ECCOMAS Congress 2016
VII European Congress on Computational Methods in Applied Sciences and Engineering
M. Papadrakakis, V. Papadopoulos, G. Stefanou, V. Plevris (eds.)
Crete Island, Greece, 5–10 June 2016

INVESTIGATIONS ON AN INNOVATIVE METAL-THERMOPLASTIC COMPOSITE ASSEMBLY

Simon Paroissien[1], Patrick Rozycki[2], Thierry Renault[3]

[1] Direction de l'Expertise et de la Recherche,
IRT Jules Verne
Chemin du Chaffault, 44340 Bouguenais
E-mail: simon.paroissien@irt-jules-verne.fr

[2] Institut de Recherche en Génie Civil et Mécanique (GeM) – UMR CNRS 6183,
Ecole Centrale de Nantes
1 rue de la Noë, 44300 Nantes Cedex 3
E-mail: patrick.rozycki@ec-nantes.fr

[3] Faurecia
2 rue Hennape, 92000 Nanterre
E-mail: thierry.renault@faurecia.fr

Keywords: Multimaterial, FEM, CMT-Pins, Assembly, analytical

Abstract. *Multimaterial assembly is one of the main answers of the car industry to weight reduction issues. As their introduction in the automotive industry is still recent, many multi-material solutions have not been tested in crash, endurance and specific loading. Pre dimensioning tools and numerical models are still very challenging. The present study aims at addressing some of these limitations. From the industrial issue of a seat back rest, an innovative multimaterial assembly based the CMT pin technology has been designed. A first model based on multilayer element is proposed to represent the global behavior of the assembly. Then, in order to investigate the multiple mechanisms (metal to composite direct adhesion, pin-composite interaction…) that rule the assembly, the present article focused on a simple unidirectional piece: a double lap shear specimen. Some studies can be found on this kind of specimen but for different materials and process. This geometry allows us to perform traction test in which both local and global data could be post treated. Several parameters have been tested to evaluate the influence of pins' arrangement on the model behavior. From this experimental campaign, a numerical model with LS Dyna software and an analytical pre dimensioning tool are then proposed and compared. These first results encourage us to continue this study with future works to find the ideal texturing answering conception specifications.*

1 INTRODUCTION

1.1 Context

During the past decades composites materials have taken more and more place in the transport industry. This is particularly true for the small series, where the price of the material have less impact on the production costs than in the large ones. One can take for example the evolution of the percentage of structural composite in the total weight of a plane in airbus production which goes from nearly 5% in the A300 (1972) to more than 50% for the more recent A350-900 XWB (2013). However, if composites are now common in this industry, their introduction in the large series of the automotive field is still very challenging. CO_2 emission standards required by various countries (Europe, USA, but also China) are forcing this sector to consider cutting edge solutions in weight reduction. The introduction of composite in structural applications is one of the main tracks, but this raises other issues like the way of joining the new composite parts with others that, for economic or structural reasons, will remain in metal. The joining method is a key choice so that the assembly can assure a safe load transfer while meeting the drastic specifications of the automotive industry.

1.2 State of the art

The two most common ways to assemble structural application in car industry are mechanical fastening and bonding. Both of them get their own advantages and limitations. Bolting benefits of a long return on experience and is a fast, cheap process but not the more efficient considering the additional weight, stress concentration and the weakening of the CFRP during the drilling process [1, 2, 5 and 13]. On the other hand bonding allowed the preservation of both substrates [7] but its efficiency decreased in extreme conditions (extremes temperature, high rate of humidity) and still need an important overlap surface. Hybrid solutions or the use of safety rivets can remedy to those limitations but, again those actions increase the weight of the structure. Apart from those industrial technologies, some other joining methods can be found in the literature. In [4] an arrow shape is laser cut and bent in a steel sheet. Those arrows are then inserted inside the composite. This study shows good results for 0° and 30° loading under various strain rates. The Surfi-Sculpt™ process, patented by The Welding Institute aims at sculpting a metal surface thanks to an electron beam. Once treated, the metal can then be assembled to a composite (Comeld™ process). Various papers explore this technology and its simulation [11 and 12]. Among the inconvenient of Comeld, the weak peeling strength, and the influence of the process on fiber orientation, and the resin distribution can be cited. The firm Alstom has also patented an assembly process between a metallic and a non-metallic part. Drops of solder formed by an arc welding process are placed, on the surface of the metallic part. Once cooled each of this droplets forms an anchoring point to the composite.

Working on the same principle, the Austrian society Fronius filled a patent (Figure 1) on a process named Cold Metal Transfer (CMT) allowing the welding of small pins on a metallic surface. This technology offers a better control of the shape of the pin formed that way than in the process mentioned above [8]. Experimental investigations have been conducted on CMT process in [10] showing promising results on thermoset CFRP – Steel assembly. The performances are then compared to those of the arrow shape in [9] showing comparable results in terms of maximum transferable joint forces and energy.

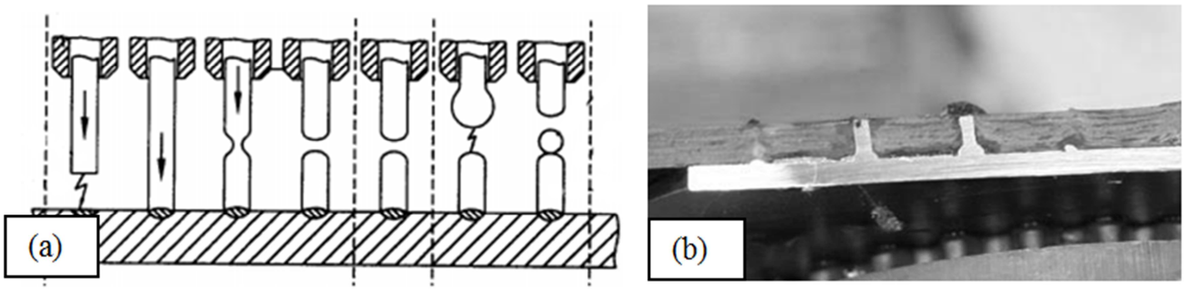

Figure 1: (a)Fronius‘ patent (b) anchoring of CMT in pins as in thermoplastic composite as developed in LIMECO project

This study focus on the evaluation and simulation of a multimaterial structure, joined with a CMT pins interface, more precisely on the case of a front seat back rest.

2 TRIANGLE CASE OF STUDY

2.1 Methodology

A thermoplastic composite triangle is assembled to the head of a steel articulation mechanism thanks to a metallic disk textured by the CMT pin technology (Figure 2). The CMT pins penetrate into the composite, locking the assembly in the transverse direction. A polyamide resin is added to avoid sliding between the two parts. Finally a second metallic triangle is welded to the articulation so that a moment could be transferred.

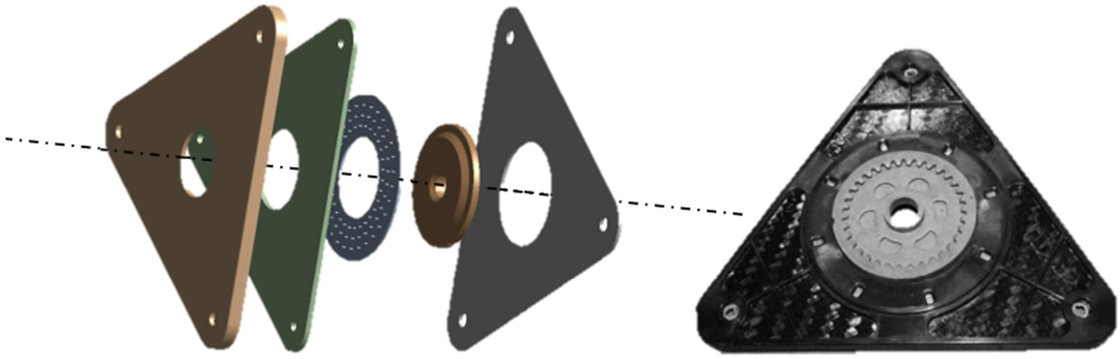

Figure 2: Assembly’s geometry

This assembly aims at transferring a torsional moment, along the rotational axis. The interface is especially complex and nonlinear, a fine representation of the local behavior of the structure would be extremely penalizing for calculation time. For this reason, it has been decided to search for the simplest possible method that conserves good global results. A first hypothesis has been to infer that the stiffness of the assembly is mainly due to interface’s texturing so the other parts of the structure can be considered perfectly bonded. The goal here is to put forward the geometrical simplifications that will reduce calculation costs. The choice has been made to represent the bonded structure with multilayer element. Those elements are an extension of the monolayer shell theory of Reissner. This type of element has the advantages to integrate different materials along its thickness (one material law per integration point), so with this element, one can represent a superposition of different parts inside a single element.

2.2 The multilayer model

The principle is to project on a plane surface the geometric profiles of the different pieces of the assembly. Thanks to multilayer elements, the section of each portion of the piece is then reconstructed. Here the assembly will be represented with two parts: the first part will represent the overmolding resin + composite, the other the metallic part. As the thickness of the shell is important regarding to the mesh size, several integration points per layer have been implemented.

The contribution of the interface is represented with a nonlinear torsional spring. The comportment law of this spring is composed of a torsional stiffness, a nonlinear moment-angle curve and a moment of failure. At first, these parameters are set up from experimental data; future works will propose an analytical model allowing identifying them.

To validate the model, numerical results have been compared to experimental ones from an experimental campaign. Specimens are fixed on a test bed with the following boundaries conditions: a constant low rotational speed is imposed to the steel triangle while composite triangle rotations are locked. This triangle remains free to translate along the rotation axis (Figure 3).

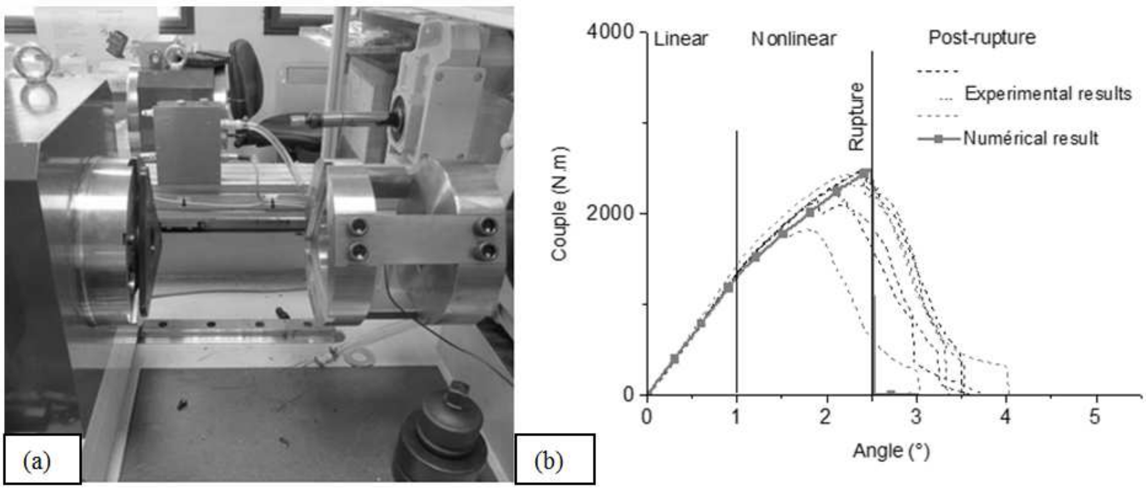

Figure 3: (a) Experimental set up, (b) Comparison between experimental and numerical results

Thanks to spring parameters set up, numerical results show good accordance with experimental ones.

2.3 Limitation of the model

A method giving spring behavior from the pins disposition is still necessary. To define this method, investigations have been made on another kind of geometry. Indeed, the assembly’s geometry leads to complex solicitations and experimentally the metal composite interface is difficult to instrument correctly. Moreover the finite element model requires an important number of elements to model both pin scale and assembly’s scale with enough precision. In order to investigate the multiple mechanisms (metal to composite direct adhesion, pin-composite interaction…) that rule the assembly, the present article focused on a simpler unidirectional piece described in the following part. Two methods of simulation, one using the Finite Element Method (FEM) and the other using an analytical algorithm, are then proposed and compared.

3 EXPERIMENTAL CAMPAIGN

3.1 The geometry tested

The geometry of the specimen should enable to isolate a single pin's behaviour, and the influence of the number of pins and their disposition on the specimen's behaviour. Moreover instrumentation should be simple and give access to the local comportment of the interface. In papers and more precisely in [13] one can find the use of a double lap shear speciemen (DLS) to characterize an interface working under a shear loading. This geometry has the advantage to limit the peeling phenomenon due to the lack of planeity in the loading. The geometry described in Figure 4, is composed of two glass fiber shells, termostamped on both sides of a metalic plate. The recovery zone is textured with the CMT pin technology. Outside of the interface area, a metal wedge is inserted between the two composites to maintain the thickness of the composite part of the specimen, as described in the figure below. Pin's height is set up at 3 mm so that the head of each pin flushes with the top of composite layer.

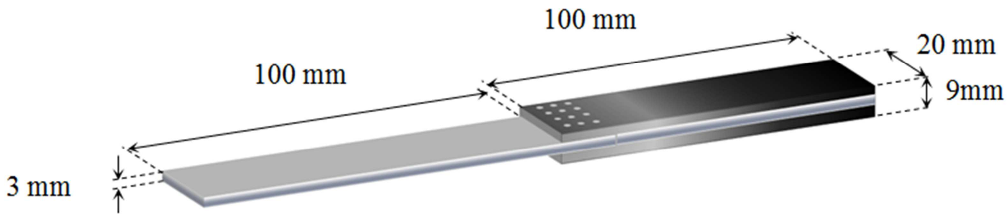

Figure 4: Specimen geometry

Three pin's dispositions have been tested: 6, 3, and 1 raw(s) of 4 pins. Only cylindrical shaped pins of 1mm diameter are welded. To ensure the repeatability five tests have been realized on each configuration.

3.2 Experimental set up

The tensile tests were carried out on a quasi-static testing machine INSTRON 5584 at a 1mm/min speed (Figure 5.a.). Resultant efforts were measured by a force cell. A measure of the relative axial displacement of the two parts is performed by digital image correlation (DIC). Two high-definition recording cameras filmed the test: one record the edge of the DLS (Figure 5.b.), the other one of the two faces (Figure 5.c.).

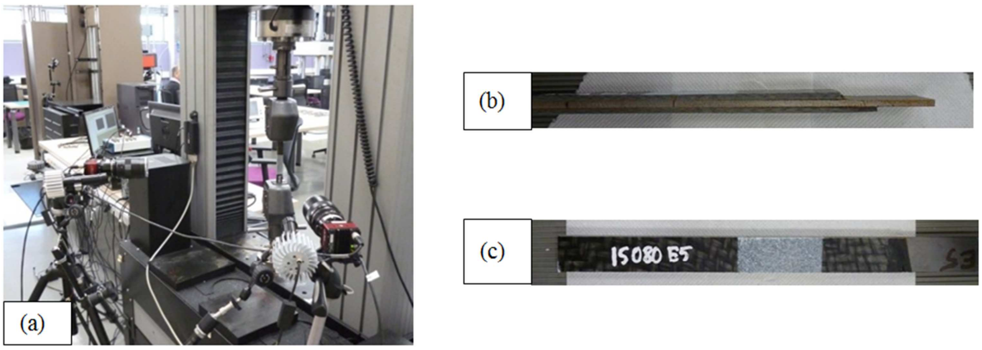

Figure 5: (a) Experimental set up, (b) DLS (edge), (c) DLS (face)

3.3 Results and discussion

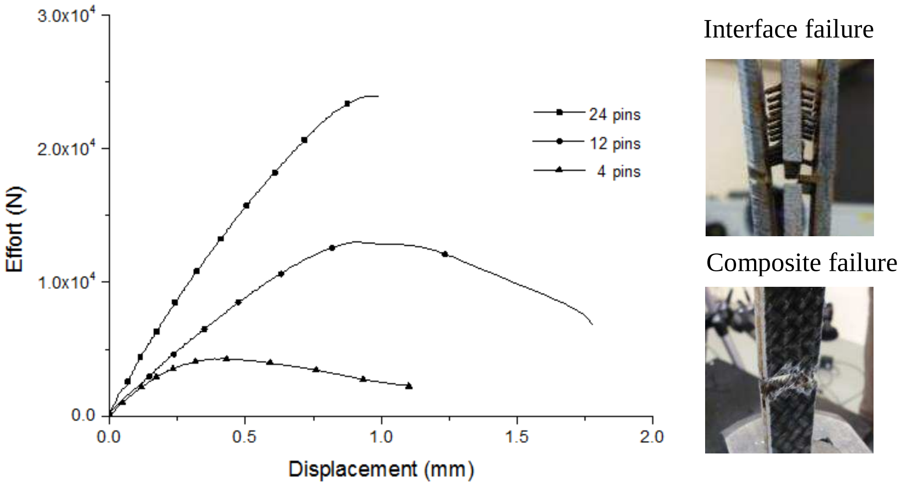

Figure 6: Effort – Displacement graph.

The experimental results are given in Figure 6, where the effort displacement law is given for the three tested configurations. For readability only one representative curve of each configuration has been displayed. Under a tensile loading, the DLS shows a linear elastic behavior, followed by a plastic softening. This softening leads to two kinds of failure, depending on fiber's orientation and effort's level. On the one hand (and in most situations) a noisy failure of the can be heard, followed by a drop of effort level. Moreover almost none normal to the interface displacement is measured by the DIC before failure. This is characteristic of a metal to composite direct adhesion [3] involved by thermo stamping. On the other hand, if fiber's orientation has been too disturbed and if the efforts are important enough, a failure of the pure composite part of the DLS can be observed.

An important dispersion can be observed between the five samples of the 24 pins configuration. This is due to resin creep during the thermo-stamping operation. In the two more flexible cases, this leads to a composite kind of failure. In the following parts of the paper, for this configuration, the simulation results will be compared to the stiffer experimental result.

4 NUMERICAL MODEL

4.1 Geometry and meshing

The geometry and meshing are realized with Ansys Workbench v15 software. The explicit solver used is LS Dyna R7.1. For this model, only the pure composite and the interface part of our DLS are modeled. The steel part is considered rigid. Two planes of symmetry allow simulating only a quarter of the specimen (Figure 7). The model is meshed with one integration point hexahedral elements. Mesh size goes from 0.1 mm on the pin to 0.3 mm on the rest of the interface.

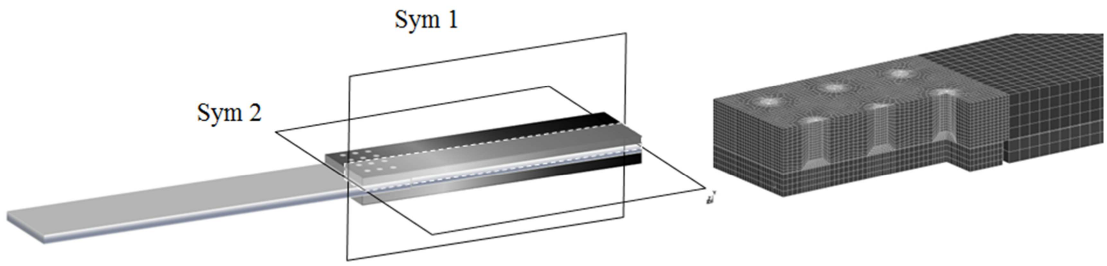

Figure 7: Geometry and meshing

To avoid initial penetrations, a small gap of 0.025mm is left between the pin and the surrounding composite. A friction coefficient of 0.05 is added. Symmetry conditions are applied to the corresponding faces. The end of composite part is considered fixed and a constant speed is imposed the other end of the DLS, on the steel section.

Concerning pins, two parts can be considered: the welding cone at pin's base and pin's stem/head. The pin, made of G_3Si_1, is characterized at solid state. A traction test performed on the welding wire shows important differences from the values given by the provider. It has been settled that the welding cone will be modeled according to providers' data and the wire, which didn't melt during the welding process, with the property obtained from the traction test. The material law n°261 of LS Dyna developed by Pinho & al in [6] is used. This material law allows the user to implement the in-plane shear behavior with a defined curve. At the neighborhood of each pin, the material law is replaced by an orthotropic linear law so that the local artificial constraints don't damage the composite. This hypothesis is justified by observations on the post rupture aspect of the interface. The direct adhesion is modeled with layer of cohesive elements. No data were given to implement the comportment law of those elements. So the general law n°138 has been chosen and its parameters were calibrated on experimental results.

The comparison between numerical and experimental results is given in Figure 8.

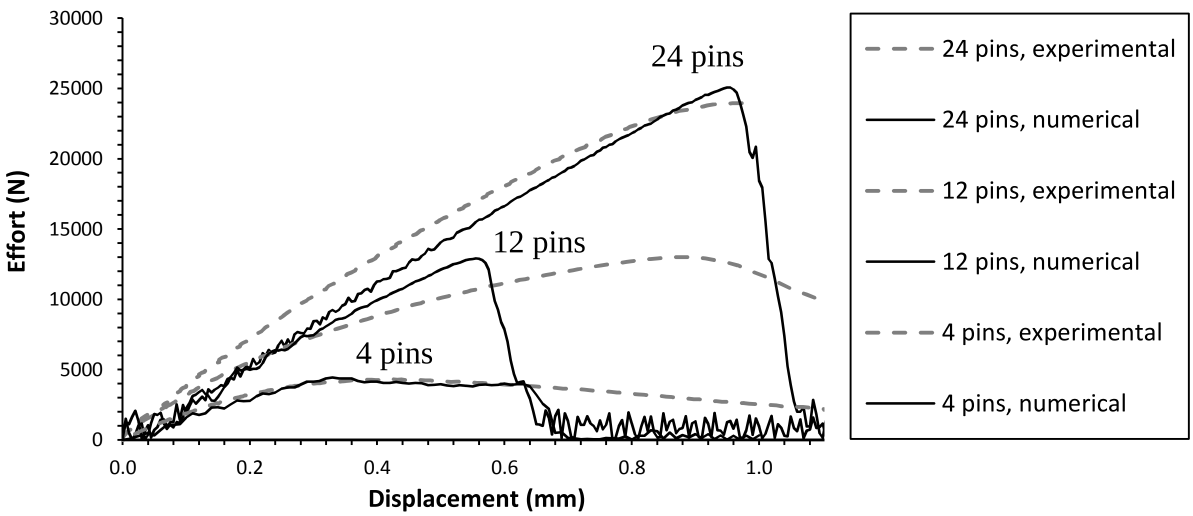

Figure 8: Comparison of the numerical and experimental results

Numerical results show a good accordance with experimental ones. Maximal efforts especially, are obtained with a 5% precision.

To link this result to the triangle issue, the local behavior of one pin in a composite environment has been studied. By refining the post processing it has been noticed, that a single force-displacement law, linking the effort at the base of the pin to its head's displacement, can be isolate for all the pins, in all the studied configurations (Figure 10). So an assumption has been made: the behavior of a pin can be found from a numerical model at pin' scale. From this hypothesis, an analytical model giving the DLS traction law has been implemented.

4.2 Scale of the pin

Inside each DLS it has been noticed that a Representative Volume Element (RVE) can be isolated, (Figure 9). Thanks to geometrical and loading symmetries according to the plan (y,z), only half of the RVE has been represented.

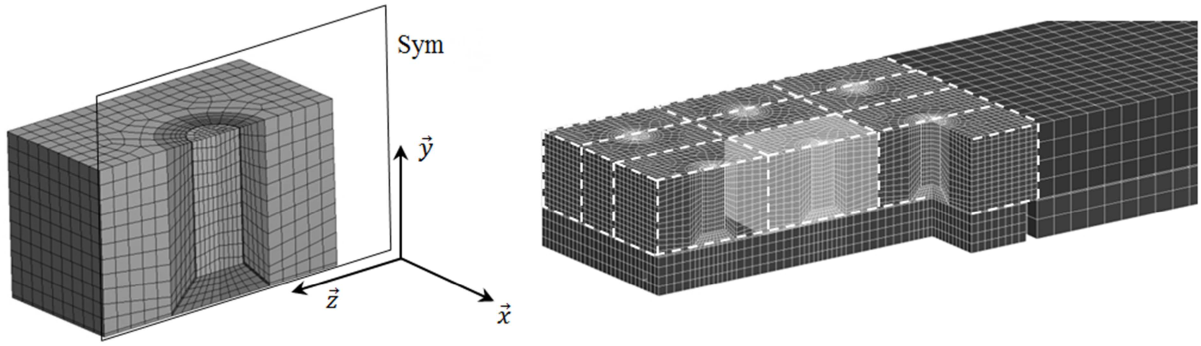

Figure 9: RVE Geometry

The same contact, material laws, and type of elements are used as in the DLS model. The bottom face of the geometry is fixed, and symmetry conditions are applied on the faces normal to x-direction. A constant speed, in the z direction is imposed on both faces normal to z. The comparison between RVE's behavior and each pins row from each configuration is given in Figure 10 (In the legend, "XX Pins RY" stands for "XX pins configuration, row number Y"):

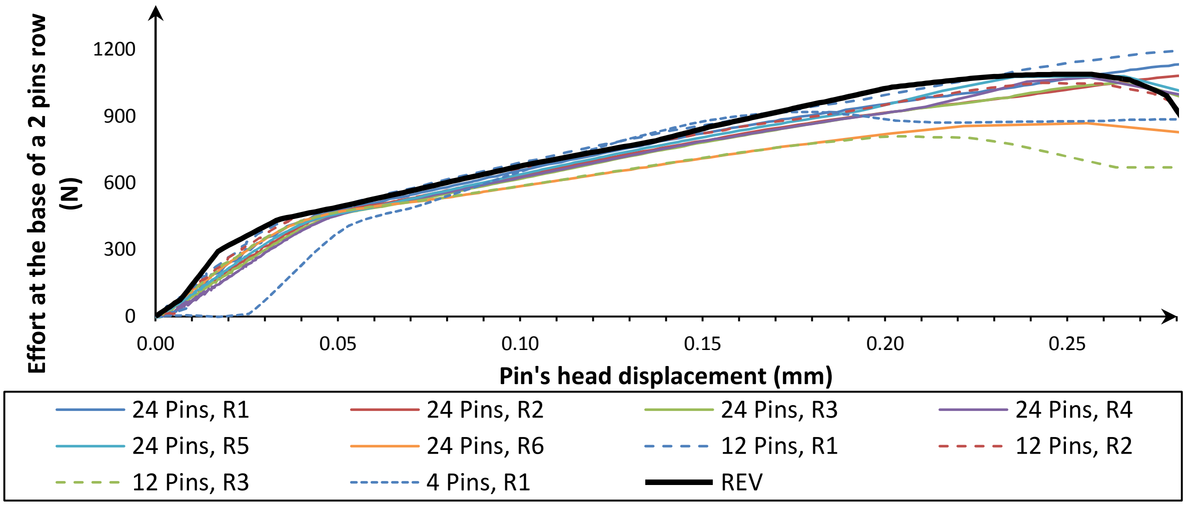

Figure 10: Pins local effort displacement law

For each pin's row, for each configuration (4, 12 and 24 pins), the Figure 10 gives the law linking pin's head displacement to the effort transferred through pin row to the metallic plate. This graph shows that the behavior of the pin inside the RVE is representative of the behavior of any pin inside the DLS. A bilinear flexion law is extracted from this graph. The rupture criterion which depends of the cohesive layer (i-e: metal to composite adhesion) is not associated to this law.

Now that the behavior of one RVE has been isolated, an analytical algorithm giving interface behavior is proposed in the next part.

5 ANALYTICAL MODEL

5.1 Presentation of the problem

For the next part, the following diagram (Figure 11: The CMT Pin link of the problem will be used:

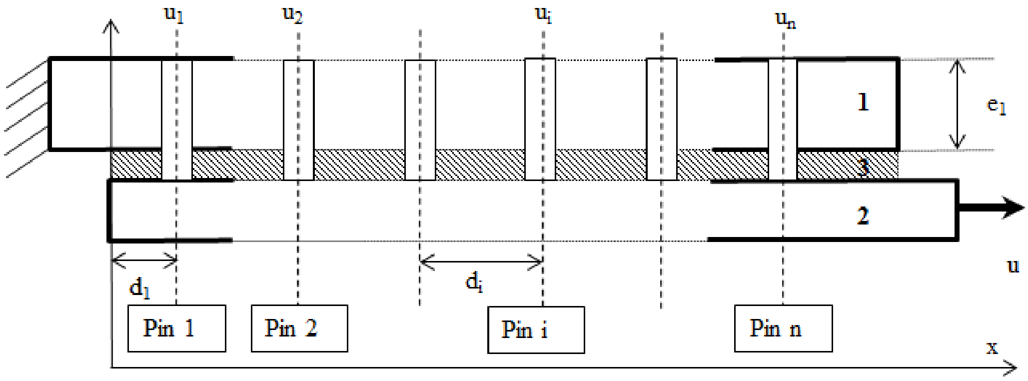

Figure 11: The CMT Pin link

5.2 Equivalent model

The model is made according to the following hypothesis:

- In comparison to other materials, the steel plate is considered rigid.
- The thermoplastic composite is supposed to be perfectly oriented, with the x direction as one of the orthotropic directions.
- The displacement of a section of the composite, surrounding a pin is equal to the displacement of the head of this pin, u_i.

Pin's behavior and cohesive layer on a RVE can be represented with a single multilinear, force displacement law $F_{RVE,i}$. The composite transmits efforts from one pin to another, with a stiffness noted K_{dyn}.

Once inserted in the global geometry, the system becomes:

$$
\begin{aligned}
&F = \sum_{i<n} F_{RVE,i}(u - u_i) \\
&u_1 = \frac{F}{K_{dyn,1}} \\
&u_2 = u_1 + \frac{K_{dyn,1}.u_1 - F_{RVE,1}(u - u_1)}{K_{dyn,2}} \\
&\forall i \in [2, n-1], u_{i+1} = u_i + \frac{K_{dyn,i}.(u_i - u_{i-1}) - F_{RVE,i-1}(u - u_i)}{K_{dyn,i+1}}
\end{aligned}
\qquad (1)
$$

5.3 Different configurations

The algorithm implemented in Matlab gives the following results (Figure 12) for the different configurations experimentally tested.

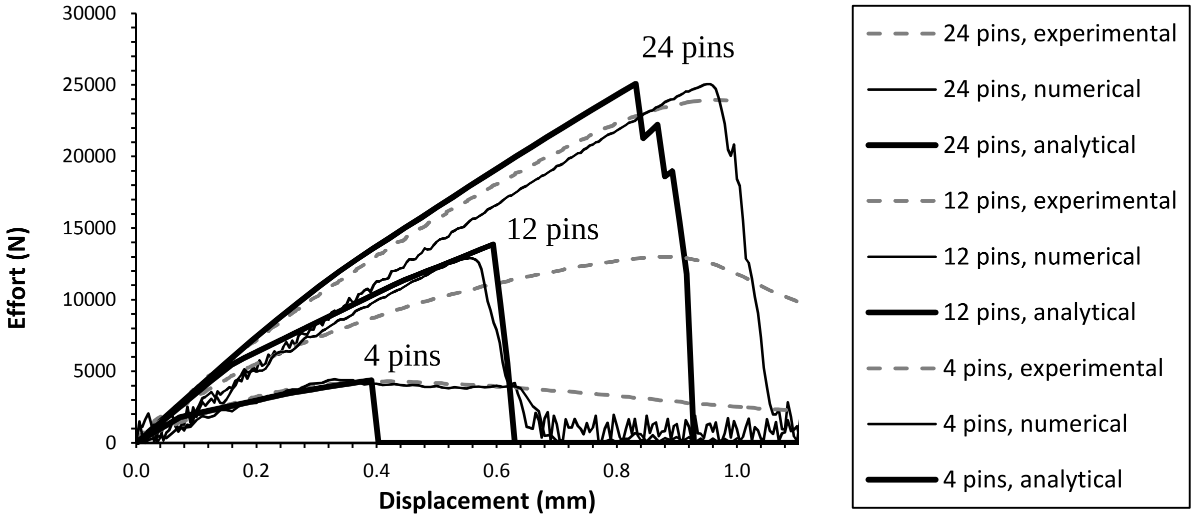

Figure 12: Comparison of analytical and experimental results for 24, 12 and 4 pins configurations

Again, the analytical results are close to experimental ones. The difference of stiffness in the 12 pins case can be due to an imperfection in fiber's orientation in the experimental test.

In this algorithm, the behavior part out of section is calculated analytically because of the simple geometry of the DLS. For a more complex case the interface can be simulated with a nonlinear spring. As pins are not modeled anymore, meshing the interface becomes much simpler and shell elements can be used to model the two parts. In the case of the 24 pin interface, the geometry used in the numerical model becomes (Figure 13):

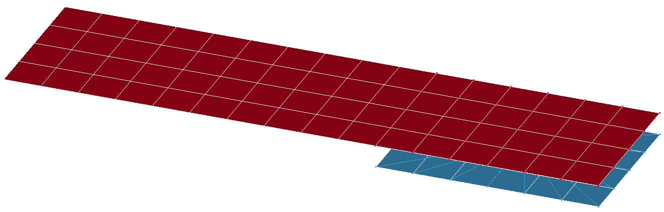

Figure 13: Simplified geometry using non linear spring as an interface

A comparison of experimental, numerical and analytical results is given in the following graph (Figure 14). In the legend, the "Ei" stands for "experimental test number i":

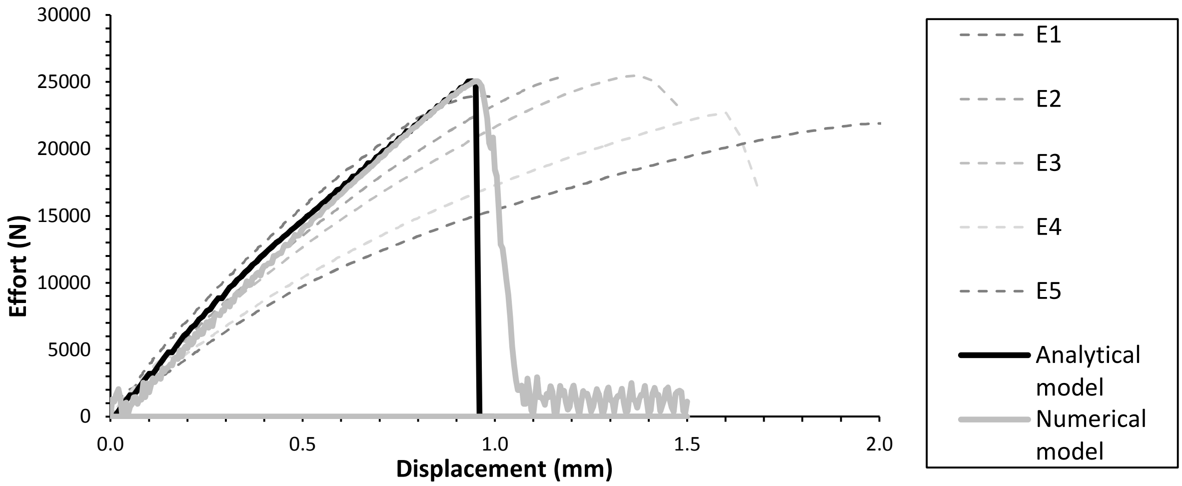

Figure 14: 24 pins configuration, experimental, numerical and analytical results

The real gain with the analytical model is on calculation time. For the numerical model the simulation was launch on a cluster with 8 nodes with a 4 x AMD Shanghaï 2380 2.5 GHz Quadcore processor and last more than 9 hours, whereas the analytical algorithm is solved quasi instantly on a single processor (i7-3630QM CPU, 2.4 GHz). This is especially convenient for the pre dimensioning or optimization phases.

6 CONCLUSION

The CMT pin technology is used to produce a multimaterial assembly between a steel part and a PA6, glass fiber thermoplastic composite. As a fine representation of the interface geometry would be complicate and expensive, a multilayer model combined with a nonlinear rotational spring is proposed to simulate the global behavior of the assembly. To implement spring parameters and find a method to study the local behavior of this assembly, double lap specimen are designed and tested on a quasi-static traction machine. The results show the important influence of the thermo stamping phase of the production on the behavior of the final specimen.

A numerical model is then proposed for these DLS, showing comparable results with experimental ones. Thanks to this model, the local behavior of a RVE, a pin stamped in thermoplastic composite has been extract. The flexural law obtained this way is then inserted in an analytical algorithm, which calculates the comportment law of the interface under shear loading. The whole interface can be then simulated with a simple nonlinear spring (whose parameters have been implemented from the previous algorithm) and inserted in another finite element model. The analytical algorithm works under any shear loading, and must be extended to torsional solicitations, so that it can be used to represent the triangle interface. These developments are still in progress.

This method shows very fast results for a given disposition of pin, shear loaded, and can be an important advantage for the pre dimensioning phases. The numerical model can be used if specific local post processing data are needed. This method represents a powerful tool for the comprehension and the development of the CMT pins assembly technology.

In future works an application to this method to a different texturing can be instigated. This method has also a high potential in optimization problems and can be used to find the ideal pin disposition answering interface specifications for both double lap and triangular specimen. Other kind of loading, as torsional or out of plane loading could be studied with a similar approach.

ACKNOWLEDGEMENT

This work was performed in the framework of the LIMECO project funded by the IRT Jules Verne. The authors wish to thank all the member of this project: Faurecia for leading the project, Compose Tools for designing and realizing the mold of the triangle shaped specimen and finally the Cetim for the torsion tests realized on the triangle geometry.

REFERENCES

[1] Camanho, P. P., & Matthews, F. L. (1997). Stress analysis and strength prediction of mechanically fastened joints in FRP: a review. *Compos Part A: Applied Science and Manufacturing*, 529-547.

[2] Dano, M. L., & al. (2000). Stress and failure analysis of mechanically fastened joints in composite laminates. *Composite Structure*, 287-296.

[3] Grujicic, M., & al. (2008). An overview of the polymer-to-metal direct-adhesion hybrid technologies for load-bearing automotive components. *Journal of materials processing technology I 97*, 363-373.

[4] Heimbs, S., & al. (2013). Failure behaviour of composite T-joints with novel metallic arrow-pin reinforcement. 16-28.

[5] Paulo, J., & al. (2004). Experimental study of drilling glass fiber reinforced plastics (gfrp) manufactured by hand lay-up. *Composites Science and Technology, Vol. 64(2),* 289 – 297.

[6] Pinho, S. T., & al. (2006). Physically based failure models and criteria for laminated fibre-reinforced composites with emphasis on fibre kinking. Part II: FE implementation. *Composites: Part A 37*, 766 777.

[7] Sohier, L., & al. (2011). Etude expérimentale du comportement d'assemblages collés de composites sous. *Comptes Rendus des JNC 17*.

[8] Somoskői, G., & Török, I. (2013). CMT pin –Define the shape of the welded pin through welding parameters. *Production Processes and Systems, vol. 6.No. 1.*, 47-56.

[9] Ucsnik, S., & al. (2014). Composite to composite joint with lightweight metal reinforcement for enhanced damage tolerance. *ECCM16 - 16TH*.

[10] Ucsnik, S., & al. (2010). Experimental investigation of a novel hybrid metal-composite joining technology. *Composites*, 369-374.

[11] Xichang , W. (2011). Study of electron beam Surfi-sculpt during composite materials joining. *Rare metal materials and Engineering, vol. 40, suppl. 4,*, 292-296.

[12] Zhang, H. (2012). Study on the strength prediction model of Comeld composites joints. *Composites: Part B43*, 3310-3317.

[13] Zitoune, R., & Collombet, F. (2007). Numerical prediction of the thrust force responsible of delamination during the drilling of the long-fibre composite structures. *Composites Part A: Applied Science and Manufacturing, Vol. 38(3),*, 858-866.