EASTERN MEDITERRANEAN UNIVERSITY
FACULTY OF ENGINEERING
DEPARTMENT OF MECHANICAL ENGINEERING

MENG 331
DYNAMICS OF MACHINERY
LABS REPORT MANUAL
(Fall 2021-2022)

Instructor: Assoc. Prof. Dr. Qasim Zeeshan
Teaching Assistant/ Lab Instructor: Mr. Tauqeer Nasir

EVALUATION	WEIGHT /100	LAB # 1	LAB # 2	LAB # 3	LAB # 4
Problem Definition					
Methodology					
Data Analysis					
Discussion and Conclusion					
TOTAL: /100					

STUDENT NO: ………………………….. NAME, SURNAME: ……………………………………………………………….

STUDENT NO: ………………………….. NAME, SURNAME: ……………………………………………………………….

STUDENT NO: ………………………….. NAME, SURNAME: ……………………………………………………………….

STUDENT NO: ………………………….. NAME, SURNAME: ……………………………………………………………….

STUDENT NO: ………………………….. NAME, SURNAME: ……………………………………………………………….

LAB PROJECT # 1: Modeling of a Discretized Lumped Parameter Spring Mass Damper System
LAB PROJECT # 2: Damped and Undamped Free vibration response
LAB PROJECT # 3: Balancing of Machines: Static and Dynamic Balancing
LAB PROJECT # 4: Mode shape analysis of cantilever beam with ANSYS

DUE DATE: 7 Jan 2022

Instructions:		**Student Outcomes**	
1. Answers are to be answered on the spaces provided under each section.		**1**	an ability to identify, formulate, and solve complex engineering problems by applying principles of engineering, science, and mathematics
2. Draw neat, labeled diagrams where necessary. Write relevant equations and write your Assumptions where necessary.		**2**	an ability to apply engineering design to produce solutions that meet specified needs with consideration of public health, safety, and welfare, as well as global, cultural, social, environmental, and economic factors
3. Be clear and specific and include units in answers. Give explanatory notes where necessary. Please carry out each step explicitly.		**3**	an ability to communicate effectively with a range of audiences
4. Please provide the references using APA Style of Referencing. References should be from the Textbook or from an authentic source.		**4**	an ability to recognize ethical and professional responsibilities in engineering situations and make informed judgments, which must consider the impact of engineering solutions in global, economic, environmental, and societal contexts
5. Please include the equations, diagrams, plots and figures in the report.		**5**	an ability to function effectively on a team whose members together provide leadership, create a collaborative and inclusive environment, establish goals, plan tasks, and meet objectives
6. All figures and Tables must be numbered and captioned.		**6**	an ability to develop and conduct appropriate experimentation, analyze and interpret data, and use engineering judgment to draw conclusions
7. Please also submit the pdf Soft copy of the report and the together with the **MATLAB codes, Simulink and Simscape Models, ANSYS Models, AVI File etc**.		**7**	an ability to acquire and apply new knowledge as needed, using appropriate learning strategies
8. The file/ folder uploaded should be named as **[MENG331-LABS-GROUP NO- STUDENT IDS]**			
9. Files with any other name or format will be disregarded.			
10. Late submissions will be penalized with reduction in total marks.			

LAB PROJECT # 1:
Modeling of a Discretized Lumped Parameter Spring Mass Damper System

Model the Single Degree of Freedom Discretized Lumped Parameter System as Second Order differential Equation in MATLAB, Simulink and Simscape.
You may use ODE45 variable step solver to solve the Second Order differential Equation.

1. **Choose any arbitrary values of Mass, Stiffness, and Damping, and plot the Amplitude, Velocity and Acceleration vs Time.**
2. **Analyze the effect of change of Mass by plotting the results.**
3. **Analyze the effect of change of Stiffness by plotting the results.**
4. **Analyze the effect of change of Damping by plotting the results.**
5. **Match the excitation force frequency with natural frequency and plot the resonance graphs.**

Mechanical Systems

Newton's laws of motion form the basis for analyzing mechanical systems. **Newton's second law**, Equation (1), states that the sum of the forces acting on a body equals the product of its mass and acceleration. **Newton's third law**, for our purposes, states that if two bodies are in contact, then they experience the same magnitude contact force, just acting in opposite directions.

$$\Sigma \mathbf{F} = m\mathbf{a} = m\frac{d^2\mathbf{x}}{dt^2} \tag{1}$$

When applying this equation, it is best to construct a **free-body diagram (FBD)** of the sysetm showing all of the applied forces.

Example: Mass-Spring-Damper System
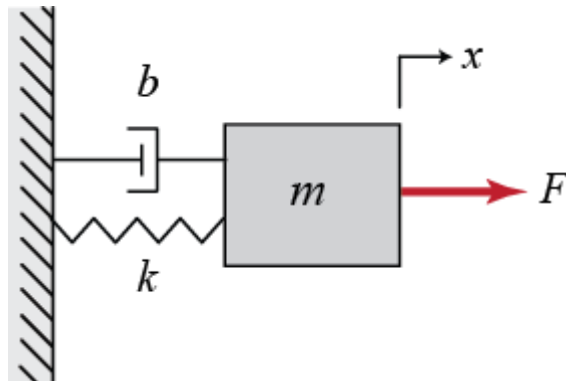
The free-body diagram for this system is shown below. The spring force is proportional to the displacement of the mass, x, and the viscous damping force is proportional to the velocity of the mass, $v = \dot{x}$. Both forces oppose the motion of the mass and are, therefore, shown in the negative x-direction. Note also that $x = 0$ corresponds to the position of the mass when the spring is unstretched.
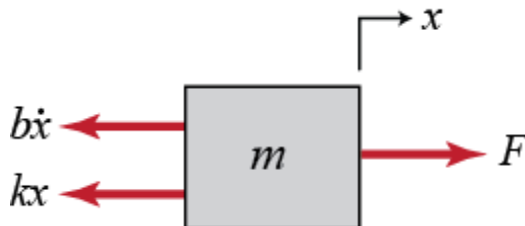
Now we proceed by summing the forces and applying Newton's second law, Equation (1), in each direction. In this case, there are no forces acting in the y-direction; however, in the x-direction we have:

$$\Sigma F_x = F(t) - b\dot{x} - kx = m\ddot{x} \tag{2}$$

This equation, known as the **governing equation**, completely characterizes the dynamic state of the system. Later, we will see how to use this to calculate the response of the system to any external input, $F(t)$, as well as to analyze system properties such as stability and performance.
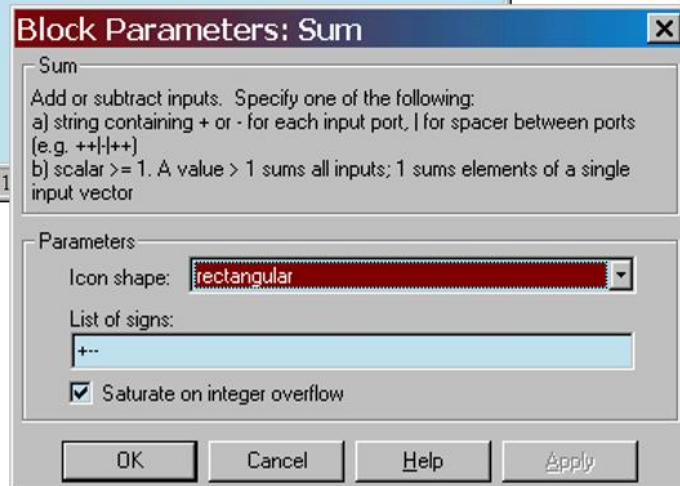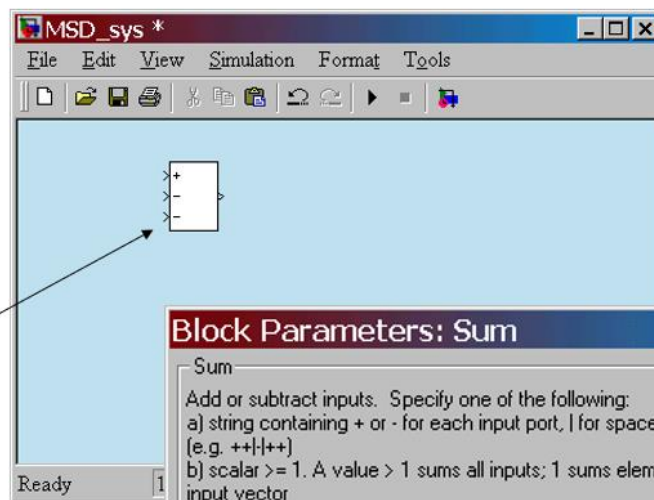
❑ First, solve for the term with highest-order derivative

$$m\ddot{x} = f(t) - c\dot{x} - kx$$

❑ Make the left-hand side of this equation the output of a summing block
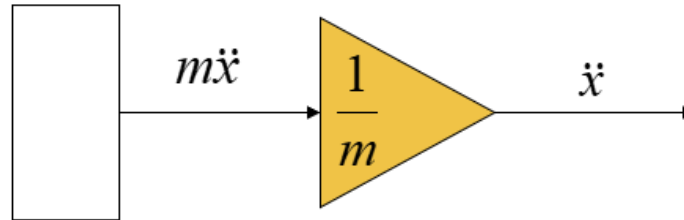
$m\ddot{x}$
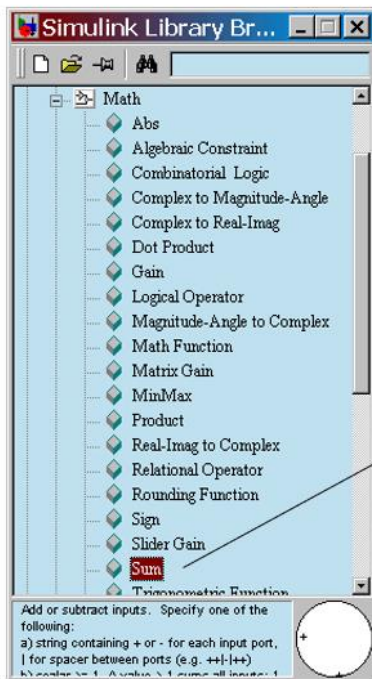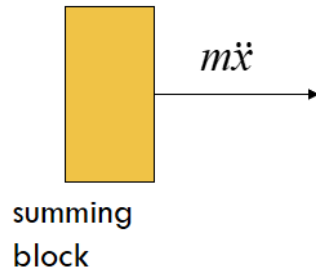
summing
block
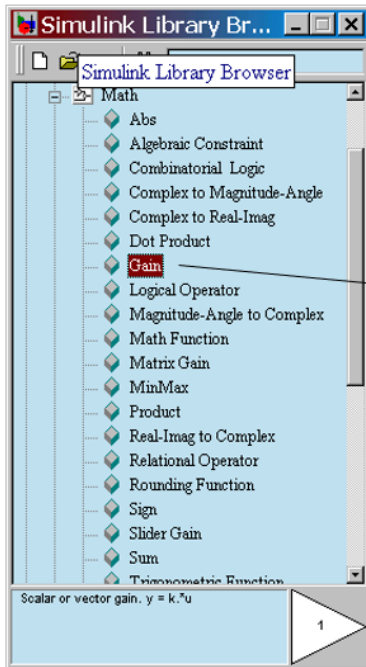
Drag a *Sum* block from the *Math* library

Double-click to change the block
parameters to *rectangular* and + - -
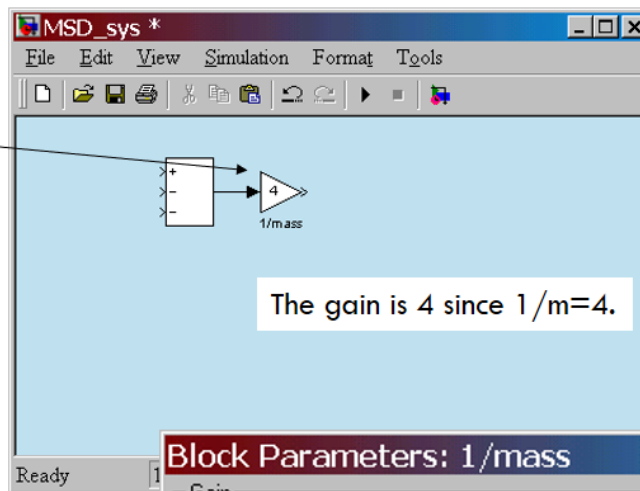
□ Add a gain (multiplier) block to eliminate the coefficient and produce the highest-derivative alone
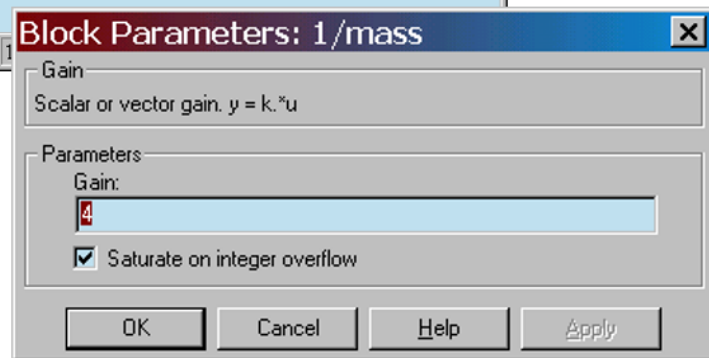
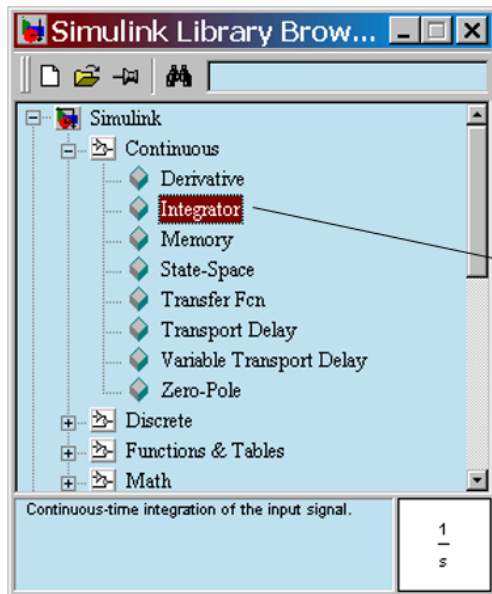$$m\ddot{x} \quad \boxed{\dfrac{1}{m}} \quad \ddot{x}$$

summing
block

Simulink Library Br... ▢ ☒

Simulink Library Browser

⊟ Math
  ◇ Abs
  ◇ Algebraic Constraint
  ◇ Combinatorial Logic
  ◇ Complex to Magnitude-Angle
  ◇ Complex to Real-Imag
  ◇ Dot Product
  ◇ Gain
  ◇ Logical Operator
  ◇ Magnitude-Angle to Complex
  ◇ Math Function
  ◇ Matrix Gain
  ◇ MinMax
  ◇ Product
  ◇ Real-Imag to Complex
  ◇ Relational Operator
  ◇ Rounding Function
  ◇ Sign
  ◇ Slider Gain
  ◇ Sum
  ◇ Trigonometric Function

Scalar or vector gain. y = k.*u

Drag a *Gain* block from the *Math* library

MSD_sys *

File Edit View Simulation Format Tools

1/mass

The gain is 4 since 1/m=4.

Ready

Double-click to change the block parameters.
Add a title.

Block Parameters: 1/mass ☒

Gain
Scalar or vector gain. y = k.*u

Parameters
Gain:
4
☑ Saturate on integer overflow

OK   Cancel   Help   Apply

□ Add integrators to obtain the desired output variable

summing
block
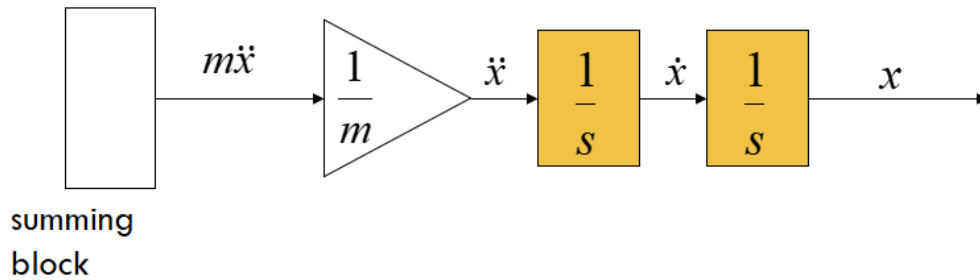
Drag *Integrator* blocks from the
Continuous library

ICs on the integrators are zero.
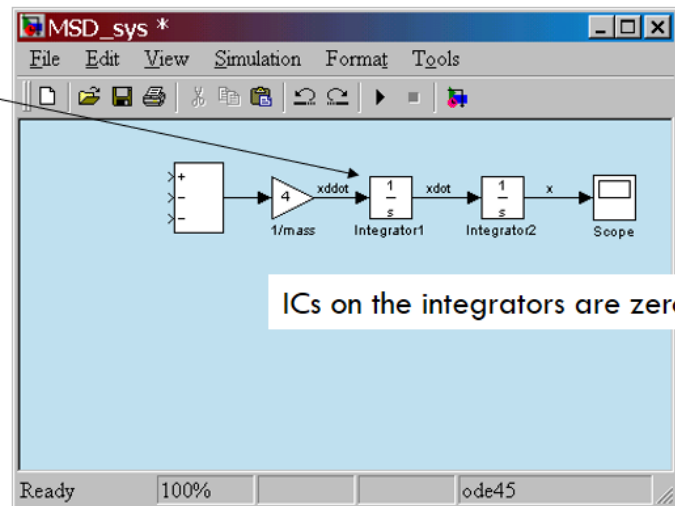
Add a scope from the *Sinks* library.
Connect output ports to input ports.
Label the signals by double-clicking on the leader line.

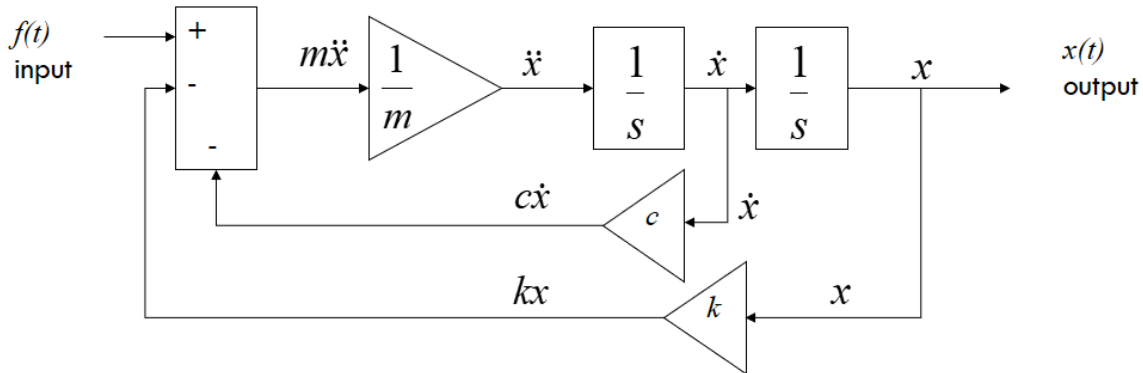- Connect to the integrated signals with gain blocks to create the terms on the right-hand side of the EOM

$m\ddot{x}$ → $\dfrac{1}{m}$ → $\ddot{x}$ → $\dfrac{1}{s}$ → $\dot{x}$ → $\dfrac{1}{s}$ → x
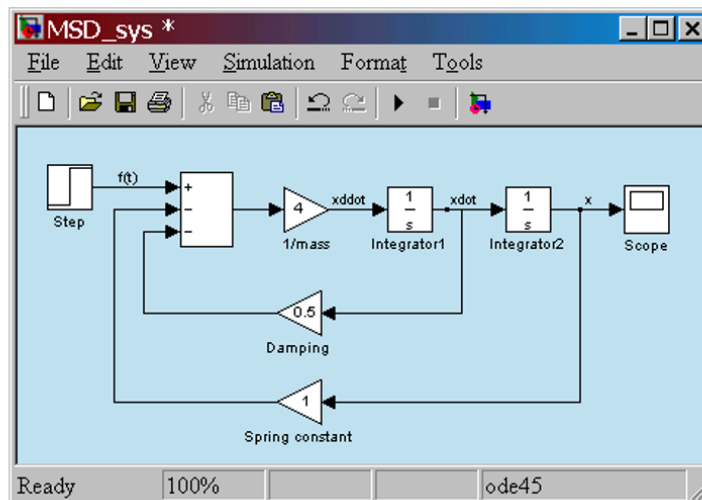
summing block

$c\dot{x}$ ← c

kx ← k
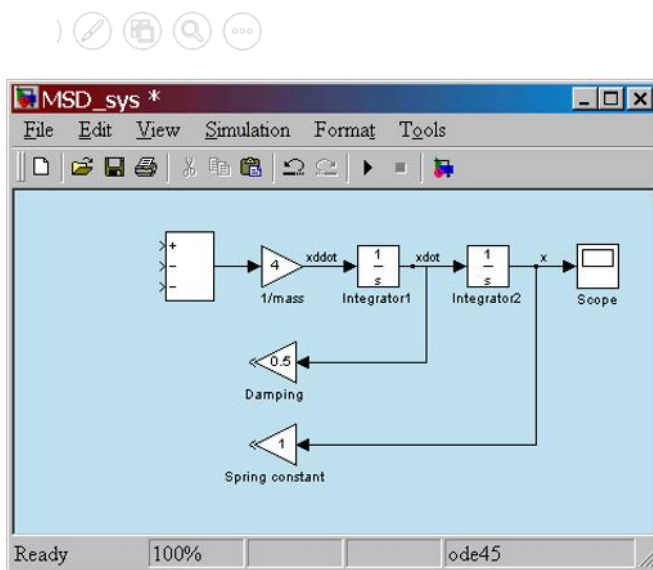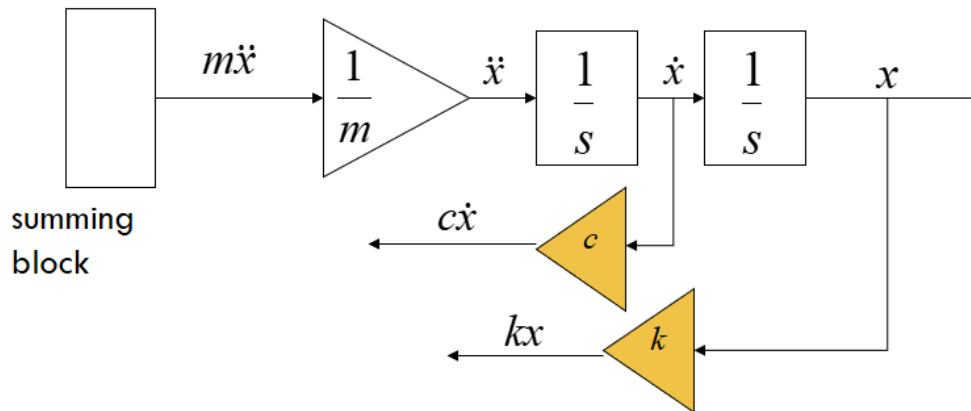
Drag new *Gain* blocks from the *Math* library

To flip the gain block, select it and choose *Flip Block* in the *Format* pull-down menu.

- Double-click on gain blocks to set parameters
- Connect from the gain block input backwards up to the branch point.
- Re-title the gain blocks.

c=0.5

k=1.0
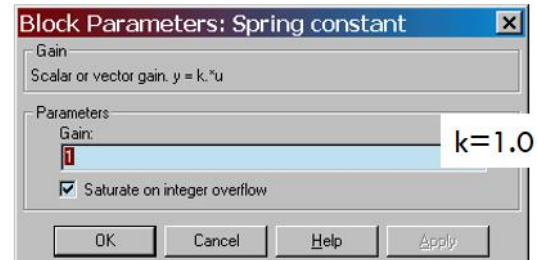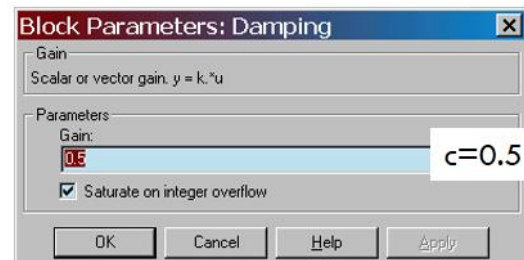
□ Bring all the signals and inputs to the summing block.

□ Check signs on the summer.

$f(t)$ input

$m\ddot{x}$ $\dfrac{1}{m}$ $\ddot{x}$ $\dfrac{1}{s}$ $\dot{x}$ $\dfrac{1}{s}$ x

$x(t)$ output

$c\dot{x}$ c $\dot{x}$

kx k x

Double-click on *Step* block to set parameters. For a step input of magnitude 3, set *Final value* to 3
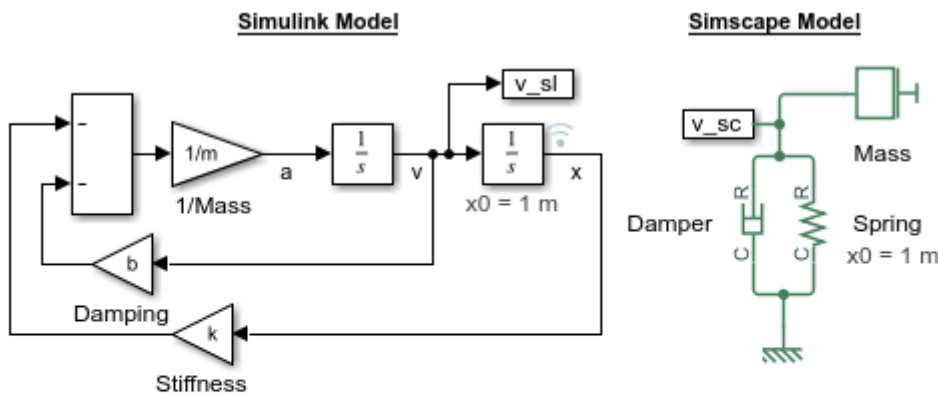
Mass-Spring-Damper in Simulink and Simscape

This example shows two models of a mass-spring-damper, one using Simulink® input/output blocks and one using Simscape™ physical networks.

The Simulink model uses signal connections, which define how data flows from one block to another. The Simscape model uses physical connections, which permit a bidirectional flow of energy between components. Physical connections make it possible to add further stages to the mass-spring-damper simply by using copy and paste. Input/output connections require rederiving and reimplementing the equations.
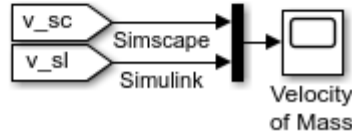
The initial deflection for the spring is 1 meter. This is shown in the block annotations for the Spring and one of the Integrator blocks.
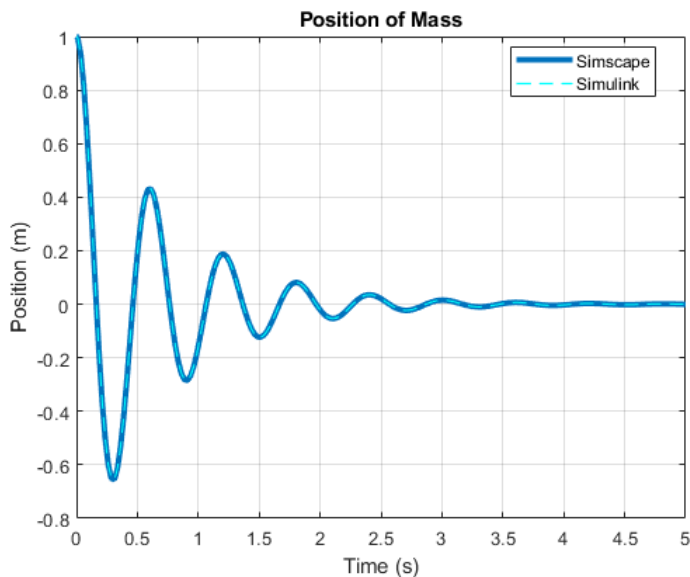
Model

Simulation Results from Simscape Logging


```
ssc_mass_spring_damper_sl
```

LAB PROJECT # 2: Damped and Undamped Free vibration response

APPARATUS: TM16 Universal Vibration Apparatus

Picture shows the Frame and Cupboard (TM16a) and the forced vibration experiment

The TM16 series is a range of products that teach different aspects of vibrations and oscillations in mechanical systems. These include pendulums, mass-springs systems and shafts and beams. The TM16 series is a modular system, based around a Frame and Cupboard (TM16a).

Objective

To determine the fundamental concepts of free vibration:

- Damped
- Undamped

Apparatus

TM16 Range Universal Vibration Apparatus

Figure 1: TM16 Universal Vibration Apparatus

Introduction

A vibration is the movement of a physical quantity in relation to a reference location in a cyclically increasing and decreasing manner as a function of time. During vibrations, energy is dissipated and so a steady amplitude cannot be maintained without continuous replacement. Viscous damping in which force is proportional to velocity affords the simplest mathematical treatment. A convenient means of measuring the amount of damping present is to measure the rate of decay of oscillation. This is expressed by the term 'logarithmic decrement', which is defined as the natural logarithm of the ratio of successive amplitudes on the same side of the mean position.

There are two types of vibration: (1) free vibration and (2) forced vibration. Free Vibration: A system is left to vibrate on its own after an initial disturbance and no external force acts on the system. Forced Vibration: A system that is subjected to a repeating external force. E.g. oscillation arises from motor.

Mechanical vibration classifications: (1) Damped Vibration: When any energy is lost or dissipated in friction or other resistance during oscillations, and (2) Undamped Vibration: When no energy is lost or dissipated in friction or other resistance during oscillations, finally (3) Resonance: It occurs when the frequency of the external force coincides with one of the natural frequencies of the system.

Parts needed form the equipment

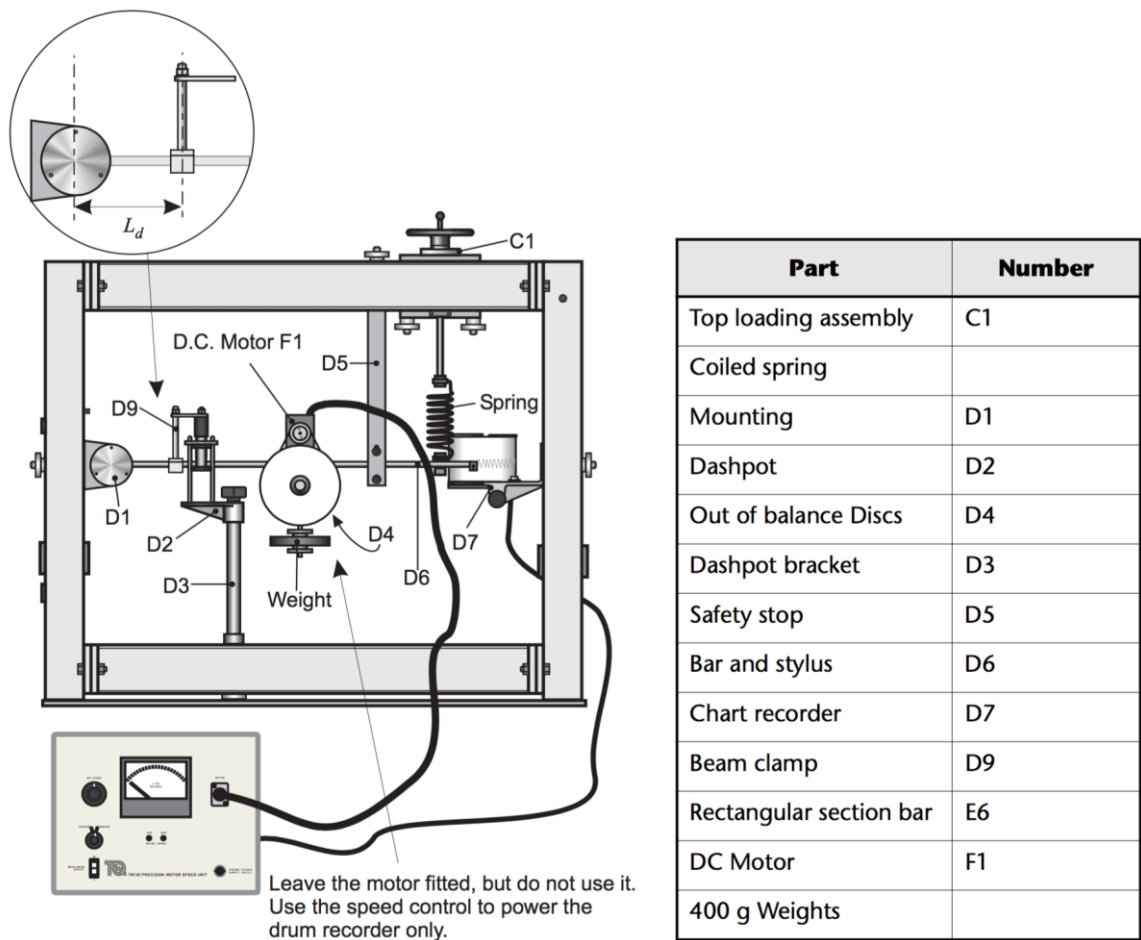| Part	Number
Top loading assembly	C1
Coiled spring	
Mounting	D1
Dashpot	D2
Out of balance Discs	D4
Dashpot bracket	D3
Safety stop	D5
Bar and stylus	D6
Chart recorder	D7
Beam clamp	D9
Rectangular section bar	E6
DC Motor	F1
400 g Weights	

Figure 2: Setup of the experiment

- The exciter motor (F1) is not going to be used (but leave it fitted), free vibrations will be used.
- To drive the chart recorder (D7) you need the Speed Control unit.
- The system is set vibrating freely by pulling down on the free end of the beam a short distance (15 – 25 mm) and releasing.
- Use the chart recorder to obtain a trace of just three successive amplitudes on the same side of the mean position
- **FOR DAMPED:** Vary the damping by moving the dashpot (D2) and its clamps along the beam, and also by relative rotation of the two orifice plates in the dashpot to increase or decrease the effective area of the piston
- **FOR UNDAMPED:** Disengaged the damper and repeat above steps.

Procedure

- Connect the lead from the motor of recorder unit D7 to the auxiliary supply socket on the Speed Control unit.
- Switch on the speed control unit.
- Set the dashpot at distance L_d (the distance from the trunnion mounting to the center of the beam clamp D9)
- Bring the recording pen into contact with the paper to produce a trace of the decaying amplitude of vibration and thus produce a trace of the decaying applied amplitude on the chart recorder paper.
- Pull the beam down a short distance, under the point of attachment of the spring, and release.
- Start observing the system, its noise and from the chart recorder.
- Gradually start increasing the speed from the motor speed control unit and observe.

QUESTIONS:
Briefly explain/ discuss the following:

1. **What type of damping are present in the system?**

2. **Why is damping considered only in the neighborhood of resonance in most cases?**

3. **How do you find the response of a viscously damped system under rotating unbalance?**

4. **The equation of motion of a machine (rotating at frequency ω) of mass M, with an unbalanced mass m, at radius e, is given by**

5. **What happens to the response of an undamped system at resonance?**

6. **Is it possible to completely avoid resonance of a damped during start up and stopping of a rotating machinery**

7. **Is it possible to reduce the amplitude of resonance during start up and stopping of a rotating machinery**

8. **The damping force depends on the frequency of the applied force in the case of a. viscous-damping b. Coulomb damping c. hysteresis damping? And Why, explain**

9. **Is it possible to find the approximate value of the amplitude of a damped forced vibration without considering damping at all? If so, under what circumstances?**

10. **Is dry friction effective in limiting the reasonant amplitude?**

LAB PROJECT # 3: Balancing of Machines: Static and Dynamic Balancing

APPARATUS: TM102 Static and Dynamic Balancing Apparatus
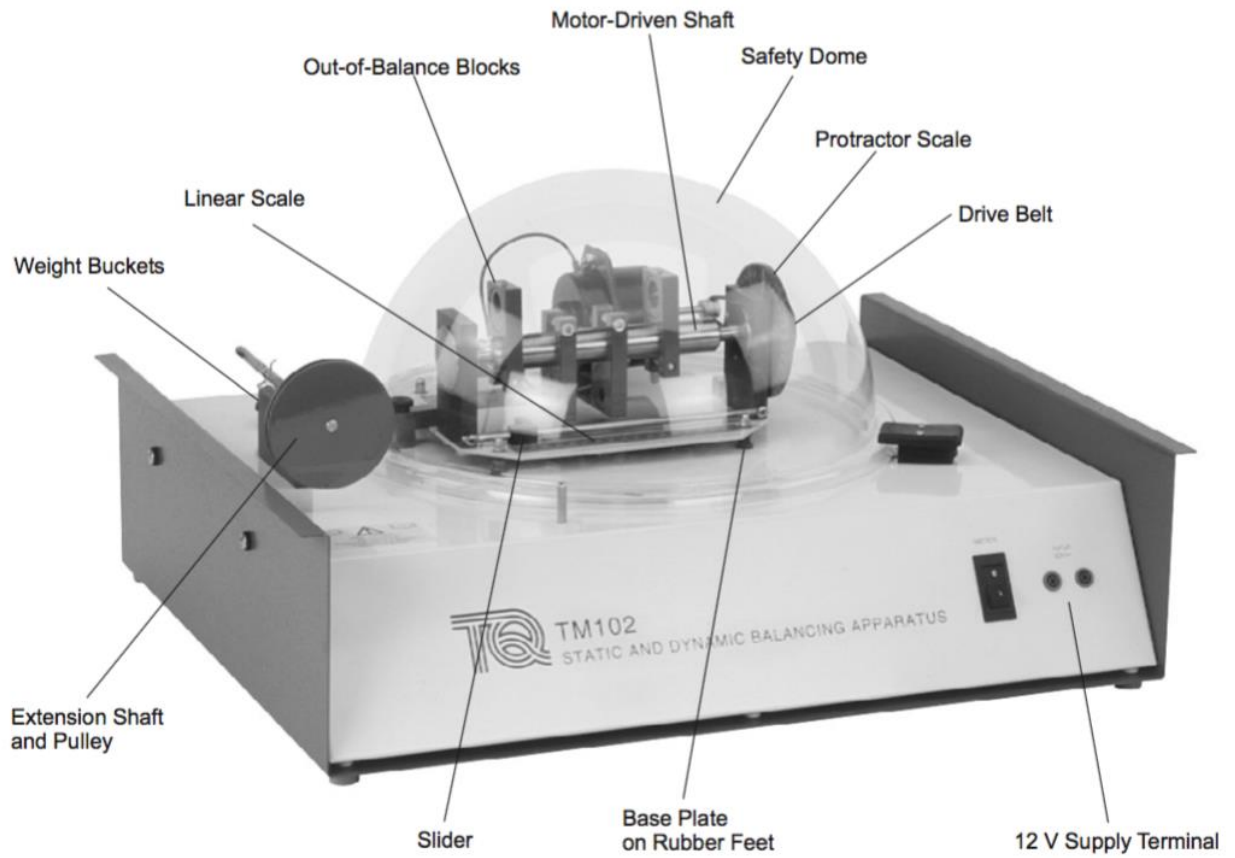
INTRODUCTION:

A shaft with masses mounted on it can be both statically and dynamically balanced. If it is statically balanced, it can stay in any angular position without rotating. If it is dynamically balanced, it can be rotated at any speed without vibration. It will be shown that if a shaft is dynamically balanced, it is automatically in static balance, but the reverse is not necessary true.

- ## Static Balancing

Figure above shows a simple situation where two masses mounted on a shaft. If the shaft is to be statically balanced, the moment due to weight of mass 1, tending to rotate the shaft clockwise, must equal that of mass 2, trying to turn the shaft in the opposite direction.

Therefore, the static balance is:

$W_1 r_r = W_2 r_2$ $\hspace{8cm}$ 1

Similarly, the same principle holds for more than two masses, as shown in Figure 2.
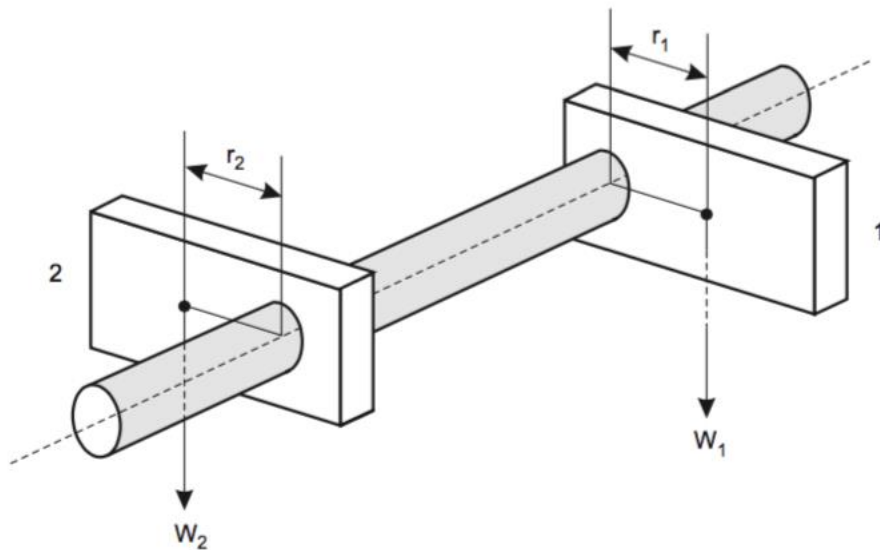

Figure 3: Static balancing of a simple two-masses system

And for the static balancing:

$$W_1 r_1 \cos \alpha_1 = W_2 r_2 \cos \alpha_2 + W_3 r_3 \cos \alpha_3$$

$$\underbrace{\hspace{2.5cm}}_{\text{anti-clockwise}} \quad \underbrace{\hspace{4cm}}_{\text{clockwise}}$$

$\hspace{14cm}$ 2

In general, the values of W, r and α have to be chosen such that the shaft is in balance. Nevertheless, for experiments using TM102 apparatus, the product Wr can be measured directly for each mass, and only the angular positions have to be determined for static balance. If the angular positions of two masses are fixed, then the position of the third mass can be found, either by trigonometry or drawing.
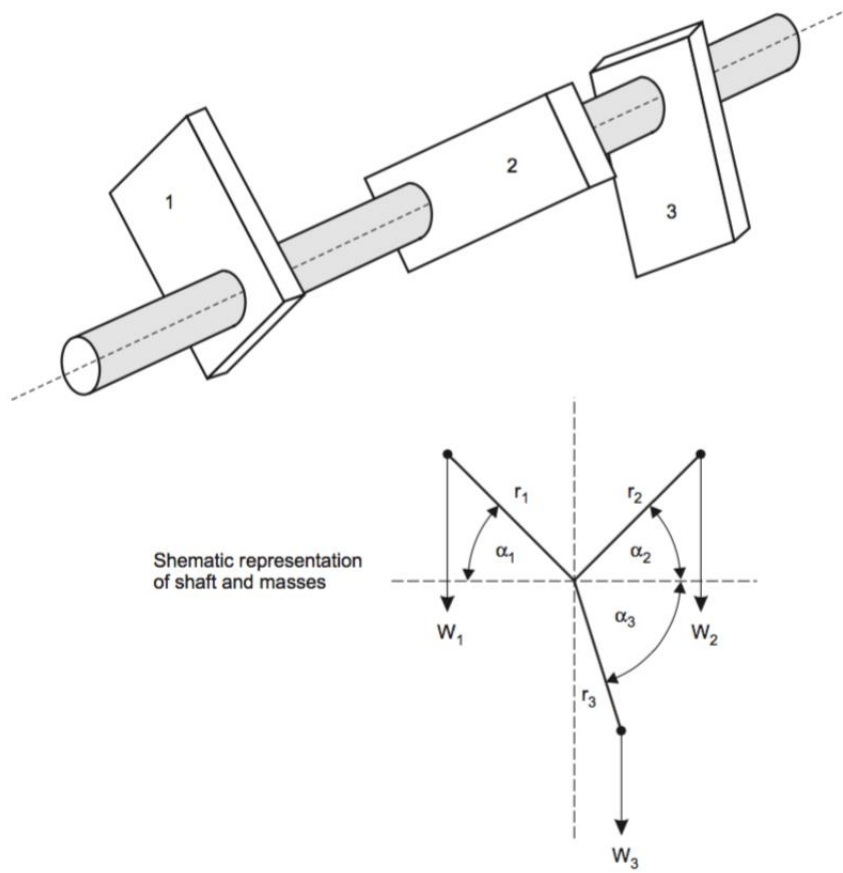
Shematic representation
of shaft and masses

Figure 4: Static balancing of three-masses system

• **Dynamic Balancing**

As the shaft rotate the masses are going to be subjected to centrifugal forces. If the shaft is not to vibrate as it rotates, then **two conditions** must be satisfied:

1. There must be no out-of-balance centrifugal force trying to deflect the shaft
2. There must be no out-of-balance moment or couple trying to twist the shaft

The shaft will not be dynamically balanced until these conditions are satisfied.
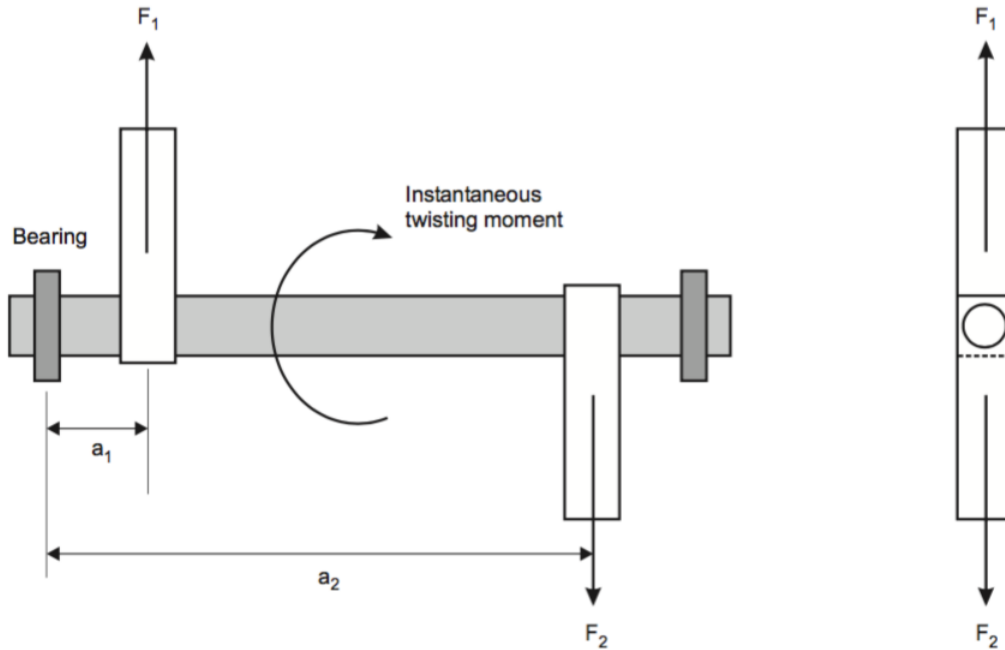
Figure 5:

• Applying condition 1 to the shaft in figure 3 gives:

$$F_1 = F_2 \qquad\qquad 3$$

The centrifugal force is:

$$mr\omega^2 \text{ or } \frac{W}{g}r\omega^2$$

Therefore;

$$\frac{W_1}{g}r_2\omega^2 = \frac{W_2}{g}r_2\omega^2$$

$$4$$

The angular speed of the rotation is the same for each mass, so that for the dynamic balance :

$$W_1r_1 = W_2r_2 \qquad\qquad 5$$

Note: If a shaft is dynamically balanced, it will also be statically balanced.

- The second condition will be satisfied by taking moments about some desired datum, like one of the bearings. Hence:

  $$a_1 F_1 = a_2 F_2 \qquad\qquad 6$$

Remember, $F_1 = F_2$, so as $a_1 = a_2$. Thus, in this case, dynamic balance can only be achieved if the two masses are mounted at the same point along the shaft.

Note: If a shaft is statically balanced, it does not mean that it is dynamically balanced.

- **Dynamic Balancing of three Masses**

It has been shown that dynamic balance can only be achieved for a two-mass system if the masses are mounted at the same point along the shaft.

It can also be shown that there are special conditions which must be satisfied for the dynamic balancing of three masses.
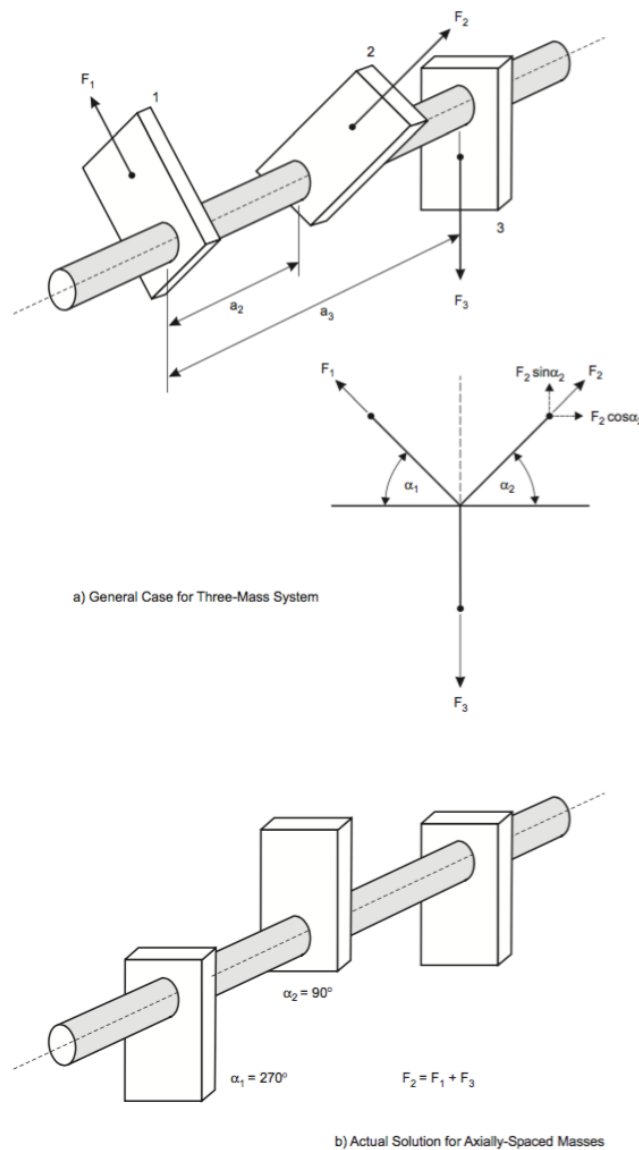

Figure 6: Dynamic balance for three mass system

From figure 4(a), mass 3 is positioned vertically for convenience. From condition 2, dynamic balance can be expressed mathematically by equating moments for centrifugal forces in both vertical and horizontal planes. In order to simplify the equations, it is convenient to take moments about mass 1, so that moments due to forces on this mass are eliminated.

$$\text{Horizontal} : a_2 F_2 \cos\alpha_2 = 0 \qquad\qquad 7$$

$$\text{Vertical} : a_2 F_2 \sin\alpha_2 = a_3 F_3 \qquad\qquad 8$$

Conditions that can satisfy equation 7 are either $a_2 = 0$, or $\alpha_2 = 90$ or 270 (that is $\cos\alpha_2 = 0$). The following conditions are obtained after substituting these in to equation 8:

a) $\alpha_2 = 0$

   a_3 must be zero for this condition. Thus, for any arbitrary values of α_2 and α_3, all three masses must be located qt the same point along the shaft.

b) $\alpha_2 = 90$ or 270

For these conditions, it is necessary to write down further equations to obtain solutions.

Applying condition 1 for dynamic balance:

$$\text{Horizontal} : F_1 \cos\alpha_1 = F_2 \cos\alpha_2 \qquad\qquad 9$$

$$\text{Vertical} : F_3 = F_1 \sin\alpha_1 + F_2 \sin\alpha_2 \qquad\qquad 10$$

If $\alpha_2 = 90$, equation 9 gives $\alpha_1 = 90$ or 270. Let's assume $\alpha_1 = 90$, then equation 10 reduces to:

$$F_3 = F_1 + F_2$$

Similarly, equation 8reduces to:

$$a_2 F_2 = a_3 F_3 \qquad\qquad 11$$

Combine above equations to solve for F_1.

Therefore, If the masses are distributed along the shaft, the following conditions must b satisfied for dynamic balancing:

1. Central mass at 180 degree to other two masses
2. Masses chosen such that:
   $$F_2 = F_1 + F_3$$
3. Masses distributed along the shaft such that:
   $$a_2 F_2 = a_3 F_3$$

• Dynamic Balancing of More Than Three Masses

If there are more than three masses, there are no special restrictions which apply to the angular and shaft-wise distributions of the masses, and the general conditions for dynamic balance have to be applied to solutions.

The angular positions of the masses can be determined by applying conditions for either static balance or for dynamic balance. The distribution of the masses along the shaft is then found by applying condition 1 for dynamic balance. This can be done by calculation.
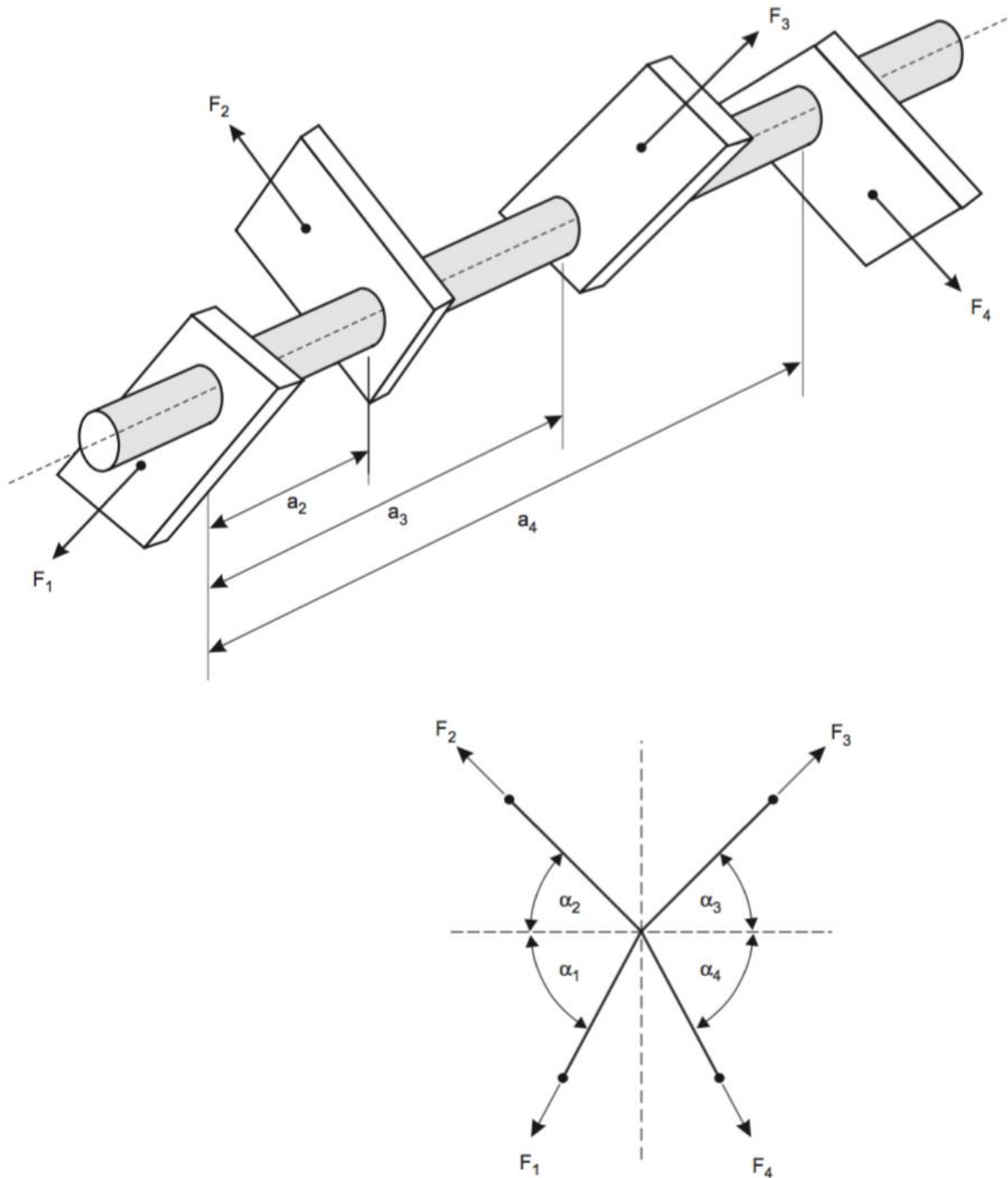
Figure 7: Nomenclature for Arbitrary Four-Mass System

The rotating moments are resolved into components tending to twist the shaft in the horizontal planes. The net moment on each plane must be zero if the shaft is to be dynamically balanced. As shown from figure 5, the appropriate equations for a four-mass system are:

Horizontal moments about mass 1

$$-a_2F_2 \cos\alpha_2 + a_3F_3 \cos\alpha_3 + a_4F_4 \cos\alpha_4 = 0$$

12

Vertical moments about mass 1

$$a_2F_2 \sin\alpha_2 + a_3F_3 \sin\alpha_3 - a_4F_4 \sin\alpha_4 = 0$$

13

PROCEDURE:

- **Demonstration of Static Balance by Dynamic Imbalance**
  - Remove the Perspex dome and the shaft drive belt.
  - Remove the disc from the four rectangle blocks using the small hexagon key. *Always take the blocks off the shaft before removing or replacing the discs.*
  - Set up two of the blocks, as shown in figure 6, with the relative angular displacement of 180 degrees and shaft-wise displacement of 120mm. use the slider to set and read off the positions of the blocks of the blocks.
  - Observe that the shaft will remain in any angular position and is therefore statically balanced.
  - Connect the apparatus to a 12 VDC supply. Make sure that the slider adjacent to the linear scale is clear of the blocks, then replace the shaft drive-belt and the Perspex dome.
  - Run the motor. Note the severe imbalance of the shaft.

Figure 8: Static Balance for Tow-Mass system

- **Dynamic Balance using Four Masses**
  - Remove the Perspex dome and set up the four rectangular blocks as shown in figure 7
  - Make sure that theoretically the shaft is both statically and dynamically balanced (refer back to **"Introduction and Theory"**)
  - Test the shaft for static balance.
  - Replace the drive-belt and the Perspex dome.
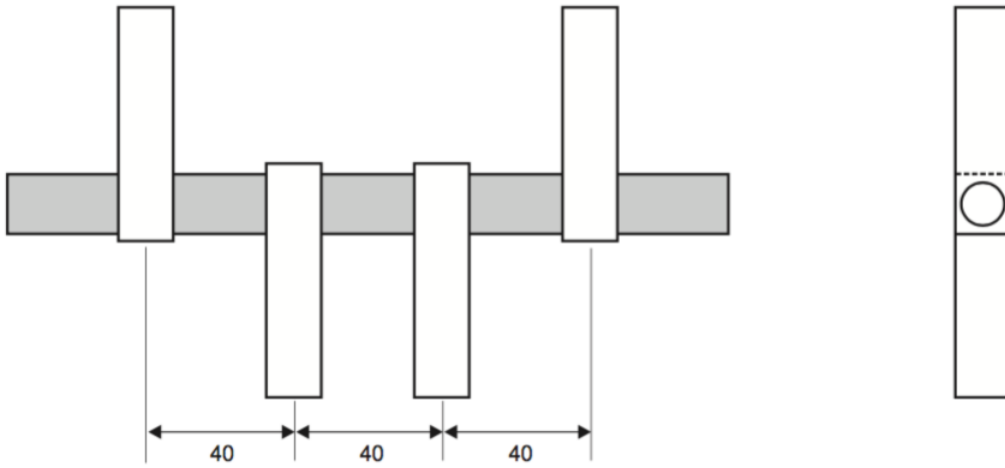 - Run the motor and observe the lack of vibration, thus, showing that the shaft is dynamically balanced.

Figure 9: Static and Dynamic Balance for Four-Mass System

Problem to be solved

Weights of 2 lb., 4 lb., and 3 lb. are located at radii 2in. ,3in. ,and 1in. in the planes C, D, and E, respectively, on a shaft supported at the bearings B and F, as shown in figure below. Find the weights and angular locations of the two balancing weights to be placed in the end planes A and G so that the dynamic load on the bearings will be zero.
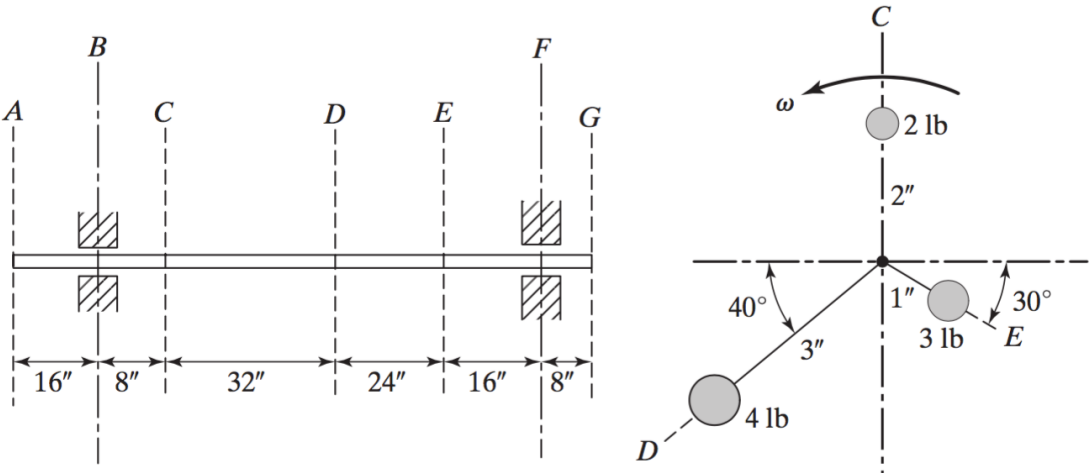
QUESTIONS:

Briefly explain/ discuss the following:

1. **What are the various methods available for vibration control?**

2. **Why does dynamic balancing imply static balancing? and Explain why dynamic balancing can never be achieved by a static test alone**

3. **What is the difference between stationary damping and rotary damping?**

4. **What causes instability in a rotor system?**

   .

5. **Why does a rotating shaft always vibrate? What is the source of the shaking force?**

LAB PROJECT # 4: Mode shape analysis of cantilever beam with ANSYS

AIM:
To model and visualize the mode shapes of a cantilever beam.

TOOL REQUIRED:
ANSYS software

BRIEF INTRODUCTION
The modal analysis deals with the dynamics behavior of mechanical structures under the dynamics excitation. The modal analysis helps to reduce the noise emitted from the system to the environment. It helps to point out the reasons of vibrations that cause damage of the integrity of system components. Using it, we can improve the overall performance of the system in certain operating conditions. We know two basic methods of the modal analysis, namely the numerical modal analysis and the experimental modal analysis. The experimental modal analysis deals with measurement input data from which a mathematical model is derived. However, it has to take different levels of analysis, from which the model is constructed.

METHODOLOGY

Step 1:
Modal (ANSYS) Project Selection
Left, click on *Modal ANSYS*, and drag it to the right of the "Cantilever" project. You should then see a red box to the right of the "Cantilever" project that says "Create standalone system"
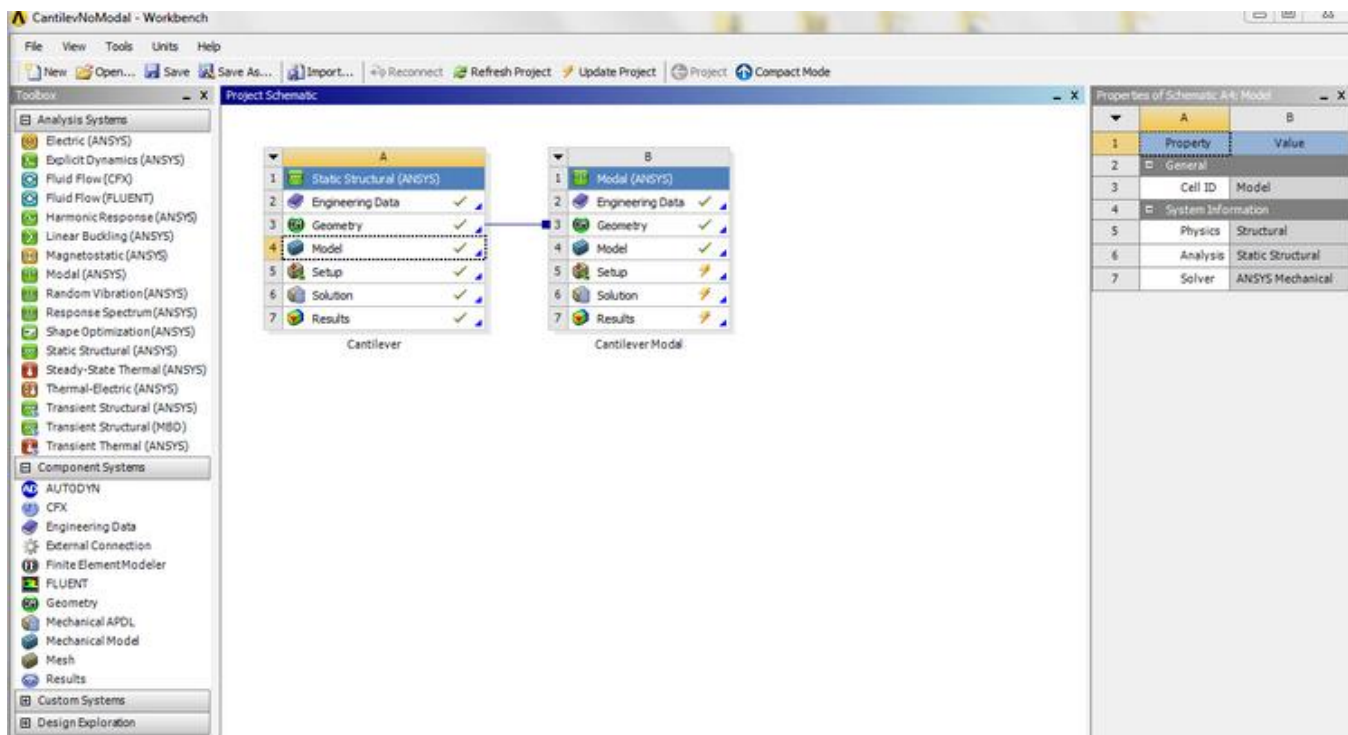

Double click on the "Engineering Data" a new window will open as shown below. Next, to add your own new material click on "click here to add new material" Name it whatever you want and press **enter**, eg

"ALUMINIUM" you should now have Aluminum listed as one of the materials in table called "Outline of Schematic B2: Engineering Data", as shown below.
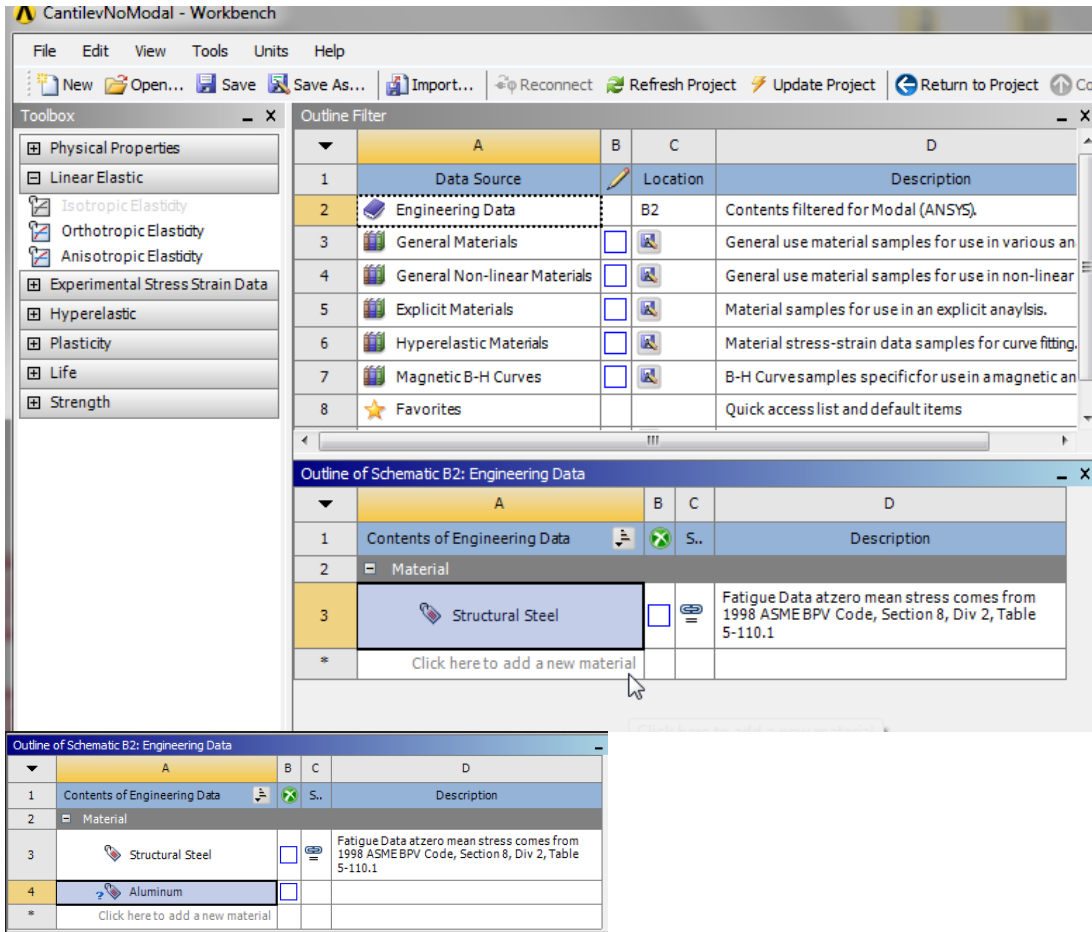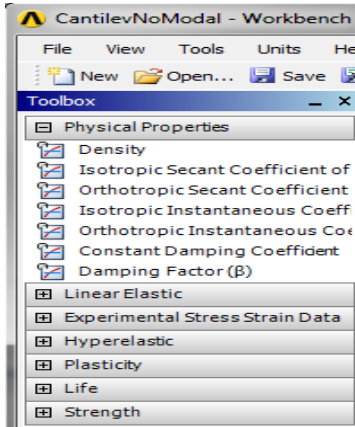

Then, *(expand) Linear Elastic*, as shown below.

Now, *(Double Click) Isotropic Elasticity*. Then set *Young's Modulus* to 70e9 Pa and set *Poisson's Ratio* to 0.35 , as shown below
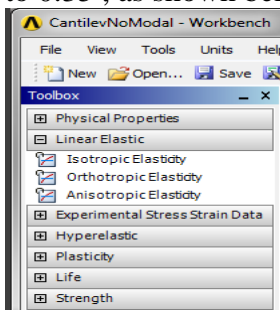
Next, *(expand) Physical Properties*, as shown below

Properties of Outline Row 4: Aluminum		_ x
▼	A	B	C
1	Property	Value	Unit
2	⊟ Isotropic Elasticity		
3	Derive from	Young's Modulus and Poisson's Ratio ▼	
4	Young's Modulus	7E+10	Pa ▼
5	Poisson's Ratio	0.35	
6	Bulk Modulus	7.7778E+10	Pa
7	Shear Modulus	2.5926E+10	Pa

CantilevNoModal - Workbench

File   View   Tools   Units   He

New   Open...   Save

Toolbox _ x

⊟ Physical Properties
- Density
- Isotropic Secant Coefficient of
- Orthotropic Secant Coefficient
- Isotropic Instantaneous Coeffi
- Orthotropic Instantaneous Coe
- Constant Damping Coefficient
- Damping Factor (β)

⊞ Linear Elastic
⊞ Experimental Stress Strain Data
⊞ Hyperelastic
⊞ Plasticity
⊞ Life
⊞ Strength

Now, *(Double Click) Density*. Then, set *Density* value , as shown below

Properties of Outline Row 4: Aluminum		_ x
▼	A	B	C
1	Property	Value	Unit
2	Density	2700	kg m^-3 ▼
3	⊟ Isotropic Elasticity		
4	Derive from	Young's Modulus and Poisson's Ratio ▼	
5	Young's Modulus	7E+10	Pa ▼
6	Poisson's Ratio	0.35	
7	Bulk Modulus	7.7778E+10	Pa
8	Shear Modulus	2.5926E+10	Pa

Now, the material properties for Aluminum have been specified. Lastly, *(Click) Return To Project*,

Save your project periodically, as you are working. ANSYS does not have an auto-save feature.

Step 2:
Geometry: Attach the Geometry
In order to attach the geometry, right click on the *"Geometry"* and either click on DM *"New DesignModular Geometry"* or import the geometry through *"Import Geometry"*
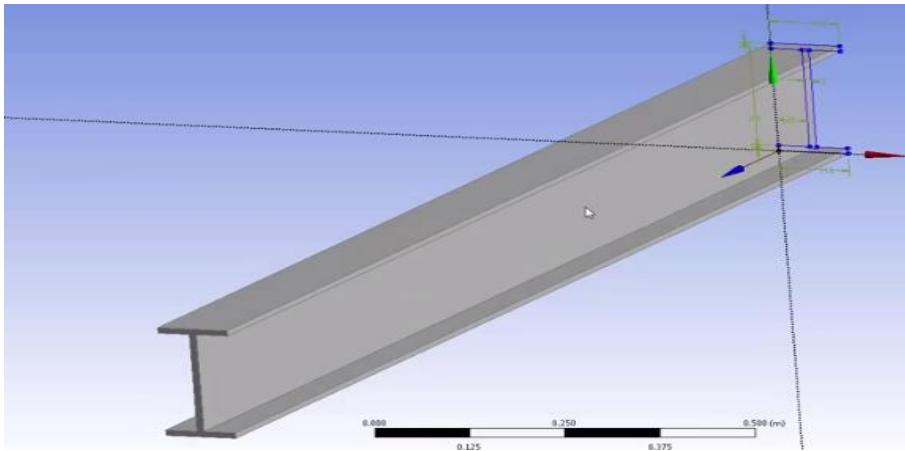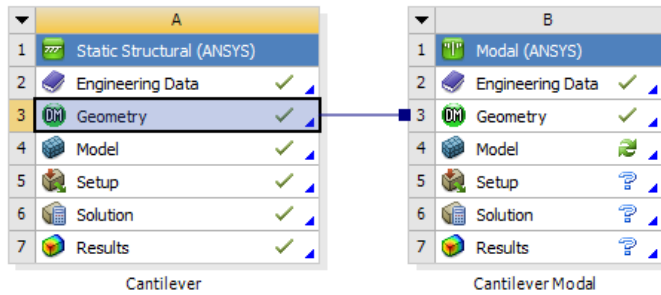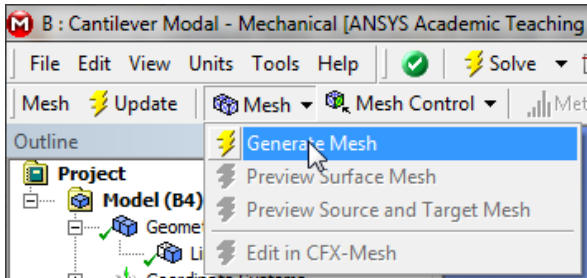
Step 3:
Generate Mesh
Go back to "project" right click or double click on "Model" and select "Edit", click and a new window will open.

To generate default mesh:
First, *(click) Mesh* in the tree outline. Next, *(click) Mesh > Generate Mesh* as shown below
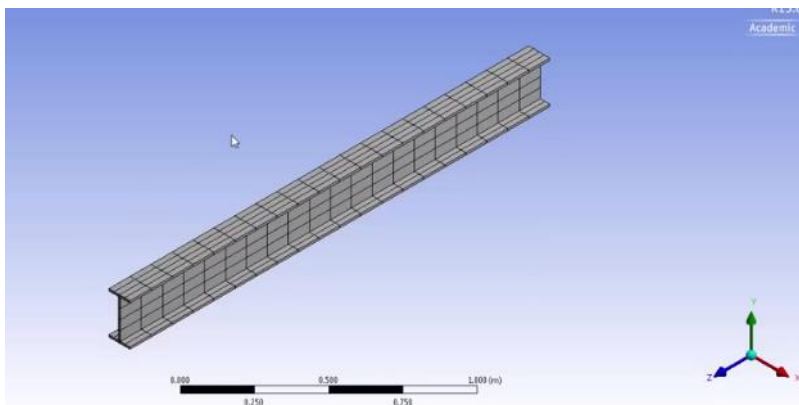

To generate Size mesh
In this section we will size the mesh, such that it has ten uniform elements. In order to size the mesh, first expand *Sizing* located within the
Details of "Mesh" table. Next, set *Element Size* to 0.40 m, as shown below.

Now, *(click) Mesh > Generate Mesh* in order to generate the new mesh. You should obtain the mesh, that is shown in the following image.

Step 4:
Material Assignment
At this point, we will tell ANSYS to assign the Aluminum material properties that we specified earlier to the geometry. First, *(expand) Geometry* then *(click) Line Body*, as shown below.
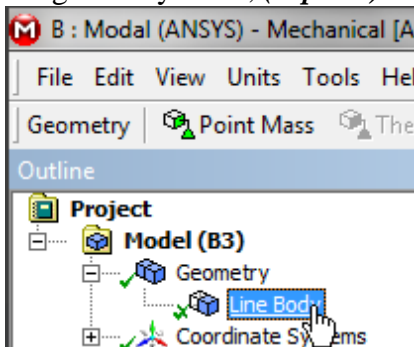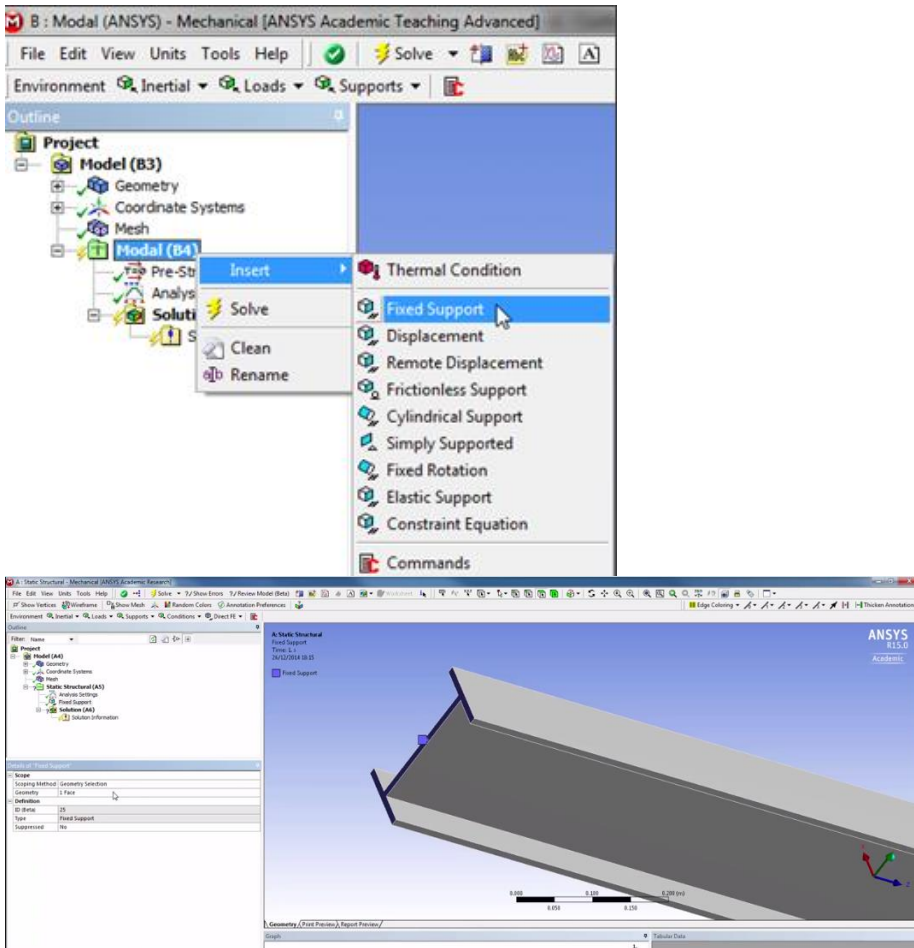

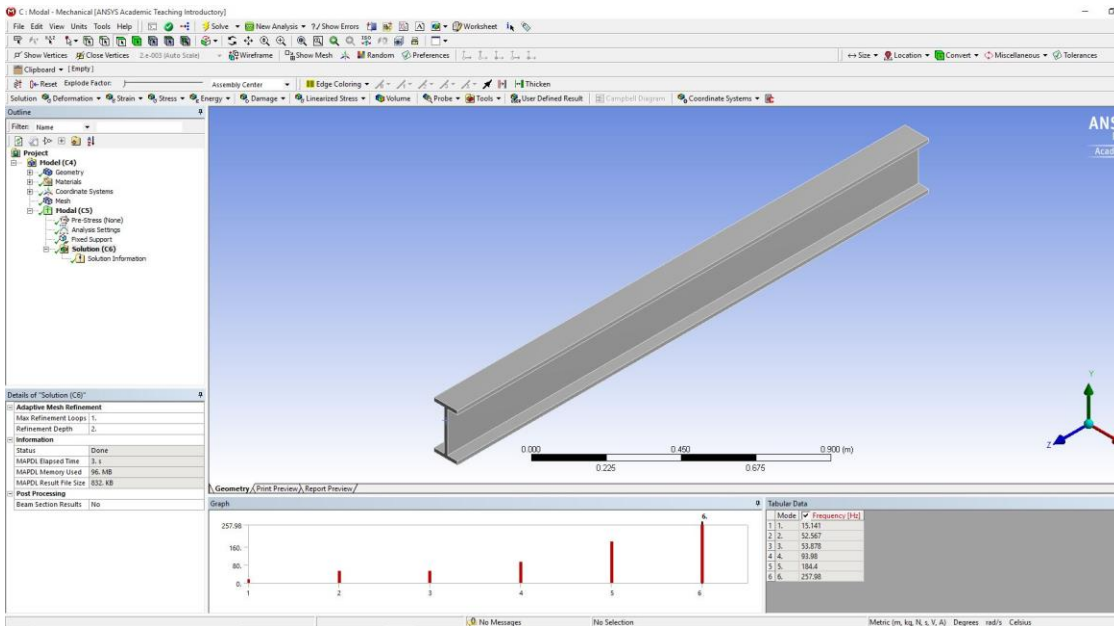Then, *(expand) Material* in the "Details of Line Body" table and set *Assignment* to Aluminum, as shown below.
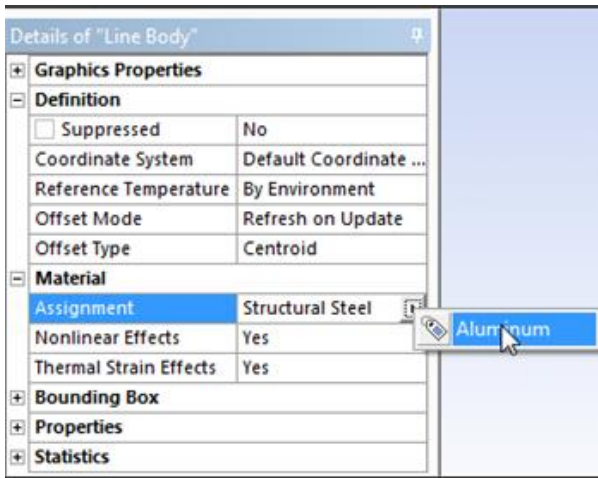
Step 5:
Assign Fixed Support
First, *(right click) Modal > Insert > Fixed Support*, as shown below.


Next, click on the "**solve**" button and get the results as shown below.

To see the model right click on the "graph" below the "geometry" window and select all. Right click again on the shaded blue region on the graph and click on the "**Create Mode Shape Results**", "**total deformation**" 1 to 6 results will appear under the "**solution (C6)" as shown below.**
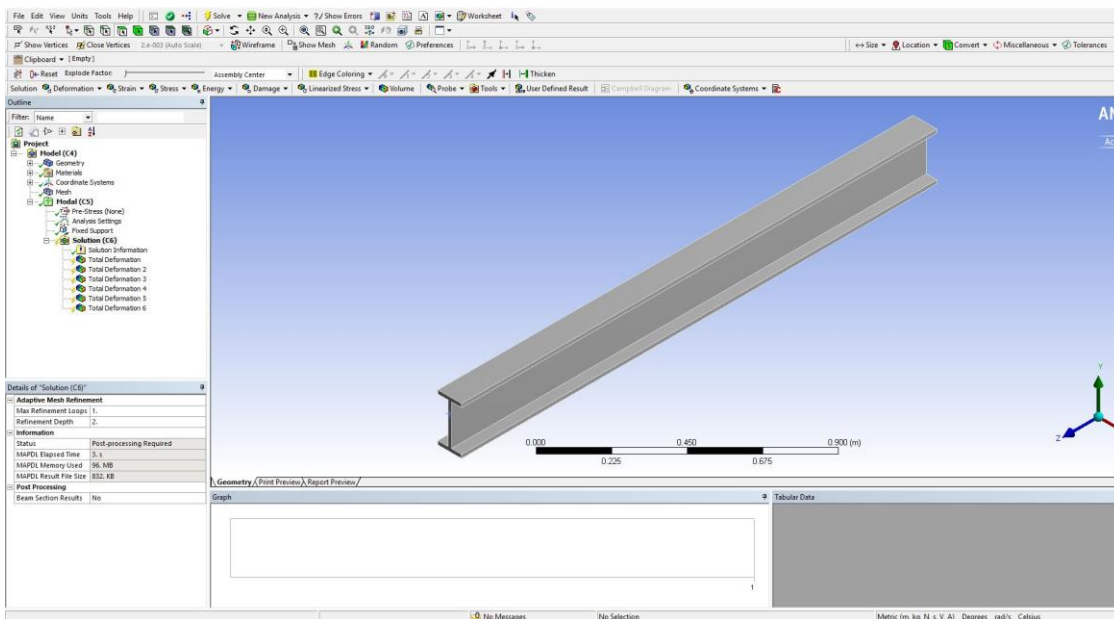

Then, click solve to obtain the mode shapes "Total Deformation" results as shown below. Here, we will tell ANSYS to find the deformation for the first six modes.
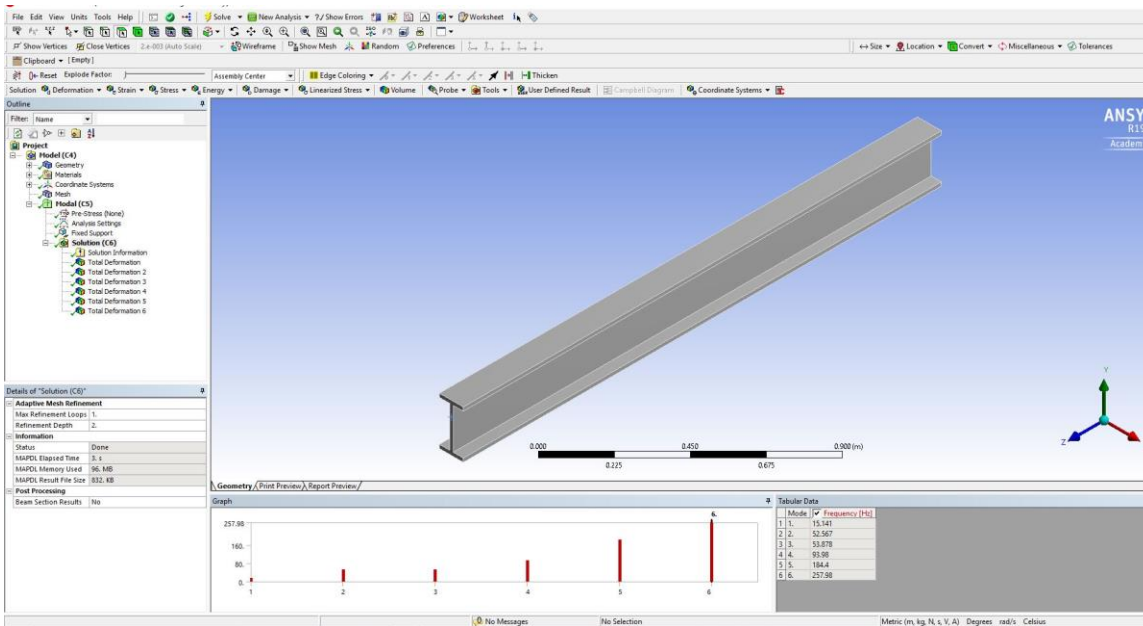
Select "total deformation 1" to visualize the mode shape result 1, select the rest for mode shape 2, 3, 4,5 and 6.
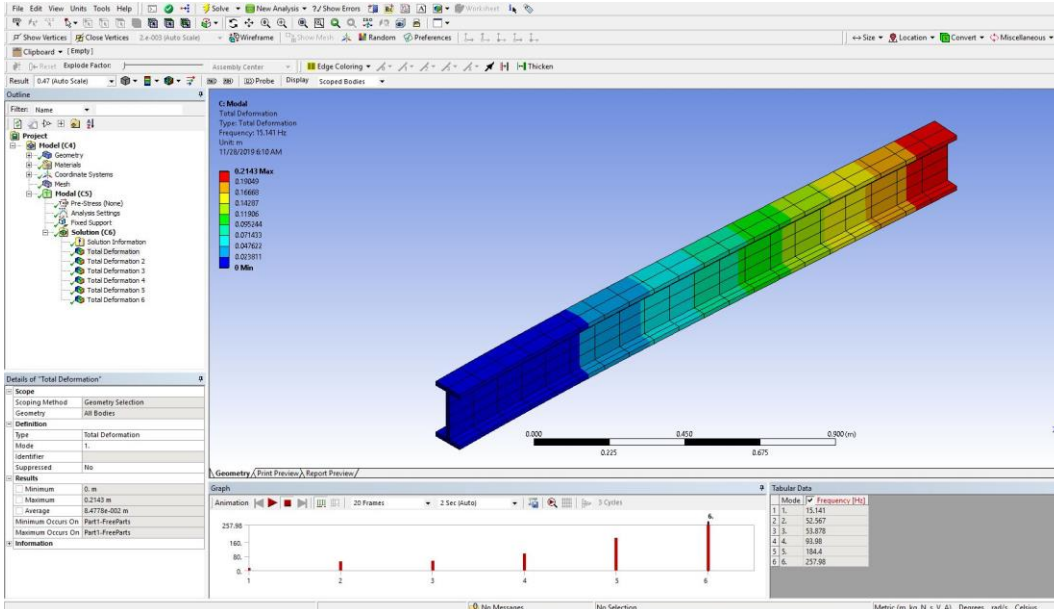
In order to visualize and run the simulation result click the "play" as shown below. Then, we will be able to see the shapes of the six modes. Additionally, we will be able to watch nice animations of the six modes.
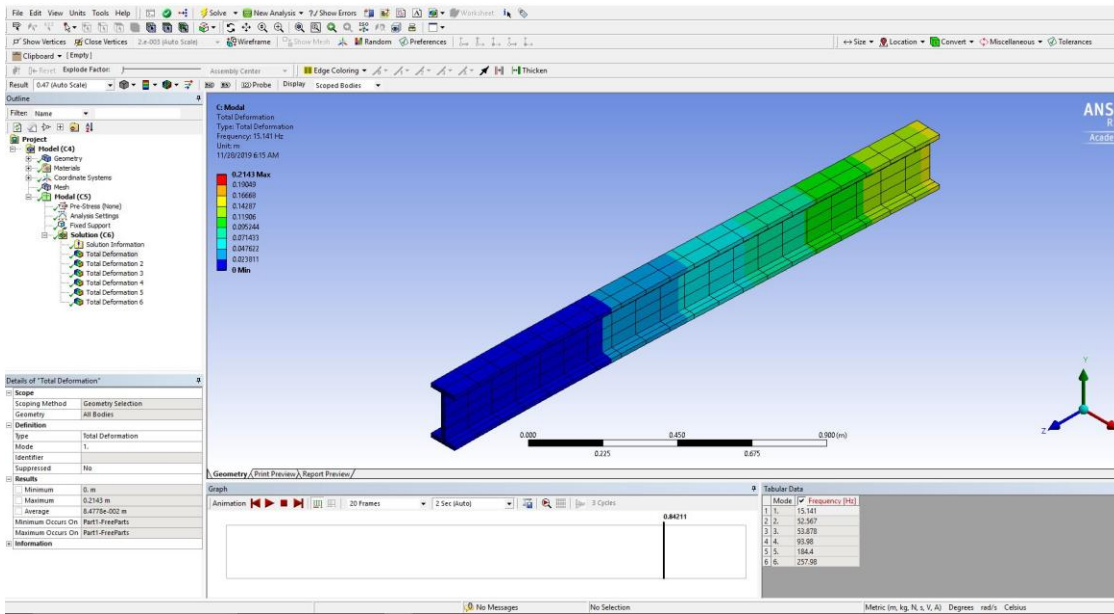
WORK TO BE CARRIED OUT:

Based on your group select and use the given specifications below, and select the material properties of the corresponding material assigned from (Table 1: Material properties). All the dimensions from the given figures below are in mm.

	BEAM TYPE	MATERIAL ASSIGNED
Group 1	I beam	Aluminum
Group 2	I beam	Steel
Group 3	I beam	Titanium
Group 4	T beam	Aluminum
Group 5	T beam	Steel
Group 6	T beam	Titanium
Group 7	O beam	Aluminum
Group 8	O beam	Steel
Group 9	O beam	Titanium
Group 10	L beam	Aluminum
Group 11	L beam	Steel
Group 12	L beam	Titanium

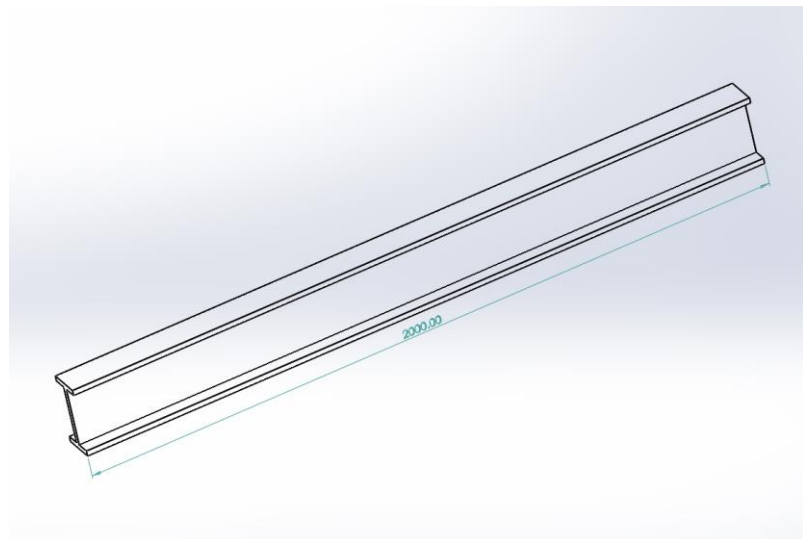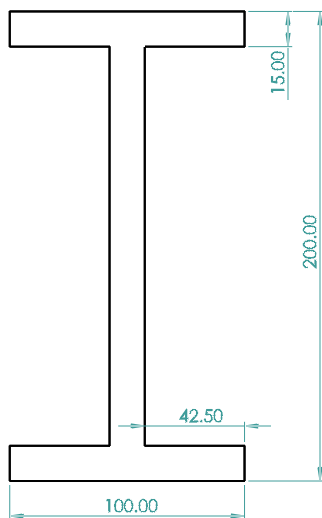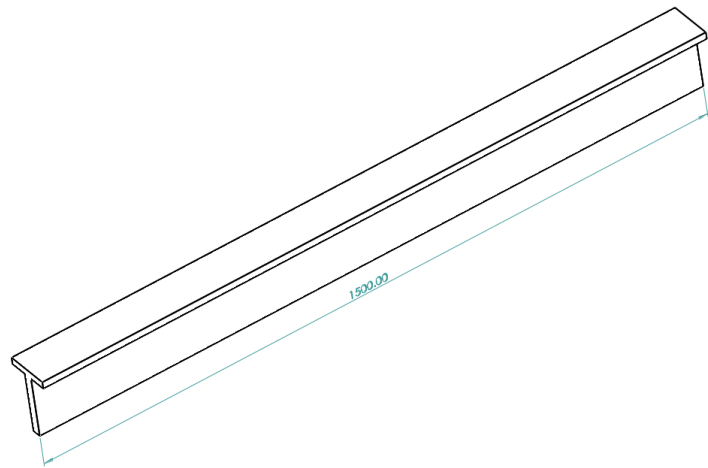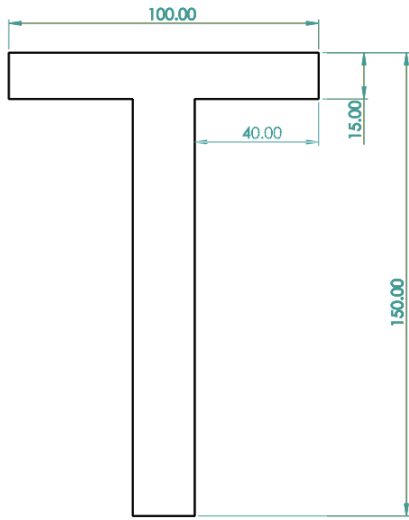*Figure 10: I Beam Dimensions*

Figure 11: T Beam Dimensions

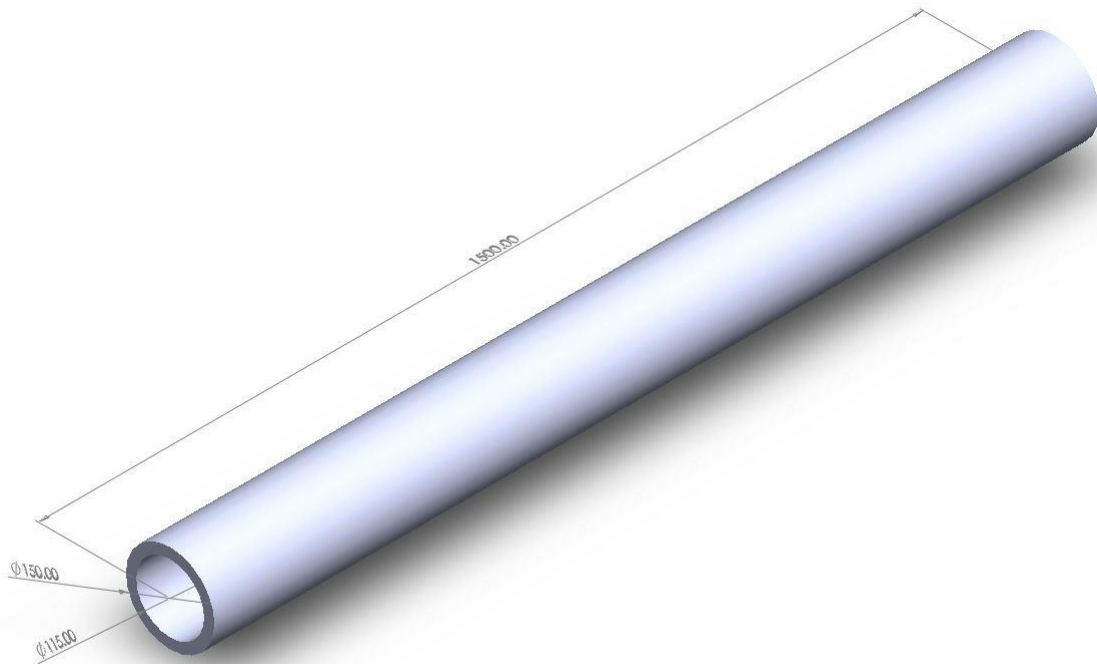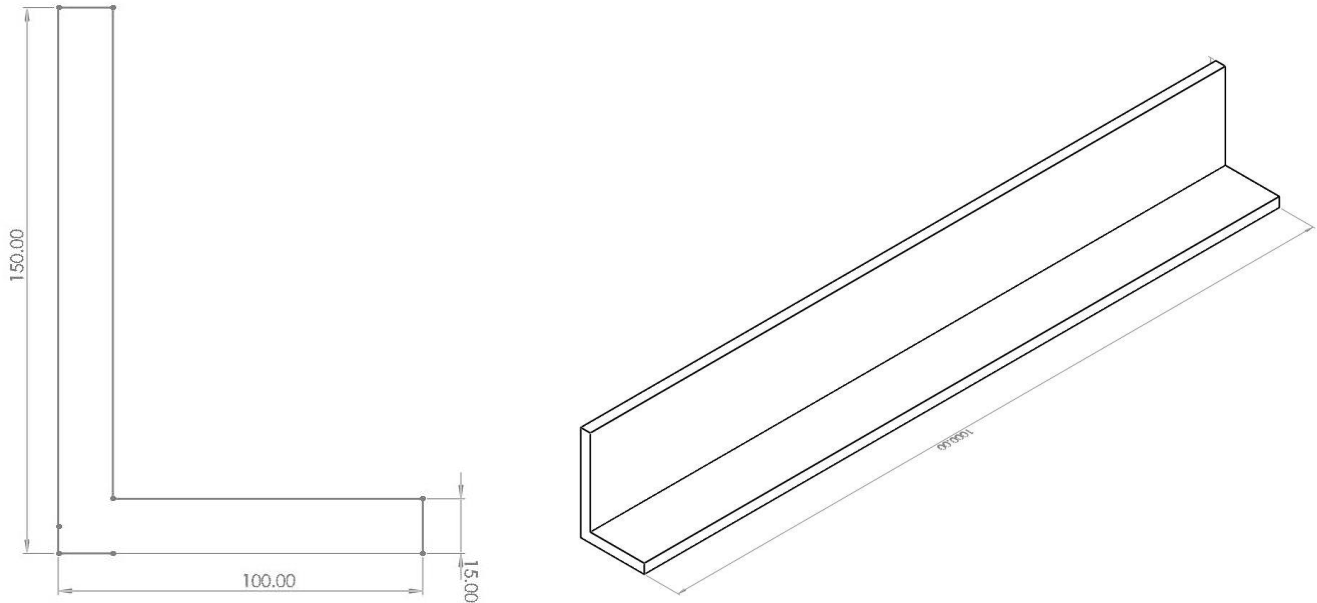*Figure 12: O Beam Dimensions*

Figure 13: L Beam Dimensions

Table 2: Material properties

Material(s)	Young s modulus (E) [GPa]	Poisson's ration (v)	Density (ρ) [Kg/m^3]
Steel (Carbon steel)	207	0.292	7850
Titanium (Aluminum alloys)	114	0.342	4430
Aluminum (Aluminum alloys)	71	0.334	2700

1. **Prepare your report based on the given template, your report should include pictures of all the major steps and they must be labelled/named.**
   a. **Abstract**
   b. **Introduction**
   c. **A brief procedure that includes (refer back to METHODOLOGY):**
      i. **Necessary steps.**
      ii. **Attach fixed support at one end.**
      iii. **All the images/pics of the main steps.**
      iv. **First, carry out the simulation based on the default setup of "default details of mesh", then**
      v. **Carry out based on the given setups from table 2 as edited "details of mesh":**

Table 3: "details of mesh"

Relevance	100
Relevance center	Fine
Element size	20 mm
Smoothing	High

   d. **Results**
      i. **Under this section you are expected to show the solution of your simulation from the "edited details of mesh" only.**

ii. Pics of <u>all the 6 mode shapes</u> analyzed from the edited "details of mesh" are expected and must be labelled.

iii. A table containing the mode shapes frequencies for both "default details of mesh" and "edited details of mesh".

Table 4: ……………………

Mode shapes	1	2	3	4	5	6
Frequencies						

e. **Discussion and Conclusion**

i. Discuss on the results obtained. Provide a comparison mode shapes table of the "default details of mesh" and the "edited details of mesh"

Table 5: ……

"default details of mesh" frequencies					
"edited details of mesh" frequencies					

ii. What Conclusions could you draw?

iii. Based on the results obtained from section "e-i", which one is better? And Why? What are the ways of improving the results?

iv. Why the choice of Element Type is important in any FEA software?

v. Which Element Type you chose and why? Please explain in detail.

All the work carried out should be burn on CD/DVD and submitted together with hard copy of the report. (Hard Copy conditional to university opening, otherwise softcopy via Microsoft Teams Assignment)

- **PDF report**
- **Figures of the main steps involved (see what is under METHODOLOGY)**
- **Figures of the 6 mode shapes**
- **Print preview and report preview files**
- **ANSYS Simulation, rendered avi video file of the simulation.**