# STAAD.Pro Tutorial
Wood Floor Problem

1. Open STAAD.Pro Connect, initially as "New". Choose a file name and directory. Enter your name as "Title:" set "Units" to English. Choose "Analytical" and click "create".

2. The basic STAAD.Pro window should open. From the menu bar at the top choose FILE->IMPORT->3D-DXF then Browse for floor22.dxf and choose "no-change"

3. Set units at inches and pounds.

4. At this point you should have the floor plan (34ft x 24ft) on the screen. Now you need to specify some parameters and set up the analysis.

5. By default it comes up as an isometric view. To get a plan view select "View" on the top menu bar and then click the box with the top view (second box).

6. The sidebar "Workflows" should be set to "Analytical Modeling". The bar just above the structure window is set by this – so "Analytical Modeling". This top bar lists the operations in setting up the model for analysis. The order that you follow is not critical. For ease start with "Supports"
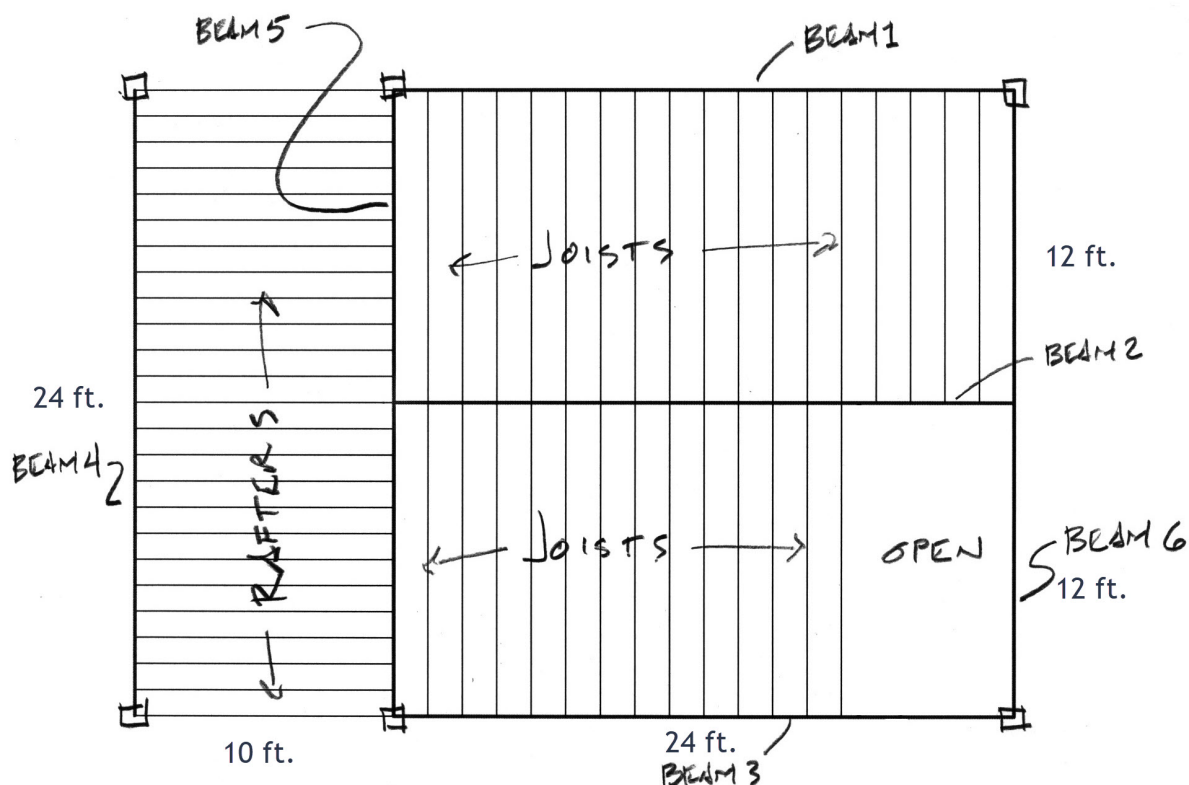
7. You need to put supports where the columns are, but like with beam
    end reactions, to avoid binding, one should be pinned and the
    others on partial rollers.

8. Pick CREATE -> PINNED -> ADD

9. This should create "support 2" on the menu. Pick that.

10. Pick the lower left corner (on B4) and ASSIGN TO SELECTED NODES.
    Then ASSIGN and YES to proceed.

11. Now you should picture the floor plan held at this point and
    shrinking a bit in scale. B4 will come straight down toward the
    pinned corner, so the roller is in the Z direction.

12. CREATE this support as FIXED BUT and release FZ. The moments get
    released later, on the ends of the beams directly. Click "Add".

13. Select this support and ASSIGN it to the upper end of B4

14. In the x direction there are 2 more columns. They both need to be
    able to slide in the x direction.

15. CREATE another FIXED BUT support and release FX.

16. ASSIGN this support to the two supports on B3. Hold CTRL to select
    more than one at a time, then "Assign".

17. CREATE fifth FIXED BUT support and release FX and FZ.

18. Select this last support and ASSIGN it to the remaining two
    supports on B1.

19. Next we want to supply information about the members. It is a lot
    easier to deal with members (beams) if they are collected into
    named groups. So first, create some groups.

20. From the top bar pic "Utilities" then click "Groups". Make eight
    groups named as below.
    RAFTERS
    JOISTS
    BEAM1
    BEAM2
    BEAM3
    BEAM4
    BEAM5
    BEAM6
    in each case start by entering the name in the box and choosing
    "beam". Then select those members with member cursor (right click
    for the courser select - beam) Use convenient view, zoom, ctrl to
    pick several, box crossing to select multiple or type the number
    if you need to add one, then ASSOCIATE. Use "create" to make the

next group. You can check by picking a Group and then Highlight. After you create all 8 close the window.

21. The next tab is Specifications. Choose that, then pick BEAM. What needs to be set here are the END RELEASES. Initially all the beams are created with fixed ends. In wood construction pinned ends are closer to the build condition, so end releases need to be set. The releases are specified by "local" axis. X is the length, z is perpendicular in the horizontal plane and y is vertical. The rafters and joists are single members and will get a release at each end(Mz, My). The beams had to be cut into several members in order to join with rafters and joists and will only get releases where they connect to other beams or columns.

22. First create the releases for the joists and rafters: Select the tab "Specifications" and then from right menu Choose: BEAM ->RELEASE pick location (first do start then do end) and pick MY MZ. Then "Add". Add both Start and End.

23. To assign, use the groups by SELECT->BY GROUP NAME then pick rafters and joists. Then in the Specification menu, "Assign to Selected Beams". Do this for both "START My MZ" and "END MY MZ".

24. In this way create a specification for MY MZ release for START and another for END. One or the other of these two get assigned to the ends of all the beams. If you hold the cursor over the end section of a beam, color will indicate green for start and blue for end. Each member has a start and an end, but it seems more or less randomly assigned (was actually set in AutoCAD).

25. Next choose the "Properties" tab. Here you set both the section dimensions and wood type and grade. The beams can be designed as Glulam sections while the smaller joists and rafters will be dimensioned lumber. To access the size table pick "SECTION DATABASE" and "TIMBER".

26. For the Beams 1-6 choose GLT-24F-V4_DF/DF. Then pick (guess) dimensions b and d (YD and ZD) for the beam. Use the standard dimensions from Table 1C in the NDS. The analysis will tell you whether your guess was safe or not, but at first just guess. (you might look up a simple span table for glulam or try d in inches = 0.9 x span in feet, a bit more or less depending on load and bracing) Initially you might make all the beams the same size or several the same size. Use the UTILITIES-GROUP as before to ASSIGN the properties to each of the beams.

27. Now choose a size for the rafters and joists. Try Hem-Fir: HMFR_SS_2X10. If this fails choose larger or if not try smaller.

28. MATERIAL has already been assigned with the wood PROPERTY tab. You can check by selecting the MATERIAL tab.

29. At this point check that units are set to feet and pounds: the top bar menu "Geometry" under Structure Input Units is the lower right icon. Open it and click foot and pounds.

30. Now set the loads. Pick the "Loading" tab. Pick LOAD CASE DETAILS and ADD. Then set load type(e.g. dead). ADD a load case for each load type you will use: DEAD use given DL (on joists and rafters) + selfweight to include members, and the given LIVE (for joists), and ROOF-LIVE (for rafters). So 5 load cases including selfload. Use loads given in the assignment sheet. Add to the load title so you know which load is what. Work through one load at a time: pick NEW and add the load case. Pick ADD and add the load. Use "Area Load" for all the surface loads and assign them to groups (either RAFTER or JOIST) Don't forget the Negative for downward in global Y. Alternately you can use the "One-way" command to do the same thing.

31. Once all of the Primary load cases are created, you need to make the combinations. With Load Case Details selected choose NEW->LOAD CASE->COMBINATIONS. Then make 1 load case combination for all loads:
1. Self + DL_roof + LL_roof + DL_floor + LL_floor
(normally you might have other combinations, but this one will do)

32. This is enough information (at last) to perform the analysis. Choose "Analysis" from the top bar and then select "Define Commands…". Then use the tab "Perform Analysis" and check "All". Click Add. Then close.

33. Now from the top-top bar choose "Analysis and Design", and "Run Analysis".

34. A window should appear (STAAD Output Viewer) and when it runs you should get 0 Errors, Maybe a Warning (about Area load being discontinued) or a Note. Select View Output File to read these. Then close and go to Post Processing.

35. Initially make sure your model was correct. Check some graphic output (deflection, moment, shear, etc.) to make sure things are reasonable – e.g. no negative moments.

36. On the Workflows left sidebar select "Postprocessing". Change the view to Isometric, check deflections (they should be downward). Right click and select "Structure Diagrams". Set scales to be visible. Check moments to be sure end releases are properly set (no negative moments).

37. Once the model is correct you can add a code check for pass/fail by the AITC Timber Code. Under the Workflows sidebar return to "Analytical Modeling" and "Design" from the top bar. On the top-top menu select "Analysis and Design". Under the material scroll down to Timber and select it. Select from the side bar tabs DESIGN and them TIMBER. At the top of the window on the right pick AITC **1994.** (it should come up by default)

38. SELECT PARAMETERS: it is better just to pick the ones you are planning to use because the output is then easier to read. You should use:
BEAM = 1
CFB = Size Factor for bending (for joists and rafters)
CR = Repetitive Member Factor (for joists and rafters)
CV = Volume Factor (for Glulam)
CDT (Load Duration Factor) 1.25 on roof   1.0 on floor
LUZ(bracing against LTB) use ½ joist or rafter length

39. DEFINE PARAMETERS: here you put numbers to the above. Some apply to different groups so some need to be set differently for different groups. CD and Luz are different for rafters and joists. There is a little Excel sheet on our website to calculate CV for the glulams.

40. COMMANDS: here add code check. That will do a check and show Pass/Fail.

41. Now run the analysis again. "Analysis and Design", and "Run Analysis".It will run in another window and give an error report.

42. Once again if there are any errors, those need to be fixed. Some warnings indicate a mistake in the model (like wrong units that produce strange values) other warnings are not important (occasional mathematical instability errors that STAAD corrects with springs). For example, the Area Load command is ok for this floor system. STAAD is also sometimes entering E in ksi for some reason which you can correct manually if it gives you that warning by adding 000 to the values for E.

43. The "output viewer" will show you the .ANL file with a side bar on left for navigation. Now go to results and see what fails and check ratios. "find" "fail". For dimensioned lumber there is no automatic member selection, so you will need to look at what fails and the ratio for passing members and adjust member sizes up or down. The goal is to get the ratio under 1.0 but above 0.7 or 0.8. But at the same time you may want to limit the number of different sizes.

44. For the glulam look to the very bottom of the list and you should see the members it "selects". You have to manually reenter those sizes into your model and assign them to the appropriate glulams.

45. After changing the sizes you need to run the analysis (and select) again because the DL changes and that will change stress. When everything passes and not too overdesigned (check ratio or compare fbz to Fbz)you are finished.

46. Post-analysis can give you some informative graphics. Deflection and Mz are the most helpful. Right-click and pick STRUCTURE DIAGRAMS to see several possibilities. You will need to SCALE the diagrams so you can see them. Animation is also possible.

47. You can capture the images you need with the CAMERA (under Utilities) and build the report (last option under Postprocessing). Images can include: Deflection, Bending z, Shear y, Full 3D Sections (to show member sizes). See assignment for list or the posted example. Utilization Ratio plot is not working for 1984.

48. Under Postprocessing tab there is a graphic version of the report generator. It will bring up a menu bar on the right. With the report setup you can choose what to add to the report. DON'T GET CARRIED AWAY. Some of the output that lists values for each node or member can be very detailed and not really what you want for an overview. You can try these categories: Job Info, Material, Supports, Primary Load Cases, Combination Load Cases, Node Displacement **Summary**, Beam Displacement **Summary**, Beam Force Detail **Summary**, Utilization Ratio Table, and the pictures (use list from assignment). This should be about 10-15 pages max. You can export it to Word and then remove useless table columns (ones that are blank or all zero) to make them fit better on LTR paper. Edit the "job information" at the top to include your names and title.