DOI: 10.24904/footbridge2017.10478

TOWARDS A FULLY DIGITAL MODELLING OF STEEL JOINTS AT ULS

Luciano TOSINI

Civil Engineer, co-founder
STRAINS
Paris, France

luciano.tosini@strains.fr

Mathieu ARQUIER

Civil Engineer, co-founder
STRAINS
Paris, France

mathieu.arquier@strains.fr

Xavier CESPEDES

Civil Engineer, co-founder, CEO
STRAINS
Paris, France

xavier.cespedes@strains.fr

Summary

For aesthetics' or structural reasons, footbridges design tends to push slenderness much further than any other structure. Moreover, it usually is the opportunity for both engineer and architect to express their creativity leading to ambitious designs. The computation of the global behaviour is generally not a problem with the use of finite element software, but when it comes to structural details, particularly to the ULS behaviour, there is discomfort. Indeed, code based formulas are most of the time inapplicable and local finite element models are time-consuming, slow down design iterations and limit the creativity of the project team.

This article describes the reasons that led structural engineers to develop a software whose goal is a fast and reliable determination of steel connection ultimate resistance, and how it can help engineers to accelerate design iterations leading to a more optimal structure. The full paper gives scientific details.

Keywords: yield analysis; computational design; steel connection analysis; cloud software

1. Introduction

The story started in 2010 in SETEC-TPI offices in Paris. The authors oversaw the independent checking of structural steel structures for the “Fondation Louis Vuitton”. The outstanding building geometry, designed by Frank Gehry, led to highly complex steel structures, with little apparent structural sense. But unlike some others “geometry-driven” steel structures, these ones are supporting heavy glazed “sails”, subjected to complex dynamic wind effects. There was therefore a real need for detailed structural checks. Checking 3D plastic capacity of complex steel assemblies undergoing complete 3D force systems, using Eurocode requirements, appeared to be a very challenging task. The problems encountered with general purpose finite element software, conducted to rely mainly on hand calculations.

It was not an isolated case and our colleagues were faced with the same difficulty of analysing complex constructions details, particularly in footbridges where the aesthetic' criteria is important, therefore most of us let the steel contractors deal with the problem. However, it is the role of the structural engineer to design up to the smallest detail. And it has become more complicated since the emergence of BIM and digital models: design and changes are faster, at a click of a mouse. The trouble for structural engineers is that our methods and computational tools have not improved while with the emergence of new digital tools it is possible to draw the most complex structures!

The difficulty today is to compute the ultimate capacity of complex structures' details. The more complex ones are found in pedestrian footbridges. On the one hand the direct application of codes is not possible except by simplifying the problem and by taking uncertain margins of security. On the other hand, the use of current computational tools does not make it possible to give a satisfactory answer within a time compatible with the project schedule.

Based on this observation, we worked on the subject, with the help of researchers from "Laboratoire Navier, Ecole des Ponts", Paris. As far as the problem to be solved is plastic capacity, the solution lies in the numerical implementation of limit analysis. We left SETEC-TPI and created STRAINS to develop the first tool that implements this method to calculate the ultimate capacity of any structural detail.

2. Limit analysis for steel structures

An example of a footbridge' 3D steel connection is given. Structural engineers have only hours not days to create the model. So the definition of specific CAD objects like beam, plate, weld or bolt is the key.

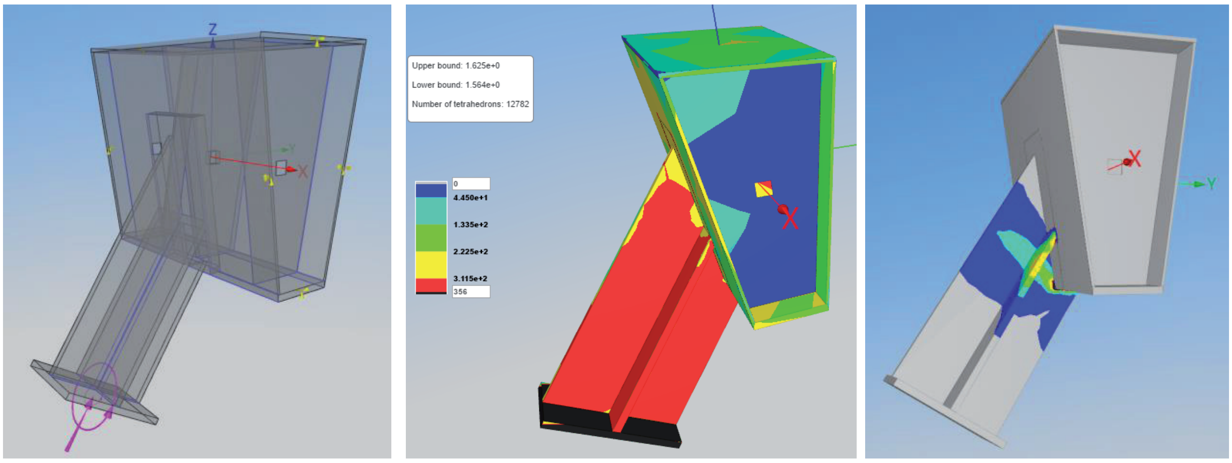

Fig. 1. 3D Strut connection on a box girder section, a) geometry and load, b) stress state at failure c) failure mechanism and plastic strains. The ultimate load factor is bounded: α^{opt}=1.56 and β^{opt}=1.63

Limit analysis aims at studying a structure at failure, assuming all materials have reached (and withstood) their limit strength criterion. The elastic behaviour is therefore not included in the analysis; as a consequence, no elasto-plastic iterations need to be performed. The underlying assumption is that the materials allow high ductility deformations and the structure behaviour is far from instability.

The plasticity is defined thanks to a criterion which limits the stresses. It's usually defined by a function *f* over an admissible stress value domain G. The limit stress for the strut in the example is 355 MPa.

$$\sigma \in G \Leftrightarrow f(\sigma) \leq 0 \quad (1)$$

Given a force systems (F) applied to the structure, the scope of limit analysis is to estimate the bounds of the load factor to failure λ by two independent computations:

- The static approach seeks the statically admissible stress field withstanding the highest load factor α^{opt} of the force systems (F). α^{opt} is a lower bound of λ and the computed stress field highlights strongly constrained zones.
- The kinematic approach seeks the velocity field that minimize the ratio "plastic dissipation/power of external forces". This ratio provides $\beta^{opt:}$ upper bound of λ. The computed velocity field emphasizes faulty areas.

Convex optimization algorithms lead with robustness to a solution. Recent progress in both Mathematics and Computer Science make possible a computation of both load factors in minutes not days, allowing the precise assessment of the steel node capacity. The computation is based on a finite element mesh, on which either the stress field or the velocity field are discretised and interpolated. The reliability of the solution can also be deduced by the α^{opt} and β^{opt} bounds; the closer they are, the more precise the load factor to failure λ is. If the precision is not satisfactory, a mesh refinement algorithm governed by the plastic dissipation density leads to a more accurate solution.

3. Conclusion

The analysis of some lacks among structural engineers' tools was made that limit the detailed design of complex projects, such as footbridges. The authors propose a steel connection software with the intention to enable the design of safe and optimized structures and to assess with liability existing structures.