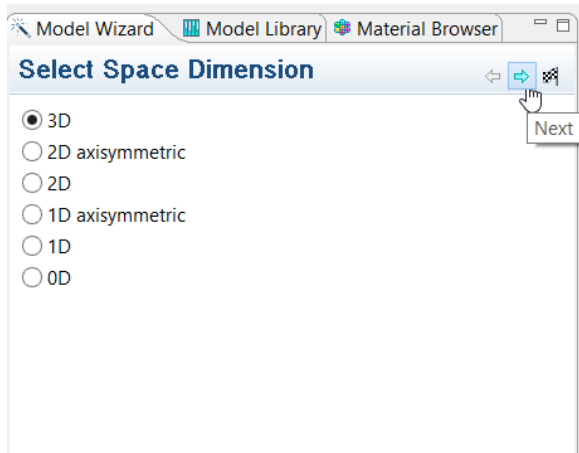


Appendix

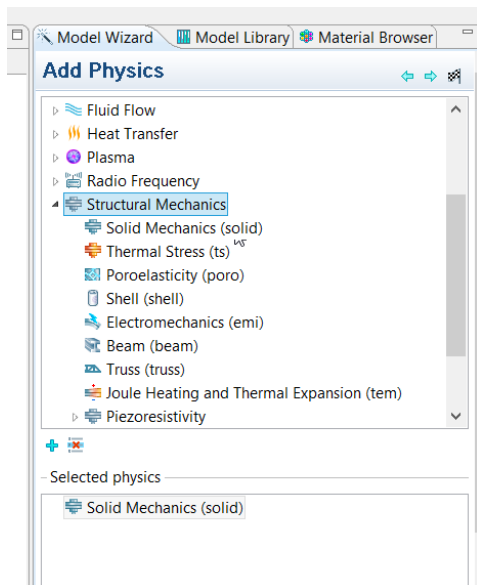
(Contributed by Mohd Hafif Basha)

Step by step method of using COMSOL simulation software.

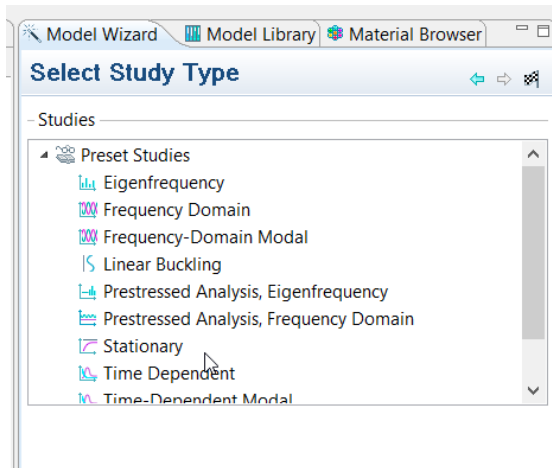
Step 1 : select 3d in select space dimension



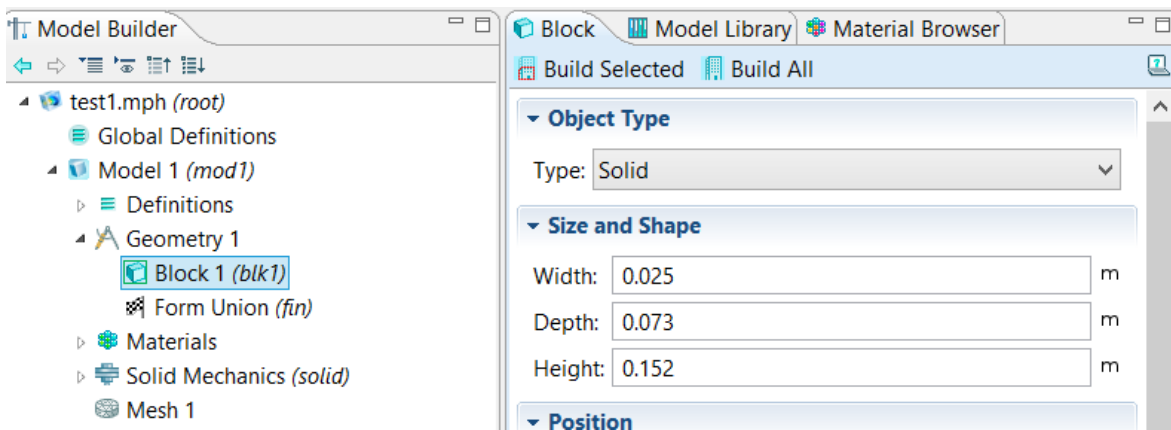
2. Select 'structural mechanics > solid mechanics' in add physics



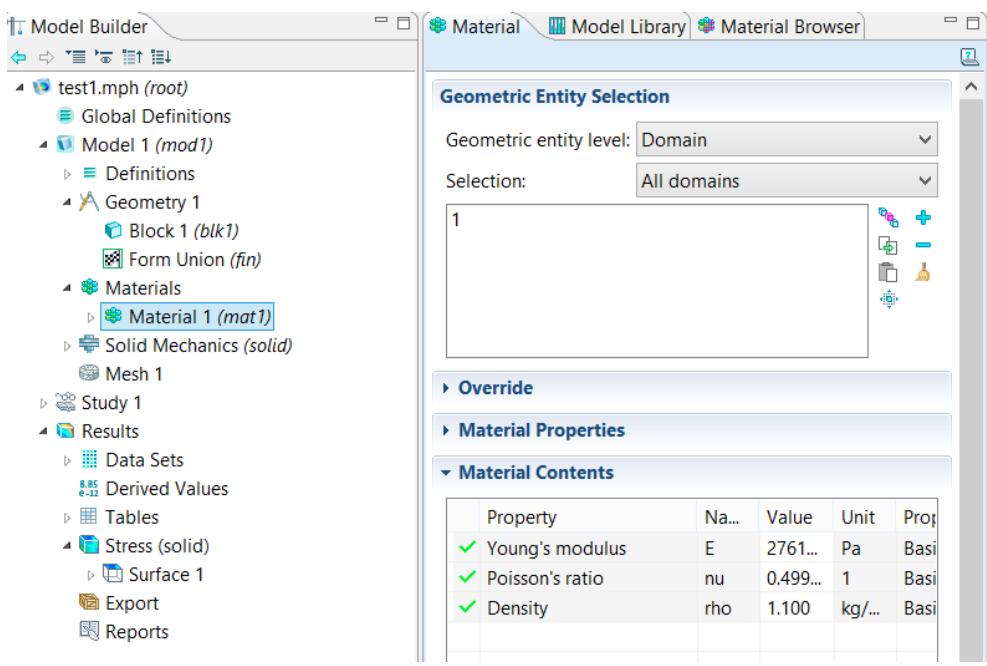
3. Select 'stationary' in study type and click finish



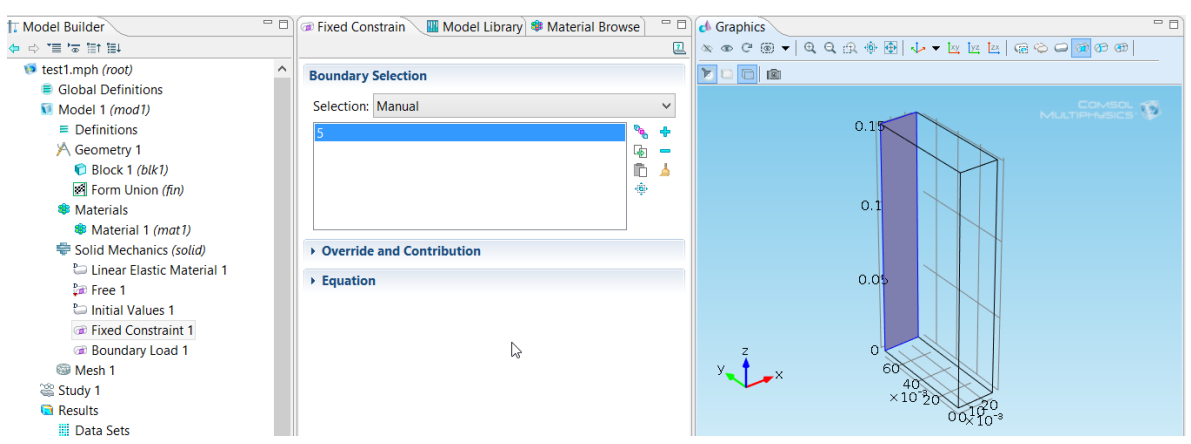
4. Geometry of 'block' is built with the following dimension



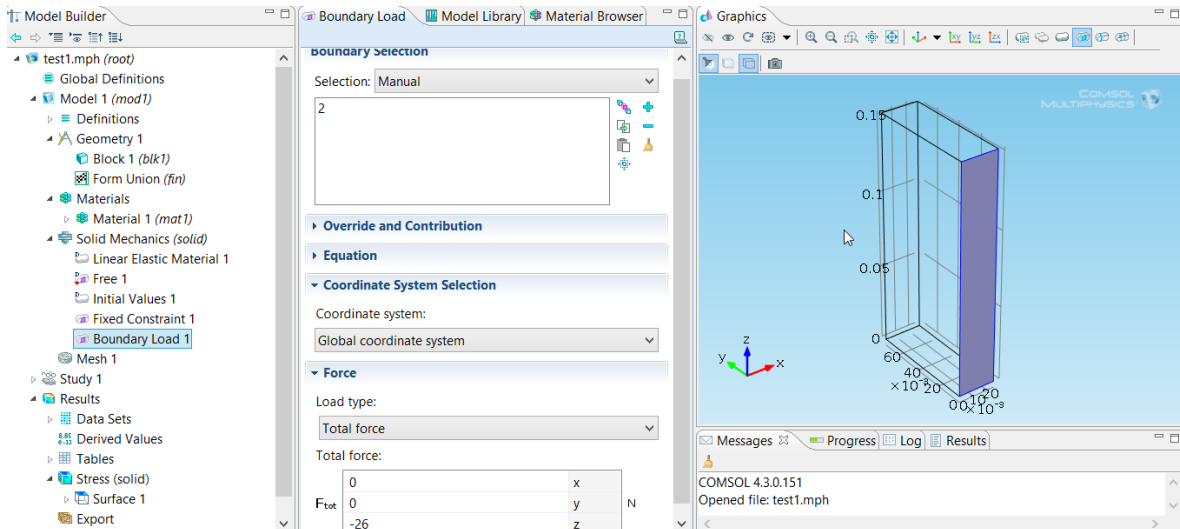
5. The material properties is set



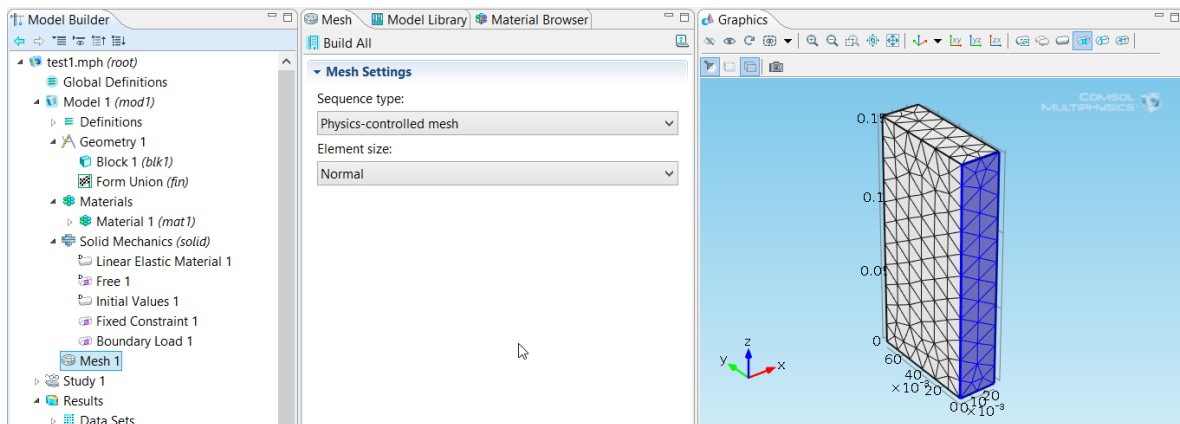
6. Fixed constraint is set



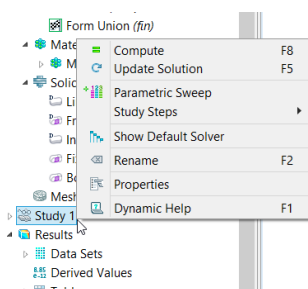
7. Boundary load is set



8. Meshing is set to normal



9. Compute



10. Result is shown as Z- component displacement field

