MODELLING SHEET METAL FORMING PROCESS USING ABAQUS

Summary – The article contains results of experimental researches and numerical simulations of forming of rectangular drawpieces with conical bottom. The numerical analysis of sheet metal forming by finite element method was carried out using Abaqus program. To describe the plastic properties of sheet metal Huber-Mises isotropic model and Hill anisotropic model were used. Similarly the friction phenomenon was modelled by using the same value of friction coefficient for all surfaces and anisotropic model. Comparison of simulation results with experimental results demonstrated that including the plastic anisotropy and the friction anisotropy in the numerical model causes that simulation results are more approximated to the experimental measurements.