Composite Design with SCIA Engineer
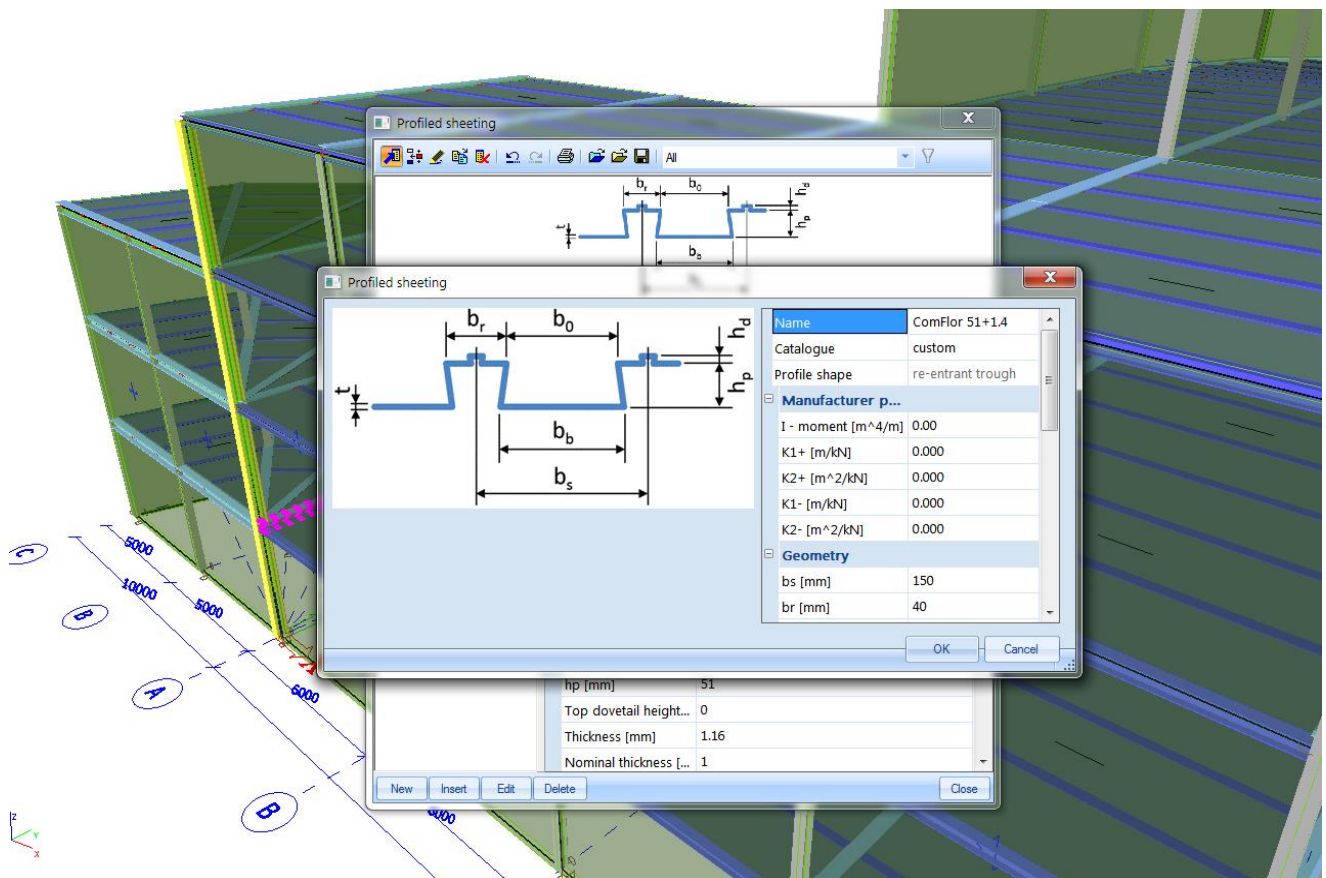
Process for calculation

1. New > Select New Project > Analysis > OK

2. Project data > Basic data > Material (Concrete and Steel for composite action)
    > Functionality > Dynamics (Seismic, General Dynamics)
        > Subsoil (Soil Interaction, Pad foundation check)
        > Stability
        > Climatic loads
        > Steel (Frame rigid connections, Frame pinned connections, Grid pinned connections, Bolted diagonal connections, Expert system, Connection monodrawings)
        > Structural model
    > Loads (Wind load, Snow load from EC1) > OK

3. Setup > Mesh

4. Setup > Solver

5. Setup > Concrete solver

6. Main > Structure > Modelling/Drawing > Import DWG, DXF, VRML97 or

    Main > Line grid and storeys > 3D Line grid or
                                File > Import > IFC 2x3

7. Cross-sections

8. Storeys

9. Main > Structure
    > 1D member
        - Select 1D members/ (Filter all)
        - Properties window Type of connections
            > With standard composite action
    > 2D member
        - Select the 2D members (slabs) and then go to properties window.
        - Element type > Composite deck
        - Element behavior > Flexible, Rigid or Semi-Rigid diaphragm
        - Profiled sheeting > Choose from the list or make one your own

10. Composite > Composite setup to choose shear connectors, etc

11. Composite > Load cases & stages (Automatically added)

12. Layers (For all sections and slabs per storey)

13. Main > Structure > Model data > Section on beam

Official Partner of SCIA in Cyprus

14. Connect Members/nodes

15. Main > Structure > Model data > Support

16. Main > Structure > Model data > [Hinge on beam](#) (i.e change fix or/and fiz to free)

17. Load > 3D wind generator > OK

            > Add load cases > Run generator (16 load cases)

18. Connect Members/nodes (1D members (beams) as ribs for composite action)

19. Check structure data

20. Main > Load Cases (Selfweight, Dead, Dead-Partitions, Dead-Roof, Live, 3D wind)

21. Main > Load Groups (Permanent, Dead-Partitions, Dead-Roof, Variable)

22. Main > Loads   > Dead > Surface load > on 2D member

           > Live > Surface load > on 2D member

           > Dead-Partitions > Line force > on 2D member edge

23. Mesh generation

24. Main > Dynamics > Mass groups (Selfweight, Dead, Live

25. Main > Dynamics > Combination of mass groups (Add All > OK)

> For ''Dead'' and ''Live'' loads below surface (also with ground floor) are NOT added to the ''Combination of mass groups'' because masses bellow ground surface are NOT calculated.

> For ''Live'' loads for seismic combination above ground (0+) > Used Coeff. = 0.3 (Auto)

> Code: $\Psi_{Ei} = \varphi \times \psi_{2i} = 0.3$ (Residential $\varphi = 1$, $\psi_2 = 0.3$) - Table A1.1 + A1.2(B) (CYS)

26. Libraries > Loads > Seismic spectrums (q-factor for steel)

           > Period

           > Eurocode

           > Code parameters > AgR*γi, q factor, Subsoil, Spectrum (Type 1), Direction.

27. Load > 3D wind generator > OK

            > Add load cases > Run generator (16 load cases)

28. Main > Load Groups (Together, Seismic)

29. Main > Load Cases (Seismic > Dynamic > Seismicity > CQC)

30. Main > Combinations (live load coeff=0.3 only for seismic combination - Auto)
      > ULS with wind, SLS, Seismic X (0.3Y), Seismic Y (0.3X)

31. Main > Stability combinations (Selfweight, Dead, Live)

32. Main > Result classes > TOTAL (ULS, Seismicity)
                                            > DYN (Seismicity)
                                            > GEO (ULS Set B, ULS Set C)
                                            > SERV (SLS-Char, SLS-Quasi)

33. Properties > Buckling data
                > Properties > Graphical input of system lengths

34. Analysis > Modal

35. Main > Results (Modal Participation factors Wxi, Wyi, Wzi_R > 0.90)

36. Main > Load Cases (Predominant mode > Mode shape per seismic direction from Results > Participation mass)

37. Analysis > Test of input data (Δημιουργεί τις σωστές μάζες ανά μέλος που έχει Line force)

38. Calculation > Batch analysis (Linear, Modal, Stability)
                          Solver setup > <u>Use IRS in case of 2 storeys or more</u>

39. Main > Results

40. View > New animation window > Play

41. Main > Steel > Steel slenderness
                      > Properties > Selection field > Standard
                                      > Values > Lam y (Λυγηρότητα στον y)
                                      > Extreme > No > Refresh
 Αλλάξτε μόνο τα Values για Ly και ly και πιέστε Refresh
                                        > Properties > Ly > Refresh
                                        > Properties > ly > Refresh

42. Main > Steel > Beams > Member check data > ULS check > Check (Preview for Results or Table results like excel)

43. Main > Steel > Connections (for IPE, HEA, SHS, RHS > Bolts, Welds, Stiffeners, Flanges,…)

44. Main > Steel > Connections
                > Select connection > Properties > Update stiffness (√)

45. Analysis > Batch analysis (Linear, Modal, Stability)

46. Main > Results

47. In case of high Unity, Section and Stability checks you may need to add <u>LTB restraints</u> and <u>Member buckling data</u> for steel members

Concrete foundation

48. Remove supports

49. Main > Structure > 2D member (Wall=Shell, Slab=Plate)

50. Main > Structure > Model data > Support > surface (el. foundation)

51. Main > Structure > Model data > Property Modifiers 1D (0.5)
    For 1D members below ground (0 <) change ''Mass factor'' to 0*.

52. Main > Structure > Model data > Property Modifiers 2D (0.5)
    For 2D members below ground (0<) change ''Mass factor'' to 0*.
    * ''Mass factor'' has to be zero (0) because foundations and basements have no movement
    because according to codes they are non-sway members.

53. Libraries > Load > Seismic spectrums (q-factor for concrete)

54. Connect Members/nodes

55. Check structure data

56. Analysis > Batch analysis (Linear, Modal, Stability)

57. Main > Results > Beams > Internal forces on beam
    > Properties > Rib/Integration strip (√)

58. Main > Results > 2D member

59. Main > Results > 2D member > Integration strip (Strip foundation)

60. Main > Concrete > 2D member > Member design > Member design ULS
    Properties > Use scale isolines

61. Main > Concrete > 2D member > Reinforcement 2D

62. Connect Members/nodes

63. Check structure data

64. Analysis > Test of input data

65. Analysis > Batch analysis (Linear, Modal, Stability)

66. Main > Results

67. Engineering Report for steel results

68. Engineering Report for concrete results

Για την γεωμετρία θα χρειαστεί να ενεργοποιήσετε:
- Snap mode 　 ή Snap mode

Official Partner of SCIA in Cyprus

- View > Set view parameters > Set view parameters for all

Ακόμη και αν εισάγατε αρχείο CAD στον κάνναβο σας προτιμότερο θα ήταν να δημιουργήσετε και κάνναβο ''3D line grid''.