12/10/22

Tabs ⟶ (File, Geometry, ...)

section (str, node, ...)

Ribbon bar (File ⟷ workflow (Red))

▽ ⟶ customize quick access toolbar

└⟶ More Commands (To add/remove tools)

→ for any tools, use search engine,

→ ~~Imp~~ Note

Only line can be import in DXF format not in DWG format.

{ Ribbon close ⟶ ~~Section~~ Recent file close

☒ ⟶ Application close }

① Geometry

using grid → Grid → beam grid → Edit (RS corner)

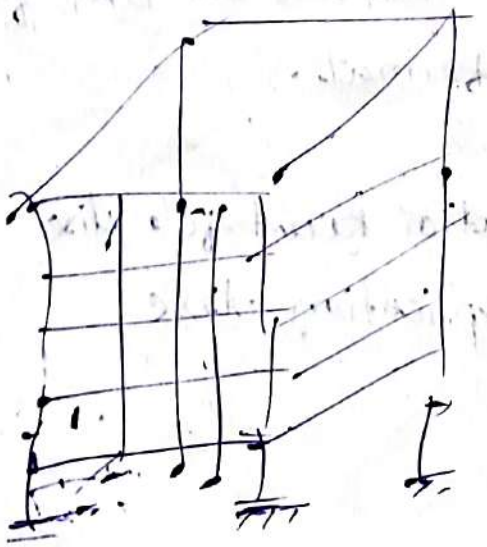
Click snap node / beam → Now join nodes using move to create beam → close

② coordinate value

③ Structure wizard

Model → Tons model → Frame (RCC)

→ Bay Frame → give details → close → Paste prototype model → OK



1,39,000/- price

(13) arch

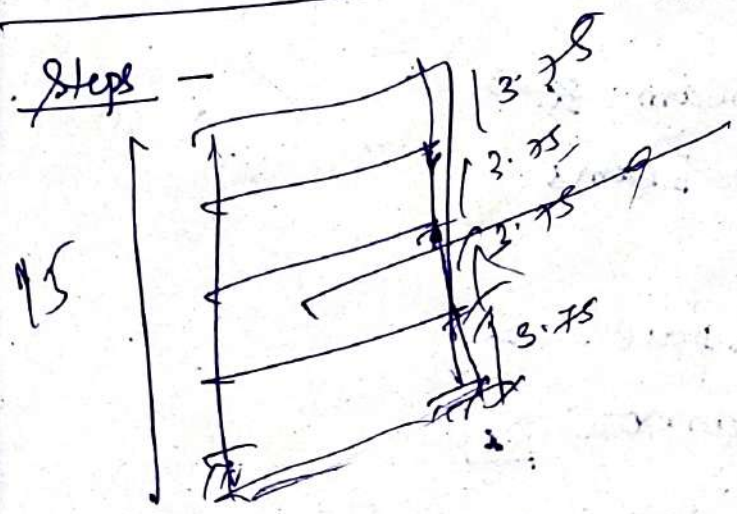
22,00/- (11)

Q//

G+4 with the Length = 12m
width = 15m
height = 12m

col. Size = 450×450 mm
beam = 380×230 mm
slab th = 160 mm
M25, HYSD 500 main, HYSD 415 sect
cc = 25 mm
DL, LL, FL, plate load,
combination
design parameters as per IS 456;

Steps –



2.25

Steps

1. Create job
   Geometry →
2. Str. wizard → frame
   (Geometry → beam section)
3. Add beam (Mid point, per. intersection curve beam)

4. Beam layout
   (To know beam locations & nodes that are. both side of beam)

5. Renumber beams
   (To change beam no)

6. Stretch beam
   (To stretch a beam in either x, y, z direction)
   (Through a dist.)

7. View → label setting →
   label → beams → beam ends

8. Merge Selected beam
   → To merge two beams

9. Rt click on workspace →
   quick comm → arrow
   → customize quick commands popup

10 → RB click on workspace and select cursor → for all cursor

11 → Break beam → node cursor
→ Right select the node (with mouse)
→ break beam

12. low thick slab → plate element (≤ 300 mm)
high thick slab → solid element (> 300 mm)

13. Mouse ball → zoom in / zoom out

14. Select any floor → RB click → New View → active window → OK

15. Go to isometric view

16. View → tab section → view management → export view

17. Geometry → plate section → add plate

18. View → display whole str.

② property

RCC → define
Steel → section delete
Slab → thickness

→ property → Rectangle → beam &
column

→ select tab

→ select tab → beam select →
parallel → X & Z (beam)
→ assign to selected beam → assign

→ ~~Select~~
→ select tab → beam select →
parallel → Y → assign to
select beam → assign

→ Slab thickness → Thickness →
add

→ plate section → parallel →
XZ section → assign to selected
plate → assign.

③ Materials

* Select concrete → edit → give fck → align to view → align

→ ↑

④ Specification

Node Spection → relationships & floor diaphragm

⑤ Support

create → fixed → view → front → select → align to select node

⑥ Loading

definition → seismic → add → (1st)

type IS-1893 (2016) → generate →

zone II (Rayagada) → gen

RRF → building with ordinary RC structural walls

→ Imp. factor → All other building → 1

→ soil → medium → str type → RC MRF building

→ DR 5% → F-depth → roof depth → 3m (Mount Rushmore)

→ generate → add

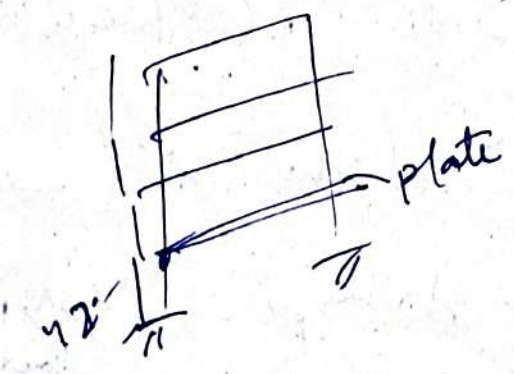
Wind defenit → add →
→ code IS-875 (2015) → add →
close

→ Type I wind → add → Intensity
→ generate → str. type →
Rec. clad bu

→ Use custom option → 50 m/s.

→ wind parameter
$K_4 = 1, K_2 \propto h$
class of str → Open
Terrain → Terrain 1-2 (medium)
Terrain 3-4 (soft)
→ K_3, K_4 → enter → add
→ factor → (1) → add → close

→ expose → align to view (whole)



Load case details

Add → Seismic H (SHX)
Seismic V (SHY)
wind → WLX, WL-X,
WLZ, WL-Z
dead load → DL
live load → LL

→ SHX → add → seismic load → 1
SHY → add → seismic load → 1
WLX → add → wind load →
(wind ward) (lee ward (inner))
→ Select type → reg. →
WLX → add → wind load → factor 1
Apply for (Z & -Z).

→ Self dead load → weight → Y → +1 (GY)
member load → weight → uniform force → 12 KN/m (out wall)
→ 2.5 KN/m (GY)

Assign

→ Assign DL → assign to view (selected)
ontward → top view option → outer wall.

→ assign to selected beam → assign

→ Similarly for inner wall.

line load

→ plate load → $W_1 = 4.5$ kN/m
(+) ve dirn (+)

→ floor load → Y range
(+Y) 7.5 m — 1 Fm
(extra for assigning self weight)

→ pressure. → (-) 4.5 kN/m²

Rt click → plate cursor → click cursor to plate → assign to selected plates → assign → yes

→ Combination load
load case → add → auto load combination → IS 456-2000 (IS 456 T-18)
~~add~~

→ Generate load case → add.

Foundation design → load envelope

Analysis

Comand → Perform analysis → all

→ add → Run analysis

Design steps

→ click on concrete on section.

→ Select parameter. (clear, Fc, fymain, fysec, Ratio) → 4

OK

→ define parameter.

put suitable values & add.

→ Clear cover → assign to view → yes
(similarly for other)

→ command

Add all parameter.

→ design beam → beam section →
// to x & // to z →
assign to selected beams → assign

→ design column → beam section →
// to z → assign to selected beam →
assign.

→ design element (plate) →
assign to selected plate →
assign

→ Analysis → Run analysis

⇒ Output file

⇒

Workflow

→ ~~post processing~~

Utilities

Geometry section → group tool →
create beam & colm → go to beam
option → go to view tap → select a
flow → go to selected geometry →
associate
(similarly for colm)

→ go to ~~geo~~ utilities → group →
beam → highlight

post prouny → result → Fx, Fy, Fz

Mx, My, Mz → Layout → All results can be seen.

1

→ Animation → deflection → apply → ok

→ Report → setup report → output → add all summary & failed member table & static check near

→ setup report

RCDC - Reinf. Concrete & Design Cal.

Workflow design → RCDC start menu.
→ staad pro limit feature

Design element → RCDC

project → P1, client → C1, engg → E1

Codes → IS 456 + IS 13920

create project → col. & wall → define level data

sheet of C4

- setting tab (from design setting to load combinations)
- Design setting → ductile → live red factor
  → seismicity both → crane width → ok
- Rein setting

  col % → (0.8 - 6) & long. repair (25 - 30 mm)

→ ok

→ Detailing & drawing setting → select all → OK → OK
(don't click use level description)

→ Zone & rebar setting → OK

→ Basic load case → give load typ → primary load case → OK → load combination.

→ load comb. → Add template → ~~proj. str~~ → OK

→ crane width → temp → proj. str (Temp.)

→ select crane width & steel limit → OK

design

→ Auto design

If column fails

→ select fail col. → Rt click →
Redesign Sect, Redesign col., Redesign Level

→ view tab

primary load case, L.C. Mem ⇒

Report

BBS - bar bending schedule → generate BBS

Note

→ Same procedure for beams also.

~~Slab design~~
(After colm → footing)
Next do all as same as column. So
~~File~~ → col. & footing simultaneously.

Slab

Work flow → Adv. concrete design

→ give the data → Create

→

Setting → design parameter → put all parmtrs & design

→ Similarly go for steps as done in beam.

Foundation

→ loading → load env. → add → envelope 1
→ type → none (only R↑ base),

→ select load from available loads ↔ consider dd & ll comb. → Run analysis →
stay in M. Mode → Workflow →
Foundation design → all support →
select env. 1 → Staad foundation adv →
limited features → ok → project info
→ Correct proj. info.

→ Re Proj info → Model view → Show load values → scale (for size)

Foundation plan

→ linear grid setup → pedestal & anchor bolt

Loads & factors

→ for all type loads -

Foundation design

→ create a new job for new foundation.

Job setup → create a new job

data input → create job

Isolated footy job → deg parameter → connect & retry.

Cohesive soil → grid data

footing geometry → "

slab & anchor → default

design

Steel structure

14/10/22

Q/ Truss → length 10m
height 5m
inclined rafter angle 5m

Use I section & angle section.

Material → steel.

load → dead (self wt = 1)

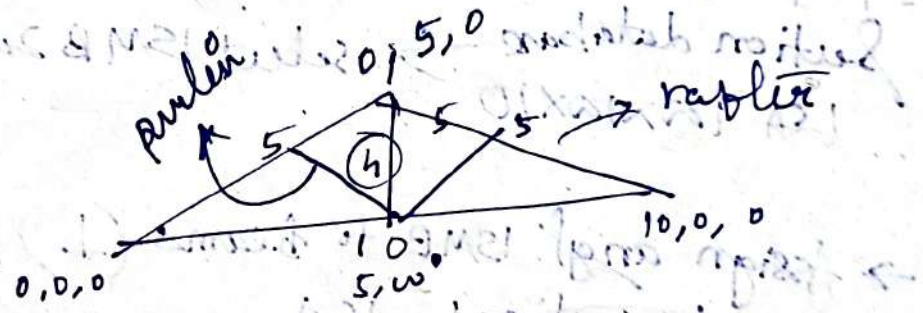
Mem. weight (dead) → 7.2 (out. truss)
→ 5 (inn. truss)
→ 3 (purlin)

live load → 3.13 (truss)
1.5 (purlin)

Wind load → Wx, Wy.

IS-800,

Soln



→ Select node cursor → give coordinate & join.

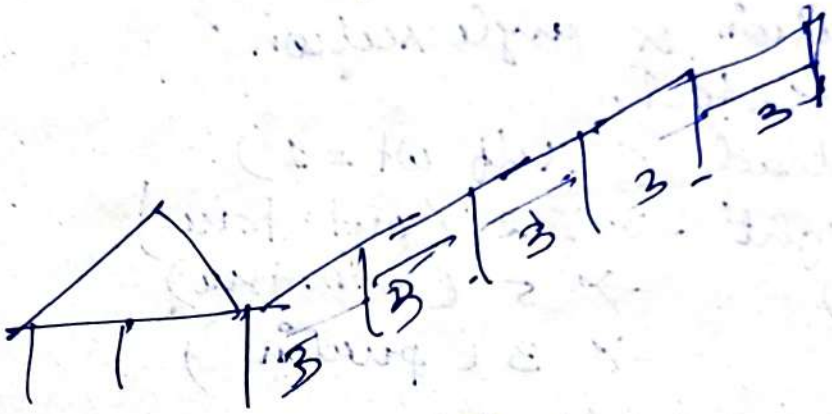
→

→ select both rafter.

→ go to beam geometry → beam setup → insert node → add (n points = 2)

→ Add beam → go to ⊥ option → joint by add beam.

→ Select node → copy → paste → give value → add beam to (use node cursor) add line.



→ go to traslational repeat → str. section select no. of step & spacing → check link steps & open base → ok

Property

Section database → select ISMB 200 & ISA 200x100x10

→ Assign angle ISMB to beams (⊥) & angle sections to ISA

→ Select tab → attributes → Mmj property

add plate (To add sheet)

geometry → plate → add plate

(See in 3D - design)

property → thickness → steel

use plate cursor → assign plate → check 3D

Material → steel edit → steel → OK

Support → select node → add. fixed Support

load

wind load

add code IS 875 → add → close

Sto type → lattice tower.

Similarly for dead & live load.

Assign load

→ While assign load rem to use ⟶ arrow to rotate & view it.

→ select a section & then select obj & after finish click whole Sto.

→ assign dl, ll of the truss.

Note

New select mode → draw new view

→ assign

→ Go for load combination

load case → add → auto load

comb → IS 456/800 → table 4

→ add

Analysis → steel → IS 800 →

parameter → Ratio 1 →

dl_max → 1m → ~~0.5m~~ MAIN YR 180 →

Profile ISMB

commands → there do we do

select

steel takeoff

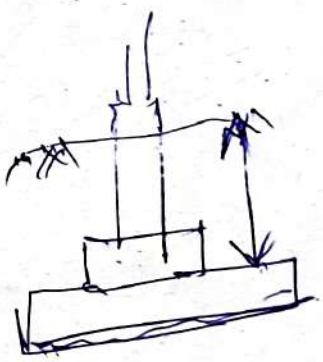
Run anly → Analyze

Combined footing design

① Analyze → workflow → foundation design → select envelope → envelope 1 → Staad Foo advance → Adv. option (not limited)

→ job setup → create new job → combined foot, IS-456, load → create new job.

② Combined footing job → Design parameters



—— ✕ ——