

COB-2021-1227

METHODOLOGY VALIDATION FOR FATIGUE DESIGN OF POLYPROPYLENE COMPONENTS

Gabriel Scomparin Silva

Department of Refrigeration Technology, Whirlpool SA, Unidade de Joinville - R. Dona Francisca, 7200, Distrito Industrial, Joinville - SC, Brasil. CEP: 89219-900
gabriel.scomparin.silva@gmail.com

Ernesto Massaroppi Jr.

Department of Mechanical Engineering, University of São Paulo (USP), Campus São Carlos - Av. Trabalhador São-carlense, 400, Pq Arnold Schmidt, São Carlos - SP, Brasil. CEP: 13566-590
massarop@sc.usp.br

Abstract. *The structural design depends on the knowledge of the efforts acting on the structure and also on the knowledge of the mechanical properties and the behavior of the material that composes it. Polymers have been widely applied in the production of several items, including structural requirements. The mechanical properties of polymers are highly dependent on process parameters and their chemical composition. In addition, some properties of polymers, such as: creep, relaxation and fatigue, are quite expensive to obtain. Due to this factor, there is low availability of this information to the engineer, who needs to design a polymeric component under fatigue. Therefore, the objective of this work is to validate a method to design polypropylene components under fatigue, through material characterization, finite element analysis, fatigue life prediction and real component testing. The results obtained were satisfactory, the failure occurred in the region predicted in the finite element modeling, and the number of cycles was compatible with the estimate.*

Keywords: *fatigue life, fatigue design, polymer testing, polymer fatigue, finite element analysis, polypropylene.*

1. INTRODUCTION

In modern industry, product development needs to be carried out quickly, economically and robustly, in order to guarantee its competitiveness. Analytical solutions can be applied to analyze and design simple mechanical components. However, for non-trivial or complex mechanical components, analytical solutions become unfeasible. In these cases, some alternatives are experiments with real components and numerical methods (Budynas and Nisbett, 2016).

One of the techniques used to design and solve complex problems, including structural problems, is the finite element method (FEM). Through which it is possible to assess in great detail and with good precision isolated components or component assemblies (Filho, 2007).

The behavior of the materials that make up the structure, as well as their shape and acting loads affect the structural behavior. It is, therefore, extremely important to have accurate and reliable material properties data. Accurately obtaining the properties requires the use of experimental tests, as these are characteristics inherent to each material (Hibbeler, 2010).

Polymers revolutionized the application of materials, replacing materials traditionally used for the most diverse purposes (Callister, 2020). Versatility is one of the great characteristics of polymers. Among the various polymers available on the market, the second most used in the world is polypropylene (PP), with a wide range of applications (Braskem, 2015).

Some PP are used for piping, due to performance requirements these materials needs high crack resistance. To ensure safety is necessary to characterize the crack growth behavior in the quasi-brittle failure regime. For some modern and optimized PP the characterization considering the quasi-brittle behavior may take an impractical time. One possibility is to use fatigue loads instead of static to accelerate testing (Arbeiter et al., 2016).

The increasing consumption of polymers leads to increasing of plastics disposal, consequently its affect negatively our environment. To minimize the effects of polymers in environment, recycling is considered one of the best solution. The PP is considered one of the most important polymer in the world, additionally has good assessed recycling process. It's generate the necessity of studying the fatigue behavior of recycled polypropylene (Abdelhaleem et al., 2017).

One promising material considered for the manufacturing of hybrid composites is the PP fiber. Many industrial applications, such as: marine, military, automotive, aerospace, etc., have been successfully used fiber-reinforced plastic composites (FRPCs). During their common application FRPCs are widely subjected to cyclic loading and fatigue is considered the main cause of failure in composites (El-Baky and Attia, 2018).

Some properties of polymers are quite costly to obtain, such as: creep, relaxation and fatigue. Due to this factor, there is low availability of this information to the engineer who needs to design a polymeric component subjected to fatigue. Therefore, the objective of this work is to validate a method for dimensioning polypropylene components subjected to variable loads, through material characterization, finite element analysis, fatigue life prediction and real component testing.

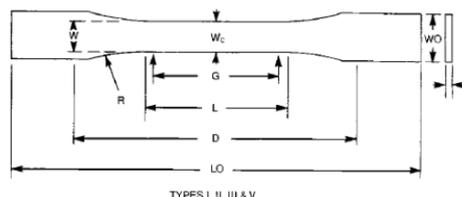
2. METHODOLOGY

2.1 Definition of process parameters and injection molding of specimens for tensile testing

The simulation of finite element injection and the injection process parameter chart of the component under study were used, with the objective of reproducing in the specimens approximately the same conditions that occur in the critical region of the component under study. The critical region was defined in a previous structural analysis of the component, through the FEM. The Moldflow 2014® software was used to model the part and its injection system; the process parameters obtained from the component's process parameters chart were considered as model input data. The critical region was monitored over time in the model, and the maximum local pressure was obtained. Subsequently, the specimen with its respective injection system was modeled. The injection pressure was varied until obtaining in the center of the specimen the same maximum pressure obtained for the critical region of the component.

2.2 Tensile test of PP specimens

The tensile test was carried out in accordance with ASTM D638 (ASTM, 2003), considering a displacement speed of 5 mm/min, as it is compatible with the load application rate to which the real component is submitted. An MTS Sintech 10/D machine was used, equipped with a 4.9 kN load cell, model S, code 17156, supplied by Panantec. An MTS 632.11F-50 extensometer was attached to the body of specimen. Type I specimens were used in accordance with ASTM D638 (ASTM, 2003), the specimens dimension are specified in Figure 1.



TYPES I, II, III & V
 Specimen Dimensions for Thickness, T , mm [in.]^A

Dimensions (see drawings)	7 [0.28] or under		Over 7 to 14 [0.28 to 0.55], incl		4 [0.16] or under		Tolerances
	Type I	Type II	Type III	Type IV ^B	Type V ^{C,D}		
W—Width of narrow section ^{E,F}	13 [0.50]	6 [0.25]	19 [0.75]	6 [0.25]	3.18 [0.125]	±0.5 [±0.02] ^{B,C}	
L—Length of narrow section	57 [2.25]	57 [2.25]	57 [2.25]	33 [1.30]	9.53 [0.375]	±0.5 [±0.02] ^C	
WO—Width overall, min ^G	19 [0.75]	19 [0.75]	29 [1.13]	19 [0.75]	...	+ 6.4 [+ 0.25]	
WO—Width overall, min ^G	9.53 [0.375]	+ 3.18 [+ 0.125]	
LO—Length overall, min ^H	165 [6.5]	183 [7.2]	246 [9.7]	115 [4.5]	63.5 [2.5]	no max [no max]	
G—Gage length ^I	50 [2.00]	50 [2.00]	50 [2.00]	...	7.62 [0.300]	±0.25 [±0.010] ^C	
G—Gage length ^I	25 [1.00]	...	±0.13 [±0.005]	
D—Distance between grips	115 [4.5]	135 [5.3]	115 [4.5]	65 [2.5] ^J	25.4 [1.0]	±5 [±0.2]	
R—Radius of fillet	76 [3.00]	76 [3.00]	76 [3.00]	14 [0.56]	12.7 [0.5]	±1 [±0.04] ^C	
RO—Outer radius (Type IV)	25 [1.00]	...	±1 [±0.04]	

Figure 1. Specimen dimensions according to the respective types (ASTM, 2003).

2.3 Tensile fatigue test of PP specimens

A universal machine MTS® model Bionix, equipped with hydraulic grips and a 15 kN load cell, was used to perform the fatigue test in tensile. The test was performed in accordance with ASTM D7791 (ASTM, 2012), with a frequency of 1 Hz and $R = 0.1$. Four levels of stress amplitude were considered, being: 13.05 MPa, 11.50 MPa, 10.90 MPa and 10.40 MPa. Type I specimens were used, according to ASTM D638 (ASTM, 2003). Data of time, elongation, strain, load, stress and number of cycles were monitored and stored. Three specimens were tested for each stress amplitude level, with the exception of the lowest stress amplitude level, for which only 2 specimens were tested due to mechanical failure in the testing machine, which made the continuity of the tests unfeasible.

2.4 Component torsion structural simulation

The model of the component under study was built considering a mesh density of at least 3 elements along its thickness, using second order tetrahedral elements, called "Solid 92" in the elements library of the Ansys 17® software.

The multilinear elastic material model was considered, and the stress curve as a function of the strain obtained from the tensile test was provided. The lower shaft coupled to the component and responsible for the transmission of torque was simplified, "pipe 16" elements were used for its modeling and typical properties of steel were considered. The upper (static) shaft was removed from the model, translation constraints in the 3 directions (u_x , u_y and u_z) were opportunely applied at the interface points between the shaft and the specimen. In the central node of the "pipe 16" elements, (representing the lower shaft) a torque of 87.75 N.m was applied (estimated value for the component to reach the yield stress), in the positive direction of the y axis (vertical axis of the model). A model for large displacements was considered and a total of 9 loading substeps was configured for the solution.

2.5 Measurement region definition and component instrumentation for the torsion test

The stress levels along with the component, obtained through finite element simulation of the component submitted to the load corresponding to the torsion test, were evaluated. Were tried to find critical regions in terms of tension. After determining the most critical regions, it was evaluated in which of them it would be possible to glue the set of electric strain gauges in the form of a rosette. The chosen region, for having presented the second-highest level of tension and allowing access for gluing the rosette, is shown in figure 2.

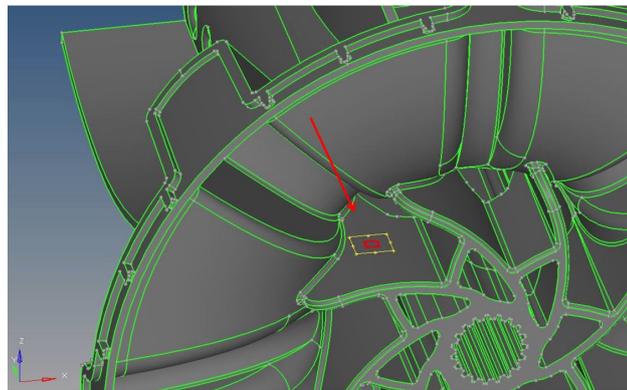


Figure 2. Rosette glue region.

The rosette model 1-RY98-3/350 manufactured by HBM was used, due to its small size and being suitable for measuring strain in polymers. The bonding surface on the component was sanded using fine sandpaper with 2000 grain, in order to guarantee the adhesion of the adhesive. After the surface was sanded, the rosette was positioned in the bonding region with the aid of adhesive tape. The sensor was removed from the bonding area, but the adhesive tape reference was maintained. Then, Super Bonder® adhesive was applied to the component's bonding surface, the rosette was positioned again with the aid of adhesive tape and the bonding process was carried out.

The electrical strain gauges were connected to the acquisition system through 15-pin connectors, one being used for each strain gauge. Figure 3 shows the rosette glued to the component and its electrical connections.

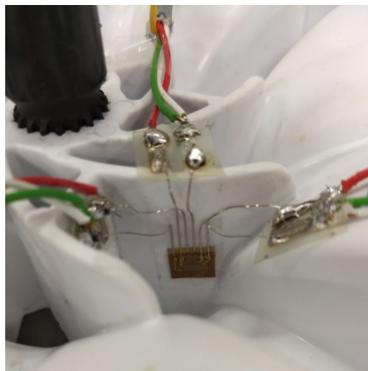


Figure 3. Rosette glued to the component and its electrical connections.

2.6 Component torsion test

A torsion machine, model MT2-E1, from the manufacturer Instron® was used. Two adapter shafts were used to enable the assembly of the component to the testing machine, these were coupled to the component and, later, the set

was assembled on the testing machine. The maximum torque was set to 87.75 N.m with a load application rate of 16.3 N.m/s, to represent approximately the same strain rate to which the tensile specimens were subjected. The strain acquisition system was not synchronized with the machine's torque acquisition system, and manual synchronization was performed later. With this objective, the machine was configured to maintain the maximum torque for 10 seconds, in order to generate a torque and deformation threshold for synchronization, the unloading was performed after this period with the same load application rate.

The acquisition rate was set to 50 Hz, with data of torque, angular position, and time being saved. Figure 4 shows the acquisition system (A) - composed of the data acquisition device (Spider - HBM) and a laptop, the testing machine (B) and the component under study (C) fixed to the machine.

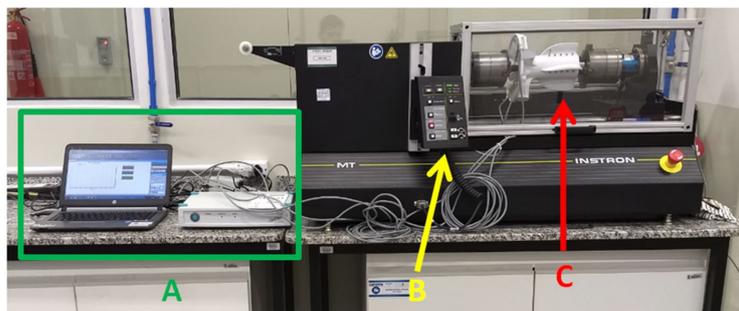


Figure 4. Acquisition system (A), testing machine (B) e component under study (C).

2.7 Definition of stress and torque level for life of 30,000 cycles in component fatigue test

Although in the real application of the component, the expected life is millions of cycles, for the purpose of methodology validation a smaller number of cycles will be used. A life of 30,000 cycles was defined in order to validate the method for high cycle fatigue and to make the tests possible economically. The determination of the necessary stress level for the failure to occur in 30,000 cycles was calculated using the Weibull model, through the JMP 13® software. The results of the tensile fatigue test were used as input to the model, considering the number of cycles to failure in relation to the applied stress amplitude and the occurrence or not of failure for the applied stress amplitude.

After obtaining the required stress amplitude for component failure in 30,000 cycles, from the failure probability analysis, the corresponding maximum stress was calculated, considering $R = 0.1$. It was decided to use the von Mises equivalent strain to allow direct comparison between the results measured by strain gauges and those calculated by the simulation.

The von Mises equivalent strain, present in the critical region of the component model, was compared with the strain required for failure in 30,000 cycles. The torque was increasingly applied to the model, until the necessary von Mises equivalent strain was reached, thus obtaining the torque required to lead the component to failure.

2.8 Spline joint strength verification and component adjustment for torsional fatigue testing

Due to the simplification adopted in the modeling of the spline joint, for safety, it was decided to verify the strength of the joint by analytical calculation. The contact pressure and the shear stress as a result of the torque application were calculated. Calculations were performed using eq. (1), (2), (3) and (4).

$$p = \frac{M_t}{\alpha \cdot h \cdot L \cdot i \cdot r_m} \quad (1)$$

$$r_m = \frac{d_1 + d_2}{4} \quad (2)$$

$$h = \frac{d_2 - d_1}{2} \quad (3)$$

$$\tau = \frac{F}{A_{cis}} \quad (4)$$

Where, p: contact pressure; M_t : Torsion moment; α : Factor related to the not perfectly simultaneous support of the splines; L: Length of spline; i: Number of splines; d_1 : Diameter of spline base; d_2 : Diameter of top of spline; τ : Shear stress; F: Normal strength; A_{cis} : Shear area.

In the calculation of the shear stress, a safety factor of 3 was considered, with 5.95 MPa being considered as an admissible value. The contact pressure was then calculated from the maximum allowable torque and dimensions of the spline joint. After performing the calculations, it was concluded that there was a risk of joint failure with the application of the required torque level for component failure in 30,000 cycles.

Thus, it was decided to adjust the component in order to reduce the torque required for failure. For this purpose, the minor ribs (indicated in red) and 4 main ribs (indicated in yellow) were removed. In figure 5 (a) the original cube is shown, with the indication of the main and secondary ribs, as well as the altered cube of the component (b) is shown.

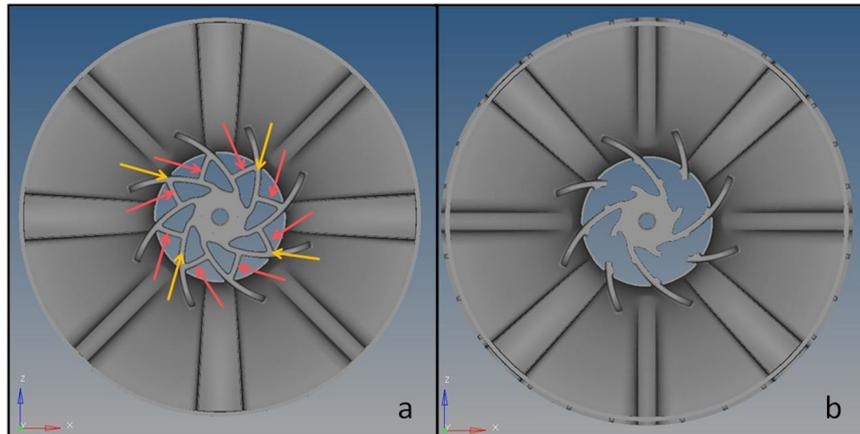


Figure 5. (a) original hub with the indication of the main ribs in yellow and the minor ribs in red, to be removed; (b) final configuration of the component's hub.

2.9 Component torsion fatigue test

Proceeded for the fixation of the real component to the testing machine, in order to carry out the fatigue test, in the same way as for the torsion test, mentioned in section 2.6. Subsequently, the testing machine was configured to apply a maximum torque of 39 N.m, $R = 0.1$, frequency of 1 Hz and 35° as the maximum torsion angle, this angle was considered as a failure criterion. The machine was programmed to run 5,000 cycles per step (due to memory limitation of the data acquisition system). The steps were repeated until the failure occurred. Torque, torsion angle and time data were stored.

3. RESULTS

3.1 Process parameters for injection molding of specimens for tensile testing

The parameters obtained, through the finite element model, for the injection of tensile and fatigue specimens are presented in table 1.

Table 1. Injection Parameters for the traction and fatigue specimens.

Parameter	Value	Unit
Melt Temperature	220	°C
Injection Gate Temperature	235	°C
Injection Pressure	48.1	MPa
Injection Time	2.8	s
Injection Flow Rate	28	cm ³ /s
Packing Pressure	48.1	MPa
Mold Temperature	30	°C
Packing Time	8.0	s

3.2 Tensile test of PP specimens

The polypropylene used for the study showed a ductile behavior, linearity in the stress curve as a function of strain up to approximately 15 MPa stress and non-linearity above this value. The polymer had an accentuated nonlinear behavior close to the yield point (considered "offset": $\epsilon = 0,05$). There was a great overlap between the curves of the

tested specimens, which indicates a good stability in the injection process of the specimens. The operating limit of the strain gauge was reached during the test, at which time the test was stopped. The stress as a function of strain curves resulting from the test can be seen in Figure 6. The results of the modulus of elasticity (E) and yield stress (σ_e) for each specimen are shown in table 2.

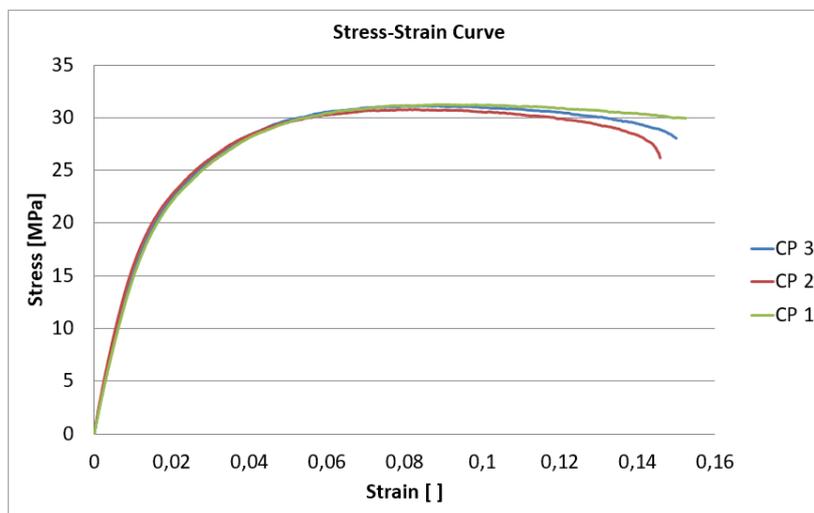


Figure 6. Curves stress as a function of strain for tensile testing of PP specimens.

Table 2. Results of the modulus of elasticity (E) and yield stress (σ_e) for the PP tensile test.

	E[MPa]	σ_e [MPa]
Specimen 1	1583	31.25
Specimen 2	1786	30.82
Specimen 3	1621	31.12
Mean	1663	31.06
Standard Deviation	108	0.22

3.3 Tensile fatigue test of PP specimens

The tensile fatigue test results for the polypropylene specimens are shown in figure 7 and table 3.

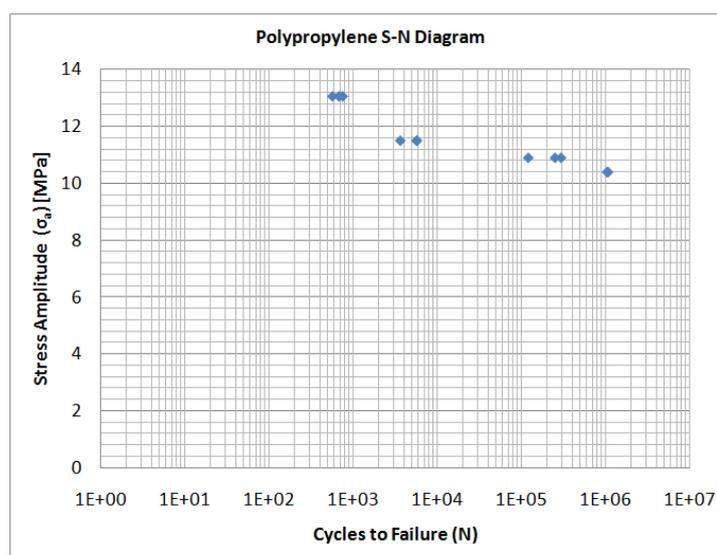


Figure 7. S-N diagram for the tensile fatigue test of polypropylene specimens.

Table 3. Results of the tensile fatigue test of polypropylene specimens.

Specimen	σ_a [MPa]	σ_{max} [MPa]	Cycles to Failure
1	13.05	29.00	750
2	13.05	29.00	680
3	13.05	29.00	570
4	11.50	25.56	5691
5	11.50	25.56	3626
6	11.50	25.56	5664
7	10.90	24.22	118575
8	10.90	24.22	289820
9	10.90	24.22	246384
10	10.40	23.10	1020919
11	10.40	23.10	1044583

3.4 Component torsion structural simulation

The maximum principal strain results for the structural model are shown in Figure 8. The highest level of strain occurs in the region close to the hub (A). The rosette gluing region (B) has the second highest level of deformation.

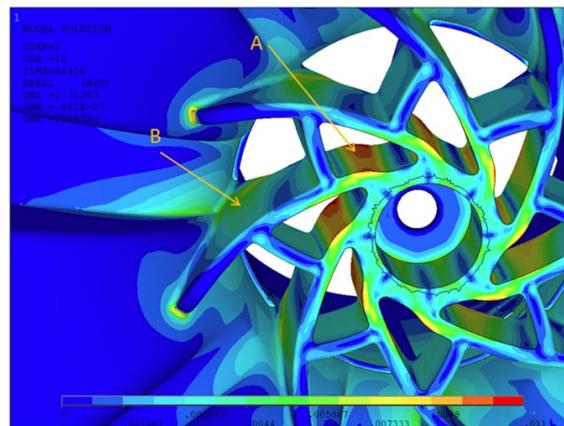


Figure 8. Results of maximum principal strain for the torsion test structural model.

3.5 Component torsion test

The results of torque as a function of time for the torsion test are shown in figure 9. The loading and unloading happened practically linearly in relation to the time, in the stabilization period the testing machine reached the previously configured torque level, having been maintained the same torque level throughout the stabilization period. The results of maximum principal strain as a function of torque are shown in figure 10. During loading and unloading, the behavior is practically linear. In the stabilization region, the torque is kept constant, but the strain presents an increase, possibly due to the phenomenon of creep or localized plastic strain. The component showed residual strain after the complete removal of the torque.

3.6 Comparison of test specimen torsion test results with the results of the structural model

The maximum principal strain obtained from the structural model, for the region of gluing of the strain gauge was 0.0047020 when applying a torque of 84 N.m. The component in the tensile test, when also submitted to a torque of 84 N.m, presented a maximum principal strain of 0.0056167.

3.7 Definition of stress and torque level for life of 30,000 cycles in component fatigue test

The result of the failure probability analysis, considering the Weibull model, showed that for a stress amplitude of 12.3 MPa the probability of failure in 30,000 cycles is 99.999%. Thus, a stress amplitude of 12.3 MPa was considered for the performance of the torsion fatigue test.

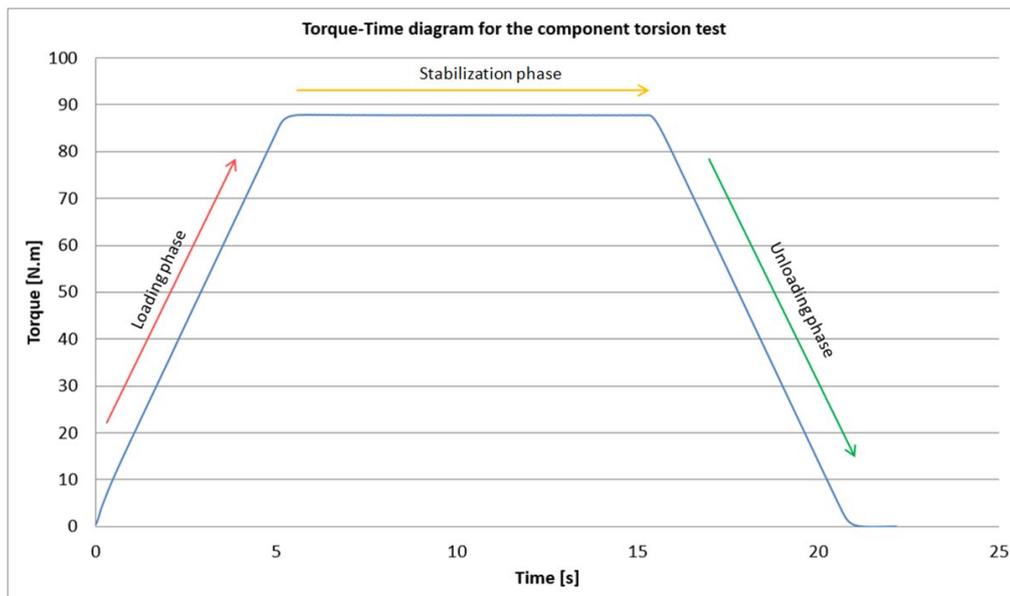


Figure 9. Torque as a function of time diagram for the component torsion test.

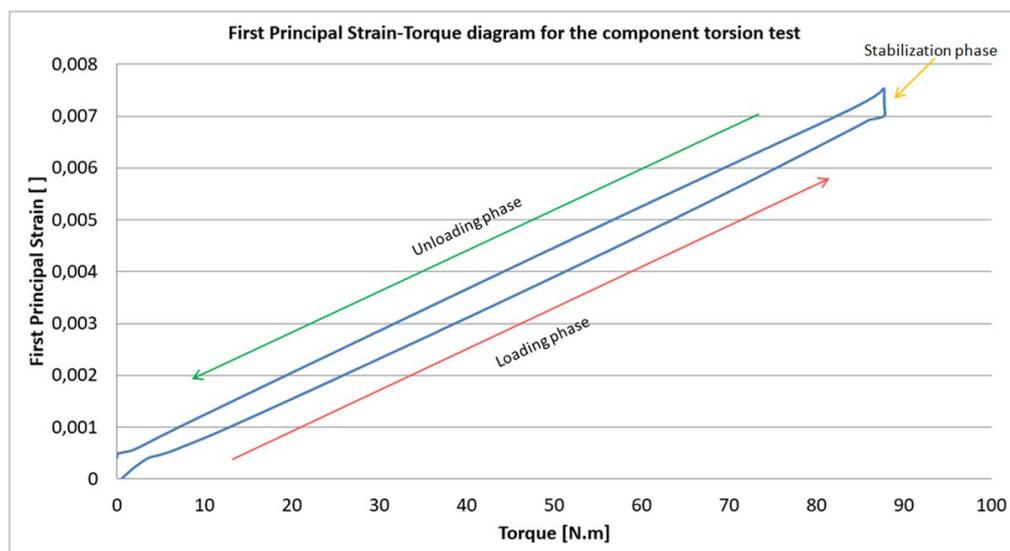


Figure 10. Maximum principal strain as function of torque for the component torsion test.

The maximum stress was calculated from eq. (5) and eq. (6), for a stress amplitude of 12.3 MPa and $R = 0.1$, the calculated maximum stress was 27.33 MPa. A strain of 0.03195 was obtained from the stress-strain curve (from the material characterization) for a stress of 27.33 MPa, to allow direct comparison with the results of the structural model.

$$\sigma_a = \frac{\sigma_{max} - \sigma_{min}}{2} \tag{5}$$

$$R = \frac{\sigma_{min}}{\sigma_{max}} \tag{6}$$

The strain level was compensated in order to consider the divergence of 16.3% found between the structural model and the experimental value. For this purpose, the strain obtained from the stress-strain curve for the stress level of 27.33 MPa was multiplied by 0.84, and the corrected strain value was 0.02684.

Through the structural model, the torque was varied until the desired von Mises equivalent strain value (0.02684) was reached, and a torque of 38.6 N.m was determined to perform the torsion fatigue test.

3.8 Component torsion fatigue test

The 3 components submitted to the fatigue test failed in the same regions, which were identified as critical in the finite element structural model. The results are shown in Table 4.

Table 4. Results for the component torsion fatigue test.

Specimens	Cycles to Failure	Torsion Angle [°]
Specimen - F1	35,065	35
Specimen - F2	21,712	35
Specimen - F3	62,417	35
Mean	39,731	-
Standard Deviation	20,749	-

The 35° angle was adopted as a failure criterion, because after this level of torsion, the component abruptly loses strength. Figure 11 shows the specimen number 1. After performing the test, it is possible to notice damage in the critical region, change in the color of the material in the critical region, reduction in section and the presence of a small crack in the indicated region. In figure 12 it is possible to see a top view of the specimen, where the section reduction in the critical region is highlighted.



Figure 11. Specimen 1 after torsion fatigue test.

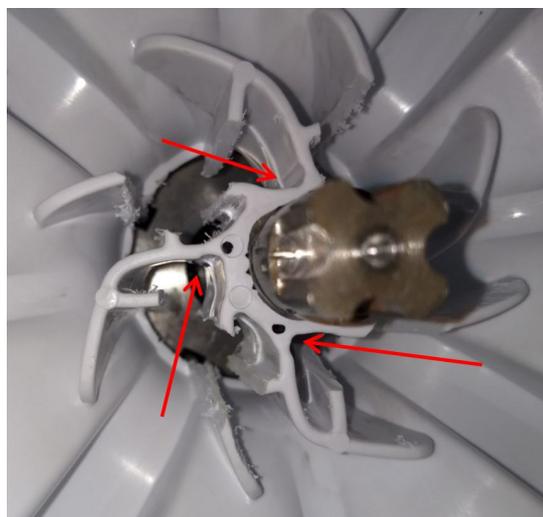


Figure 12. Specimen 1 top view after torsion fatigue test.

4. CONCLUSION

The method used was able to predict with high accuracy the location of the fatigue failure and with good accuracy the number of cycles to failure, occurring in the same order of magnitude as predicted. Thus, as long as the finite element structural model is representative and the mechanical properties of the material are known, it is possible to predict with good precision the fatigue life of a given polypropylene component.

5. ACKNOWLEDGEMENTS

To the Postgraduate Program in Mechanical Engineering at the School of Engineering of São Carlos (EESC) of the University of São Paulo (USP). To Whirlpool SA for the supply of materials, laboratories and financing of the specimens and, jointly, to the Applied and Computational Mechanics Laboratory of the Mechanical Engineering Department of the School of Engineering of São Carlos/USP for the collaboration with software, hardware, licenses and standards to carry out the studies. To SENAI from Joinville-SC for providing the equipment to carry out the torsion test.

6. REFERENCES

- Abd, El-Baky. M., Attia, M., "Flexural fatigue performance of hybrid composite laminates based on E-glass and polypropylene fibers". Journal of Thermoplastic Composite Materials. 2019;32(2):228-247. doi:10.1177/0892705717751020
- Abdelhaleem, AM., Megahed, M., Saber, D., "Fatigue behavior of pure polypropylene and recycled polypropylene reinforced with short glass fiber". Journal of Composite Materials. 2018;52(12):1633-1640. doi:10.1177/0021998317729888.
- Arbeiter, F. J., Frank, A. and Pinter, G., 2016. "Influence of molecular structure and reinforcement on fatigue behavior of tough polypropylene materials". J. Appl. Polym. Sci., 43948. doi: 10.1002/app.43948.
- ASTM D638-03, *Standard Test Method for Tensile Properties of Plastics*, ASTM International, West Conshohocken, PA, 2003, www.astm.org
- ASTM D7791-12, *Standard Test Method for Uniaxial Fatigue Properties of Plastics*, ASTM International, West Conshohocken, PA, 2012, www.astm.org.
- Braskem, 2010. *Braskem works cited: Polipropileno Verde: Relatório Anual e de Sustentabilidade 2010*. <http://www3.braskem.com.br/upload/rao/2010/pt/polipropileno-verde.html>. Accessed 30 march 2015.
- Budynas, R.G. and Nisbett, J.K., 2016. *Shigley's Mechanical Engineering Design*. Ed. Mc Graw-Hill, New York.
- Callister, W.D.Jr., 2020. *Materials Science and Engineering: An Introduction*. Ed. LTC, Rio de Janeiro.
- Filho, A.A., 2007. *Finite Elements – The base of the CAE technology*. Ed. Érica, São Paulo.
- Hibbeler, R.C., 2010. *Mechanics of Material*. Ed. Pearson Prentice Hall, São Paulo.

7. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.