A Three Phase Conjugated Heat Transfer Solver Applied To Additive Manufacturing

Andreas Mark*, Johan Göhl, Simon Ingelsten, Tomas Johnson and Fredrik Edelvik

Fraunhofer-Chalmers Centre, Chalmers Science Park, Göteborg, 412 88, Sweden. *Email: andreas.mark@fcc.chalmers.se

Keywords: Conjugated heat transfer, immersed boundary, volume-of-fluid, three phase flow, phase transition.

Abstract

Additive manufacturing (AM) is growing rapidly in the industry and is dependent on a number of process parameters and complex multiphysics. In the process, metal powder is melted by a laser beam forming a metal structure. Optimization of the process is very hard and is currently dependent on simple melt pool models. Therefore, there is a lot of trial and error involved in finding a robust AM process. To increase the understanding of the process we propose a physical and CFD-based melt pool model. The model includes a three phase conjugated heat transfer transfer solver, where the gas and melted solid (fluid) are treated with the volume-of-fluid method. To accurately capture the flow of the fluid a temperature dependent rheology model is employed. The phase transition occurs over a small temperature span, in which affected cells are transferred to the other phase solver. Finally, a ray trace based heat source model is used to simulate how an array of powder particles on a substrate are partially melted by a laser beam and the final solidified structure is formed.

Introduction

Productivity in powder additive manufacturing is strongly dependent on process parameters, powder properties as well as the required component properties. In general, the standard processes established for powder bed fusion additive manufacturing are empirical compromises between powder size, bed thickness and process parameters, aiming to reach the highest productivity (by increased build speed) and desired properties of the components. Lack of theoretical understanding on the complex interplay between micro-scale and macro- scale properties makes it almost impossible to define an optimal process window for a robust AM process. Hence, a multiphysics model of AM, which takes process parameters such as powder particle size, powder bed thickness, heat input on the local scale determined by the spot size and energy of the energy source (laser or electron beam) as well as scanning speed into account, is required. Until now, AM simulations typically use approximate melt pool models as input. However, to understand the complex physics in the AM process physical, CFD-based, injection and melt pool models are required. With CFD, the flow and heat transfer in the melt pool can be investigated, but due to the fact that melted metals cannot be treated as standard fluids novel techniques need to be developed. With a physical melt pool model, the impact of the process parameters can be studied and used as input for thermo-mechanical AM simulations.

In this work the in-house multiphysics software IPS IBOFlow® is extended to handle AM. The immersed boundary techniques and adaptive octree mesh together with advanced rheology models makes IBOFlow perfectly suited for AM applications. Examples of previous IBOFlow applications include deposition of sealing, adhesive material and 3D bioprinting of nano-cellulose.

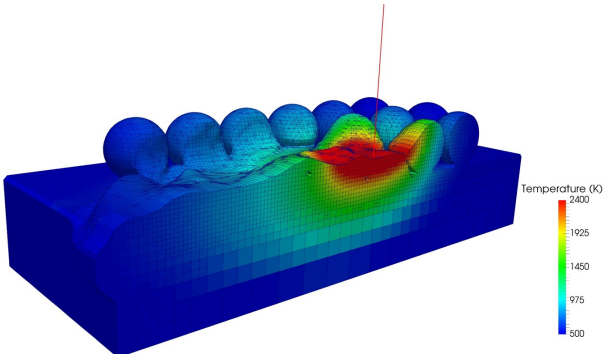

Figure 1: Metal powder and substrate are partially melted by a laser beam (red line) during an AM process simulation. The melted and solid metal are coloured by temperature.

Numerical method

The AM multiphysic model consists of five parts. The first part is the solid volume fraction model, where the fluid and the solid volume fractions are dynamically calculated for all cells and faces. The second part is the three phase conjugated heat transfer model that simulates the momentum and temperature transport in the gas, fluid and solids. The third part is the rheology model that handles the shear and temperature dependent viscosity. The fourth model handles the heat coupling between the fluid and solid and the energy phase change process. The final model, handles the beam model (energy source) that heats up and melts the powder and the solid substrate.

Initially the solid objects are described by a surface triangulation, then the solid volume fraction algorithm approximates the intersection of the local triangle mesh with each

cell by a plane. This is done by fitting a least square plane to the intersection between the cell and the triangle mesh. The plane splits the cell and its faces in two parts, which are used to compute the volume and area fractions of the solid (Svelander et al. (2017)). In the conjugated heat transfer solver the solid volume fraction is used to determine active solid cells and is included in the discretized heat equation.

In IPS IBOFlow® the Navier-Stokes and heat equations are solved on an adaptive Cartesian octree grid that can be refinements to resolve boundary layers and fluid interfaces. Further, the grid is automatically generated and internal boundary conditions are handled with the mirroring immersed boundary method (Mark and van Wachem (2008); Mark et al. (2011)). Gas and fluid phase are treated with the volume-of-fluid method. The convection and diffusion of the temperature is modelled by the coupled temperature equations. The heat transfer solver couples to the momentum equation through the buoyancy/natural force added to the external body force in the momentum equations.

The fluid and solid temperatures are discretized in two different matrices and the two phases are coupled with physical boundary conditions. Hence, one cell that is intersected by a fluid boundary has both a fluid and a solid temperature. In the direct numerical simulation coupling the fluid temperature at the solid/fluid interface is set to the solid temperature using the immersed boundary technique and the solid temperature is coupled to the fluid temperature with a heat flux boundary condition (Mark et al. (2013)). When calculating the flux, the local fluid temperature gradient is used and if grey body radiation is included it is also added. In the melting and solidification model the phase change enthalpy is added to the heat capacity during a temperature span. When the temperature of a fluid or solid cell have reached the upper or lower temperature limit it is dynamically transferred to the other other phase solver.

The rheology of the fluid metal is modelled by a temperature dependent rheology model, where the exponential temperature dependency is shown in Figure 2. The laser beam is modelled as a fluid and solid heat source. Ray tracing is adopted to find the impacting fluid and solid region where the source is applied uniformly as a first approximation. The active depth of the source is estimated from experiments.

Results and Discussion

To demonstrate the potential of the framework an array of solid powder particles are placed on substrate. The array represents an idealised positioning, in reality the powder is randomly placed. A laser beam with 200 W is applied and after enough energy is absorbed by the powder and substrate, they melt and begin to flow according to the rheology model. When the beam moves away the melted metal solidifies and a new surface is generated. In the next step of the AM process, new powder is placed on the formed surface and the process is repeated until the object is manufactured. During the simulation internal temperature gradients and fluid stresses can be tracked and used as an input to thermo-mechanical AM simulations on a larger scale.

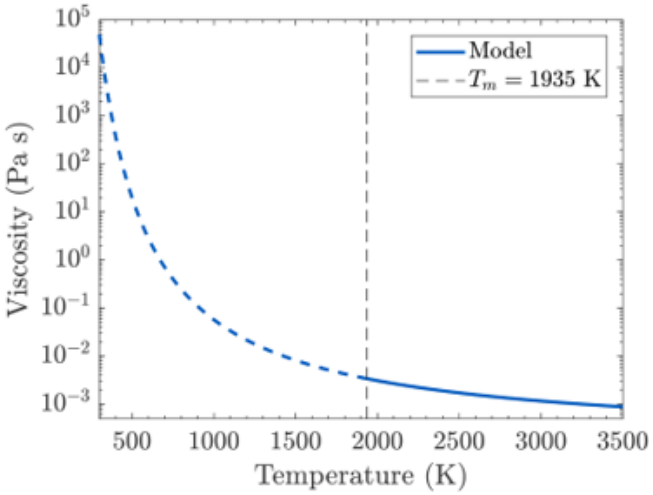

Figure 2: Temperature dependent rheology model where T_m is the phase transition/melting temperature.

Acknowledgement

This work has been supported in part by the Swedish Governmental Agency for Innovation Systems, VINNOVA, through the FFI Sustainable Production Technology program, and the Compentence Centre for Additive Manufacturing - Metal, CAM^2. The support is gratefully acknowledged.

References

A. Mark, E. Svenning, and F. Edelvik. An immersed boundary method for simulation of flow with heat transfer. *International Journal of Heat and Mass Transfer*, 56:424–435, 2013.

Andreas Mark and Berend GM van Wachem. Derivation and validation of a novel implicit second-order accurate immersed boundary method. *Journal of Computational Physics*, 227(13):6660–6680, 2008.

Andreas Mark, Robert Rundqvist, and Fredrik Edelvik. Comparison between different immersed boundary conditions for simulation of complex fluid flows. *Fluid Dynamics & Materials Processing*, 7(3):241–258, 2011.

Frida Svelander, Gustav Kettil, Tomas Johnson, Andreas Mark, Anders Logg, and Fredrik Edelvik. Robust intersection of structured hexahedral meshes and degenerate triangle meshes with volume fraction applications. *Numerical Algorithms*, pages 1–40, 2017.