

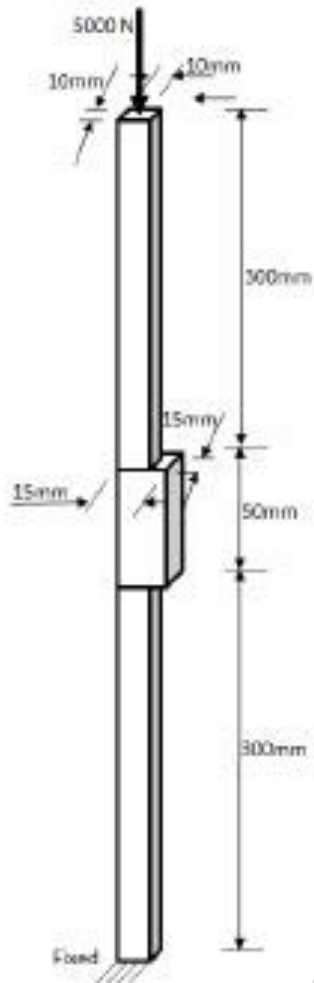
Buckling Analysis of a Column in ANSYS

Armin Yousefi Kanani



DESIGN MODELER

The information of question

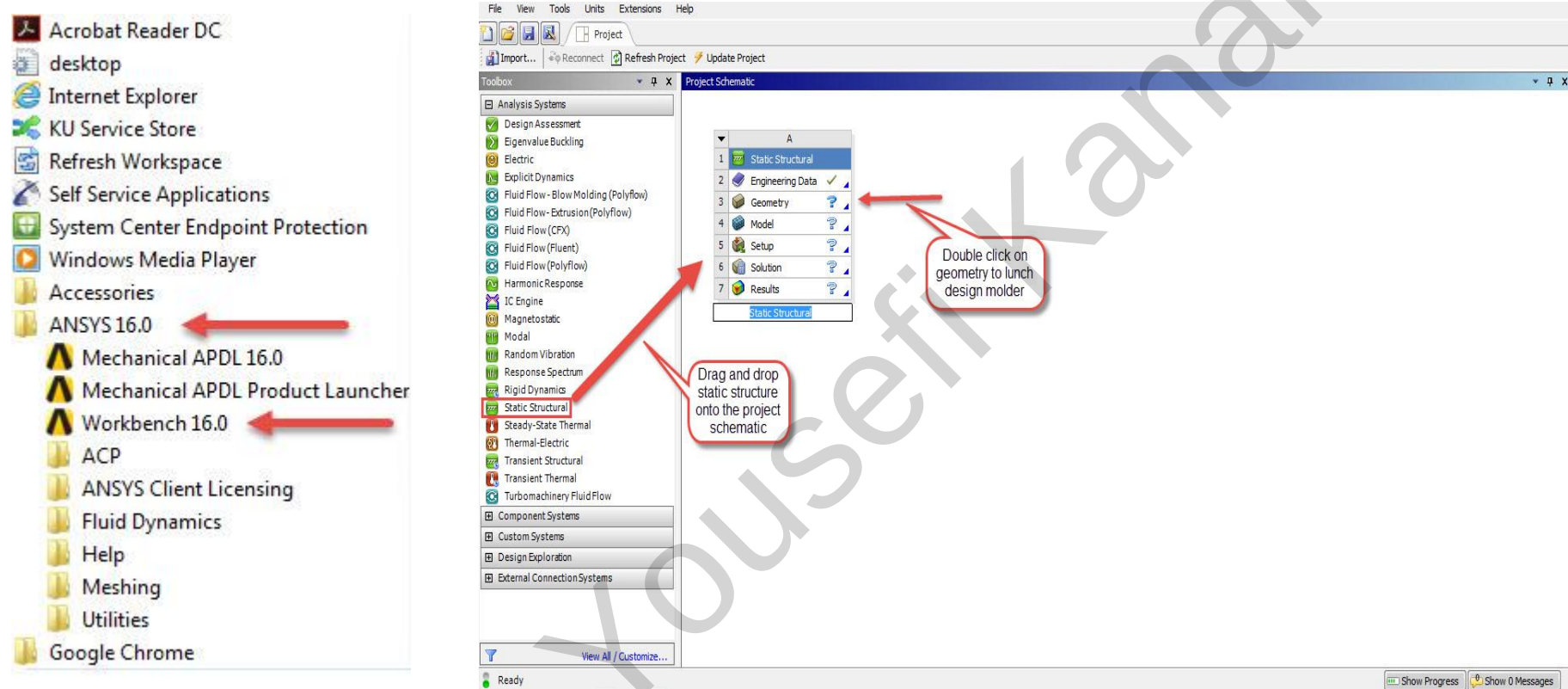


Young's modulus, $E = 2.1 \times 10^5 \text{ N/mm}^2$

Poisson's ratio, $\nu = 0.3$

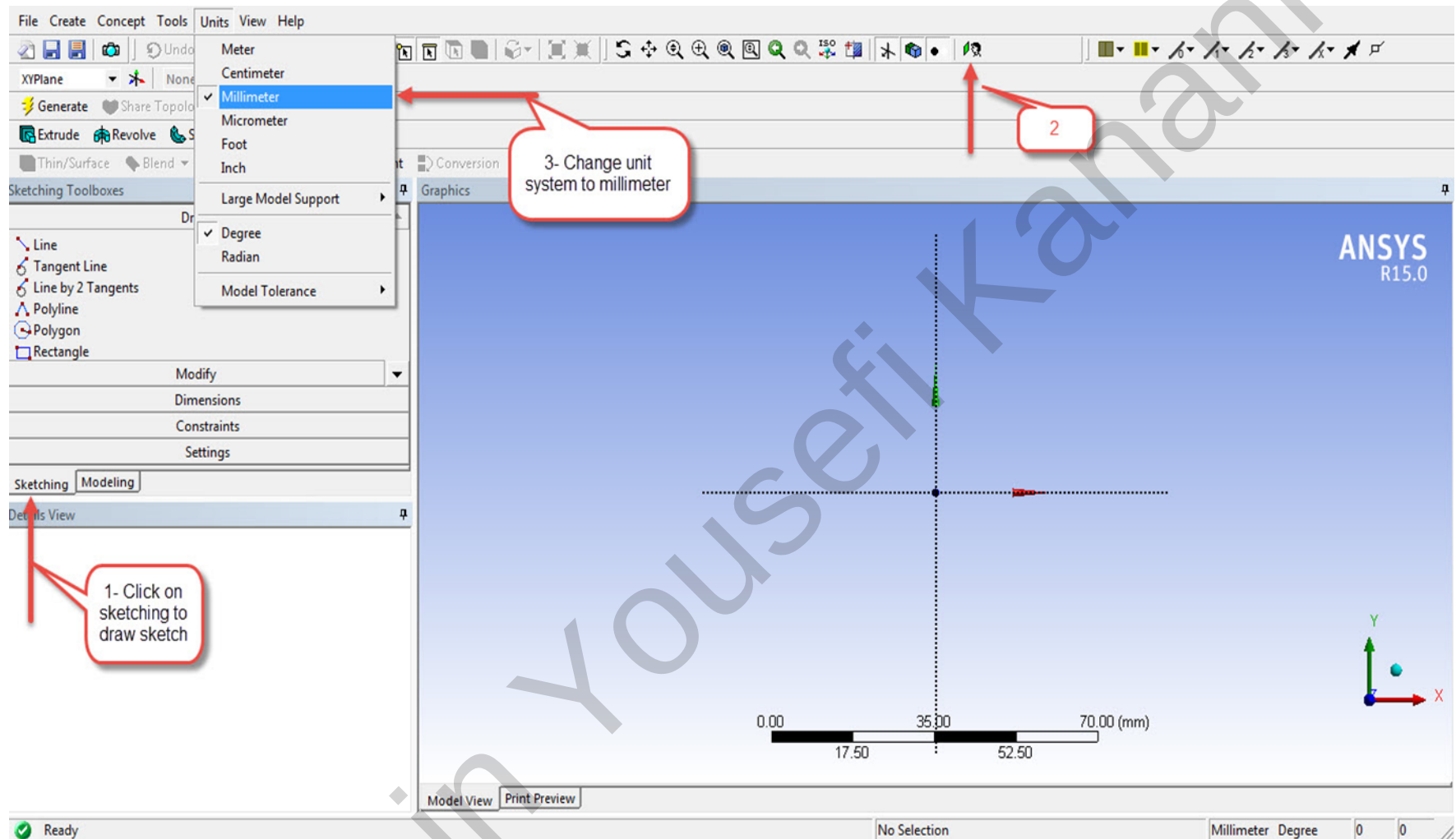
Yield strength = 250 N/mm^2

2- Open Workbench

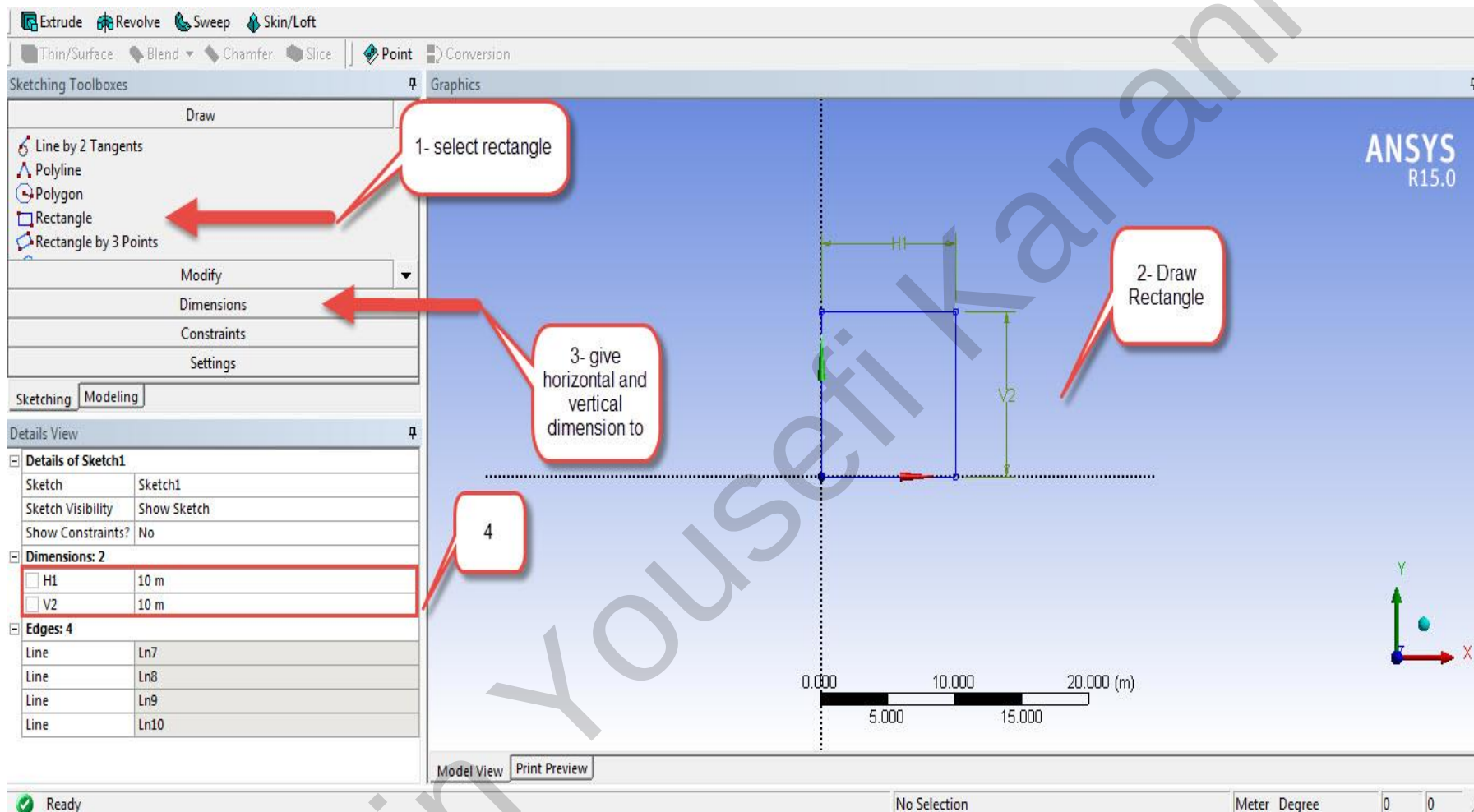


Open ANSYS workbench and drag and drop static structure and double click on geometry to lunch design molder

3- Design Modeler

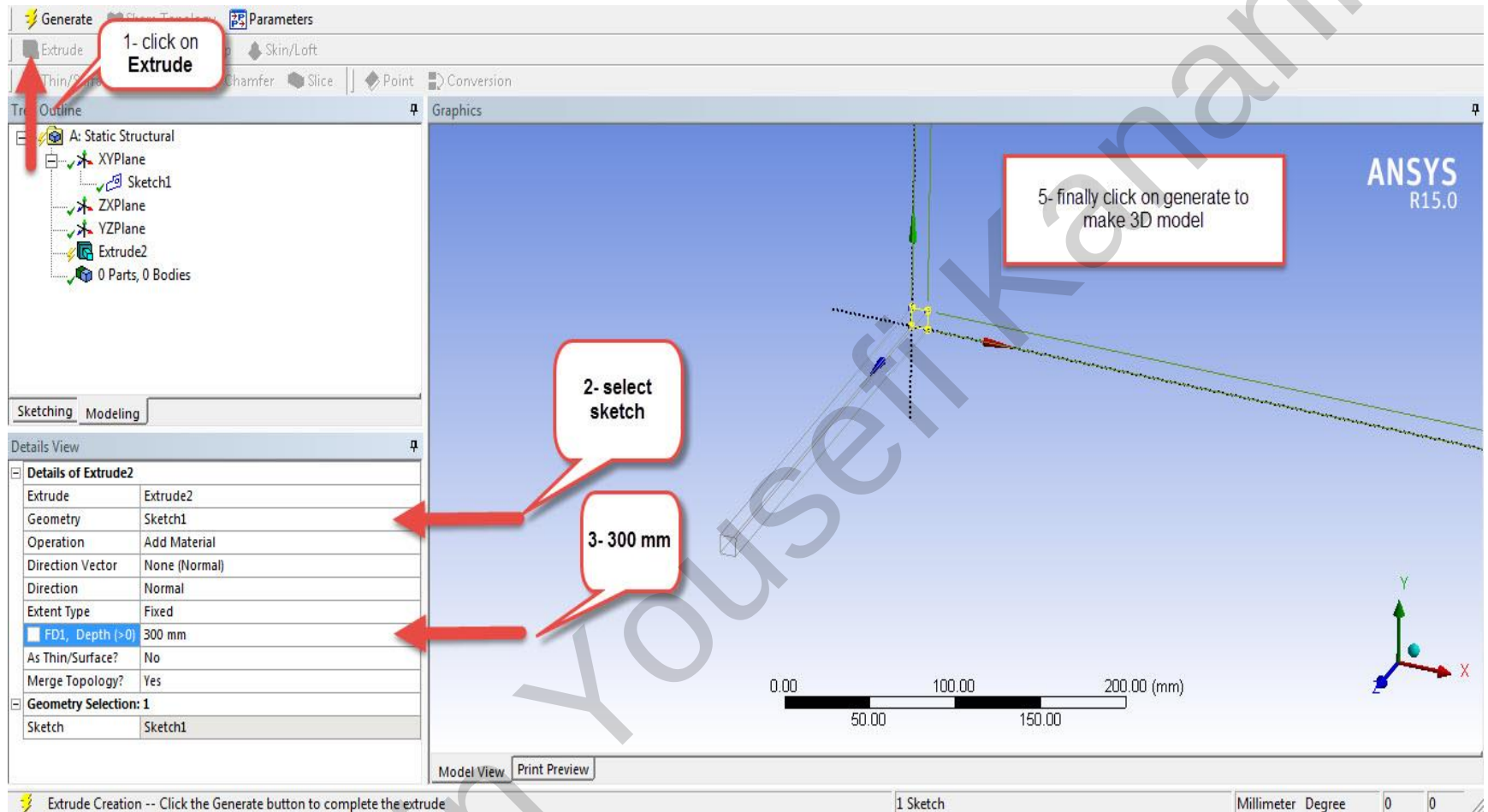


4- Sketch



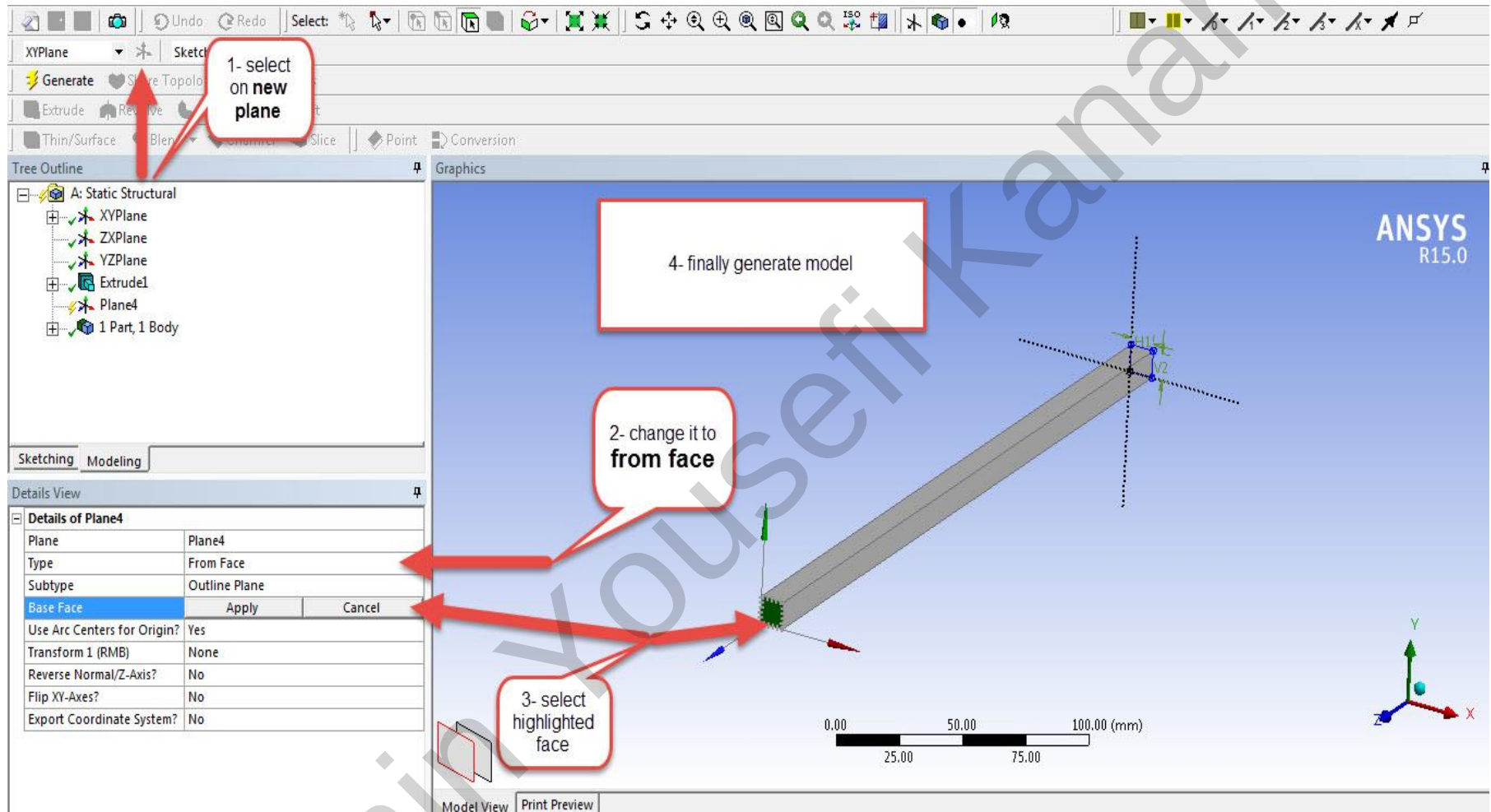
Select rectangle from draw toolbars then give horizontal and vertical dimension to rectangle L:10 mm, w: 10 mm

5- Extrude

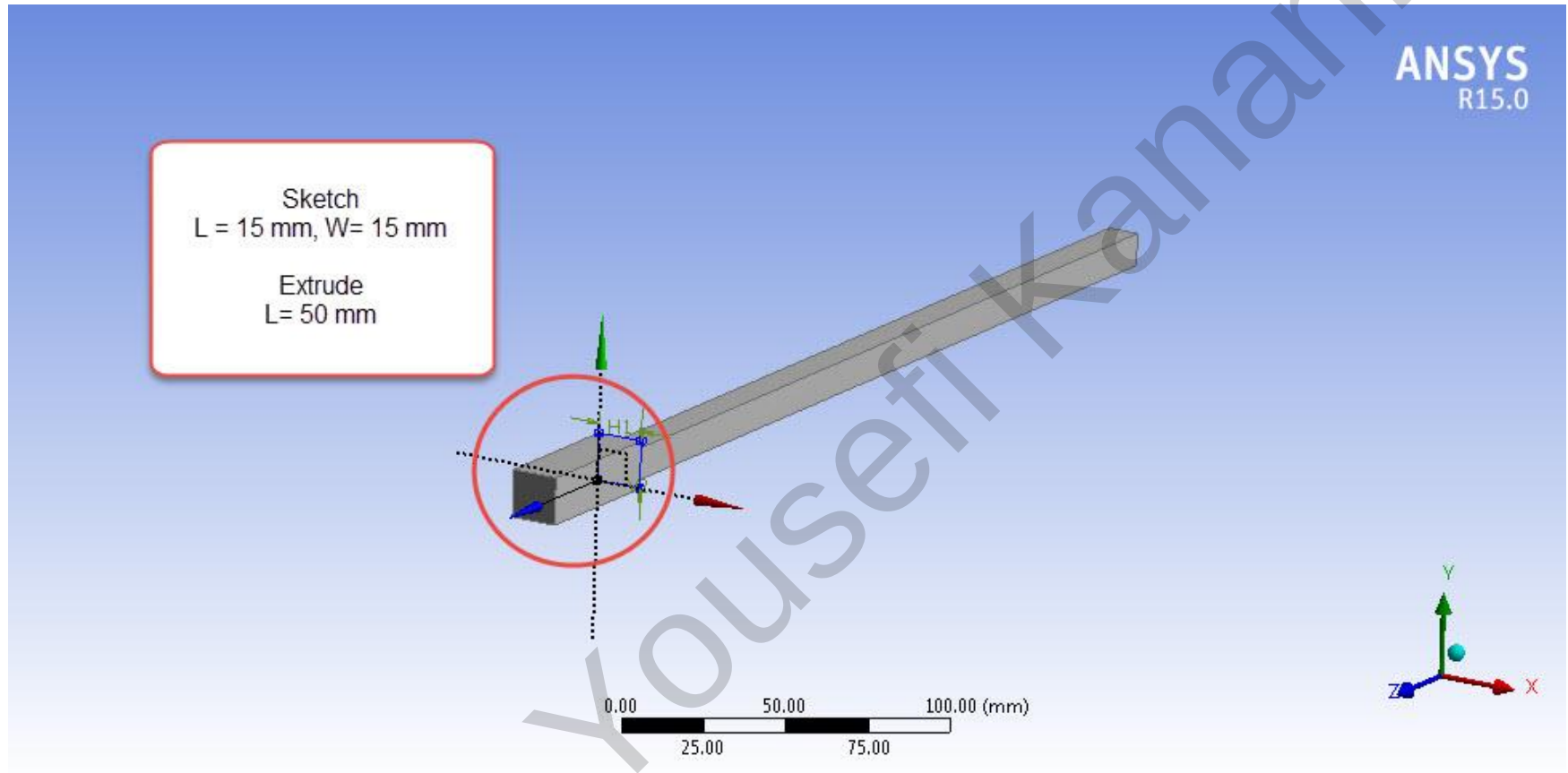


First click on extrude and select sketch from graphic view. Then give 300 mm depth and generate 3d model

6- Making a New Plane

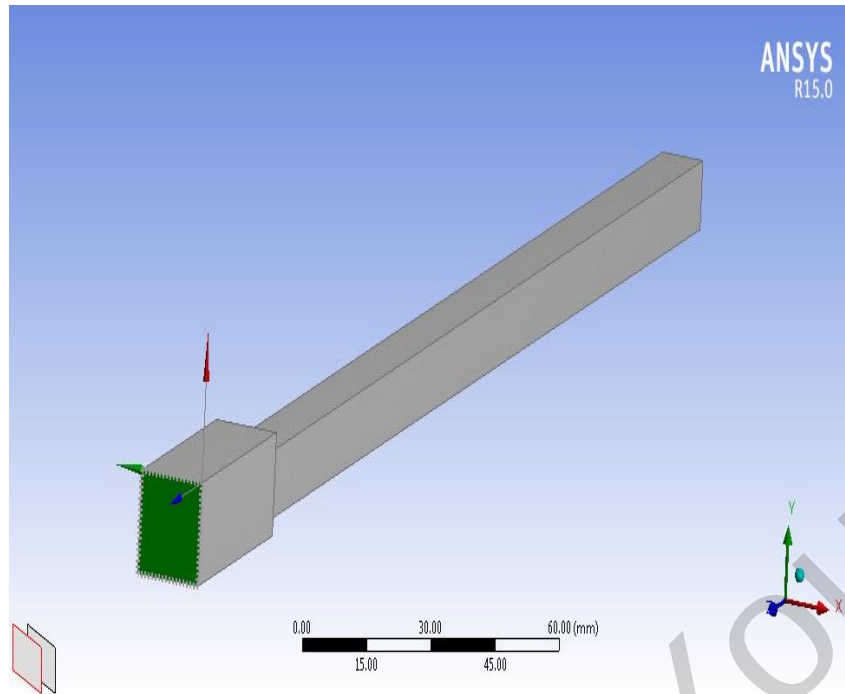


7- Geometry-Section 2

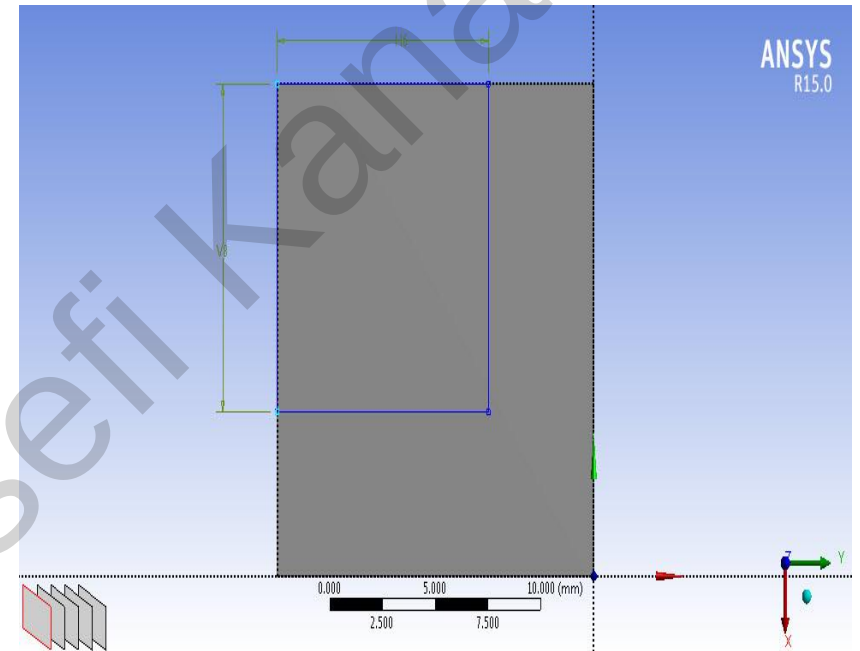


Now we are going to make another rectangle on new plane. Same as previous slides. **1- select on new plane** **2- draw rectangle in size of L: 15 mm W: 15 mm** **3- Extrude it by 50 mm**

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

File View Tools Units Extensions Help

Project

Import... Reconnect Refresh Project Update Project

Toolbox

Analysis Systems

- Design Assessment
- Electric
- Explicit Dynamics
- Fluid Flow - Blow Molding (Polyflow)
- Fluid Flow - Extrusion (Polyflow)
- Fluid Flow (CFX)
- Fluid Flow (Fluent)
- Fluid Flow (Polyflow)
- Harmonic Response
- Hydrodynamic Diffraction
- Hydrodynamic Time Response
- IC Engine
- Linear Buckling
- Linear Buckling (Samcef)
- Magnetostatic
- Modal
- Modal (Samcef)
- Random Vibration
- Response Spectrum
- Rigid Dynamics
- Static Structural
- Static Structural (Samcef)
- Steady-State Thermal
- Steady-State Thermal (Samcef)
- Thermal-Electric
- Throughflow
- Transient Structural
- Transient Structural (Samcef)
- Transient Thermal
- Transient Thermal (Samcef)

Project Schematic

A

- Static Structural
- Engineering Data
- Geometry
- Model
- Setup
- Solution
- Results

Static Structural

Messages

	A	B	C	D
1	Type	Text	Association	Date/Time

Progress

	A	B	C
1	Status	Details	Progress

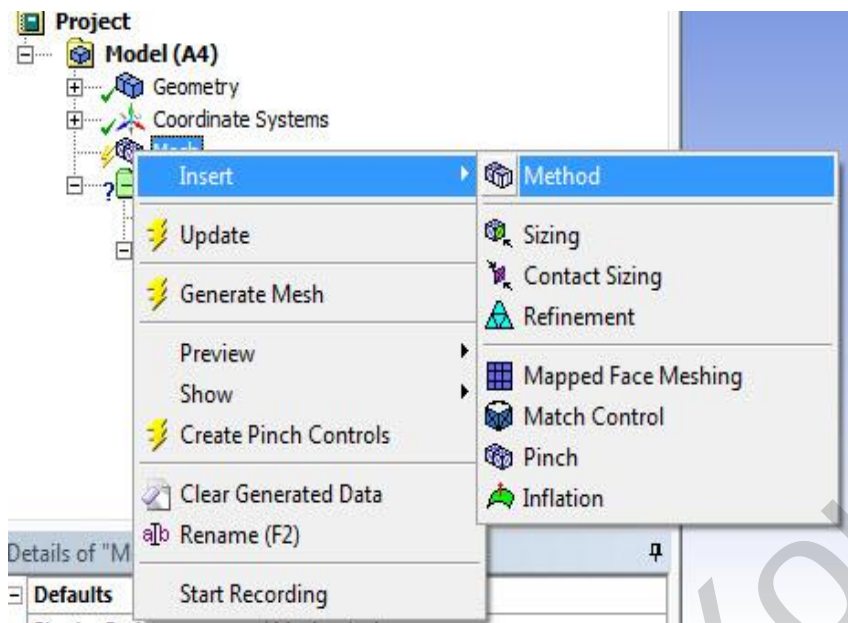
Properties of Schematic A3: Geometry

	A	B
1	Property	Value
2	General	
3	Component ID	Geometry
4	Directory Name	SYS
5	Notes	
6	Notes	
7	Used Licenses	
8	Last Update Used Licenses	
9	Geometry Source	
10	Geometry File Name	C:\Users\1\AppData\Local\Temp\WB_1+P_1_9212_2\unsaved_project_files\dp0\SYS\DM\SYS.agdb
11	CAD Plug-In	DesignModeler[4456]
12	Basic Geometry Options	
13	Attributes	☐
14	Material Properties	☐
15	Advanced Geometry Options	
16	Analysis Type	3D
17	Use Associativity	☒
18	Import Coordinate Systems	☐
19	Import Work Points	☐
20	Reader Mode Saves Updated File	☐
21	Import Using Instances	☒
22	Smart CAD Update	☐
23	Compare Parts On Update	No
24	Enclosure and Symmetry Processing	☒
25	Decompose Disjoint Geometry	☒

Double-click component to edit.

Hide Progress Hide 0 Messages

11- Mesh



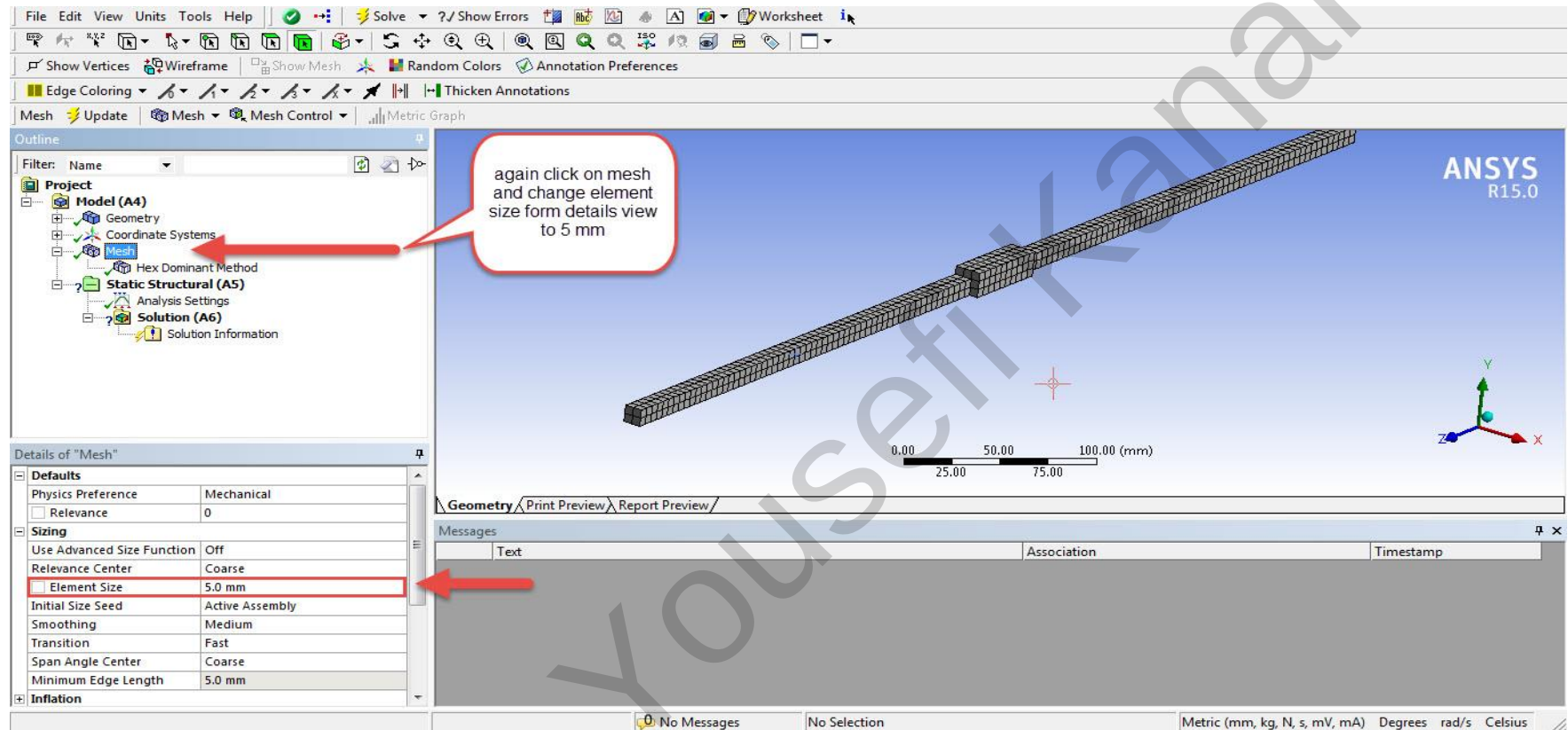
Right click on mesh and insert method

[-] Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
[-] Definition	
Suppressed	No
Method	Hex Dominant
Element Midside Nodes	Use Global Setting
Free Face Mesh Type	Quad/Tri
Control Messages	Yes, Click To Display...

Select model from graphic view then change method to Hex Dominant

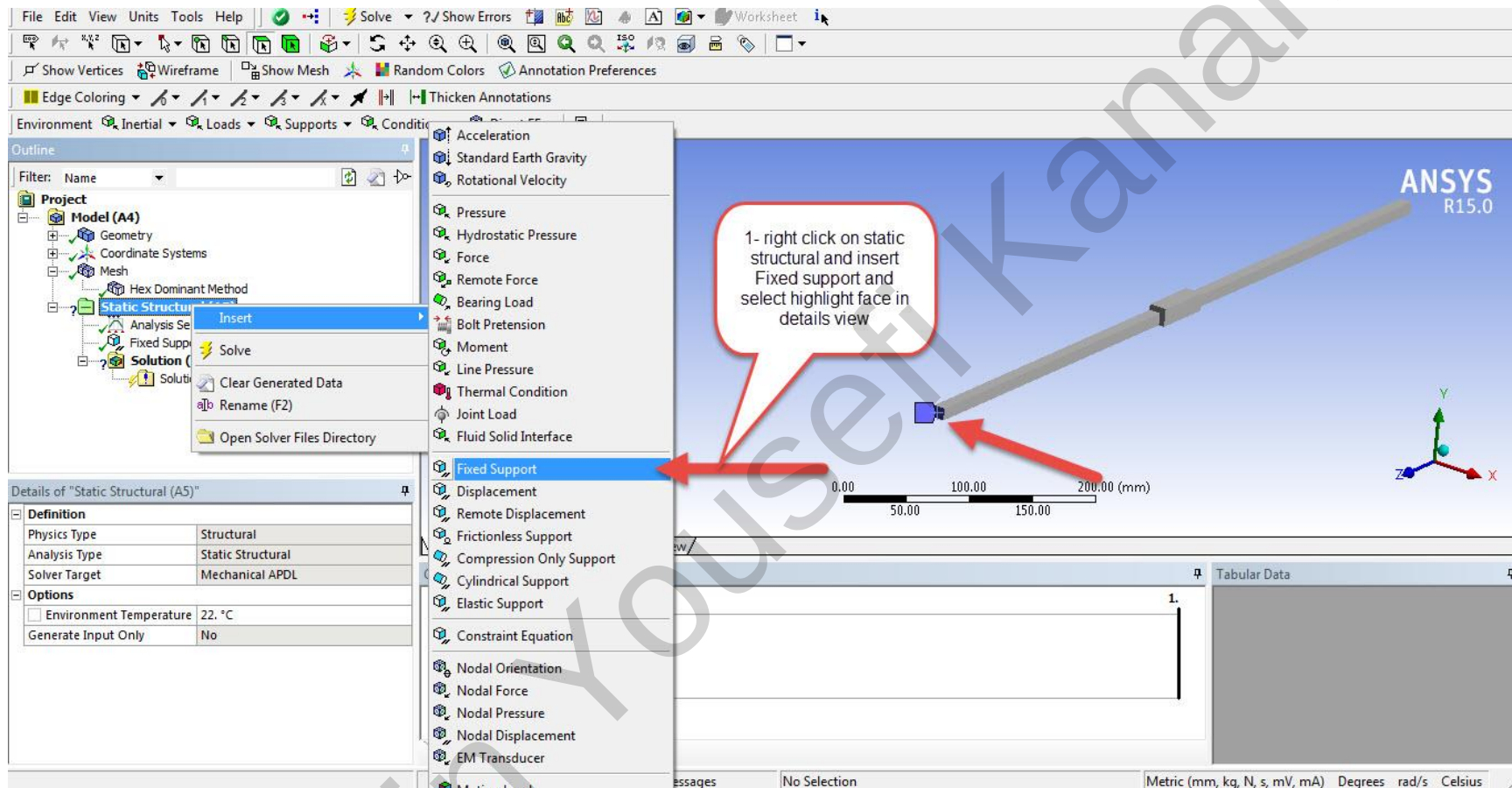
In details view select model from graphic view then change method to Hex Dominant

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

1- same as previous section right click on static structural and insert force

2- select other face for applying load

3- change define from vector to components and give 5000 N on Z direction

Components: 0., 0., 5000. N

0.00 100.00 200.00 300.00 400.00 (mm)

5000. 2000. 0.

1.

1.

Messages Graph

No Messages No Selection Metric (mm, kg, N, s, mV, mA) Degrees rad/s Celsius

Details of "Force"

Scope

Scoping Method Geometry Selection

Geometry 1 Face

Definition

Type Force

Define By Components

Coordinate System Global Coordinate System

☐ X Component 0. N (ramped)

☐ Y Component 0. N (ramped)

☒ Z Component 5000. N (ramped)

Suppressed No

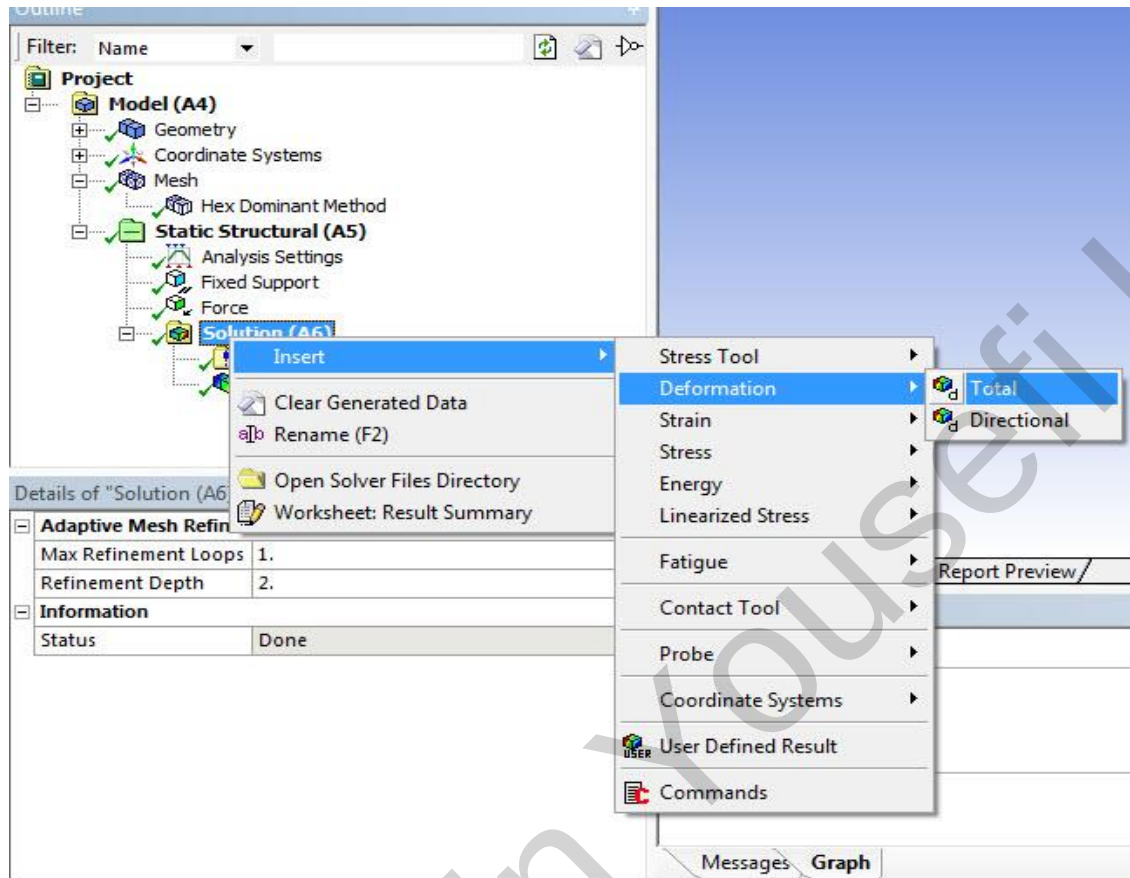
Tabular Data

Steps	Time [s]	X [N]	Y [N]	Z [N]
1	0.	0.	0.	0.
2	1.	0.	0.	5000.
*				

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

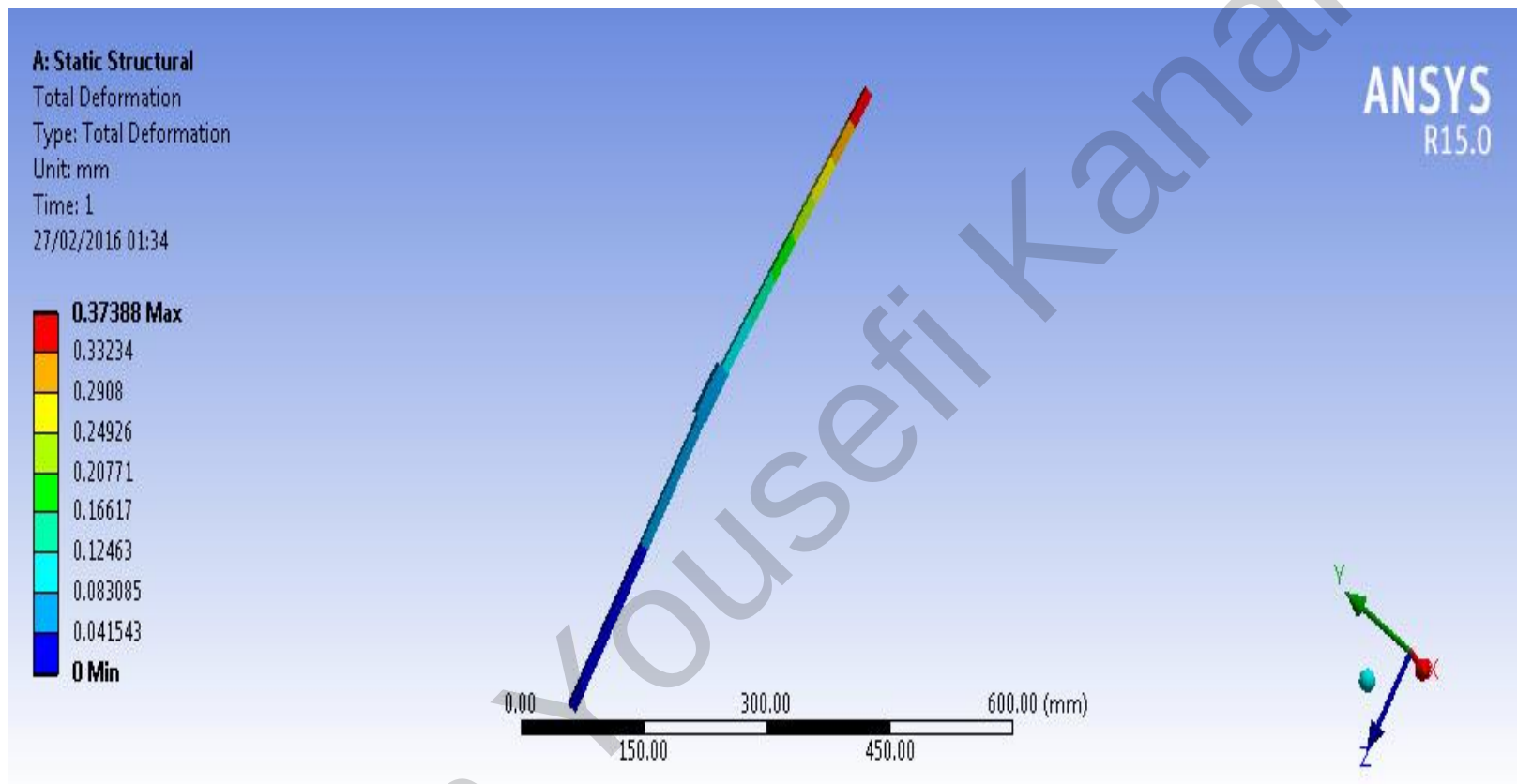
SOLUTION

13- Solution



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

13- Result



Close Solution

Linear Buckling

14- Open Linear Buckling

Drag and drop Linear Buckling on Solution and

Share A2:A4 Transfer A6

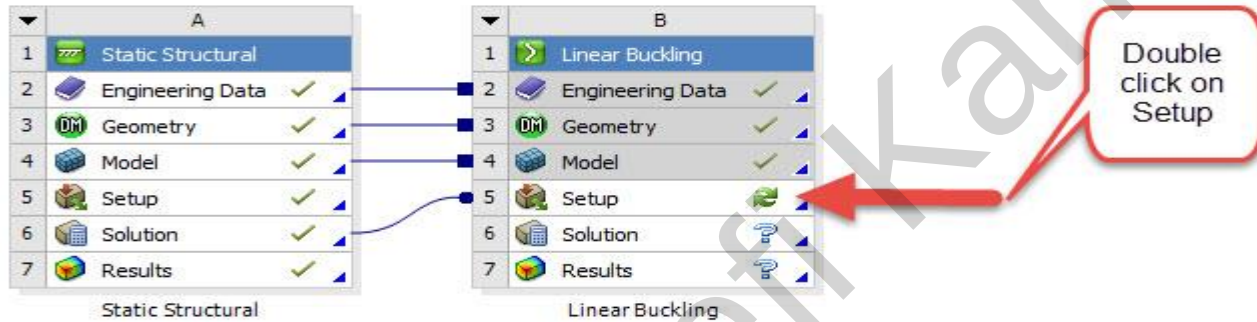
	A	B
1	Property	Value
2	Notes	
3	Notes	
4	Project Update	
5	Update Option	Run in Foreground

	A	B	C
1	Status	Details	Progress

Ready

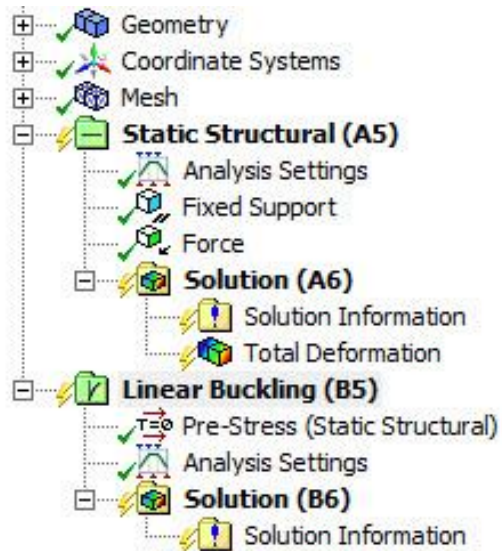
Hide Progress Show 0 Messages

14-1 Linear Buckling

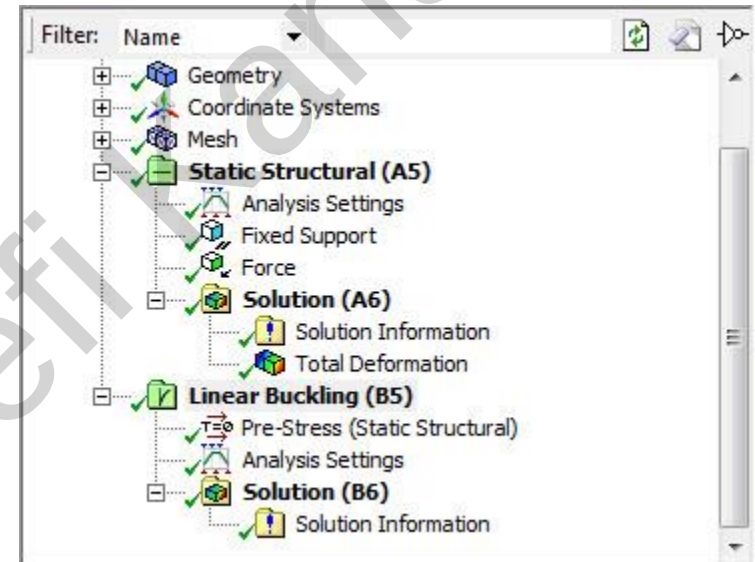


If you enter linear buckling correctly, you should have linear buckling as project B then double click on Setup

14-2 linear Buckling

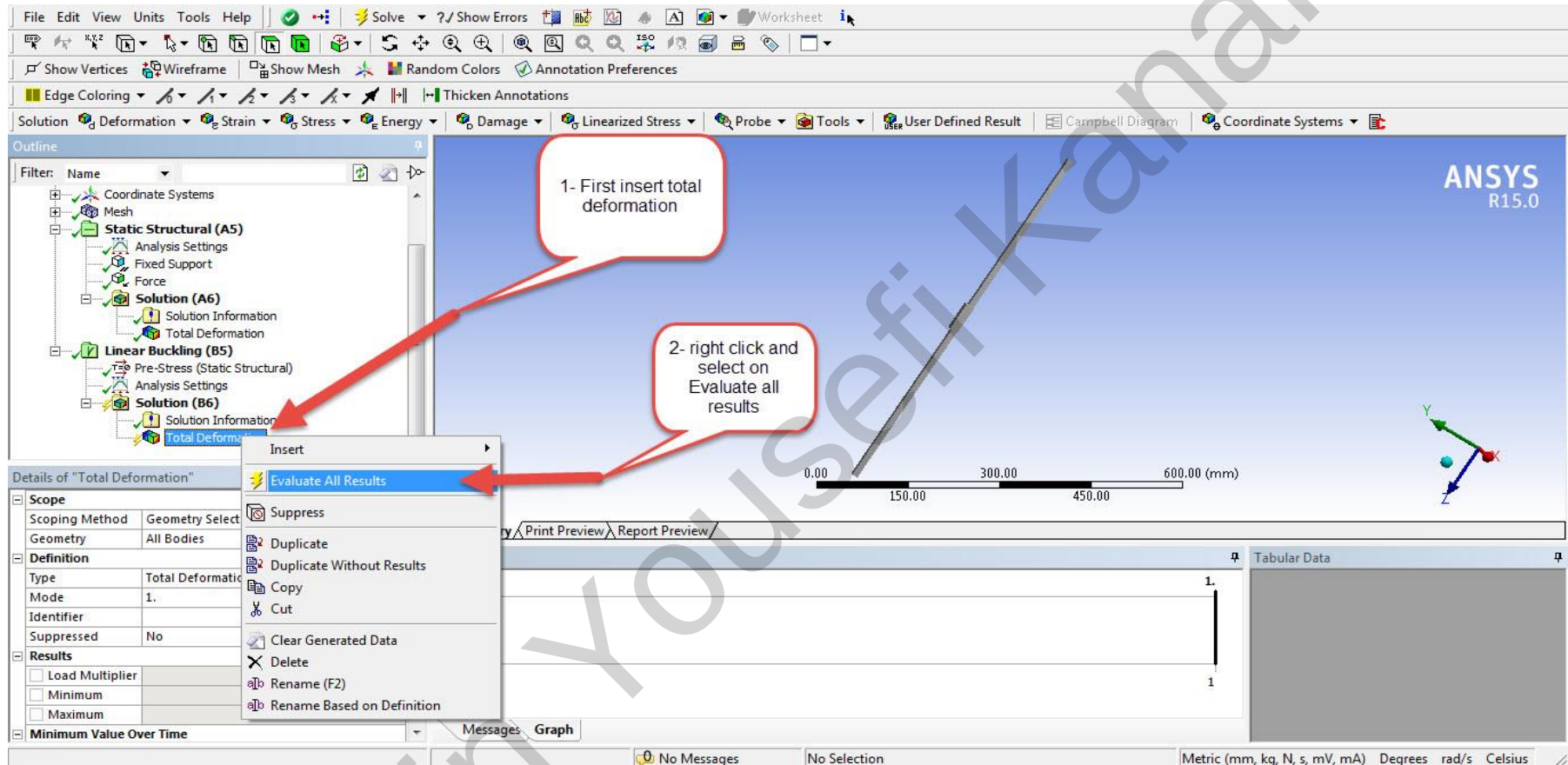


Click on solve without adding any solution



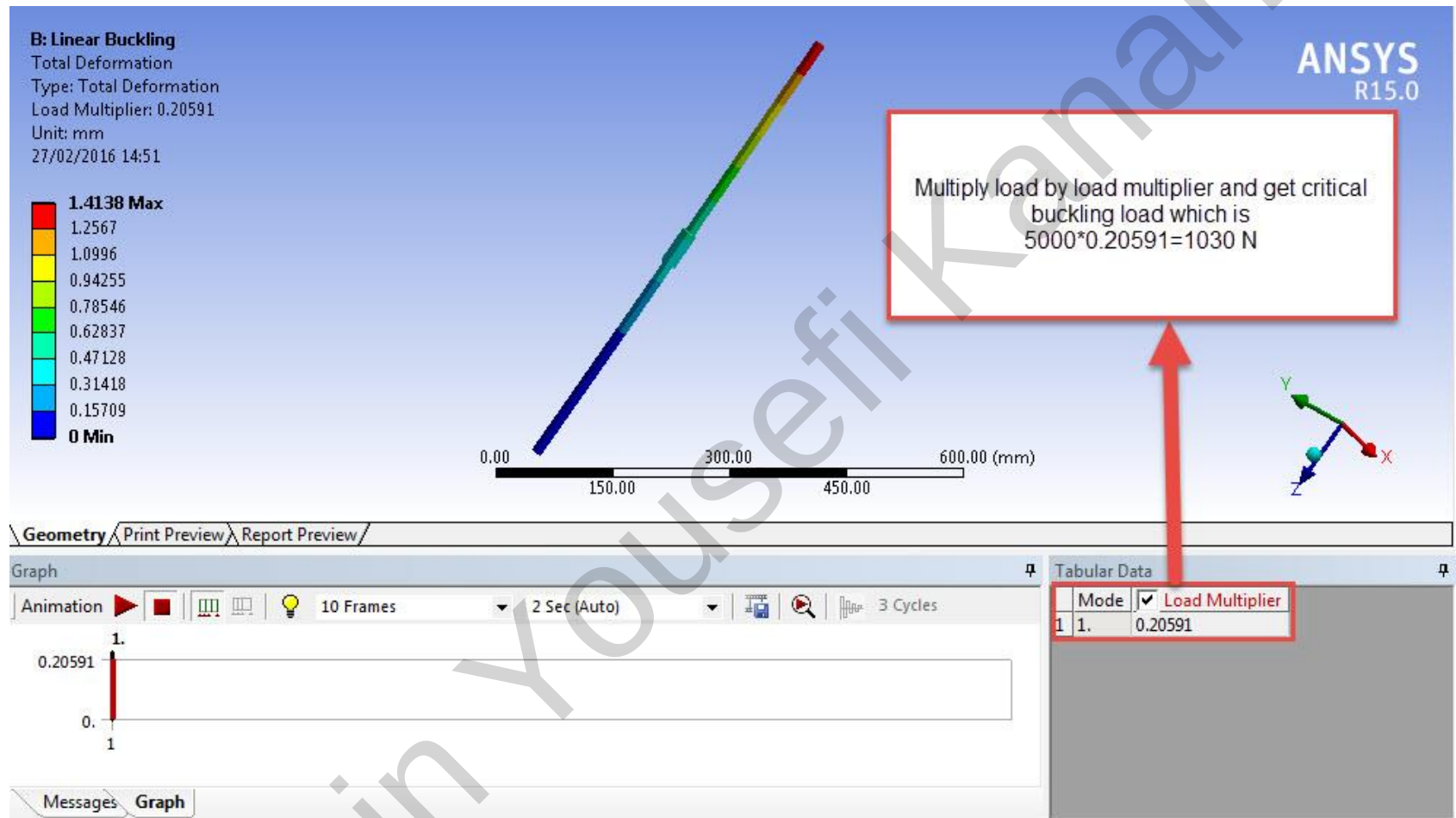
After solving

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

The screenshot displays the ANSYS Workbench Project Schematic. On the left, the Toolbox contains various analysis types. A red arrow points from 'Static Structural (Samcef)' in the Toolbox to the 'Model' component of the 'Linear Buckling' analysis in the Project Schematic. A red box highlights the 'Model' component in the Linear Buckling analysis, and another red box highlights the 'Static Structural' component in the Project Schematic. A text box with a red border contains the instruction: 'Drag and drop static structural on model of linear buckling to find deformation on critical buckling load'. A red box labeled 'Share B2:B4' is also present. The Properties of Schematic B5: Setup panel on the right shows the following table:

	A	B
1	Property	Value
2	General	
3	Component ID	Setup 1
4	Directory Name	SYS-1
5	Notes	
6	Notes	
7	Used Licenses	
8	Last Update Used Licenses	
9	System Information	
10	Physics	Structural
11	Analysis	Linear Buckling
12	Solver	Mechanical APDL

At the bottom, the Progress panel shows the status of the analysis components:

	A	B	C
1	Status	Details	Progress

Buttons at the bottom right include 'Hide Progress' and 'Show 1 Messages'.

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

17- Boundary condition and Load

The image shows the ANSYS R15.0 interface for a Static Structural analysis. The main window displays a 3D model of a beam with a fixed support at one end and a force applied at the other. The beam is oriented along the X-axis, with a coordinate system (X, Y, Z) shown at the bottom right. The force is applied in the Z direction.

1- insert force and fix support same as previous section

2- Fix support

3- Load

Details of "Force"

Scope	
Scoping Method	Geometry Selection
Geometry	1 Face

Definition	
Type	Force
Define By	Components
Coordinate System	Global Coordinate System
☐ X Component	0. N (ramped)
☐ Y Component	0. N (ramped)
☒ Z Component	1030. N (ramped)
Suppressed	No

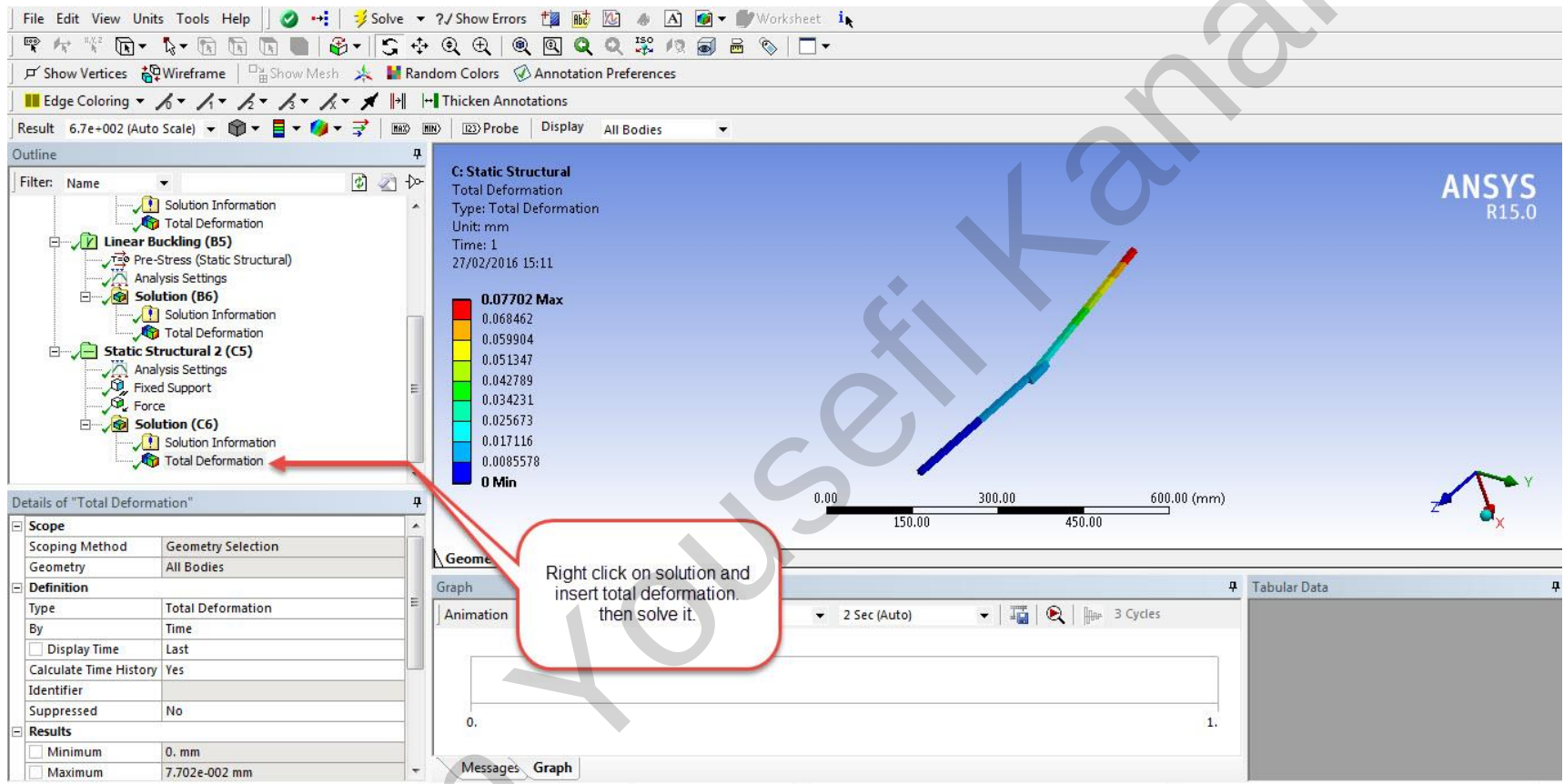
Graph

The graph shows the force components over time. The Z component (blue line) increases linearly from 0 to 1030 N over 1 second. The X and Y components remain at 0 N.

Tabular Data					
Steps	Time [s]	X [N]	Y [N]	Z [N]	
1	1	0.	0.	0.	
2	1	0.	0.	1030.	
*					

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