FACULDADE DE CIÊNCIAS E TECNOLOGIA
UNIVERSIDADE DE COIMBRA
DEPARTAMENTO DE ENGENHARIA CIVIL

Civil Engineering Department

Faculty of Science and Technology

Coimbra University

STSM

Scientific Report – COST TU0904

	Host:	Guest:
Name:	Dr. Milan Veljkovic	João Ribeiro
Institution:	Lulea University of Technology	Coimbra University of Technology

JUNE 2012

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

PEOPLE

From Coimbra University:

DR. SANTIAGO, Aldina	Coordinator for COMPFIRE and IMPACTFIRE.
Ph.D. LOPES, Fernanda	Ph.D. Student and researcher within COMPFIRE project performing experimental tests.
Res. RIBEIRO, João	Researcher within IMPACTFIRE project performing finite element analysis and grant holder for this short-mission.

From Lulea University:

DR. VELJKOVIC, Milan	Coordinator for Lulea University Steel Research Department and host for this short-mission.
Ph.D. HEISTERMANN, Tim	Ph.D. Student and researcher within COMPFIRE project performing finite element analysis for component tests.
Ph.D. IQBAL, Naveed	Ph.D. Student and researcher within COMPFIRE project performing finite element analysis for frame tests.

DEFINITIONS

Shear Lock – Inability of element type to deform in bending (curvature deformation). Element becomes overly stiff under bending moment potencially producing wrong displacements, false stresses and spurious natural frequencies

Hourglass – Is represented by a zero-energy mode at the integration point, producing deformation but no strain is generated due to the deformation. This phenomena may propagate in coarse mesh producing false results.

ACKNOWLEGEMENT

The grant holder would like to present his personal thanks to Dr. Aldina Santiago for the proposal of such interesting and personal enrichment abroad experience and to Dr. Milan Veljkovic for receiving me in such short notice and for all his preoccupation, care and shared experience.

I would also like to address the PhD students at Lulea Steel Structure Research Group for all the time, kindness and knowledge sharing.

Finally I'd like to gratefully acknowledge the support of COST – European Cooperation in Science and Technology under grant TU0904: http://www.cost.eu/domains_actions/tud/Actions/TU0904

João Ribeiro

1. PRESENTATION

Grant holder João Ribeiro, 27 years of age, born in Coimbra, Portugal, is currently a researcher for the project IMPACTFIRE which is studying the behavior of two connection geometries – ‘extended end plate’ and ‘reverse channel’ – subjected to impact loading. The main target is to understand the behavior and capacity of such connections to withstand fast applied loading due to accidental action and verify if current design guide lines need to contemplate further checks to prevent progressive failure.

Experimental tests will be conducted at the University of Coimbra and finite element analysis will be calibrated against these tests. Parameterized analysis shall be conducted proven that the model can depict connection behavior. It is worth mentioning that the t-stub component will be tested subjected to impact loading and also to impact loading with the specimen at high temperatures.

2. INTRODUCTION

This COST STSM had three main objectives:

i. To establish contact with the partners involved on research projects related with fire response;
ii. To discuss problems related to the numerical modeling with software ABAQUS, taking into consideration the high temperature problems;
iii. To enlarge the research cooperation between the University of Coimbra and the Technical University of Lulea.

In order to find common ground, the works during this short-mission are related to the project COMPFIRE which is studying the reverse channel connection type in order to assess its behavior and ductility limits to provide consistent information for design guide-lines and avoid premature progressive structural collapse when subjected to fire hazard. Therefore, a series of experimental tests are being conducted at the University of Coimbra Steel Structure Laboratory providing base results for finite element analysis which are being performed by PhD students at

Lulea University, namely *Tim Heistermann*, who is conducting reverse channel components FEA, and *Naveed Iqbal* who will be performing the frame tests FEA.

3. TEST DESCRIPTION

Finite element analysis conducted during this short-mission is related to the reverse channel section 1-a), highlighted in Figure 1. Both tension and compression tests have been conducted at ambient and high temperature for thicknesses of 8, 10 and 12 mm for the front plate.

The work focused mostly in the simulation of 'test 5' – reverse channel with plate t = 10mm – for compression loading due to previous FEA analysis mismatch to the test behavior. Test specimen can be seen on Figure 2 and test rig dimensions in Figure 3.

Test results at ambient temperature for both compression and tension are shown in Figure 4 showing a rather ductile behavior in tension, which is due to different load carrying behavior, as tension tests are clearly governed by the front plate bending, and for the compression tests it gives the impression that the load is being carried by plate and weld in pure shear.

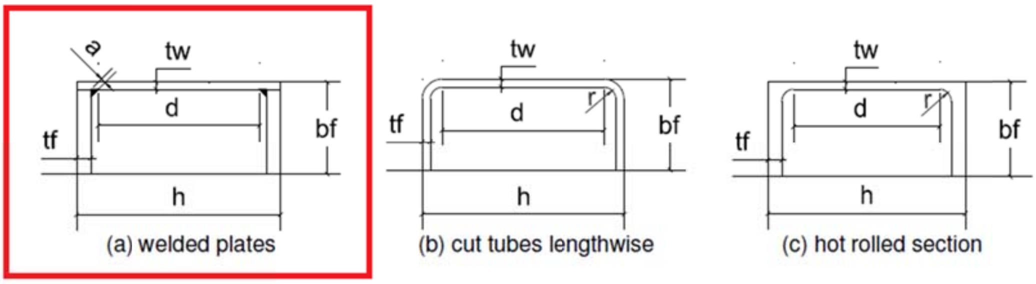
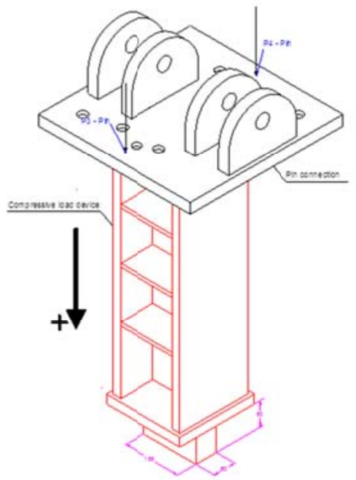


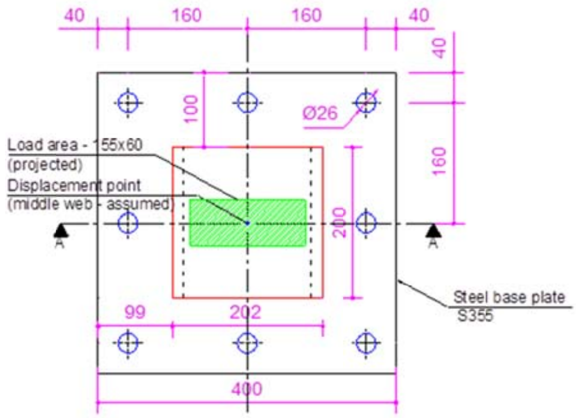
Figure 1 – Different reverse channel sections – COMPFIRE.

a) test 2 – plate 10mm in tension b) test 5 – plate 10mm in compression

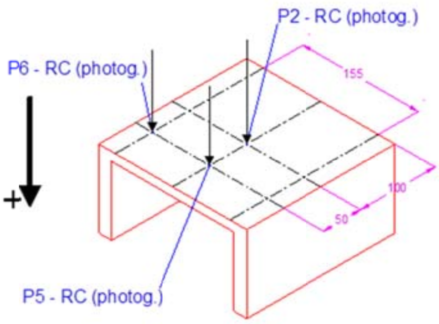
Figure 2 – Specimens after testing – ambient temperature.

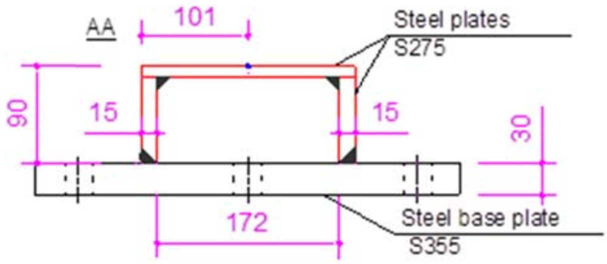


a) Loading device



b) Top view





c) Measuring points

d) Elevation view

Figure 3 – Compression tests loading rig and specimen dimensions.

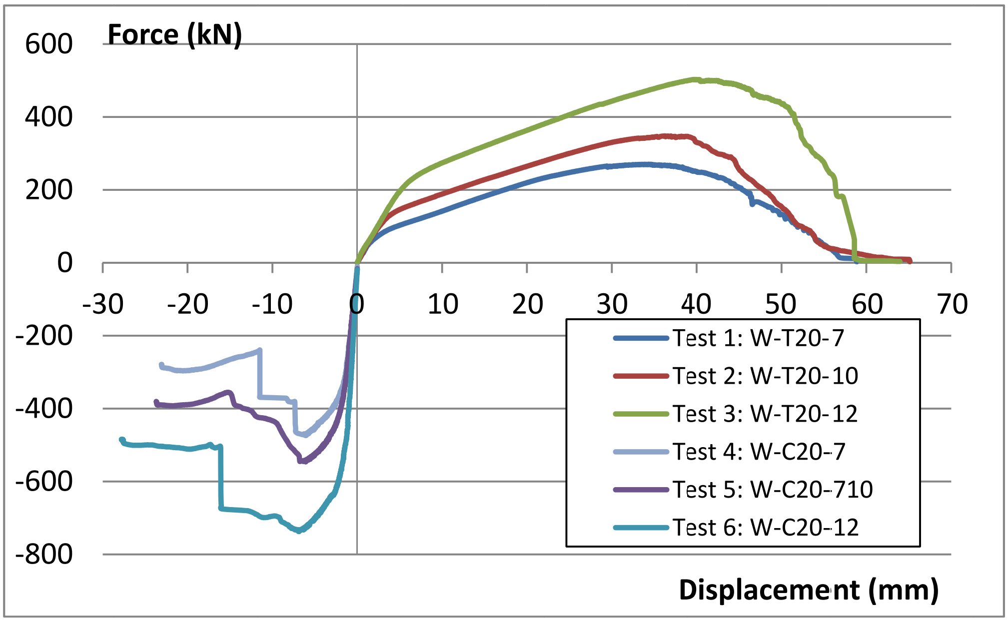


Figure 4 – Force x Displacement curve for the for welded plate component tests conducted at ambient temperature.

4. FINITE ELEMENT ANALYSIS - DESCRIPTION

During the short mission five models were built and its results compared. The evolution from one model to the other followed validation of the model and uncertainties encountered during discussion mostly with colleague *Tim Heistermann*, who is modeling the component test, and as such the models follow some of his assumptions, as well as those stated by colleague *Fernanda Lopes* (researcher at University of Coimbra – performing COMPFIRE experimental tests) in the report **WP2** for '*COMPFIRE - Component Tests*'

Model #1 -	Whole specimen was modeled using the nominal weld referred in WP2 of a = 9mm – leg = 12mm. The weld was considered to be only in the interior of the reverse channel.
Model #2 -	Quarter model of specimen using nominal weld – a = 9mm. Weld assumption as in previous model.
Model #3 -	Quarter model of specimen using a weld leg of 6mm – a = 4 mm. Weld assumption as in previous model.
Model #4 -	Quarter model using weld throat as in the previous model. Weld assumption to be fully penetrating, completely tying the front plate to the flanges of the reverse channel
Model #5 -	Assuming nominal weld throat (a=9mm), complete tie between front plate and flanges, and using EN 1993-1-2 high temperature material parameters provided by *Tim*, a high elevated temperature (550ºC) model was successful run.

Material Models:

Material models are the same as *Tim*'s so that different values for the true stress-true plastic strain relationship wouldn't occur. The material follows the data retrieved from the coupon tests realized in Coimbra - Table 1:

Table 1 – Material mechanical properties at ambient temperature.

Material type	$f_{y,nom}$ MPa	Thickness mm	$f_{y,test}$ MPa	$f_{u,test}$ MPa	E_{test} MPa
Plates	275	8	246.3	421.0	195.0
		10	249.3	379.0	201.8
		12	236.3	373.3	198.9
		15	312.0	527.0	211.6

Finite Element Mesh:

The finite element type used was the C3D8R which accounts for an extra integration node in the center in order to avoid shear-lock. It also uses reduced integration for faster calculation but hourglass behavior should be checked, although the algorithm has control parameters to prevent it. This finite element type as proven to be accurate for bending situation, if discretization of elements through the thickness is above 3 to elements.

The mesh has been assumed to be of 1mm were small elements were need. The finite element mesh is present on Figure 5.

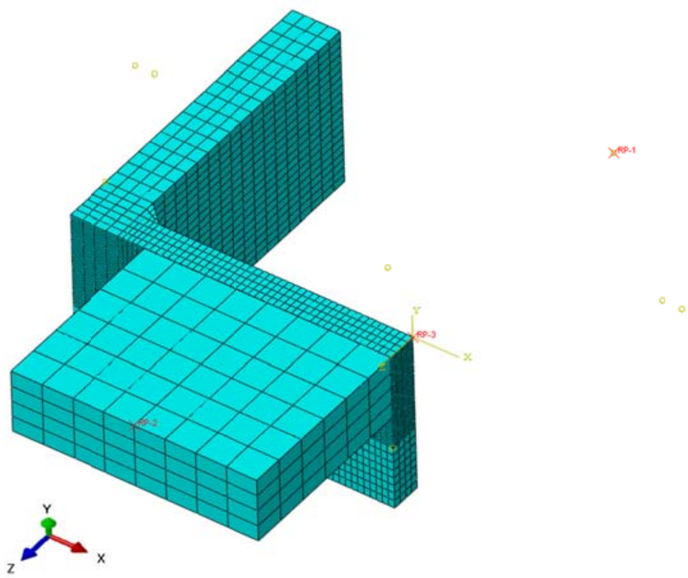


Figure 5 – Mesh size

5. RESULTS

The results were taken from both reference points RF2 and RF3, which can be seen on Figure 6 below. Test measurements were taken from point RF3.

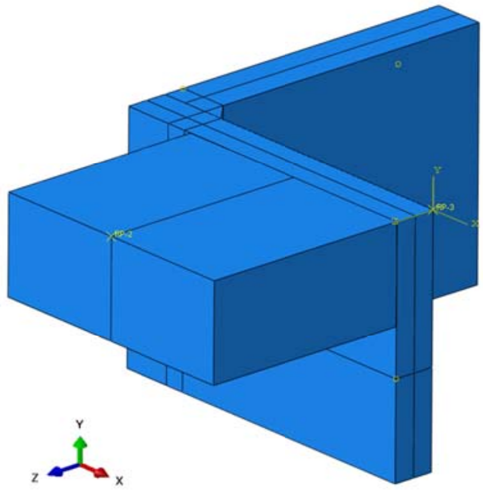
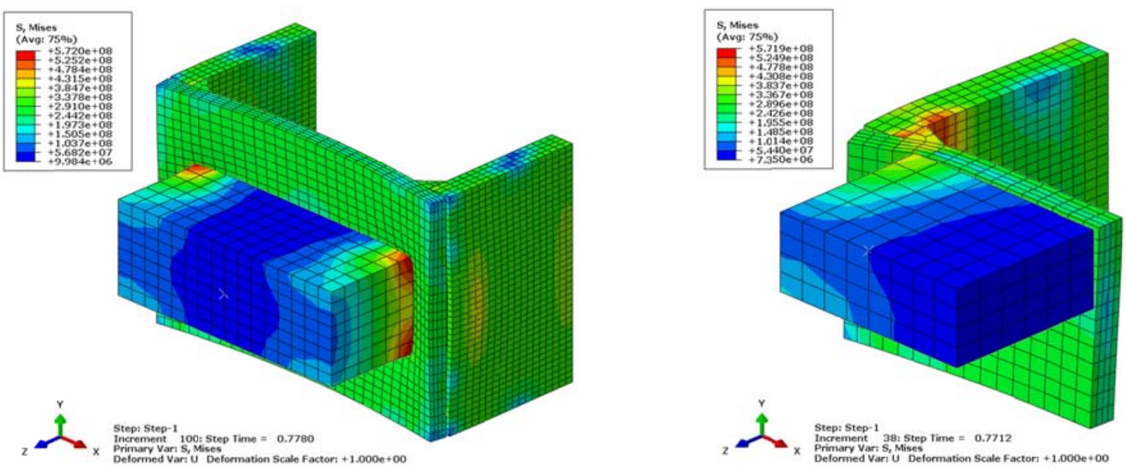


Figure 6 – Measuring points – RF2 & RF3

Figure 7 shows geometry of models #1 & #2 and Figure 8 compares results from Model #1 with Model #2 showing that the quarter part model can perfectly reproduce the results.



a) Model #1 b) Model #2

Figure 7 – Model #1 Vs. Model #2 - Geometry.

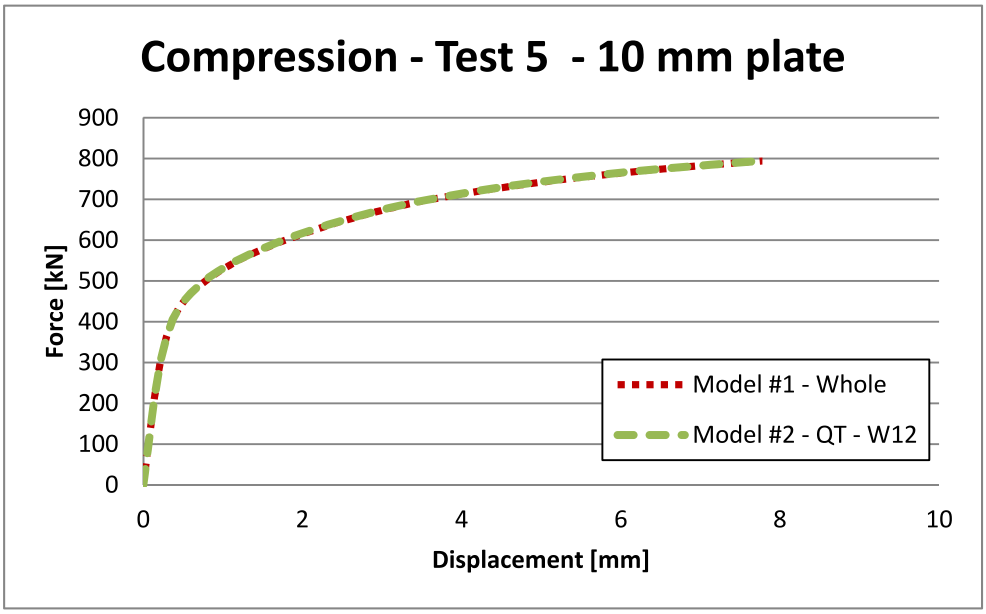


Figure 8 – Model #1 Vs. Model #2

Since the nominal weld throat referred in testing report is of a=9mm, model #2 uses a weld leg of 12mm. Due to lack of agreement with the test results and to verify the influence of the weld in the behavior, the weld was reduced to 6mm (a=4.2mm), and this differences can be seen in the Figure 9 below, showing a reduction of the initial stiffness and also smaller yield force.

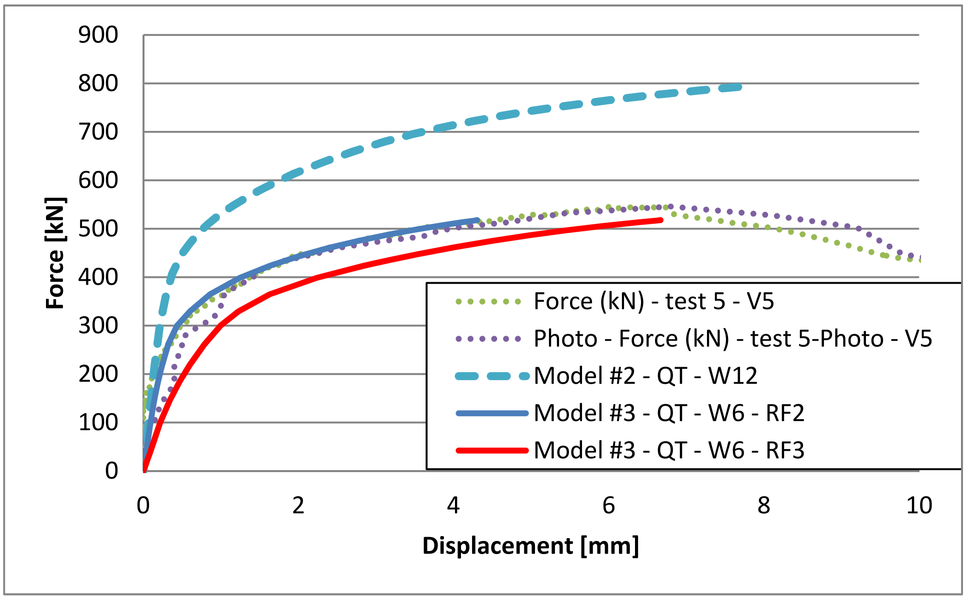
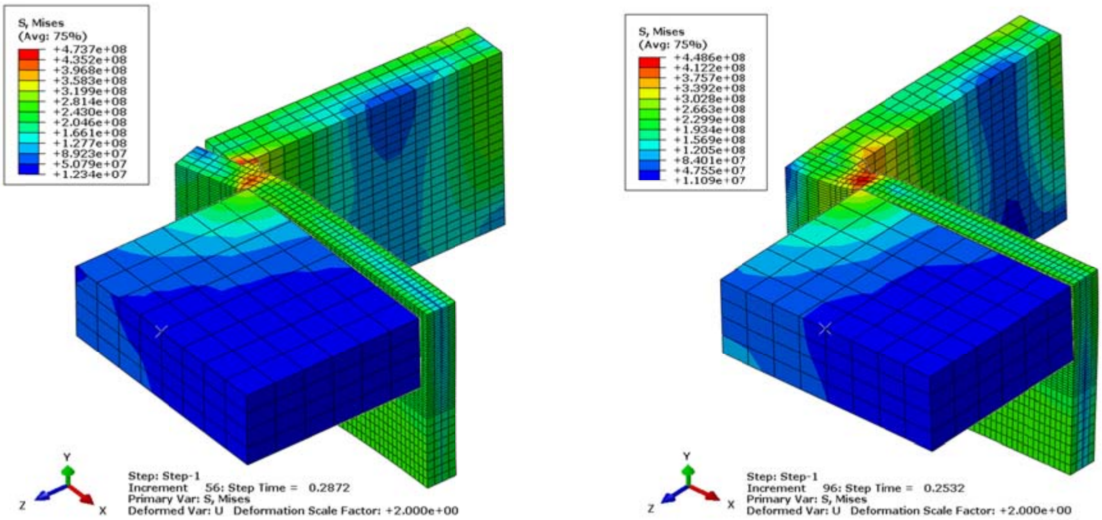


Figure 9 – Model #2 Vs. Model #3 – Weld leg influence

After this, it was noted that the specimens would also have an exterior weld which would prevent the rotation of the front plate, therefore mode #4 reflects the behavior of a perfect tie between the front plate and the flanges of the reverse channel - Figure 10. Result comparison between Model #3, Model #4 and test data is shown below - Figure 11, where we can see an increase of stiffness (as expected) and also a smoother difference between RF2 and RF3 displacement.



a) Model #3 – 'free' weld b) Model #4 – 'fixed' weld

Figure 10 – Weld modeling detail - deformed shape

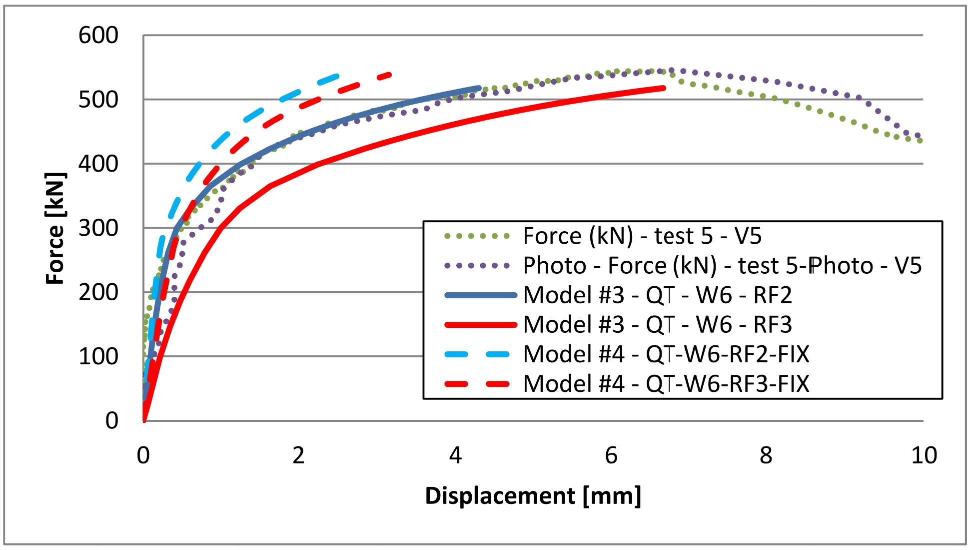


Figure 11 – Model #3 Vs. Model #4 – Weld scheme influence

6. RESULT DISCUSSION

Assuming that the real weld throat is of 4mm (leg=6mm) instead of nominal 9mm, as the material stress-strain relationships are taken from coupon tests for each piece, the model should now be representative of the tests. However there are still some mismatches at several key points - Figure 12, such as, yielding point and failure load. Also, improvements in the models convergence are needed so that it would continue the calculation for increased loading.

There is also some doubt about the test equipment database collector which leads to unexpected initial increase of force without any displacement. Such situation lead the partners to calculate the expected initial stiffness through the elastic modulus and yield stress relation which leads to the results in Figure 13, showing even greater mismatch of FEA models and test results.

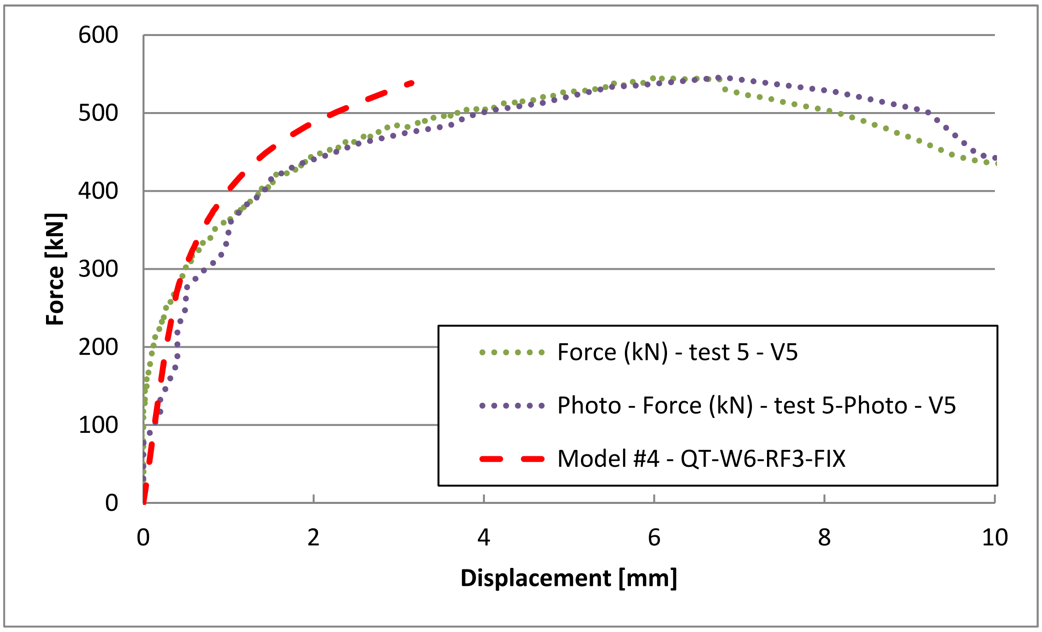


Figure 12 – Model #4 Vs. Test Data

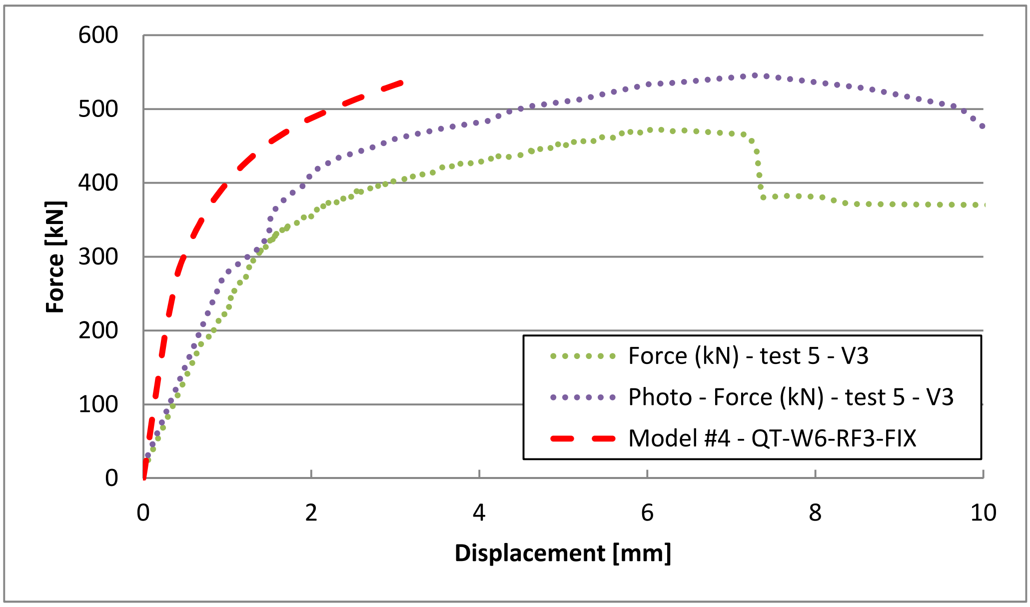


Figure 13 – Model #4 Vs. Test data – correction

7. HIGH-TEMPERATURE MODEL

Results for Model #5 - Figure 14, show that further comprehension of the experimental test is needed. In the model, a temperature pre-defined field of 550 ºC was firstly applied and thermal expansion was allowed, which is why a negative displacement is developed before the loading step begins. Although it does not depict the test, it is of personal interest due to lack of experience in the development of high-temperature FEA with ABAQUS.

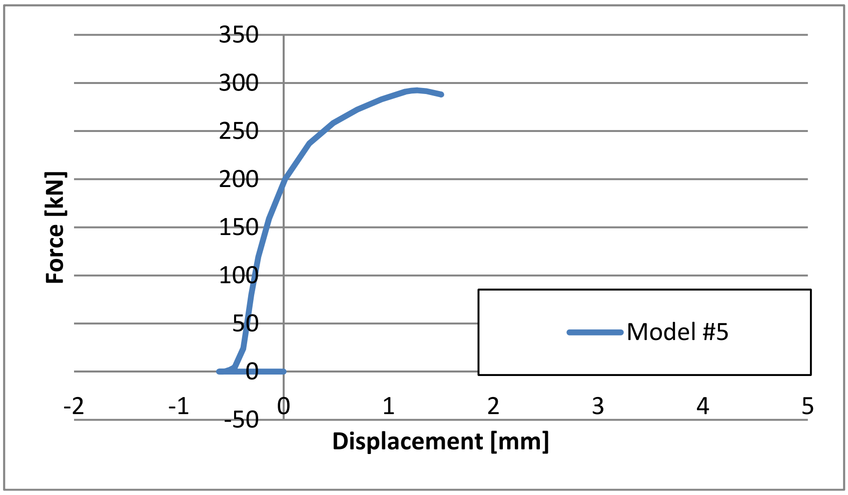
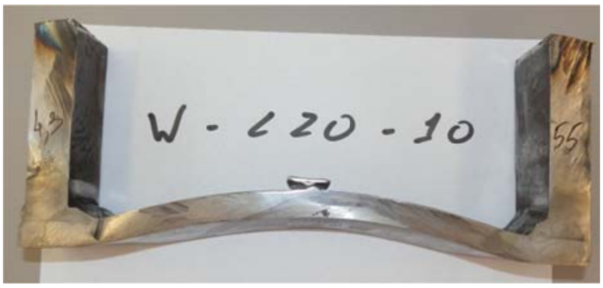


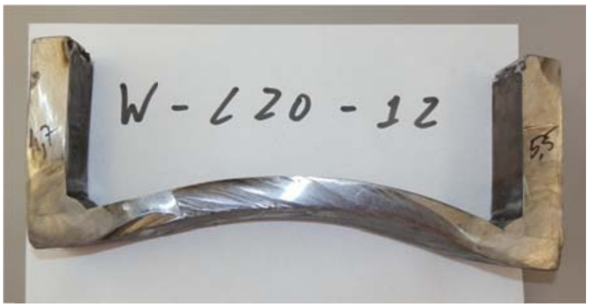
Figure 14 – Model #5 – elevated temperature model

8. POST SHORT-MISSION

After this short mission and bearing in mind the doubts encountered, mostly related to the welds throat, the test specimens at Coimbra University were cut in order to verify the weld throat. Figure 15, showing the weld throat details, confirm suspects that the welds' throat is not 9mm but around 5 mm (leg = 7mm) for both 'test 5 – 10 mm plate', and also 'test 6 – 12 mm plate':



Test 5 - W-C20-10 – specimen cut



Test 6 - W-C20-12 – specimen cut

Detail of left side weld – a = 4.3 mm

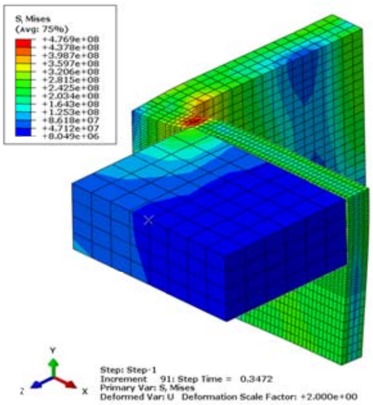
Detail of left side weld – a = 4.7mm

Detail of right side weld – a = 5.5 mm

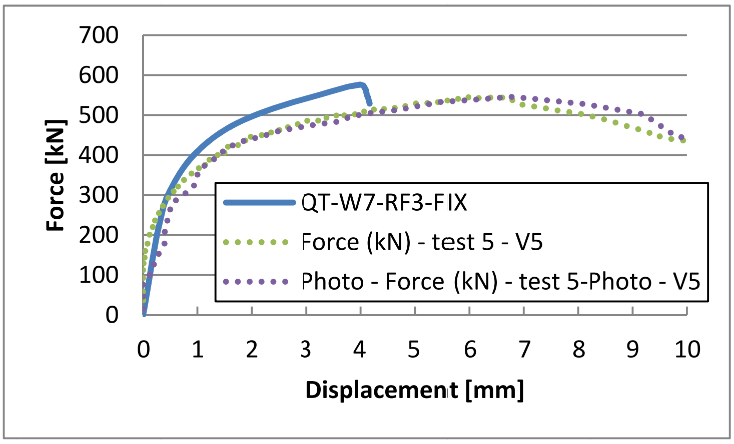
Detail of right side weld – a = 5.5 mm

Figure 15 – Specimen weld detail.

With this information a model of ‘test 5’ with a weld leg of 7mm was run - Figure 16. Afterwards, it was found of interest to run a model replying ‘test 6 -12 mm plate’, as the test results do not have any initial displacement blockage. The model was run with a weld leg of 7mm, perfectly tied front plate, and keeping material properties of the 10mm steel plate (since coupon data value are pretty much alike -Table 1). The model results agree well with the tests - Figure 17.

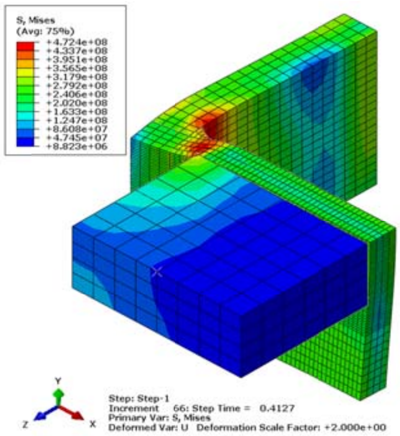


a) Stresses on deformed shape;

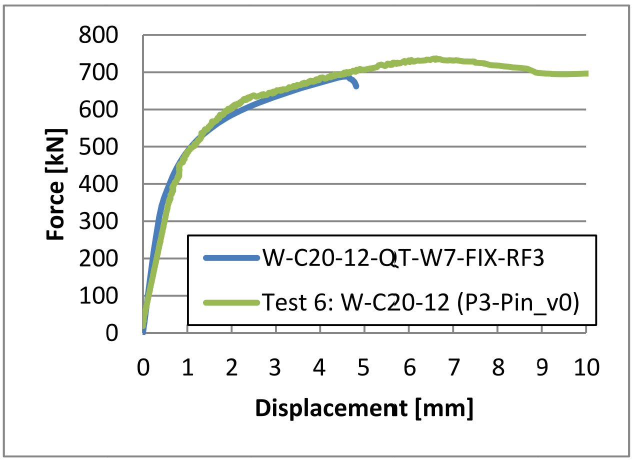


b) FEA Results Vs. Test results

Figure 16 – Model results of test 5 – 10 mm thick front plate – Ambient temperature.



a) Stresses on deformed shape;

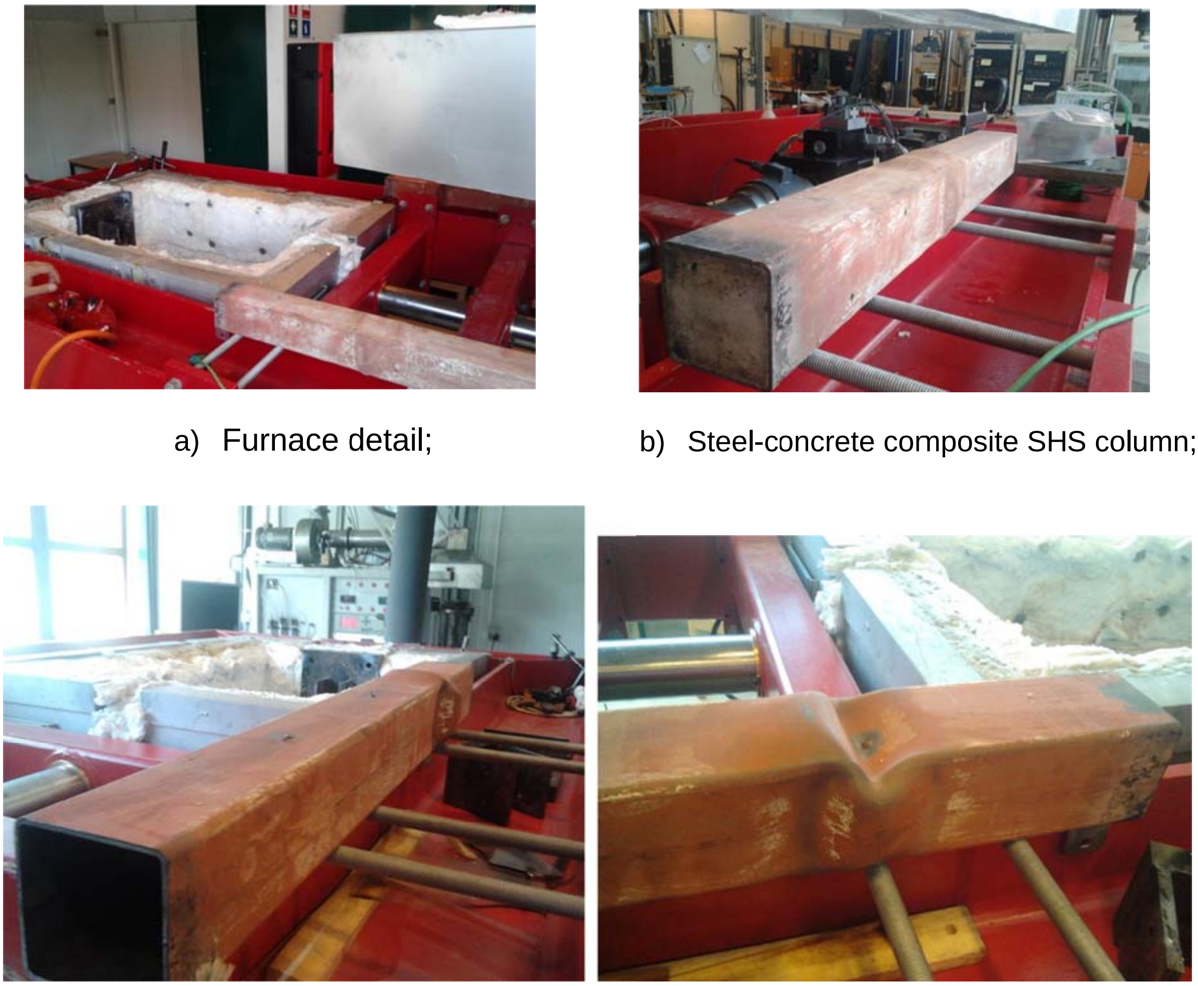


b) FEA Results Vs. Test results

Figure 17 – Model results of test 6 – 12 mm thick front plate – Ambient temperature.

9. LABORATORY VISIT

During the short mission we also took time visit Lulea's University laboratory, and as for fire matters is concerned, the laboratory is equipped with a small furnace with a load cell incorporated which allows for small scale tests at high temperatures (approx. 1200 mm specimen). Pictures below - Figure 18 - show the equipment and also test specimen from current works, in which both composite and steel column are subjected to a predefined axial load and subjected to temperature increase until failure.



a) Furnace detail;

b) Steel-concrete composite SHS column;

c) Steel SHS column;

d) Steel SHS column damage.

Figure 18 – Photograph from Lulea University Laboratory equipment.

10. FINAL REMARKS

During this short-mission there has been a straightening of the relationship and cooperation between Lulea University and Coimbra University. Cultural differences and technical matters were addressed and with the little time available the results of this cooperation have proven to be successful.

We have been able to identify possible causes for mismatch between the FEA and test results within the COMPFIRE project and shared finite element modeling with ABAQUS techniques, finding personal enrichment for future works for both institutions and the people involved.