A METHODOLOGY FOR RETICULAR STRUCTURES MODELING APPLIED TO THE FATIGUE ANALYSES OF RIVETED BRIDGES

R. C. Guedes Leite [1], R. Natal Jorge [2], A. M. P. de Jesus [3]

[1] Departamento de Engenharia Civil, CEFET-MG,
Belo Horizonte, Minas Gerais, Brazil,
Av. Amazonas, 7565, 30.421-169 Belo Horizonte, Brasil
E-mail: rleite@civil.cefetmg.br

[2] IDMEC/ Departamento de Engenharia Mecânica,
Faculdade de Engenharia da Universidade do Porto,
Rua Dr. Roberto Frias, 4200-465 Porto, Portugal
E-mail: rnatal@fe.up.pt

[3] IDMEC/Departamento de Engenharias,
Escola de Ciências e Tecnologia/UTAD,
Quinta de Prados, 5001-801 Vila Real, Portugal
E-mail: ajesus@utad.pt

ABSTRACT

Riveted bridges are ancient metallic structures that have been subjected to a long operational period, with increasing traffic loads and intensity. The fatigue damage levels experienced by these structures may be significant. S-N approaches are usually used to assess these structures. However, Fracture Mechanics and local strain approaches are gaining an increasing interest. The application of these alternative approaches requires detailed stress/strain analysis of the riveted connections, which demands considerable computation resources. This paper proposes an integrated CAD/FEA methodology for stress analysis of joints from reticular structures, based on a sub-modelling approach to mitigate the need for extensive computational resources. Detailed local three-dimensional solid finite element models of the joints are proposed, the respective boundary conditions being derived automatically from a global beam model of the structure. The procedure is demonstrated for the Trezói riveted railway bridge. The procedure to carry out a detailed stress analysis of a significant joint of the bridge for a moving train loading is illustrated.

KEY WORDS: fatigue analysis, riveted bridges, sub-modelling, finite element analysis.

1. INTRODUCTION

Today, riveted steel bridges are, mostly, centenary structures. The life time of this structures have been extended to respond to economical concerns, although, changes on traffic loads along the years, have exposed these structures to different overloads from that originally considered in design. In addition to the utilization of those bridges under new load conditions, theirs original designs, generally of the end of the 19^{th} century or the beginning of the 20^{th} century [1] did not considered fatigue as a damaging mechanism. Therefore, these bridges are an important group of metallic structures very likely to present high levels of fatigue damage. Fatigue assessment studies of existing bridges are consequently important to assess the current fatigue damage state as well as to estimate the remaining fatigue life of those structures. Adequate analysis models are then required. The usual model is based on the so-called S-N approach, which requires the nominal stress evaluation on members under analysis, for comparison with the fatigue strength data [2].

Recently, research projects have proposed the use of Fracture Mechanics and strain-based local approaches [3, 4]. However, these approaches demand for considerable computing resources since they require detailed stress/strain analysis at the joint. To reduce these resources needs, sub-modelling techniques can be used to perform detailed stress/strain analysis using three-dimensional solid finite element models only at the joints of interest, while a global model of the structure, using beam or shell elements, is used to derive the boundary conditions of the detailed model [5, 6].

In order to facilitate the application of the local methods for fatigue analysis of reticular structures, an integrated CAD system have been developed to build a geometric model of the structure, allowing specification and modification of geometry, boundary conditions, materials and sections. An exporting data module, that organizes the structure of the data in a compatible format with ANSYS® input [7], was developed and added to the global system. This module automates the exportation, for the finite element analysis, of the global structure data as the structure is submitted to multiple simulation scenarios (moving loads). In sequence, the system gathers the displacements results of a selected group of nodes for each one of the simulation scenarios. These results are organized in an ABAQUS® [8] input file, and are applied to the refined model of the selected structural connection. This methodology contributes to reduce the computational costs involved at the applications of the local approaches in the fatigue analysis of riveted bridges.

The developed methodology is demonstrated for a riveted bridge, namely the Portuguese Trezói railway bridge. This bridge is probably one of the last riveted bridges built in Portugal.

2. DESCRIPTION OF THE PROPOSED METHODOLOGY

In order to assist the application of the local methods to fatigue analysis of riveted bridges, an integrated CAD system has been developed to build the geometric model of reticular structures, using beam elements, and to manage and transfer data results to a local detailed solid finite element model, in the framework of sub-modelling techniques in finite element analysis.

To use the methodology, initially, a global model of the structure is built. This model represents the geometry of the structure. The system allows the definition, the application and the modification of actions, boundary conditions, materials and sections of each element of the model. A refined model with beam elements may be defined from the geometrical model, with automatic numbering of elements and nodes, and with a control that can recognize the original geometric elements and the refined elements obtained from that. With this functionality it is possible to refine only desired parts of the model.

This CAD system was developed from a framework applied to automation and integration systems applied to the reticular three-dimensional structural project process [9]. The modules of the framework responsible for the data control and the graphical view of the structure were adapted for the specific necessities of the riveted bridges. The data structure of the system had been complemented with the specific data required by the ANSYS® software and new functionalities were added to the global system, in order to better represent the model generated by that software. Those functionalities were not existent at the original systems developed using the framework. The final methodology automates not only the construction of the three-dimension reticular model of the structure but also the data export for the finite element analysis. An exporting data module was developed and added to the global system. This module organizes the structure of the data in compatible formats with the input for the ANSYS® and ABAQUS® software. The exporting data module automates the transportation of the global structure data, submitted to multiple loading scenarios (moving loads), for the finite element analysis at the software ANSYS®, and gathers the displacement results at a select group of nodes for the simulated several loading scenarios. This set of results is then written in an input file to be automatically applied at the refined model of the selected structural joint in the ABAQUS® software.

The traffic loads may be simulated by the CAD system that automates, in an iterative way, the application of an array of loads along a set of elements chosen by the user. At each step, the loads are applied at a specific position distant for the previous step position by a value defined by the user. Each position of the loads corresponds to a load step defined at the ANSYS® software.

After applying the traffic loads along the bridge, from typical trains, the system organizes and writes out the structural data to a file as well as the code that automates the finite element analysis process and gathers the displacements values at each load step, for a set of nodes selected by the user. The process may be repeated for different train types, with different distances between loads steps and different initial and final points of application. In order to obtain the critical load step, it is possible to refine the global structure defining the desired nodes near a structural joint and refine the application of the traffic load. After the simulation of the traffic loads at the structure, the system gathers the displacements for each node around the desired structural joint, at each load step. The displacements are written in an input file for the ABAQUS® software and are automatically applied to the refined model with solid elements of the analyzed structural join [5, 6]. The stress histories may then be evaluated for the traffic loads.

The proposed methodology consists of an object oriented integrated system [10, 11, 12] developed from an application framework, REMFrame [9]. This system integrates a modeller model, the Geometric Modeller, designed as a CAD system that uses the graphic platform AutoCAD® [13, 14] as a graphic interface, with two external analysis systems: the software ANSYS® for the global analysis and the software ABAQUS® [8] for the local analysis.

3. A CASE STUDY: THE TREZÓI BRIDGE

The proposed methodology was applied to model probably the latest riveted bridge built with riveted joints, the Trezói railway bridge, constructed at the Beira Alta line, in Portugal, just after the Second World War. The availability of the original drawings of the bridge and previous researches carried out on this bridge [15] allowed the construction of the Trézoi Bridge geometric model, with some precision.

After constructing the geometric model of the bridge, the CAD system allows the definition and modification the actions (loads), boundary conditions, materials, sections and type of elements. The geometric model can then be transformed in a refined model, both with beam elements. A strict correspondence between the original geometric element and the refined elements generated is managed by the system, allowing successive refining. This functionality was used to define a set of nodes near the desired structural joint (see Figure 1).

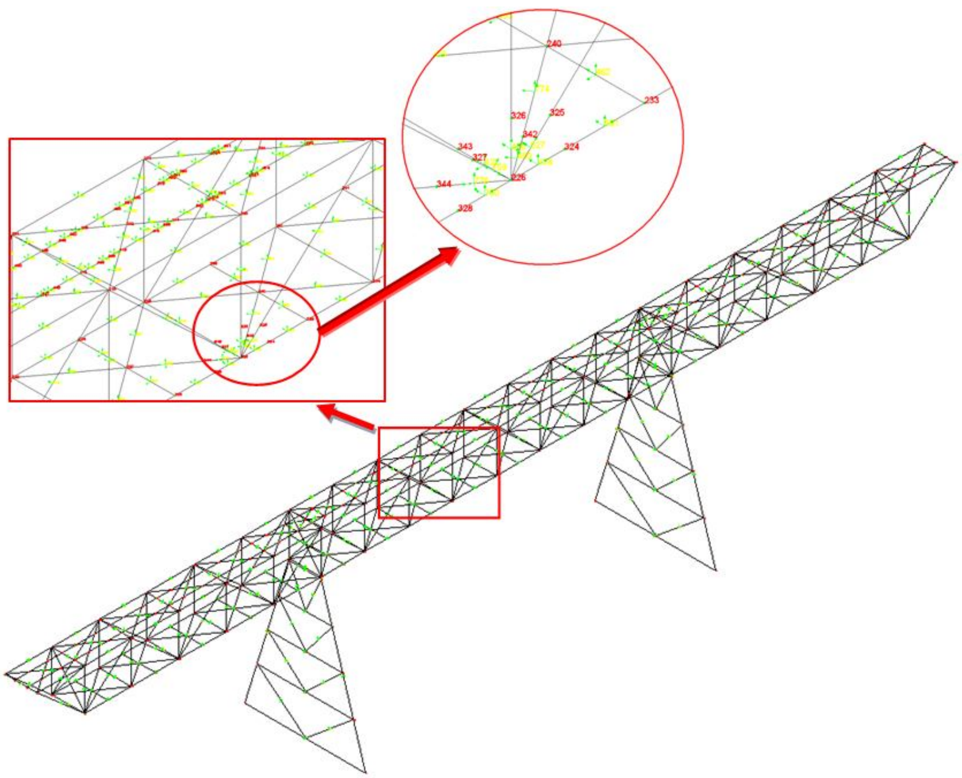

Figure 1. Geometric and refined models generated by the CAD system.

After having defined the refined model it is possible to carry out sensitivity analysis and investigate multiple simulation scenarios by the application of different load cases of different load steps. This functionality allows the simulation of a train crossing the bridge.

In the study of the Trézoi bridge, three types of trains were chosen, according to the Eurocode 3, respecting the actual maximum train speeds at the bridge: 90 and 110 Km/h in different directions [15]. The chosen train types are: type 2 - a locomotive-hauled passenger train; type 7 - a locomotive-hauled freight train; type 11 - a heavy traffic with 250kN – axles [16] (see Figure 2).

The CAD system allows the application of a train type through a set of consecutive selected elements, in a regular interval of load, generating different load steps. The refined model had to be modified at each load step to insert nodes just at the application point of the loads (see Figure 1). Those modifications are iterative and automatic, managed by the developed CAD system. For each load step the model of structure is saved in an appropriate format allowing posterior recuperation (see Figure 3).

After the application of a specific train type through all the bridge, the exporting module is automatically called to write the structural data with all the loads steps in the appropriate format required by the software ANSYS®. In sequence, the original model is recovered, allowing the application of other train types to the structure.

The exporting file written by the CAD system in the APDL Language carry out not only the structural data, but has also some line commands to automate the analysis of the structure at the software ANSYS® (see Figures 4 and 5), as well as to gather the displacements and rotations at a set of nodes chosen by the user, writing them in a specific format at a ASCII file (see Figures 1 and 6).

Using the same exporting module, the user can ask the system to read the deformations written at a specific file and to organize them in a input file, allowing the automatic application of a set of deformations to a specific node of a solid model, analyzed at the software ABAQUS® (see Figure 7).

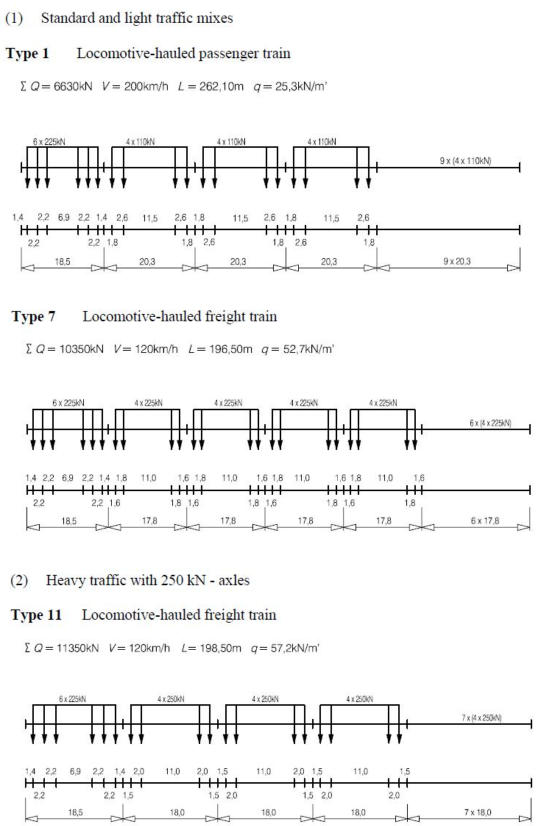

Figure 2. Train types (extract of European Committee for Standardization - EN1991-2 (2003).

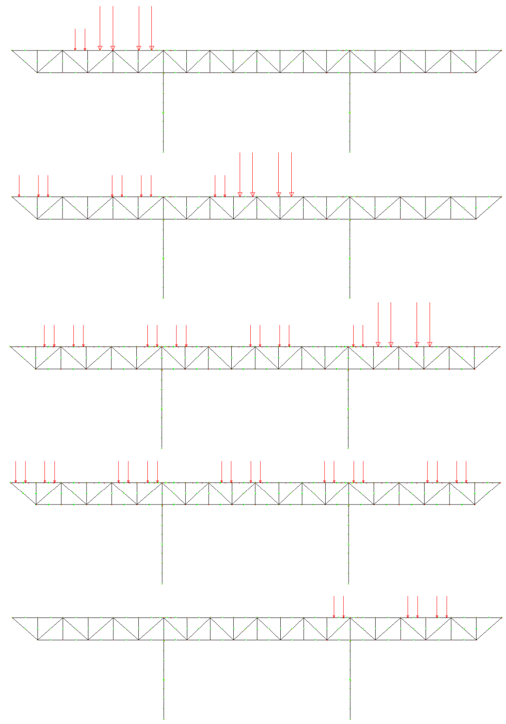

Figure 3. Load steps 2, 3, 4, 8 and 11 for train type 2, with 36m between steps.

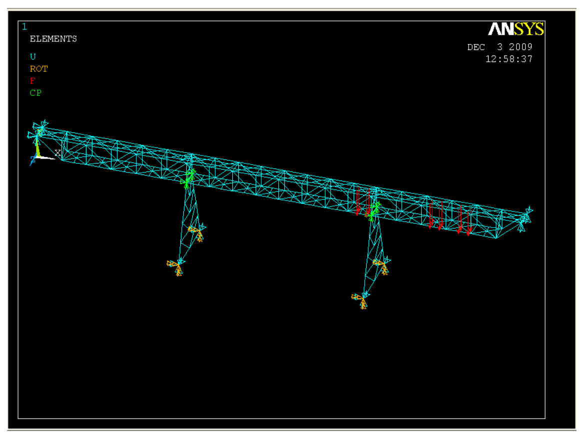

Figure 4. Analysis of the global structure: load step 11 and boundary conditions

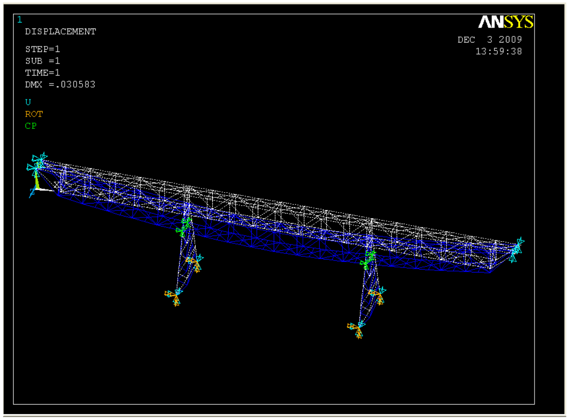

Figure 5. Analysis of the global structure: deformed shape for load step 11.

NODAL DISPLACEMENT [mm]

STEP	NODE	Ux [mm]	Ux [mm]	Ux [mm]	ROTX	ROTY	ROTZ
1.	207.0	-.2268E-02	-.28E-01	-.2918E-04	0.62E-04	-.1188E-04	-.39E-03
1.	208.0	-.2039E-02	-.27E-01	0.7885E-05	0.47E-04	-.2982E-05	-.39E-03
1.	274.0	-.1949E-02	-.27E-01	0.2290E-04	0.41E-04	-.6515E-07	-.24E-03
1.	278.0	-.4881E-02	-.30E-01	0.6481E-05	0.46E-04	0.3174E-05	0.53E-02
1.	323.0	-.2353E-02	-.27E-01	-.3102E-04	0.32E-04	0.1194E-04	-.41E-03
1.	345.0	-.2287E-02	-.27E-01	-.3994E-04	0.14E-03	-.2864E-04	-.43E-03
1.	346.0	-.2315E-02	-.27E-01	-.1794E-04	-.11E-03	-.4886E-06	-.38E-03
1.	347.0	-.2341E-02	-.27E-01	-.4003E-04	0.21E-03	0.2779E-04	-.28E-03
2.	207.0	-.4183E-09	-.54E-08	0.7324E-08	0.30E-08	0.1699E-10	-.19E-10
2.	208.0	-.4150E-09	-.54E-08	0.9976E-08	0.28E-08	-.5268E-09	-.35E-10
2.	274.0	-.4286E-09	-.54E-08	0.1136E-07	0.36E-08	-.2185E-10	0.76E-11
2.	278.0	-.4905E-09	-.54E-08	0.1009E-07	0.28E-08	0.7610E-09	0.30E-10
2.	323.0	-.4589E-09	-.52E-08	0.7300E-08	0.28E-08	-.7248E-10	-.20E-09
2.	345.0	-.4065E-09	-.33E-08	0.7276E-08	-.20E-10	0.1648E-10	0.20E-08
2.	346.0	-.3642E-09	-.20E-08	0.7228E-08	0.27E-08	-.1232E-09	0.37E-11
2.	347.0	-.3725E-09	-.33E-08	0.7324E-08	-.17E-08	-.1052E-09	-.33E-08
3.	207.0	-.1729E-06	-.18E-05	0.7270E-07	0.34E-07	-.1591E-08	0.10E-07
3.	208.0	-.1769E-06	-.18E-05	0.9964E-07	0.31E-07	-.2387E-08	0.63E-08
3.	274.0	-.1800E-06	-.18E-05	0.1119E-06	0.33E-07	-.1061E-08	0.10E-08
3.	278.0	-.1707E-06	-.18E-05	0.9938E-07	0.31E-07	0.2638E-08	-.50E-08
3.	323.0	-.1791E-06	-.18E-05	0.7097E-07	0.29E-07	0.5625E-10	-.34E-09
3.	345.0	-.1731E-06	-.18E-05	0.7158E-07	0.44E-07	-.2623E-08	0.48E-09
3.	346.0	-.1739E-06	-.18E-05	0.7229E-07	0.30E-07	-.2091E-08	0.72E-08
3.	347.0	-.1769E-06	-.18E-05	0.7063E-07	0.56E-08	0.1208E-08	-.13E-07
4.	207.0	-.3613E-06	-.50E-05	0.9342E-07	0.48E-07	-.3229E-08	0.22E-07
4.	208.0	-.3681E-06	-.50E-05	0.1300E-06	0.44E-07	-.3263E-09	0.11E-07
4.	274.0	-.3751E-06	-.50E-05	0.1446E-06	0.39E-07	0.5910E-10	-.39E-08
4.	278.0	-.3403E-06	-.50E-05	0.1383E-06	0.46E-07	0.1437E-07	-.33E-07
4.	323.0	-.3802E-06	-.50E-05	0.9390E-07	0.41E-07	0.2293E-08	0.24E-08
4.	345.0	-.3643E-06	-.49E-05	0.9151E-07	0.79E-07	-.6419E-08	-.54E-08

Figure 6. Nodal displacements of a user selection set of nodes near a structural joint.

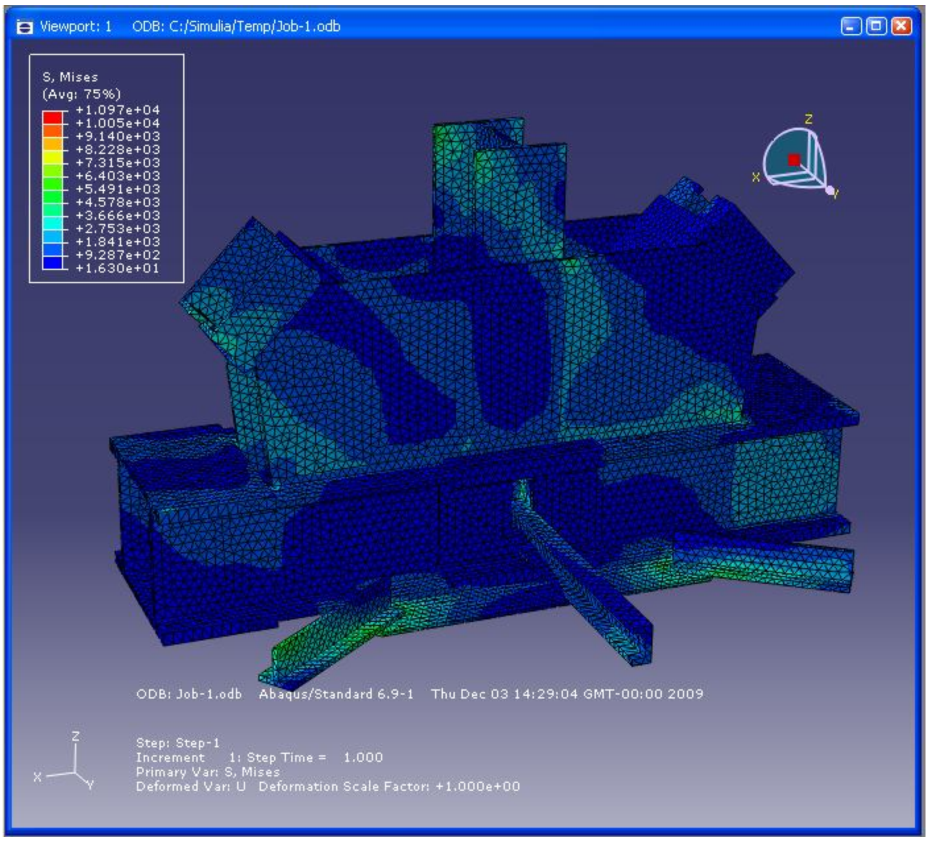

Figure 7 – Detailed FE solid model of a structural joint: Von Mises stresses.

4. FINAL REMARKS

The presented methodology allowed the application of the sub-modelling techniques to the modelling of the Trezói Railway Bridge in Portugal. The analysis of this bridge has been carried out with the construction of two basic models: a global model of the structure with beam elements and a local model of the desired joint with solid elements. The developed integrated CAD system permits the construction and management of the structural data for the global model, as well as it automates the transportation of a huge quantity of data between different external systems: one utilized for the analysis of the global structure and other to the local analysis. The methodology contributed to reduce, significantly, the computing resources required to the application of the local approaches in the fatigue analysis of this riveted bridge, as also permitted the simulation of different load scenarios applied through all the bridge and allowed the automatic local analyses with the corresponding deformations of a set of nodes selected by the user through the structure.

In the case study the application of the sub-modelling permitted the local approaches whit a low computational cost: the global model has only about 600 nodes and 1000 beam elements, while the joint model has about 25 000 nodes and 85 0000 elements. A first simulation of the traffic load utilized a distance between positions of loads applications of 36m and generates 12 load steps through all the bridge, for the train type 2, 9 load steps for the train type 7 and 10 load steps for the train type 11. For each train type and for each load step the deformations of eight nodes around a structural join in a principal beam in the central section were obtained from the software ANSYS® and transferred to the local model at the software ABAQUS® and all stages of the analysis runs in a single computer, with no special configuration. Actually new approaches have being done with narrow space between loads steps and with a large local model. Those new approaches are possible with a little work due to the use of the developed CAD system.

The proposed integrated CAD system will be used in future works to assess the stress field histories at the main joints of the bridge. These stress histories will be used to assess the critical locations of the joint and the fatigue damage at those locations. The model presented in Figure 7 is a continuous model of the joint, however it is expected in future developments to include rivets in critical branches of the joint.

5. RERERENCES

[1] J. Moreno, A. Valiente, 2004, "Stress intensity factors in riveted steel beams", Engineering Failure Analysis, 11, pp. 777-787

[2] DiBattista, J.D., Adamson D.E.J. & Kulak G.L. 1998. Evaluation of remaining fatigue life for riveted truss bridges. Canadian Journal of Civil Engineering 25(4):678-691.

[3] De Jesus, A.M.P., Correia, J.A.F.O. 2008. *Fatigue assessment of riveted railway bridge connections. Part II: numerical investigations*. In P.J.S. Cruz, L.S. Silva & F. Schröter (eds) Steel bridges. Advances solutions & technologies: 339-48. European Convention for Constructional Steelwork.

[4] De Jesus, A.M.P., Pinto, H., Fernández-Canteli, A., Castillo, E., Correia, J.A.F.O. 2010. Fatigue assessment of a riveted shear splice based on a probabilistic model. International Journal of Fatigue 32: 453-462

[5] B.M. Imam, T.D. Righiniotis, M.K. Chrysssanthopoulos, 2007, "Application of probabilistic fracture mechanics in evaluation of existing riveted bridges", Engineering Structures, 29, pp. 3071-3081.

[6] T.D. Righiniotis, B.M. Imam, M.K. Chrysssanthopoulos, 2008, "Fatigue analysis of riveted railway bridge connections using the theory of critical distances", Engineering Structures, in press.

[7] Swanson Analysis Systems Inc., Version 12.0, ANSYS (2009) Houston.

[8] Dassault Systèmes Simulia Corp., Version 6.9-1 , ABAQUS/CAE (2009) Providence, RI, USA.

[9] LEITE, R. C. G. "Um Framework Para Automação/Integração do Processo de Desenvolvimento de Projetos de Estruturas Reticuladas Tridimensionais". 2007. Tese (doutorado em Engenharia de Estruturas) Escola de Engenharia, UFMG, Belo Horizonte.

[10] ECKEL, Bruce, "Thinking in C++ - vols 1 & 2", 2nd Edition, Prentice Hall, 1999. Disponível em:<http://www.mindview.net/Books/TICPP/ThinkingInCPP2e.html> Acesso em mai 2002

[11] GAMMA, E., HELM R., JOHNSON R., VLISSIDES J. "Padrões de Projeto: soluções reutilizáveis de software orientado a objetos." Porto Alegre: Bookman, 2000. 363p.Trad.: Salgado, Luiz. A. Meirelles.

[12] BUSCHMANN, F. MEUNIER, R. ROHNERT, H. SOMMERLAD, P. STAL, M. "Pattern - Oriented Software Architecture A system of Patterns". Chichester: John Wiley & Sons, 1996. 467p

[13] AUTODESK Inc., "ObjectARX for AutoCAD 2009". Disponível em < http://usa.autodesk.com/adsk/servlet/index?id=773204&siteID=123112> Acesso em Abr 2009.

[14] AUTODESK Inc., Version C.56.0 (UNICODE), AutoCAD 2009.

[15] MARQUES, F. M. de S. "Avaliação do Comportamento Estrutural e Análise de Fadiga em Pontes Metálicas Ferroviárias". 2006. Tese (Mestrado em Estruturas de Engenharia Civil) Faculdade de Engenharia da Universidade do Porto. Porto

[16] European Committee for Standardization (CEN) – EN1991-2 (2003) – "Actions on Structures – Part 2: General Actions – Traffic loads on bridges", Brussels