus.com*
[b)]*charles.mareau@ensam.eu*
[c)]*yessine.ayed@ensam.eu*
[d)]*sjoerd.van-der-veen@airbus.com*
[e)]*philippe.dalsanto@ensam.eu*

Abstract. Additive manufacturing processes allow developing complex optimized geometries, hence making them very attractive for aeronautical applications. In recent years, the possibility of fabricating secondary structural parts with additive manufacturing techniques has been explored by Airbus. Because of the thermal gradients caused by local heating, these techniques also have some drawbacks such as residual stresses and distortions. To solve these issues, post treatments, like stress relieving and Hot Isostatic Pressing (HIP) treatments, are usually applied. The present work deals with the development of a numerical tool for the prediction of the final state of Ti-6Al-4V industrial components produced by Selective Laser Melting (SLM). This study focuses on the simulation of the stress relaxation occurring during heat treatments. The objective is to analyse the creep phenomenon of the Ti-6Al-4V alloy at high temperatures (between 500°C and 900°C). Creep tests are performed under a constant stress (tensile and compression), using a Gleeble© machine. Specimens initially exhibit a fully martensitic microstructure resulting from the SLM process. Observations show that the microstructure evolves according to the temperature and influences the creep strain rate. A constitutive equation, based on the Arrhenius law, is thus proposed to describe the material behaviour during stress relief heat treatment and HIP. The model parameters are fitted using a least squares method showing a good agreement with experimental data. This phenomenological model is implemented in ABAQUS with a user subroutine. Numerical simulations are performed on simple test cases inspired from literature. The simulation results correspond well to the experimental data. In a near future, the model will be used to simulate complex shaped parts.