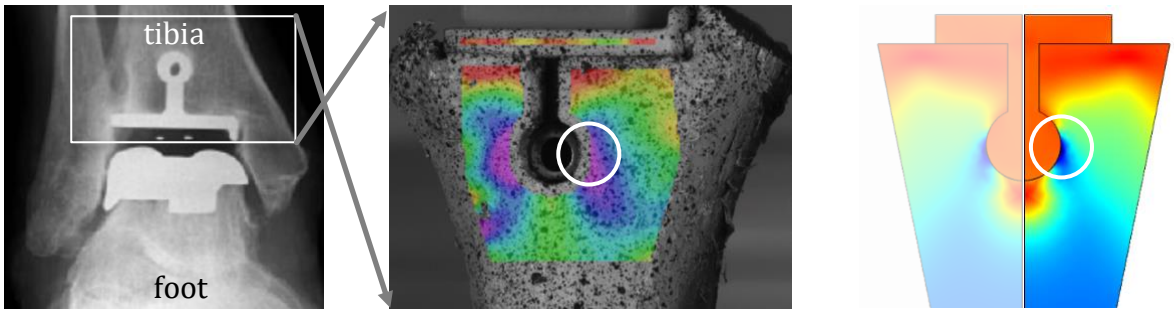
**FEM: mesh convergence analysis**

A simplified numerical model of an ankle prosthesis has been developed. The model contains the tibia and the tibial component. Because of the assumed symmetry, only a quarter of the system is geometrically modeled. The model accounts for the cortical and trabecular bone. The model will be validated with experimental measurements of strain on the anterior surface of a cadaveric tibia. In this experiment, an axial force of 2 kN was applied on the metallic tibial component, while the other (proximal) end was fixed in resin.

Before the experimental validation phase, the model must be verified. In this exercise you have to check the sensitivity of the numerical model to the mesh refinement. Open the model (ankleProsthesis.mph). Apply the correct symmetry and boundary conditions. Evaluate the axial strain ($E_{zz}$) in the most lateral point around the implant cylinder (center of white circle in the figure below), where the strain is maximal. Start with the coarsest mesh, and refine to the finest one. Plot the axial microstrain versus the number of degrees of freedom (numberofdofs). Use the ppr() operator (polynomial-preserving recovery[1]) to evaluate the stain. What can you deduce about the accuracy of the numerical prediction? Do the same with the volume of bone with first principal strain (tension) above a value of 1000 microstrain.




Ankle Arthroplasty Experiment axial strain Numerical axial strain

[1] More information on ppr operator in COMSOL Documentation/COMSOL Multiphysics/COMSOL Multiphysics Reference Manual/Global and Local Definitions/Operators, Functions, and Constants/Built-in Oprators/ppr (online or pdf).