FE-Simulation of the Tension Levelling Process with Abaqus Explicit

Lorenz Steinwender[1], Alexander Kainz[1], Konrad Krimpelstätter[2], Klaus Zeman[1]

[1] ***Institute of Computer-Aided Methods in Mechanical Engineering***
Johannes Kepler University of Linz
Altenbergerstr. 69
4040 Linz, Austria

[2] ***Siemens VAI Metals Technologies***
Turmstr. 44
4031 Linz, Austria

Within the production chain of cold-rolled metal strips, tension levelling is either applied as scale breaker at the entry of the pickling section in continuous pickling lines or as final process step in continuous galvanizing or finishing lines. The main objective of tension levelling is to improve the flatness (shape) of the strip by minimizing the residual stresses. When running through a tension leveller, the strip is bent alternately around multiple rolls with small diameters, while typically high tension stresses (in the range of up to 70 % of the yield strength) are applied. Due to the combination of bending and tensile stresses, the strip is deformed plastically at each roll resulting in a small elasto-plastic strip elongation which typically does not exceed a value of five percent.

Almost all tension levelling setups are based on rough offline calculations which have to be supplemented by trial and error procedures during the operation. To improve the design process of tension levelling machines, a precise model is essential. Key objectives of adequate models are the determination of the bending line including the maxima of strip curvature, the analysis of the necessary level of tension, the tension losses due to the plastic deformation as well as the calculation of reaction forces at the bending rolls. All models comprise a large number of parameters describing strip thickness, the geometrical setup of the tension levelling machine, the selected roll positions, the desired degree of elongation, the strip speed as well as elastic and inelastic material properties.

To attain a comprehensive understanding of the tension levelling process, different models were created, analyzed and compared utilizing the non-linear capabilities of Abaqus Explicit. As the flatness of the strip is the result of a sequence of small deformations along the bending line of the strip, the longitudinal displacements required to reach the desired steady state are generally very large. Hence, a long piece of strip has to be modelled. Additionally, the multiple contacts between strip and rolls require a very fine mesh. Both characteristics yield the necessity of up to several hundred thousand

degrees of freedom even for plane strain calculations. Therefore - in addition to concepts using 2D-continuum elements - models with beam elements were developed to explore the potential of computational cost savings. Extensive analyses as well as consistency checks confirmed the reliability and the comparability of continuum and structural elements. The models created are capable to obtain the relevant process information taking into account the several non-linear characteristics of geometry, contact and material as well as dynamic effects.