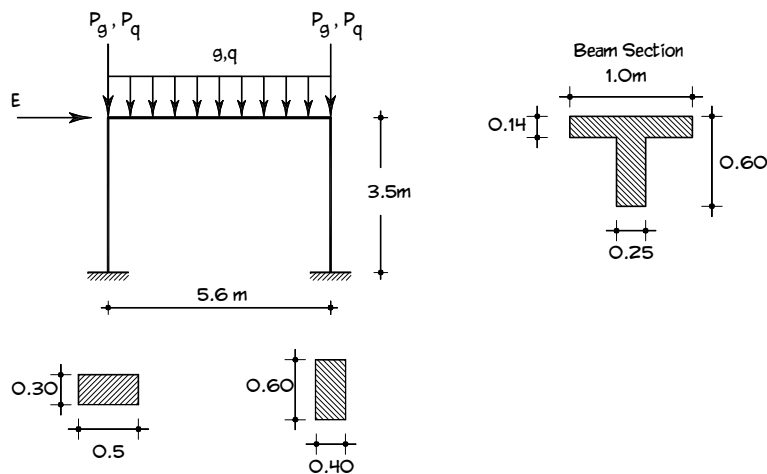


MODELING WITH SAP2000

1. Select units
2. Create geometry of the model
3. Assign restraints
4. Define materials
5. Define sections
6. Assign sections
7. Define load patterns
8. Assign loads
9. Define load combinations
10. Analyze
11. Review results

EXAMPLE-I


1. Start SAP2000
2. Set Units as kN,m,C
3. File→New Model→2D Frames
4. Set
 - Number of Stories 1, Number of Bays 1
 - Story Height 3.5 Bay Width 5.6 Click OK
5. Select support joints.
6. Assign→Joint→Restraints...
 - Check All Translation and Rotation boxes. Click OK.
7. Define→Materials
 - Add New Material...
 - Material Name C30
 - Material Type Other
 - Weight per Unit Volume 0
 - Modulus of Elasticity, E 3E7
 - Poisson's ratio, U 0.2
 - Coefficient of Thermal Expansion, A 1E-5 Click OK twice.
8. Define→Section Properties→Frame Sections... → Add New Property
9. Frame Section Property Type → Concrete
10. Click Rectangular section
 - Section Name C50X30
 - Material C30
 - Depth 0.50
 - Width 0.30 Click OK.
11. Add New Property
12. Click Rectangular section
 - Section Name C40X60
 - Material C30
 - Depth 0.40
 - Width 0.60 Click OK.
13. Add New Property
14. Frame Section Property Type → Steel
15. Click Tee section

- Section Name B
 - Material C30
 - Outside stem (t3) 0.6
 - Outside flange (t2) 1.0
 - Flange thickness (tf) 0.14
 - Stem thickness (tw) 0.25 Click OK twice.
16. Select left column. Assign→Frame→Frame Sections→C50X30. Click OK.
 17. Select right column. Assign→Frame→Frame Sections→C40X60. Click OK.
 18. Select beam element. Assign→Frame→Frame Sections→B. Click OK.
 19. Define→Load Patterns...
 - Write G in Load Pattern Name textbox. Click Add New Load Pattern.
 - Write Q in Load Pattern Name textbox. Type=LIVE. Click Add New Load Pattern.
 - Write E in Load Pattern Name textbox. Type=QUAKE. Click Add New Load Pattern. Click OK.
 20. Select the beam element.
 21. Assign→Frame Loads→Distributed.
 - Load Pattern Name G
 - Direction Gravity
 - Uniform Load 60. Click OK.
 22. Select the beam element again.
 23. Assign→Frame Loads→Distributed.
 - Load Pattern Name Q
 - Direction Gravity
 - Uniform Load 40. Click OK.
 24. Select the top joints of columns.
 - Assign→Joint Loads→Forces.
 - Load Pattern Name G
 - Force Global Z -200. Click OK.
 25. Select the top joints of columns.
 - Assign→Joint Loads→Forces.
 - Load Pattern Name Q
 - Force Global Z -100. Click OK.
 26. Select the top joints of left column.
 - Assign→Joint Loads→Forces.
 - Load Pattern Name E
 - Force Global X 180 and Force Global Z 0. Click OK.
 27. Define→Load Combinations
 - Add New Combo...
 - Load Combination Name 1.4G+1.6Q
 - Load Case Name=G, Scale Factor=1.4. Click Add.
 - Load Case Name=Q, Scale Factor=1.6. Click Add. Click OK.
 - Add New Combo...
 - Load Combination Name G+Q+E
 - Load Case Name=G, Scale Factor=1. Click Add.
 - Load Case Name=Q, Scale Factor=1. Click Add.
 - Load Case Name=E, Scale Factor=1. Click Add. Click OK.
 - Add Copy of Combo...
 - Load Combination Name G+Q-E
 - Load Case Name=E, Scale Factor=-1. Click Modify. Click OK.
 - Similarly create 0.9G+E and 0.9G-E load combinations.
 28. Save model file.
 29. Analyze→Set Analysis Options... →XZ Plane. Click OK.
 30. Analyze→Run Analysis. Run Now.
 31. Display→Show Forces/Stresses→Frames/Cables
 - Case/Combo Name=G
 - Moment 3-3. Show Values on Diagram. OK
 32. Display→Show Forces/Stresses→Frames/Cables
 - Case/Combo Name=E
 - Moment 3-3. Show Values on Diagram. OK
 33. Display→Show Forces/Stresses→Frames/Cables
 - Case/Combo Name=E
 - Shear 2-2. Show Values on Diagram. OK

Note: If you want to ignore the contribution of axial and shear deformations (to obtain hand calculation results), before starting analysis Select all members Assign→Frame→ Property modifiers → Axial 1E9 , Shear Area2=0, Shear Area3=0
In practice do not ignore the contribution of axial and shear deformations.