FEA of Rat Femur: Outline of Steps in ANSYS

1. Start Workbench 2019R2
2. Under Component Systems, choose Geometry and then double click to open SpaceClaim
3. Open STL file in SpaceClaim
4. Use ShrinkWrap tool (“Facets” tab) to smooth the shape using a 0.2 mm gap size.
5. Add coordinate axes planes using the “plane” tool in the “Design” tab by clicking on the origin of the secondary coordinate axis (you have to zoom out to see it).
6. Add planes to segment the bone into parts (add two more XY planes by clicking on the existing XY plane while using the plane tool)
7. Press Esc until you exit the plane function. *It’s very easy to add new planes by accident if you don’t press Esc.*
8. Use the “Skin Surface” tool in the “Tools” tab to connect the planes.
9. Use the “Stitch” tool in the “Repair” tab to connect the surfaces together to form one surface.
10. Highlight *Surface* in the tree and then click Create Coordinate System.
11. Use the “Pull” tool, “Scale Body” option, and then choose the new coordinate system origin for the anchor point. Specify a scaling of 0.75.
12. Define the surfaces where you are going to apply the boundary conditions.
    a Add two 1 mm x 4 mm rectangles in “Sketch” mode by clicking on the plane in front of the femur. Move them so they are 15 mm apart and centered vertically on the femur. Project the rectangle onto the surface.
    b Create another plane parallel to the one you just worked with, move it and insert a 1 mm x 3 mm rectangle. Center it vertically and horizontally on the femur. Project the rectangle onto the surface.
13. Suppress all surfaces except the main bone surface (translucent green surface).
14. Suppress Shrinkwrap body
15. Save project now and often
16. In Workbench, drag “Static Structural” onto the geometry.
17. Mesh the shell with element size set to, say, 0.3 mm. MAKE SURE THE UNITS ARE CORRECT.

18. Specify thickness of, say, 0.7 mm and set Offset Type: Top.
19. Specify material properties: $E = 5\ \mathrm{GPa} \quad \nu = 0.4$
20. Specify the boundary conditions:
    a "Frictionless Support" for the pair of rectangles where the model is going to be held
    b "Displacement" for the single rectangle where the force is going to be applied (x = desired value, say, 0.3 mm; others = 0).
21. Click on Solve to have the solver calculate nodal displacements and rotations.
22. Post-process the results by adding deformation, stress, strain and reaction objects to the "Solution" part of the project tree
23. Refine the mesh, re-run and check sensitivity of results to the mesh