



Software Lab:

Modeling: ★☆☆☆☆
Mathematics: ★★☆☆☆
Programming: ★★★★★

Implementation of the Finite Cell Method in ANSYS®

Setting

The FCM is an embedded domain approach for high order finite elements [1]. As such, it embeds the physical domain in a simple Cartesian mesh. The original geometry is recovered at the integration level using adaptive methods, which are easy to implement for Cartesian grids. This makes the FCM very attractive for engineering simulations, where currently most of the time is spent in mesh generation.

Task

Make the FCM available for users inside the commercial FEA software ANSYS®. Therefore, you will have to

- get familiar with the basics of the Finite Cell Method.
- develop a strategy to implement the FCM into the workflow of ANSYS®.
- implement User-Defined-Elements for linear elasticity and weakly enforced boundary conditions.
- integrate these elements into the ANSYS®' workflow.



Supervisors

Tino Bog, Simulation in Applied Mechanics Group, tino.bog@tum.de

László Kudela, Simulation in Applied Mechanics Group, laszlo.kudela@tum.de

Nils Zander, Simulation in Applied Mechanics Group, nils.zander@tum.de

References

[1] A. Düster, J. Parvizian, Z. Yang, and E. Rank, "The finite cell method for three-dimensional problems of solid mechanics," *Computer Methods in Applied Mechanics and Engineering*, vol. 197, no. 45–48, pp. 3768–3782, Aug. 2008.

[2] ANSYS, Inc., ANSYS Release 14.0, Help System, Element Reference. 2011.