

A finite element model of the full 96th floor of WTC 1 was translated from the SAP2000 reference models into ANSYS 8.1 for detailed structural evaluation (Figure 6–12)¹⁴. The two models generated comparable predictions of the behavior under dead or gravity loads.

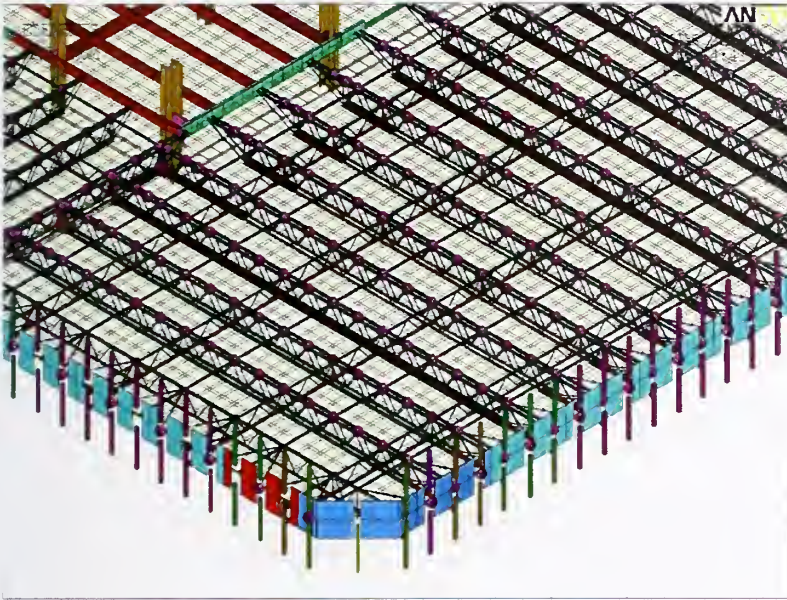


Figure 6–12. ANSYS model of 96th floor of WTC 1.

The model was used to evaluate structural response under dead and live loads and elevated temperatures, identify failure modes and associated temperatures and times to failure, and identify reductions in modeling complexity for global models and analyses. The structural response included thermal expansion of steel and concrete members, temperature-dependent properties of steel and concrete that affected material stiffness and strength, and bowing or buckling of structural members. The deformation and failure modes identified were floor sagging between truss supports, floor sagging resulting from failure of a seat at either end of the truss, and failure of the floor subsystem truss supports.

Exterior Wall Subsystem

The exterior walls played an important role in each tower's reaction to the aircraft impact and the ensuing fires. Photographic and video evidence showed inward bowing of large sections of the exterior walls of both towers just prior to the time of collapse.

A finite element model of a wall section was developed in ANSYS for evaluation of structural response under dead and live loads and elevated structural temperatures, determination of loads that would have caused buckling, and identification of reductions of modeling complexity for global models and analyses. The modeled unit consisted of seven full column/spandrel panels (described in Section 1.2.2) and portions of four other panels. The model was validated against the reference model developed by LERA (Section 6.6.2) by comparing the stiffness for a variety of loading conditions.

¹⁴ ANSYS allowed including the temperature-varying properties of the structural materials, a necessary feature not available in SAP 2000.