Steel beam crack propagation simulation and experimental verification

Sebastián Moya Lazo[1], Mauricio Jara Ortiz[2], Ignacio Fuenzalida Henríquez[2]

[1]*School of Civil Engineering, Faculty of Engineering, University of Talca*
Camino a Los Niches Km 1, Curicó 3340000, Chile.
smoya15@alumnos.utalca.cl
[2]*School Civil Mining Engineering, Faculty of Engineering, University of Talca.*
Camino a Los Niches Km 1, Curicó 3340000, Chile.
mauriciojara@utalca.cl
[2]*Engineering Systems Doctoral Program, University of Talca. Engineering Building and Management Department, Faculty of Engineering, University of Talca.*
Camino a Los Niches Km 1, Curicó 3340000, Chile.
ifuenzalida@utalca.cl

Abstract. Steel is a structural material par excellence due to the great ductility it offers, and its excellent behavior under tensile and bending loads. However, steel can also crack after a significant number of static and dynamic load cycles. These cracks can be the cause of a later structural failure and, being part of important constructions, they bring with them a considerable economic loss. During the last years, engineering has studied with determination the concept of damage tolerance. In this work, two types of commercial steel beams were selected, where with small interventions to their geometry and inducing a crack in their center, it was possible to study the propagation of said crack. At the same time, the geometric dimensions and constitutive properties of the tested beams were entered into the Salomé-Meca finite element software, seeking to compare the experimental behaviors with those of the simulation. Finally, the tests carried out in the laboratory and the computational simulation indicated a crack propagation and a new location of the crack front that is millimetrically similar. This shows that the experimental test is verified with respect to the computational simulation, validating this method to study tolerance to structural damage.

Keywords: Salomé-Meca, X-FEM, damage tolerance, crack propagation, steel.

1 Introduction

It is very common to find various types of failures in structures, although it is easy to identify them, the complex thing is to be able to understand their origin, their behavior over time and the possible difficulties that they can bring to the project. One of the most common and important failures to analyze are cracks.

The crack propagation analysis consists of an iterative computational process, composed of several stages that show its progress for each load case until convergence is reached. The classical analysis consists of partitioning the analysis mesh manually every time the crack changes dimension in the computational software.

In this project, a crack will be induced in a steel beam in the shape of a dog-bone to then be tested in the laboratory with bending machinery. Subsequently, the results obtained in the laboratory will be verified with the results of the crack propagation carried out in the computer software.
In order to perform the computational calculations, the free license Salomé-Meca software will be used. To achieve the simulation of crack propagation, the extension of the finite element methods of this computer program called X-FEM [3][4] will be used, which consists of using shape functions, not requiring an extra meshing cut on them [5]. This analysis has an important relevance for different kinds of industry, such as mining, railway and civil

works. Carrying out a good crack simulation and verification provides optimal results in order to avoid the demolition of the structure in question, that is, prevent the stoppage of said industrial sector and thus avoid losing money.

2 State of the Art.

One of the main causes of failure in structural elements is fatigue [1]. Therefore, it is a phenomenon that affects all types of buildings, which requires an engineering response or solution. There are two ways to solve this dilemma: demolition and reconstruction, or the analysis of the crack and its behavior in order to avoid the need for intervention.

On the other hand, the use of computer software based on finite elements is of vital importance in order to make the decision to demolish or rebuild. A finite element analysis is commonly performed to predict the mechanical behavior of a structure. Subsequently, said analysis is verified with respect to the current regulations in each country.

2.1 Introduction to the fatigue phenomenon.

Structural fatigue originates from cyclical stresses that fluctuate between tension and compression. The accumulation of all the stresses applied to the structure in its useful life is what is called fatigue resistance.

On many occasions, the fatigue measurement is obtained from the difference in the stress intensity factor, which is composed of the applied stresses and the respective geometric dimensioning of the crack [2]. This factor is represented by the following equation:

$$\Delta k = K_{max} - K_{min}$$

Where:

- Δk: Stress intensity factor.
- K_{max}: Maximum value of the stress intensity factor of a cycle.
- K_{min}: Minimum value of the stress intensity factor of a cycle.

The stress intensity factor (SIF) determines the stress state in the contour of the end of the crack.

2.2 Fatigue resistance in fracture mechanics.

In fracture mechanics, this method evaluates fatigue resistance with or without previous problems and is mainly based on the relationship between the speed of crack propagation and the stress state to which the element is subjected [7].

Crack propagation consists of the following stages:

<u>Initiation</u>: It can start at any point in the material, but cracks are usually generated in the places where the stresses are concentrated and on the outer surface. At this point, there are surface crack repair treatments that prolong the useful life of the element.

<u>Growth</u>: The crack tends to develop perpendicular to the application of stress, which is divided into two stages:

<u>Microscopic growth:</u> It is not related to fracture mechanics, but if different elastoplastic laws are also applied, the growth is at the grain level of the material.

<u>Macroscopic growth:</u> There is a rapid growth, and the Paris law is applied: [6]

$$\frac{da}{dN} = C\Delta K^{m}$$

Where:

- ΔK= Stress intensity range.
- $\frac{da}{dN}$ = Crack length growth rate.

The other variables are constants that depend on the type of material.

<u>Breakage:</u> It happens when the size of the section fails to withstand the load of the cycle and the remaining sector breaks.

3 Methodological Framework

Studies using finite element methods are carried out to predict the physical behavior of a structural element or a structure itself. Laboratory tests are a tool where the properties and characteristics of a material are determined in a tangible way. This is why a large number of samples are studied in the industry to ensure the quality of the elements that are being used.

3.1 Research design.

In the framework of this project, a research design based on experimentation is developed, where a crack propagation will be analyzed according to the amount of load applied to a beam. In this study, two research fronts will be carried out, which will later be verified. The first focuses on the laboratory, where bending tests will be carried out on a steel beam with a dog-bone shape [9] with an induced crack. The second will be done using the finite element software called Salomé-Meca [8]. In order to begin the research in this computer program, the idealized model to be studied must be carried out, that is, that the elements generated in the finite element software are understandable for it. Immediately, the number of nodes and the type of mesh to be used for the analysis must be determined. Subsequently, the loads and the boundary conditions of the problem will be defined, in addition to applying the analysis functions for the numerical simulation. Finally, the results obtained will be verified with standards and/or some control element of the simulation. In this research, the computational results will be verified with the test carried out in the laboratory.
To understand the study, two variables are identified: amount of applied load (independent variable) and advance length of the crack (dependent variable), in which the following is described:

- Amount of applied load: This is the control variable of the experimentation, since it is constantly changing and it is the action carried out in the laboratory to obtain the consequences of the research.
- Crack progress: This corresponds to a dependent variable that develops as a consequence of load application.

In addition to the variables mentioned, there are other parameters that depend on the material that must be taken into consideration, for example: strange variables (elasticity modulus, Poisson's modulus, steel composition, natural imperfections and cracks, radius in the fracture criterion, etc.) and intervening variables (geometric shape of the sample, dimensions of the element, inertia, among others). These parameters are important to understand crack propagation in the material.

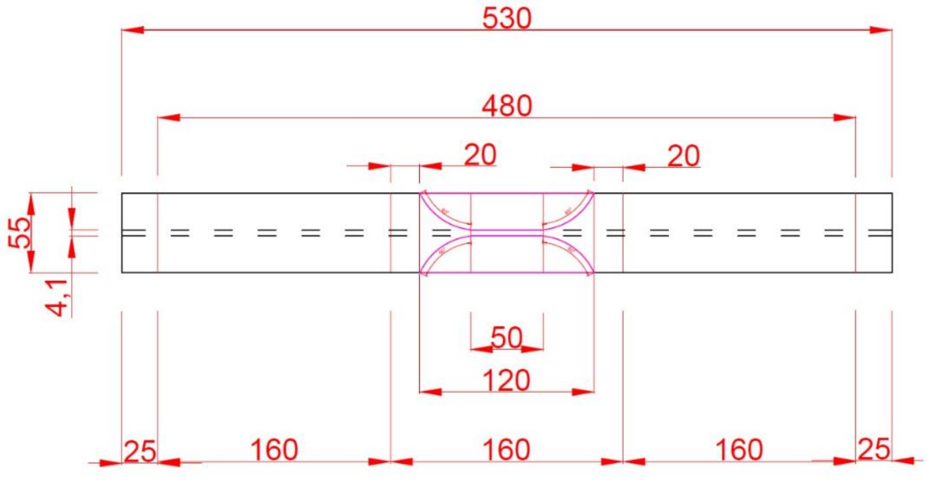

Figure 2 IPE 100 specimen model and dimensions.

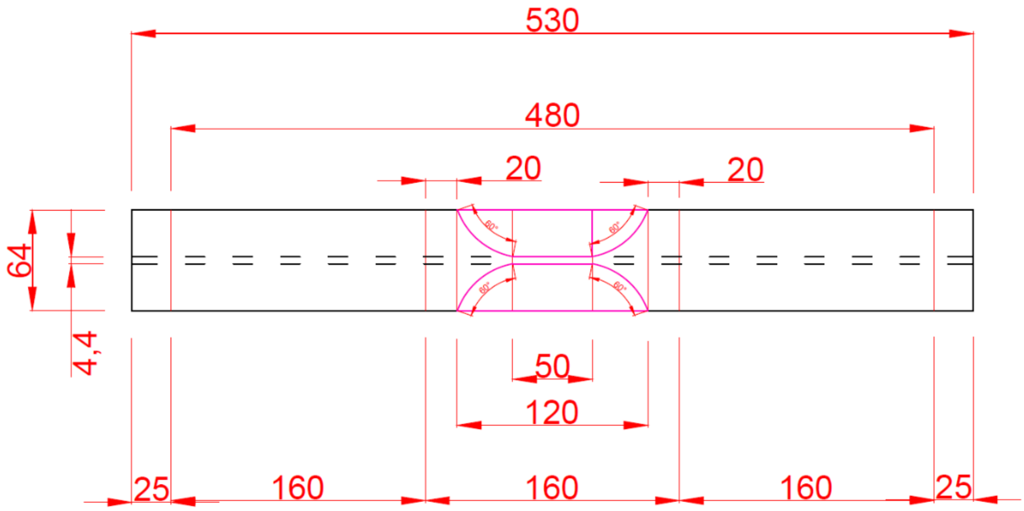

Figure 3 IPE 100 specimen model and dimensions.

4 Analysis of Results

4.1 Analysis of results in laboratory bending tests.

4.1.1 IPE 120 Beam

The three IPE 120 steel beams were the first to be tested in the hydraulic press. The objective of the test carried out was to measure the maximum applied load (KN), the crack propagation in (cm), and the coordinates of the crack front with respect to an axis assigned at one end of the specimen. The results of the tests were those shown in table number 1.

Table 1. Experimental results for IPE 120 beam.

N° of tested specimen	Specimen 1	Specimen 2	Specimen 3	Average of specimens
Maximum applied load (KN)	70.22	60.14	59.49	63.28
Coordinates on the length of the beam (cm)	27.80	27.80	27.80	27.80
Coordinates at the height of the beam (cm)	3.70	3.60	3.70	3.67
Crack propagation (cm)	1.30	1.20	1.30	1.27

Regarding the location coordinates of the already propagated crack front, it is identified that similar values are obtained in the three specimens, where the average location is 27.80 centimeters in its length and 3.67 centimeters in its height. Additionally, the average crack propagation was 1.27 cm.

4.1.1 IPE 100 Beam

Subsequently, the three IPE 100 beams were tested. In the same way as the previous samples, the applied load measurement (KN), the crack front coordinates (cm) and the crack propagation (cm) were obtained. The results of the experimentation can be seen in the following table.

Table 2. Experimental results for IPE 100 beam.

N° of tested specimen	Specimen 1	Specimen 2	Specimen 3	Average of specimens
Maximum applied load (KN)	34.00	31.46	37.20	34.22
Coordinates on the length of the beam (cm)	27.90	27.70	28.00	27.87
Coordinates at the height of the beam (cm)	3.90	3.80	3.80	3.83
Crack propagation (cm)	1.40	1.20	1.00	1.20

Regarding the crack propagation, it was recorded between 1.40 cm and 1.00 cm, obtaining an average propagation of 1.20 centimeters. Finally, for the coordinates, it was obtained, on average, that the location of the already propagated crack front was at the coordinate 27.87, measured in centimeters, with respect to the length of the beam, while, regarding the height, the average of coordinates was 3.83 centimeters.

4.2 Analysis of results in Salomé-Meca bending tests.

Bending tests were carried out with several iterations for each type of sample, varying the length of the crack allowed for a constant load step number. This variation generated the various propagations in the samples. The parameters of the Paris law were kept constant for all simulations with values of $C = 3.8x10^{-12}$ and $M = 3$ for A36 steel. The applied load for each type of beam was obtained experimentally in the laboratory, for both types of constant beams.

4.2.1 IPE 120 Beam

For the IPE 120 beam, the crack propagation in the experimental test was 1.27 centimeters, so the convergent results are as follows:

Table 3. Convergent results of the IPE 120 beam simulation.

Meshing dimension	Crack Progress Length (mm)	Propagation (mm)	Final coordinate in beam length (mm)	Final coordinate at beam height (mm)
Meshing 50 (mm)	4.9	12.30	280.95	27.32
Meshing 25 (mm)	5.8	12.27	281.95	31.85
Meshing 12.5 (mm)	6.1	12.33	284.42	36.88
Meshing 10 (mm)	6.1	12.35	277.86	37.22
Meshing 5 (mm)	6.1	12.20	280.02	37.15

To validate the results of Table 3, the error associated with meshing is calculated; what is known as analysis of results. As to perform this calculation, a comparison is made with an exact result. In this case, it is verified with respect to the laboratory test. The following table shows the associated errors.

Table 4 Approximate meshing error for IPE 120 beam.

Meshing dimension	Final coordinate error in beam width (%)	Final coordinate error at beam height (%)
Meshing 50 (mm)	1.06	25.55
Meshing 25 (mm)	1.42	13.22
Meshing 12.5 (mm)	2.31	0.49

Meshing 10 (mm)	0.05	1.42
Meshing 5 (mm)	0.73	1.23

4.2.2 IPE 100 Beam

As the meshing dimension gets smaller, the lead length tends to converge near 6, similar to the IPE 120 beam simulations. Meanwhile, the crack front coordinates after the simulation vary depending on the length of the mesh. With respect to the IPE 100 beam, the crack propagation in the experimental test was 1.20 centimeters, so the convergent results are as follows:

Table 5 Convergent results of the IPE 100 beam simulation.

Meshing dimension	Crack Progress Length (mm)	Propagation (mm)	Final coordinate in beam length (mm)	Final coordinate at beam height (mm)
Meshing 50 (mm)	4.6	11.99	277.52	29.36
Meshing 25 (mm)	5.8	11.96	276.80	33.80
Meshing 12.5 (mm)	5.9	12.04	276.51	38.13
Meshing 10 (mm)	5.9	12.04	278.74	38.86
Meshing 5 (mm)	6.0	12.00	280.13	38.81

Finally, to validate the results of table 5, the error associated with meshing is calculated, that is, how close the simulations are with respect to laboratory experimentation. The following table shows the associated errors.

Table 6 Approximate meshing error for IPE 100 beam.

Meshing dimension	Final coordinate error in beam width (%)	Final coordinate error at beam height (%)
Meshing 50 (mm)	0.42	23.34
Meshing 25 (mm)	0.68	11.75
Meshing 12.5 (mm)	0.79	0.44
Meshing 10 (mm)	0.01	1.46
Meshing 5 (mm)	0.51	1.33

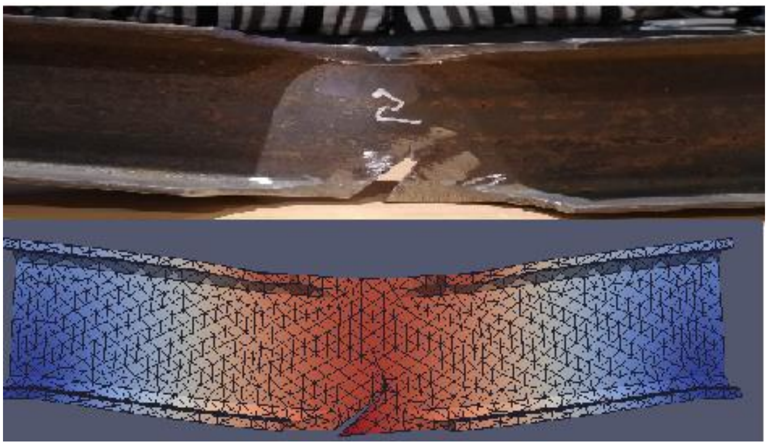

Figure 2 Verification of the experimental and simulated IPE 100 sample.

5 Conclusions

When comparing the results of both procedures, it was possible to validate the results with each other, since crack propagations and the location of the new crack fronts in the two studies are very similar for both types of samples. This validation allows considering this computational analysis to study structural elements, verifying their damage tolerance and the behavior over time of cracks and irregularities of the material.

In laboratory bending experimentation, crack propagation was achieved in all samples, finding new coordinates for the crack front after load application. By having crack propagation, selected business models are accepted for experimentation. On the other hand, a deformation occurs in the center of the beam according to the application of bending loads in a gravitational way. Additionally, a secondary phenomenon of elastic instability occurs in the area weakened by the dog-bone. This phenomenon is called buckling.

Regarding the simulation in Salomé-Meca, a study was carried out with materials and properties of a linear nature. The analysis carried out was of the linear static mechanical type, which is commonly performed on structures. It was possible to propagate the induced crack in the simulated model, finding new coordinates for the front of the virtually loaded crack, which is why the software is accepted as a crack propagation analysis tool and fatigue study in structural elements.

On the other hand, when studying a problem of finite element methods, it is necessary to perform an analysis of the results on the simulations generated by the different sizes of assigned meshes, since, with a greater number of nodes and elements, the error associated with the meshes decreases, approaching the real results. Notwithstanding the above, not necessarily assigning more nodes and elements will considerably improve the results, but it may cause a computational overload, slowing down the iterations. For this research, both models were assigned a 50 mm, 25 mm, 12.5 mm, 10 mm and 5 mm mesh. Although the first presents a percentage of error in the coordinate of considerable height, as the meshes of smaller lengths were used, this was progressively decreasing, managing to visualize the process of reducing the meshing error through the simulation of both models, so that the 5 types of mesh element sizes associated with the virtual test are accepted.

6 Acknowledgements

This work was partly funded by the international project ECOS-CONICYT C17E04 between LMT and Universidad de Talca, ANID/CONICYT grant number ECOS C17E04. In addition, this research was funded by ANID, grant number PFCHA / DOCTORADO BECAS CHILE / 2018 – 21181707. Finally, the authors would like to thank Carolina Fuenzalida and the help provided for the English translation of our work.

References

[1] Balseiro, J. W. (2011). Fatigue: Main cause of failure of mechanical elements. *Engineering and Development Scientific Journal; No 1 (1995): January-June.*

[2] National Highway Laboratory. (Registration N° 181143). Road Laboratory Course. Volume VI. Concrete and its components. Chapter VI. Concrete pavement restoration techniques. 151-160 pp. Santiago Chile

[3] Andrade, A. A., Mosquera, W. A., & Vanegas, L. V. (2015). Models of fatigue crack growth. Between Science and Engineering, 9(18), 39–48. Recovered from: http://www.scielo.org.co/scielo.php?script=sci_arttext&pid=S1909-83672015000200006&lng=en&nrm=iso&tlng=es

[4] Belytschko, T., & Black, T. (1999). Elastic crack growth in finite elements with minimal remeshing. *International Journal for Numerical Methods in Engineering*, *45*(5), 601–620. https://doi.org/10.1002/(SICI)1097-0207(19990620)45:5<601::AID-NME598>3.0.CO;2-S

[5] Trujillo Montes, D. (2014). *Fatigue analysis in metal joints using the Salomé-Meca environment*. http://hdl.handle.net/10481/33391

[6] Andrade, A. A., Mosquera, W. A., & Vanegas, L. V. Models of fatigue crack growth.

[7] Darcis, P., Santarosa, D., Recho, N., & Lassen, T. (2004). A fracture mechanics approach for the crack growth in welded joints with reference to BS 7910. In *ECF15, Stockholm 2004*.

[8] Open code structural numerical analysis (FEM) - Code_Aster. EDF (2016).

[9] Rodriguez, H. Mechanical fatigue resistance. Arahal, Spain: *Ingemecanica.* Recovered from: https://ingemecanica.com/tutorialsemanal/tutorialn217.html