MODELLING OF FRICTION CONNECTION FOR WIND TOWERS
Finite Element Simulation

Marouene Limam [a], Milan Veljkovic [a] and Lars Bernspång [a]

Carlos Rebelo [b] and Luis Simoes da Silva [b]

[a] Luleå University of Technology, Div. of Structural Engineering, Sweden
[b] University of Coimbra, Dept. of Civil Engineering, Portugal

INTRODUCTION

The tubular steel towers supporting wind turbines account for about 15 to 20% of the total installation costs and their optimisation may lead to substantial savings with regard to costs and use of material. An important aspect of the design is the connection between tower's sections. The present assembling detail between two segments of the tubular tower is accomplished by conventional ring flange connection where pairs of heavy steel flanges are welded on the inside of the tubes and bolted together with pre-tensioned high strength bolts as shown in *Fig. 1*.
A new friction joint has been developed in the recently finished RFCS research project HISTWIN (High-Strength Steel Tower for Wind Turbine) where open slotted holes has been used to facilitate the assembly of the tubular tower as shown in *Fig. 2*.
This paper focuses on the Finite Element (FE) modelling of the friction connection using the commercial software Abaqus. An axial force on the tower is considered to illustrate the behaviour of the connection.

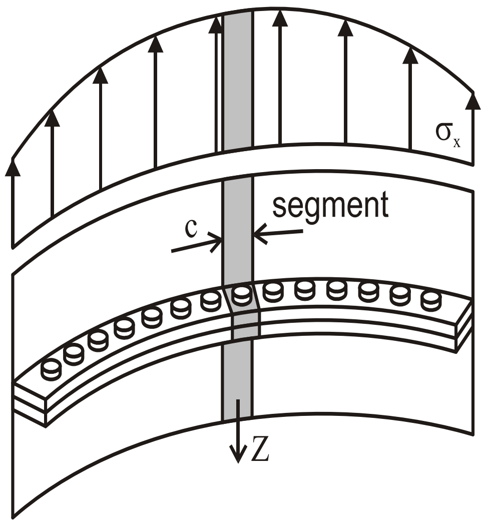

Fig. 1. Ring flange connection

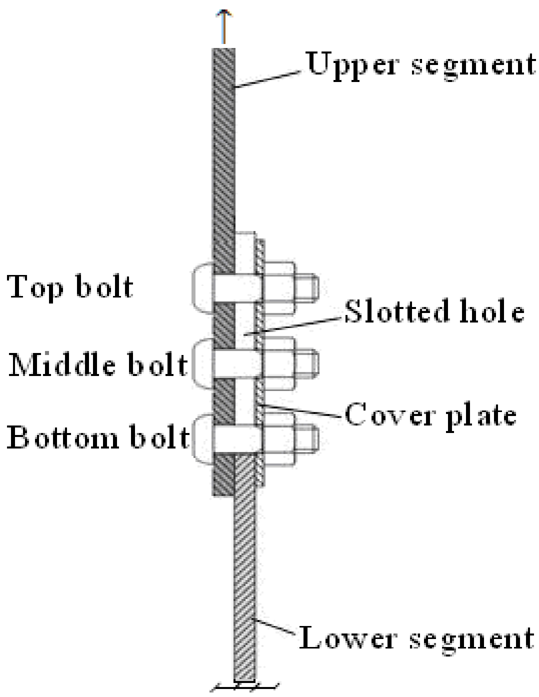

Fig. 2. "HISTWIN connection", the lap connection

1 DESCRIPTION OF A FINITE ELEMENT MODEL OF THE LAP CONNECTION

The lap connection shown in Fig. 2 is considered in the paper. The name "HISTWIN connection" comes from the RFCS project under same name where the connection is developed:
- 8mm thick plate with the normal clearance bolt holes,
- 8mm thick plate with a long opened slotted holes,
- 6mm cover plate, with material properties of a washer, is placed on the plate with the slotted hole
- High-strength bolts, M20 and M30, 10.9 grade are considered for a sake of illustration. The normal clearance holes, 22mm and 33mm diameter respectively allow the bolt movement relative to the plates when the slip load is reached.

Steel grade S355, elasto-plastic isotropic hardening model is used for the plates.

1.1 Contact interactions

The contact surfaces are modelled with a master-slave algorithm having the bolt shank, bolt head, and nut to be master surfaces (as the bolt is of stiffer material) with all the other contact surfaces considered as slaves [1].The contact areas comprise the bolt shanks-to-bolt holes, bolt heads-to-upper segment, nuts-to-lower segment, and upper segment to lower segment.

The tangential behaviour is used to enforce contact constraints based on penalty method, which approximately enforces the contact constraints by use of springs without adding degrees of freedom to the matrix structure. Friction coefficient equal to 0.35 is assumed using the basic Coulomb friction model.

Contact discretization is another important configuration. *“surface to surface”* contact is used for all the contact surfaces. Finite Element models including contact problems are very sensitive to any applied load and may have problems to numerically converge at the beginning of a calculation. The reason is that the parts are not fully in contact (the contact pressure is zero at the contact surfaces) therefore rigid body modes may exist due to the lack of boundary conditions. It may be necessary to stabilize the model by activating damping in normal and/or tangential direction.

1.2 Element types

A fin mesh in the FE model is necessary to get convergence to the true solution and also to avoid contact problems on expense of computation time.

In order to have optimum mesh it is necessary that the mesh is more refined near to the connection area to avoid high stress concentrations during the slip. Since the connection is designed for tensile loading (*Fig. 2*) and based on our practice it is good to use C3D8 elements in the contact zones, C3D8I elements for the bolts and C3D8R (reduced integration) elements outside of the connection region which can be used to decrease the computational time.

1.3 Numerical procedure and boundary conditions

The analyses are performed in the following two sequential steps.

1. Pre-tensioning of bolts: In this first step a pre-tensioning load of 160kN is applied in all the bolts. The method used for pre-tensioning can be either *“Adjust length”* or *“Apply force”*. The purpose of this step is to initialize all surface to surface contact interactions.
2. Fixing the bolt length: The bolt length is fixed at its current length by using the option *“Fix at current length”* in Abaqus.

The model is fixed in one plate and a displacement-controlled loading is applied to the other plate at one reference point until either a bolt-hole bearing or a bolt shear limit state is reached. The displacement-controlled is recommended instead of force-controlled since it provides a higher numerical stability to the system.

1.4 FE results

The global behaviour of the lap joint in tension is shown in Fig. 3 and Fig. 4 where two different levels of the gap between the “inner” and the “outer” plates are considered. A size of a gap between plates is chosen so it corresponds to dimple imperfection of two extreme fabrication classes of the tower, 1.5mm and 3.9mm for class A and C respectively, acc. to EN1993 part 1-6 [3]. This choice is made for the sake of comparison of the connection behaviour of a tower in the compression and tension.

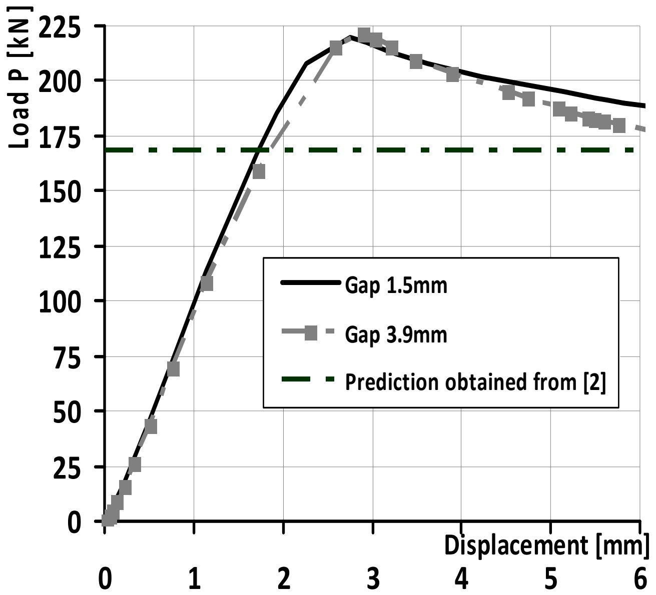

Fig. 3. Load-displacement curves

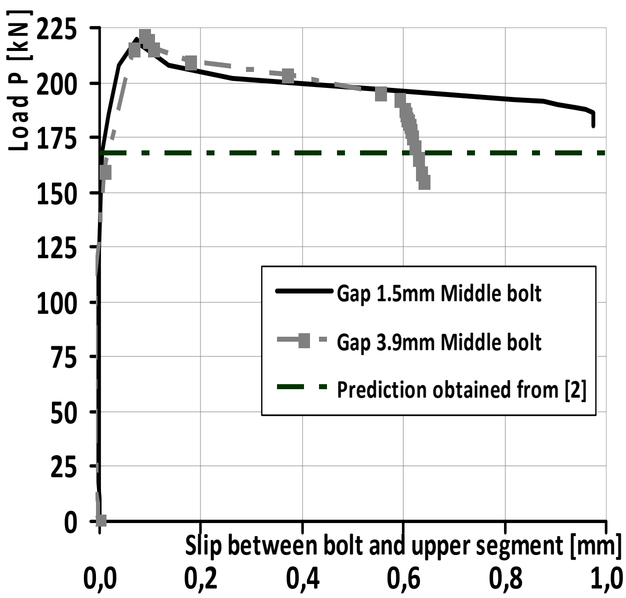

Fig. 4. Load slip curves

The slip load of one bolt, in a symmetric connection, is predicted by *Eq. (1)*, acc. to EN 1993-1-8 [2],

$$F_{\mathrm{s.Rc}} = n \mu k_{\mathrm{s}} F_{\mathrm{p.C}} \tag{1}$$

where n is the number of friction interfaces,

$F_{\mathrm{p.C}}$ is the pre-tension force in the bolt,

$\mu = 0.35$ is the slip factor (slip coefficient) of the faying surfaces, assumed in this paper,

$k_{\mathrm{s}} = 0.63$ is experimentally established in HISTWIN project [4], for normal holes $k_{\mathrm{s}} = 1.0$,

Due to the eccentricity of the connection, shown in *Fig. 5*, the axial force in the connection consists of two components; the externally applied force and contribution from the eccentricity.

$$e = t + g \tag{2}$$

$$\sigma_{\max} = \sigma_{\mathrm{N}} + \sigma_{\mathrm{M}} \tag{3}$$

$$M = F_{\mathrm{N}} \cdot e \tag{4}$$

where M is the additional bending moment, caused by the eccentricity, that influence increase of the contact pressure,

e is the eccentricity of the connection, including the gap, g, between plates, see *Fig. 5*,

σ_{M} is the bending stress component in the plate,

b is the plate width,

t is the plate thickness,

σ_N is the axial stress component in the plate.

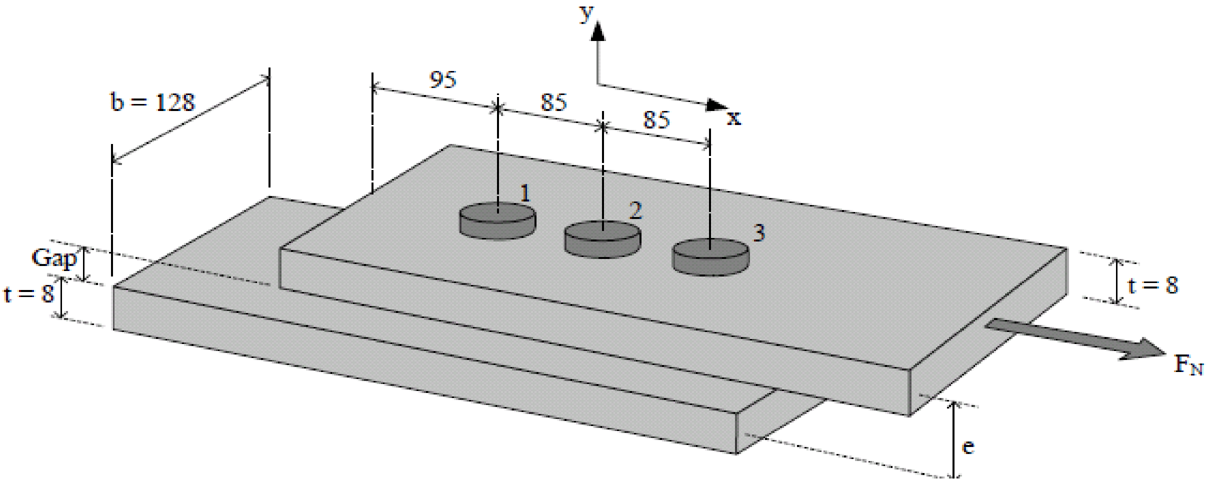

Fig. 5. The lap joint before pretensioning, dimensions are in mm

From diagram in *Fig. 3*, it is clear that prediction based on symmetric friction connection [2] which predicts 168kN as the maximum load underestimates the total lap joint resistance F_{N}. For the purpose of this comparison $k_{\mathrm{s}} = 1.0$.

2 FINITE ELEMENT MODELLING OF FRICTION CONNECTION OF AN AXIALLY LOADED TOWER SEGMENT IN COMPRESION

A very detailed three-dimensional (3-D) FE model is prepared where the geometry is modelled with solid elements and a very realistic model of the connection is achieved by using contact elements and a special option to model pretension in the bolts.
The connection consists of two tubular segments (*Fig. 6.*). The segment A, (see *Fig. 7*) has 72 equidistant normal clearance holes where 72 bolts are installed with outer diameter 1008mm and 8mm thickness. The segment B has 24 long open slotted holes, the outer diameter is 1000mm and 8mm thickness (see *Fig. 8*)
Instead of standard washers for each bolt, 24 cover plates with 250mm length, 78mm width and 6mm thickness are used; they are supposed to hold the bolt group together during the assembly.

Fig. 6. Mesh of the model

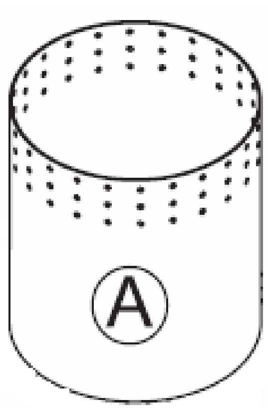

Fig. 7. Upper segment

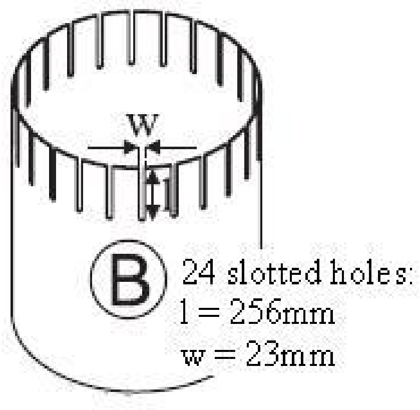

Fig. 8. Lower segment

The FEA analysis is performed in two sequential steps.

- In the first step, the pretension forces of 160kN have been introduced in bolts.
- In the second step, at the lower boundary the Segment B is clamped, boundary condition called BC1r in [3]. At the top of the Segment A is pinned, boundary condition called BC2f in [3])

are applied where a displacement-controlled loading is modelled at one reference point. Two alternative failure modes are possible, either bolt-hole bearing or bolt shear limit state. The displacement-control is recommended instead of force-controlled since it provides a higher numerical stability to the system.

All FE calculations clearly show the maximum load, because the descending part of the load-displacement curve or the plateau is obtained as shown *Fig. 9* where results of the axially compressed ”tower” are shown. The total length of the tower is 5000mm.

Slip between middle bolt and upper segment vs. total load curves are presented in *Fig. 10* for two different gap sizes and two different bolt dimensions. The slip between the bolt and the upper segment reaches its maximum at 2mm, at this state the bolt shank gets in contact with bolt holes surface for the models using M20 bolts.

Influence of the gap size on longitudinal stress distribution along the shell is shown using results obtained from the first step of the FEA. In *Fig. 11* just one strip around the connection is shown and corresponding diagram of the longitudinal stress. It is clear that the longitudinal stresses are closely distributed at the connection area (approximately 1m) independently of the size of the gap. The holes with normal clearance are in the segment A and the open slotted holes are in the segment B, respectively. Therefore higher longitudinal stresses are in the segment A because it is the stiffer part of the connection.

The ultimate plastic resistance of the tower is 8850kN, as shown in *Fig. 9* to *Fig. 11*. Influence of geometrical imperfection introduce as a gap between cylinders do not have a big effect on the maximum load. The predicted resistance due to shell buckling acc. to [3] is 7625kN and 7322kN for class A and C, respectively. These predictions are clearly overestimated in FEA, and the local buckling is obtained after the plastic resistance is achieved in the region out of connection.

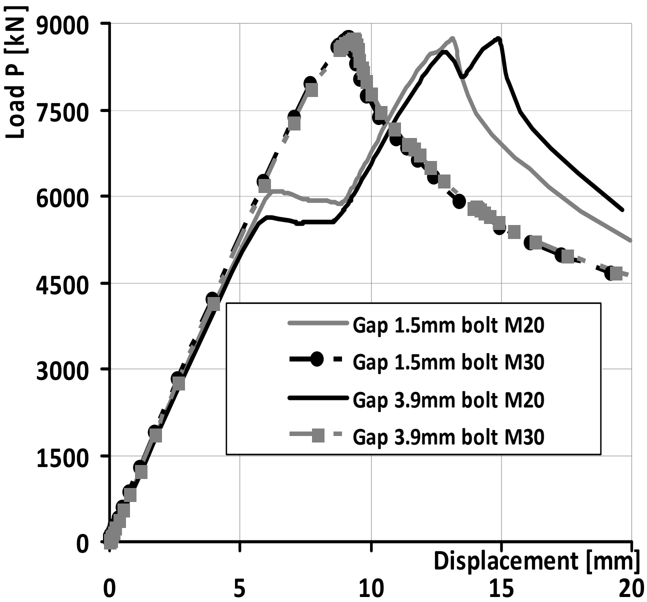

Fig. 9. Load-displacement curves

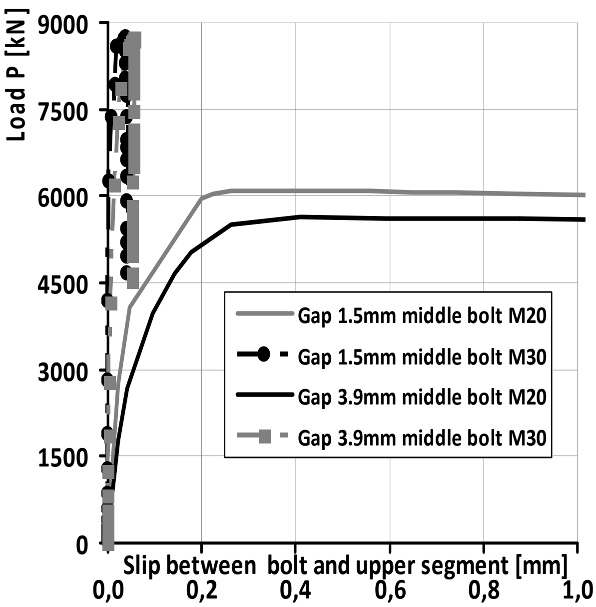

Fig. 10. Load slip curves

Similar tendency of overestimating the strength of the connection shown in Ch 1.4, is obtained for the tower segment in axial compression. Predicted forces for symmetric lap connection where pretension force of 160kN and 400kN are used for M20 and M30 respectively, should lead to a failure in connection at the total load 4032kN and 10080kN respectively. It is obvious that the slip failure in the connection with M20 bolts occurs and after that the bearing of bolts allows the plastic resistance to be reached. It is clear from *Fig. 11* that the normal stresses due to closing the gap are locally distributed and the maximum stress is about 150MPa.

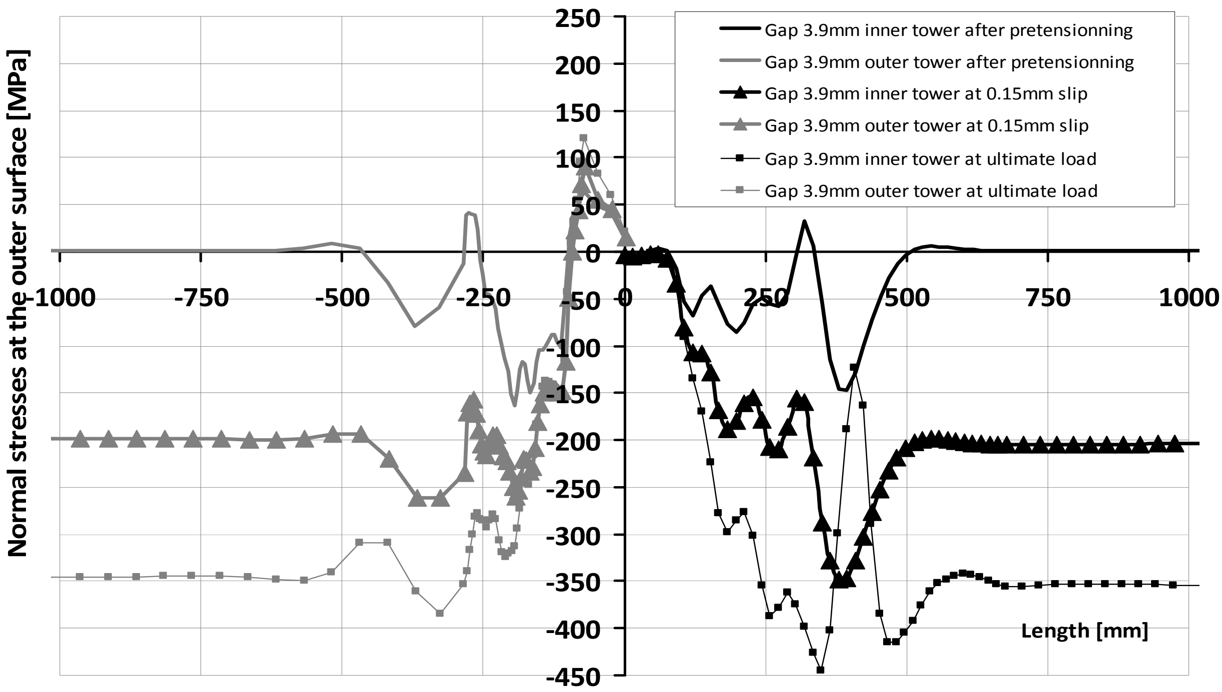

Fig. 11. Longitudinal stress distribution at the inner and outer cylinder in the region of the connection at the maximum load achieved during the pretension

3 CONCLUSIONS

FE-analysis of friction connections have been compared with prediction using hand calculation models available in [2] and [3]. For the comparison two different bolt sizes and two different gap sizes have been used. Very realistic FE model of the HISTWIN connection allows better insight into the behaviour of the connection and the internal forces distribution. Results obtained in the FEA are in agreement with experimental prediction in [4]. Special attention of this paper is given to:

- Slip in the connection and the corresponding applied force.
- Distribution and size of the bending stresses in the lap connection caused by closing the gap during the assembling of the tower and at the various levels of applied force.

4 ACKNOWLEDGMENT

The research leading to these results has received funding from the European Community's Research Fund for Coal and Steel (RFCS) under grant agreement n° RFSR-CT-2006-00031. Furthermore, the first two authors gratefully acknowledge very good collaboration with the project partners on the HISTWIN project: Institut für Stahlbau der RWTH Aachen, Germany, Germanischer Lloyd Industrial Services GmbH, Hamburg, Germany, Aristotle University of Thessaloniki, Thessaloniki, Greece, Repower Portugal Equipamentos Eólicos SA, Oliveira de Frades, Portugal, and Rautaruukki Oyj, Hämeenlinna, Finland.

REFERENCES

[1] Abaqus Analysis User's manual version 6.10, Dassault Systèmes Simulia Corp., Providence, 2010

[2] European Committee for Standardization (CEN), EN 1993-1-8, "Eurocode 3: Design of steel structures. Part 1-8:Design of joints", Brussels, 2005

[3] European Committee for Standardization (CEN), EN 1993-1-6, "Eurocode 3: Design of steel structures. Part 1-6:Strength and stability of shell structures", Brussels, 2005

[4] Veljkovic, M, [et al.], "HISTWIN – High-Strength Steel Tower for Wind Turbine", Grant agreement n° RFSR-CT-2006-00031, RFCS Publications, European Commission, Brussels, 2011