University of Stuttgart
Germany

Material Modeling in LS-DYNA

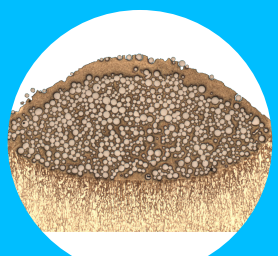
This student research project is part of a project funded by the Ministry of Economics and Technology that deals with the simulation and experimental validation of eigenstresses in laser-generated composite materials. Laser dispersion is the primary process used in this project, where working surfaces of already manufactured parts are coated by adding tungsten powder into the melt pool. This coating serves as a wear protection layer that helps in extending the part's lifetime. Such coating can lead to high economic savings that correspond to a potential saving of up to 200 billion Euros in Germany in 2016.

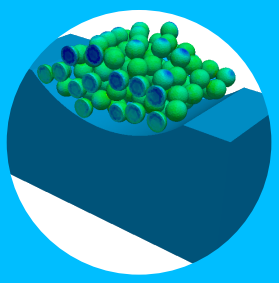
Assuming that artificial structures are already available (this is a topic of another project), the goal here is to simulate an anisothermal problem using the finite element method (FEM) in LS-DYNA. Hence, the first step is to implement a simple anisothermal material model (possibly in Python). The model is then to be used in thermo-mechanical simulations including an in-depth analysis of the results. Optionally, the consideration of particle cracking is given. After that, an optional part of the project is to investigate possibilities of extending LS-DYNA's solver in order to incorporate customed numerical schemes such as model order reduction ones.

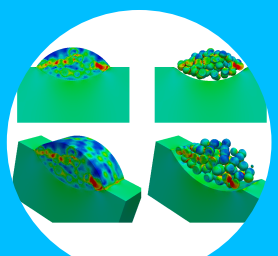
Technical requirements

- Basic Python knowledge
- Interest in simulations and efficient numerical schems
- Starting examples and access to LS-DYNA will be provided

LS-DYNA
Python







Shadi Alameddin • **alameddin@mib.uni-stuttgart.de**