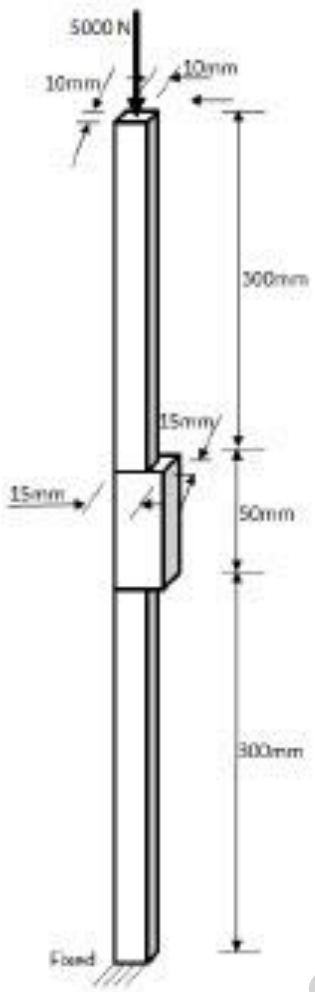
Buckling Analysis of a Column in ANSYS

Armin Yousefi Kanani

DESIGN MODELER

The information of question

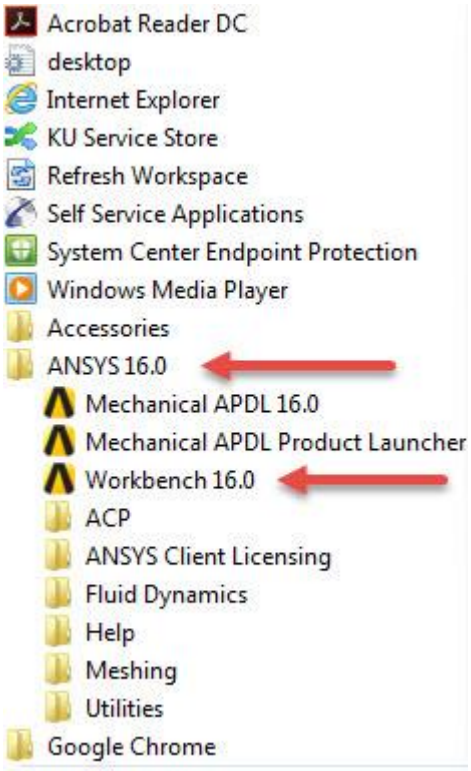


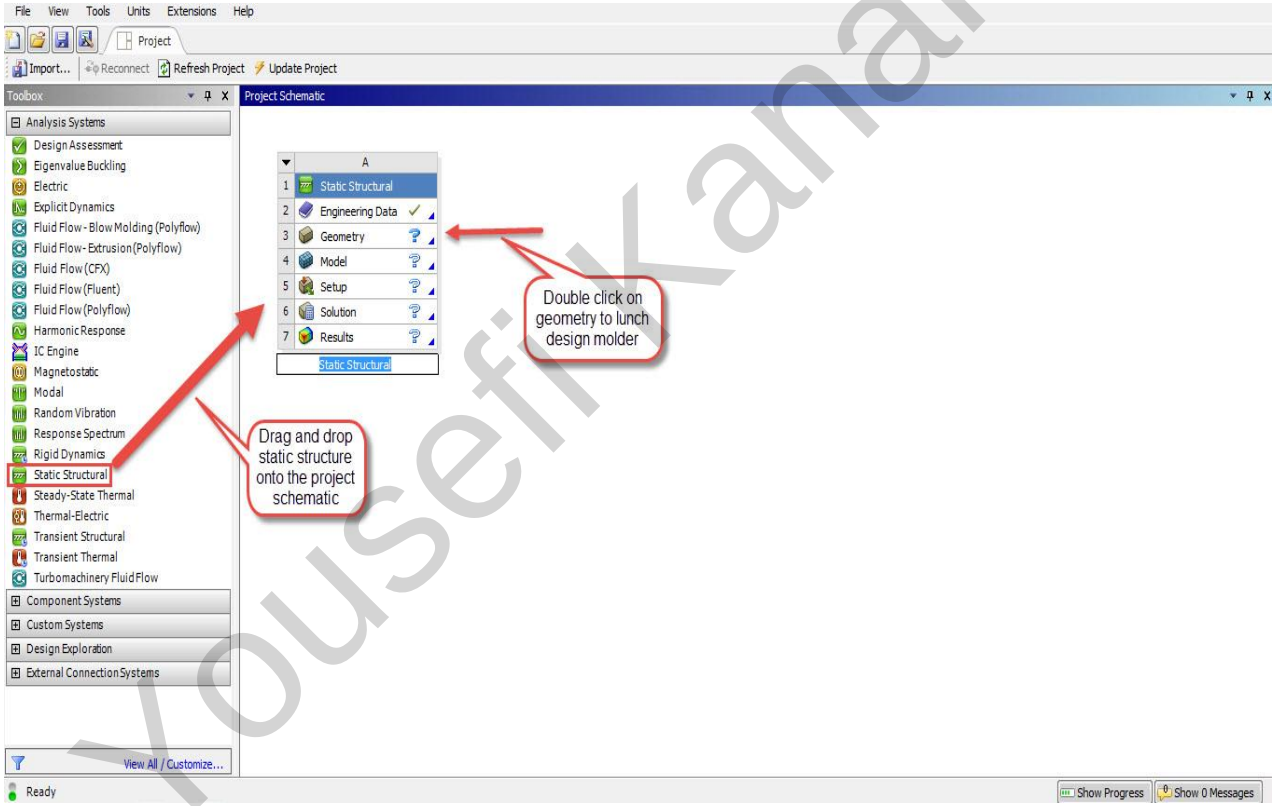
Young’s modulus, E = 2.1e5 N/mm2

Poisson’s ratio, v = 0.3

Yield strength =250 N/mm2

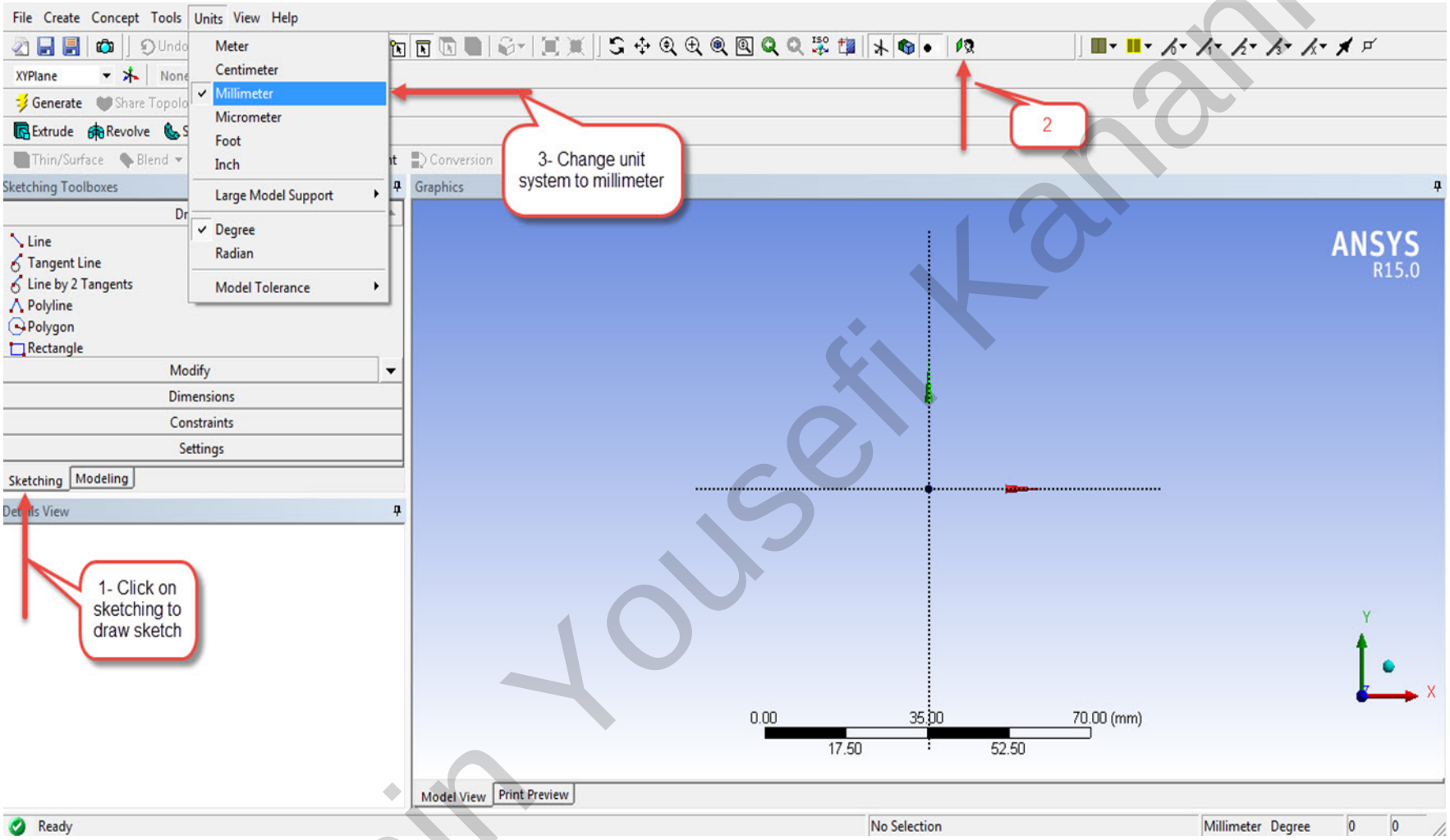
2- Open Workbench



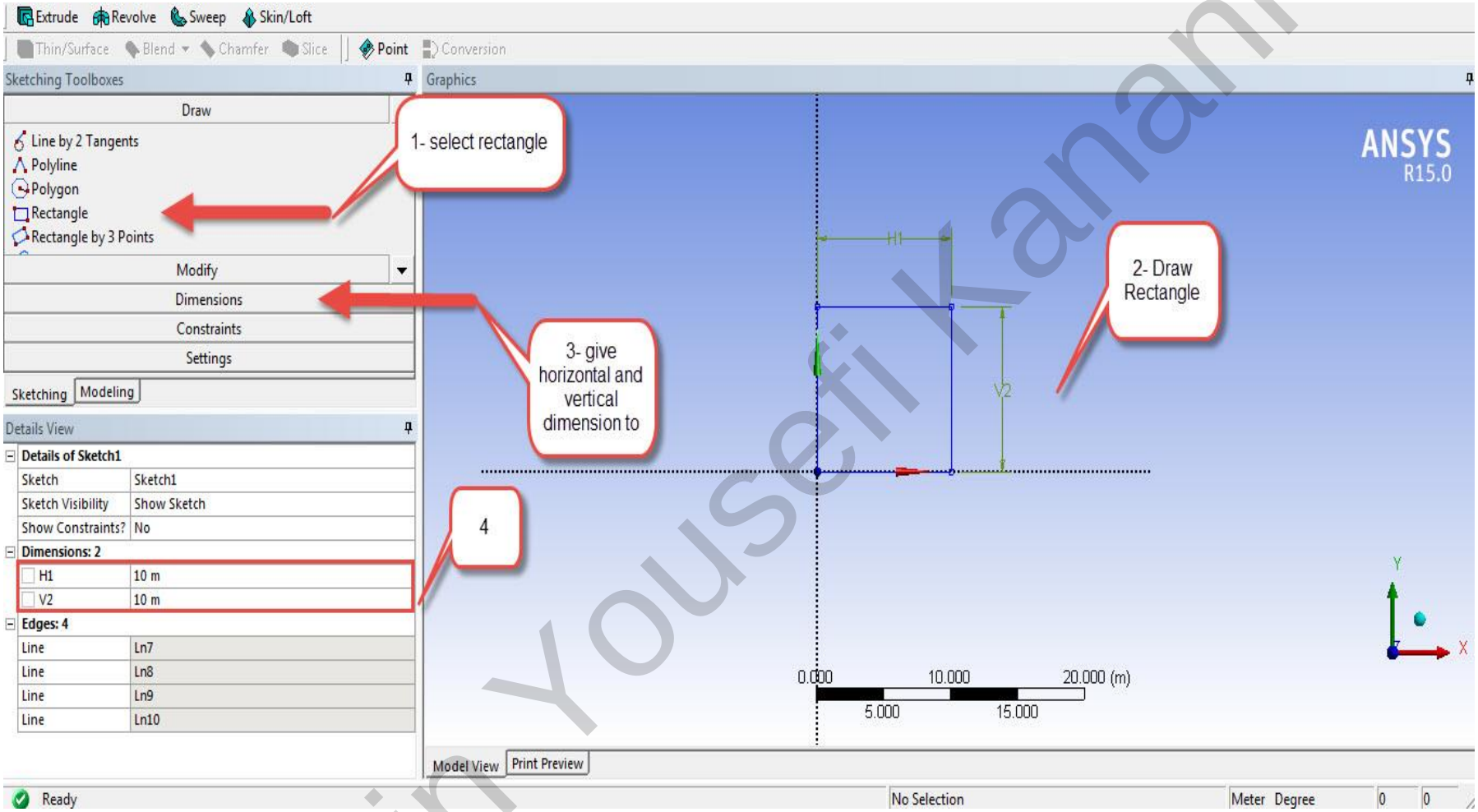


Open ANSYS workbench and drag and drop static structure and double click on geometry to lunch design molder

3- Design Modeler

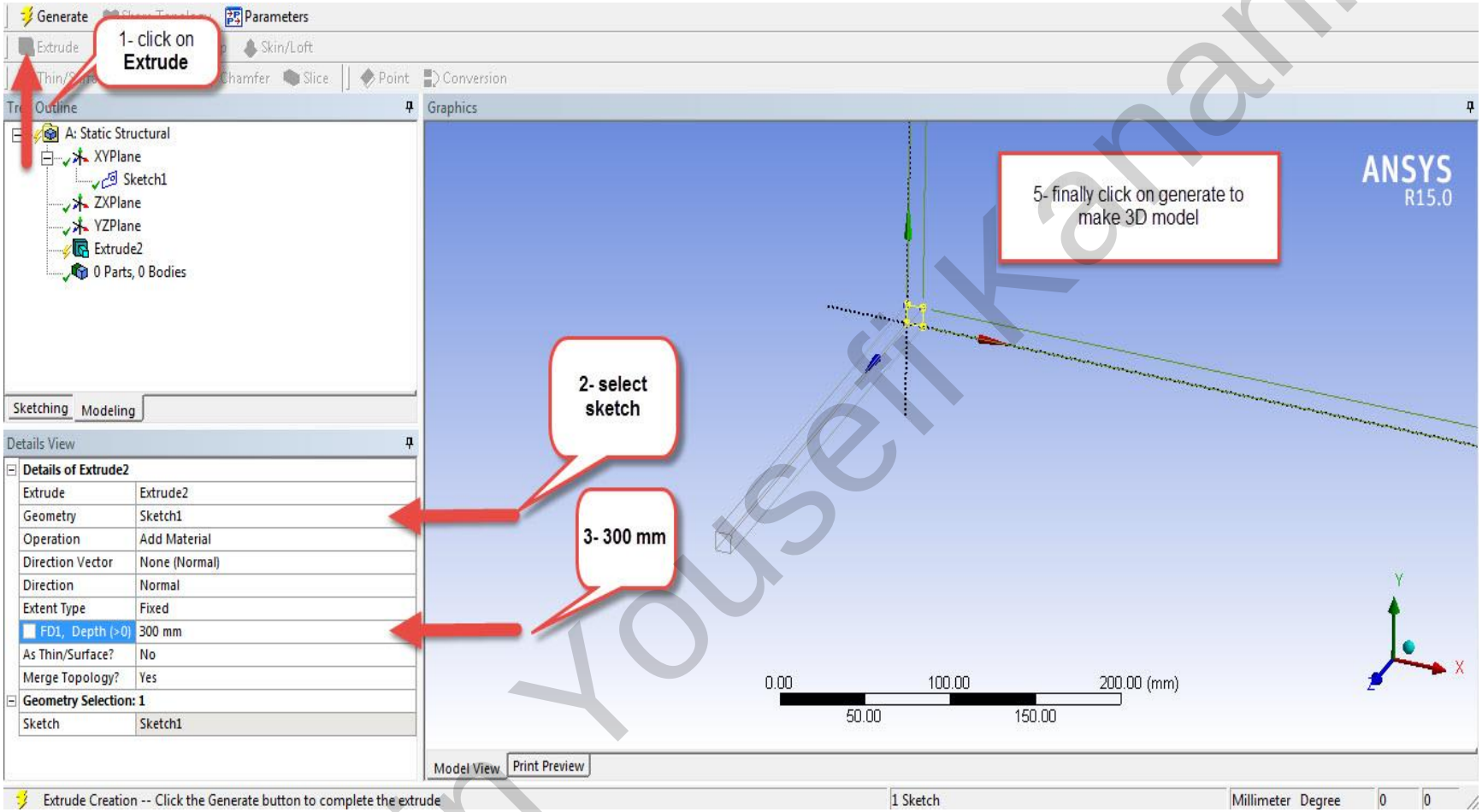


4- Sketch



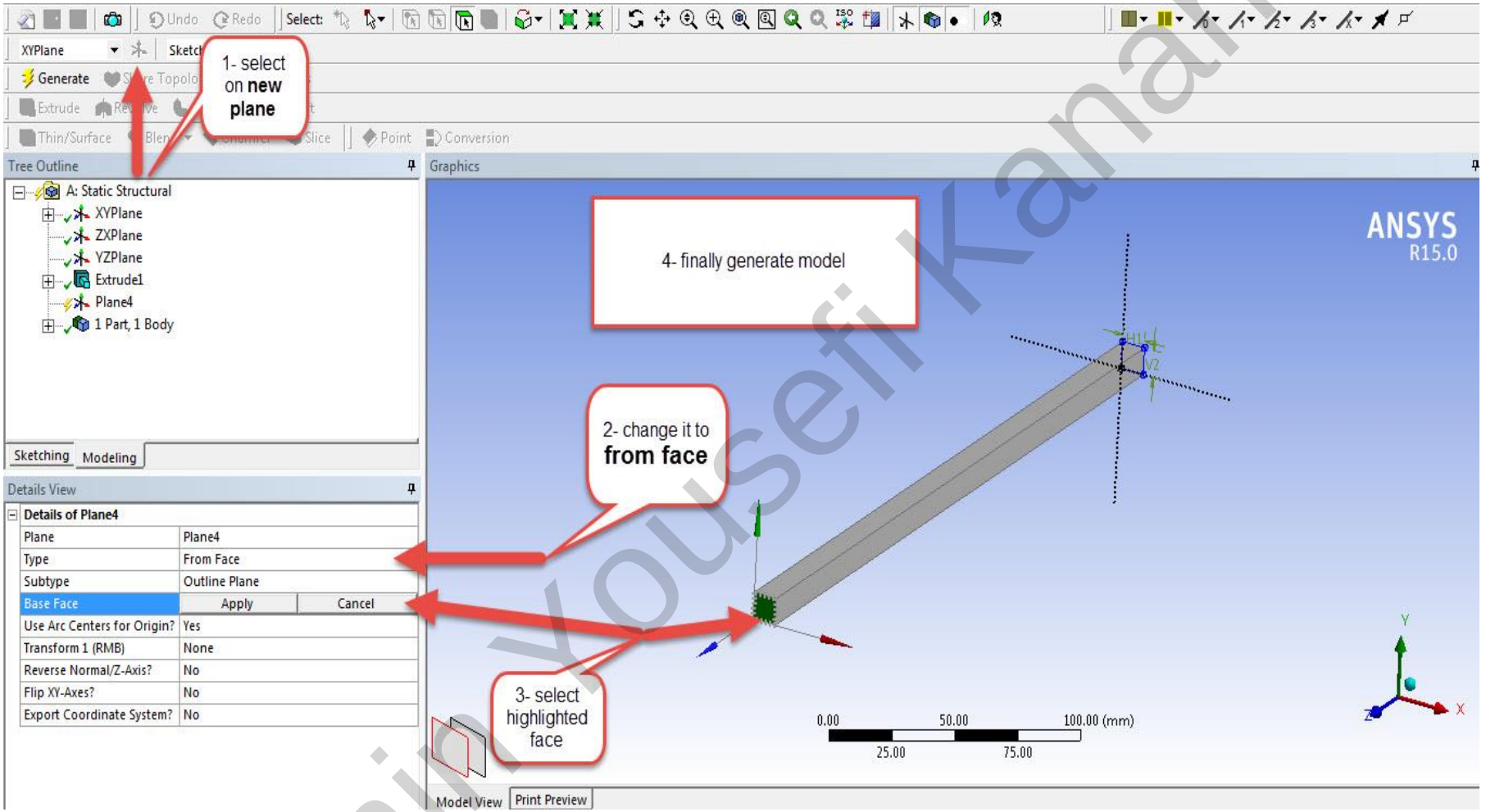
Select rectangle from draw toolbars then give horizontal and vertical dimension to rectangle L:10 mm, w: 10 mm

5- Extrude

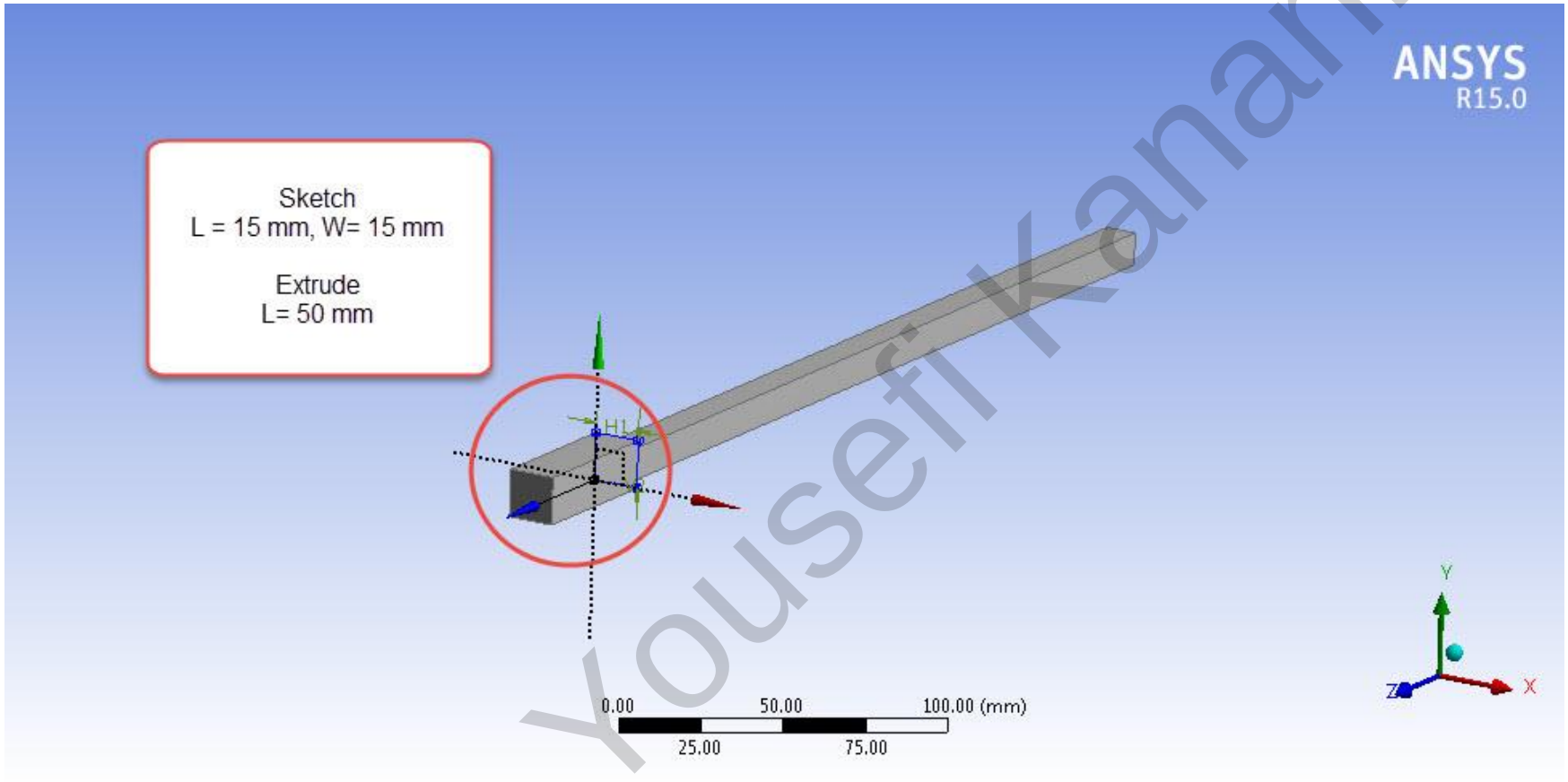


First click on extrude and select sketch from graphic view. Then give 300 mm depth and generate 3d model

6- Making a New Plane

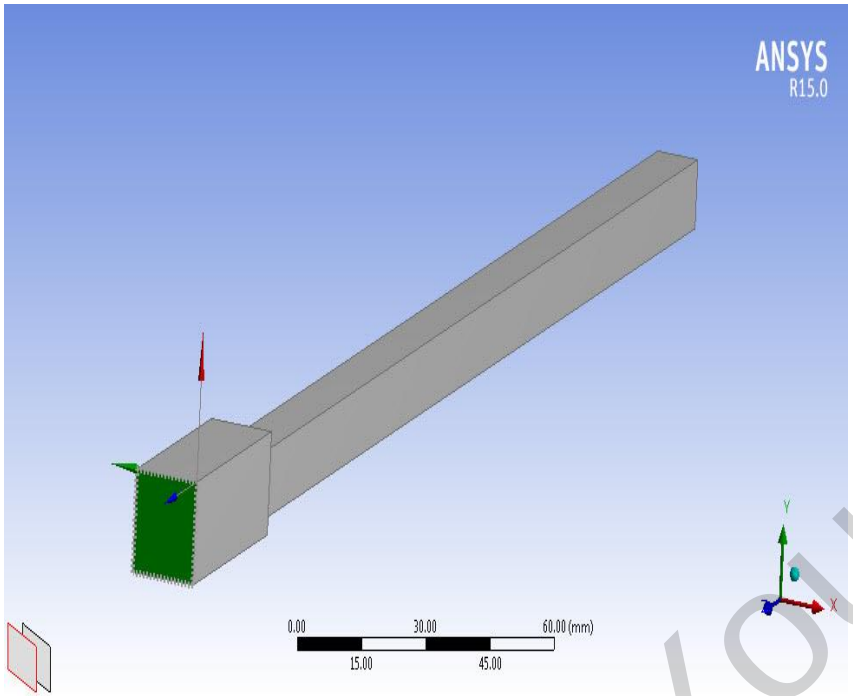


7- Geometry-Section 2

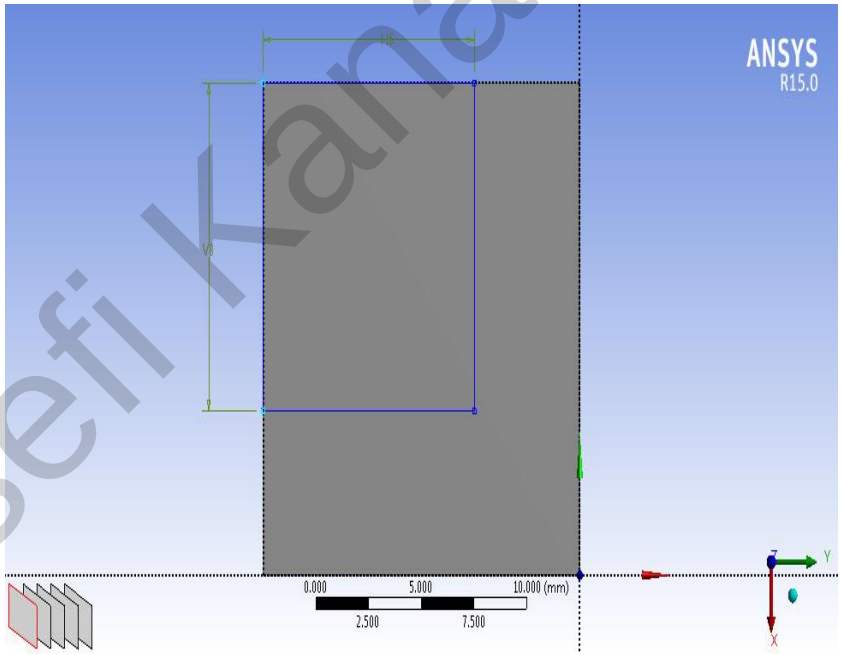


Now we are going to make another rectangle on new plane. Same as previous slides. 1- select on new plane 2- draw rectangle in size of L: 15 mm W: 15 mm 3- Extrude it by **50 mm**

8- Geometry- Section 3



1- Make another plane on highlighted face



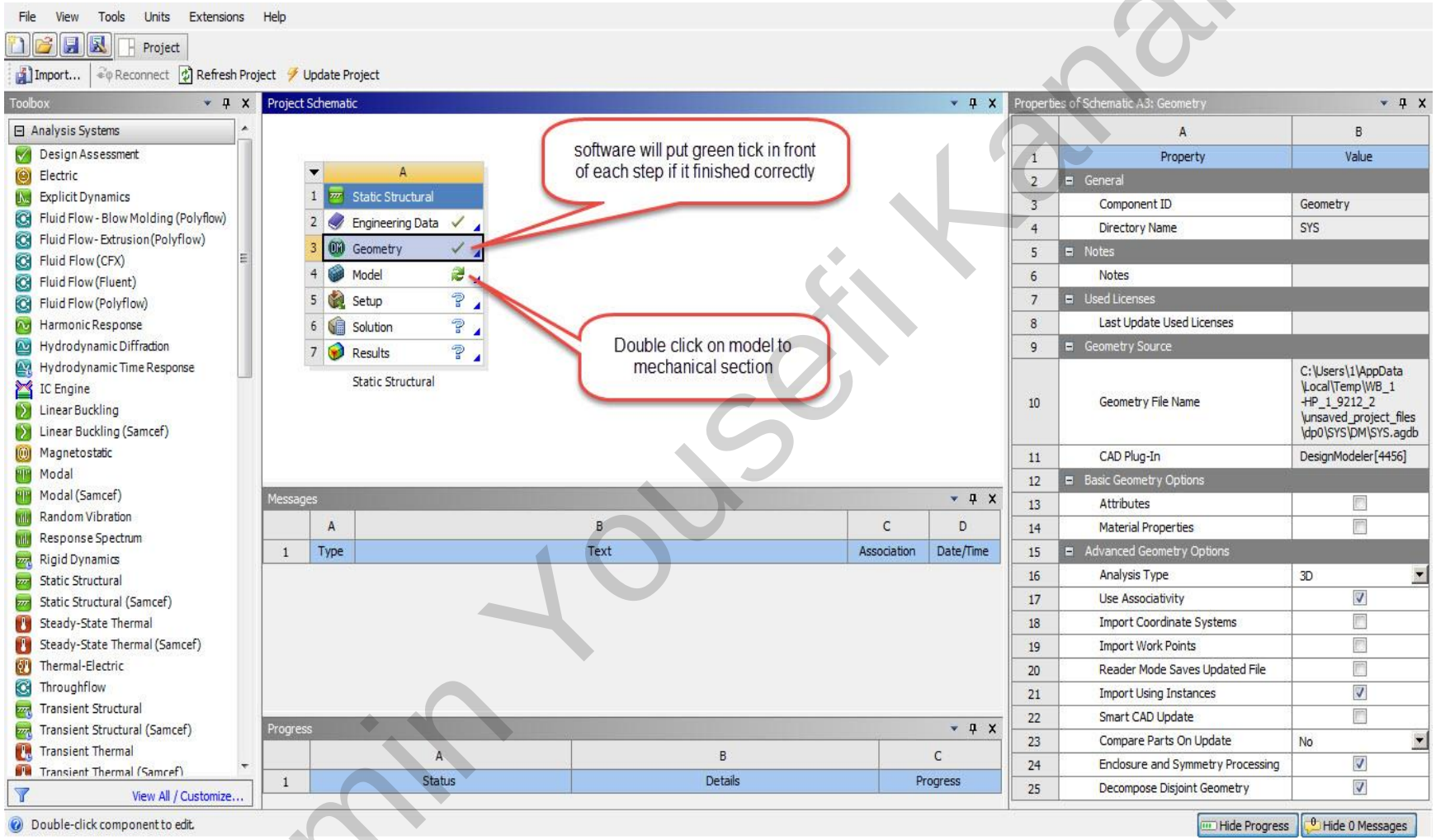
2- draw Sketch L:10 mm and W: 10 mm

9- Final model

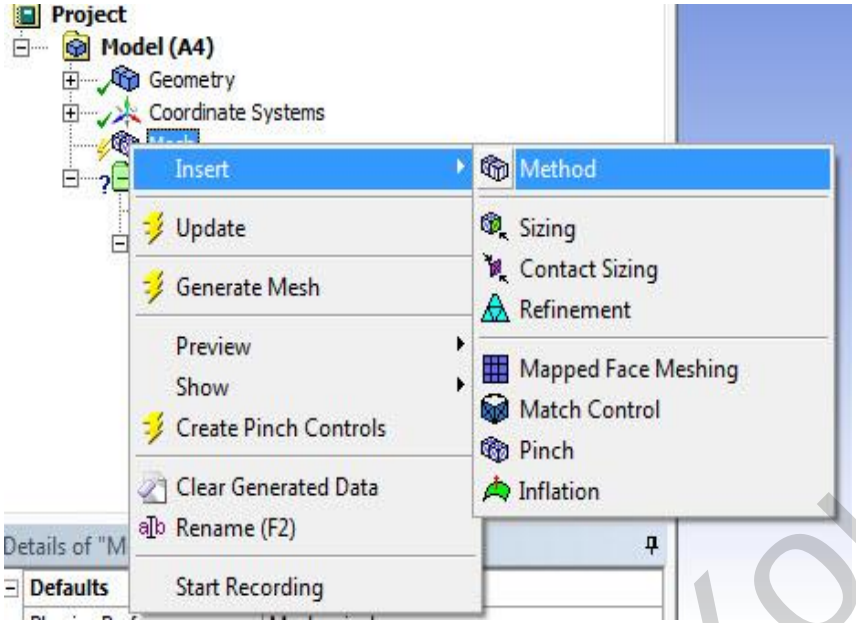
Extrude sketch by 300 mm

STATIC STRUCTURAL ANALYSIS

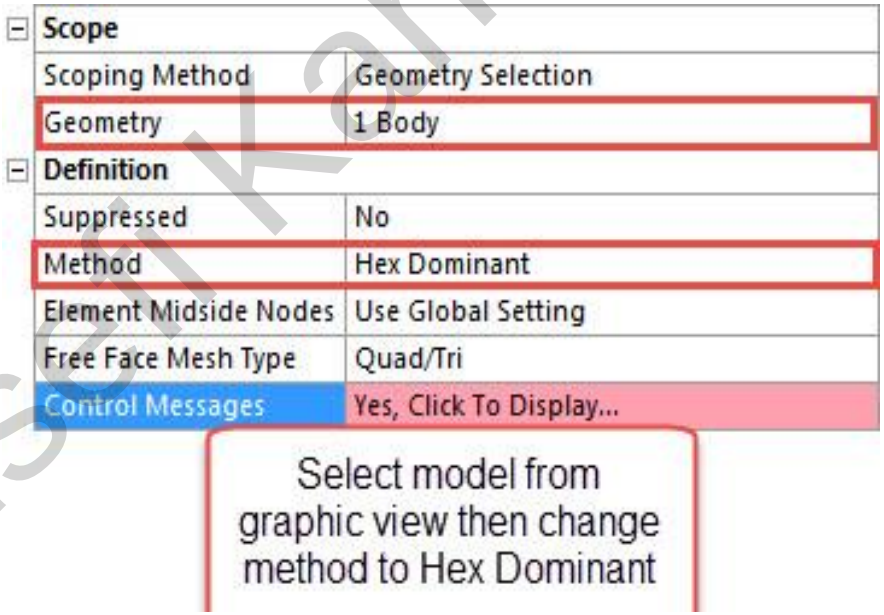
10- Open Model Section



11- Mesh

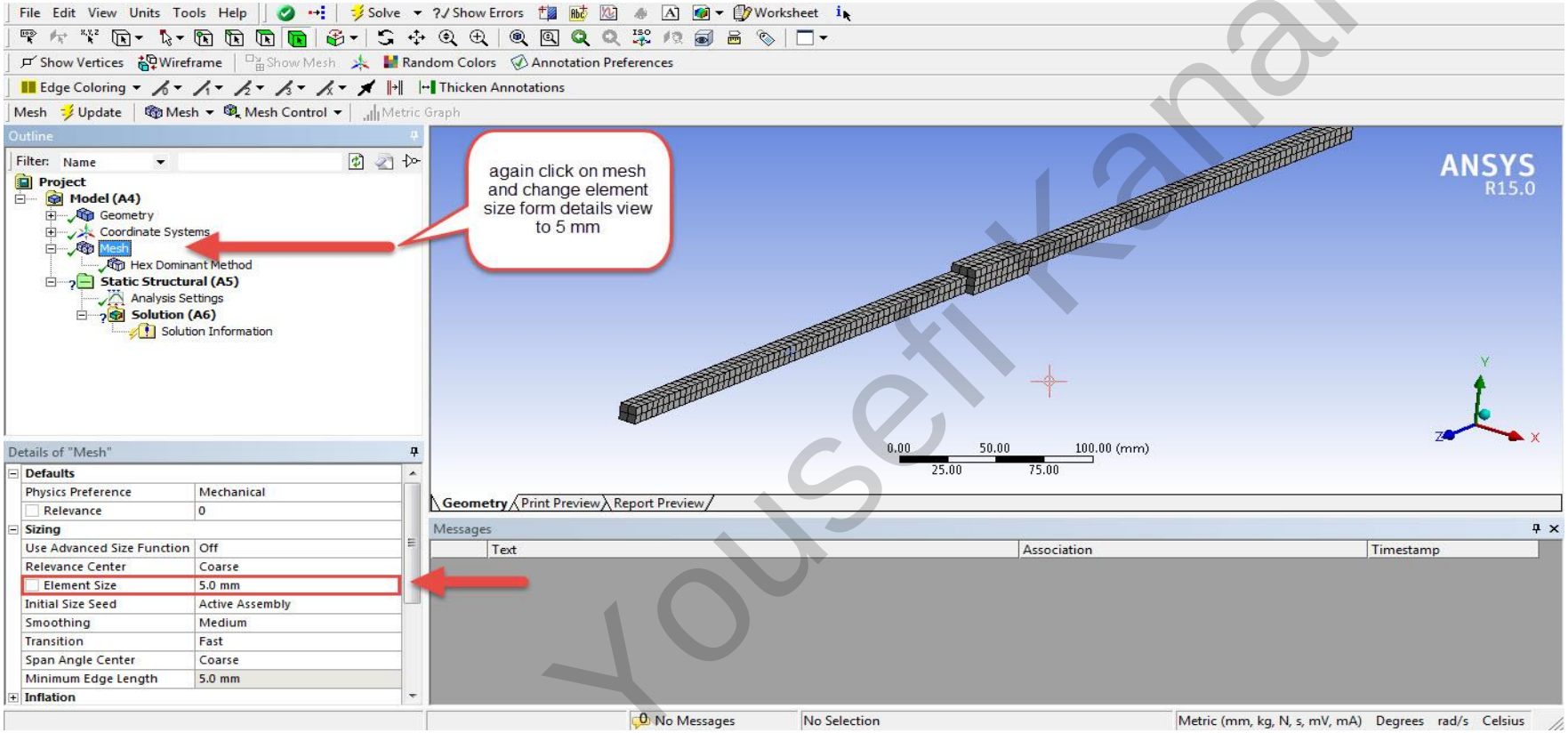


Right click on mesh and insert method



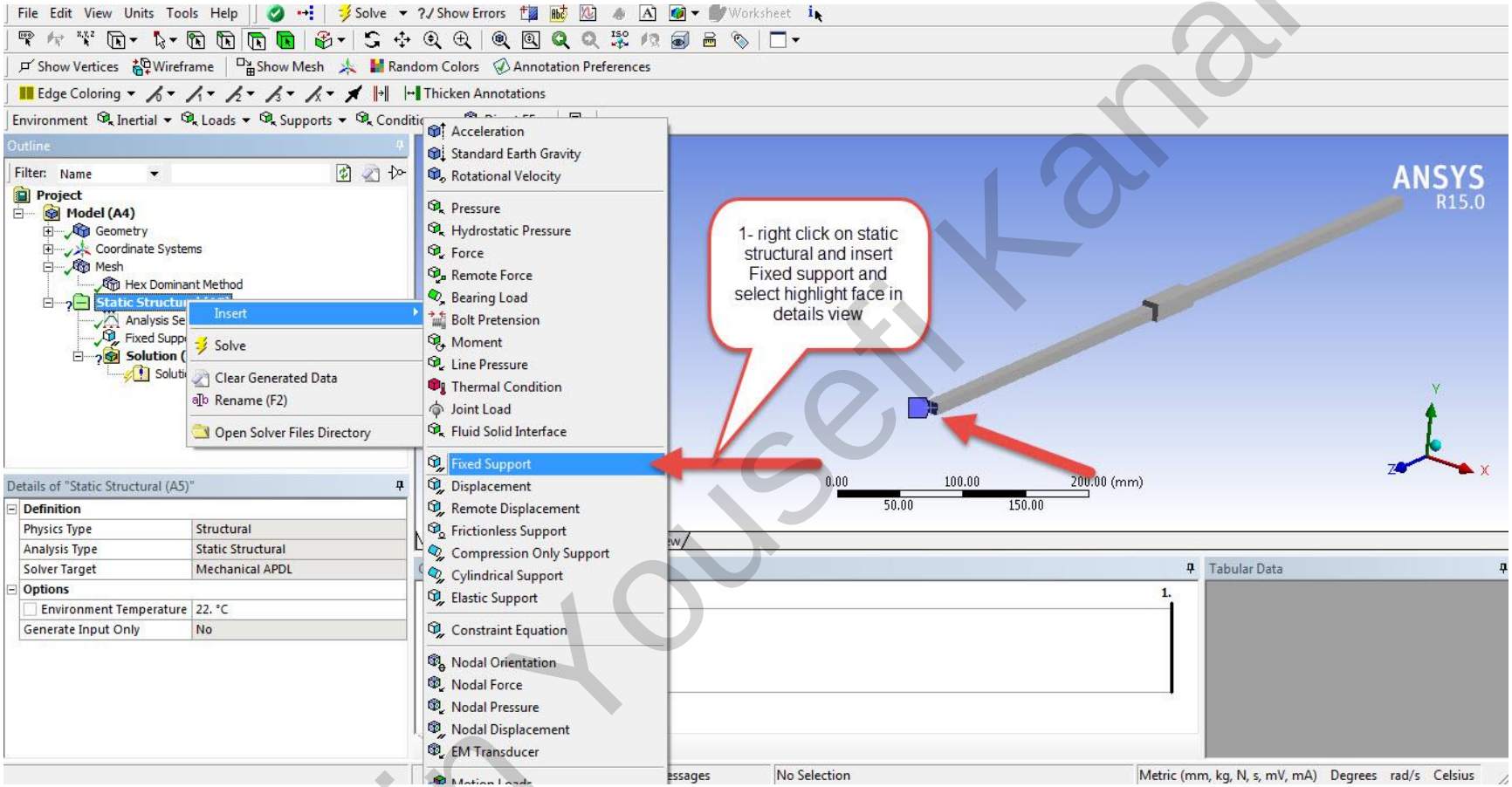
In details view select model from graphic view then change method to Hex Dominant

10-1 Mesh



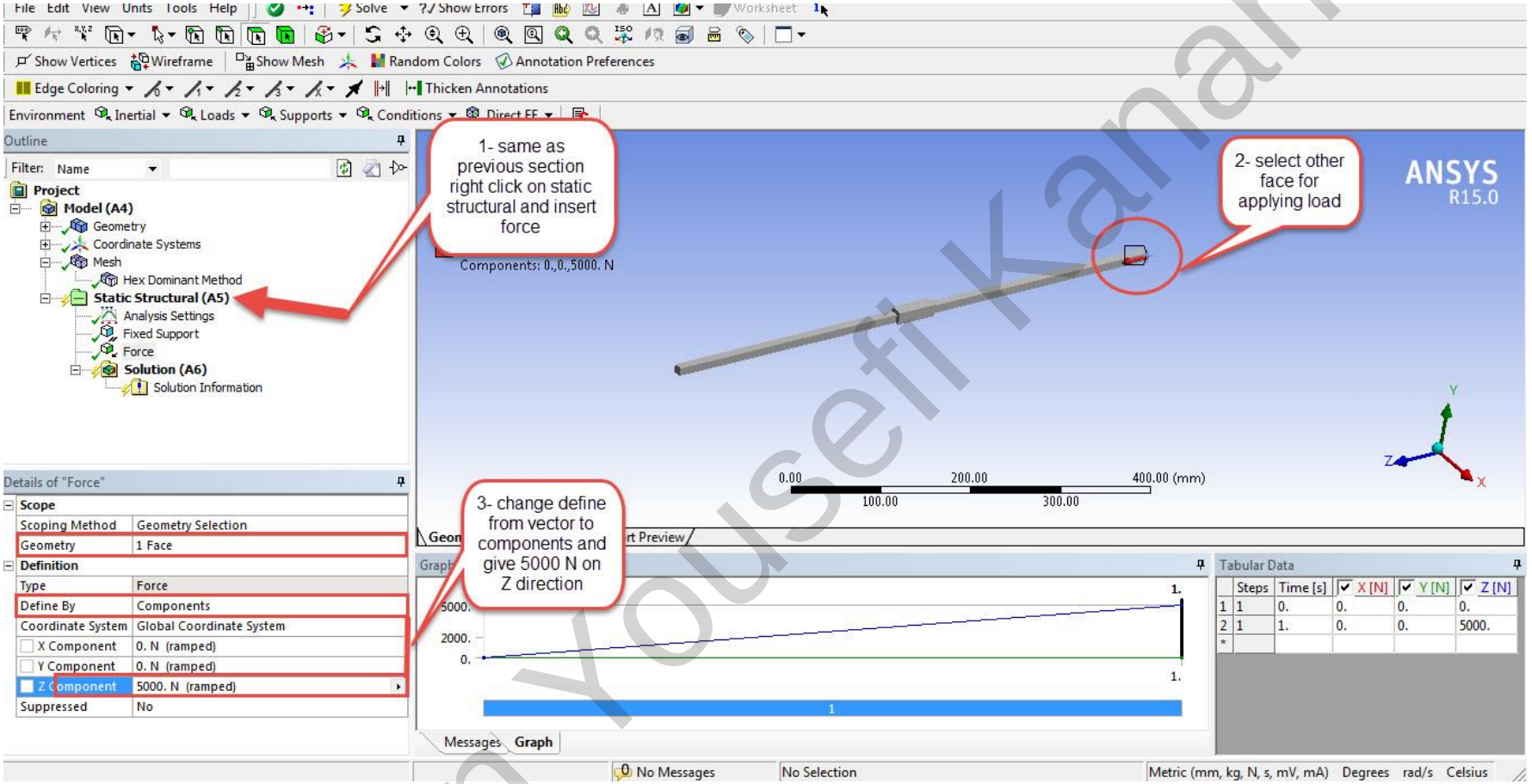
Again click on mesh and change element size form details view to 5 mm then click on generate mesh

11- Boundary condition



First right click on static structural and then insert Fixed Support. Then in details view select highlighted view as geometry and click on apply

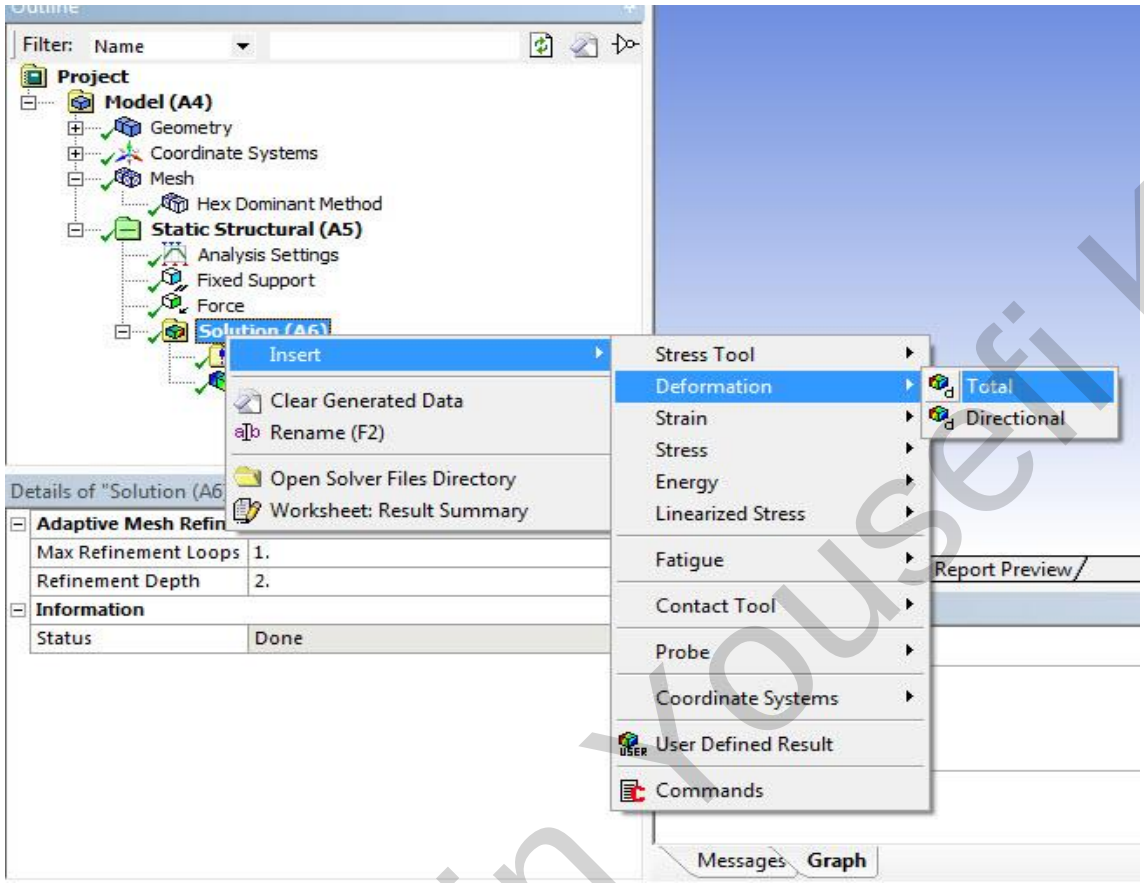
12- Force



Same as previous section insert force and then select other face for applying load then change define by to component and give 5000 N in Z direction for force

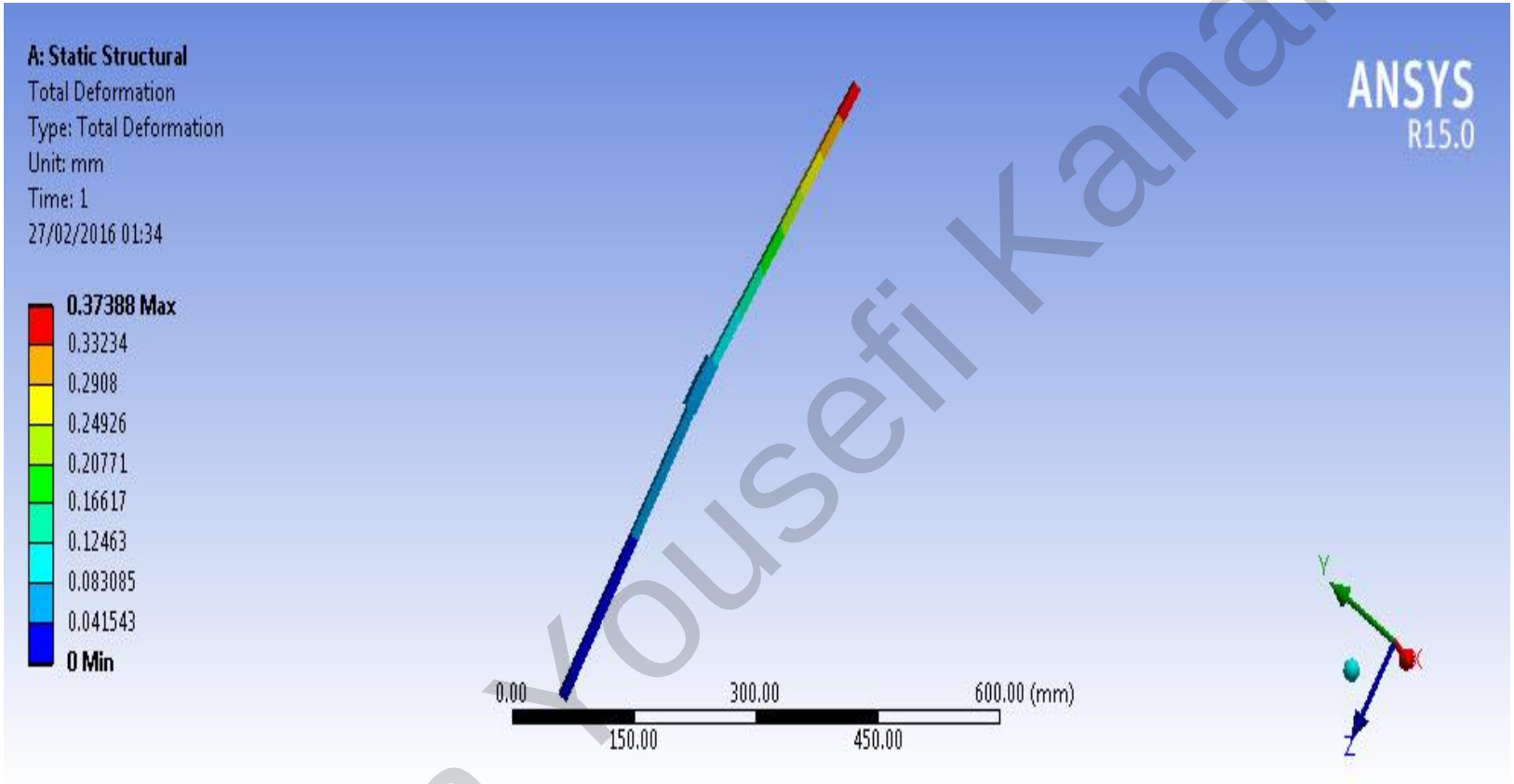
SOLUTION

13- Solution



Right click on solution and insert total deformation and click on solve

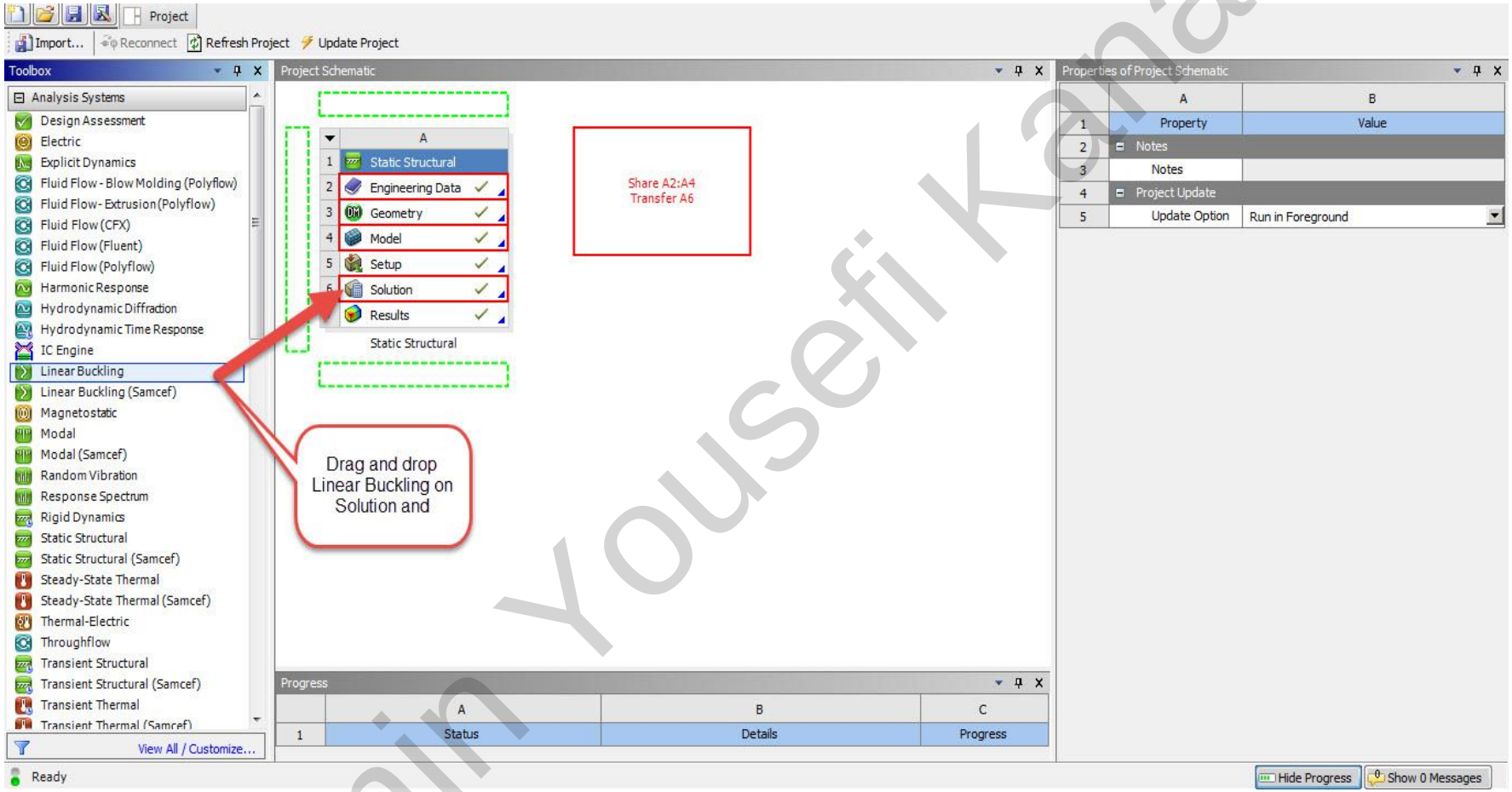
13- Result



Close Solution

Linear Buckling

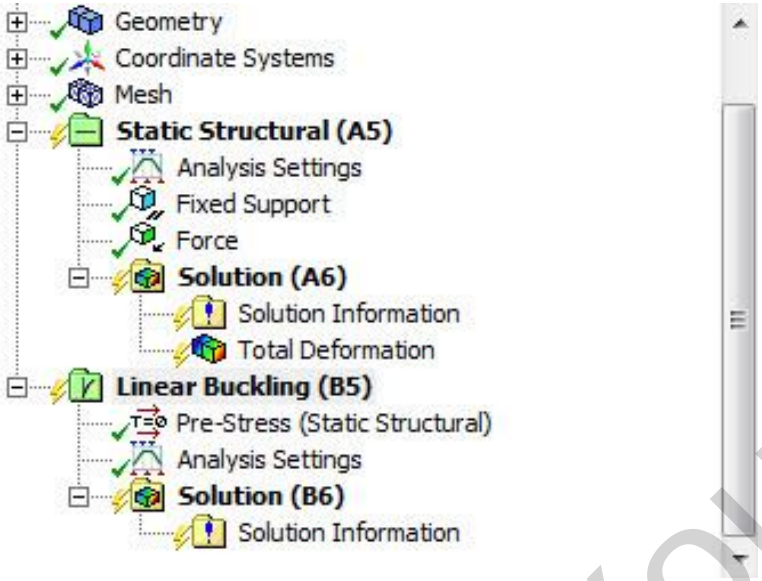
14- Open Linear Buckling



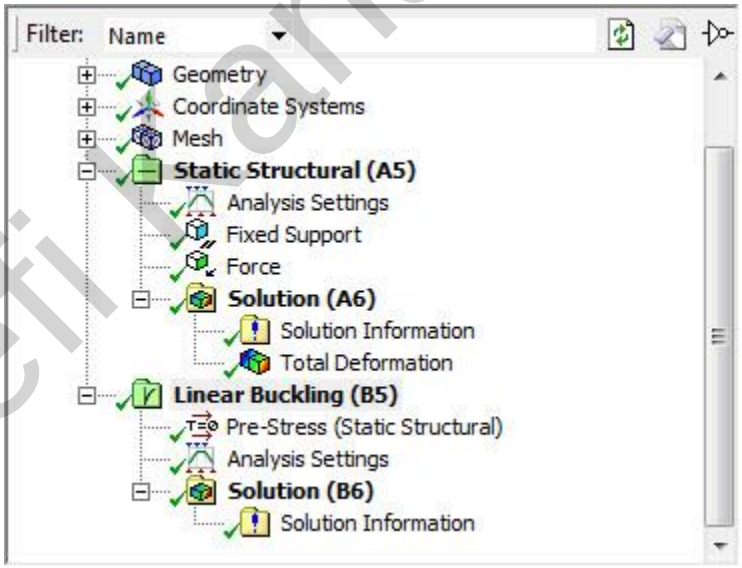
14-1 Linear Buckling

If you enter linear buckling correctly, you should have linear buckling as project B then double click on Setup

14-2 linear Buckling

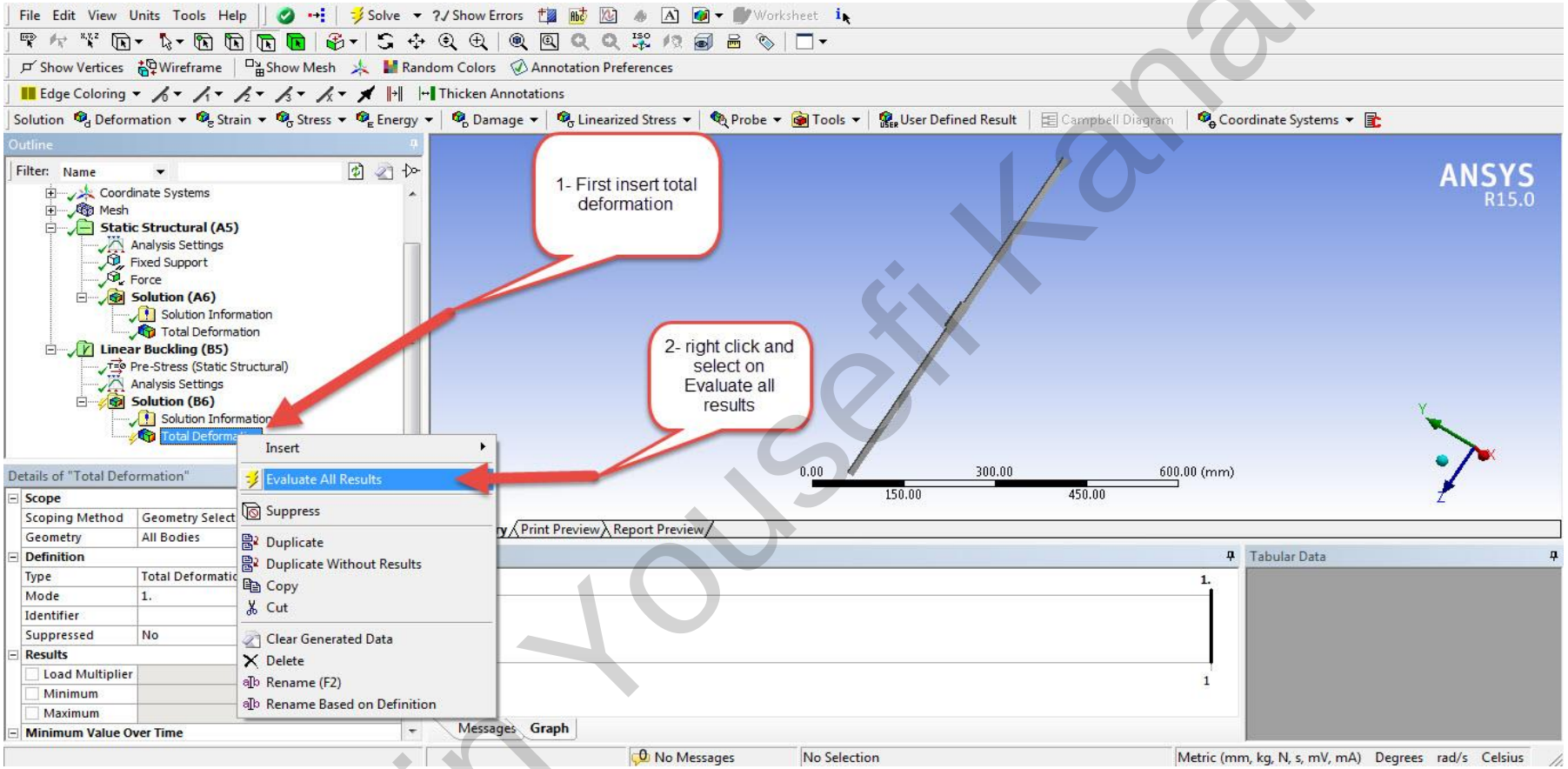


Click on solve without adding any solution



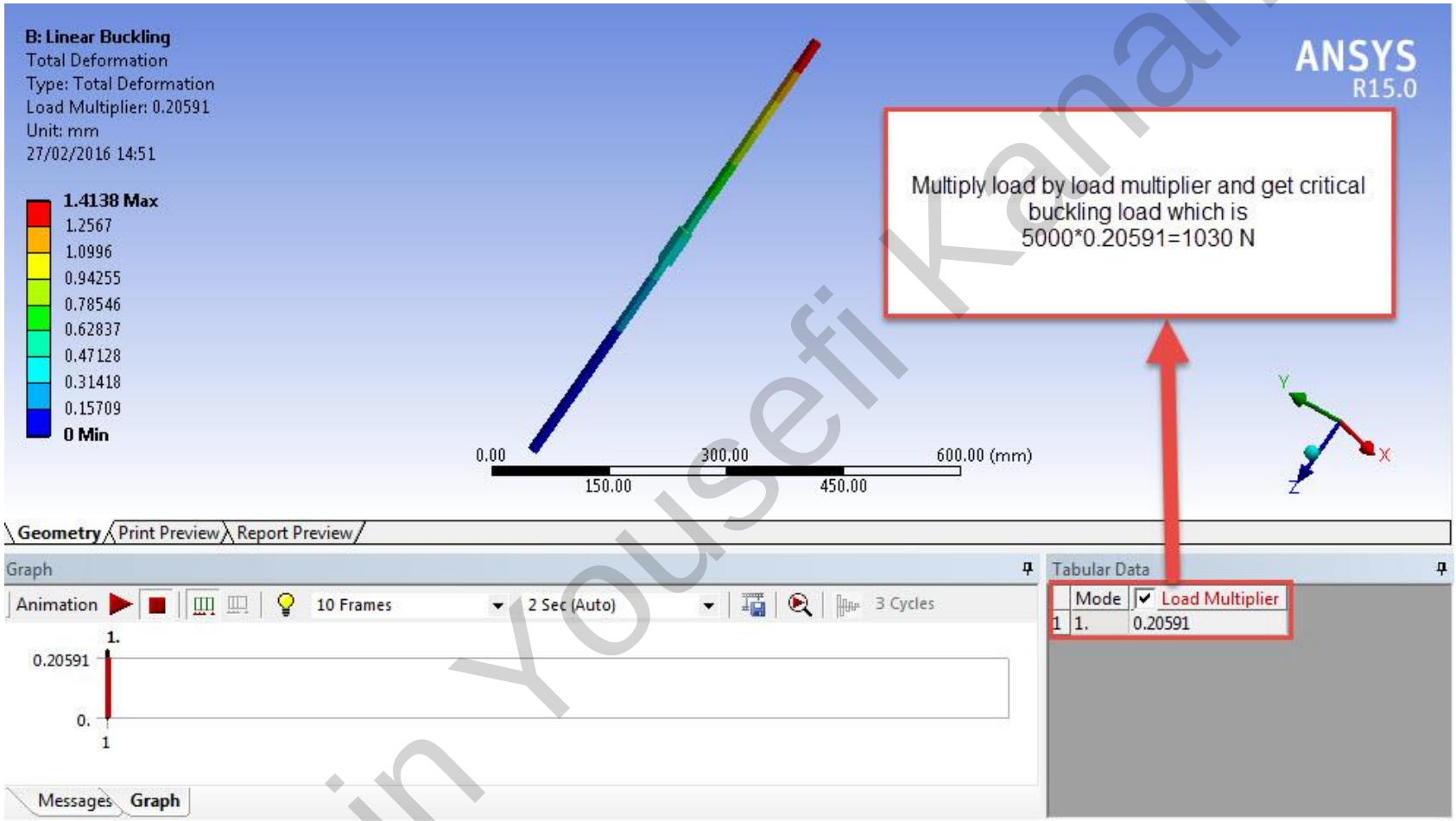
After solving

14-3 Linear Buckling



After solving, insert total deformation then right click on it and select evaluate all results

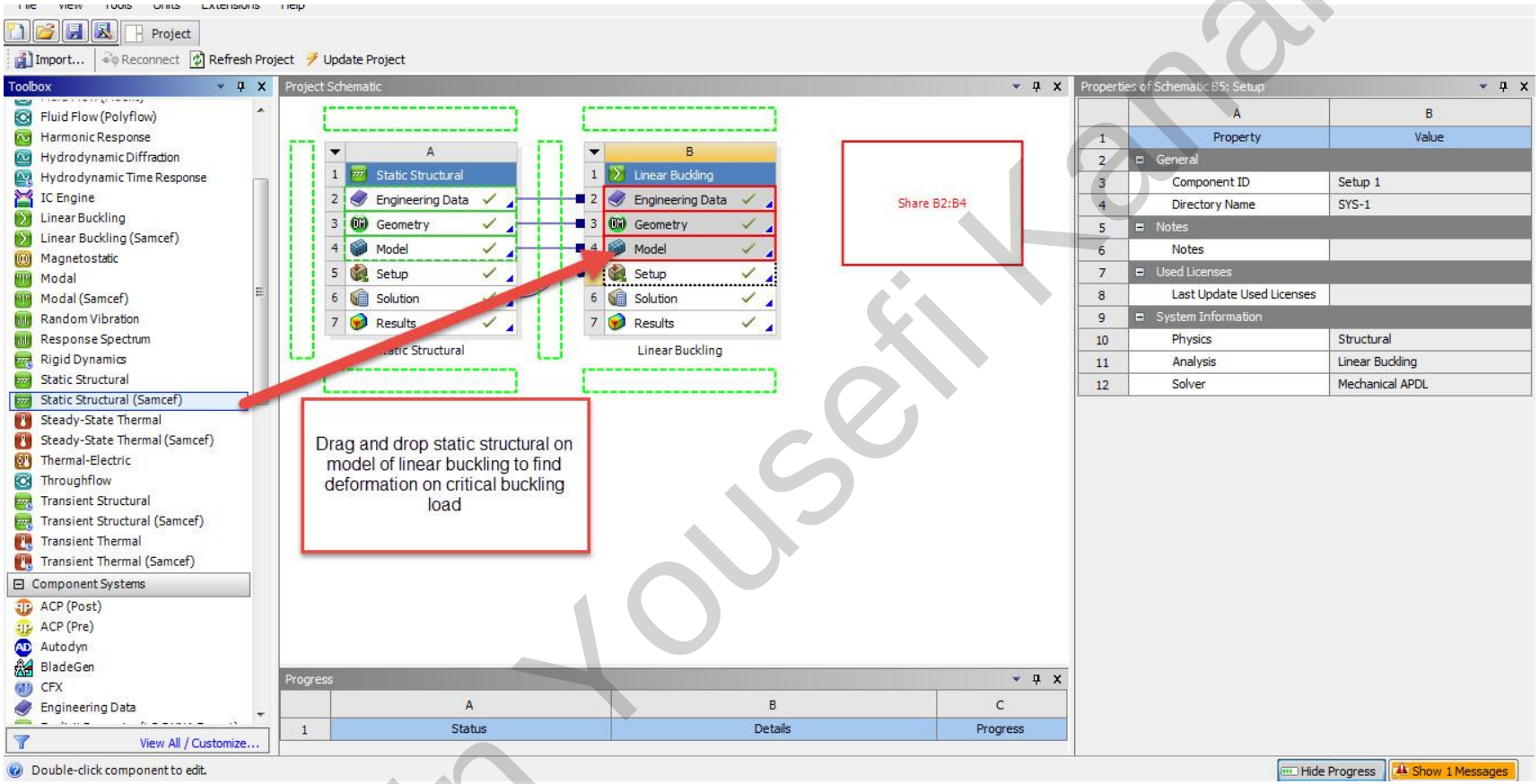
15- Load amount



After selecting evaluate all results, click on solve and you will see load multiplier on left side and finally close linear buckling. Do not forget to project

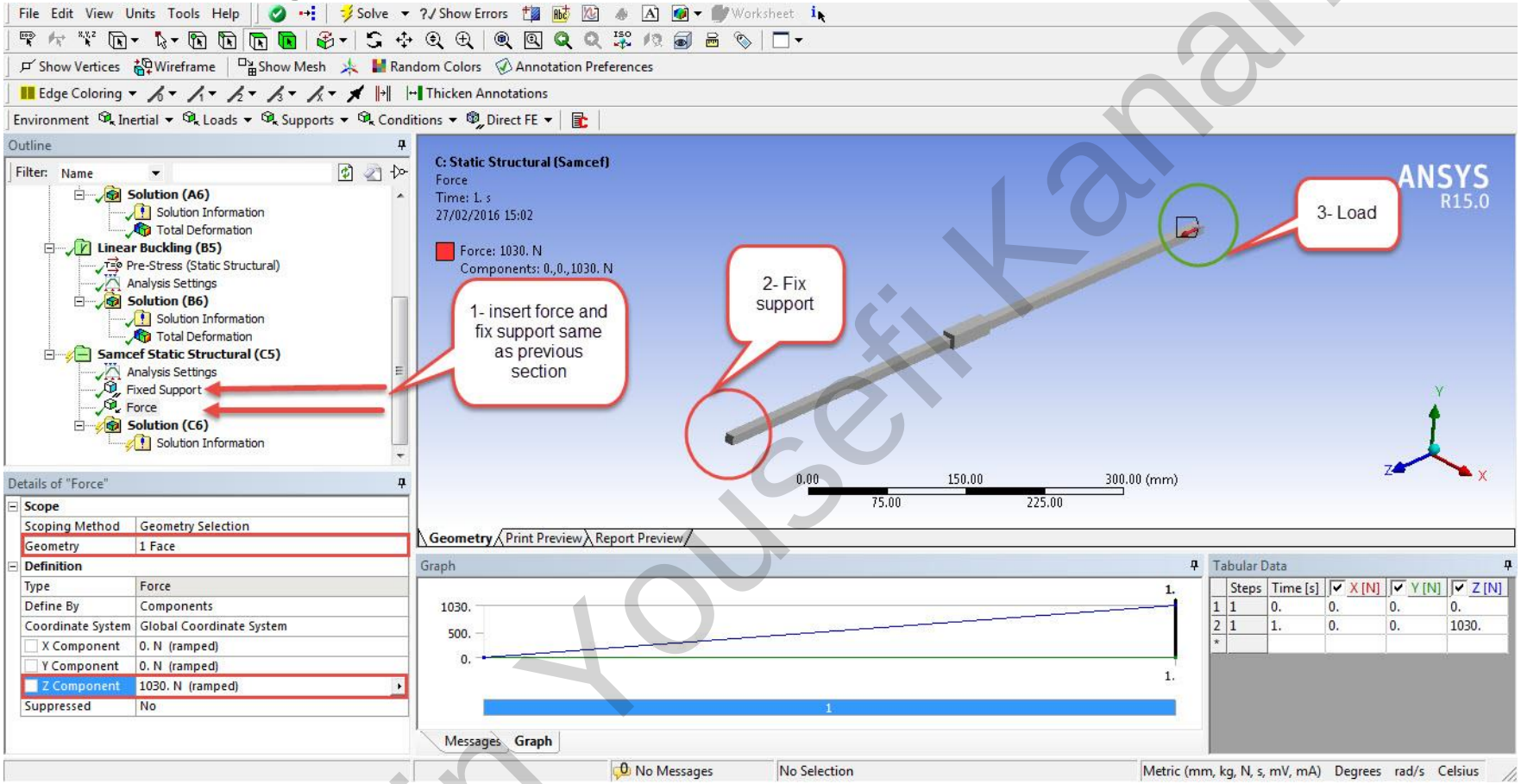
Finding deformation on critical buckling load

16- Deformation on critical buckling load



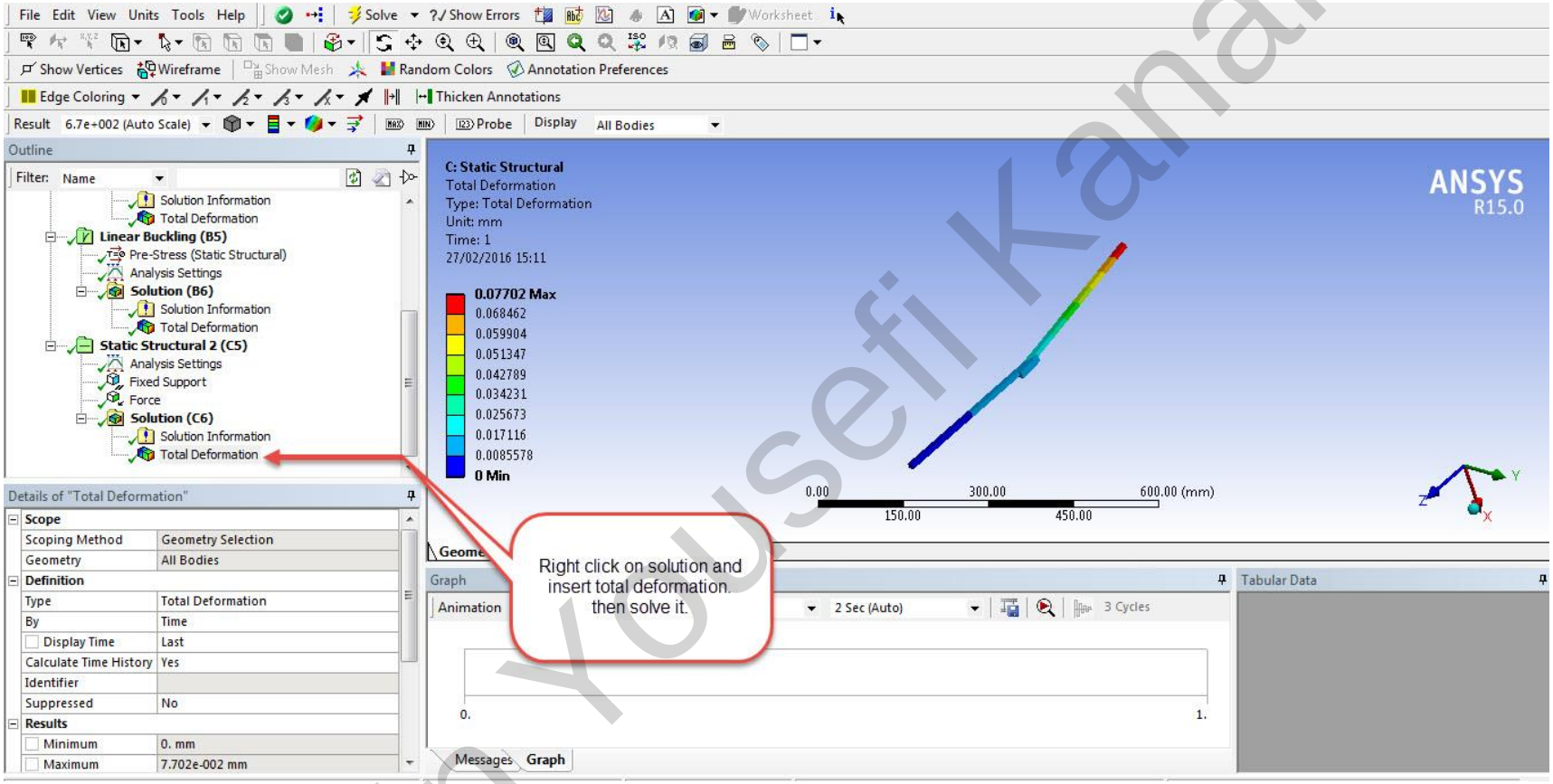
drag and drop static structural on model of linear buckling to find deformation on critical buckling load

17- Boundary condition and Load



After opening model, insert fixed and force same as previous slides and select highlighted faces. Critical buckling load was calculated from linear buckling 1030 N in Z direction

18-Solution



Right click on solution and insert total deformation. then solve it.

FINISH