After Mid.

Finite Element Method (FEM) Module):-

→ If we take infinitly small we get analytical i.e actual value.

But when we take a large piece and apply simpson rule we get approximate solution.

→ In ordinary differential we have slope in one variable.
while in partial differential we have more number of variables.

(*). Finite E.M is numerical method used to solve
Partial differential eqns.

→ In PDE we represent problem that involves space & time.
→ Structural Eng. problems can be represented by PDE.

(*) FEM Give approx sol.

First we get displacement.
We diff to get moment.
& We again diff to get shear.

$F = Kx$ → disp.
Known force, Known stiffness

8/5/2019.

Steps:-

① Apply force.

② Then provide support in form of reactions.

③ We find resisting displacement (x). & resisting forces of structure.

This is called analysis we do bcz.

① In order to change dimension

② & Material which help in sustaining of its self weight.

④ This help in designing

→ If we get quantity estimate from which we get cost which is 5-D of BIM.

In Freecad:-

Export objs from Part design

① module to part module.

(2) Apply analysis solve ③ used FEM.

④ Apply material. (Steel or concrete)

(4) Apply constraints (boundary condition).

⑤ Apply load.

⑥ Mesh (discretize), it → divide a str. into

(7) Apply Section (geometry/dimension), finite

⑧ Perform analysis.

⑨ Then check analysis

Used:-

(1/2) This analysis help in designing to find appropiate dimension & material.

Steps:#

i. Open new document.
→ Give name as caltiliver.

ii. Move to part design.

iii. Click create new sketch.
iv) Select XZ plane & press OK.

v) Create a line from origin horizontal.
Press escape.

vi) Select a line. (⊢|)→ Fix horizontal distance. → Increase 3000mm.

Then closed.
→ Press VF → to show Comptelly

(1/2) Then move to part.

Advance utility shape.

edge from vertices.
Press control & select edge.
Green dot show.
→ create.
Model.

Body select & press space
box → Invisible.

→ (3) Used FEM module.

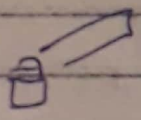
Select FEM.

"A" Analysis solver.
Select. → software used
= Calculac ((x).

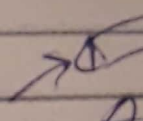
Solve $F = kx$.
we get x after solving.

○ → create FEM material.

Name → & select Concrete - generic.
Y. M = 32GPa.
P. R = 0.17.


Create FEM constrain
click.

"Add." → select corner edges
a. Rim. then add.
Press ok


Create FEM force
Add reference
→ click next corner.
Give value & 10,000 → OK

Select edge → Green.

Now mesh. to convert it into small space in order to integral.

Select ⬡ Mesh

Size 300mm

& apply. → Preob ok.

→ Apply section.

FM1 been Create section.

250mm width.

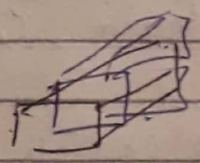
300mm height.

→ 1 ~~ft~~ length → 9" width.

Press ok.

Now add orientation

Rotate simply.



→ Save.

select first.

select 3 dots → give path.

→ static (No change & position & load constant).

Select write input file

Objective

→ To Create matrices & spring & applied

Edit input file.

Nodes → end point.

B 32 R → add R.

68 → Node on which support.

Beam section

250, 300 → Invert.

Boundry. → restrains.

6 → unknows as in 3D.

- 10000 load.
in down word.

→ Run Calculate.

Calculate result.

Deflection = 0.5.