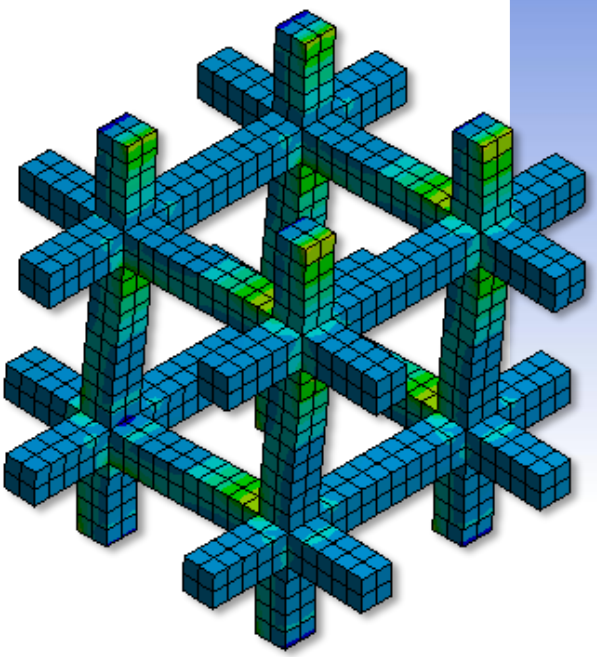
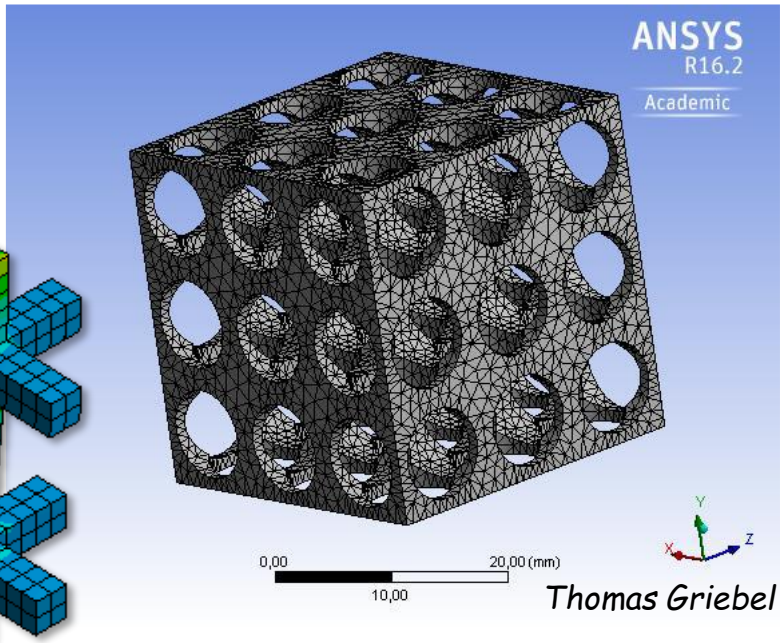
Lab for Lecture 4

Modeling of Artificial Trabecular Bone Structure

Use (A) Ansys Workbench or (B) Ansys Classics to model and analyze an artificial trabecular bone structure.





Thomas Griebel

Hints for Modeling

- You may start with a 2D or 3D case.
- You may start with a solid cube and cut out regular positioned spheres or boxes.
- You may construct beam like structures in all three directions and put them together (see Fig.)
- You may combine 1D beam elements (instead of 3D solids) in a 3D structure.
- The model should be a so called representative volume. Symmetry extension of the cut out in a periodic manner should be possible.

Tasks

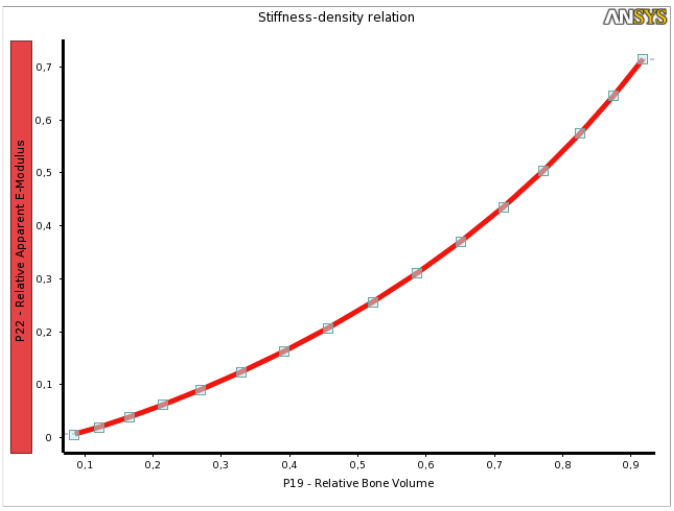
It is not necessary to finish with all tasks within the official lab time. The ECTS points for the course a taking 2 hours "homework" each week into account.

1. Using the FE model, determine the global (apparent) elastic behavior of the trabecular bone sample. Assume a bone volume content of 20% to 30%. Is it isotropic?
2. Perform mesh convergence analyses.
3. Use the parameterization options to perform a series of FE analyses with changing bone volume content. Evaluate the stiffness-density relation (see figure below) and compare with the one from Carter & Hayes, 1977

$$E = 3790\ \dot{\epsilon}^{0.06}\ \rho^{3}.$$

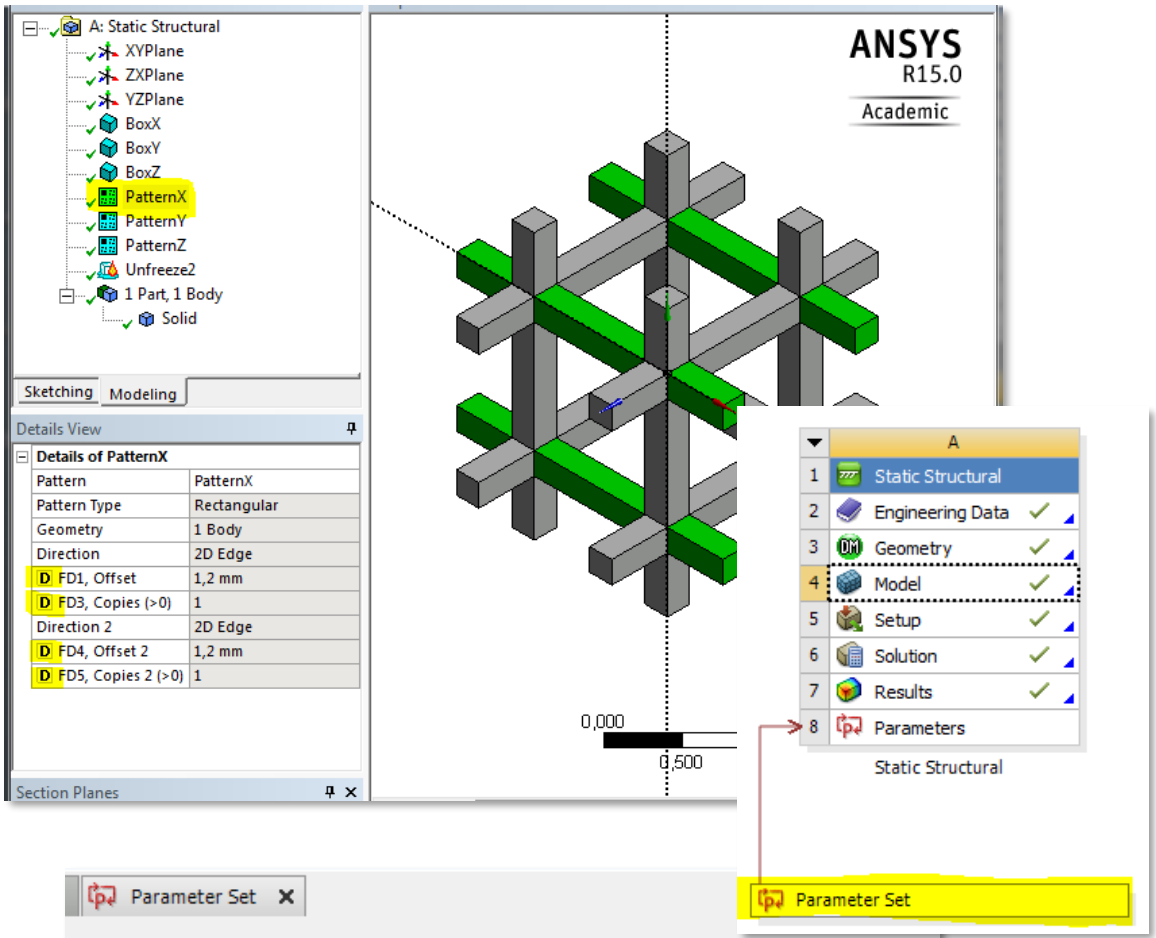
4. Homework: Calculate global deformations, stresses and strains of both geometries loaded by the three different load cases. Use the information given in the lecture.
5. Homework: Implement a plastic material law for the bone tissue and try to evaluate the ultimate mechanical behavior of trabecular bone. Produce a stress-strain curve of the global apparent behavior.
6. Homework: Perform a series of FEAs of last working step and compare to the Carter-Hayes-Relation for the strength:

$$S = 68\ \dot{\epsilon}^{0.06}\ \rho^{2}$$



A) Modeling with Ansys Workbench

Use the parameterization options to allow for easy reruns with changed parameters. See figures for inspiration or refer to a lecture at UZWR homepage.



Outline of All Parameters

	A	B	C	D
1	ID	Parameter Name	Value	Unit
27	P42	BoxX.FD5	0,5	mm
28	P43	BoxX.FD6	2,4	mm
29	P44	BoxX.FD7	0,2	mm
30	P45	BoxX.FD8	0,2	mm
31	P16	TrabekelDicke	0,2	mm
32	P39	PorenGroesse	1	mm
33	P40	AnzahlTrab	2	
*	New input parameter	New name	New expression	
35	Output Parameters			

B) Modeling with Ansys Classics and APDL Script

If you are not familiar with the classical Ansys GUI nor the usage of Ansys Parametric Design Language APDL refer to a lecture at UZWR homepage

Pictures that might help: