

ANSYS I-BEAM TUTORIAL IN WORKBENCH

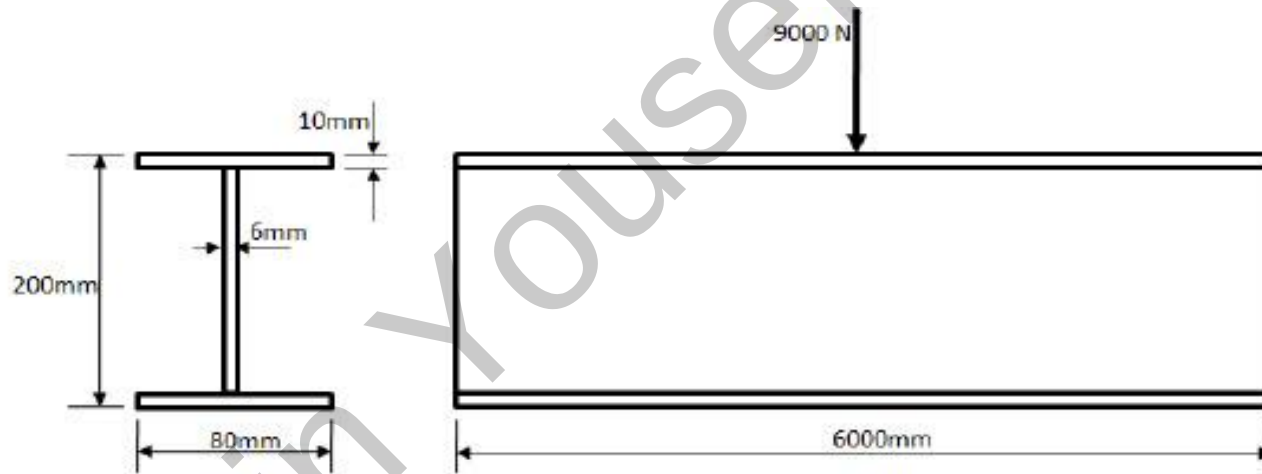
Armin Yousefi Kanani



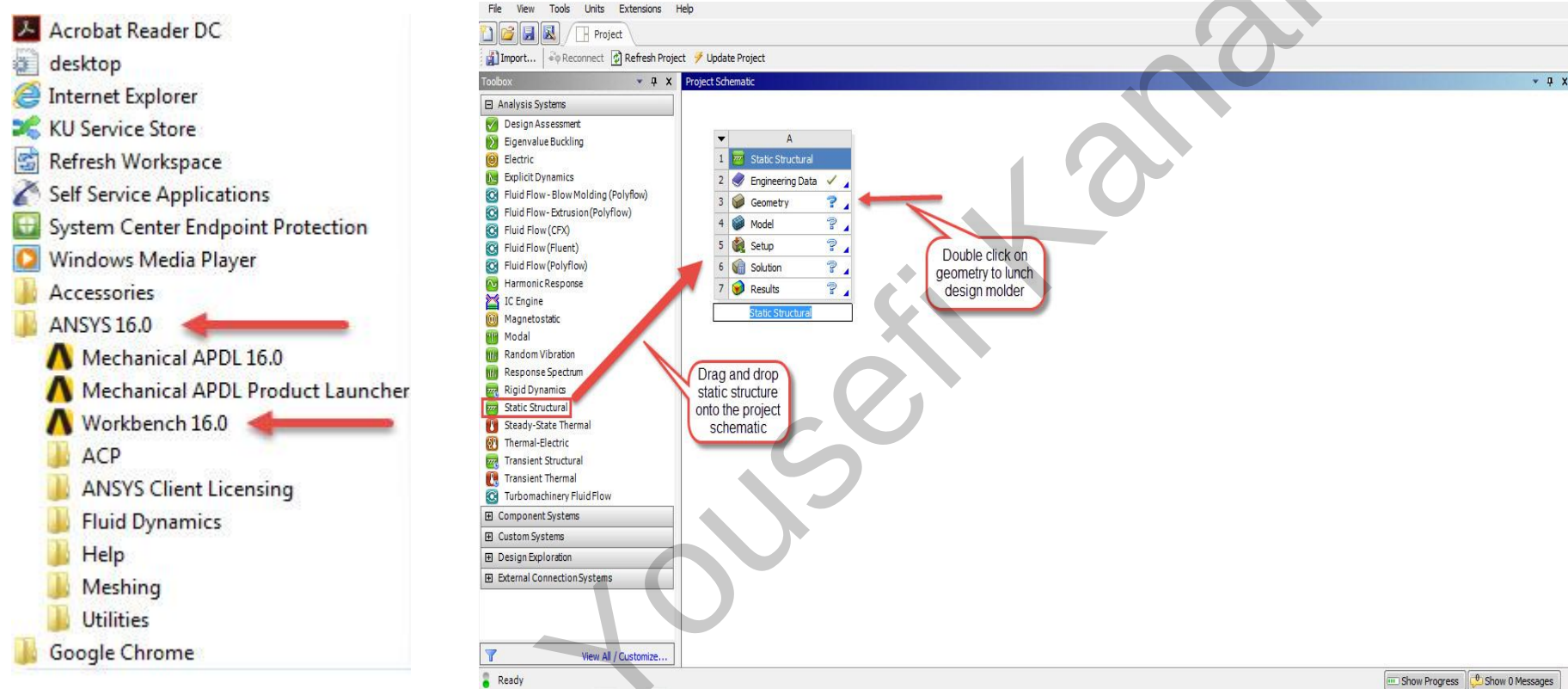
DESIGN MODELER

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 $= 250 \text{ N/mm}^2$

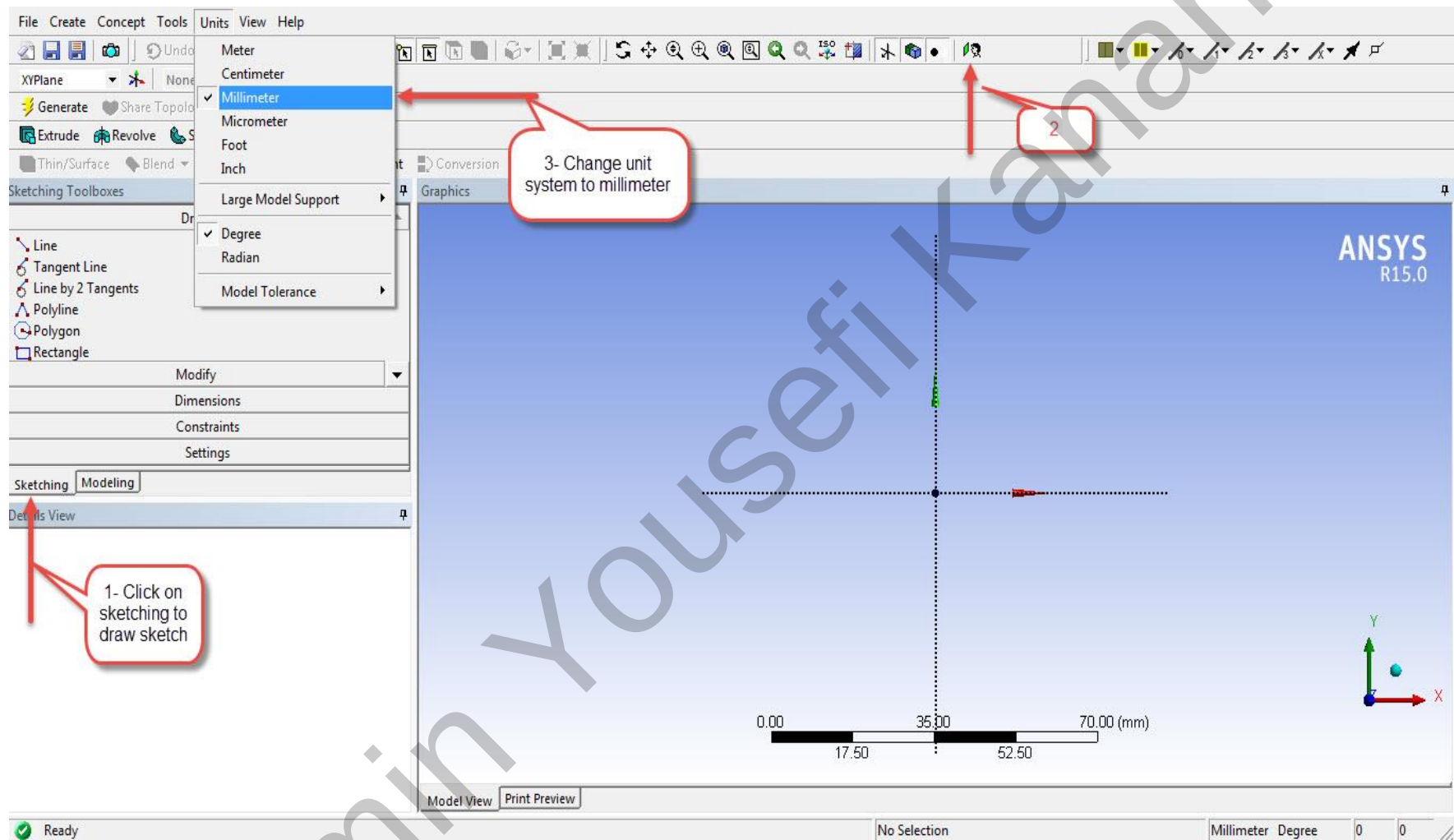


2- Open Workbench

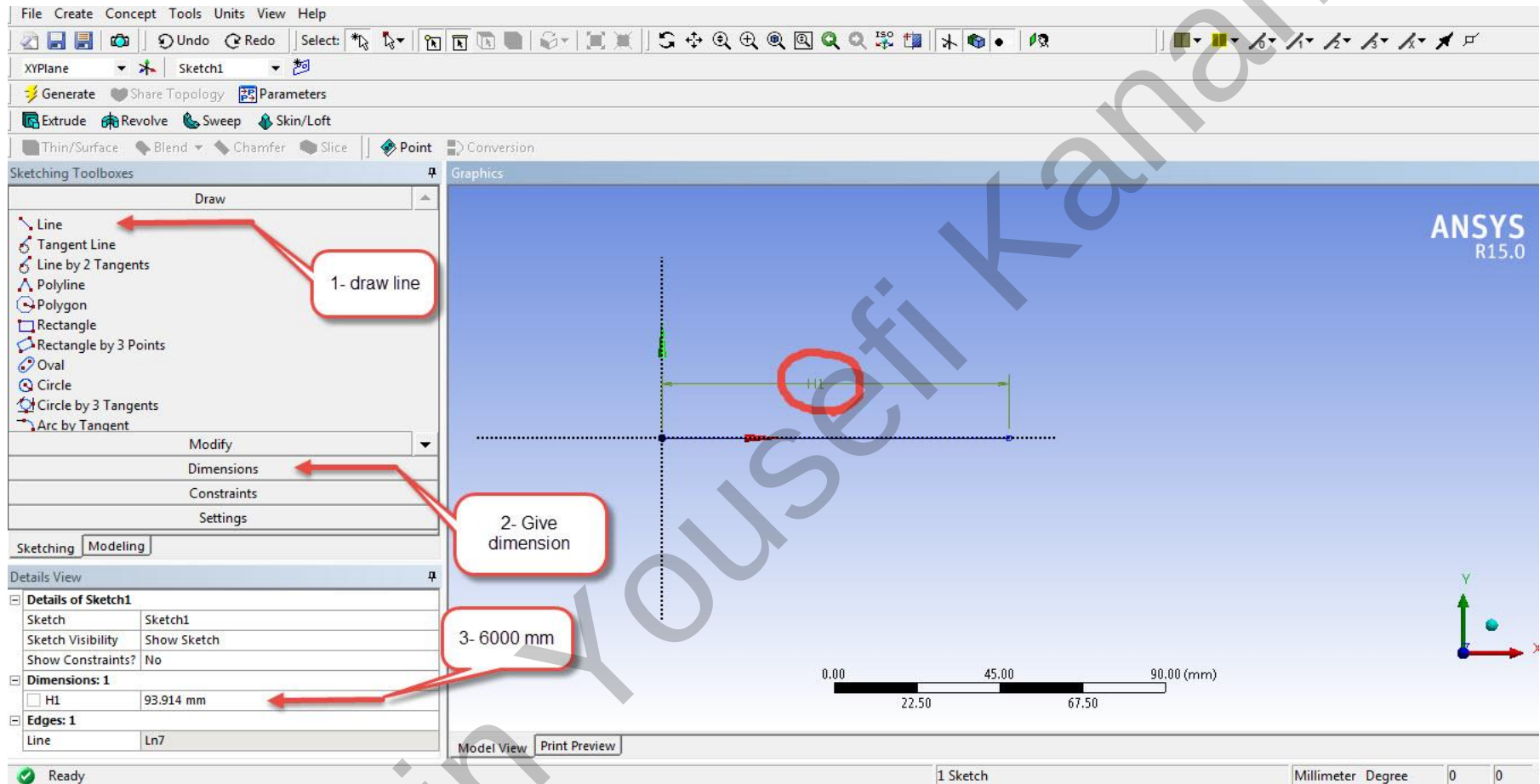


Open ANSYS workbench and drag and drop static structure and double click on geometry to lunch design molder

3- Design Modeler

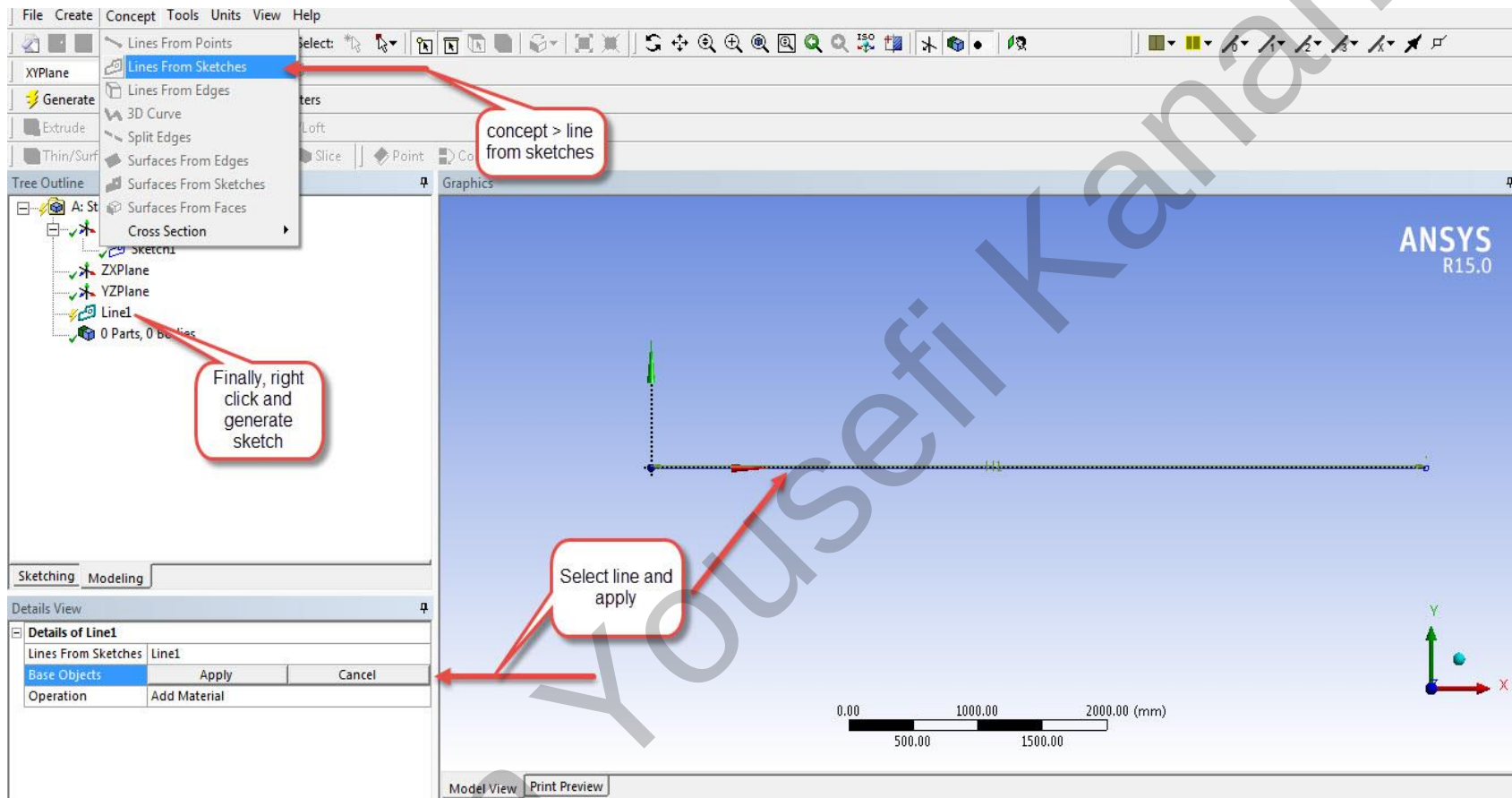


4- Making line in x-y plane



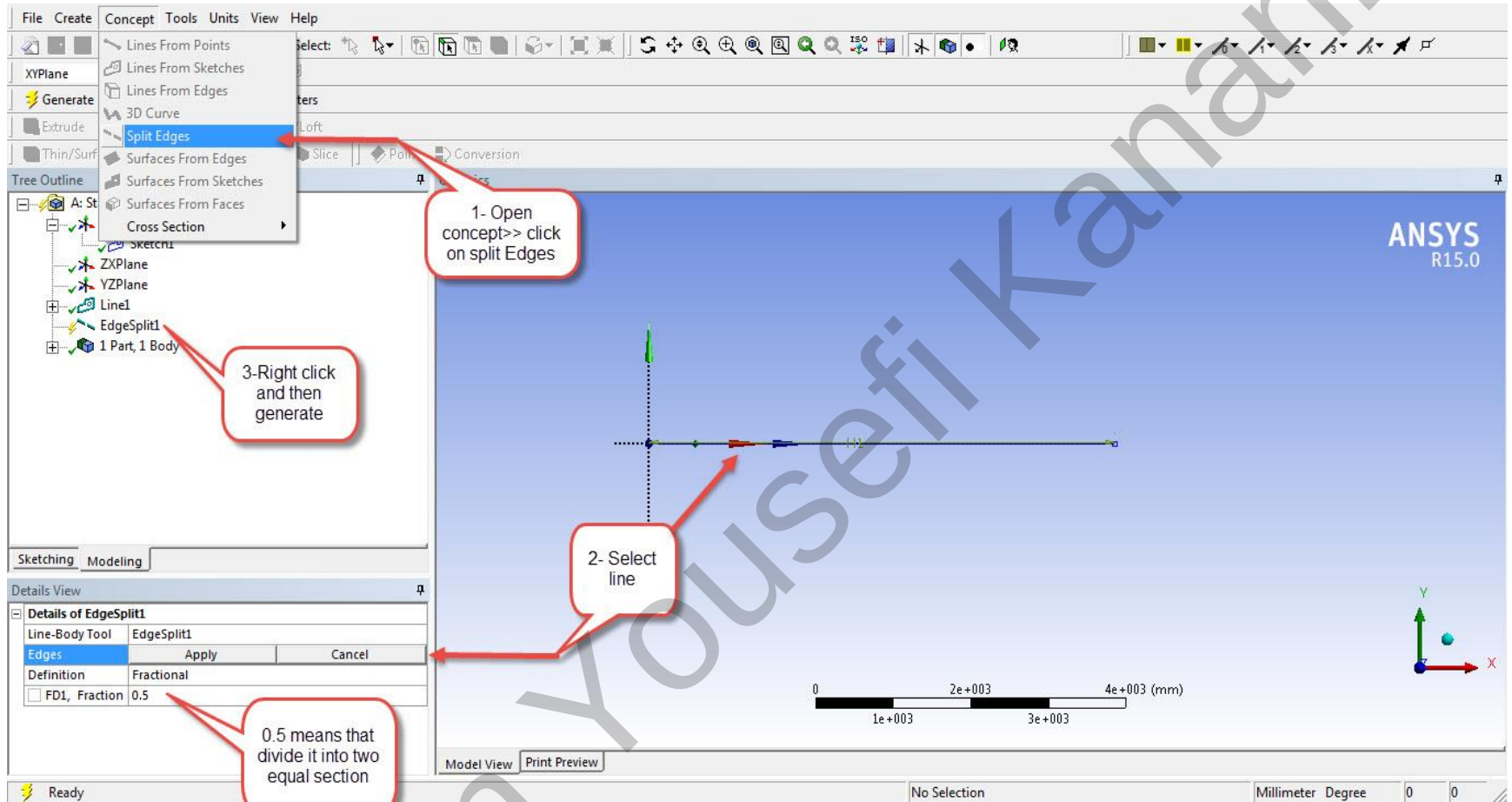
First draw line then change dimension to 6000 mm by horizontal dimension

5- Making Sketch from Line

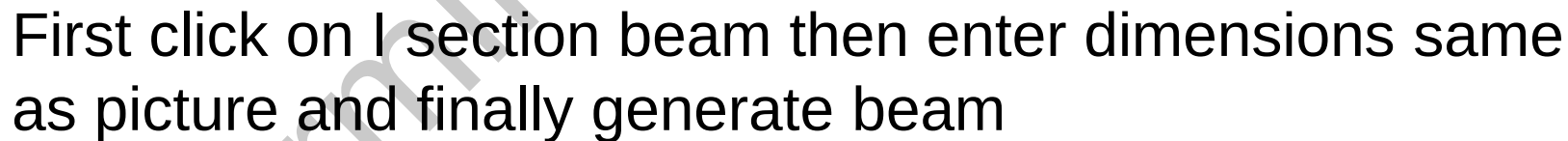


First click on concept and line from sketches to make sketch then select line and generate sketch.

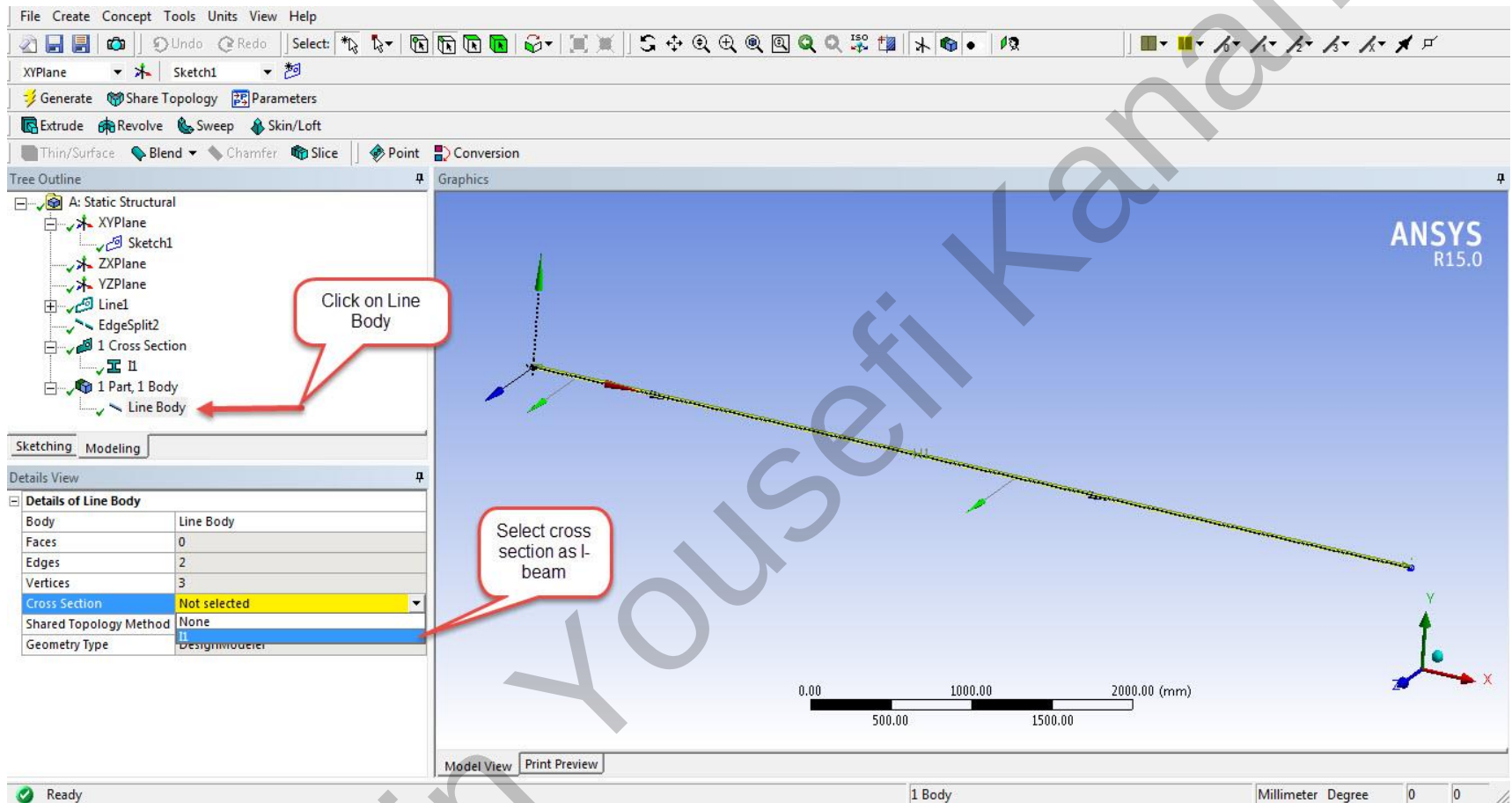
6- Split Line



Dividing line into two equal sections by using split edges command



8- Complete Geometry



Save and close design modeler after doing this step

9- Material

The screenshot shows the ANSYS Workbench interface. The 'Toolbox' on the left contains a list of analysis systems. The 'Project Schematic' in the center shows a hierarchy with 'Model' selected. A red arrow points to the 'Model' cell with a callout saying 'Double click'. The 'Properties of Schematic A4: Model' panel on the right displays the following information:

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

9-1 Selecting Material

Filter Engineering Data Engineering Data Sources

Toolbox

- Physical Properties
- Linear Elastic
 - Isotropic Elasticity
 - Orthotropic Elasticity
 - Anisotropic Elasticity
- Hyperelastic Experimental Data
- Hyperelastic
- Chaboche Test Data
- Plasticity
- Creep
- Life
- Strength
- Gasket
- Viscoelastic Test Data
 - Shear Data - Viscoelastic
 - Bulk Data - Viscoelastic
- Viscoelastic
- Shape Memory Alloy
- Damage
- Cohesive Zone
- Fracture Criteria

Outline of Schematic A2: Engineering Data

	A	B	D
1	Contents of Engineering Data		Description
2	Material		
3	Structural Steel		Fatigue Data at zero mean stress comes from 1998 ASME BPV Code, Section 8 Div 2, Table 5-110.1
*	Click here to add a new material		

Click on material

Properties of Outline Row 3: Structural Steel

	A	B	C
1	Property	Value	Unit
7	Derive from	Young's ...	
8	Young's Modulus	2E+11	Pa
9	Poisson's Ratio	0.3	
10	Bulk Modulus	1.6667E+11	Pa
11	Shear Modulus	7.6923E+10	Pa
12	Alternating Stress Mean Stress	Tabular	
16	Strain-Life Parameters		
17	Display Curve Type	Strain-Life	
18	Strength Coefficient	9.2E+08	Pa
19	Strength Exponent	-0.106	
20	Ductility Coefficient	0.213	
21	Ductility Exponent	-0.47	
22	Cyclic Strength Coefficient	1E+09	Pa
23	Cyclic Strain Hardening Exponent	0.2	
24	Tensile Yield Strength	2.5E+08	Pa

In this tutorial you do not need to change figure

Chart: No data

View All / Customize...

* After modifying material close **engineering data** or **click** on **project** to come back to project schematic

STATIC STRUCTURAL ANALYSIS

10- Open Model Section

The screenshot displays the ANSYS Workbench interface. The **Project Schematic** panel shows a sequence of steps: 1. Static Structural, 2. Engineering Data, 3. Geometry, 4. Model, 5. Setup, 6. Solution, and 7. Results. The 'Geometry' step is highlighted with a green checkmark, and a red callout indicates that the software will put a green tick in front of each step if it is finished correctly. The 'Model' component is also highlighted with a red callout, instructing the user to double-click on the model to open the mechanical section.

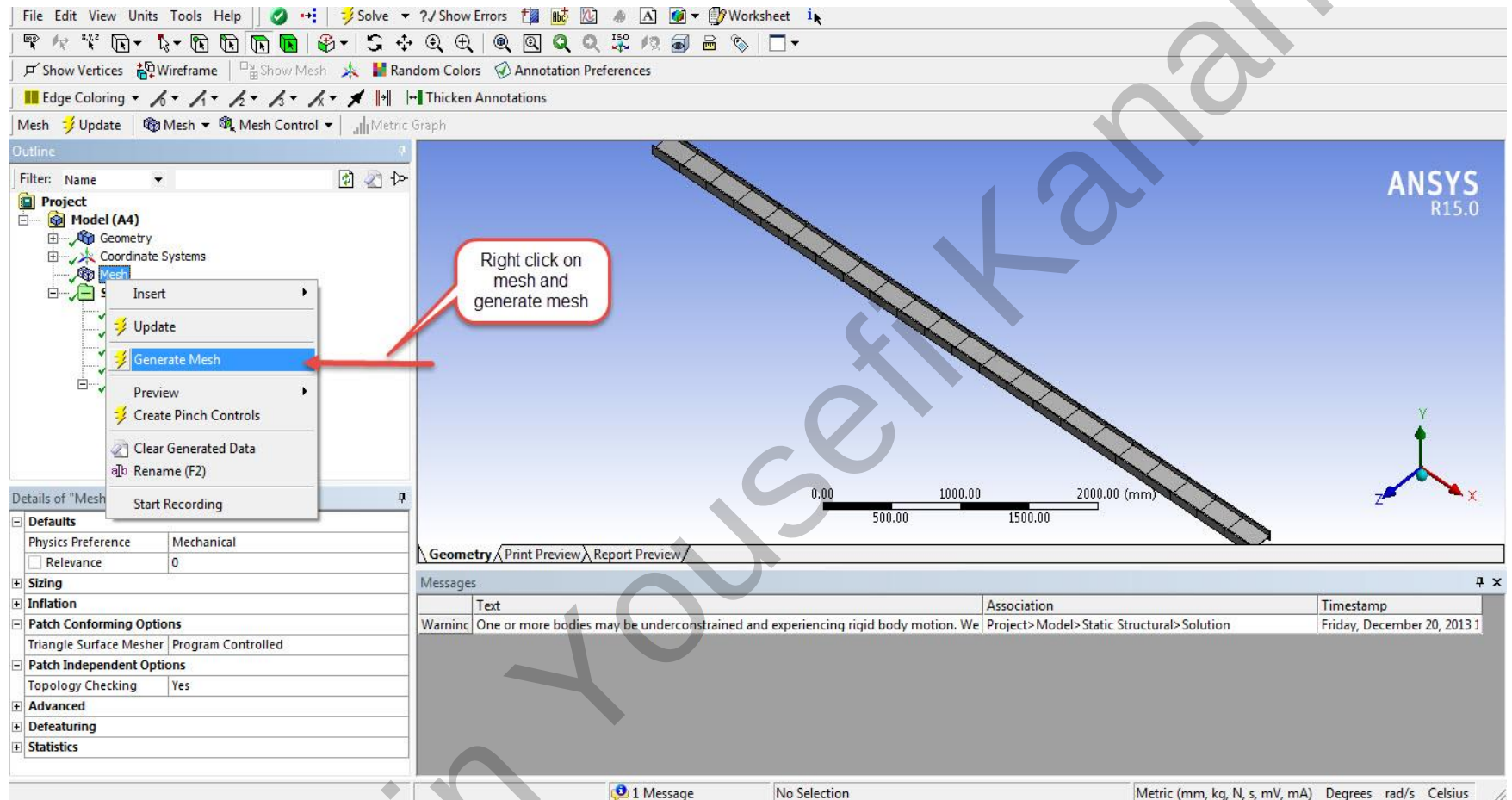
The **Messages** panel at the bottom shows a table with columns A, B, C, and D. The first row is labeled '1' and contains the following data:

	A	B	C	D
1	Type	Text	Association	Date/Time

The **Properties of Schematic A3: Geometry** panel on the right shows various settings. The 'Geometry File Name' is set to 'C:\Users\1\AppData\Local\Temp\WB_1-HP_1_9212_2\unsaved_project_files\dp0\SYS\DM\SYS.agdb'. The 'CAD Plug-In' is set to 'DesignModeler[4456]'. The 'Analysis Type' is set to '3D'. The 'Use Associativity' checkbox is checked. The 'Import Coordinate Systems' checkbox is unchecked. The 'Import Work Points' checkbox is unchecked. The 'Reader Mode Saves Updated File' checkbox is unchecked. The 'Import Using Instances' checkbox is checked. The 'Smart CAD Update' checkbox is unchecked. The 'Compare Parts On Update' is set to 'No'. The 'Enclosure and Symmetry Processing' checkbox is checked. The 'Decompose Disjoint Geometry' checkbox is checked.

At the bottom of the interface, there are buttons for 'Hide Progress' and 'Hide 0 Messages'.

11 - Meshing



You can change view from view >> Shaded Exterior

12- Boundary Condition

1- Click on displacement to apply boundary condition

2- select node filter

3- Select point A and B

Hold ctrl and select B

Zero movement in Z direction

Details of "Displacement"

Scope	
Scoping Method	Geometry Selection
Geometry	2 Vertices

Definition	
Type	Displacement
Define By	Components
Coordinate System	Global Coordinate System
X Component	Free
Y Component	Free
Z Component	0. mm (ramped)
Suppressed	No

Graph

Zero movement in Z direction

Tabular Data

Steps	Time [s]	Z [mm]
1	0.	0.
2	1.	0.

13- Applying Load

ANSYS R15.0

File Edit View Units Tools Help | Solve ? Show Errors | Worksheet

Show Vertices Wireframe Show Mesh Random Colors Annotation Preferences

Edge Coloring 1 2 3 Thicken Annotations

Environment Inertial Loads Supports Conditions

Outline

Filter: Name

Project

Model (A4)

Geometry

Coordinate Sys

Mesh

Static Struct

Analysis S

Displacement

Force

Solution

Solve

Pressure

Pipe Pressure

Hydrostatic Pressure

Force

Remote Force

Bearing Load

Bolt Pretension

Moment

Generalized Plane Strain

Line Pressure

Thermal Condition

Pipe Temperature

Joint Load

Fluid Solid Interface

Detonation Point

1-applying force

Force: 9000. N
Components: 0., 0., -9000. N

2- Select point in the middle of beam

Details of "Force"

Scope

Scoping Method Geometry Selection

Geometry 1 Vertex

Definition

Type Force

Define By Components

Coordinate System Global Coordinate System

X Component 0. N (ramped)

Y Component 0. N (ramped)

Z Component -9000. N (ramped)

Suppressed No

3- -9000 in z direction

Graph

Graph

Tabular Data

Steps	Time [s]	X [N]	Y [N]	Z [N]
1	1	0.	0.	0.
2	1	0.	0.	-9000.
*				

Messages Graph

No Messages No Selection

Metric (mm. ka. N. s. mV. mA) Degrees rad/s Celsius

14- Fixing Rotation in X Direction

ANSYS R15.0

Outline

- Project
 - Model (A4)
 - Geometry
 - Coordinate Systems
 - Mesh
 - Static Structural (A5)
 - Analysis Settings
 - Displacement
 - Force
 - Fixed Rotation
 - Solution (A6)
 - Solution Information

Details of "Fixed Rotation"

Scope	
Scoping Method	Geometry Selection
Geometry	2 Vertices

Definition	
Type	Fixed Rotation
Coordinate System	Global Coordinate System
☐ Rotation X	Fixed
☐ Rotation Y	Free
☒ Rotation Z	Free
Suppressed	No

Graph

1.

Messages **Graph**

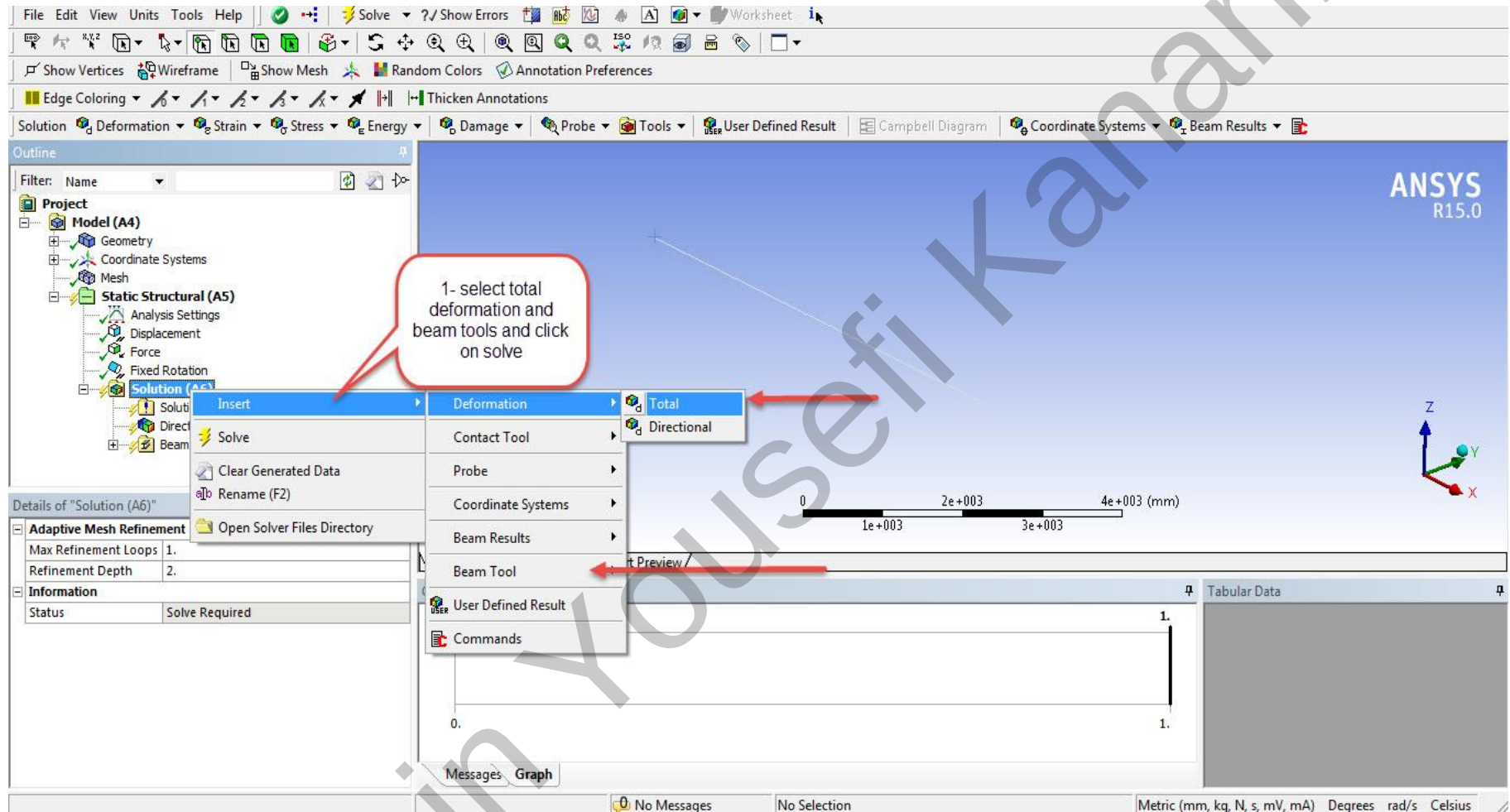
No Messages No Selection Metric (mm, kg, N, s, mV, mA) Degrees rad/s Celsius

1- Fixing rotation in x direction

2- select two point showed in beam

3- Rotation is zero in x direction

15-Solution



16-Results

