Proceedings of the International Association for Shell and Spatial Structures (IASS) Symposium 2010, Shanghai
Spatial Structures – Permanent and Temporary
November 8-12 2010, Shanghai, China

The Freeform Structure of the UAE Pavilion at the Shanghai EXPO 2010

Soeren STEPHAN[1*2*], Feng PAN [2*], Yongqiang HUANG[3*]

[1*] Member of IASS, ASCE, AISC, VDI

[2*] Novum Structures LLC
W126 W8585 Westbrook Crossing
Menomonee Falls, WI 53051, USA
soeren.stephan@novumstructures.com

[3*] ECADI East China Architectural Design and Research Institute Co. Ltd
151 Hankou Road, Shanghai, China

Keywords: UAE Pavilion, Freeform Structure, Semi-rigid Connection, Rotational Stiffness

1 Introduction

Organic building shapes with surfaces arbitrarily curved in two directions, also known as freeform structures, have gained enormous popularity in the last decade due to the emerging availability of Computer Aided Design software utilizing NURBS surfaces. The majority of freeform building envelopes are featuring either double glazed units or insulated metal panels and are therefore requiring the transition from smooth geometric shapes to facetted shapes. The development of the facetted shape is a crucial step in the design process as it is strongly influencing the visual appearance of the building as well as the feasibility of the nodal connections of the freeform structure and the facet joints. Therefore the design of freeform structures is requiring a very close cooperation of the architect, the engineer and the contractor in order to address these integration issues in the execution process as early as possible.

Nodal connections of freeform structures can be either bolted or welded. Welded node connectors are easier to design due to the somewhat higher rigidity of the structure. However the fabrication and installation of such welded structures is usually imprecise, slow and labor intensive. Bolted node connectors are more challenging during the design process due to the semi-rigid connection behavior with the related implications on the global stability of the structure. The fabrication and installation process of bolted structures is very fast, precise and efficient though. Therefore a proprietary bolted node connector was used for the freeform portions of the pavilion structure.

The UAE pavilion for the Shanghai EXPO 2010 shown in Fig. 1 & 2 is a sustainable temporary exhibit building designed by Foster + Partners, London. The pavilion will be disassembled after the exhibition and reassembled on Saadiyat Island in the UAE.

The initial structural engineering of the pavilion was done by Halvorson & Partners, Chicago. The local architect and engineer of record was ECADI - East China Architectural Design and Research Institute Co. Ltd. The general contractor wholly responsible for this pavilion was CSCEC China Construction Industrial Equipment Installation Co. Ltd Shanghai.

Fig. 1: UAE Pavilion, Dune Shapes

Fig. 2: UAE Pavilion, Louver Wall

The general contractor appointed Novum Structures China for the final design of the freeform structure, the complete fabrication and the supervision of the installation. The engineering teams of Novum Structures GmbH & LLC engineered the structure in close cooperation with ECADI and CSCES in China. The responsible project management team of Novum Structures China executed the project within only 4 months from contract award to installation finish.

2 Pavilion Structure

The pavilion structure shown in Fig. 3 has an almost circular footprint with a diameter of about 65 m and is featuring three sand dunes, thus replicating natural formations typical for desert biomes in the UAE. Two smaller dunes rising to a height of about 18 m occupy the northern half of the building. The other dune in the southern half has a height of 20 m. The dune shapes replicate the distinct features of natural sand dunes formed by steady desert winds: solemn curved windward sides, sharp ridges and almost flat leeward sides.

The windward dune sides are framed as single layer freeform grid shell structures, the leeward dune sides are made as louver structures with vertically inclined straight I-beams braced by smaller horizontal members. The northern and southern dunes are subdivided by a central spine structure. Crescent shaped cantilevered canopy structures are covering the building entrances at the northern and southern building edges.

The lateral and vertical stiffness of the spine and the canopy structures is crucial for the structural efficiency of the freeform grid shells as it significantly depends on the stiffness of the grid shell edges.

Therefore the spine structure is made of stiff portal frames at 6 m intervals which are connected by two arch shaped I-beams. These arches are densely braced among themselves and their end points are fixed to the foundation.

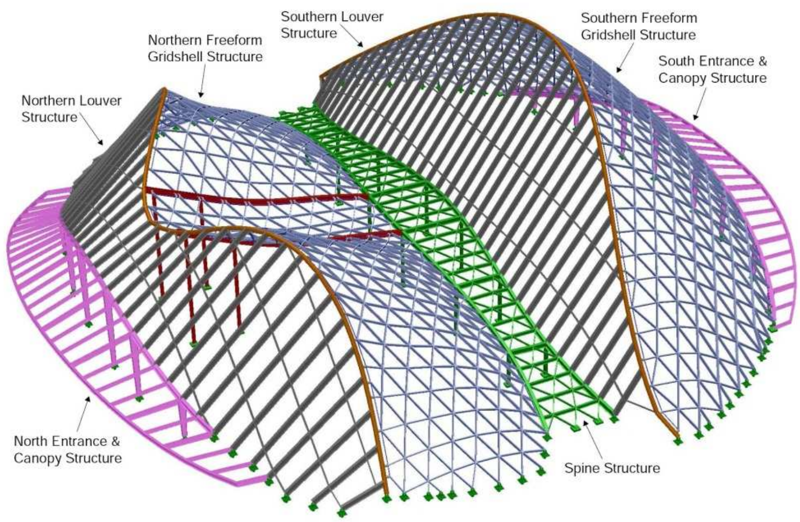

Fig. 3: Pavilion Structure

Both crescent shaped canopies are also featuring two arch shaped I-beams at their edges, one of which is vertically supported by entrance wall columns. The two canopy arch beams are interconnected by straight beams, which are moment connected to the entrance wall columns where possible. The end points of the canopy arches are fixed to the foundation as well, thus creating rigid tension ring belt supports for the dune structures above. The dune ridges are made of curved, large diameter pipes.

The northern freeform grid shell is additionally supported by two spatially curved I-beams at the inflection zone of the double dune shape. Each of those support beams is spanning from the spine edge to two vertical columns underneath the northern ridge.

The portal posts and arch beams of the spine structure, the entrance wall columns and the arch ring belt beams of the canopy structures and the eastern and western edges of the freeform grid shell structures are supported by reinforced concrete stripe foundations.

The initial structural design of the freeform grid shell structure by Halvorson & Partners called for square tube members 200 x 200 x 6 mm. Since this would have lead to large node connectors, the cross section of the freeform members were changed to rectangular tubes 240 x 80 x 8 mm of steel grade Q345. The proprietary freeform node connector shown in Fig. 4 typically consists of two forged steel discs of grade #45 with machined faces which are precisely inclined toward each member end. All those machined faces feature one threaded hole. Special precision cast steel adapters of grade GS-20Mn5V are welded to each member end. Support nodes are made of weldable steel grade Q345.

The members are bolted to the machined faces of the top and bottom node discs using grade 10.9 socket head bolts M24 or M27 inserted into the adapters via small rectangular openings in the top chord of tubes and screwed into the threaded holes of the node faces. All bolts are pretensioned with special proprietary tools. A mockup of the grid shell structure with the roof cladding can be seen in Fig. 5.

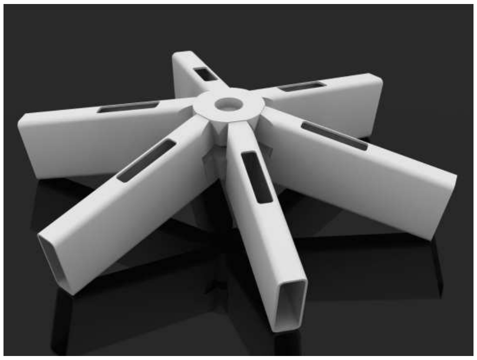

Fig. 4: Freeform Node Connector

Fig. 5: Freeform Roof Built-Up Mockup

3 Grid Shell Geometry

There are two basic methods to develop a grid on a smooth freeform surface: the planar grid projection method and the surface partitioning method. The planar grid projection method is simply projecting a planar grid onto the smooth surface. Therefore this method is only suitable for surfaces with small curvatures. Since the footprint of the freeform surface is ignored during the projection, there are usually many problematic nodes along the perimeter of the freeform grid. The problems at these nodes occur due to too short member lengths for practical realisation or due to too tight angles between grid and perimeter members. Fig. 6 shows the grid of the south freeform structure as initially developed by the architect. The problematic nodes are marked with clouds.

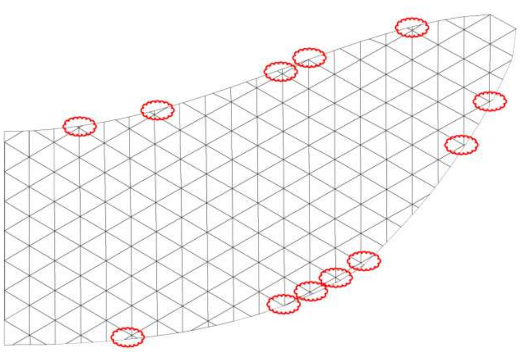

Fig. 6: Planar Projection Grid

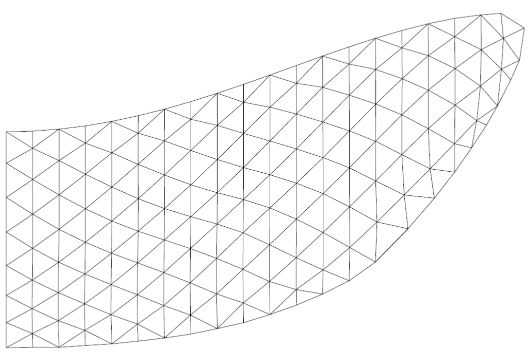

Fig. 7: Surface Partitioning Grid

The surface partitioning method is developing the grid from the perimeter. All perimeter sections get subdivided into portions of similar length so that those subdivisions match an "orthogonal" auxiliary grid directly on the freeform surface. One direction of this auxiliary surface grid has to be selected to be directly used for the final grid. The grid elements generated this way serve as base edges of the final grid triangles. All these triangle base edges get subdivided by a constant ratio. The other two directions of the final triangular grid are generated by connecting all these subdivision points with the two end points of the corresponding adjacent triangle base edge. The use of this grid generation method minimizes the amount of resulting problematic perimeter nodes.

Fig. 7 displays the grid of the southern freeform structure developed with the Surface Partitioning method.

Unfortunately the very tight realisation schedule of the pavilion did not allow for a re-development of the initial planar projection grid by using the surface partitioning method. Instead all the problematic nodes of the initial grid were manually corrected.

4 Freeform Grid Shell Node Connector

Once the wireframe geometry of the pavilion structure representing member centerlines was established, refined and coordinated, a comprehensive structural model including all loads and load combinations required by the local building code had to be generated. In order to adequately consider the semi-rigid connection behavior of the freeform grid shell node connections in this structural model, a series of bending stiffness and connection capacity tests had to be done beforehand.

4.1 Node Connection Bending Stiffness and Capacity Tests

These tests were performed using a 4-point bending test scheme as shown in Fig. 8. The specimen was supported on the test machine base on a sliding and a fixed support. A spreader beam on top of the specimen distributed the downward force of the test ram.

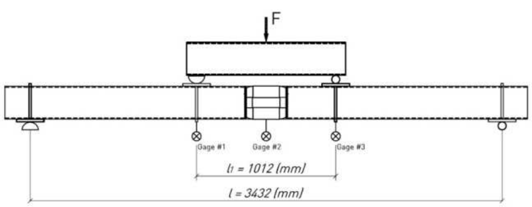

Fig. 8: 4-Point Bending Test Scheme

Fig. 9: Test Setup in Laboratory

Three displacement gages were mounted underneath the load introduction points and at the mid span of the specimen as shown in Fig. 9. By this arrangement the shear force was eliminated from the node connection and only determined bending moments were introduced. The observed failure loads and displacements are shown in Fig. 10 & 11.

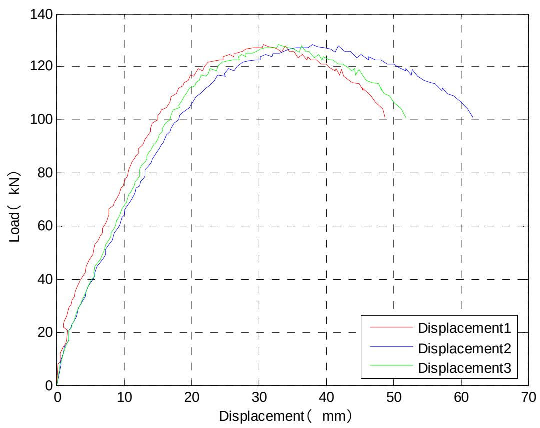

Fig.10: Ram Load - Displacement Curves

Test #	**Failure Mode**	**Ram Load, at Failure**	**Bending Moment, at Failure**
1	Bolt	127 kN	76.8 kNm
2	Bolt	128 kN	77.4 kNm
3	Bolt	129 kN	78.0 kNm

Fig. 11: Bending Test Results

4.2 Numerical Calculation of Node Connection Bending Capacity

The bending moment capacity of the node connection is apparently dependent on the axial force transfered by this connection. A direct analytical structural calculation of the node connection capacity is impossible due to the non-linear character of this bolted connection. Therefore an iterative numerical analysis as described in [1] was performed for 3 different limit state conditions - elastic limit, plastic limit and failure. The results of the failure load calculation can be directly compared with the bending test results.

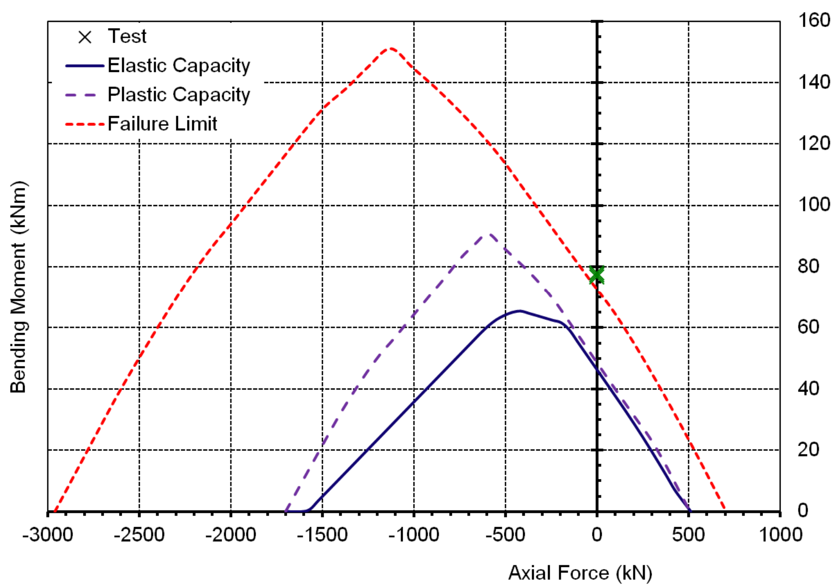

Fig. 12: Calculated Bending Moment Capacity vs. Axial Force

The numerically calculated bending moment capacity of the node connection for varying axial forces is shown in Fig. 12. The bending moment capacities observed in the tests are apparently conservatively close to the estimated failure moments.

4.3 Analytical Calculation of Node Connection Stiffness

The measured load - displacement curves are the basis for determining the rotational stiffness of the node connections, which is needed for the accurate structural analysis of the freeform grid shell.

The measured mid span displacement is apparently caused by a linear combination of the flexural deformation of the tube members and a rotational deformation of the node connection. Thus the mid span deflection Δ_{Rot} caused by the rotational deformation of the node connection has to be determined by:

$$\Delta_{Rot} = \Delta_{Total} - \Delta_{Flex} = \frac{M \cdot l}{2 \cdot K_{Rot}} \tag{1}$$

Δ_{Total} - total measured mid span deflection, m
Δ_{Flex} - flexural mid span deformation of the tube members, m
M - bending moment at node connection, Nm
l - span, distance from support to support
K_{Rot} - rotational node connection stiffness of (Nm/rad)

The flexural mid span deformation of the tube members is calculated via:

$$\Delta_{Flex} = \frac{M}{24EI} \cdot \left(3l^2 - 4a^2\right) \tag{2}$$

E - modulus of elasticity of steel, N/m^2
I - moment of inertia of tube section, m^4
a - distance of load introduction point from support, m

By combining and simplifying formula (1) and (2), the rotational node connection stiffness can be derived as:

$$K_{Rot} = \frac{M \cdot l}{2 \cdot \left[\Delta_{Total} - \frac{M}{24EI} \cdot \left(3l^2 - 4a^2\right)\right]} \tag{3}$$

The average measured mid span deflection was 18.5 mm for a test ram load of 100 kN, which caused a node connection bending moment of 60.5 kNm. Using formula (3), the corresponding rotational node connection stiffness can be calculated as 16110 kNm/rad.

4.4 Finite Element Analysis of Node Connection

As an additional verification a non-linear Finite Element analysis of the test specimen was performed, using appropriate contact elements for the bolted connection.

The Finite Element analysis for a test ram load of 100 kN resulted in a total mid span deflection of 14.8 mm (0.582 in) as shown in Fig.13. This deflection is only 80% of the measured average mid span deflection of 18.5 mm.

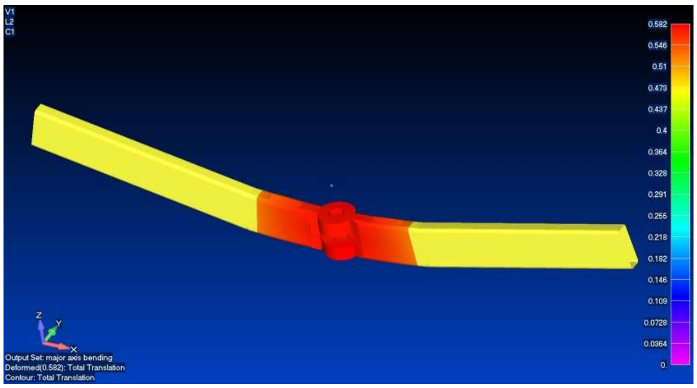

Fig. 13: Finite Element Model Displacements for 100 kN Test Ram Load

The reasons for this underestimated displacement are probably imperfections of the real test specimen and the real node connections which cannot be considered in a Finite Element model, like small initial gaps of the node connections or little initial settlements due to uneven or rough contact surfaces etc.

This comparison of measured and directly calculated deflections was then utilized to calibrate any other Finite Element model of the node connection used for the structural calculation of the pavilion.

5 Structural Analysis

The original structural analysis of the pavilion done by Halvorson & Partners had to be adopted considering local code requirements by ECADI - East China Architectural Design and Research Institute Co. Ltd.

A major update of the structural design calculations became necessary in order to take into account the modified cross section RHS 240 x 80 x 8 mm of the freeform members instead of the previously assumed SHS 200 x 200 x 6 mm as well as the semi-rigid node connection characteristics instead of the previously assumed fully rigid node connection characteristics, which could have been achieved only with welded nodes.

Due to the complex behaviour of the pavilion structure with significant dependencies between all portions, a meaningful structural model of the freeform portions would need to include all other parts of the structure as well. Therefore ECADI and Novum agreed to closely coordinate their structural models and perform the structural analysis simultaneously - ECADI using the Finite Element software SAP2000 and Novum using the FE software Dlubal RStab for it.

The experimentally determined rotational node connection stiffness was applied to each freeform member end in the structural model. Load combinations for limit state design using factored loads as required by the Chinese building code were generated. Using appropriate Finite Element analysis software, internal forces and moments, support reactions as well as deflections for all those load combinations were calculated using second order theory (static equilibrium at the deformed structure with small deflections).

The internal forces and moments at all node connections were compared to the calculated & tested connection capacity (as described before under 4). Also the conventional stability analysis and the stress analysis of individual structural members as per the Chinese steel design code were performed. The analysis results did confirm that the selected profiles and connection components had sufficient capacity. However, due to the single layer configuration of the freeform grid shell structure, special attention had to be paid to global stability issues.

5.1 Global Stability Analysis according to Chinese Building Code

ECADI established in their initial structural analysis that the governing load case for the global stability of the south freeform grid shell is wind load in 45 degree direction. The corresponding governing load combination 1.0*DL+1.0*WL45+0.7*LL$_R$+0.7*TL(+), which is a service load combination as required by code. For illustration refer to Fig. 14.

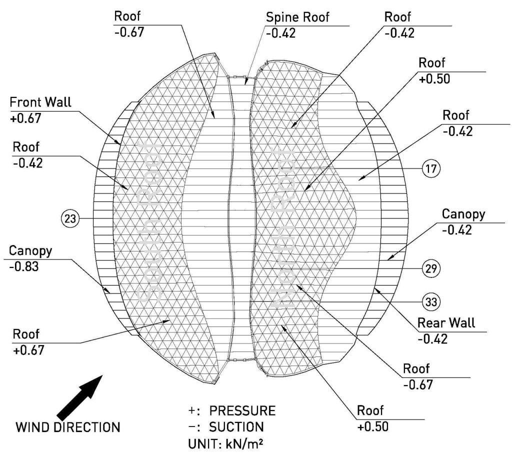

Fig. 14: Governing Wind Load Case for Global Stability

The Chinese Building Code Spec JGJ 61-2003 "Technical Specification for Lattice Shells" as well as the project specific “UAE Pavilion Structural Design Specification” require a critical load factor of at least 4.2 against the lowest elastic buckling load using the governing load combination.

The elastic buckling analysis computes the critical load factor for a structure with imperfections subjected to a particular set of applied loads. This critical load factor is the ratio by which the axial forces in the structural members must be increased to cause the structure to become unstable due to the flexural buckling of one or more members.

The elastic critical load factor of the structure is a function of the structural geometry, the elastic properties of the structure and the loading pattern. Lateral torsional buckling of individual members is not considered here. For the freeform grid shell structure this is justified since only RHS tubes are used, which are not sensitive to lateral torsional buckling.

As per JGJ 61-2003 a global imperfection of L/300 has to be considered for the computation of the critical load factor of the structure. Hereby L is the span of the freeform grid shell structure. Initially the first buckling mode for the south grid shell structure without any imperfections and regarding the governing load combination was determined. The resulting buckling mode shown in Fig. 15 was then scaled to a maximum imperfection value of L/300 = 66 mm in order to generate a modified structural geometry of the south grid shell with imposed imperfections. Then the first buckling mode was computed again for the imperfect structure.

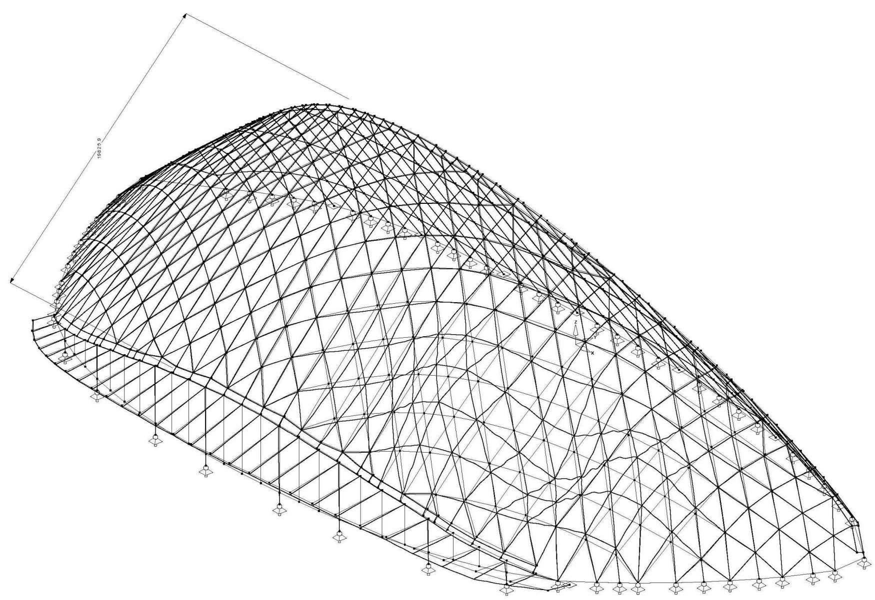

Fig.15: First Global Buckling Mode of South Freeform Grid Shell

The first buckling mode of the imperfect south grid shell structure under the governing load combination was established with a critical load factor of 4.6 which is meeting the requirement of the Chinese building code.

5.2 Stability Analysis according to Eurocode

The computation of the critical buckling load factor is a linear elastic calculation which is not considering nonlinear second order effects when increasing the axial loads. Therefore the critical load factor cannot be interpreted as a safety factor against global buckling, which will in fact happen at a lower load level than predicted by the critical load factor. In order to assess the safety margin against global buckling a direct buckling analysis according to Eurocode EN 1993-1-1 was done.

The global imperfection for the direct buckling analysis of the structure was determined to L/300 according to EN 1993-1-1, table 5.1 "bow imperfections", buckling curve "a". Due to the nature of the direct buckling analysis, the governing load combination had to be changed to the limit state combination 1.2*DL+1.4*WL45+ 0.98*LL_R+0.7*TL(+) corresponding to the service load combination used for the elastic buckling analysis. The first buckling mode for the south grid shell structure without imperfections was computed for this limit state load combination, scaled to a maximum imperfection value of L/300 = 66 mm and the modified imperfect structural geometry of the south freeform grid shell was generated. In addition to these global imperfections of the structure local bow imperfections of individual members were imposed as per EN 1993-1-1 in order to perform a simultaneous global and local buckling analysis. The modulus of elasticity of the steel material was reduced to 90% of the nominal value. The resulting structural model was analysed using second order theory. If the analysis is converging and the member stress is below the design limits, then the global stability of the structure as well as the local stability of the individual members can be confirmed.

This structural analysis was repeated with incrementally increased loads until the analysis is not converging anymore. The last load step before this failure load is the stability load limit. The ratio of this stability load limit to the limit state design loads is apparently the true safety factor against buckling. For the south freeform grid shell this true buckling safety factor was established with 2.3. Using the partial safety factor of the wind load of 1.4 this gives us a total safety factor of 3.22 which is significantly lower than the critical load factor of 4.6 found in the linear-elastic buckling analysis.

6 Fabrication and Installation

As described in [2] the node connector is the most important structural component of any single layer freeform structure as it is directly and very significantly influencing the geometric flexibility, the structural performance, the production and installation cost and last but not least the visual appearance of the structure.

Fig. 16: Check of CNC Program

Fig. 17: CNC Node Fabrication

The proprietary Novum grid shell design software program was used to generate the fabrication data for all node connectors of the pavilion structure using only the wireframe CAD model of the structure and the member profile and connection data from the structural analysis. The generated fabrication data was then inserted into specially prepared software programs which control the fabrication process of an automated 5-axis CNC machining center. These special programs for each node were checked by virtual simulations in a CAM software as shown in Fig. 16 before the node fabrication started which is illustrated in Fig. 17. The ready fabricated and numbered nodes as shown in Fig. 18 were arranged in the installation sequence in batches, coated and shipped to the site.

The fabrication data of the members was as well generated by the proprietary Novum grid shell design software program. The member fabrication is a conventional steelwork production process using parametric drawings together with the generated member data. The length tolerances of those members need to be between +0 and -1 mm to ensure the fit-up with the node faces which are machined with very high accuracy. The ready fabricated and numbered members as shown in Fig. 19 were again arranged in the installation sequence in batches, coated and shipped to the site.

Fig. 18: Fabricated Nodes

Fig. 19: Fabricated Members

The installation process of a freeform structure typically starts at three building support points which are preferably on a bigger distance to each other. Those supports have to be kept floating until a sufficiently rigid preassembled grid shell "patch" connecting those three supports is lifted into position. Then the three supports will be fixated by site welding or bolting. After that the initial grid shell patch will be incrementally extended at the outer edges toward the adjacent support points by adding clusters of 3 or 4 members which are preassembled to one node adjacent to the already assembled grid shell portion.

This way the labour intensive and slow work of picking the right members connected to a particular node out of the batch, orientating these members and the node correctly to each other, inserting the correct bolts and tightening it etc. can be done on the ground at special cluster preassembly points close to the storage area.

The output of those cluster preassembly points is naturally limited, but by having several preassembly points which are parallel working in a coordinated way the installation speed can be significantly increased, since the process of lifting preassembled clusters to their final position and fixing the bolted node connections of the free member ends is very fast.

Fig. 20: South Freeform Structure

Fig. 21: Pavilion Structure

This incremental installation process which is relying on the accuracy of the machined node geometry and the rigidity of the pretensioned node connection has to be simulated upfront in order to determine where temporary supports would be needed to avoid significant deflections of the free patch edges under the dead weight of the structure. This structural analysis of the installation process is also delivering the information needed to design those temporary supports which can be scaffolding platforms or individual scaffolding towers. The installed pavilion structure is shown in Fig 20 and 21.

References

[1] Soeren STEPHAN et al: General Method for the Design of Bolted Connections for Space Frames, Space Structures 5, Telford Publishing, London 2002, p. 759-773

[2] Soeren STEPHAN et al: Reticulated Structures on Freeform Surfaces, Proceedings of the IASS Symposium "Shell and Spatial Structures From Models to Realization", Montpellier / France 20 Sep 2004 - 24 Sep 2004

[3] Kyung-Ju HWANG et al: Influence of various types node connectors on the buckling load of grid shells, Proceedings of the IASS Symposium "Evolution and Trends in Design, Analysis and Construction of Shell and Spatial Structures", Valencia / Spain 28 Sep 2009 - 02 Oct 2009

[4] Catherine A CARDNO: Traveling Single-Layer Diagrid Structure to Be Built For 2010 World's Fair, Civil Engineering Oct 2009, p. 21-22

[5] Lianjin BAO et al.: Design of Single Layer Lattice Shell with Semi-Rigid Connections, Submitted to Journal of Building Structures (in Chinese)

[6] Stephen KENNETT: Sands of Time, Building UK 28 May 2010, pages 60-62