Embracing Complexity

BY KERMIN CHOK

Judicious use of a suite of design and analysis tools on a stadium project gave a consulting engineering firm a whole new way of looking at engineering workflows.

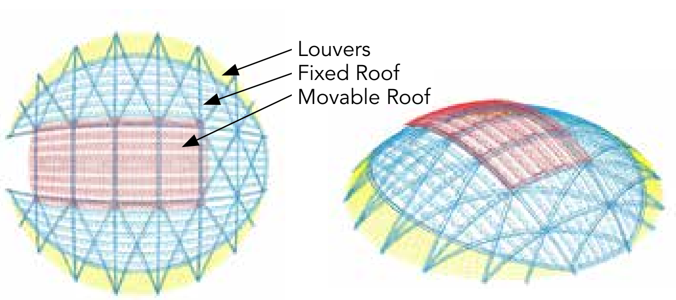
ELEGANT ENGINEERING IS often the byproduct of conscious and disciplined efforts to reduce complexity. However, in certain situations, it is necessary to fully embrace it—as well as the opportunities it provides for new ways of thinking.

For example, Meinhardt Group recently completed a finite element analysis (FEA) of the Singapore Sports Hub Stadium Roof, designed by Arup and scheduled to open in 2014. (The analysis was actually done as a peer reivew; according to local building regulations, all projects in Singapore are required to undergo a second-party peer review certifying that the major structural elements have been adequately designed and detailed.)

The review involved verifying that all members have adequate capacity and the welded hollow structural section (HSS) connections (welds, joint cans, end thickening, etc.) have been appropriately designed and detailed. The main stadium roof has a diameter of approximately 300 m (984 ft) and rises to over 70 m (230 ft) in elevation. The covered surface area of the stadium is approximately 71,000 sq. m (764,238 sq. ft) and will have a seating capacity of 55,000. The roof is exclusively comprised of round HSS (main chord members are approximately 18 in. in diameter and the secondary members are 11 in. in diameter; total steel tonnage for the roof is 9,000). At the node points, thickened joint cans are introduced to receive the incoming brace members. The three components of the stadium are shown in Table 1 and illustrated in the figure. The scale of the project can be demonstrated by the size of the structural analysis work, as illustrated in in Table 2.

Roof Component	Number of Elements	Number of Connection Nodes
Fixed (Blue)	13,102	4,490
Movable (Red)	5,394	2,178
Louvers (Yellow)	6,858	3,960
Total	25,354	10,628

Table 1: Stadium Roof Components



A model of the roof (colors correspond to Table 1).

Table 2: Structural Analysis Dataset

Total number of beams	25,354
Number of load cases	26
Number of load combinations	96
Number of models (open/closed position, fixed/flexible support condition)	4
Number of output stations (start, end)	2
Number of result types (axial, moment, shear, torque)	6
Number of sets of beam results	25,354×(96+26)×4×2= 24.7 million
Total number of results items	24.7×6 = 148.2 million
Maximum number of MS Excel rows	1,048,576
Maximum size of MS Access database	2 GB

The primary codes of practice used on the project are listed in Table 3. Depending on the type of check, enveloped forces or load case forces were used for evaluation. Using an enveloped set of forces condensed the result set of 24 million into a manageable set of 25,000.

Images: Courtesy Meinhardt Group

Application	Design Code/Procedure	Comments	Forces Required
Member Checks	BS 5950: Structural use of steelwork in building	Primary code for verification of strength verification of tubular members	Envelope Min/Max
Connection Checks	CIDECT Design Guide 1: Circular Hollow Section (CHS) Joints under predominantly static loading	Primary code of reference for standard Joint configurations	Envelope Min/Max
Connection Checks	AWS D1.1: Structural Welding Code-Steel	Secondary code of reference for verification of joint can capacity	Load case
Connection Checks	Finite Element Analysis	Procedure for evaluation of stress with multiple overlapped members	Load case

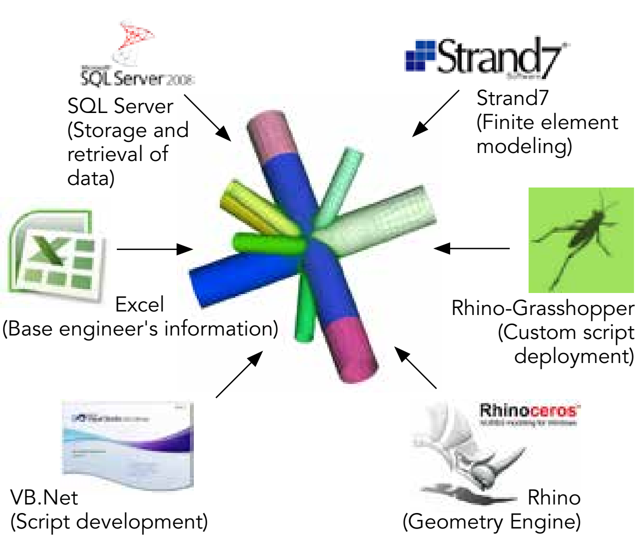
Table 3: Design Codes

Storage and Checks

Traditional data storage means (MS Excel, MS Access) were grossly inadequate for the analysis model dataset. Besides simply storing the data, the team needed the ability to quickly query the dataset to obtain specific load case forces or enveloped forces for a particular set of members. More importantly, any process automation depended on the ability to reliably query the dataset in a structured manner. SQL Server 2008 was chosen since our office already had a license of this software used in accounting and other back end office roles.

CIDECT-based checks generally approach connections based on their classification, depending on the incoming brace geometry and force distributions. A subroutine was written in VB.net, leveraging the Rhino geometry engine, to calculate relative angles, overlaps between incoming beams and any other geometrical information required for code based checks; envelope forces were used in this case.

When using AWS D1.1 to compute joint capacities, the joint needs to be evaluated on a load case by load case basis. Such an approach required more than 1,000,000 unique code based evaluations. A complete AWS code check script was thus written in VB.net, again using the Rhino geometry engine for geometrical evaluations and SQL Server for brace force extraction.



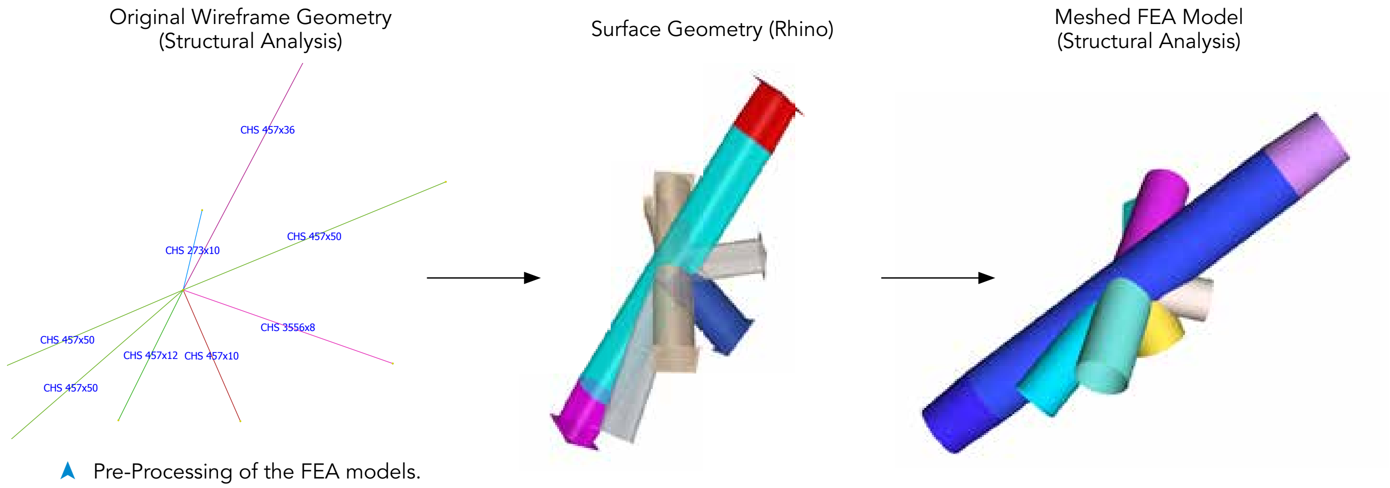
The various software packages used for the review.

The finite element approach was used for joints where complicated overlaps between incoming beams occurred or additional verification was deemed necessary due to the criticality of the joint or the magnitude of forces being transferred. The FEA of a joint requires significantly more effort since the creation of the surface geometry, transfer of geometrical information to an FEA package and post-processing are traditionally very time-consuming and manual tasks. As much of the FEA process as possible was automated. The only thing that was not automated was the verification of stress levels in the particular model after it was run.

FEA Model Pre-Processing

Pre-processing was heavily automated and primarily required the creation of a surface model from the base wireframe information, which also required outer diameter and thickness information of the incoming members. The sequence of primary and secondary members was read from the base engineer's information (referred to as the "cutting sequence"). The cutting sequence determined the continuous brace and the other braces that get profile cut against the primary member. The surface model was transferred to the FEA package (Strand7) via a custom VB.net script hosted in Rhino-Grasshopper. All analysis options and property information (such as plate thickness) were automatically set, and the only manual intervention was confirmation of the finite element mesh quality. FEA models typically consisted of 20,000 to 60,000 Quad8 finite elements.

Kermin Chok (kc@meinhardt.com.sg) is a technical director in the Singapore office of Meinhardt Group.



Pre-Processing of the FEA models.

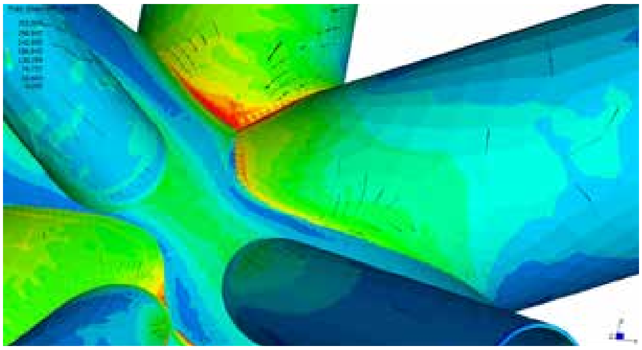
Load application of the 520 coincident load cases to the model was also automated. The automation of load application not only made the actual analysis possible but, once qualified and checked, also eliminated human error. While the model was loaded with all 520 load cases, a select few cases (approximately 30 to 50) were run due to time and practical considerations of result file size. The load cases were selected based on minimum/maximum forces and previously estimated AWS usage.

FEA Model Post-Processing

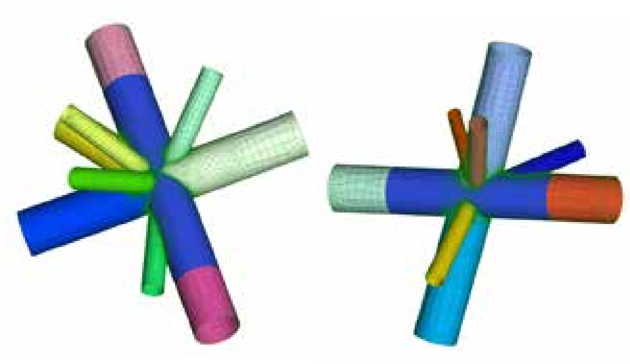
Documentation of the FEA models was required after the manual verification of the stress contours. In traditional FEA post-processing, an engineer typically takes screen shots of various angles of the model and different performance measures such as stress and deformation. If only a small number of models were involved in our analysis, a manual process would have been acceptable. However, given the large number of FEA models (more than 200), a systematic process was needed to capture the stress contours and the model definition. A standalone program, which interacted with the Strand7 FEA model through the application programming interface (API), was developed. This program opened the model, captured images of the individual beams from different angles and stored the images in a JPEG format, with standard file prefixes for easy identification. This program allowed multiple models to be queued up at the end of the workday for post-processing overnight. Four viewing angles of a particular brace were captured, which resulted in anywhere from 60 to 120 images per FEA model being created.

New Approaches

Thanks to this project, we've been able to develop new workflows based on other disciplines; our team tapped database approaches (SQL Server) typically used in the back end of many commercial applications such as accounting and web services. We also adopted 3D surface modeling software typically used in architectural and marine design applications (Rhino). However, creativity and flexibility were the most important tools in our arsenal, and the human mind was the indispensable judge to determine which tasks could be and should be automated. Regular engineering tasks will continue to be more efficient as technologies develop. However, the appropriate application of the relevant technologies (human or computer) will ultimately be the differentiating factor in the pursuit of engineering excellence. MSC



FEA post-processing, indicating the stress contours of an HSS node.



Sample of FEA connections and actual HSS.