TECHNISCHE
UNIVERSITÄT
MÜNCHEN

**Chair of
Structural Analysis**

Prof. Dr.-Ing.
Kai-Uwe Bletzinger

Topic 21:

Implementation of automated meshing algorithms into OpenFOAM

The aim of this software project is to develop a module for automated meshing within the c++ software environment OpenFOAM, which is used for CFD simulations.
The specifications are given as follows:

- geometric description of the domain boundaries in stl-format and OpenFOAM format
- output in OpenFOAM format
- direct implementation within the solver routines of OpenFOAM
- using NetGEN routines as black-box meshing tool for the generation of free meshes
- for changes of the boundary geometry: use of existing mesh movement algorithms to adapt the mesh, check mesh quality and decide for local remeshing
- extension of the module for use within the existing coupling environment for fluid structure interaction simulations

Supervision by:
Thomas Gallinger
gallinger@bv.tum.de
089/289-22418