Available online at www.sciencedirect.com

ScienceDirect

Procedia Engineering 213 (2018) 846–855

Procedia Engineering

www.elsevier.com/locate/procedia

7th International Conference on Fatigue Design, Fatigue Design 2017, 29-30 November 2017, Senlis, France

# Internal voids as a stress reliever and palliative in fretting fatigue

Diego Erena, Jesús Vázquez, Carlos Navarro*, Jaime Domínguez

*University of Seville, Camino de los Descubrimientos, Seville 41092, Spain*

**Abstract**

Currently, additive manufacturing with metals is an increasingly popular technique that allows the manufacturing of pieces of difficult shapes, nearly impossible to make with other techniques. Usually, these shapes try to optimize the solid to have the same strength with a lower weight. The fatigue behavior of the material of the components manufactured with this technique is a field in development. On the other hand, fretting fatigue is a common type of fatigue where a "stress concentration" appears due to the contact between two components. There are some procedures used to increase fatigue life in this situation (shot peening, surface knurling, etc.). This paper tries to analyze the possible beneficial effect on fatigue life of introducing voids inside the material in components under fretting, which is feasible now thanks to additive manufacturing. The problem under study is, for now, a 2D simplification where a cylinder is in contact with a half plane and a normal constant load and a variable tangential load are applied. This geometry has been numerically simulated, introducing a circular hole below the surface. The effect of this hole is to make the contact more elastic, which decreases the stresses near the surface. This work analyses and compares the stress and strain fields and Smith-Watson-Topper multiaxial fatigue parameter in the areas sensitives to fretting with respect to a case with homogeneous material (no internal voids). Various configurations changing different parameters like size and position of the hole, friction coefficient and the size of the slip zone have been considered. The problem analyzed in this paper is two-dimensional, therefore there would be no need to use additive manufacturing in a real situation. However, the results obtained in this paper indicate that it could also work in 3D. Actually, it is in a real three-dimensional problem where the additive manufacturing would be necessary for the introduction of voids inside the material to improve fatigue life.



*Keywords:* Fretting; voids; numerical model; cylindrical contact; palliative.

* Corresponding author. Tel.: +34-954481365.
*E-mail address:* cnp@us.es

1877-7058 © 2018 The Authors. Published by Elsevier Ltd.
Peer-review under responsibility of the scientific committee of the 7th International Conference on Fatigue Design.
10.1016/j.proeng.2018.06.001

## 1. Introduction

Fretting damage is a phenomenon that may occur on surfaces pressed together when subjected to cyclic loads. In addition to the bulk stress and the contact stresses produced by the pressure that keeps the surfaces together, components under the action of fretting are subjected to relative displacements. This combination leads to different types of damage such as wear, oxidation and the nucleation of cracks [1].

Various works shed light on different palliatives in fretting fatigue [2]. The palliatives can be divided between the ones that modify the geometry to in turn change the stress/strain fields, the ones that modify the properties of the material and surfaces (hardness, friction coefficient, etc.) or the ones that introduce residual stresses (shotpeening, etc). This paper focuses strictly on the modification of the geometry of the elements in contact so that it might be possible to mitigate, to the extent possible, the influence of fretting in crack initiation. This idea is linked to progress in additive manufacturing, thanks to which it can be made almost any type of geometry quickly, accurately and with a wide variety of metals, such as steel, aluminium and titanium.

However, this is a field that is developing and there are not many accurate data about the behaviour of the material. The fatigue behaviour of the material manufactured in the traditional way differs from the behaviour of the components created by additive manufacturing [3]. This is due to the method of manufacturing layer by layer, causing an apparent anisotropy to the component as well as the inclusion of unwanted defects, porosity and residual stresses. In general, the new material is not completely homogenous and must undergo heat treatments to improve their mechanical properties. However, there is no doubt that this method will be the most interesting manufacturing process in the very near future so it is important to study the benefits that offers. In addition, this method of manufacturing offers each day more competitive prices so soon will begin to gain foothold in the manufacturing market.

The literature usually differentiates the case where a bulk load is applied to one of the solids in contact and where there is not. As a first approximation to the technique proposed in this paper, the second case will be analysed, i.e. with no bulk load. In a usual fretting test of this kind, two surfaces are placed face to face, in this case one is cylindrical and the other completely flat. A constant normal load per unit thickness, *N*, is applied to the system. Then an oscillatory tangential load, *Q*, is applied as shown in Figure 1. Tests experience shows that cracks will always appear near the edge of the contact area. Based on these results, it is possible to introduce one or more holes in the vicinity of these areas, and thus modify strain and stress fields. So that it is possible to assess in which cases the introduction of those holes can be or not beneficial, facing an increase in fatigue life.

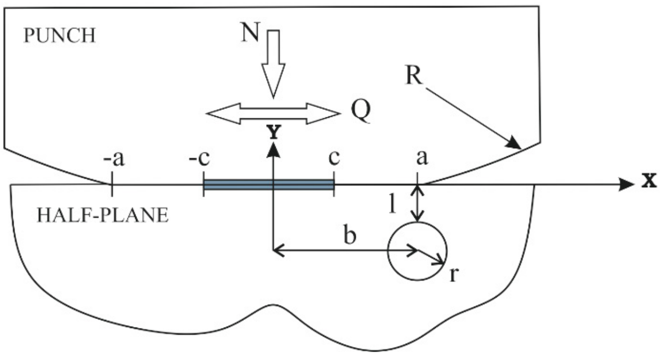


Figure 1. Geometry and parameters.

To analyze the behaviour of this new geometry the combination of parameters that are shown in Figure 1 are used, whose values are referred to the half-width of the theoretical contact area, $a$, which in the case between a cylinder and a half-plane is given by equation (1).

$$a^2 = \frac{8RN(1-\upsilon^2)}{\pi E} \tag{1}$$

Where, $R$ is the punch cylinder radius, $\upsilon$ the Poisson coefficient and $E$ the Young's modulus.

Using a finite element model, the fretting behaviour of different cases, in which the relative position of the hole and the conditions of contact have been changed, have been simulated. For this purpose, the following parameters have been studied (see Figure 1): the resulting ligament between the contact surface and the upper edge of the hole, $l$, the radius of the hole, $r$, and the horizontal distance of the hole, $b$. On other hand, some simulations have been performed for different load cases, i.e., for various combinations of $Q/\mu N$, parameter that governs the size of the stick zone according to equation (2).

$$\frac{c}{a} = \sqrt{1 - \left|\frac{Q}{\mu N}\right|} \tag{2}$$

Where $c$ is the half-width of the stick zone. The results here obtained open the doors to the study of a system subject also to bulk loads and 3D problems.

**Nomenclature**

| | |
|---|---|
| a | semi width of the contact zone |
| b | horizontal position of the hole |
| c | semi width of the stick zone |
| l | depth of the hole |
| r | radius of the hole |
| E | Young's modulus |
| N | normal load |
| Q | tangential load |
| R | radius of the punch |
| SWT | Smith-Watson-Topper parameter |
| ε | normal strain |
| ν | Poisson's ratio |
| σ | normal stress |

## 2. Finite element model description

There is an analytical study that analyses a contact pair similar to the considered [4]. This paper studies the influence of a hole and an inclusion, considering conditions of global sliding and a rigid punch. The conclusion is that the influence of both defects cause important alterations in the contact distribution of stresses and strains. The conditions studied here are different so it is not possible to compare this analytical method with the results shown in this paper, however, they support the initial idea that gives rise to this study.

There are two different models of simulations, Figure 2. One of them is the common case of the pair of contact consisting of a cylinder and a half-plane, which will be used later to compare the results with respect to cases with one hole. In addition, this simple model also is used to compare their results with those obtained by analytical models [5], thus certifying its validity. The other model reproduces the behaviour of the system for one hole. The models have been done assuring a coincident mesh in the contact zone in order to avoid numerical problems.

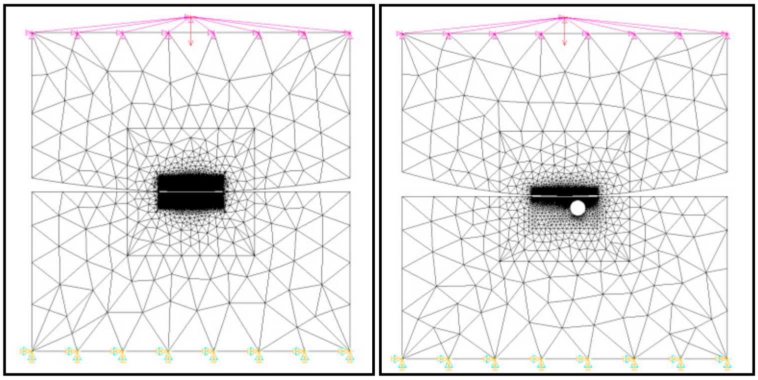

Figure 2. Finite element model mesh.

The models used are completely parametric so that the width and height of models are ten times the semiwidth of the contact zone, $a$, size that ensures a behaviour similar to a half-plane [6]. The contact pad radius, $R$, is 10 mm and the material is assumed to be a typical steel ($E$ = 210 GPa, $\nu$ = 0.3). The behaviour of the material has been considered elastic and linear in a state of plane-strain. This state has been selected to reduce the computational cost, as well as to compare the results of the numerical model with the analytic results from the case without hole.

The model is reproduced in three load steps. In the first one is applied a normal load, $-N$, keeping it constant for the remaining steps. In the second step a tangential load, $Q$, is applied to the left, and the third step applies the same load but in this case to the right to reproduce the conditions of a fretting test.

In the bottom line of the half-plane movements in both directions, as well as turns, are restricted. The punch boundary conditions vary from the first to the second load step. In the first step horizontal movements and turns are restricted. In the second and third load steps it is necessary to delete the horizontal displacement restriction, which is applied directly to a master node that controls the movement of all the top line of the punch.

Simulations have been made in the commercial software ANSYS™ 15, using for this purpose the APDL interface. The element used for modelling the components is the PLANE182, for the contact elements CONTA169 and TARGE171 have been used. The value of $N$, is fixed for all simulations in 500 N/mm, which gives a maximum pressure of contact, $p_0$, for the case without a hole, of 1354 MPa and a half-width contact zone of 0.235 mm. The half-width has been discretized with 150 elements, being the length of each element of 1.5 μm approximately. The total number of elements varies depending on the size of the hole and its position, from 40000 to 160000. Figure 3 shows mesh near hole.

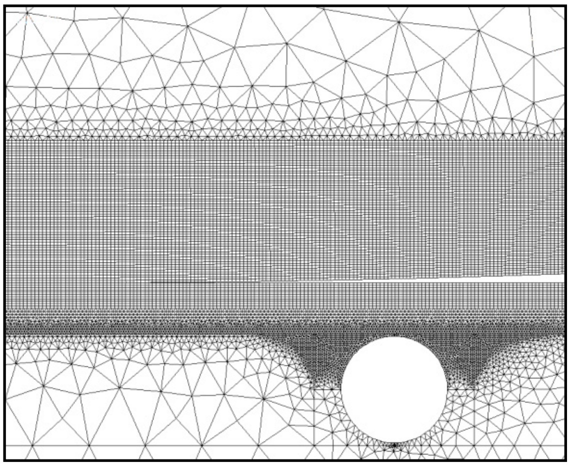

Figure 3. Finite element model, contact area and hole detail.

## 3. Fatigue method

To determine whether the influence of the hole is favourable, it is necessary to establish a comparison criterion between cases with hole and without it. So in this case it is used a parameter of multiaxial fatigue, the SWT parameter [7], which is shown in equation (3), but used with the procedure developed in [8].

$$\mathrm{SWT}=\left(\sigma^{n}\frac{\Delta\varepsilon}{2}\right)_{\text{máx}} \tag{3}$$

An outline of this method is shown in Figure 4. Firstly, with the results produced by a finite element model it is necessary to look for the most unfavourable point of the surface, which will be taken as that which produces the greatest value of the SWT parameter, evaluated only in the horizontal direction. This point will be considered to be the origin of the different lines that will be drawn. They are then plotted, distributed homogeneously between θ = 60 ° and θ = - 60 ° (one for each grade level), with a specified length, in this case 25μm. In this way the swept area is of the order of the microstructural typical size for a metal.

Then the SWT parameter is calculated along each of the lines at different points, but with the peculiarity that, for all these points, the orientation of the material plane considered to evaluate the parameter is not the critic one, i.e. which meets the equation (3), but is one that coincides with the orientation imposed by the θ orientation.

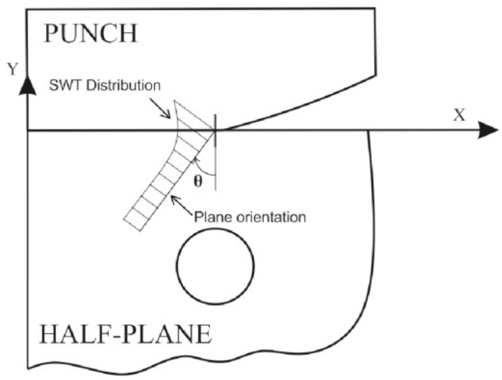


Figure 4. Crack initiation analysis procedure.

Once the parameter for each point are obtained, they are averaged for each of the considered material planes. The maximum average value from all levels will mark the more likely direction for the initiation of cracks. The result of this procedure is shown in Figure 5. Hereafter the SWT parameter references refer to the value of the parameter calculated by this method.

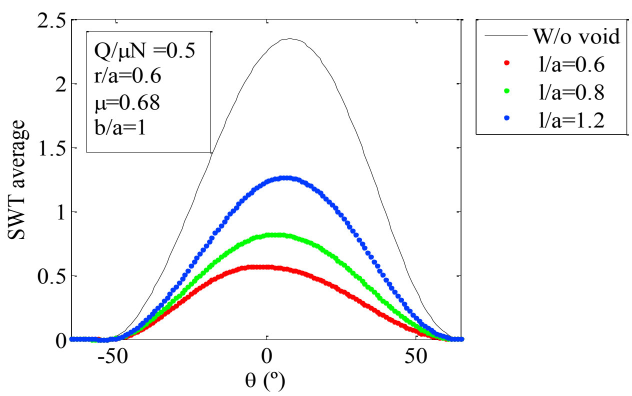


Figure 5. SWT average as a function of the orientation.

## 4. Simulation schedule

There have been done a series of simulations to check the improvement possibilities that can offer this method. First, the cases with one hole and two different friction coefficient are studied, being the first block of simulations. The values of geometric parameters for these simulations are taken from Table 1. Each of these simulations will be repeated for four different values of $Q/\mu N$, such as 0.2, 0.5, 0.8 and 1. As the value of the unit case is a total slide situation, it is convenient, to avoid problems of convergence, to use a value slightly smaller than the unit. Then the influence in the horizontal position, *b*, of the hole will be studied. This will be done once optimized values of the ligament and the radius of the hole are known, for each of the friction coefficients.

The friction coefficient is not constant throughout the test. Typically, in the case of steel, the friction coefficient at the beginning of the test is approximately 0.2, increasing then gradually to a value between 0.55 and 0.8 in the first loading cycles [9]. To check the influence of this parameter, two cases will be studied: 0.2 and 0.68. The latter is an intermediate value, to be able to reproduce approximately the real behaviour of the steel in a fretting situation.

Table 1. Simulation parameters

| | | | | | |
|---|---|---|---|---|---|
| **l/a** | 0.15 | 0.3 | 0.6 | 0.8 | 1.2 |
| **r/a** | 0.15 | 0.3 | 0.6 | 0.8 | |

## 5. Results

The results obtained in the simulations can be first separated into two groups: $\mu = 0.68$ and $\mu = 0.2$. Afterwards, once the size and depth of the hole have been optimized, the position *b* will be studied. To quantify the improvement of the geometry with the hole, the variation of the SWT parameter, calculated as in section 3, with respect to the case without a hole is calculated in each case, Equation (4).

$$\Delta\overline{SWT}_{\text{máx}} = \frac{(\overline{SWT})_{\text{máx}}^{\text{con a.}} - (\overline{SWT})_{\text{máx}}^{\text{sin a.}}}{(\overline{SWT})_{\text{máx}}^{\text{sin a.}}} 100 \tag{4}$$

### 5.1 Friction coefficient 0.68

Figure 6 shows the variation of SWT with different sizes, position and loads. Each figure corresponds to a different loading case, which in turn gives a different stick zone calculated with Equation 2 (Table 2).

Table 2. Simulation parameters

| ***Q/ μN*** | 1 | 0.8 | 0.5 | 0.2 |
|---|---|---|---|---|
| ***c/a*** | 0 | 0.45 | 0.71 | 0.89 |

Each of these figures show 20 different cases corresponding to the different combinations of the parameters in Table 1. Negatives values mean that the multiaxial fatigue parameter decreases with respect to the case without a hole and therefore fatigue life increases.

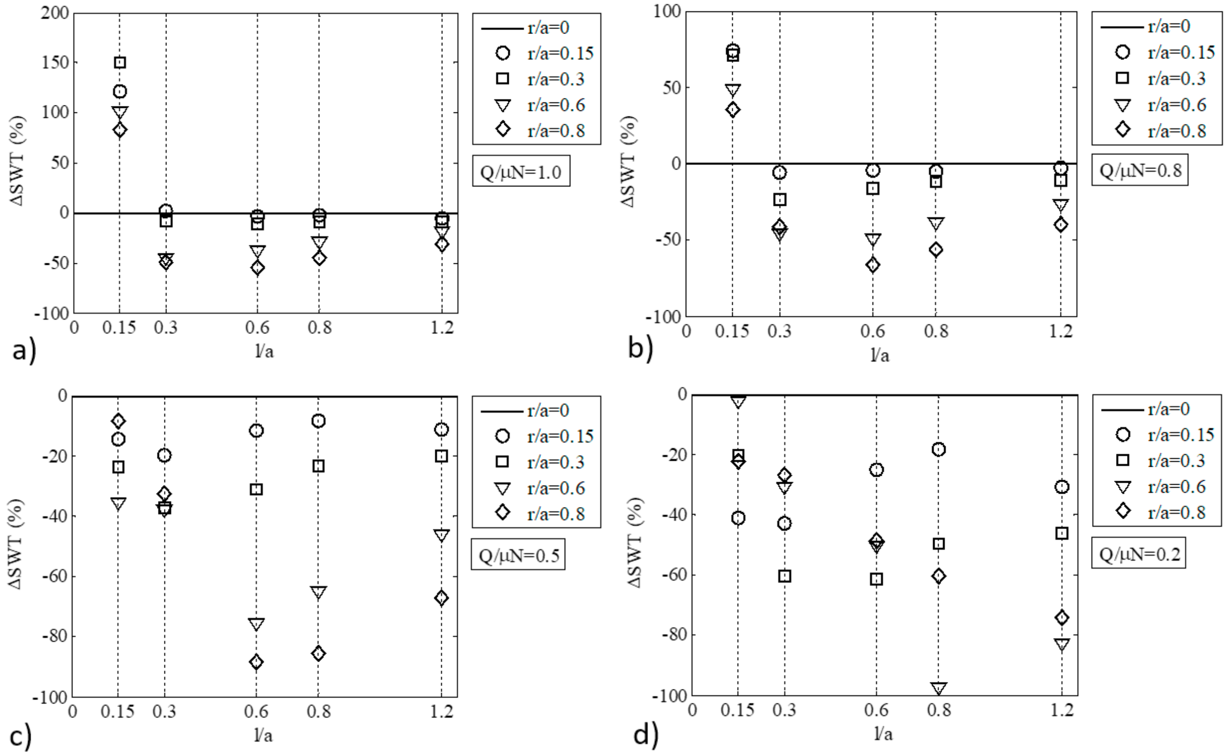


Figure 6. a) $\mu$=0.68 and $Q/\mu N$=1.0; b) $\mu$=0.68 and $Q/\mu N$=0.8; c) $\mu$=0.68 and $Q/\mu N$=0.5; d) $\mu$=0.68 and $Q/\mu N$=0.2.

In general, the parameter SWT decreases when the hole is far from to the surface and when it increases in size. Values of $r/a$ > 0.6 and $l/a$ > 0.6 are needed for a decrease in SWT higher than 20%. In some cases even a decrease of 90% can be achieved.

## 5.2 Friction coefficient 0.2

Figure 7 shows the results obtained for a lower friction coefficient. The behavior is different from the one shown in Figure 6, i.e., the friction coefficient has a considerable influence on SWT parameter and hence on fatigue life. Nevertheless, for designing purposes, a friction coefficient of 0.68 should be used since this would be the value during most of the fatigue life of the component.

## 5.3 Optimal horizontal position

This section analyzes the effect of changing the horizontal position, $b$, once the depth, $l$, and size, $r$, of the hole is chosen so that the reduction in SWT is maximum. To do this, the best solution for each loading case is selected and the hole is allocated at three different positions: $b/a$ = 1, $b/a$ = 0.8 and $b/a$ = 1.2. Figure 8 shows the results obtained for a friction coefficient of 0.68. There is not a clear trend but it seems that, in general, the best option would be to put the hole right below the edge of the contact zone.

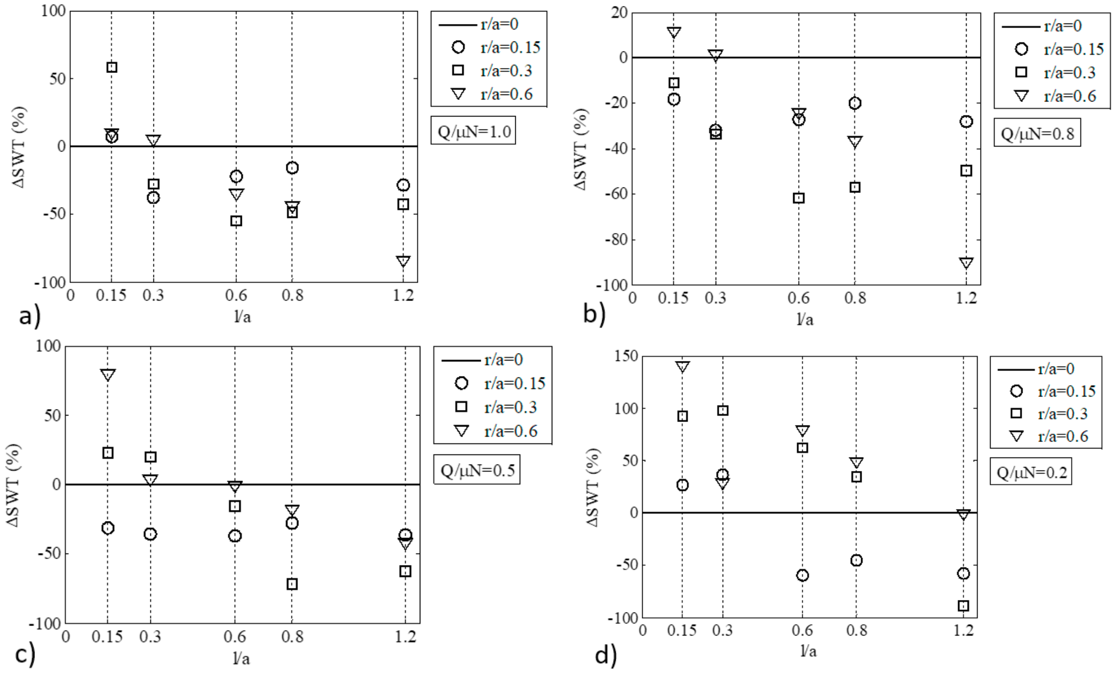


Figure 7. a) $\mu$=0.2 and $Q/\mu N$=1.0; b) $\mu$=0.2 and $Q/\mu N$=0.8; c) $\mu$=0.2 and $Q/\mu N$=0.5; d) $\mu$=0.2 and $Q/\mu N$=0.2.

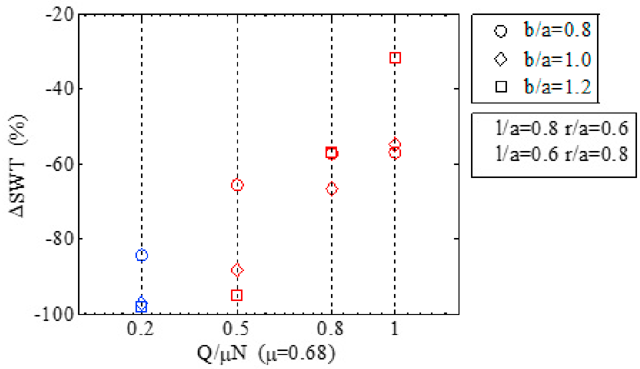


Figure 8. Influence of parameter $b/a$. $\mu$ = 0.68.

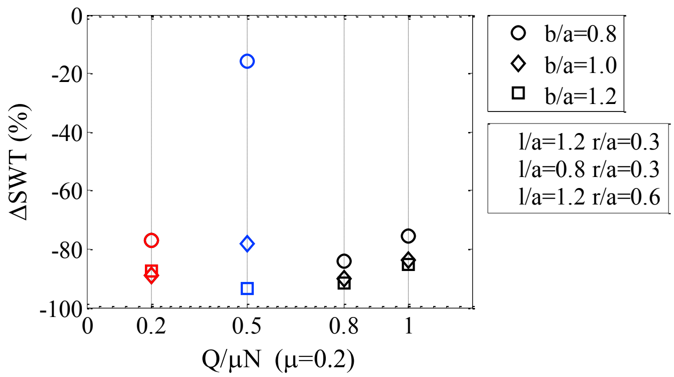


Figure 9. Influence of parameter $b/a$. $\mu$ = 0.2.

If the value of $\mu$ = 0.68 is chosen, there is not a clear trend as before, but the best choice would be *b/a* = 1.2, Figure 9.

### 5.4 Summary

As a summary of all the results shown in previous sections, Table 3 and 4 show the optimum values of the parameters and the reduction in the multiaxial fatigue parameter SWT.

Table 3. Optimum set of parameters for $\mu$ = 0.68.

<table>
<tr><th>Q/μN</th><th>0.2</th><th>0.5</th><th>0.8</th><th>1</th></tr>
<tr><td>l/a</td><td colspan="4" rowspan="2">≥ 0.6</td></tr>
<tr><td>r/a</td></tr>
<tr><td>b/a</td><td>[1-1.2]</td><td>[1-1.2]</td><td>1</td><td>[1-1.2]</td></tr>
<tr><td>l/r</td><td>[1.3-2]</td><td colspan="3">[0.75-1.3]</td></tr>
<tr><td>-ΔSWT (%)</td><td>[97-75]</td><td>[95-88]</td><td>[66-39]</td><td>[57-29]</td></tr>
</table>

Table 4. Optimum set of parameters for $\mu$ = 0.2.

<table>
<tr><th>Q/μN</th><th>0.2</th><th>0.5</th><th>0.8</th><th>1</th></tr>
<tr><td>l/a</td><td>0.15</td><td>0.6</td><td>1.2</td><td>1.2</td></tr>
<tr><td>r/a</td><td>0.6</td><td>0.3</td><td>0.6</td><td>0.6</td></tr>
<tr><td>b/a</td><td colspan="4">1.2</td></tr>
<tr><td>l/r</td><td>4</td><td colspan="3">2</td></tr>
<tr><td>-ΔSWT (%)</td><td>90</td><td>94</td><td>92</td><td>85</td></tr>
</table>

## 5. Conclusions

This paper shows that introducing a hole or a void inside the material may be beneficial for the fatigue life of a component subjected to fretting fatigue. This effect has been studied through the multiaxial fatigue criteria Smith-Watson-Topper. Different positions and sizes of the hole and different loads and friction coefficients have been studied, showing that not every option is valid. But for certain cases the reduction may be higher than 50% for all loading conditions.

Other influencing factors have not been considered, like the internal surface roughness of the hole that would appear from the fabrication process. Also, the mechanical and thermal treatments applied to the material may introduce some defects that have not been taken into account. All these should be introduced in the simulations in the future to obtain more realistic results.

## Acknowledgements

This research was supported by the Ministerio de Economía y Competitividad through the project DPI2014-59160-P.

## References

[1] J. Vázquez, C. Navarro, J. Dominguez, Experimental results in fretting fatigue with shot and laser peened Al 7075-T651 specimens, International Journal of Fatigue 40 (2012) 143-153.
[2] J. Domínguez, Algunas Consideraciones Sobre la Fatiga Por Fretting y Sus Paliativos. Anales de Mecánica de la Fractura 17 (2000) 89-102.
[3] A.Yadollahi, N. Shamsaei, S. M. Thompson, A. E. Linkan Bian, Effects of building orientation and heat treatment on fatigue behavior of selective laser melted 17-4 PH stainless steel, International Journal of Fatigue 94 (2017), 218–235.
[4] G. R. Miller, L.M.Keer, Interaction between a rigid indenter and a near-surface void or inclusion, J. Appl. Mech 50 (1983), 615-620.

[5] D. Nowell, D.A. Hills, Mechanics of fretting fatigue test, Int. J. Mech. Sci, 29 (1987) 355-365.
[6] L.J. Fellows, D. Nowell, D.A. Hills, Contact stresses in a moderately thin strip (with particular reference to fretting experiments), Wear 185 (1995) 235-238.
[7] R.N. Smith, P. Watson, A. Topper, A stress strain function for the fatigue of metals, Journal of Materials 5 (1970) 767–78.
[8] C. Navarro, J. Vázquez, J. Dominguez, Nucleation and early crack path in fretting fatigue, International Journal of fatigue 100 (2017) 602-610.
[9] J. Juolsukangas, A. Lehtovaara, A Comparison of relative displacement field between numerical predictions and experimental results in fretting contact, Journal of engineering tribology 230 (2016) 1273-1287.