

10.3 SIMULATED FIRE-INDUCED THERMAL RESPONSE OF WTC 7

10.3.1 General

It was assumed that the structure was initially at 23 °C. The thermal analysis covered the approximately 21,600 s (6 h) period over which the FDS fires were present on Floors 7, 8, 9, 11, 12, and 13. The minimum time step for the thermal analysis was 1 ms, while the maximum time step was limited to 50 s. As was done for the towers, the output file for use in the structural analysis contained thermal data at 12 instants in time, spaced at 30 min (1,800 s) intervals, beginning at 12:30 p.m.

The modeling of the steel elements was similar to that in NIST NCSTAR 1-5G. However, in WTC 7, there was no evidence of damage to the thermal insulation (away from the immediate impact area of debris from WTC 1). Since heat transport across the insulation layer was much slower than heat transport into the depth of the steel, temperatures across column sections differed by less than 1 °C. Temperature gradients through the depth of the steel beams and girders were affected by the presence of the floor slab. Temperatures were uniform (within 1 °C) across the bottom flange and web, but the top flange temperature was less by up to several hundred degrees, because the slab acted as a heat sink (see Figure 4-6 to Figure 4-8).

As described in Chapter 2, the floor systems had a concrete slab on a corrugated steel deck. For Floor 8 and above, the overall concrete thickness was 140 mm (5.5 in.), with the corrugations being 75 mm (3 in.) deep. The FSI simulated these slabs as uniformly 100 mm (4 in.) thick. The thickness was subdivided into eight elements, each 12.7 mm (0.5 in.) thick. For the north side of the 7th floor, the overall concrete thickness was 200 mm (8 in.), with the corrugations being 75 mm (3 in.) deep. The south side had a formed 200 mm (8 in.) slab without a metal deck. The FSI simulated this slab as uniformly 200 mm (8 in.) thick. This thickness was also subdivided into eight elements, each 25 mm (1 in.) thick.

For each step, a set of thermal load data was generated to completely specify the thermal state of the lower 16 stories of the building. The thermal load data were in a format that was consistent with the structural models and could be readily read with the ANSYS finite element software to specify body loads (temperature and temperature gradient information) on the entire structure, for use in the simulations recounted in Chapter 11. ANSYS performed a linear interpolation between two consecutive thermal load steps. A second set of thermal load data was generated as input for the LS-DYNA simulations of the building collapse (Chapter 12).

The Investigation Team estimated the sensitivity of the thermostructural simulations to the temperatures developed from the fires. Accordingly, three different thermal response computations were performed that were consistent with a realistic and reasonable range of expected fires. Case A used the temperature data as obtained from the FDS simulation. Case B incremented the Case A gas temperatures by 10 percent. This increased the severity of the fire in three ways. It increased the local heat transfer to the structural members, it increased the area over which the structural members were heated, and it increased the chance of hot zone overlap on adjacent floors. A 10 percent increase or decrease in gas temperatures resulted in a roughly 30 percent increase or decrease in the heat flux to structural members. For example, a gas temperature of 1000 °C results in a heat flux to nearby objects of about 150 kW/m². A 10 percent increase in gas temperatures to 1100 °C leads to a heat flux of about 200 kW/m²; a 10 % decrease in temperature leads to a heat flux of just over 100 kW/m². This range of thermal exposure is fairly broad,