Towards a fully digital modelling of structural joints at ULS

Xavier Cespedes, Luciano Tosini, Mathieu Arquier

Strains, Paris, FRANCE

Contact: luciano.tosini@strains.fr

Abstract

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 the engineers to stay connected to the BIM digital workflows.

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.

Indeed, the emergence of BIM as a collaborative, interactive way of designing and checking projects, highlights the need of fully digital tools for all project members. Design is faster, checks are faster, and changes are faster. One fundamental idea behind the process is that once a design change is proposed, it can be treated very quickly by all members, and the project is updated permanently.

Although having been the first to get professional numerical tools in the 70's, the structural engineer is now among the ones with the biggest lacks in his numerical chain, and remains often reluctant to the fast-iterative management of complex BIM projects.

Another idea behind BIM is that it can be used during the building's life, for maintenance and, from the engineer's point of view, to assess structural health, for example when coupled to wireless sensors of all kinds that are now rapidly spreading. Structural health monitoring normally relies on reverse engineering for interpreting the data, and on direct simulation to analyze the possible impact of the changes on the structure's safety. Automatizing the whole chain is, here also, the key for the supervision of infrastructures, and developing, for example, alert thresholds.

After the end of this project, it was thus observed that this problem was becoming rather common, Not only for bolted assemblies of steel structure, but also for 3D parts of concrete structures, timber connections, structural connections to RC blocs, anchorages of prestressing cables or bars…

Structural Engineers are very trained to the use of structural software at the global scale of their structure, dealing with member forces, determined by static or dynamic calculations, generally under elastic assumptions. But when it comes to the design or assessment of local features, the same software is hardly useable, because of poor CAD capabilities, not allowing efficient description of the geometry, and because local design is generally a capacity problem, out of the scope of elastic analysis, and even hardly reachable by common implementation of elasto-plastic algorithms. In practice, many engineering softwares propose code-based checks for the most standard details, and leaves the complex ones to the (hand-based) expertise of the engineer.

Restoring the digital workflow continuity for structural analysis could theoretically rely on existing general-purpose commercial finite element software. But one can observe that this is not generally done. Two main reasons can explain this reluctance: First, representing the geometry of bolted assemblies or 3D reinforced concrete elements on a general-purpose CAD interface may take days or even weeks, and rarely fits in a project timetable. Secondly, as far as force capacity is concerned, nonlinear computations are required, the convergence of which should be driven as far as possible. At this point, it is often the case that the convergence cannot be obtained by automatic software procedures, and requires engineer's control, adding extra days or weeks to the calculation cost: the cost of using numerical software is often much higher than the cost of hand calculations. Finally, when presenting numerical results, one must fight against a common distrust in the engineer's community, because of mesh sensitivity, difficult convergence control, the frequent absence of error estimates etc., questioning the result's reliability.

Thus, assuming that new numerical methods are needed, the team looked for applicable theories, hopefully leading to the design of a dedicated tool.

Brainstorming was extended to the "Laboratoire Navier", at "Ecole Nationale des Ponts et Chaussées" in Paris, where it appeared that from the numerical point of view, as far as the problem to be solved is the plastic capacity, the use of "yield design" limit analysis should be the best choice: It enables faster computations, safe convergence without need for user control, and, above all, the opportunity to get values for the "lower bound" and "upper bound" capacity, giving at the same time a "safe value" (the lower bound), and an error estimate that can be reduced by mesh refinement. The security given by the existence of this "lower bound" is inestimable for engineering experience, and unreachable with classical elastoplastic FE modelling. Furthermore, this method could be used for similar problems encountered for reinforced or prestressed concrete structures, where 3D parts are still often designed or checked using struts and ties with manual search of the good resistant pattern.

Given the rather difficult experience of the team in modelling 3D joints with the built-in CAD tools of FE softwares, it was believed that useable software should not only have efficient and safe solver, but also a dedicated CAD interface, making 3D modelling much easier, and hopefully allow BIM interaction.

Such software would help engineers designing and checking steel joints and 3D concrete parts. It would also, by reducing and progressively eliminating hand calculations, help make the whole checking process digital, and drastically reduce the time engineers need to adapt to any design modification, finally accelerating the whole BIM process. To this purpose, connections with global structural software solutions should be developed to avoid the re-introduction of member forces.

The final project complexity, made it unlikely to be developed as a new "in-house" software restrained to Setec employees. Following this logic, the team decided to create a whole-new company, to test the commercial perspectives for this innovation – and some others not in the scope of this paper.

2 Limit analysis for steel structures:

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, therefore no elasto-plastic iterations need to be performed. But the underlying assumption is that the materials allows 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.

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

As material criterion, here is a list of few examples:

- $N_c \leq N \leq N_t$ for 1d beam under normal force
- $M_1 \leq M \leq M_2$ for 1d beam under moment
- Von Mises for 3d steel model
- Drucker-Prager for 3d soil model
- …

Limit analysis combines two approaches:

- The "static approach" looks for an admissible stress state withstanding the highest possible external load. Its principle is commonly used in civil engineering hand calculations, for example when looking for "struts and ties" scheme in RC elements, where it relies obviously on the engineer's "educated guess", the results being possibly different when 2 engineers try to solve the same problem.
- The "kinematic approach" postulates a structural movement, and compares the energy dissipated by the plastic flow to the work of external forces. This method is also quite common for soil stability analysis (slope stability, etc.) and steel connections analyses, per Eurocode requirements.

When used in hand calculations, both "approaches" have a major drawback: the engineer must guess what the solution is likely to be. Incorrect guess for the stress field of the static approach leads to overconservative results, whereas an incorrect guess for the structural movement in the kinematic approach leads to unsafe conclusions.

Hence, limit analysis is the research of an optimal solution of both approaches. This research can benefit from a finite element mesh, on which both the static stress field or the kinematic displacement field are discretised and interpolated.

2.1 Static approach

In a static approach and for a given external load *F*, one wants to find the safety factor λ which leads to failure. To do so, a first stress distribution σ^1, in equilibrium with F, is guessed. Then, the maximal multiplier α^1, such that $\alpha^1\sigma^1$ withstands the given material criterion $f(\alpha^1\sigma^1) \leq 0$, is found.

As only one stress distribution has been analysed, one does not know if it is the optimal distribution. Hence $\alpha^1 \leq \lambda$, and, by implementing this method with another distribution σ^2, still in equilibrium with *F*, one gets α^2 which can be higher or lower than α^1.

Trying all admissible distributions in equilibrium with *F* would lead to the calculation of the maximum of all multipliers α^i. But as all possible distributions cannot be tried, $\alpha^{opt} = \max(\alpha^i)$ is computed as a lower bound of λ.

The output of this approach is α^{opt}, lower bound of the load multiplier, and the corresponding stress distribution. The drawback is the lack of information regarding the displacement field.

2.2 Kinematic approach

The kinematic approach is based on the research of an optimal displacements field which minimizes the deformation energy. For a given external load *F*, the Virtual Work Principle (VWP) over the structure gives:

$$F.\hat{U} = \iiint_V \sigma:\varepsilon(\hat{U})dV + \iint_\Sigma (\sigma.n).[\![\hat{U}]\!]d\Sigma \quad (2)$$

REMARK: If there is no discontinuity surface Σ, the last term of the above equation vanishes.

As, by hypothesis, all materials withstand their criterion:

$$\sigma:\varepsilon(\hat{U}) \leq \max[\sigma:\varepsilon(\hat{U}), \forall\sigma \in G] \stackrel{\text{def}}{=} \pi(\hat{\varepsilon}) \quad (3)$$

with π the support function of the criterion.

The same inequality can be written for the discontinuity surfaces Σ:

Defining:

$$\pi(n,[\![\widehat{U}]\!]) = \max[(\sigma.n).[\![\widehat{U}]\!], \forall\sigma \in G] \quad (4)$$

We have:

$$(\sigma.n).[\![\widehat{U}]\!] \leq \pi(n,[\![\widehat{U}]\!]) \quad (5)$$

The two last equations combined leads to this inequality:

$$F.\widehat{U} \leq P_{\mathrm{m}}(\widehat{U}) \quad (6)$$

With

$$P_{\mathrm{m}}(\widehat{U}) = \iiint_V \pi(\hat{\varepsilon})dV + \iint_\Sigma \pi(n,[\![\widehat{U}]\!])d\Sigma \quad (7)$$

If it is verified for all virtual displacements $\widehat{U}$, the structure withstands the load *F*. On the contrary, if at least one displacement $\widehat{U_0}$ that does not verify this inequality is found, that means the structure will collapse under *F*, with $\widehat{U_0}$ as failure mechanism.

Let's define β as follow:

$$\beta(\widehat{U}) = \frac{P_{\mathrm{m}}(\widehat{U})}{F.\widehat{U}} \Rightarrow \forall\widehat{U}\ \beta(\widehat{U}) \geq 1 \quad (8)$$

So, $\beta^{opt} = \min \beta(\widehat{U})\ \forall\widehat{U}$ is computed. Two possibilities appear:

- $\beta^{opt} \leq 1 \Rightarrow$ collapse of the structure under *F*
- $\beta^{opt} > 1 \Rightarrow$ the structure withstands *F*

This can also be written:

$$\beta^{opt} = \min \frac{P_{\mathrm{m}}(\widehat{U})}{F.\widehat{U}} \Rightarrow \min \frac{P_{\mathrm{m}}(\widehat{U})}{\beta^{opt} F.\widehat{U}} = 1 \quad (9)$$

Hence, β^{opt} can be seen in (9) as the load multiplier to the failure. As in the static approach, all possible $\widehat{U}$ cannot be tried, and β^{opt} is calculated as an upper bound of the safety factor $\lambda \leq \beta^{opt}$.

The output of the kinematic approach is an upper bound of the safety factor and the corresponding failure mechanism. The drawback is the lack of information regarding the stress distribution.

2.3 Optimal safety factor

The combination of these two approaches, provides the bounds of the safety factor λ:

$$\alpha^{opt} \leq \lambda \leq \beta^{opt} \quad (10)$$

An optimal analysis would give the same value for the upper and lower bounds. But, as all possible solutions cannot be tried, the equality between the bounds is seldom found.

Hence, the precision on the safety factor depends on the values of both bounds: the closer they are, the more precise the safety factor is.

2.4 Optimisation problem

2.4.1 Conic quadratic optimization

As seen previously, for a given exterior load F, the aim is to calculate the mathematical optimum of two functions, with imposed constraints on σ and U:

- $\alpha^{opt} = \max_{\sigma} \alpha(\sigma)$ in the static approach (with σ in equilibrium with F and $f(\sigma) \leq 0$)
- $\beta^{opt} = \min_{U} \beta(U)$ in the kinematic approach (with U admissible)

By discretising the mechanical problem over a 3D solid mesh, these two problems can then be written in the form of conic quadratic optimization:

$$\min_{\substack{x \\ Ax=b \\ x\in C}} c^T x \quad (11)$$

The domain C is the dot product of several sub-domains C_i which can either be the positive orthant $C_i = \mathbb{R}^+$, or the quadratic cone of dimension

$$d_i, C_i = \left\{ y \in \mathbb{R}^{d_i}, y_0 \geq \sqrt{\sum_{j=1}^{d_i-1} y_j^2} \right\} \quad (12)$$

This optimization is computed thanks to the interior-point algorithm.

2.4.2 Different criteria (or laws) to consider

To be realistic, the physical behaviour of all components of the structure must consider:

- Rigid-plastic behaviour of steel (3D Von Mises)
- Coulomb law (for friction)

3D Von Mises law

The Von Mises criterion can be written as:

$$f(\sigma) = \sqrt{\frac{3}{2}}\|\sigma^{\text{dev}}\| - f_y \leq 0 \quad (13)$$

with σ^{dev} the deviatoric part of σ.

In the static approach:

We then have σ belonging to the intersection of a hyperplane and a cone of dimension 7.

To ensure the equilibrium, we add the constraints $[\![\sigma.n]\!] = 0$ between two adjacent finite elements, and the constraint $\text{div}(\sigma) = 0$ on all elements.

In the kinematic approach:

We have

$$\pi(\varepsilon) = \sqrt{\frac{2}{3}} f_y \|\varepsilon\| \quad (14)$$

We can rewrite the minimum of this function as:

$$\min_{U} \pi(\varepsilon) \Leftrightarrow \min_{\substack{U \\ \pi(\varepsilon) \leq t}} t \quad (15)$$

Hence for Von Mises law, (t, ε) belongs to a cone of dimension 7 and $\text{tr}(\varepsilon) = 0$.

If the displacements are discontinuous between two adjacent finite elements, the plastic energy is given by $\pi(n, [\![U]\!]) = \frac{f_y}{\sqrt{3}}[\![U]\!]$ and $(t, [\![U]\!])$ belongs to a cone of dimension 4 and $n.[\![U]\!] = 0$.

Friction behaviour

On a given surface dS of normal $\underline{n}$, the stress vector can be written:

$$\underline{t} = \underline{\underline{\sigma}}.\underline{n} = \sigma_n \underline{n} + \underline{\tau} \quad (16)$$

The Coulomb friction law is generally modelled thanks to a friction angle φ, or a friction coefficient $f_r = \tan(\varphi)$, by the following law (compressive stresses are counted negatively):

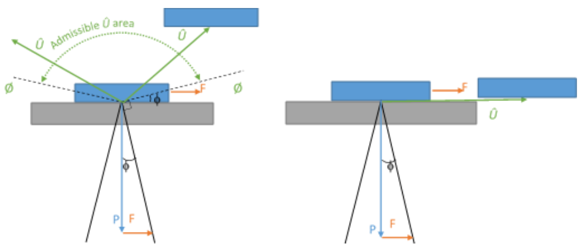
$$f(\underline{t}) = f_r \sigma_n + \|\underline{\tau}\| \leq 0 \quad (17)$$

In the static approach:

From the Coulomb law, we can deduce than $(-f_r \sigma_n, \underline{\tau})$ belongs to a cone of dimension 3.

In the kinematic approach:

In theory, the plastic flow must be normal to the criterion surface ("admissible $\widehat{U}$ area"), which leads to the loss of contact in case of friction. However, a volume can slide tangentially to another volume without losing contact. Hence, the real physical behaviour doesn't follow the plastic flow rule. This is called a non-associated law.



Frictional behaviour: associated law (left), non-associated law (right).
P is the normal force, F the friction force and φ the friction angle.

Normally, limit analysis theory only considers associated law. To tackle this issue, a solution is to use an adaptive Von Mises law which allows to slide tangentially. To do so, we need to iterate to find a Von Mises limit and to converge to an equilibrium state.

Hence, the modified law will be the same Von Mises law for the discontinuity of displacements between two adjacent finite elements, but with a different limit value.

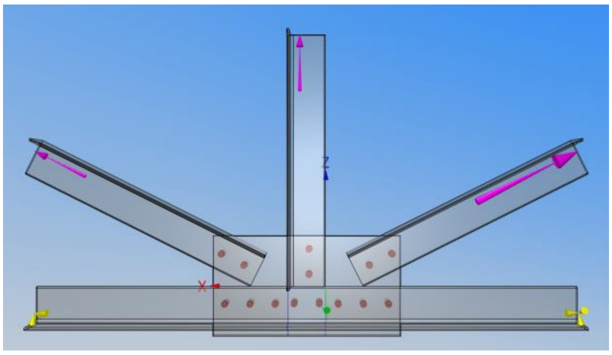
2.5 Anisotropic mesh refinement

A key feature of finite elements is the ability to refine the mesh per analysis results. This must be done automatically, focusing on the most stressed or deformed areas, and letting loose the mesh on rigid parts. Mesh refinement is controlled using the second derivative (hessian tensor) of plastic energy over the structure.

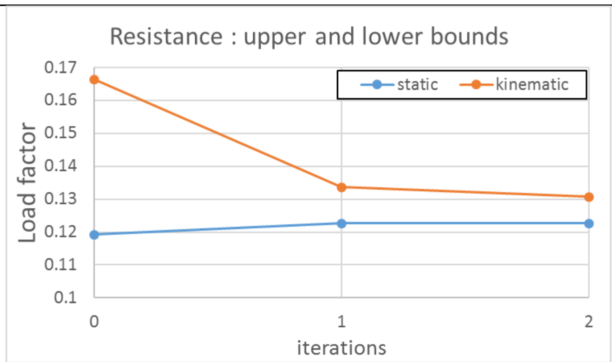
In the static approach, we can use the dual field of the stresses of the optimization problem solution to assess the plastic energy. In the kinematic approach, the plastic energy is calculated directly thanks to the displacements fields.

Moreover, as the hessian of plastic energy gives a 3D tensor, this enables to use anisotropic remesh. Solid elements are distorted on purpose, along the minimal principal direction of the tensor. Hence it is possible, for example, to create a refined mesh over a beam cross section, but a coarse mesh along the beam axis.

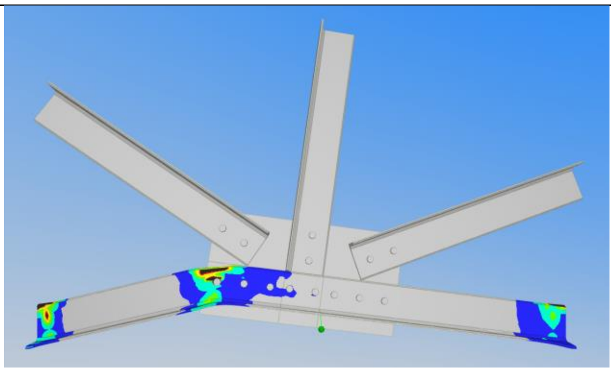
Not only this saves a lot of engineer-time and computer-time but it also allows to have better accuracy on the results because the static and kinematic bounds will be closer.



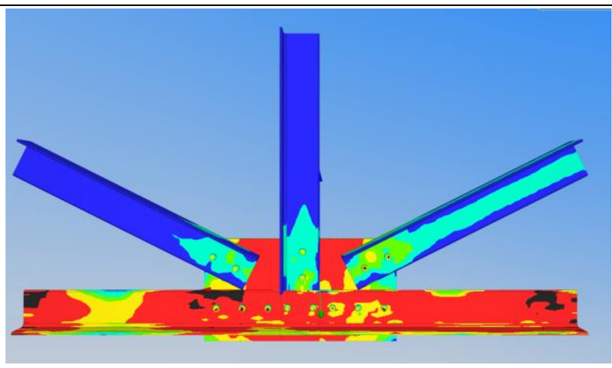
Load case



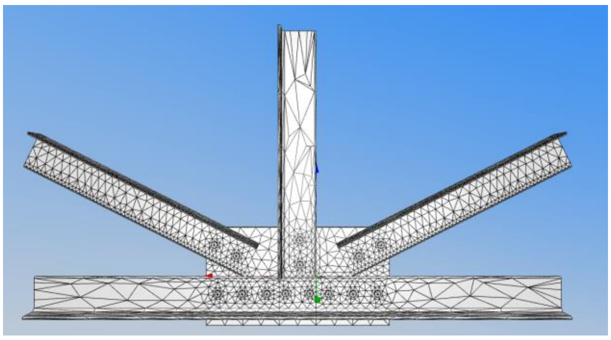
α^{opt} et β^{opt} multipliers for successive mesh iterations



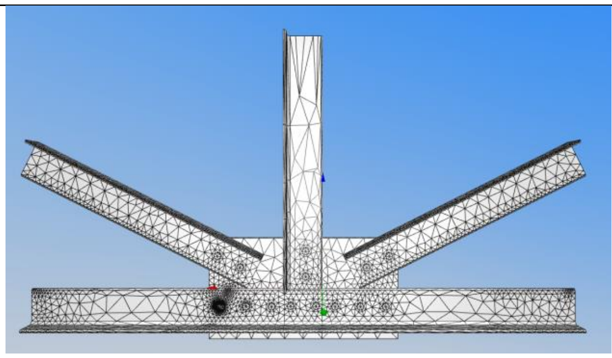
Kinematics resulting from the kinematic approach



Stress distribution resulting from the static approach



Mesh at iteration 0



Mesh refinement in the relevant zones (iteration2)

Figure 1. Re-meshing iterations and the convergence of static and kinematic results

3 CAD and model definition:

CAD is one of the most time-consuming parts of finite element computations, and the most human-intensive. Although not an academic challenge, making something easy to use remains a difficult task, and a major milestone on the way to an efficient engineering tool.

Embedded CAD of commercial mechanical software allow creating almost any kind of geometry, using complex sets of operations, including 2D sketches, extrusions along lines, boolean operations between solids, etc. But the power of such capabilities may become a drawback when it comes to design classical steel connections. Even though it makes it possible to represent nearly anything, the number of operations involved for rather simple designs becomes incompatible with engineering productivity.

The focus, though, was to make something solely for steel joints. Simplicity would come from the fact that only a relatively small number of standard elements are commonly used, and modelling efficiency could be better reached using steel manufacturer's concepts and vocabulary: beams, plates, bolts, welds, etc.: standard steel beam profiles are given in a catalogue. They should be accessible on a few clicks, directly as volume elements, that would be set in place and oriented by simple "drag and drop" actions. Bolts also are standardized elements. Declaring bolts through a set of plates or flanges should automatically adjust their length and create the holes at the correct diameter. Weld volumes are created together with the corresponding chamfers, etc., avoiding as much as possible abstract CAD concepts.

On the other hand, even steel joints may be designed with non-standard elements, either for aesthetic or mechanical reasons. These elements would be imported as IFC or STEP files, and intersected by and connected to other objects.

It seemed important to us that the model could be modified and use parameters. To that end, the model should translate into a chronological script (in Python), which could be understood and modified by the user. If any part of the model is edited, the script would be simply re-executed. Depending on the relations between objects, this helps prevent the user from breaking the model's logic and having to redo all the work.

4 Cloud computing and the global software architecture

Limit analysis, using the method described above in §2, is faster, more precise and safer than traditional non-linear finite element. But even though, determining the resistance of a single joint for a single load case on standard mesh can take 30 to 60mn on a standard laptop computer. Moreover, this analysis needs be performed twice (static+kinematic), and a second calculation on optimized mesh is usually needed for convenient precision. And one usually wants to analyse different load combinations.

Speeding up the computation can be achieved on cloud servers: they are more powerful than laptops computers (up to 32 cores, with faster memory access), and, using a convenient architecture, one can launch different load combinations simultaneously on different servers, so that the results of the whole analysis, with all load cases, each one computed on its optimized mesh, can be obtained in the order of 30mn.

This choice (cloud computing) then governs the whole software architecture: the "client" computer only runs the user interface, displaying the structure in 3D throughout the modelling process, and the deformed structure with all stress and strains isolevels after the computation is performed, whereas distant servers set up the actual CAD model, calculate the structural load cases, store the results, and send them to the "client" on request.

5 Conclusion:

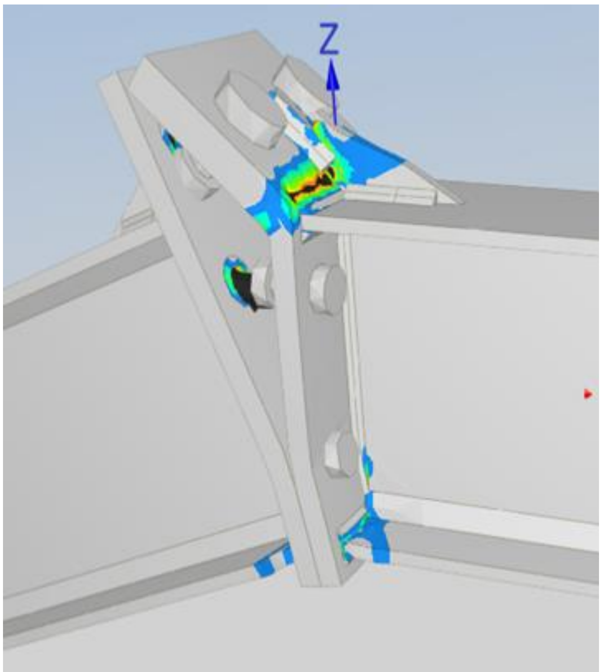


Figure 2: ULS plastic flow from kinematic approach on a simple beam-to-beam bolted connexion. Grey zones undergo zero plastic strains

Considering the emergence of BIM and collaborative design, and the growing demand for automatic assessment of old structures, continuous digital workflow becomes essential for modern engineering software, so that modifications are converted in results changes without heavy hand calculations. Several lacks of this "digital continuity" have been identified in the most widely used structural computer programs, many of them related to ULS analysis of structural details.

The authors then tried to figure out what those missing chain links should look like to benefit the best to the engineering community. The main reflexions guiding the definition of a new computer program, dedicated to the ULS analysis of steel connexions, have been exposed. Considering the different aspects, from user interface to numerical methods and the global software architecture,