DS '11 KOBE
International Symposium on
Disaster Simulation & Structural Safety in the Next Generation (DS'11)
September 17-18, 2011, JAPAN

The Analysis Method in the structural design of Tokyo Sky Tree

KONISHI, Atsuo, WATANABE, Kazunari, NAKANISHI, Norio* and ESAKA, Yoshisato

Keywords: *Tokyo Sky Tree, Tubular Joint, Buckling Modal Analysis, Detail drawing*

SUMMARY

Construction of Tokyo Sky Tree, a digital broadcasting tower with a height of 634 m, broke ground in July 2008 in Sumida City, Tokyo. The tower will not only transmit digital terrestrial broadcasting beginning in the spring of 2012, but will also serve as a symbol of efforts to revitalize this area of Tokyo.

The area in and around Oshiage and Narihirabashi Stations, the site of the construction of Tokyo Sky Tree, is located in the center of Sumida City, Tokyo.

The tower configuration progressively changes from a triangular shape at the base to a circular form higher up giving the tower a unique configuration not found anywhere else in the world.

The changing configuration of the tower from a triangular to a circular shape produces “warping” in the tower design, forms that are found in traditional Japanese architecture and culture.

The main structural frame of Tokyo Sky Tree is a truss structure employing steel pipes. The steel pipe system was chosen due to steel’s high strength to weight properties and durability. The steel products have a yield strength ranging from 325 to 630 MPa . The size of the largest steel pipe is 2300mm in diameter and 100mm thick.

The characteristic elevation of Tokyo Sky Tree is a cylindrical cage shaped truss structure composed of small pipe members. These small pipes make efficient use of the changing tower configuration, and add ductility and redundancy to the discrete structure system.

In the structural analysis of the main structural frame, various types of analysis models were used for different areas of the design.

A complete global 3D frame analysis model was used for static and dynamic response to check the stress of each member.

A partial 3D frame analysis was utilized for an elastic buckling analysis. This analysis confirms that local buckling of each members occurs before buckling of the whole frame.

A finite element method (FEM) analysis model of tubular joints was used to check the strength of the tubular joints and their weld area.

During the structural detailing phase, careful attention to the steel detail drawings was also necessary to verify our analysis assumptions were consistent with the fabricated design.

An important issue when using complicated 3D analysis models is to check the structural response using simple models and calculations. When designing, one must grasp the expected response of the completed structure as well as interpreting the numerical analysis results.

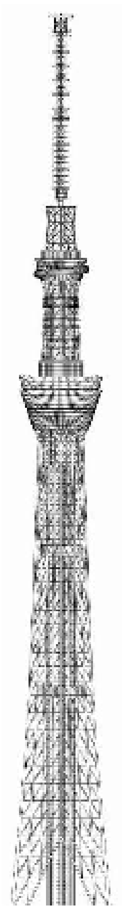
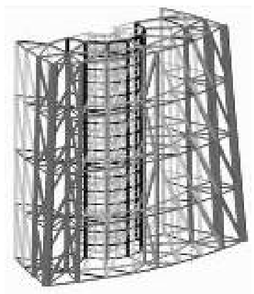
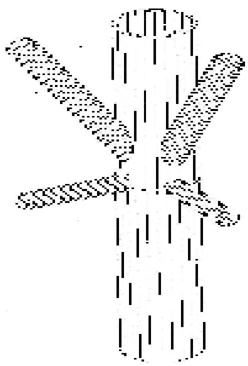
	MODEL IMAGE	NAME	ANALYSIS	DESIGN ISSUE
		ALL FRAME MEMBER 3D ANALYSIS MODEL	STATIC ANALYSIS DYNAMIC ANALYSIS ・WIND ・EARTHQUAKE	・MEMBER STRESS ・TUBULAR JOINTS ・PANEL STRENGTH and FATIGUE
		PARTIAL FRAME ELASTIC BUCKLING MODAL ANALYSIS MODEL	BUCKLING MODAL ANALYSIS	・ BUCKLING MODE
		STEEL PIPE TUBULAR JOINTS FEM MODEL	FEM ANALYSIS	・STRENGTH of TUBULAR JOINTS ・STRENGTH of WELD AREA

TABLE 1 ANALYSIS MODEL SUMMARY