Failure Analysis of World Trade Center 5

Acknowledgement

SIMULIA would like to thank Kevin J. LaMalva, Dr. Jonathan Barnett, and Donald O. Dusenberry of Simpson Gumpertz & Heger Inc. This work was performed by Mr. LaMalva while he was a student at Worcester Polytechnic Institute and advised by Dr. Barnett while he was a professor at Worcester Polytechnic Institute. Mr. Dusenberry co-advised the work. Funding was provided by the SFPE Educational and Scientific Foundation.

Summary

On September 11, 2001, flaming debris from the World Trade Center tower collapse ignited fires in World Trade Center 5. These fires burned unchecked and caused a localized collapse from the 8th floor to the 4th floor in the eastern section of the building.

A failure analysis of the internal structural collapse was conducted using Abaqus. Specifically, a sequentially-coupled, thermal-stress simulation was completed to study the structural performance during the fire and estimate the time to catastrophic failure. With respect to a critical structural detail, the deformed shape predicted with Abaqus was very similar to that of a specimen collected from the failed building. Based on the Abaqus results and forensic evidence, it was hypothesized that the steel column-tree assembly (Gerber beam design) failed during the heating phase of the fire.

Figure 1: Internal collapse area in WTC 5 [1]

Key Abaqus Features and Benefits

- Ability to include non-linear, temperature-dependent material properties
- Heat transfer and sequentially-coupled thermal stress analysis capability
- Bolt pretension modeling
- Contact interactions with static and kinetic friction modeling

Background

World Trade Center 5 (WTC 5) was a nine-story building located in New York City, NY. On September 11, 2001, due to uncontrolled fires, the eastern portion of WTC 5 experienced an internal progressive collapse. Structural impact was not a factor in this failure; the collapse was caused by fire alone. A portion of the collapsed area is shown in Figure 1.

The purpose of this failure analysis is to determine whether the collapse of the steel assembly occurred during the heating phase of the fire or when the building cooled down.

The design of WTC 5 employed steel column-tree assemblies, which is a common construction method. Forensic evidence suggests that the collapse occurred during the heating phase of the fire, which is atypical. In general, this scenario represents a higher risk to firefighters and occupants, as it would occur at an earlier time in the fire event.

Accurate, detailed finite element modeling can be used to assess the fire endurance of steel structures. Such simulations can be used to develop pre-engineered solutions to building codes. Additionally, knowledge of the fire endurance behavior of WTC 5 can further the understanding of firefighting risk in existing buildings of similar design.

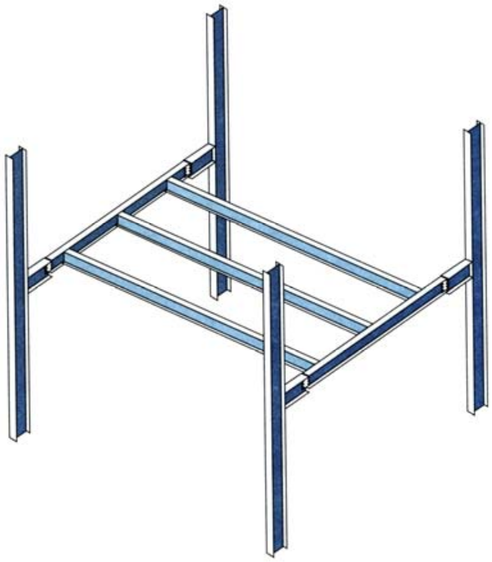

Figure 2: Typical interior bay framing in WTC 5 (Floors 4, 5, 6, 7, and 8) [1]

In this Technology Brief, the nonlinear heat transfer and stress analysis capabilities of Abaqus/Standard are used to study the mechanisms that caused WTC 5 to collapse.

Finite Element Analysis Approach

A sequentially coupled thermal stress-analysis consists of two separate analyses; a heat transfer simulation to determine temperature history, followed by a stress analysis that incorporates the temperature history as part of the loading.

The same WTC 5 structural mesh (with appropriate element types) was used for both the heat transfer and stress analyses. Four structural bays on the 8th floor of WTC 5 were modeled; a typical bay is shown in Figure 2, with mesh detail shown in Figure 3.

Heat Transfer Analysis

For the heat transfer analysis, common spray-applied mineral fiber fireproofing insulation is explicitly included in the thermal model. Thermal contact between the insulation and steel was modeled. Temperature-dependent specific heat and conductivity properties of A36 steel were derived from literature.

By referencing the 2005 National Institute of Standards and Technology (NIST) report on WTC 1 and 2, the effective heat of combustion and the peak heat release rate of the fire that occurred within WTC 5 were estimated.

Using the information from the NIST reports, heat release rate versus time curves were derived. Using these curves

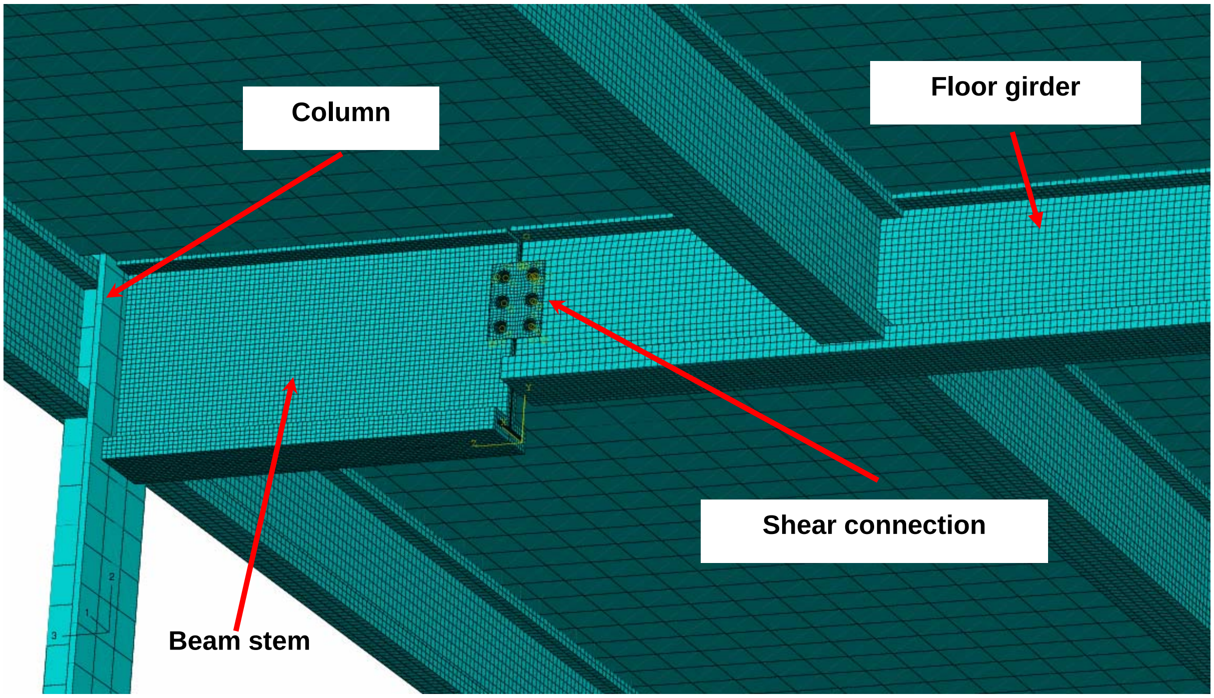

Figure 3: WTC 5 Structural assembly with spray-applied insulation and concrete slab included

as input, the Consolidated Fire and Smoke Transport Model (CFAST) from NIST was used to determine temperature history scenarios of the reconstructed fire.

The Abaqus/Standard thermal loading was applied as a convective film condition on the faces of the assembly that were exposed to the fire environment. The heat transfer coefficient selected is characteristic of turbulent, natural convective heating. Moreover, the results of the CFAST simulation were used to prescribe the sink temperature history.

Stress Analysis

The detailed three-dimensional temperature history of the steel assembly is applied as part of the loading in the structural analysis. This is integral for understanding the thermal-stress behavior, since the steel strength is highly temperature-dependent.

The structure is also loaded by gravity, and 39 kips of pretension is applied to each shear connection bolt. Static and kinetic friction between the faying surfaces was modeled as well.

The stress analysis model utilized nonlinear, temperature-dependent stiffness and thermal expansion properties of A36 steel based on experimental data from Harmathy [2]. Geometric nonlinearity was accounted for in the analysis.

Simulation Results

The temperature distribution near a shear connection after one hour of fire exposure is shown in Figure 4. In general, the average temperature of the steel was found to agree well with estimates from hand calculations.

It was observed that the insulation functions as predicted: it delays the transmission of heat to the steel during the fire exposure. Moreover, it can be observed that the top of the assembly remains relatively cool due to the heat sink effect of the overhead concrete slab.

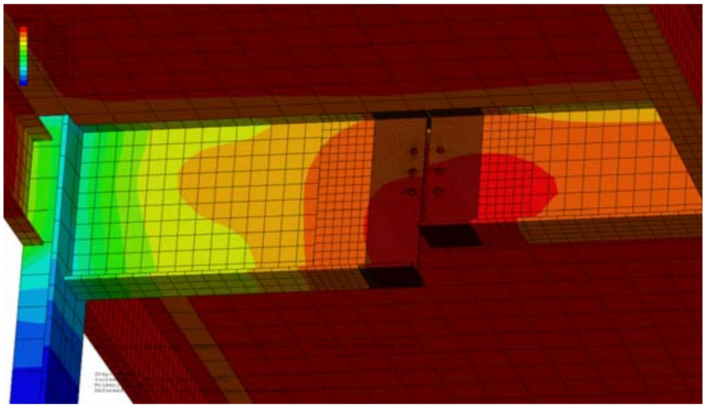

Figure 4: Temperature distribution near a shear connection after one hour of fire exposure

The temperature difference between the end of the beam stem welded to the column and its other end at the shear connection is approximately 450 ºC after one hour of fire exposure. Additionally, the interface between the beam stem and column remains relatively cool due to the heat sink effect of the structure beyond the column. Since heat at the shear connection must be conducted across the length of the beam stem before it reaches the column, heat accumulates quickly in the vicinity of the bolt holes. This effect is exacerbated by the fact that the beam stem itself is being heated, which reduces the thermal flow from the connection.

The structural assembly is initially at ambient temperature (20 °C) and carrying the specified gravity loads. As the steel in the compartment heats up in the first hour of fire exposure, it undergoes thermal expansion which causes the floor girder to elongate significantly and close the gap between it and the beam stem. This elongation causes relatively harmless compressive stress concentrations as the bolts push into the beam stem web.

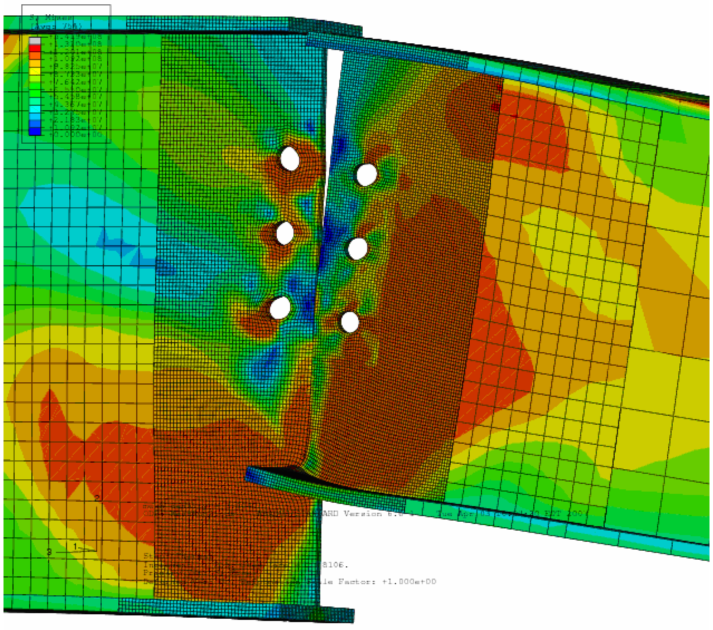

Figure 5: Mises stress distribution near a shear connection after two hours of fire exposure

As the temperature of the steel assembly increases, its rigidity decreases steadily and the floor girder begins to undergo significant deflection. This deflection causes the lower flange of the floor girder to contact and deform the beam stem web, which forms a fulcrum at the contact point. After 2 hours of fire exposure, the loss of rigidity in the steel “outpaces” its thermal expansion and the top bolt of the shear connection undergoes a stress reversal and begins to pull toward the end of the beam stem web, as shown in Figure 5.

The stress reversal described above can be attributed to the fulcrum action, which has fully formed at this stage of the fire. The fulcrum mechanism forms completely be-

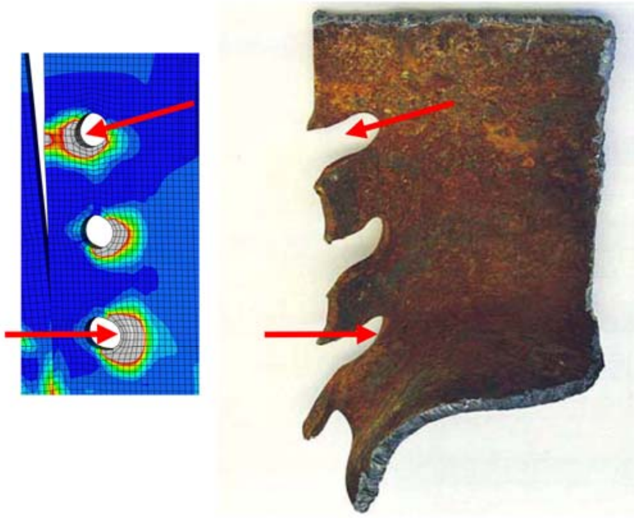

Figure 6: Comparison of Abaqus model results and forensic evidence

cause tensile stresses surrounding the lower flange's penetration into the beam stem web arrest further deformation in this region.

It was observed that the deformation in the Abaqus model is very similar to that of a collected structural specimen from WTC 5 that underwent failure. More precisely, the angles at which the bolts pried against the bolt holes are similar in the model and the specimen, as shown in Figure 6. Moreover, photographs of failed beam stems show evidence of fulcrum point deformations.

Discussion of Results

The shear connection failures in WTC 5 were due to shear rupture of the web portion of the beam stems. This type of failure occurs when the bolts bear against the weak side (i.e., acting toward the free end of the member) of the bolt holes. Bearing stress of this type causes shear planes to form in the web steel. Finally, cracks along the shear planes cause catastrophic failure of the shear connection.

An indirect technique was developed to predict the time at which connection failure occurred. The failure criterion model for a single bolt and hole was based the AISC Steel Manual (LRFD) [3]. A failure load was derived and applied to the failure criterion model. The plastic shear strain at failure was then used as the failure criterion for shear rupture at a bolt hole. According to the failure criterion established, the shear connection experienced complete failure after approximately two hours of fire exposure.

Conclusion

The sequentially-coupled, nonlinear thermal stress modeling capabilities in Abaqus/Standard have been used to study the quasi-dynamic structural behaviors that led to the progressive collapse of WTC 5. It is hypothesized that the structure collapsed during the heating phase of the fire as a result of the formation of fulcrum point mechanisms.

References

1. World Trade Center Building Performance Study: Data Collection, Preliminary Observations, and Recommendations, Federal Emergency Management Agency (FEMA), New York, 2002
2. Harmathy, T.Z. Fire Safety Design and Concrete, Longman Scientific and Technical, United Kingdom, 1993
3. AISC Manual of Steel Construction: Load and Resistance Factor Design, 3rd ed., American Institute of Steel Construction (AISC), 2001

Abaqus References

For additional information on the Abaqus capabilities referred to in this brief, please see the following Abaqus Version 6.7 documentation references:

- 'Heat transfer analysis procedures,' Section 6.5.1 of the Abaqus Analysis User's Manual
- 'Sequentially coupled thermal-stress analysis,' Section 6.5.3 of the Abaqus Analysis User's Manual

---

About SIMULIA

SIMULIA is the Dassault Systèmes brand that delivers a scalable portfolio of Realistic Simulation solutions including the Abaqus product suite for Unified Finite Element Analysis, multiphysics solutions for insight into challenging engineering problems, and lifecycle management solutions for managing simulation data, processes, and intellectual property. By building on established technology, respected quality, and superior customer service, SIMULIA makes realistic simulation an integral business practice that improves product performance, reduces physical prototypes, and drives innovation. Headquartered in Providence, RI, USA, with R&D centers in Providence and in Suresnes, France, SIMULIA provides sales, services, and support through a global network of over 30 regional offices and distributors. For more information, visit www.simulia.com