

Universidade do Minho
Escola de Engenharia

Cedrico de Resende Oliveira

**Study of the behavior of a thermoplastic
injection mold and prediction of fatigue
failure with numerical simulation**

Tese de Doutoramento
Doutoramento em Engenharia Mecânica

Trabalho efetuado sob a orientação do(a)
Professor Doutor José Filipe Bizarro Meireles
Professor Doutor Nuno Ricardo Maia Peixinho

Agradecimentos

Gostaria de agradecer a todos aqueles que de alguma forma contribuíram para a realização deste trabalho.

Em primeiro lugar, aos meus orientadores, Professor Doutor José Filipe Bizarro Meireles e Professor Doutor Nuno Ricardo Maia Peixinho, pelo apoio e disponibilidade demonstrados ao longo destes anos. As inúmeras conversas e partilha de ideias que tivemos enriqueceram os meus conhecimentos e forma de pensar, trazendo-me novas formas de abordar os desafios do meu dia-a-dia.

Ao Grupo Simoldes, por disponibilizar os meios necessários para a realização deste trabalho. Queria agradecer em especial ao Engenheiro José Velhas, por sempre ter acreditado no meu trabalho e me ter apoiado na realização do mesmo.

À minha família, por estar incondicionalmente ao meu lado, nos bons e maus momentos. Também por sempre terem contribuído de forma construtiva ao longo de toda a minha vida académica e profissional.

À Catarina, que partilhou comigo muitos momentos e nos períodos mais difíceis foi capaz de me mostrar o que já tinha conquistado. Um grande obrigado pela paciência e por me ter acompanhado nesta caminhada com um sorriso na cara.

Aos meus amigos:

- Eduardo Pereira, Rui Barbosa e Vitor Pais, que começaram por ser colegas de trabalho e com o tempo se tornaram bons amigos, sempre disponíveis para ajudar e capazes de transformar momentos aborrecidos em boas horas de diversão.
- Ao João Xará, que me tirou da frente dos livros e do computador quando eu estava mais desgastado. Essas horas de convívio contribuíram bastante para o meu bem-estar.
- À Rosana, por sempre ter uma palavra motivacional a dar.
- A todos os restantes amigos, com quem passei menos tempo durante a realização deste trabalho. Um muito obrigado pela compreensão e apoio.

STATEMENT OF INTEGRITY

I hereby declare having conducted this academic work with integrity. I confirm that I have not used plagiarism or any form of undue use of information or falsification of results along the process leading to its elaboration.

I further declare that I have fully acknowledged the Code of Ethical Conduct of the University of Minho.

Resumo

O objetivo deste trabalho é a criação de uma metodologia de análise da resistência à fadiga de moldes de injeção de termoplásticos. Uma metodologia capaz de satisfazer o mercado atual que exige a diminuição do tempo de entrega e custos de moldes de injeção, sem comprometer a sua fiabilidade.

Para o desenvolvimento desta metodologia, foram utilizados modelos digitais. Com estes modelos é possível executar-se várias iterações sem os custos de um modelo físico. Além do menor custo dos modelos digitais, também é possível compreender o comportamento de cada molde no decorrer da fase de projeto.

Com o aumento da complexidade dos componentes injetados, o estudo da resistência à fadiga tende a ser cada vez mais importante. Neste trabalho serão apresentados cuidados a ter na preparação dos modelos digitais, de forma a obter-se resultados fiáveis.

No desenvolvimento desta metodologia, usaram-se dois softwares de simulação numérica para gerar os modelos digitais. Um deles dedica-se ao estudo reológico de peças termoplásticas e outro ao comportamento estrutural dos moldes de injeção.

A execução de simulações numéricas requer uma boa caracterização dos materiais usados. No caso dos termoplásticos, os fabricantes têm uma grande base de dados com a informação necessária para as simulações numéricas. No entanto, para as simulações estruturais, os fabricantes tendem apenas a fornecer os dados das curvas monotónicas, os quais não fornecem qualquer informação sobre o comportamento à fadiga. Portanto, neste trabalho foram estudados modelos empíricos que se adaptam aos aços usados em moldes de injeção, a partir dos quais é possível gerar as curvas S-N e ε -N. De modo a avaliar qual o modelo empírico que se adaptaria melhor a esta área, foram realizados ensaios experimentais com provetes feitos em EN 1.2311. A partir destes ensaios, escolheu-se o modelo empírico mais conservador. Com base no modelo empírico escolhido, foi desenvolvida uma aplicação capaz de gerar as curvas S-N e ε -N, a partir das informações fornecidas pela aciaria.

Além da caracterização dos materiais, também é importante que as condições de carregamento do modelo numérico estrutural sejam o mais aproximadas possível do que irá ocorrer no modelo físico. Como as cargas deste modelo numérico podem ser previstas a partir do modelo numérico reológico, a criação de uma ponte entre estes dois modelos numéricos é imprescindível. Logo, neste trabalho foi construída uma aplicação capaz de converter os dados gerados pelo software comercial Moldflow em ficheiros capazes de serem lidos por softwares comerciais de simulação numérica estrutural. Usando esta aplicação para a conversão dos dados, foram realizadas simulações e comparadas com os

respetivos modelos físicos. Verificou-se que é possível replicar o comportamento do molde em modelos digitais. No entanto, os modelos digitais dos moldes de injeção estudados tenderam a apresentar resultados conservadores quando comparados com os modelos físicos.

Por fim, foi desenvolvida uma aplicação capaz de usar dados calculados a partir de softwares comerciais de cálculo numérico estrutural para a determinação da resistência dos moldes à fadiga. Aqui foi tido em conta o modelo para geração das curvas de fadiga dos materiais validado. Os modelos de cálculo à fadiga na aplicação baseiam-se na regra de Palmgren – Miner para a determinação dos ciclos até à nucleação da fissura. O cálculo das tensões alternadas foi realizado a partir de dois métodos, o critério da tensão de corte octaédrica e o método de Sines. Para testar a aplicação foram escolhidos cinco moldes que apresentaram falhas por fadiga. Em seguida, foi aplicada a metodologia proposta neste trabalho para a determinação da resistência dos mesmos à fadiga. A partir da aplicação desta metodologia e das ferramentas desenvolvidas para o seu emprego, foi possível verificar que esta é capaz de prever as zonas onde ocorreram as falhas, bem como outras com probabilidade de nucleação de fissuras.

Portanto, no decorrer deste trabalho foi possível criar uma metodologia e ferramentas de apoio para o cálculo de moldes à fadiga. Assim, projetistas de moldes podem ter uma boa perspetiva da resistência à fadiga de moldes de injeção ainda em projeto, tendo por base métodos científicos.

Palavras-chave: Fadiga de metais; Moldes de injeção; Análise estrutural; Simulação numérica; Caracterização de metais.

Abstract

The objective of this work is to create a methodology to analyze the fatigue resistance of thermoplastic injection molds. A methodology capable of satisfying the current market that demands a decrease in the delivery time and costs of injection molds, without compromising their reliability.

To develop this methodology, digital models were used. With these models it is possible to execute several iterations without the costs of a physical model. Besides the lower cost of digital models, it is also possible to understand the behavior of each mold during the design phase.

With the increasing complexity of injected components, the study of fatigue resistance tends to be more and more important. In this work, care will be presented in the preparation of the digital models, in order to obtain reliable results.

In the development of this methodology, two numerical simulation software's were used to generate the digital models. One of them is dedicated to the rheological study of thermoplastic parts and the other to the structural behavior of injection molds.

The execution of numerical simulations requires a good characterization of the materials used. In the case of thermoplastics, manufacturers have a large database with the information needed for numerical simulations. However, for structural simulations, manufacturers tend to provide only monotonic curve data, which do not provide any information about fatigue behavior. Therefore, in this work, empirical models that fit the steels used in injection molds were studied, from which it is possible to generate the S-N and ε -N curves. In order to evaluate which empirical model would best fit this area, experimental tests were performed with specimens made in EN 1.2311. From these tests, the most conservative empirical model was chosen. Based on the chosen empirical model, an application capable of generating the S-N and ε -N curves from the information provided by the steel mill was developed.

Besides the characterization of the materials, it is also important that the loading conditions of the numerical structural model are as close as possible to what will occur in the physical model. Since the loads of this numerical model can be predicted from the rheological numerical model, the creation of a bridge between these two numerical models is essential. Therefore, in this work was built an application capable of converting the data generated by the commercial software Moldflow into files capable of being read by commercial structural numerical simulation software. Using this application for data conversion, simulations were performed and compared with the respective physical models. It was found that it is possible to replicate the mold behavior in digital models. However, the digital models of the injection molds studied tended to present conservative results when compared to the physical models.

Finally, an application capable of using data calculated from commercial numerical structural calculation software was developed for determining the fatigue resistance of molds. Here the validated model for generating the fatigue curves of the materials was taken into account. The fatigue calculation models in the application are based on the Palmgren - Miner rule for the determination of the cycles until crack nucleation. The alternating stresses calculation was performed from two methods, the octahedral shear stress criterion and the Sines method. To test the application, five molds that presented fatigue failures were chosen. Then, the methodology proposed in this work was applied to determine their fatigue resistance. From the application of this methodology and the tools developed for its use, it was possible to verify that it is able to predict the areas where the failures occurred, as well as others with a probability of crack nucleation.

Therefore, during this work it was possible to create a methodology and support tools for the calculation of fatigue molds. Thus, mold designers can have a good perspective of the fatigue resistance of injection molds still in project, based on scientific methods.

Key words: Metal fatigue; Injection mold; Structural analysis; Numerical Simulation; Metal characterization.

Index

List of figures.....	XII
List of tables.....	XXVIII
List of abbreviations.....	XXX
List of notations.....	XXXI
1 Introduction	1
1.1 Motivation	1
1.2 Work context	2
1.3 Thesis framework.....	4
2 State of art and injection mold design concepts	8
2.1 State of art.....	9
2.1.1 Mold design	11
2.1.2 Use of FEM in injection mold design	13
2.1.3 Metallurgy, surface wear and corrosion.....	23
2.1.4 Structural mechanics	24
2.1.5 Compression loading.....	25
2.1.6 Fatigue	25
2.1.7 Fracture	31
2.1.8 Surface roughness	31
2.2 Injection mold design and components.....	32
2.3 Rheology study during the design process	36
2.4 Injection mold thermal concepts	41
3 Fatigue design theoretical concepts	46
3.1 Nucleation of fatigue fractures	51
3.2 Principal mechanisms of nucleation and microscopic growth of fatigue cracks	51
3.3 S-N curves.....	57
3.3.1 Basquin equation	59
3.3.2 Palmgren, Kohout and Vechet equations.....	59
3.3.3 Simplified full range S-N curve	60
3.3.4 Strzelecki model.....	63

3.3.5	Hardness method.....	64
3.4	ϵ -N curves	65
3.4.1	Coffin-Manson relationship.....	65
3.4.2	Four-point correlation method	67
3.4.3	Modified Four-point correlation method.....	69
3.4.4	Universal slopes method	69
3.4.5	Hardness method.....	70
3.5	Comparison between S-N and ϵ -N curves.....	70
3.6	Factors affecting the fatigue life.....	71
3.6.1	Loading type (k_c).....	72
3.6.2	The component size and stress or strain distribution (k_b)	73
3.6.3	Surface finish (k_s)	73
3.6.4	Effects of notches (k_t)	76
3.6.5	Temperature effects (k_d)	79
3.6.6	Environmental effects.....	80
3.6.7	Reliability factor (k_r).....	82
3.6.8	Effect of mean stress	82
3.7	Variable amplitude loading.....	85
3.7.1	Cumulative damage	85
3.7.2	Cycle counting methods.....	88
3.8	Multiaxial stresses.....	91
3.8.1	Stress-based criteria.....	93
3.8.2	Strain-based approach	94
3.8.3	Critical plane approach	95
4	Computer applications and methodology to conduct the fatigue analysis.....	100
4.1	Application for fatigue data creation	100
4.1.1	Empirical S-N curve validation	103
4.2	Link between Moldflow and structural software.....	109
4.3	Fatigue endurance limit calculation algorithm	111

4.4	Proposed methodology for fatigue calculation of injection molds.....	120
5	Injection mold structural analysis.....	124
5.1	Plastic part rheology.....	129
5.1.1	Rheology numerical analysis from tool 1	129
5.1.2	Rheology numerical analysis from tool 2	133
5.2	Numerical models for the structural analysis	138
5.2.1	Structural numerical analysis from tool 1	138
5.2.2	Structural numerical analysis from tool 2	147
5.3	Experimental results.....	154
5.3.1	Experimental results from tool 1	154
5.3.2	Experimental results from tool 2	157
5.4	Chapter conclusions	162
6	Fatigue theoretical model analysis	164
6.1	Tools under study	164
6.1.1	Rheological numerical simulation results.....	164
6.1.2	Structural simulations	179
6.1.3	Fatigue endurance limit calculations	197
6.2	Chapter conclusions	242
7	Conclusions	246
8	Future research	250
	References	251
	Appendix A	265
	Appendix B	284
	Appendix C	297

List of figures

Figure 1 – Schematic of the design phase of an injection mold.....	8
Figure 2 – Example of a plastic part and its injection system.	9
Figure 3 - Design and manufacturing process using traditional CAD and CAE systems for injection molding parts.[3].	10
Figure 4 - Various factors influencing molding quality [6].....	11
Figure 5 – Scheme of the mold design based in previous knowledge [14] (adapted).....	12
Figure 6 – The data flow diagram in mold-base design [11]	13
Figure 7 – Injection mold of a bumper; a)Injection mold for a car bumper [22] b) One example of the core deflection of a bumper injection mold at the maximum injection pressure.....	14
Figure 8 - Four commonly used rheometers; from left to right: coaxial cylinders in Couette flow, coaxial parallel plates, coaxial cone and plate and cylindrical capillary [28].	15
Figure 9 - Unstable flow which causes surface defects [35].....	16
Figure 10 - Valve gate injection system for thermoplastic injection molds [38]	17
Figure 11 - Plastic part with different corner concepts to study the geometry effect in sink marks (LIU, LIN, & WU, 2001).....	18
Figure 12 – Cooling system optimization in plastic injection molding using CAE tools [58] (adapted).	21
Figure 13 - Conformal cooling channels example made in one insert.	21
Figure 14 - Scheme of the estimation method for fatigue life [106]	27
Figure 15 – 3D CAD drawing of an injection mold.	32
Figure 16 – Representation of the a) cavity side of the injection mold and b) cavity side without the cavity block.....	33
Figure 17 – Representation of the core side of the injection mold.	33
Figure 18 – Core side of the injection mold a) without the core block and b) without the core block, rail and one of the extraction plates.....	34
Figure 19 – Molding components in the core where a) moving components during the extraction phase and b) are the inserts with the representation of half of the plastic part.	35
Figure 20 – Detail of sliders in an injection mold. Where a) are the sliders, b) one section of a slider and c) the detail of the slider and part.	36
Figure 21 – Detail of lifters in an injection mold.	36
Figure 22 – Filling pattern of a plastic part.	37

Figure 23 – Isochronous lines of the filling pattern.	37
Figure 24 – Pressure at the instant of switching between injection in velocity and injection in pressure.	38
Figure 25 – Temperature at front flow.	38
Figure 26 – Shear rate at the cold gates.	39
Figure 27 – Volumetric shrinkage at ejection.....	39
Figure 28 - Weld lines.....	40
Figure 29 – Pressure at the injection location.....	40
Figure 30 – Minimum injection machine clamping force.	41
Figure 31 – Elements used in thermal simulations in Moldflow.	41
Figure 32 – Representation of the functioning a) a baffle and b) a bubbler [42].	42
Figure 33 – Cooling channels analysis results, where a) is the coolant flow rate, b) is the circuits pressure, c) the Reynolds number at each location and c) is the coolant temperature.	43
Figure 34 – Relationship between the Nusselt number in function of the Prandtl and Reynolds numbers [149].	43
Figure 35 – Temperature distribution in the part and molding region of the cavity side and core sides.	44
Figure 36 – Temperature profile between the core and cavity side of the part.....	45
Figure 37 - Pressure at the injection mold entrance during five consecutive cycles.	46
Figure 38 - Fatigue design flow chart created for use by the Society of Automotive Engineers Fatigue Design and Evaluation [151].....	48
Figure 39 – Fatigue cracks stages.....	51
Figure 40 - Schematic of slip due to external loads. a) static load and b) cyclic load [154] (adapted).	53
Figure 41 - Fatigue progression and detail of the appearance of inclusions and extrusions under cyclic loading [151].....	54
Figure 42 - Schematic of stages I and II of the propagation stages for transcrystalline microscopic fatigue crack, and the indication of the fatigue resistance in terms of difficulty of stage I crack pass to a stage II crack [155].	55
Figure 43 – Two different fatigue crack propagations and representation of the first and second stages for each one. Where a) is the representation of crack that grow parallel to the free surface and b) is for cracks that propagate away from the free surface [155] (adapted).....	57

Figure 44 – Representation of a S-N curve of a low-alloy steel SAE 4130 [156] (adapted).	58
Figure 45 – Fatigue diagram for different mean stresses where a) is the representation of the S-N curves for different S_m values and b) is the representation of the fatigue diagram obtained from the S-N curves [156].	58
Figure 46 – S-N curve representation where is shown the regions of the curve [135] (adapted)....	59
Figure 47 – Representation of the full range S-N curve and some related terms [157] (adapted)...	60
Figure 48 – Predicted S-N curves using the proposed formula using the global value for b and experimental S-N data [157].	62
Figure 49 – Illustration of the S-N curve with accordingly with the Strzelecki formulation [158]	63
Figure 50 – Plot of the relations between the ultimate strength and steel hardness [139].	64
Figure 51 – Stress-strain loops during constant strain amplitude cycles and stress history for cyclic a) hardening and b) softening [156].	66
Figure 52 – Schematic of fatigue life curve with the four points highlighted for the elastic and plastic strain lines [162] (adapted).	68
Figure 53 – Two similar load histories diverging only in the first cycle (OABC) applied to a notch member [163] (adapted).	71
Figure 54 – Correction factors applied to a S-N curve obtained from a smooth test probe.	72
Figure 55 – Size factor (C_s) for steels [164] (adapted).	73
Figure 56 – Surface factor that accounts for the surface finish on the steel fatigue endurance [151].	75
Figure 57 – Stresses distribution in a bar with symmetric notches: a) bar shape and loading direction; b) photo-elastic fringe photography; c) Stresses distribution obtained from a finite element solution [167].	77
Figure 58 – Example of a bar with a centered hole under a longitudinal load with the respective stress distribution at the plane which crosses the center of the hole [167].	77
Figure 59 – Isochrone lines representing the stress concentration at a notch from a finite element model.	78
Figure 60 – Curves of lifetime of a 316L stainless steel probes tested at 1Hz under cyclic loading in air and vacuum at different temperatures [173]	81
Figure 61 – Effect of mean stress on fatigue life [151].	83
Figure 62 -Constant life plots on normal stress in S_a and S_m coordinates [166] (adapted).	83

Figure 63 – various forms of presenting mean-stress data, where line “a” represents de Soderberg curve, line “b” the Goodman curve, line “c” the Gerber parabola and line “d” the Morrow curve [163] (adapted).	84
Figure 64 – Examples of variable loading spectrums [173].	86
Figure 65 – Example of a block with bilevel loading sequence [151].	87
Figure 66 – Influence of the load spectrum on the fatigue limit [177].	88
Figure 67 – Peak counting method where a) is the load cycling history and the peak count table and b) is the resulting cycling counting [178].	89
Figure 68 – Level crossing counting method where a) is the load cycling history with the levels and the crossing counting table and b) is the resulting cycling counting [178].	89
Figure 69 – Rainflow counting method where a) is the load cycling history and b) is the Rainflow counting [166].	90
Figure 70 – Example of a proportional loading with the normal and shear stresses in-phase.	93
Figure 71 – Example of a non-proportional loading with the normal and shear stresses out-of-phase.	93
Figure 72 – Algorithm of the data creation application.	101
Figure 73 – Comparison between the S-N models and the Roessle and Fatemi ε -N model, using the equivalent stress as the term of comparison.	102
Figure 74 - Application interface window to input the steel monotonic data.	102
Figure 75 – Relationship between the steel hardness and the fatigue endurance limit [182].	103
Figure 76 – Application interface output windows of the fatigue data.	104
Figure 77 – Dimensions of the steel probe used to obtain the monotonic properties following the EN ISO 6892-1 standard.	105
Figure 78 – Trial set-up where a) shows the probe in the trial machine during tensile trial with mechanical strain gauge and b) the mechanical strain gauge.	106
Figure 79 – BAQ dynaROCK II portable hardness measuring device.	107
Figure 80 – Dimensions of the steel probe used to verify the S-N curve following the E606/E606M – 12 standard.	107
Figure 81 – Full range S-N curves obtained by the Simplified full range S-N curve (model 1) and the hardness method (model 2) for the steel EN 1.2311.	108
Figure 82 – Plot with the simplified full range S-N curve (Model 1) and the trials results.	108
Figure 83 – Plot with the hardness method curve (Model 2) and the trials results.	108

Figure 84 – Algorithm for the link interface between Moldflow and FEM software.	110
Figure 85 – Application interface window to generate the input file for the structural numerical simulation.....	110
Figure 86 – Mapped pressures of one time-step in a mold engraving region using the application presented in this chapter and the table presenting the other mapped time-steps in ANSYS Mechanical.	111
Figure 87 – Three phases of the application for fatigue calculation.	112
Figure 88 – Example of the maximum principal stress exported from ANSYS Mechanical.....	112
Figure 89 – Scheme of the procedure for the fatigue calculation application and its outputs.	113
Figure 90 – Management of the maximum principal stress data.	114
Figure 91 – Algorithm of the fatigue endurance limit application.	115
Figure 92 – Application procedure to calculate the tool fatigue endurance.....	116
Figure 93 – Example of the counting cycles mechanism.	117
Figure 94 – Output from the algorithm to detect the peaks.	118
Figure 95 – Generation of the output files.	118
Figure 96 – Application graphical user interface.....	119
Figure 97 – Example of the csv file output for the equivalent alternated stress.....	119
Figure 98 – Scheme to analyze the results outside the FEM software.	120
Figure 99 – Diagram of the proposed methodology for the determination of the fatigue endurance limit of injection molds.....	122
Figure 100 – Diagram for all the structural simulations made for the structural analysis.	124
Figure 101 – Full model of the injection tools under study for study. a) is the tool 1 and b) the tool 2.	125
Figure 102 – Moldflow models from the plastic parts produced a) by tool 1 and b) by tool 2.	125
Figure 103 – Section of tool 1 where is represented the fixation between the bottom clamp plate, the rails and the core. The orange arrows represent the locked directions by the components.....	126
Figure 104 – Section view of one example of a) an insert and b) a slider. Where the red arrows represent the pressure load caused by the molten polymer, the blue arrow represents the movement made by the extraction component during the ejection of the plastic part and the orange line represents the walls in contact between the core/extraction component.	127

Figure 105 – Core side CAD from tool 1 prepared for the numerical analysis where a) is the model for the numerical analysis with lower simplification and b) for the numerical analysis fully simplified.	128
Figure 106 – Section of the simplified CAD of tool 1 detailing the position of the indirect pressure sensor.	129
Figure 107 – Packing pressure profile of tool1.	130
Figure 108 – Filling pattern of the plastic part from tool 1.	131
Figure 109 – Pressure distribution during the part injection cycle of tool 1.	132
Figure 110 – Pressure at defined locations through time. The black line is the pressure at the injection tool entrance, the red line at the cold gate, the others are at important locations at the engraving region.	133
Figure 111 – Injection pressure curve and the dashed lines represent the time-steps used on the structural simulations.	133
Figure 112 – Ram speed Vs ram position in tool 2.	134
Figure 113 – Packing pressure profile of tool1.	135
Figure 114 – Filling pattern of the plastic part from tool 2.	135
Figure 115 – Pressure distribution during the part injection cycle of tool 2.	136
Figure 116 – Pressure at defined locations through time. The black line is the pressure at the injection tool entrance, the others are at important regions at the engraving region.	137
Figure 117 - Injection pressure curve and the dashed lines represent the time-steps used on the structural simulations of the tool 2.	138
Figure 118 – Boundary conditions for the analysis of tool 1.	139
Figure 119 – Injection tool under study a) photo of the tool, b) model under study where the red highlighted region is the position of an indirect pressure sensor and the yellow highlighted region marks the position of the Inductive length measuring probe.	139
Figure 120 – Injection mold deformations in mm at 1.5329 seconds after the beginning of the injection.	140
Figure 121 – Stresses [MPa] distribution at 1.5329 seconds after the beginning of the injection.	141
Figure 122 – Locations chosen to perform the comparison through the injection cycle.	142
Figure 123 – Displacement though the injection cycle of the point N1.	143
Figure 124 – Displacement though the injection cycle of the point N2.	144
Figure 125 – Displacement though the injection cycle of the point N3.	144

Figure 126 – Displacement though the injection cycle of the point N4.	145
Figure 127 – von Mises stress distribution in the cavity insert through three different instants of the injection cycle.	146
Figure 128 – Detail of a pressure plate with a) LGS where its pocket is present and with b) FGS where the pressure plate is merged to the core.	146
Figure 129 – Gap between the core and cavity in the adjustment zone.	147
Figure 130 – Boundary conditions for the analysis of tool 2.	148
Figure 131 – Injection tool under study a) tool core side where the red highlighted region is the position of an indirect pressure sensor and b) is the tool cavity side where the yellow highlighted regions mark the position of the Inductive length measuring probes.	148
Figure 132 – Injection mold deformations in mm at 6,4701 seconds after the beginning of the injection.	149
Figure 133 – Stresses [MPa] distribution at 6,4701 seconds after the beginning of the injection.	150
Figure 134 – Displacement though the injection cycle of the point LOT.	152
Figure 135 – Displacement though the injection cycle of the point LOB.	152
Figure 136 – Displacement though the injection cycle of the point LOPT.	153
Figure 137 – Displacement though the injection cycle of the point LOPB.	153
Figure 138 – Comparison of the displacement between the four locations.	153
Figure 139 – Pressure at the indirect sensor, comparison between numerical and experimental data.	154
Figure 140 – Displacement at the partition plane.	155
Figure 141 – Displacement in mm relative to the global coordinate system where a) is the cavity and b) the core.	157
Figure 142 – Displacement at the partition plane for one injection cycle.	158
Figure 143 – Comparison of the displacement at the partition plane in the LOT position between the numerical and experimental results.	159
Figure 144 – Comparison of the displacement at the partition plane in the LOB position between the numerical and experimental results.	160
Figure 145 – Comparison of the displacement at the partition plane in the LOPB position between the numerical and experimental results.	160
Figure 146 – Pressure at the indirect sensor, comparison between numerical and experimental data.	161

Figure 147 – Filling pattern of the part.	166
Figure 148 – Pressure distribution a) at the velocity/pressure switchover and b) at the end of filling.	167
Figure 149 – Pressure through time in critical locations of the engraving region.	168
Figure 150 – Needed force to keep the injection tool closed through the injection cycle.	168
Figure 151 – Filling pattern of the part.	169
Figure 152 – Pressure distribution a) at the velocity/pressure switchover and b) at the end of filling.	170
Figure 153 – Pressure through time in critical locations of the engraving region.	170
Figure 154 – Needed force to keep the injection tool closed through the injection cycle.	171
Figure 155 – Filling pattern of the part.	172
Figure 156 – Pressure distribution a) at the velocity/pressure switchover and b) at the end of filling.	172
Figure 157 – Pressure through time in critical locations of the engraving region.	173
Figure 158 – Needed force to keep the injection tool closed through the injection cycle.	173
Figure 159 – Filling pattern of the part.	174
Figure 160 – Pressure distribution a) at the velocity/pressure switchover and b) at the end of filling.	174
Figure 161 – Pressure through time in critical locations of the engraving region.	175
Figure 162 – Needed force to keep the injection tool closed through the injection cycle.	175
Figure 163 – Filling pattern of the part.	176
Figure 164 – Pressure distribution a) at the velocity/pressure switchover and b) at the end of filling.	177
Figure 165 – Pressure through time in critical locations of the engraving region.	178
Figure 166 – Needed force to keep the injection tool closed through the injection cycle.	178
Figure 167 – von Mises stress distribution through the core and cavity sides of Fat_1 at the instant where only the clamp force is applied and at the instant of highest pressure through the engraving region.	180
Figure 168 – von Mises stress distribution through the core and cavity engraving regions of Fat_1 at the instant where only the clamp force is applied and at the instant of highest pressure through the engraving region.	181

Figure 169 – von Mises stress distribution through the core and cavity fitting regions of the partition plane of Fat_1 at the instant where only the clamp force is applied and at the instant of highest pressure through the engraving region.	182
Figure 170 – von Mises stress distribution through the core and cavity sides of Fat_2 at the instant where only the clamp force is applied and at the instant of highest pressure through the engraving region.	183
Figure 171 – von Mises stress distribution through the core and cavity engraving regions of Fat_2 at the instant where only the clamp force is applied and at the instant of highest pressure through the engraving region.	184
Figure 172 – von Mises stress distribution through the core and cavity fitting regions of the partition plane of Fat_2 at the instant where only the clamp force is applied and at the instant of highest pressure through the engraving region.	185
Figure 173 – von Mises stress distribution through the core and cavity sides of Fat_3 at the instant where only the clamp force is applied and at the instant of highest pressure through the engraving region.	186
Figure 174 – von Mises stress distribution through the core and cavity engraving regions of Fat_3 at the instant where only the clamp force is applied and at the instant of highest pressure through the engraving region.	187
Figure 175 – von Mises stress distribution through the core and cavity fitting regions of the partition plane of Fat_3 at the instant where only the clamp force is applied and at the instant of highest pressure through the engraving region.	188
Figure 176 – von Mises stress distribution through the core and cavity sides of Fat_4 at the instant where only the clamp force is applied and at the instant of highest pressure through the engraving region.	189
Figure 177 – von Mises stress distribution through the core and cavity engraving regions of Fat_4 at the instant where only the clamp force is applied and at the instant of highest pressure through the engraving region.	190
Figure 178 – von Mises stress distribution through the core and cavity fitting regions of the partition plane of Fat_4 at the instant where only the clamp force is applied and at the instant of highest pressure through the engraving region.	191

Figure 179 – von Mises stress distribution through the core and cavity sides of Fat_5 at the instant where only the clamp force is applied and at the instant of highest pressure through the engraving region.	192
Figure 180 – von Mises stress distribution through the core and cavity engraving regions of Fat_5 at the instant where only the clamp force is applied and at the instant of highest pressure through the engraving region.	193
Figure 181 – von Mises stress distribution through the core and cavity fitting regions of the partition plane of Fat_5 at the instant where only the clamp force is applied and at the instant of highest pressure through the engraving region.	194
Figure 182 – Pressure load, von Mises stress distribution and total deformation of the most stressed components of Fat_5.	196
Figure 183 – Fatigue reduction coefficients for the Fat_1 tool.	198
Figure 184 – Maximum alternated stress in MPa at each location by the Sines method at the cavity engraving region of Fat_1.	199
Figure 185 – Damage determined by the octahedral shear stress method at the cavity engraving region of Fat_1.	199
Figure 186 – Damage determined by the Sines method at the cavity engraving region of Fat_1.	199
Figure 187 – Number of cycles before crack nucleation determined by a) the octahedral shear stress and by b) the Sines method at the cavity engraving region of Fat_1.	200
Figure 188 – Safety factor determined by a) the octahedral shear stress and by b) the Sines method at the cavity engraving region of Fat_1.	200
Figure 189 – Maximum alternated stress in MPa at each location by the Sines method at the fitting region on the cavity side of Fat_1.	200
Figure 190 – Damage determined by the octahedral shear stress method at the fitting region on the cavity side of Fat_1.	201
Figure 191 – Damage determined by the Sines method at the fitting region on the cavity side of Fat_1.	201
Figure 192 – Number of cycles before crack nucleation determined by a) the octahedral shear stress and by b) the Sines method at the fitting region on the cavity side of Fat_1.	202
Figure 193 – Safety factor determined by a) the octahedral shear stress and by b) the Sines method at the fitting region on the cavity side of Fat_1.	202
Figure 194 – Core engraving region and the sections under individual analysis.	202

Figure 195 – Maximum alternated stress in MPa at each location by the Sines method on the core engraving region 1 of Fat_1.....	203
Figure 196 – Damage determined by the octahedral shear stress method on the core engraving region 1 of Fat_1.....	203
Figure 197 – Damage determined by the Sines method on the core engraving region 1 of Fat_1.	203
Figure 198 – Number of cycles before crack nucleation determined by a) the octahedral shear stress and by b) the Sines method on the core engraving region 1 of Fat_1.....	204
Figure 199 – Safety factor determined by a) the octahedral shear stress and by b) the Sines method on the core engraving region 1 of Fat_1.....	204
Figure 200 – Maximum alternated stress at each location by the Sines method on the core engraving region 2 of Fat_1.....	204
Figure 201 – Damage determined by the octahedral shear stress method on the core engraving region 2 of Fat_1.....	205
Figure 202 – Damage determined by the Sines method on the core engraving region 2 of Fat_1.	205
Figure 203 – Number of cycles before crack nucleation determined by a) the octahedral shear stress and by b) the Sines method and c) is the region under analysis on the core engraving region 2 of Fat_1.	205
Figure 204 – Number of cycles before crack nucleation determined by a) the octahedral shear stress and by b) the Sines method and c) is the region under analysis on the core engraving region 2 of Fat_1.	206
Figure 205 – Safety factor determined by a) the octahedral shear stress and by b) the Sines method and c) is the region under analysis on the core engraving region 2 of Fat_1.	206
Figure 206 – Safety factor determined by a) the octahedral shear stress and by b) the Sines method and c) is the region under analysis on the core engraving region 2 of Fat_1.	207
Figure 207 – Maximum alternated stress in MPa at each location by the Sines method on the core engraving region 3 of Fat_1.....	207
Figure 208 – Damage determined by the octahedral shear stress method on the core engraving region 3 of Fat_1.....	208
Figure 209 – Damage determined by the Sines method on the core engraving region 3 of Fat_1.	208

Figure 210 – Number of cycles before crack nucleation determined by a) the octahedral shear stress and by b) the Sines method on the core engraving region 3 of Fat_1	208
Figure 211 – Safety factor determined by a) the octahedral shear stress and by b) the Sines method on the core engraving region 3 of Fat_1	209
Figure 212 – Maximum alternated stress in MPa at each location by the Sines method at the fitting region on the core side of Fat_1	209
Figure 213 – Damage determined by the octahedral shear stress method at the fitting region on the core side of Fat_1	209
Figure 214 – Damage determined by the Sines method at the fitting region on the core side of Fat_1.	210
Figure 215 – Number of cycles before crack nucleation determined by a) the octahedral shear stress and by b) the Sines method at the fitting region on the core side of Fat_1.	210
Figure 216 – Safety factor determined by a) the octahedral shear stress and by b) the Sines method at the fitting region on the core side of Fat_1.	210
Figure 217 – Maximum alternated stress in MPa at each location by the Sines method at the cavity engraving region of Fat_2	211
Figure 218 – Damage determined by the Sines method at the cavity engraving region of Fat_2 ..	212
Figure 219 – Number of cycles before crack nucleation determined the Sines method at the cavity engraving region of Fat_2	212
Figure 220 – Safety factor determined by the Sines method at the cavity engraving region of Fat_2.	212
Figure 221 – Maximum alternated stress in MPa at each location by the Sines method at the fitting region on the cavity side of Fat_2	213
Figure 222 – Damage determined by the Sines method at the fitting region on the cavity side of Fat_2.	213
Figure 223 – Number of cycles before crack nucleation determined the Sines method at the fitting region on the cavity side of Fat_2	214
Figure 224 – Safety factor determined by the Sines method at the fitting region on the cavity side of Fat_2	215
Figure 225 – Maximum alternated stress in MPa at each location by the Sines method at the core engraving region of Fat_2	215
Figure 226 – Damage determined by the Sines method at the core engraving region of Fat_2	216

Figure 227 – Number of cycles before crack nucleation determined the Sines method at the core engraving region of Fat_2.....	216
Figure 228 – Safety factor determined by the Sines method at the core engraving region of Fat_2.	217
Figure 229 – Maximum alternated stress in MPa at each location by the Sines method at the fitting region on the core side of Fat_2.....	217
Figure 230 – Damage determined by the Sines method at the fitting region on the core side of Fat_2.	218
Figure 231 – Number of cycles before crack nucleation determined the Sines method at the fitting region on the core side of Fat_2.....	218
Figure 232 – Safety factor determined by the Sines method at the fitting region on the core side of Fat_2.....	218
Figure 233 – Maximum alternated stress in MPa at each location by the Sines method at the cavity engraving region of Fat_3.....	219
Figure 234 – Damage determined by the Sines method at the cavity engraving region of Fat_3..	220
Figure 235 – Number of cycles before crack nucleation determined the Sines method at the cavity engraving region of Fat_3.....	220
Figure 236 – Safety factor determined by the Sines method at the cavity engraving region of Fat_3.	221
Figure 237 – Maximum alternated stress in MPa at each location by the Sines method at the fitting region on the cavity side of Fat_3.....	221
Figure 238 – Damage determined by the Sines method at the fitting region on the cavity side of Fat_3.	222
Figure 239 – Number of cycles before crack nucleation determined the Sines method at the fitting region on the cavity side of Fat_3.....	222
Figure 240 – Safety factor determined by the Sines method at the fitting region on the cavity side of Fat_3.....	222
Figure 241 – Maximum alternated stress in MPa at each location by the Sines method at the core engraving region of Fat_3.....	223
Figure 242 – Damage determined by the Sines method at the core engraving region of Fat_3....	223
Figure 243 – Number of cycles before crack nucleation determined the Sines method at the core engraving region of Fat_3.....	223

Figure 244 – Safety factor determined by the Sines method at the core engraving region of Fat_3.	224
Figure 245 – Maximum alternated stress in MPa at each location by the Sines method at the fitting region on the core side of Fat_3.	224
Figure 246 – Damage determined by the Sines method at the fitting region on the core side of Fat_3.	225
Figure 247 – Number of cycles before crack nucleation determined the Sines method at the fitting region on the core side of Fat_3.	225
Figure 248 – Safety factor determined by the Sines method at the fitting region on the core side of Fat_3.	225
Figure 249 – Fatigue reduction coefficients for the Fat_4 tool.	227
Figure 250 – Maximum alternated stress in MPa at each location by the Sines method at the cavity engraving region of Fat_4.	227
Figure 251 – Damage determined by the Sines method at the cavity engraving region of Fat_4.	228
Figure 252 – Safety factor determined by the Sines method at the cavity engraving region of Fat_4.	228
Figure 253 – Maximum alternated stress in MPa at each location by the Sines method at the fitting region on the cavity side of Fat_4.	228
Figure 254 – Damage determined by the Sines method at the fitting region on the cavity side of Fat_4.	229
Figure 255 – Number of cycles before crack nucleation determined the Sines method at the fitting region on the cavity side of Fat_4.	229
Figure 256 – Safety factor determined by the Sines method at the fitting region on the cavity side of Fat_4.	229
Figure 257 – Maximum alternated stress at each location by the Sines method at the core engraving region of Fat_4.	230
Figure 258 – Damage determined by the Sines method at the core engraving region of Fat_4.	230
Figure 259 – Number of cycles before crack nucleation determined the Sines method at the core engraving region of Fat_4.	230
Figure 260 – Safety factor determined by the Sines method at the core engraving region of Fat_4.	231

Figure 261 – Maximum alternated stress at each location by the Sines method at the fitting region on the core side of Fat_4.....	231
Figure 262 – Damage determined by the Sines method at the fitting region on the core side of Fat_4.	231
Figure 263 – Number of cycles before crack nucleation determined the Sines method at the fitting region on the core side of Fat_4.....	231
Figure 264 – Safety factor determined by the Sines method at the fitting region on the core side of Fat_4.....	232
Figure 265 – Images of the locations where fatigue cracks appeared in the tool.	232
Figure 266 – The a) engraving region is made by two different manufacturing processes where the region b) is milled and the region c) is made by EDM.....	233
Figure 267 – Maximum alternated stress in MPa at each location by the Sines method at the cavity engraving region of Fat_5 a) in the milled region and b) on the EDM region.....	234
Figure 268 – Damage determined by the Sines method at the cavity engraving region of Fat_5 a) in the milled region and b) on the EDM region.....	234
Figure 269 – Number of cycles before crack nucleation determined the Sines method at the cavity engraving region of Fat_5 a) in the milled region and b) on the EDM region.....	235
Figure 270 – Safety factor determined by the Sines method at the cavity engraving region of Fat_5 a) in the milled region and b) on the EDM region.....	235
Figure 271 – Maximum alternated stress at each location by the Sines method at the fitting region on the cavity side of Fat_5.....	236
Figure 272 – Damage determined by the Sines method at the fitting region on the cavity side of Fat_5.	236
Figure 273 – Safety factor determined by the Sines method at the fitting region on the cavity side of Fat_5.....	237
Figure 274 – Maximum alternated stress at each location by the Sines method at the core engraving region of Fat_5 a) in the milled region and b) on the EDM region.....	237
Figure 275 – Damage determined by the Sines method at the core engraving region of Fat_5 a) in the milled region and b) on the EDM region.....	238
Figure 276 – Number of cycles before crack nucleation determined the Sines method at the core engraving region of Fat_5 a) in the milled region and b) on the EDM region.....	239

Figure 277 – Safety factor determined by the Sines method at the core engraving region of Fat_5 a) in the milled region and b) on the EDM region.....	239
Figure 278 – Maximum alternated stress at each location by the Sines method at the fitting region on the core side of Fat_5.....	240
Figure 279 – Damage determined by the Sines method at the fitting region on the core side of Fat_5.	240
Figure 280 – Number of cycles before crack nucleation determined the Sines method at the fitting region on the core side of Fat_5.	241
Figure 281 – Safety factor determined by the Sines method at the fitting region on the core side of Fat_5.....	241
Figure 282 – Predicted locations for fatigue crack nucleation in the injection mold Fat_5.....	242
Figure 283 – Image of the cracked component where the region with the visible crack is highlighted.	242

List of tables

Table 1 - Failure modes of metals [120].....	48
Table 2 - Metallurgical feature scales [152].....	50
Table 3 - Schematic fatigue fractures for axial and bending loads for round specimens [153].....	52
Table 4 - Surface finish parameters for steels [112].....	74
Table 5 – Surface treatment factors for steels with surface treatments [166] (adapted).....	75
Table 6 – Comparison of Kt between four different specimens obtained by three different techniques [156].	79
Table 7 – Comparison of the fatigue endurance strength of steels exposed to air and under a corrosive environment of complete immersion in 6,8% salt water [174] (adapted)	82
Table 8 – Reliability factors for a standard deviation at 8% deviation from the strength endurance limit.	82
Table 9 – Reversal counts using the Rainflow counting method [166].....	91
Table 10 – Cycles obtained from the Rainflow counting method [166].....	91
Table 11 – Monotonic data from the steel provider and from tensile trials from EN 1.2311.	106
Table 12 – Number of components for each tool at each level of simplification.	129
Table 13 – Number of contacts for each tool at each level of simplification.	129
Table 14 – Input data for numerical simulation obtained from trial.	130
Table 15 – Input data for numerical simulation obtained from trial.	134
Table 16 – Polymer parts and respective tools.....	165
Table 17 – Input data for numerical simulation of Fat_1.....	166
Table 18 – Input data for numerical simulation of Fat_2.....	169
Table 19 – Input data for numerical simulation of Fat_3.....	171
Table 20 – Input data for numerical simulation of Fat_4.....	173
Table 21 – Input data for numerical simulation of Fat_5.....	176
Table 22 – ExxonMobil Chemical - Exxtral BNU011 material properties.	285
Table 23 – ExxonMobil Chemical - Exxtral HMU202 material properties.....	287
Table 24 – SunAllomer Ltd - TRC104N material properties.....	289
Table 25 – Mitsubishi Chemical Corporation - Durabio D7340R material properties.	291
Table 26 – Borealis Europe – BD950MO material properties.....	294
Table 27 – Tool_1 with FGS simulation statistics in quasi-static regime.	298
Table 28 – Tool_1 with FGS simulation statistics in transient regime.	298

Table 29 – Tool_1 with LGS simulation statistics in quasi-static regime.	298
Table 30 – Tool_1 with LGS simulation statistics in transient regime.	298
Table 31 – Tool_2 with FGS simulation statistics in quasi-static regime.	298
Table 32 – Tool_2 with FGS simulation statistics in transient regime.	298
Table 33 – Tool_2 with LGS simulation statistics in quasi-static regime.	298
Table 34 – Tool_2 with LGS simulation statistics in transient regime.	299
Table 35 – FAT_1 simulation statistics.	299
Table 36 – FAT_2 simulation statistics.	299
Table 37 – FAT_3 simulation statistics.	299
Table 38 – FAT_4 simulation statistics.	299
Table 39 – FAT_5 simulation statistics.	299

List of abbreviations

1D, 2D, 2.5D and 3D – One dimension, two dimensions, two and one-half dimensions and three dimensions.

ANOVA – Analysis of variance.

CAD – Computer aided design.

CAE – Computer assisted engineering.

CAM – Computer aided manufacturing.

DOF – Design of experiment.

EDM – Electrical discharge machining.

FEM – Finite element method.

FGS – Full geometric simplification.

GCF – Giga cycle fatigue.

HCF – High cycle fatigue.

HDPE – High density polyethylene.

LCF – Low cycle fatigue.

LGS – Low geometric simplification.

LPDE – Low density polyethylene.

PS – Polystyrene.

PSB – Persistent slip bands.

R – Ratio.

S-N – Stress-number of cycles.

VOF – Volume of fluid.

ε -N – Strain-number of cycles.

List of notations

- A – Constant from Basquin equation.
- a , b and c – Material related parameters.
- a_s and b_s – Constants for the surface finish.
- cf – Fatigue ductility exponent.
- B – Number of cycles at the first knee point.
- bf – Fatigue strength exponent.
- C_1 – Constant in the critical plane approach.
- D_{PM} – Palmgren-Miner linear damage rule.
- E – Young's Modulus [Pa].
- e – Thickness [m].
- G – Shear modulus [Pa].
- HB – Brinell hardness.
- Hv – Vickers hardness [kg.f/mm²].
- K – Material constant in the Fatemi-Socie critical plane approach.
- k_c – Loading type.
- k_b – The component size and stress or strain distribution.
- k_a – Surface finish.
- k_t – Effects of notches.
- k_d – Temperature effects.
- k_e – Reliability factor.
- l – Liter [l].
- m – Slope of the S-N curve.
- m_s – Coefficient of mean stress influence.
- N – Number of cycles.
- N_f – Number of repetitions of the same cycle that equals the median life to failure.
- N_{GCF} – Number of cycles where the GCF region starts.
- N_k – Number of cycles at the second knee point.
- N_{Sy} – Number of cycles to failure at stress amplitude equal to $0,9S_y$;
- Nu – Nusselt number.
- Pr – Prandtl number.

R^2 – Least squares fit.

Re – Reynolds Number.

S – Material dependent constant.

S_a – Stress amplitude [Pa].

S_m – Mean stress [Pa].

S_{ut} – Ultimate tensile stress [Pa].

S_y – Yield stress [Pa].

S_e – Fatigue endurance limit [Pa].

$S(N)$ – Stress amplitude for a certain number of cycles [Pa].

S_k – Stress amplitude at the second knee point [Pa].

S_{GCF} – Stress amplitude at the GCF region [Pa].

S_f – Completely reversed stress amplitude for a given life [Pa].

S_{qm} – Nominal equivalent stress [Pa].

$\frac{\Delta\varepsilon}{2}$ – Represents the total deformation.

$\frac{\Delta\varepsilon_e}{2}$ – Represents the elastic deformation.

$\frac{\Delta\varepsilon_p}{2}$ – Represents the plastic deformation.

$2N_f$ – Number of cycles to reach failure.

σ'_f – Fatigue strength coefficient [Pa].

ε'_f – Fatigue ductility coefficient.

ε_f – True fracture ductility.

σ_{max} – Maximum stress [Pa].

σ_{nom} – Nominal stress [Pa].

σ – Stress [Pa].

ε – Strain.

$\sigma_1, \sigma_2, \sigma_3$ – Principal stresses [Pa].

σ_f – Fatigue strength [Pa].

σ_{a1}, σ_{a2} and σ_{a3} – Alternated main stresses [Pa].

σ_{m1}, σ_{m2} and σ_{m3} – Principal Mean stresses [Pa].

$\varepsilon_{a1}, \varepsilon_{a2}$ and ε_{a3} – Principal alternated strains.

$\frac{\Delta\tau}{2}$ – Shear stress amplitude [Pa].

σ_n – Normal stress acting on the $\Delta\tau$ plane [Pa].

$\frac{\Delta\gamma_{max}}{2}$ – Maximum shear strain amplitude [Pa].

$\Delta\varepsilon_n$ – Normal strain range on the $\Delta\gamma_{max}$ plane.

$\sigma_{n,a}$ – Alternated normal stresses [Pa].

$\sigma_{n,m}$ – Residual normal stresses [Pa].

ν_e – Plastic Poisson ratio.

ν_p – Elastic Poisson ratio.

$\Delta\tau/2$ – Shear stress amplitude [Pa].

$\Delta\tau_{max}$ – Maximum shear stress range [Pa].

$\sigma_{n,max}$ – Normal stress on the maximum shear stress range [Pa].

$\Delta\gamma_{max}/2$ – Maximum shear strain amplitude [Pa].

τ'_f – Shear fatigue strength coefficient [Pa].

1 Introduction

With the appearance of the first injection molding machine in 1946, developed by James Watson Hendry [1], the injection molding industry began to be indispensable in the production of several types of polymeric parts. This process brought numerous advantages, both in economic terms and in increasing the ability to produce plastic parts with complex shapes and a high repeatability rate. Over the years the evolution of the injection mold industry was indispensable so that it could address the market demands.

1.1 Motivation

Nowadays the use of plastics is very common and one of the most used processes to produce the plastic parts is through injection molding. For this production process, the design and production of injection tooling is very important to reduce the production costs and increase the quality of the plastic parts. With the introduction of new tooling production techniques the complexity of the parts is increasing, so the design process is becoming more complex and indispensable.

For the development of plastic parts there is a lot of research on that topic, but for the tool that will mold the plastic part most of the research is limited to some parts of the mold design. In most cases the work done with injection molding is linked to the quality of the plastic part and not to its behavior or structural strength. As parts are designed with increasingly complex geometries, the study of all phenomena is indispensable as they tend to have an impact on the quality of the plastic part. Currently, the injection process is studied independently of the structural performance of the injection mold. Considering that the injection mold is responsible for the quality of the part to be injected, any excessive deformation or cracking will lead to the appearance of defects in the plastic part.

In the course of this work, it is intended in the first place to create a mechanism to be able to use data from rheological studies, which tend to be exclusively used for the study of the plastic part, in the study of the behavior of the injection mold during the injection cycle. With this, it is also intended to understand which simplifications can be made to the injection mold models to carry out numerical studies without compromising the results. Also, within the theme of numerical simulations, it is intended to understand what differences exist between the quasi-static and transient regimes in order to optimize the studies both in terms of results and computation time.

Finally, another area that will be the target of study is the fatigue analysis of injection molds, since studies will be done with all the factors that can lead to fatigue failures in injection molds. With this is intended to create tools that help designers predict points susceptible to fatigue failures at the design stage.

At the end of this work, it is intended to have a methodology that can be used as a guide for the analysis of injection molds using numerical methods to increase the scientific basis during the injection mold design. With it is expected to reduce typical wastes related to design based on experience, which tend to be very conservative.

1.2 Work context

Nowadays it is extremely necessary that in the design phase besides taking functionality into account, economic and environmental factors must also be considered. The economic factors are mainly to do with reducing the quantity of used raw materials as well reducing the manufacturing time. In the injection molding industry the reduction of expenses with raw materials and manufacturing time has a direct impact on the environmental component, because here the energy costs involved in the production of raw materials as in its transformation are high. Besides that, the market is increasingly demanding when it comes to reliability, so improving the mold design with better studies and better information is fundamental. It is also necessary to keep in mind that injection molds are tools that will be used to manufacture thermoplastic parts. Using as example the injection molds for the manufacture of components for the automotive industry, automobiles must be economical and environmentally friendly which leads to their components having more complex shapes and with it a more demanding manufacturing. So, injection molds need to be able to withstand the stress caused by injection over thousands or millions of cycles. Since injection molds are unique tools, it is extremely important that they can withstand the stresses to which they are subjected throughout their life cycle without fatigue failures that end up compromising the logistics of the production chain in which they are inserted.

However, to be able to perform the necessary calculations for the sizing of thermoplastic injection molds, it is very important to have a good knowledge about the behavior of the used materials. In this case in the behavior of steel alloys. More concretely for the construction of these tools the most common is the use of tool steels. Therefore, considering the need to be able to perform reliable studies in a short period of time, and as it is not possible to perform experimental tests on the materials due to the associated costs and because it is not compatible with the time available for the project, the knowledge of the behavior of the alloys used is very important. First, at the time of the analyses performed during the mold design phase, there may be changes in the type of steel used taking into account its mechanical strength or thermal properties depending on the type of problem encountered. Therefore, it is important to have algorithms capable of constructing the curves that model the fatigue behavior of the steel from the information provided by the steel mills.

As computational power increases, it becomes possible to perform more and more refined calculations on more and more complex models. In this industry the same happens, both in terms of

specific software to study the behavior of polymers during the injection process and in terms of software for structural calculations. This evolution is fundamental to be able to meet the requirements of current markets by being able to predict from digital models the behavior of physical models and thereby reduce or eliminate the need to build test models. However, while for the study of polymer behavior during the injection process there are dedicated software's properly developed for this purpose, for the calculation of structural behavior this is no longer the case. Therefore, beyond the training and experience of the analyst engineer there are no dedicated tools. Thus, one of the principal points of this work is the study and determination of models that can characterize the fatigue behavior of tool steels. So, during the design phase different alloy steels from various steel mills can be studied without having to resort to destructive testing. And also, with this it will be possible to create models for calculating the fatigue endurance of injection molds that have as input data information that is easily obtained by the designers.

During this work, the focus will be on creating methodologies that are able to optimize the design and consequently the manufacturing of injection molds. As nowadays the use of digital models is quite common, and the use of numerical simulation is recurrent but many times in a dispersed way and without a good connection between the different areas involved in the project. Therefore, in a first phase, we intend to create tools that are able to connect the two major areas related to the design of injection molds. That is, it is common that rheological studies developed from numerical models and structural studies of the injection mold are performed separately. When one wants to create more efficient and cheaper injection molds it does not make sense to separate these two areas of study.

As part of the input data for the structural calculation of the injection mold is directly related to the behavior of the polymer during the injection phase, it is very important that these data are properly used in the structural calculation. Therefore, it is essential to create connections between rheological and structural simulation software in order to avoid over-dimensioning. With this it is expected that it can directly reduce the costs associated with alloy steel expenses and you can also size the mold according to the needs for the work you will be doing. Also, by using these coupled numerical simulation tools it is possible to reduce the number of tests that the mold will perform, because the problems that could appear in the tests related to excessive deformation of the mold components can be detected early in the design phase and solved at this early stage and not when the mold is already built, which can lead to numerous limitations in addition to the associated costs.

The methodology to determine the fatigue strength of injection molds is a very important point although it does not have a direct impact on reducing the mold manufacturing costs. Since injection molds are tools that will be used to produce thousands of parts and are subject to the loads inherent to their production, they end up being subject to alternating cycles that are well known. That is, once the

process is fine-tuned, the variation of loads inside the mold should be residual, because a change would imply a change in the injected part. Considering that the regions more prone to the appearance of fatigue cracks are not the molding region and its surroundings, the fatigue study in principle will not bring a great impact in reducing the mold manufacturing costs. But preventing the injection mold from having a fatigue failure during its lifetime leads to supply chains not being affected during the phase when the mold is producing parts. Given that parts stocks are very low these days, it is imperative that the possibility of a component failure in the production chain is eliminated or at least anticipated. In the case of injection molds there are two possibilities, if the possibility of fatigue failure is detected either the mold/part is modified to avoid the failure, or no change is made but replacement components are produced for when the failure occurs. Therefore, the study of fatigue molds is important if supply chain failures triggered by fatigue failures of injection molds are to be avoided.

With the realization of this work, it is intended to minimize the costs related to the manufacture of the mold using the computing power existing in the market and the software of numerical methods. With this it will be possible to build more reliable and robust injection molds and at the same time reducing the cost of manufacturing them, because both the consumption of raw material can be reduced like the number of tests performed to approve the mold.

For the studies of the fatigue resistance of injection molds, it is anticipated that it will be possible to significantly reduce this type of failure. That cost reduction that in many cases is extremely difficult to determine due to the impact they might have on supply chains. So, these studies are meant to avoid any negative impacts caused by fatigue failures in the supply chains. However, there are many factors that must be studied for a fatigue analysis. While material characterization is very important, it is not the only one. Here, it is necessary to analyze which theories are best suited to this type of problem as well as to study the factors that influence the fatigue resistance of injection molds. With this work is intended to simplify this process, by creating an application that will perform the fatigue calculations of injection molds based on a set of assumptions that will make it dedicated to this industry. With this, it is intended that the designer will only have to enter a set of data that does not require an exhaustive knowledge about alloy fatigue and thus be able to draw results that help him to perform a reliable analysis. However, it is necessary to take into account that the results will always tend to be conservative. But they help to avoid this type of failure in injection molds and thus increase the reliability of the product being developed.

1.3 Thesis framework

Given that the design of injection molds involves several areas and that so far tend to be studied separately or almost separately, this work will address several topics involved in the injection molds

design. In this section an approach will be made to what will be presented in each section so the reader can better follow this work and understand how each section will interact with the others.

Initially, the state of the art will be presented so that it is possible to have a perspective of the work done in this area. However, as this is a market niche, it was not possible to find many studies based on the study of the fatigue behavior of molds. So, the literature research conducted had to be a more subjective in order to take into account the various factors that might influence the behavior of injection molds. Still within the same section, so that the readers can better understand the terms used, a description of the basic components related to mold mechanics is made. Here it is intended to explain the reason for the use of each component as well as how it works in the injection mold. Finally, a brief description of how the numerical studies related to rheology and thermal efficiency of the mold are performed will be given. This type of numerical studies is the most common to be performed in injection molds, because they are directly related to the quality of the part that will be injected. But there are several nuances that should be analyzed because they have an influence on the structural behavior of the mold and consequently on its fatigue resistance. Although many of the concepts presented here are generic and theoretical, they are fundamental in order to understand how the design of the filling of plastic parts is made and how the cooling circuits are designed.

In the next section a theoretical approach will be made to fatigue concepts that will be useful in the way they will be used in this work to create an algorithm that will perform the necessary calculations to validate the fatigue resistance of the mold. Here we will start by presenting some theory about crack nucleation and how they propagate, so that it can serve as a handbook at the design stage for the explanation of the problems related to metal alloy fatigue that may be detected. Also in this chapter, a presentation of formulations related to the creation of S-N and ϵ -N curves will be made. Here the goal is to present to the reader of this work formulations capable of modulating the curves, where some of the curves can be built from parameters that can be easily obtained from tensile tests or even from non-destructive testing. Considering that a major goal of this work is the creation of an algorithm for calculating the fatigue strength of molds, the creation of the curves that model the fatigue behavior of steel is a very important and essential step.

Next, still in the theoretical concepts chapter, a brief compilation of the factors that affect fatigue life is made. Here a compilation will be made of the values to be used and a description of how they affect the resistance of steels to alternating cycles of loading. This section will help both in obtaining the values for the practical part of this work, and in designers using this work as a guide to have some of the main factors compiled in order to facilitate the design phase.

Finally, in the theoretical concepts section a brief description of variable amplitude loading will be presented as well as cycle counting methods and how to consider them in order to determine the life of the component under study. As well the description of some models of multiaxial stresses that will serve as a basis for the construction of the algorithm for calculating the fatigue strength of molds. Also will be presented some models that can be used in future work because they can bring more value in terms of accuracy of calculation and predict how the crack will form.

Then in section 4 the algorithms and applications that were developed in this work will be presented. Here we begin with the presentation of the application that will generate the data necessary for the construction of the S-N and ε -N curves from information easily obtained from the steelworks. Also presented in this section is an experimental validation of the curves and where the reason for the empirical model used in the algorithm is explained. This validation is important because the empirical model chosen will be used again in the construction of the application that will be used to calculate the fatigue resistance of the molds. Also in this section, an application will be presented that aims to connect the commercial software Moldflow, which is used to perform rheological numerical simulations, to structural simulation software, which in the case of this work will be the commercial software ANSYS. The last item of this section presents the algorithm and application that will be responsible for calculating the fatigue resistance of injection molds. This application will be the culmination of all the research work done and will then be tested in order to verify how it can help injection mold designers to prevent potential fatigue failures right at the design stage.

Sections 5 and 6 aim to test the applications developed and to study a methodology to be used in structural numerical studies in injection molds. In the first phase, it is intended to study how the use of loads obtained from the numerical software Moldflow can improve the structural analysis of injection molds and simultaneously understand how the simplifications can affect the structural numerical studies. These studies will be compared with experimental results in order to have a term of comparison and to understand how the results reflect the behavior of the physical model. At the end of this section a methodology will be obtained that will be applied to calculate the fatigue endurance limit of injection molds. In section 6, using the methodology defined in section 5, the structural calculations will be performed from where the necessary values will be obtained to perform the calculations to determine the fatigue strength limit. Here will be used for study five injection molds that presented fatigue damage before fulfilling their lifetime. Whereas it will be verified later the use of the application conceived for this effect was capable to identify several points where the fatigue damage would be probable.

Finally, the last chapter summarizes the conclusions that were reached throughout this work. Since in this work besides the theoretical framework there is a chapter where the presentation of the developed

applications is made and two chapters where these applications will be tested and also a methodology for conducting this type of study, a conclusion that incorporates all these related themes is important to better understand how these themes are intrinsically linked. At the end of this section some proposals for future work are also made so that there can be an evolution in this area of work and thus be able to make this industry more competitive and more capable of responding to market needs.

2 State of art and injection mold design concepts

To be able to analyze and understand the structural behavior of an injection mold, it is necessary to understand the components of an injection mold and what their functions are and how they work. It is also important to understand how the design of an injection mold is developed and why each type of components is used.

In Figure 1 is possible to observe the interactive process of the design of an injection mold. Because, to meet the design requirements for the injection mold to be delivered to the customer, it is necessary to perform several operations in parallel that may have an influence on each other.

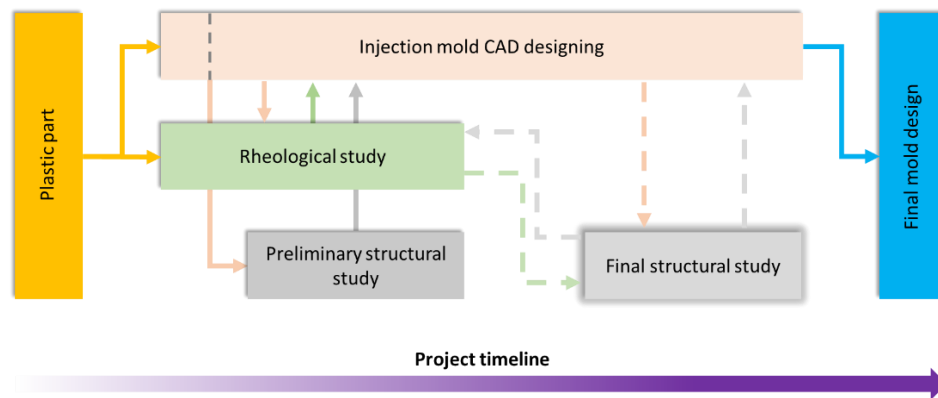


Figure 1 – Schematic of the design phase of an injection mold.

Once the 3D drawing of the part is received, the mold development work and the rheological study of the part begin. During mold development, all the necessary mechanisms for molding and demolding the part to meet the predefined criteria, are outlined. At the same time, a rheological study is carried out where the injection points will be defined to obtain a part that can meet the desired aesthetic and dimensional criteria. In addition to ensure that the pressures inside the mold is enough to meet the previous criteria but not too high, considering the clamping force of the injection machine and polymer material limits. During the rheological study, input data from the mold development will be used in order to understand if the used injection points and injection type can be applied to the designed mold layout. This process should be repeated until agreement between these areas of the project is reached.

In the first phase of the injection mold design a preliminary design is made, because according to the mold designer or due to indications given by the rheological study changes might be made to the injected part. But with this preliminary design it is already possible to analyze whether the injection mold is structurally well dimensioned, taking into account the loads to which it will be subjected. This study can also lead to changes in the injection mold design or rheological study taking into account the problems detected. Although the problems related to the filling of the part are more difficult to detect, because this study tends to be performed using a constant pressure along the molding region, making it a conservative

study. The study is conservative because a pressure value based on the experience of the designer is used, since at this stage usually there is no final rheological study that can be used.

In a final phase a structural study of the mold can be performed if the changes made between the preliminary and the last mold design phase are significant. At this stage there is already information about the pressures to which the molding area will be subjected, and all the components are modeled in its final form. Through this phase the study is done to validate the injection mold or to verify if it is necessary to make slight modifications or even the care to be taken in the injection profile in order to avoid aesthetic problems in the parts.

In order to understand the terms used throughout this work and to give a better understanding of the studies performed on the injection molds, the rheological and thermal studies will be addressed, since the structural studies will be covered later in this work. And finally, a presentation of the main components that are used in injection molds.

Figure 2 shows a plastic part that will be used throughout this section as an example for the presentation of the rheological and thermal studies of the mold. A preliminary mold design (Figure 15) for this part will also be used as a support to demonstrate different structural components that are relevant to the structural analysis and to understand the expressions used throughout this work.

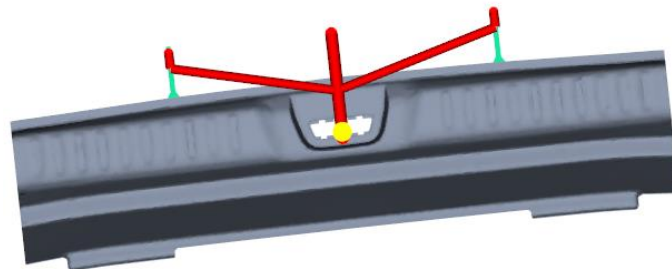


Figure 2 – Example of a plastic part and its injection system.

Before describing the work done in the different project phases, the state of the art will be presented so that the readers of this work can have a perception of the work already done within this theme or that may be important for the realization of this work.

2.1 State of art

The injection mold is a precision tool and one of the most common process to fabricate thermoplastic parts [2]. But to create such a tool an appropriate design process is mandatory since there are many variables and areas of physics involved. To understand the variables needed for the design of an injection tool, first is needed to understand the plastic part and how the mold is designed around the molded part.

The first stage in the mold design is to study the part geometry to create a sketch of the mechanics

needed to demold the plastic part and then create the first drawing of the mold. At the same time the injection concept is also being studied to integrate the first mold drawings where the general injection concept is made. If the part doesn't present any significant changes up to the end this first concept will stand with low or no modifications but if the part presents significant changes the injection concept should follow them, in Figure 3 is possible to see a scheme of the mold design process with the use of FEM analysis during the process.

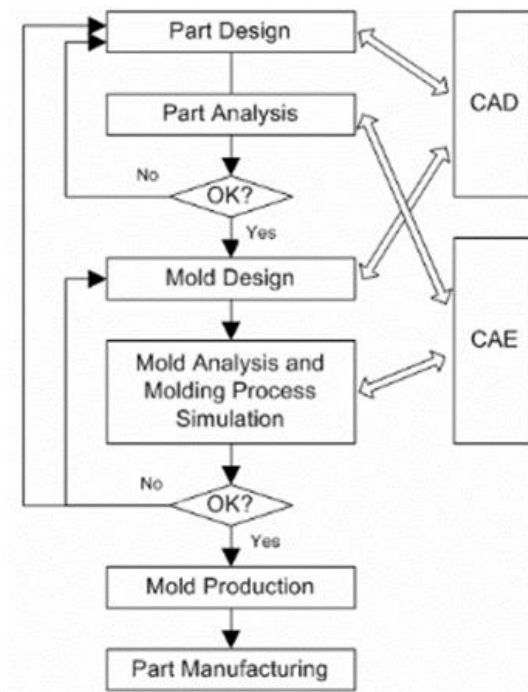


Figure 3 - Design and manufacturing process using traditional CAD and CAE systems for injection molding parts.[3].

As stated, the injection mold is a mechanical system that involves many parts of physics so all of them should be studied to work together as a whole. The main areas that are studied during the mold design are the feeding system, heat-transfer system, structural system and ejection system (mold dynamics) [4]. But most of the times they are studied separately which can cause delays and costs to the production of the mold and nowadays, the design and production time is reducing as the cost to produce the mold [4].

The main tasks involved in the project of an injection mold are the design of the geometry of the part (functional changes to eliminate ejection or injection problems), the mold type, machining, polishing of cavity/core surfaces, cooling lines, assembly of plates, pins and mold bases, and prototype tests. In injection molding, the design of the molds is of critical importance for product quality and to have an efficient production process. The mold design presents a big impact in production efficiency, as for the tool production as for the plastic part production [5], Figure 4 shows the three main studied areas in the injection mold design.

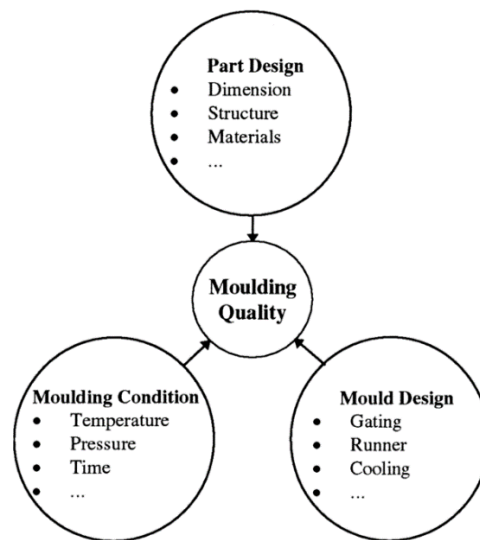


Figure 4 - Various factors influencing molding quality [6].

The other main areas that will be presented, are the injection process, the part cooling process, tool steel grade metallurgy and the structural mechanics design. The steel wear and corrosion as the surface rugosity will also be addressed because they can present an influence in other areas of study but they are also hard to quantify in a way they can be placed in the numerical models so designers have to keep this concepts in mind when analyzing the results.

2.1.1 Mold design

The design of thermoplastic injection molds needs to be done in the lowest time possible and most of the mold concepts are based in the designer's experience. Sometimes the experience comes from the organization that creates standards for each type of mold or by the lack of conception time which makes the designer use previous mold knowledge to make de mold concept without thinking in other conceptions and this method of work can create future problems that can lead to other costs. This procedure based in previous mold knowledge can be schematized as presented in Figure 5.

Nowadays the recourse to CAD (computer assisted design)/CAE software's to reduce the design and manufacturing cost is unavoidable. Matin, et al. [7] presented a work where a software with a specially developed module for the calculation of injection molding parameters, mold design, and selection of mold elements were used to present a software solution for multiple cavity mold of identical molding parts, the so called one product mold.

There are also research made about the introduction and application of CAD, CAM (computer assisted manufacturing) and CAE solution in the production of injection molds [8]–[12]. They present concepts of using CAD/CAM/CAE technology for design the injection mold where it is possible to agree that this new technology improves the capability to design molds, easier and with more complex injected parts.

As stated by Alexandra [8], with “the software’s capabilities, we can easily generate the core and cavity blocks. As a result, this new system provides a powerful, feature-based solid modeler that allows users to make modifications to parts for mold design”. Deng et al. [9] made a research on mold design with the objective of creating a CAD-CAE integrated systems, which they reached to a feature based integration model and an iterative design analysis process based on the model.

With this research, it is possible to conclude that CAD/CAE/CAM can reduce the design time and the cost by finding the problems in the virtual model instead of realizing of these problems in the real mold. The level of integration in the software is mostly related with the company size, small companies can have all features integrated in the same software but when they reach to a certain size, they need to spread these features to optimize their work and specialize their employers on each area. Zhou et al.[13], presented a paper where they intent to make a low-cost virtual injection system. With this work is possible to have an overview of the formulation and simplifications made to calculate the cooling and filling results. Another focus of their work is the simulation of motion to detect collisions, but here they weren’t able to find a suitable solution. The detection of collisions is very useful since in injection molds could have innumeros moving parts and if there are any mistake in the design of their path it could compromise the mold integrity and functionality.

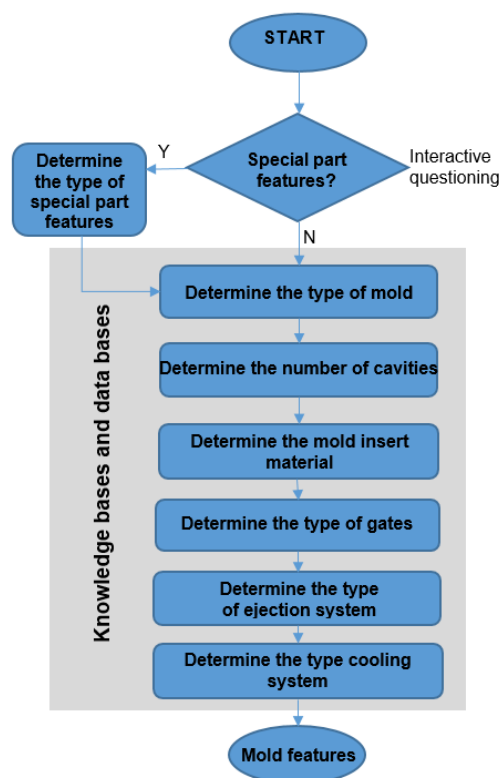


Figure 5 – Scheme of the mold design based in previous knowledge [14] (adapted).

Other base of work is the standardization and automatization of several steps [15], [16]. The mold is a mechanical system that involves innumeros parts and each mold maker have his own rules as his

clients, but even with this different sets of rules there is always ways to create standards. With the generalization of CAD systems and information present on the internet [17], [18] this standardization is getting easier to make and there are many studies about this theme since this is one of the processes where nothing can fail and the time to deliver results is getting smaller. Which is becoming more and more indispensable to facilitate the design of a complex system as is possible to observe in Figure 6.

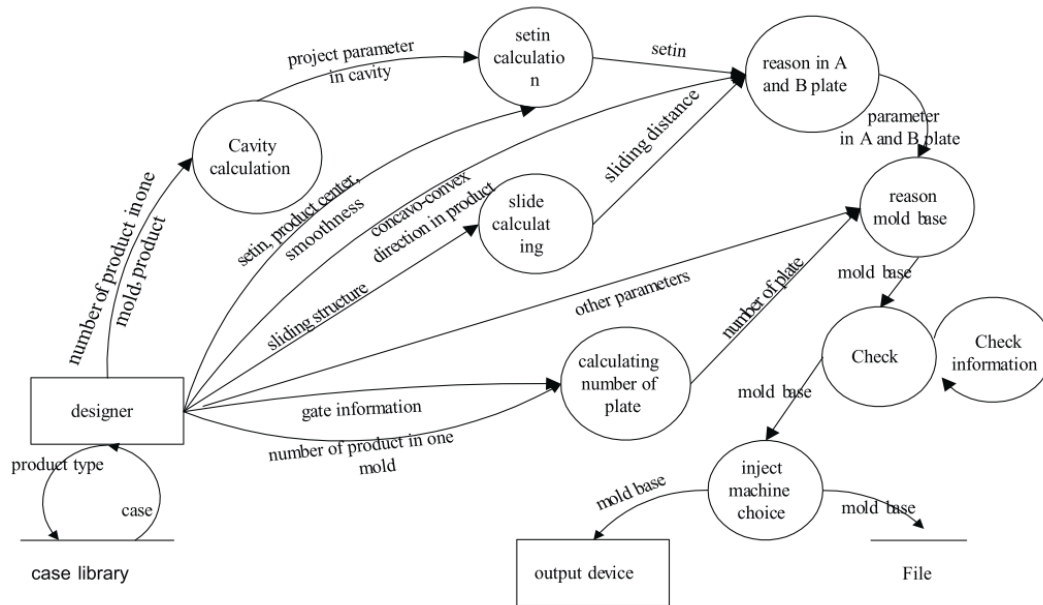


Figure 6 – The data flow diagram in mold-base design [11]

2.1.2 Use of FEM in injection mold design

As stated before, the mold industry to be competitive must be able to do it right the first time. To do so, simulations are a huge tool to make sure that for mold trials there are no problems. So as stated by Bakharev and Astbury in their paper, “analysis of ejection is an important step in designing injection molds. Ejection forces, stresses and strains during ejection are difficult to estimate without a rigorous finite element analysis. Incorrect estimation of those parameters can lead to a number of molding defects including: ejection marks; ejectors punching through the moldings; and increased warpage of moldings during ejection phase” [19], so Bakharev and Astbury made a research about this theme where they could conclude that a combination of mold filling and mechanical simulation can be used to make numerical simulations during the ejection phase of the injection molding process.

For instance, as can be seen in the paper presented by Giacomini, et al. [20] where they studied the core deflection of an injection mold using a fifth order nonlinear ordinary differential equation using a Newtonian fluid to fill the mold and considering the cavity as a rigid body. Chen, et al. [21] also made a work with the same objective but here they design the experiment with an insert, simulating the core where they made numerical studies as well trials with an injection mold. This evaluation brings good results when the cavity presents a rigidity so high when compared with the core as they had in their

research. But this is a huge simplification to make, for instance, if it is considered an injection mold for a car bumper the cavity will be subjected to huge forces in the plane XY (considering the vector Z in the injection direction) as it is possible to see in Figure 7. In Figure 7 b) can be observed that there is a significant displacement of the injection mold core caused by the pressure exerted by the molten polymer.

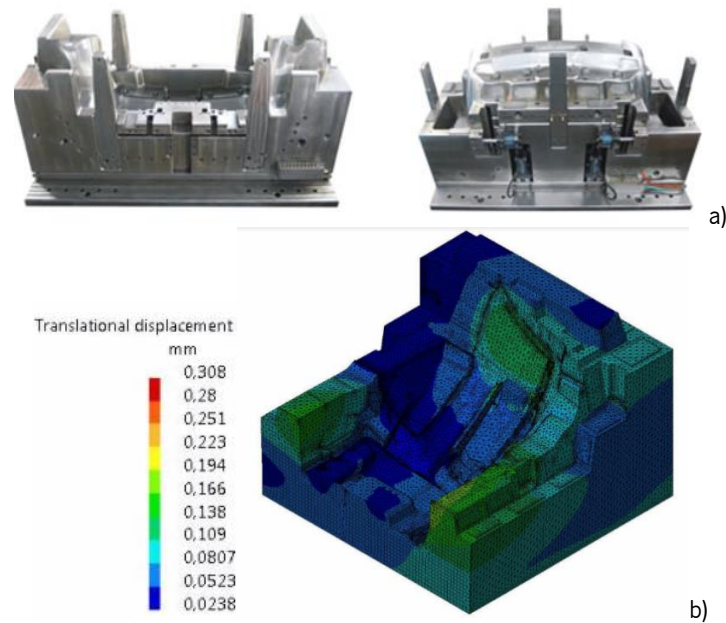


Figure 7 – Injection mold of a bumper; a) Injection mold for a car bumper [22] b) One example of the core deflection of a bumper injection mold at the maximum injection pressure.

The use of CAD and CAE systems are getting more common in the mold industry but most of the times they are used separately which it can reduce the flexibility of the work flow and the synergy that could be present with these systems can be lost. An overview of the synergy with the use of CAD-CAE integrated systems is presented by Deng, et al. [23] in their paper. But it is also possible to find some works where the authors resort to coupled simulations in their studies, i.e., they perform the structural simulations using the loads obtained from rheological simulations. Works such as the one presented by Carpenter, et al. [24] where a study of the deflections of a mold during the compaction phase is performed, where without a good definition of the pressures in the molding region, it would be very difficult to perform a study capable of modeling the problem. Or the work done by Wu, et al. [25], who using coupled simulations can make predictions about the effect of mold deformation on the shrinkage of the part and warpage.

2.1.2.1 Use of CAE systems in the part design

The use of CAE systems is commonly used to filling, cooling, warpage analysis and others. The injection process has many variables and some of them have to be taken under account at the design. If don't, it could cause the need to make big changes on the existing mold or even the obligation to make a new mold. So to prevent this from happening the injection concept should be made and tuned at the

same time as the mold design and here the simulation results help mold makers to optimize the plastic part, the injection system and the mold cooling system [26], [27] and with this all production costs can also be reduced.

2.1.2.2 Material characterization

To create numerical simulations with good results there is a need to create experiments to characterize the materials, the “fluid is defined as a material that deforms continuously if a shear stress is applied” [28] and in the case of mold injection there are innumerable factors that can influence the results.

One of the main problems in rheology numerical simulations is the data quality as stated by Sethuraman & Palanisamy [29]. In their paper they bring out the challenges that are encountered in the current simulation methodologies and discusses some methods practiced in the industry to overcome these challenges. They also stated “that traditional material characterization methods might under-predict or over-predict the stiffness of a structure depending upon the fiber orientation and direction of loading” [29]. So, as it is possible to see in the Gou et al. paper [30] a characterization is made of a material and compared with the online data and with the data presented in Moldflow (commercial software). Some differences could be found between them in most characterization parameters. They stated “that the choice of the correct rheological data was significantly important to accurately predict the performance of shrinkage and warpage and to evaluate the effect of varying the processing parameters variation on final part properties” [30]. Although the material characterization method is relatively standard, where the use of rheometers is fundamental, such as those shown in Figure 8, the data obtained must be usable in numerical simulation software, usually from mathematical models. The most common method to model the viscosity is presented in this work, most software’s use the Cross-WLF model to characterize the viscosity. This model presents a good approximation of the behavior of melted polymers, but other viscoelastic models can be applied as it is possible to see in the work presented by Isayev and Hieber [31] where they use the Lenov viscoelastic constitutive equations and they also present a comparison between elastic and inelastic models.

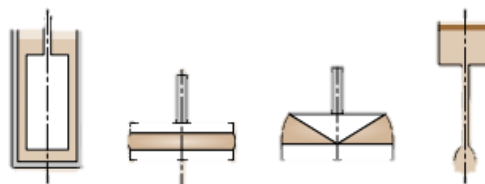


Figure 8 - Four commonly used rheometers; from left to right: coaxial cylinders in Couette flow, coaxial parallel plates, coaxial cone and plate and cylindrical capillary [28].

2.1.2.3 Rheology simulations

For the part filling, there are innumerable factors that could affect the plastic part quality and “the

injection mold design is critically important to product quality and efficient product processing” [32]. With the appearance of new technical polymers, the knowledge of how the material behaves during the injection and how the injection parameters influence this behavior is fundamental. The study of the injection parameters as well the gate dimensions have been studied for many researchers and mold makers but the results aren’t always consensual [33]. Adding to this problem, some ways to improve the injected part quality can create problems in the mold resistance, so proper studies of the injection parameters must be performed to balance the problems.

Lakkanna, et al. [34] presented a work about the design of mold sprue injection molding parts. This is an important system in the thermoplastic injection molds since this is one of the main factors that will influence the quality of the plastic part and the stress which the mold will be subjected. This work was made taking under account a resin injection and they presented a new outlook to assess its injection moldability. This cold sprue systems in some industries are being replaced by hot runner systems due to the capability of injecting in different part positions and giving the control of choosing the time of opening/closing of each nozzle to improve the plastic part quality. As stated by Grillet et al. [35] flow instabilities during injection molding of polymer melts like the ones shown in Figure 9 can cause non-uniform surface reflectivity on the final part, and here, this new technology can reduce this problem.

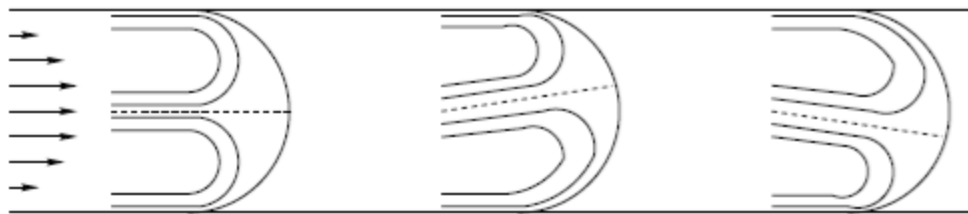


Figure 9 - Unstable flow which causes surface defects [35].

To reduce injection problems the introduction of hot runner systems, like the one presented in Figure 10, with open gates or valve gates have proven to be a good choice to reduce or eliminate many injection defects with the benefit of reducing the waste of plastic since there are no sprues or they are very small, these systems are widely used by the automotive industry [36]. Actually, nowadays electric valve gate systems are being introduced to give a bigger control of the injection pattern, instead of having a gate that opens at a steady rate it is possible to have a control of how much material can pass through the gate at a determined instant.

Transient finite element techniques can be a powerful tool to examine the stability behavior of complex flows. Linear stability analysis shows that the most unstable eigenmode is a swirling flow near the free surface which is in qualitative agreement with the experimentally observed flow instability [35].

An experimental work made by Hassan [37] with two polymer materials, one being the polystyrene (PS) and the other low density polyethylene (LDPE). The mold cavity has a cuboidal form with two different

thicknesses. The cavity pressure is measured through time using a Kistler pressure sensor at different injection molding cycles. The results show that the increase of packing pressure by 100 % increases the cavity pressure 50 % in the case of PS and 70 % in the case of LDPE. They also show that the increase of injection pressure by 60 % increases the cavity pressure 36 % in case of PS and 90 % in case of LDPE at an injection temperature of 220 °C. The results indicate that process parameters have an effect on the product weight for LDPE greater than PS. These studies are also relevant to understand how the tool should be designed to support these pressures.

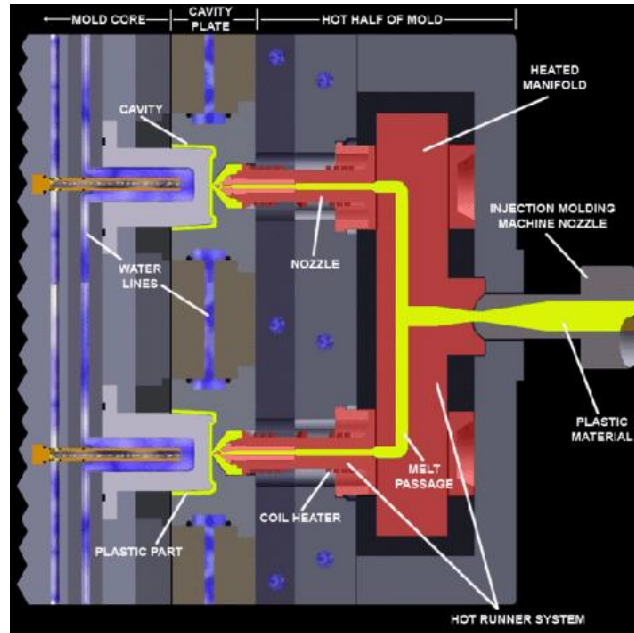


Figure 10 - Valve gate injection system for thermoplastic injection molds [38] .

Bociaga, et al. [39] made an investigation about the influence of injection molding parameters (injection velocity, mold temperature and injection temperature, and additionally as a result of these three parameters change, injection time, hold time, and cooling time) and blowing agent percentage on selected properties of HDPE (high density polyethylene) molded parts such as weight, density, mechanical properties (tensile strength and elongation at maximum force), surface state (gloss and color), and structure. Since blowing agents are known by reducing the mechanical properties of the plastic part, they have made some conclusions about the ratio of blowing agent the mixture should have to balance the loss of mechanical properties and the reduction of the part weight. With this they could get to the conclusion that the mold temperature has a significant impact on the quantity and size of pores in the parts. Also, the addition of 2% of the blowing agent for HDPE is sufficient to obtain parts with the good surface quality, and with the mechanical properties similar to the properties of solid plastic parts. The advantage of using this quantity of blowing agent is the possibility of reducing packing time from 31 seconds for solid parts to 24 seconds for porous parts, with no significant impact on their properties. As for Yuwen et al., they presented a study of Cast filling [40], although this simulation is based in the filling

a mold with liquid metal, the filling principles are the same and in their paper it is possible to see mathematical models and the simplifications made in this simulations where they also were able to present a basis to process optimization that in part could be used in thermoplastics simulations.

The use of finite element methods (FEM) in this area is very important to prevent defects and to optimize time and costs, so there are researches in this area to check the best techniques and parameters to reduce this design part. The most simplistic simulation that can be performed is using 2D elements like in the research made by Shin and Lee [41] where was used this mesh type and considering a modified volume of fluid (VOF) method. They made a comparison with experimental results which corroborated their model. But this kind of mesh could present problems with chunky parts [42], so it is needed to pay attention to the part geometry before defining the mesh type to used.

Liu, et al. made experiments and numerical simulations to understand how a rib thickness and geometry influences the sink marks in thermoplastic injection parts [43]. Their work was based in the sink marks made by corner surfaces like the ones seeing in the Figure 11. With this work, they were able to reach some conclusions about the effect of the corner geometry on the sink marks. Also, they used different materials, injection pressures, packing pressures, melt temperatures and mold temperatures. As for the cooling time they used different times but that's related to the thermoplastic material.

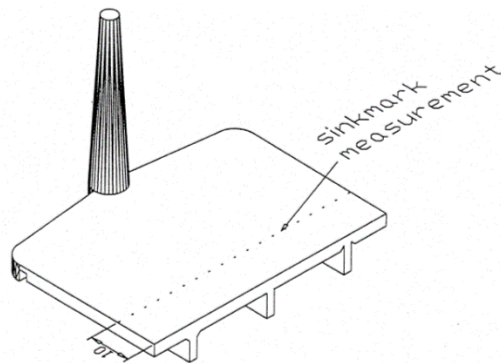


Figure 11 - Plastic part with different corner concepts to study the geometry effect in sink marks (LIU, LIN, & WU, 2001).

With this work, they draw some conclusions like the big influence of the ribs corner geometry in the sink mark effect where they could conclude that a rib with an undercut shows lower sink marks. Also, they stated that a numerical analysis can help to reach to conclusions during the design phase. As for the paper presented by Tang, et al. [44] where is studied a plastic part considering the thermal load and stresses in the mold to check its influence in the part warpage. From the study it was concluded that cooling channels also have a major part in the shrinkage since its more likely to occur nearby the cooling channels.

FEM analysis are also used to study the influence of parameters during the injection like the work made by Rusdi et al. [45], where they studied the effects of pressure and temperature on the melt filling

for a thermoplastic. With this work they could state that pressure is the dominant factor during the filling since this is the factor that presented a highest influence in filling patterns and velocity profile. Taking under account a real example can also be observed in the work made by Aisa, et al. [46] where rheologic simulations of a 2400l solid container were made and also considering the cooling stage during their simulations. In this works the authors concluded that the use of CAE tools didn't get the intended results when compared to the previous knowledge from other tools or even the injection mold trials.

The use of finite elements is broadly used in this area, but with the computational power of nowadays even with more complex parts it's possible to make good optimizations of the injection parameters during the mold design. One of the methods to optimize the filling parameters is with the Taguchi optimization technique, for instance Oktem, et al. used this method to reduce the part warpage problem related to the shrinkage variation [47]. This method is based on orthogonal arrays and the number of experiments which may cause to increase the time and cost can be reduced by using Taguchi technique. It employs a special design of orthogonal arrays to learn the whole parameters space with a small number of experiments only. So, Oktem, et al. showed with their work the powerfulness of this technique to optimize the part quality, in this case to solve the warpage problem with their part. Chen, et al. [48] have used a DOF (design of experiment) also with the Taguchi method and the ANOVA (Analysis of variance) method. That is a more complex and accurate method to minimize the warp in a thermoplastic part where they were able to reach great results even when they made an experiment the measured results and the theoretical results were close, so it was possible to prove the obtained results. These techniques can also be used in other areas of the mold design to improve the tool performance such as the mold cooling and the structural design. These are the more used techniques to reduce the part deformations, but other techniques are also used as it can be seen in the work of Gao & Wang [49], where they used a modified rectangular grid and the Kriging model which presented satisfactory results in minimizing the warp in a cellular cover.

Another area under development is automatic gate positioning, instead of using the experience to get the starting point for the gate locations, there are many works made to create an algorithm to present an optimal positioning of the gates [50]–[53], these algorithms can present good results for simple parts but with more complex parts and with sequential injection the results aren't satisfactory. Another area which presents a challenge and can reduce the design/optimization time, is the automatic settings for the injection process parameters [6], [54], [55] these works, like the works made for the gate automatic positioning aren't able to present good solutions for sequential injections and nowadays this kind of injection is very common for complex or aesthetical parts. The determination of the injection parameters isn't a trivial process since there are many factors that can influence the part quality such as melt

temperature, injection pressure, screw velocity, holding pressure and duration, cooling time and temperature and many others. All of these factors interact directly or indirectly with each other [56], [57], which makes their tuning a very complex problem.

2.1.2.4 Mold cooling

During the plastic part design the cooling process is very important to reach a good quality part and production cycle [58], even with a good filling parameters if the cooling isn't homogeneous the plastic part could present esthetical or structural defects. Sometimes non-homogeneous cooling is also used to reduce the part deflection, but this could also induce internal stresses that later will prevail, and the part will deflect into its resting position. This process can take long processing times and computational resources so, some works on this process have been done to improve this process like the one presented by Zhou et al. [59]. Where in their work it is presented an acceleration method in order to improve the computation time where they reach to a cost-effective method by splitting the dense BEM matrix into a sparse dominant matrix and a dense residual matrix. Other important research for the cooling cycles is the transient simulations, with this is possible to have an idea of the behavior of the mold since the beginning of the injection process until stabilization so it is possible to check at the work made by Tang et al. [60] where they studied the mold cooling using a three dimensional by making a transient simulation, for this model they used the Galerkin finite element formulation. With this work, they could validate the algorithm for the studied mold.

As stated, the cooling process is very important but also hard to design. There are innumerable problems during the design of the cooling systems being the space needed for the cooling channels the biggest one. It isn't just the plastic part geometry that creates difficulties to place cooling channels in a way they can cool down the part homogeneously. The traditional way of designing the cooling channels is based in the milling of the steel block and there are researches made in that way but, two different approaches can be seen. The first is based in drilling the mold block (more traditional). The research is made in order to reduce the time to design the channels proposing algorithms to create an efficient cooling [58], [59], [61]–[63] where all of this methods are based on the principle of reducing the time of design and optimize the cooling of the plastic part. Where in Figure 12 is represented a scheme with a common method to make the cooling system optimization.

The other approach is the change of the way the cooling channels are made [64], instead of drilled cooling channels it's proposed grove milled channels. With this is expected to increase of the cooling rate and have the cooling more homogeneous through the surface of the plastic part. This last method could be very expensive and sometimes almost impossible to do in most injection molds, so the design of inserts with more conductive materials, sometimes are used to improve the cooling near the most

complex geometrical regions. Materials such as copper-alloys which can have an impact on the cooling process, that can be improved up to 29% when compared to conventional tool steel [65], but this technique could lead to visible marks in the plastic part that are not aesthetically acceptable. So, the introduction of new manufacturing methodologies like the rapid prototyping, came to help solving that problem. With this technique, the change of cooling rate isn't so high, but it helps to keep a more homogeneous surface visual quality by its ability to place cooling channels near the most critical regions.

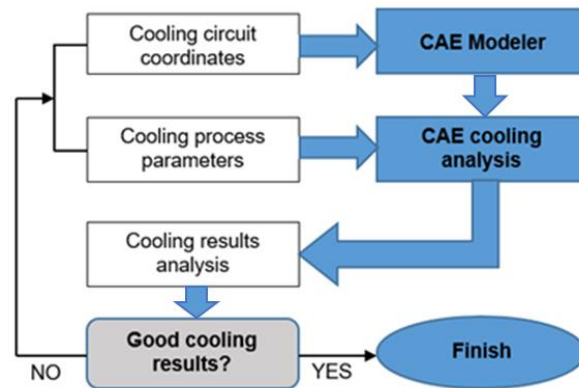


Figure 12 – Cooling system optimization in plastic injection molding using CAE tools [58] (adapted).

With the evolution of the production systems nowadays it's possible to design cooling channels without the limitation that the conventional method presents. When channels are drilled, the channels are made along a straight line. But with the 3D printed inserts (made by rapid prototyping) with conformal cooling channels, they can follow a path along the insert surface making the cooling more efficient like the one presented in Figure 13.



Figure 13 - Conformal cooling channels example made in one insert.

Some research made about this theme where the authors checked diverse ways and materials to improve the cooling rate. The materials are very important due to their thermal conductivity and most of the printed metallic materials present a lower thermal conductivity than tool steels so it is important to study the most significant parameters as studied by Kovács et al. [66]. Nowadays there are tool steels with a high thermal conductivity allied to good mechanical strength but, the conformal cooling enables to place cooling channels in places where traditional drilled channels don't reach, so understanding which parameters are more influent is very important. Since the steels used in this technique present a

lower thermal conductivity, the ability of this production method to print different materials in the same part, could be a way to work around the problem of the heat exchange rate as can be seen in the work made by Saifullah et al. [67]. Where they studied an insert with a different material in contact with the refrigerant liquid where they could conclude that this technique can increase the heat exchange and with this reduce the cycle time.

Also to improve and optimize the shape and path of the cooling channels since there are rules to follow, approaches for the design of these channels have been proposed and studied and this might lead to a set of rules for each kind of printed material or fabrication methodology [68], [69].

2.1.2.5 Simplifications made in numerical analysis for rheology

When creating numerical models sometimes it is too difficult to recreate the problem as it is, or sometimes simplifications are made to reduce the computation time, so it is important to understand these simplifications and how they can influence the results. For instance, “mathematically a single node definition for a finite radius injection gate imparts a singularity” [70]. In the paper of Modi, et al. an approach to avoid this singularity by modelling the injection gate with more than one node is presented. An analytical solution relating the fill time to the injection gate radius is developed for a constant pressure injection from a spherical injection gate into an isotropic media. The accuracy of the results depends on the ratio of the flow-front radius to the injection gate radius as well on the mesh density level. In many situations, a spherical injection gate may not represent the correct physics and a different model simplification may be necessary. It is recommended to use proper geometric definition for the injection gate, wherever possible. If proper modelling of the injection gate is not possible, the injection gate should be modelled using all the nodes at a distance less than or equal to the injection radius. In extreme cases, where only one node is positioned within the injection radius, it is permissible to define the injection gate using a single node. The results from such models should be analyzed with extra precaution [70]. This study was made for an RTM-injection process but this is also valid for the thermoplastics injection process since the injection gate is a region where the plastic will be forced to pass through different shapes, fast and with a lot of pressure. So, this is one of the regions where extra precaution should be taken to reduce the study errors.

Taking into account the material properties and numerical simplifications Zhuang et al. [71] made simulations to predict the flow-induced stresses during the mold filling, in this simulation they considered a non-isothermal two-phase flow model in a three-dimensional mesh. They concluded that the 3D simulations presented more accurate results than 2.5D simulations and could effectively predict the flow-induced stresses.

In software's like Moldflow, three mesh types can be used to mesh parts. Being these meshes 2D, 2.5D and 3D where each one of them have an area of application. Of course, the 3D mesh can be used at any situation, but it increases the calculation time. So usually if the filling is the only problem to be studied the mesh to be used will be chosen taking into account the best correlation between the results accuracy and running time. The 2D mesh can be used in flat parts or parts where the thickness doesn't present significative variations, the 2,5D mesh can be used in almost all the plastic parts presenting good results and reduced computation time when compared to 3D mesh. But the results might not be very trustful when presented for chunky parts, here the 3D mesh should be used since it can represent what happens inside the part without any estimations like the ones made in 2D and 2,5D mesh [42].

Pillai, et al. [72], [73] presents an extensive work about the injection/compression of a liquid composite as Han, et al. [74] made a similar work using the control/volume finite element method. Lam, et al. [75] made a paper in which they used numerical simulations to recreate the mold filling in a resin-transfer molding process based on the finite element control volume approach, as for Voller & Peng [76] used a volume of fluid (VOF) free surface technique to formulate the problem and they were able to validate the results with experiments. Yoo & Lee [77] presented in their paper a study using the boundary element method for the numerical model also for a resin transfer mold. With the works made by Simacek & Advani and by Tan & Pillai [78], [79] is possible to get an overview of the model constraints and an overview of the mesh parameters. Although these papers are about a different plastic processing method most of the simulation principles are similar.

To create the numerical models is fundamental to have formulations that can describe the phenomenon's during the filling process, so the creation of formulations with the parameters which can describe the "behavior of fluid polymers in an unbounded range of variation of recoverable strains" [80] as it is possible to check in the work presented by Leonov [80].

2.1.3 Metallurgy, surface wear and corrosion

To understand the mechanical behavior of the alloys used by the mold makers must be understood how materials are made and their metallurgical properties, this are important factors to comprehend the differences between alloys.

O'Brien & Hosford [81] presented a paper about the Spheroidization cycles for medium carbon steels in the bolt industries, this is a very important component in molds since it is responsible to the good functioning of the mold, keeping elements together and supporting the mold during transportation. With this paper, they show some conclusions but the most important for the wear and resistance theme is that formability correlates well with degree of spheroidization but not with hardness changes or particle size. As for Kim, et al. [82] made a study of a multi-step forging process of subminiature Torx screws

where they also made improvements in the die to remove crack defects and folding defects. Here, with their research they present theories that a two-step forging process can be used to reduce the formation of cracks.

The steel composition and treatment have a huge impact in the mold performance, as by its stress resistance, thermal conductivity as by its corrosion and wear resistance. So there is a need for a constant research in this area to improve the tool steel performance like the work made by Cao, et al. [83] where they study the hardening response for metal alloys. The introduction and the percentage weight of alloy elements present a challenge when there is a need of a good balance of so many factors.

The wear of the mold surface is a very important factor, Brezinová & Guzanová [84] studied this factor with tow alloys used in molds, such as Alumecc 89 alloy and Moldmax HH alloy, with different filler contents. With this work, they were able to make a relationship between the wear resistance and the heat exchange of this alloys. As for the work presented by Lindwal, et al. [85] presented experimental results and calculations to corroborate the conclusions from the experimental results about corrosion in tool steel. Where they state that carbon steel grade presents a more critical influence on the corrosion resistance of the stainless 316L steel in comparison to the nitrogen tool steel grade.

Different working conditions for the mold require different steel grades and their resistance to the wear and corrosion are crucial factors for the steel grade choice. Usually the wear resistance is the principal factor for the choice of steel, since the used tool steels usually present a good corrosion resistance and as stated in Polyanskaya paper [86] “the impact wear rate depends on the impact energy, ductility, hardness, and phase composition of steels”. So, the steel grade choice is dependent of the material to be injected and the injection conditions. In some cases, common milling operations can’t be made due to the part geometry, so the EDM process (electrical discharge machining) has to be used to create the pretended shape. Very often, these are regions subjected to high pressures and wear so the finishing rugosity needs to be studied to reach to the best rugosity parameters, like the work made by Schneider, et al. [87], [88] where laboratory tests were made with different materials and surface finishing’s so it could be discovered the influence of surface finishing in these materials and propose finishing methods.

2.1.4 Structural mechanics

A big part of the injection tool design is in the capability to support the loads inside the mold during the injection process. To understand that it’s needed to understand how materials behave when they are loaded as it was done in the paper of Fukaura, et al. [89]. These works help in the understanding of materials behavior when subjected to alternated loads and with improve the choice of the steel grade and treatment of the steel.

2.1.5 Compression loading

There are innumerable regions in injections molds that are subjected to compression loads, and as known the metals have a poor resistance to compression loads. So, the comprehension of correlations between the mechanical properties and its failure properties under different loads is indispensable [90], [91]. Usually most of these works are made for production processes that involves the deformation of the part until it reaches its plasticity like in the forging process. In this processes it is also important to understand the crack formation as the work made by Fuertes, et al. [92] where they made their research using FEM analysis taking under account different geometries in a forging process.

Toda, et al. made experiments about the energy absorbed by a specimen during static and dynamic compression loads [93]. With this work it's pretended to show the influence of the mechanical properties of steel sheets and the influence of the thickness. As for Liu *et al.* [94] also made a work about the deformation of low carbon steel, but with the effect of the temperature, i.e., the hot deformation behavior during compression at elevated temperatures.

As for Dorogoy & Rittel [95] they made a validation of numerical study of the shear compression in a specimen with a quasi-static large strain where they made accurate correlations between the averaged Von Mises stress, strain and the applied experimental load and displacement. They state that these relations depend on the specimen geometry and the material properties, and were able confirm the potential of the shear compression specimen for large strain testing of materials.

2.1.6 Fatigue

The mold failure by fatigue is one of the main problems when mold makers are making the mold design. So, it's important to understand how to prevent and what are the main factors. Some factors are studied in the work made by Vosilovsky [96] with the study of the stress ratio on fatigue crack growth rates, or the effect of small defects on fatigue strength as studied by Murakami & Endo [97]. The work presented by Knoerr, et al. [98] where fatigue failure of cold forging tooling is studied and causes where presented with some possible solutions through fatigue analysis. Or Eid & Thomason [99], where they studied the nucleation of fatigue cracks when a low alloy steel is subjected to high cycle loads. All of this works help to create tools and methods to help to predict fatigue cracks during the design.

Zheng in his paper presents some basic problems of fatigue [100], such as “the basic fatigue formula, the criterion for omitting small loads, the rule for cumulative fatigue damage, and models for predicting the fatigue crack nucleation, life and probability distribution under variable-amplitude loading are comprehensively studied and summarized”, but at the end of his paper he states a very important problem especially in the main core of this paper that is the “structural elements often operate under multi-axial stress conditions” which makes harder to study by conventional methods the life cycle of the

mold elements. To have a better comprehension about the theories Cui [101] made an extensive research about metal fatigue in particular in the life prediction methods. In this paper was stated that the main factors that affect the fatigue life of metal structures can be grouped in 4 categories: material factors; structural factors; loading factors; environmental factors. At the time the author found innumerable problems with the theories in study such as the prediction of fatigue life under variable amplitude loading was unsatisfactory, the transferability of fatigue data from small specimens to actual components or structures and the corrosion fatigue were completely unresolved. Also, the combination of high temperature and fatigue was largely unsolved. The multiaxial fatigue especially when there are no fixed correlations between forces in different directions had work to be done to have good results. Other works such as the one presented by Colin and Fatemi [102] based on the study of the fatigue behavior of a stainless steel 304L alloy for different alternating loading conditions. The ones presented by Borodii and Shukaev [103] and McClaffin and Fatemi [104] that studied the effects of the strain hardening on the material fatigue endurance resistance and many others helped on the evolution of the models and how to consider the many parameters related to this theme by improving the capabilities of fatigue failure prediction.

Lynn & Duquesnay and Jono [105], [106] checked the crack initiation behavior when a specimen is subjected to variable amplitude load. But, Lynn and Duquesnay used computer simulations, from where they were able to make quick fatigue crack predictions for a wide range of alloys using inputs based on nominal stress spectrum, a maximum spectrum stress and a net stress concentration factor. To validate their results, they made experiments to check the correlations. As for Jono, experimental studies were performed where they studied the plastic strain cycling and the effective stress intensity factor range, being these the important factors for fatigue damage accumulation and fatigue crack growth. In his paper it's presented a scheme of the method for fatigue life prediction as shown in Figure 14, and he was able to reach to interesting conclusions for injection mold fatigue design: "the linear cumulative damage law, when expressed in terms of the plastic strain range-pair was confirmed to give a good approximation for life prediction under variable amplitude loading through from the low-cycle to the high-cycle fatigue regimes"; there exist various complex factors which are difficult to understand such as environmental effects, effects of the transition of cracking mechanism due to load variations, effects of non-stationary load variations, and differences of material behavior, and others.

For the study of the linear cumulative damage law, there are numerous works that vary in their conclusions according to the test conditions and/or materials used. One can find works such as Manson and Halford [107] who present a work where the double linear damage rule is studied and conclude that this model tends to be more conservative but also more accurate than linear damage rule. However the

linear damage rule is one of the most used due to its simplicity. As presented in some works, it can be observed that this model in most cases is able to predict with a good degree of confidence the fatigue endurance of components [108]–[111].

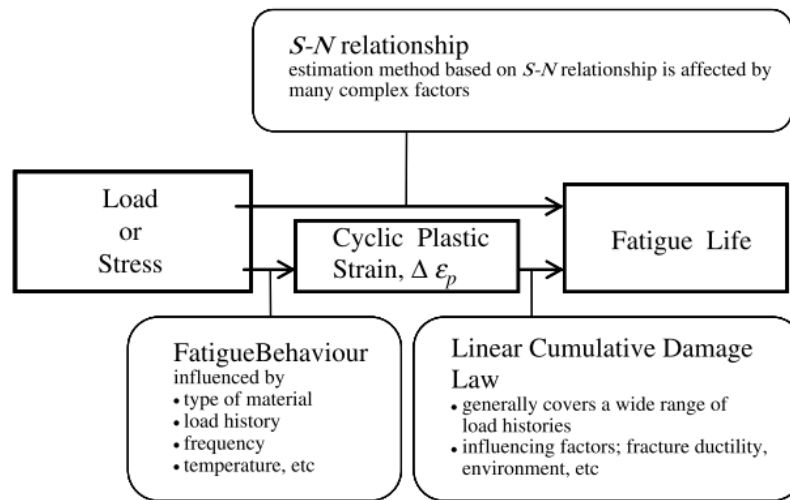


Figure 14 - Scheme of the estimation method for fatigue life [106]

Since there are elements that can fracture at a low cycle, *i.e.*, in steels it's considered that low cycle fatigue is when it's estimated that they crack by fatigue between 1 and 10^3 cycles [112]. In this theme Kuroda [113] presents a study where he aims to predict, for an extremely low life fatigue life of annealed low carbon steel, by presenting a new cumulative fatigue damage model. And states that “the predicted fatigue life”, using the proposed model “coincided well with the experimental results obtained by conventional fatigue tests and fatigue tests under mean strain conditions”. As for the surface crack evolution, Polák et al. [114] studied the profile evolution and fatigue crack initiation when applied a cyclic loading on a specific steel where they present three dimensional information about the persistent slip markings profiles obtained by their experiments and discuss their results comparing them to the physical models of surface relief formation and fatigue crack initiation. Other works where studies are presented for the determination of low cycle fatigue failure can be easily found, such as the one presented by Chen, et al. [115] where their studies are based on non-proportional alternating loads where is proposed a novel fatigue endurance prediction model based on the critical plane method. But also other works that are dedicated to the subject of low cycle fatigue failure in non-proportional alternating loads present parameters to be taken into account in this type of analysis and models for determining fatigue endurance considering the range of materials and test conditions [116], [117]. McKelvey, et al. [118] present a paper where uniaxial loads are studied but based on the Forschungskuratorium Maschinenbau guideline. In this work the guideline method is addressed and a presentation of the effects that impact the fatigue strength and respective reduction factors to be applied to the S-N curves is also made. But considering that in many components the type of loading to which they are subjected is multi-axial, the study of their

behavior and determination of their resistance to alternating loads is very important. Li, et al. [119] in his paper presents some approximations performed in the study of multiaxial fatigue life, as well as a computer-based procedure for determining the life span. This work was tested on a casing compressor from which it was possible to derive interesting results to proceed with this work. Papers in a strand of presenting various models for determining multiaxial fatigue strength can also be found, where comparisons corroborated by experimental tests are also presented [120], [121]. Other works in which these models are applied in a more dedicated way to steel types or to connections can also be found. As for example the work of Cruces, et al. [122] that is dedicated to the study of two stainless steel alloys subjected to multiaxial loading where it uses the critical plane method for the determination of their fatigue resistance. In the works of Sonsino [123] and Lee, et al. [124] studies are conducted on welded joints subjected to multiaxial loading. In these works, where linear cumulative damage models are used, it is possible to observe the differences between the results obtained for in-phase and out-of-phase multiaxial loading as well as when subjected to bending or torsion. Pellinghelli, et al. [125] also conducted a fatigue study of a welded joint subjected to multiaxial loading but using three different criteria, with numerical models and experimental tests to compare the results in order to understand which criterion best fit the case.

There is one kind of model that it wasn't presented for the estimate fatigue cracks, that is the coupled damage-viscoplasticity. It's known that most of the injection mold parts doesn't present a significant geometrical change when they crack by fatigue so this type of analysis is not relevant but, when analyzing ejector shafts, if they have any tune or design problems they could crack by fatigue and here the change of the section area at the most stressed region is an important factor. Khoei and Eghbalian [126] present a study where they show the evolution of ductile damage and the failure in cyclic loading of ductile materials using the rate-dependent elasto-viscoplastic damage model. Vansovich et al. [127] studied surface cracks grow model caused by fatigue taking under account the plastic deformations for a cylinder where they use specimens to study the effects of biaxial loadings in the crack grow rate and compared the results with uniaxial loadings, they also use a numerical method to get more information about the stresses at the crack zone. These studies might be useful to understand why some elements of the injection mold might crack during production, but this shouldn't be a problem if everything is designed and produced with the proper tolerances.

In the papers presented before, the authors showed the theories and worked to validate them or to find their limitations but it is also important how some parameters that can be controlled during the design affect the tool lifetime. Newman Jr. & Newnan III [128] presented a study of the two-parameter fracture analysis recurring to numerical analysis. Where a study using three different materials were

made and they reach to an interesting conclusion in this paper that was: “the relation between the elastic stress-intensity factor at failure, K_{Ic} , and the net-section stress, S_n , at failure was “linear” for net-section stresses less than the yield stress”. These relations are helpful when it’s needed to work with low safety coefficients and with this, designers can use the relations to reduce the design time which in this area is still made by trial-and-error recurring to FEM analysis.

Studies about experimental evidences or numerical models of fatigue crack initiation and orientation like the ones made by Lopez-Crespo et al. and Polák & Man [129], [130] help to introduce easier methodologies and to prove concepts of how to predict the crack initiation. As for Lopez-Crespo et al. they also made a more complex study using biaxial loads which is common in injection molds, especially when the molds are created for curved parts (for instance bumpers, some kinds of spoilers, vacuum cleaners’ plastic parts and others) as for Polák & Man their study showed evidence about the importance of the localized cyclic plastic straining in the formation of surface relief and initiation of fatigue cracks.

In some injection processes the temperature amplitude is very high and that can cause cracks by thermal fatigue. Petráš, et al. [131] made a paper where they study the effects of thermomechanical fatigue on one specific steel, in this study they subjected the specimen to in-phase and out-of-phase thermomechanical loadings where in in-phase as in out-of-phase loadings the specimen only started to present cracks after the formation of the oxidized layer appeared. Most of mold materials are treated to present oxidation resistance but where the treatment doesn’t reach or where there is no treatment at all this could be a problem. For instance, the injection molds have refrigeration channels in the steel and these regions are very favorable to oxidation and with cyclic thermal load this is a problem to take under account. Nowadays to avoid this problem mold makers use a limit distance between cooling channels and the engraved region but in many situations, they use empirical values reached by previous experiences, that could lead to an over safety coefficient, and they only know after the mold is in production.

Nowadays with the use of new technologies to produce metal parts it is needed to understand how these materials behave. For instance the metal additive technologies are starting to be widely used to produce more complex geometries so, in the paper of Akita, et al. [132] it is possible to see a comparison of the fatigue of a specimen made by selective laser melting and conventional-melted stainless steel. With this work, it is possible to have an idea of the life-time differences between these two processes but, this is a recent technology in the mold industry and needs further studies. But with the evolution of this technology and the introduction of new powders it is hard to have these materials as well studied as the common produced steels. Since the parts are becoming more and more complex these additive

technologies are more used. So, they should have more studies to check how the current models behave and if there are more factors to have under account.

But, all of the physical models and numerical models used by researchers or designers need the materials information and it is important to create good methodologies to get this parameters in order to have accurate results, as Stoudt, et al. [133] made when they presented a way to get an innovative way to measure those parameters for a thin sheet metal. These researches are important to create methodologies for measuring the needed material properties faster, less expensively and with accurate results. The fatigue characterization of materials is a central issue because without this characterization it is not possible to make failure predictions at the design stage. Basan, et al. [134] present considerations to be taken into account when estimating fatigue parameters. In this work several methods are evaluated to obtain the parameters from the monotonic curves of the materials. But there are several papers that are dedicated to the presentation of models capable of generating S-N or ε -N curves that characterize the fatigue behavior of materials, but they either have some limitations or need to be validated on a larger spectrum of metallic alloys. For example the work of Kohout and Věchet [135] who present a new formulation for the creation of the S-N curve from the monotonic steel properties. In these works methods based on obtaining the parameters from the monotonic curves of the steels were presented, curves obtained either from axial or torsion tests, or even from web-based data [136]–[139]. Jeon and Song [140] presented in their work a system that can help designers on the creation of the ε -N curves for metallic materials accordingly to the available input data.

Until now it was presented some research papers about the fatigue theme in general, now it's going to be presented a couple of works made in specific for injection molds. Li et al. [141] made a research about the fatigue lifetime of an injection mold but in an very specific injection process that is a rapid heating cycle injection where as stated before the thermal loads are expected to have an impact at the steel life time. The author made his research using the panel of large-size liquid crystal display TV as his specimen. They made numerical simulations to get the results and to make improvements suggestions to increase the lifetime of the injection mold. With this work is possible to check how the temperature cycles can reduce the mold lifetime since with the temperature variation. The mold increases or decreases in volume depending on the temperature which can lead to the increase of the internal efforts. As for Kim and Lee [142], they studied the fatigue of an injection mold with numerical simulations. They made a stress analysis and fatigue analysis, in order to estimate the fatigue life of the mold and they also presented proposals to improve the core design to prevent its fatigue failure.

In this chapter, it was presented researches to understand the methods to determine the fatigue lifetime of a specimen, works made with numerical methods since the injection mold have a very complex geometry to calculate the values analytically. So, the FEM analysis are indispensable to reach into results. Also, presented works which shows us the main factors that can influence the fatigue lifetime of a specimen.

2.1.7 Fracture

To understand how to reinforce the tools in order to support the cyclic loads that they will be working, it's needed to understand how materials reach to fracture to get a better idea of how to make a proper design of the tool. So here a brief description of works made in this area that can be helpful to reach into some conclusions.

Zhang et al., made a research about the constrained intrinsic cohesive finite element method with little stiffness reduction for fracture simulation [143], with this work was presented some research about the use and considerations to make when using finite elements in the study of a fracture. At this work, it was possible to see that the flow stress increases with the increasing strain rate and decreasing deforming temperature. And with increasing deformation temperature and decreasing strain rate, the grain size as well as the volume fraction of the recrystallizes grains increase. As for Eshraghi & Soltani [144] made a description of the intensity factors obtained for functionally graded cylinders with internal circumferential cracks using the weight function method for several combinations of cylinder geometry's, crack depth and material gradations. With this work they were able show that the weight function method along with the proposed fitting technique are in good agreement with the results directly obtained from the finite element analysis.

2.1.8 Surface roughness

The rugosity of the mold surface has an impact on its mechanical and thermal behavior. This parameter usually is dependent of the desired part surface finish in the engraved surface, as for the rest of the mold surface usually depends on where the surface is working.

The surface roughness has an impact on the heat transfer from the injected part to the mold surface. Coates and Argyropoulos [145] present in their paper the heat transfer coefficient at the metal mold interface, here they made experiments where they measure temperature profiles to reach the conclusions. And they stated that with an increase in mold surface roughness results shows a decrease in the heat-transfer coefficient at the metal mold interface, this work was made with aluminum instead of plastic. But, with plastic the same can be stated, since with the increase of the surface rugosity the plastic slows down faster, and it cools down faster near the walls, and since the plastic is a good insulant, it slows the heat transfer rate from the melt plastic on the inside of the plastic part.

For certain plastic parts, there is need to ensure specific surface rugosity to mold a part with good surface quality, for instance in some car bumpers the mold cavity is polished to ensure a very low rugosity so they can have a mirrored look. But for some parts the very low rugosity can perform as a demolding agent that helps to eject the plastic part without having regions imprisoned in groves or other regions that presents problems for demolding as it is possible to see in the work made by Masato, et al. [146]. To help this surface finishing there are some surface coatings that improves these factors as the paper made by Chen, et al. [147] presents. Surface coating is a good solution to improve demolding capabilities without the need of making some changes of the geometry.

But the surface finish also affects the surface resistance to crack nucleation as will be discussed again in this paper. But in the work of Javadi, et al. [148] results were presented that demonstrate the influence of surface roughness on fatigue behavior.

2.2 Injection mold design and components

Along with the rheological study, the injection mold design is the first part of the project to be carried out. During this chapter the main components of the mold will be presented in order to understand the terms used throughout this work and also, a brief description of the operation of some components will be made in order to understand their function.

Injection molds are divided into two parts, one called the fixed part and the other the moving part. Generally, the moving part corresponds to the core and the fixed part to the cavity. Figure 15 shows a 3D drawing of an injection mold and a representation of the machine columns, these are represented to get an idea of the dimensions of the injection mold inside the injection machine so that its sizing can be done without exceeding the limits of the machine or, if this is not possible, to propose a change of the injection machine that can accommodate the injection mold.

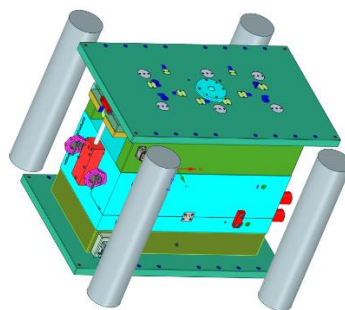


Figure 15 – 3D CAD drawing of an injection mold.

Figure 16 shows the cavity of an injection mold (Figure 16 a)) and a detail of the manifold plate and clamping plate (Figure 16 b)). The manifold plate can be on the fixed or movable side of the injection mold, this depends on the side on which the injection of the part is outlined and is also only used when the injection mold has an injection system (hot runners).

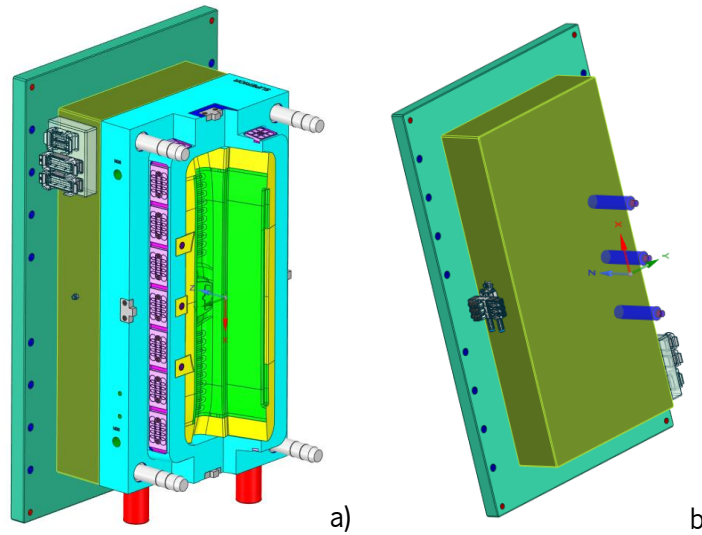


Figure 16 – Representation of the a) cavity side of the injection mold and b) cavity side without the cavity block.

Generally, the movable side, or the core side shown in Figure 17, is the most complex both in terms of sizing and mechanics. Usually, the regions of the injected part that require mechanisms in order to demold it, are on the core side. For this reason, it is necessary to design components and their kinematics so that when the mold is opened, it is possible to extract the injected part without damaging it.

In Figure 16 a) and Figure 17, it can be observed that there are plates along the partition plane. Where the partition plane, as the name implies, is the region where the side of the core and the cavity touch when the mold is closed. The pink colored plates are called adjustment plates, while the dark gray colored plates are the pressure plates. The adjustment plates that are applied to the mold cones and are intended to fine-tune and ensure proper mold closure, while the pressure plates are responsible for ensuring that the clamping force applied to the mold is properly distributed along the partition plane.

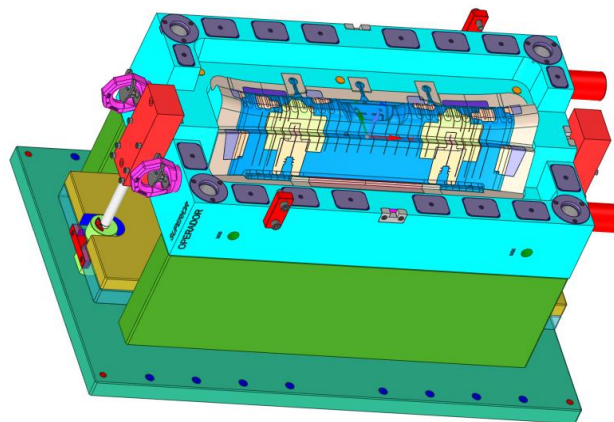
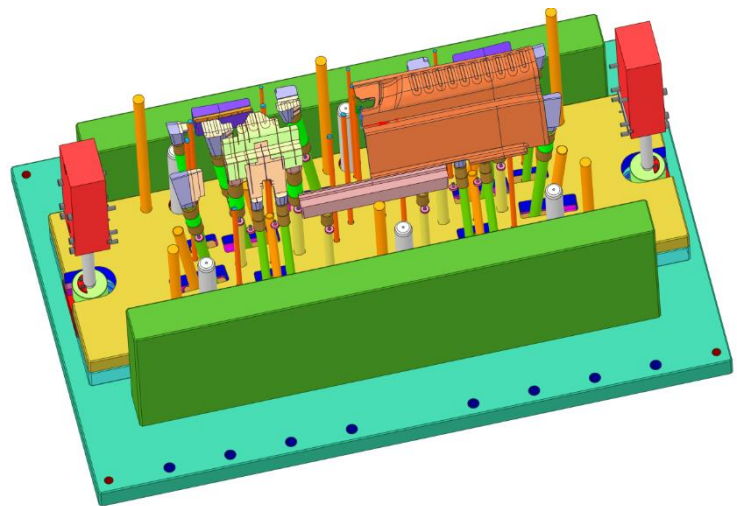


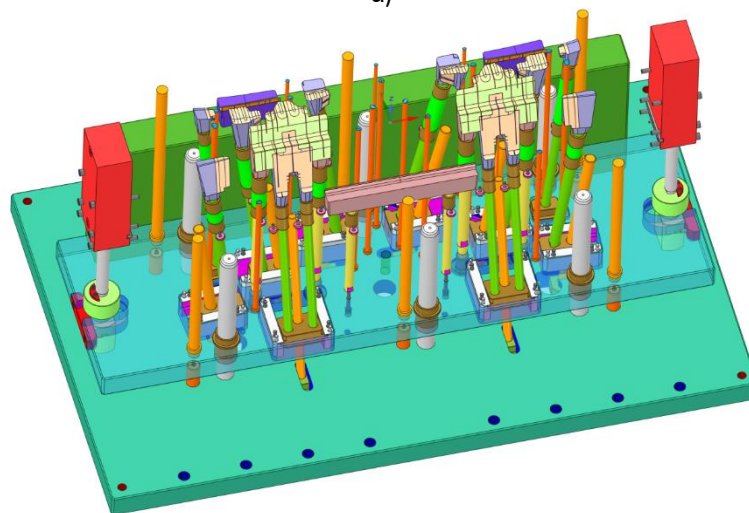
Figure 17 – Representation of the core side of the injection mold.

Figure 18 a) shows the movable side of the mold, i.e., the core side, without the core block. In this picture it is possible to observe the components of the core where the inserts are, sliders, lifters, extraction plates, the core clamping plate, and the rails. Also represented is half of the plastic part that the mold is going to inject, to show how all the moving components are positioned along the part. To move the

components responsible for extracting and demolding the injected part, there are two plates, called extraction plates. These can be driven by hydraulic cylinders or by a component of the injection molding machine, in the case shown in Figure 18 the drive is hydraulic. One of the plates is where the shafts are positioned as shown in Figure 18 b), where for the lifters the shafts are only fixed to the plate, for the extractors, they are positioned because they have a groove in their base that does not allow them to spin, but they are not fixed to the plates, but trapped between the two extraction plates. In the case of the sliders, there is a component that is responsible for ensuring that they are always guided as the extraction advances, because they have a linear stroke that is not perpendicular to the extraction plates.



a)



b)

Figure 18 – Core side of the injection mold a) without the core block and b) without the core block, rail and one of the extraction plates.

Figure 19 shows the molding and part extraction components in a mold, although no mechanical slider is shown, these are part demolding components where their actuation is performed from the mold opening using guides that are applied on the fixed side of the mold and with the moving part of the mold the mechanical movement slides away from the part to allow its extraction. Figure 19 a) shows the sliders and lifters and where the guides of the sliders can be better observed. Figure 19 b) shows two inserts

with half of the part on one side to better understand their positioning. The inserts are used to simplify the mold construction, for example when there are small areas that need other manufacturing/finishing processes or their access would be difficult if it had to be machined in the core/cavity or even in the case of having to use another metal alloy in that region, but these components are fixed to the core/cavity.

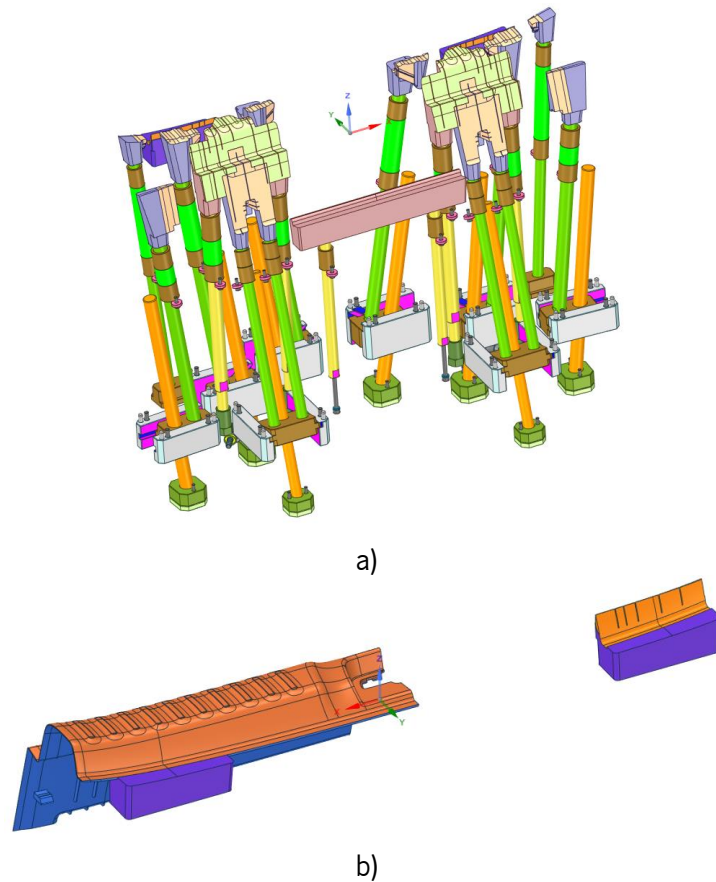


Figure 19 – Molding components in the core where a) moving components during the extraction phase and b) are the inserts with the representation of half of the plastic part.

Sliders are used to demold areas of the part that cannot naturally come out in the demolding direction, i.e., the direction of the mold opening. Figure 20 b) shows a detail of a slider where the arrow represents the travel direction and it is noticeable that the path is oblique to the demolding axis. In Figure 20 c) where the extraction movement and the part in cut can be analyzed in detail, it is possible to see that there are some areas of the part that do not allow it to be extracted without removing the steel from its path, so during extraction the sliders are moving away from the part in order to clear the path so it can be freely extracted.

The lifters, like the ones shown in Figure 21, are used to remove the part from the mold so that it can be removed from the mold. These can be accompanied by extractors, but this depends on the geometry of the part, because if it has tubular areas, these must be demolded by extractors. But for these components the movement is in the direction of demolding, that is, they move perpendicular to the extraction plates and accompany the part during the entire extraction stroke.

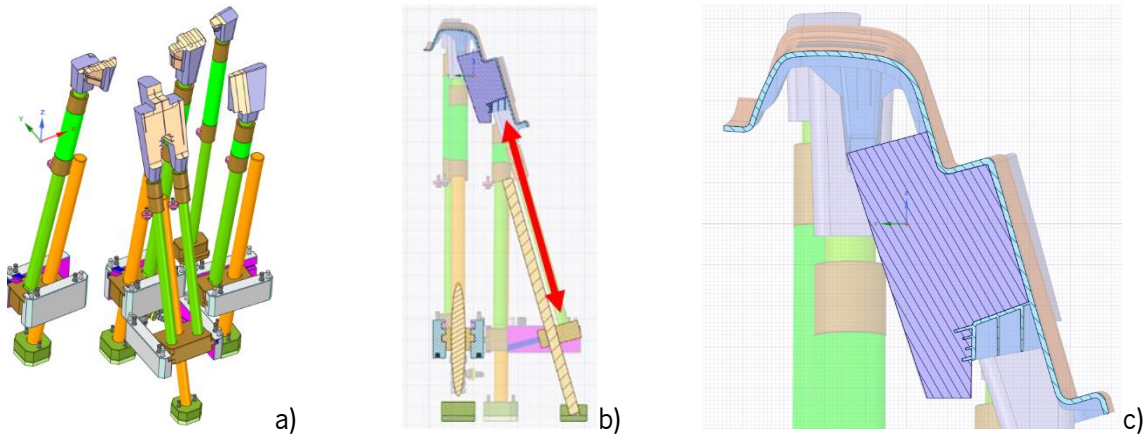


Figure 20 – Detail of sliders in an injection mold. Where a) are the sliders, b) one section of a slider and c) the detail of the slider and part.

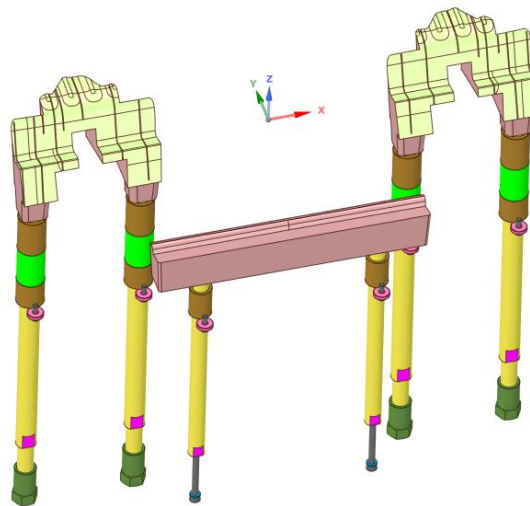


Figure 21 – Detail of lifters in an injection mold.

2.3 Rheology study during the design process

After obtaining the 3D drawing of the part, the mold maker can start with rheological studies to understand the difficulties and peculiarities of the part. At this stage besides the part design, it is very important to know what material or materials are planned for its injection. The characterization of the material is of great importance to obtain a good rheological study, because it will have an influence on the type and the injection system, and it may lead to the need to make modifications to the part in order to ensure that it meets the structural, dimensional or aesthetic requirements.

In a first analysis the filling pattern is checked. Here the aim is to achieve as balanced a filling as possible, taking into account any limitations to the injection points that may exist. By combining the filling pattern (Figure 22) and the isochronous lines of the filling pattern (Figure 23), it is intended that the front of the molten polymer flow to have as uniform a velocity as possible. Obviously plastic parts can have obstacles to the progress of the molten polymer such as holes, variations in thickness and significant variations in shape that will impact the homogenization of the velocity of the flow front. Still analyzing these two results, it is necessary to look for hesitation or acceleration lines caused by the sequential

opening of the hot nozzles. If the opening of these hot nozzles occurs too early or if the positioning of the injection points into the part are not properly designed, these types of defects can occur. With the analysis of this result this type of problem can be avoided.

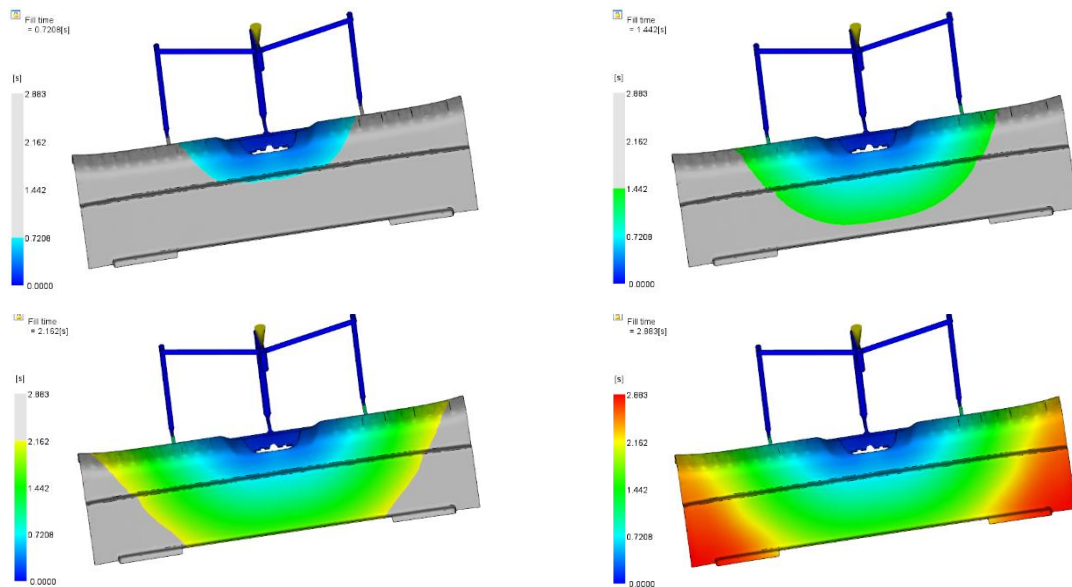


Figure 22 – Filling pattern of a plastic part.

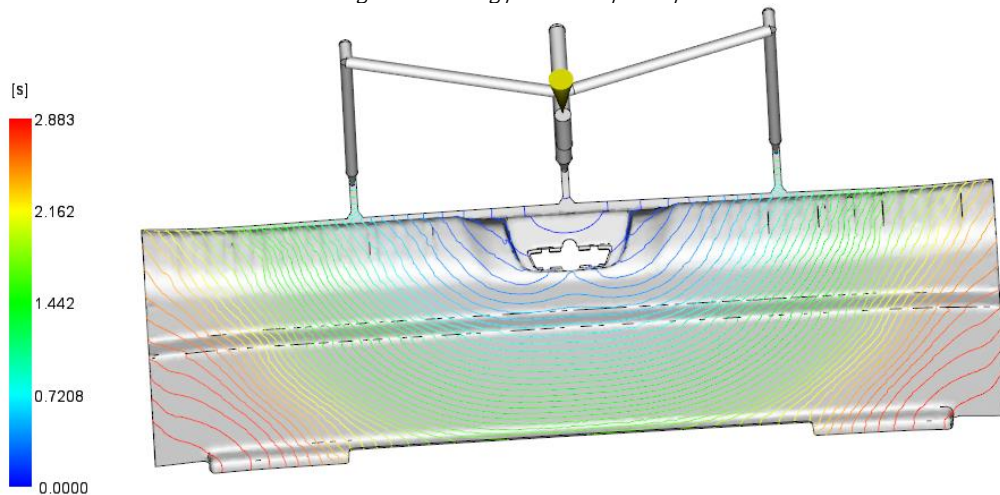


Figure 23 – Isochronous lines of the filling pattern.

Figure 24 shows the part volume filled by polymer at the instant of the switchover between speed injection and pressure injection, which is usually called V/P switchover. This term appears because during the first phase of filling the part it is intended to fill the part as quickly as possible, to avoid the appearance of a significant solidified layer that makes it difficult to fill the part and, in the end, to pack it. When most or all the molding zone is filled, it passes into the injection in pressure, i.e., here molten polymer is injected at a slower speed into the part, but it is guaranteed that it is injected at a predefined pressure. At this stage, the molding zone is fully filled, and the part is packed. Still in the analysis of the part filling phase, the temperature of the flow front of the molten polymer is studied as shown in Figure

25. With this it is intended to know if between the entry of the material into the injection mold and the most distant points there is a large temperature variation.

Here either heating of the material within the molding region or cooling of the material, can result in defects. The heating of the material is due to the heat generated by the shear effects caused by the displacement of the polymer, which if high can lead to material degradation. Cooling of the material is caused by heat transfer between the molten polymer and the injection mold, if the cooling of the flow front is significant it can lead to either aesthetic defects as well as problems in situations where there are welding lines, or even problems with packing of the part.

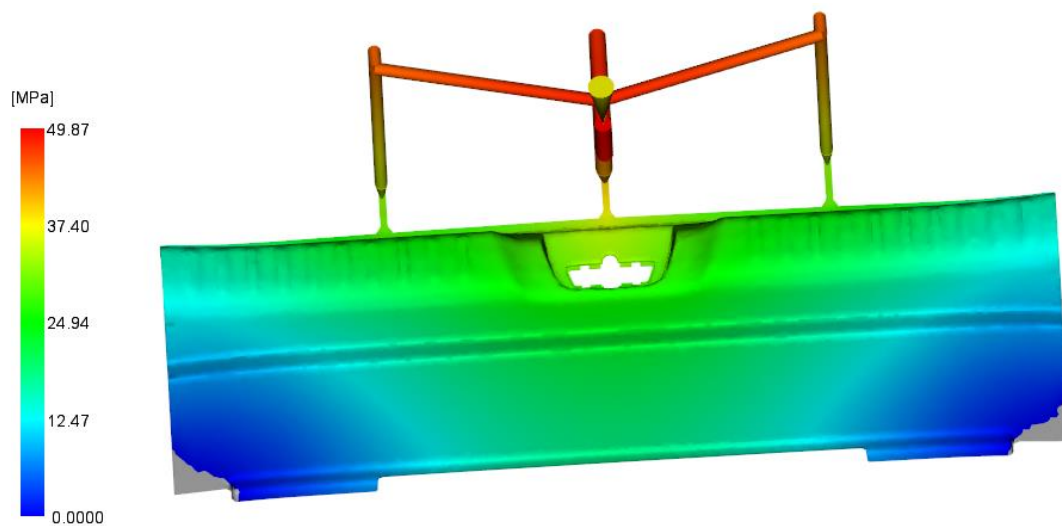


Figure 24 – Pressure at the instant of switching between injection in velocity and injection in pressure.

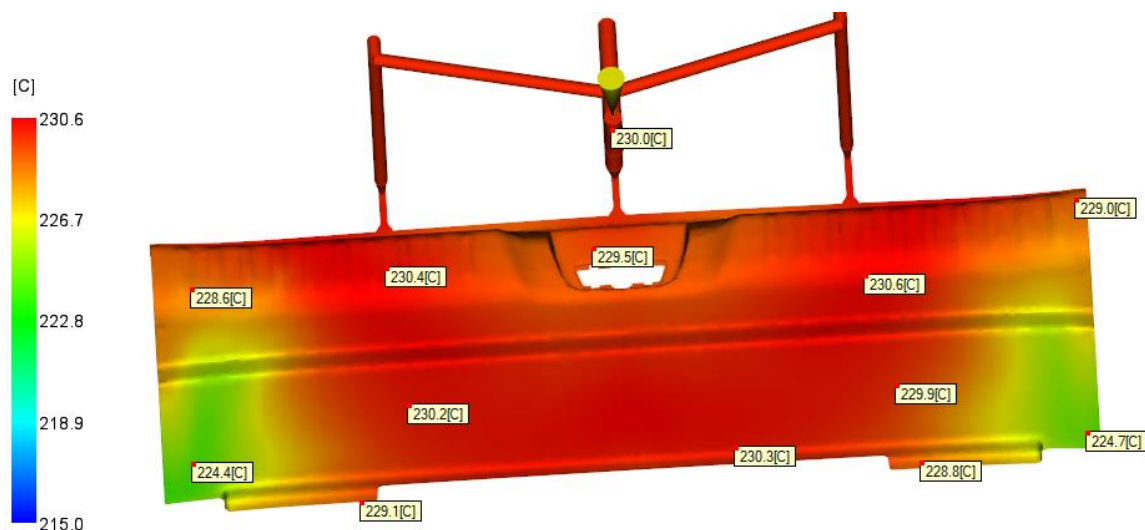


Figure 25 – Temperature at front flow.

The shear rate of the material is analyzed for the cold channels, that is, at the locations where material begins to enter the part as shown in Figure 26. This analysis is due to the changes in section that the molten material goes through that usually culminates in a region of smaller section at the attack to the part. The reduction in cross-section causes the shear rate of the material to increase substantially at that

location, if the value exceeds the limits of the material the size or shape of the attack must be modified to prevent material degradation.

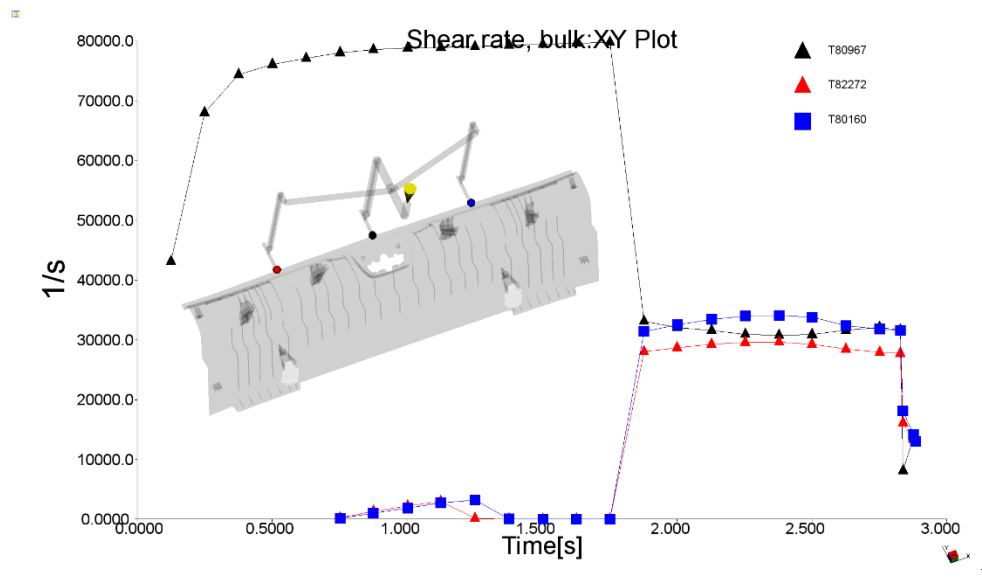


Figure 26 – Shear rate at the cold gates.

Another result that is very important for predicting problems related to part quality is volumetric shrinkage as presented in Figure 27. Parts can have variations in thickness that lead to different regions having different volumetric shrinkage values. In addition to the thickness variation, with this result it can be verified if the packing of the part is being efficient and if not, with this result is possible to outline strategies to obtain the most uniform volumetric shrinkage possible.

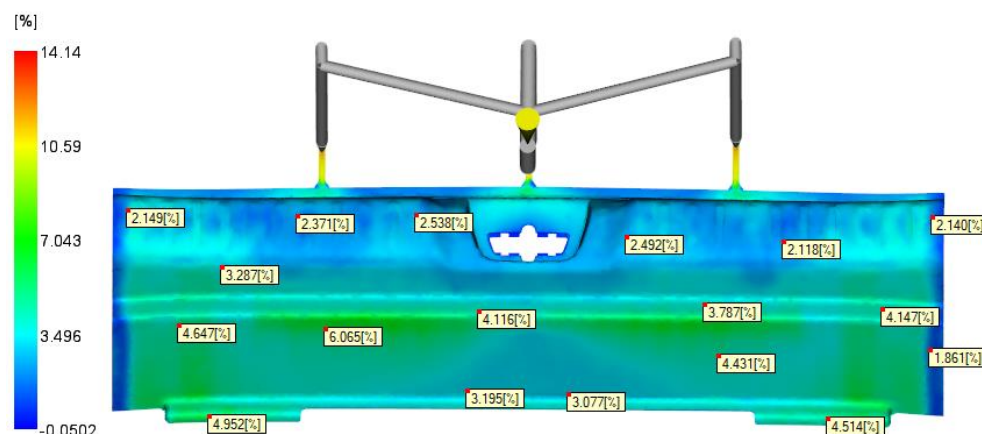


Figure 27 – Volumetric shrinkage at ejection.

Weld lines are one of the points of analysis in rheological studies for two main reasons, one is the structural inefficiency of the part, and the other is the aesthetic component. The way the flow fronts come together, and the temperature of the flow fronts have a very significant influence on the strength of the part at the weld line area. Figure 28 shows the result for weld line analysis in the Moldflow software, where according to the notation used, the greater the angle of formation of the weld line the better the material will merge. But this result must be combined with the temperature of the flow front, because if

the flow fronts come together at very different temperatures, it will not be possible to obtain a good fusion of the material. In addition to the angle of formation of the weld line and the temperature of the flow fronts, the position of the line is also important. This can be done by repositioning the injection points or changing the injection sequence in order to control the flow front, but the solutions must be analyzed part by part.

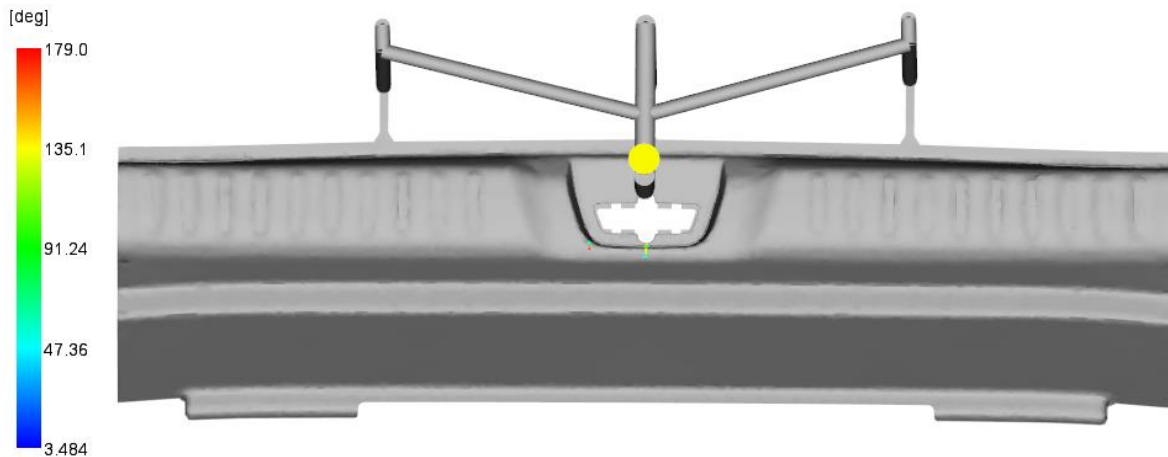


Figure 28 - Weld lines.

The clamping force that the machine needs to apply to keep the mold closed can also be obtained from these studies, in Figure 30 the force needed to keep the injection mold closed during the entire cycle is graphically shown. Usually, this result is combined with the injection pressure at the inlet to get an insight into the critical instants. Figure 29 shows the graph with the injection pressure at the inlet, which in addition to being used to compare with the clamping force of the mold, should also be analyzed in order to understand if there are significant pressure drops that could lead to the appearance of aesthetic defects.

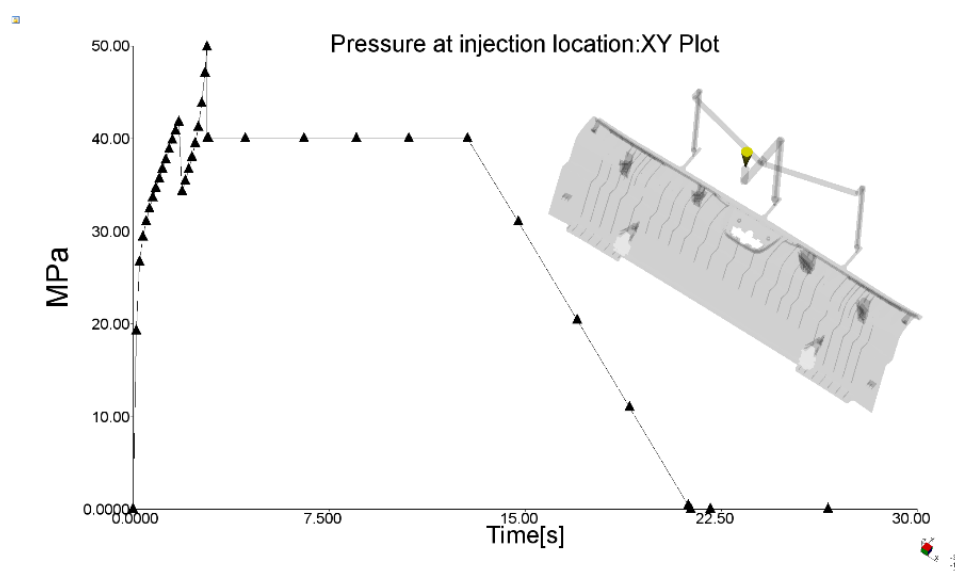


Figure 29 – Pressure at the injection location.

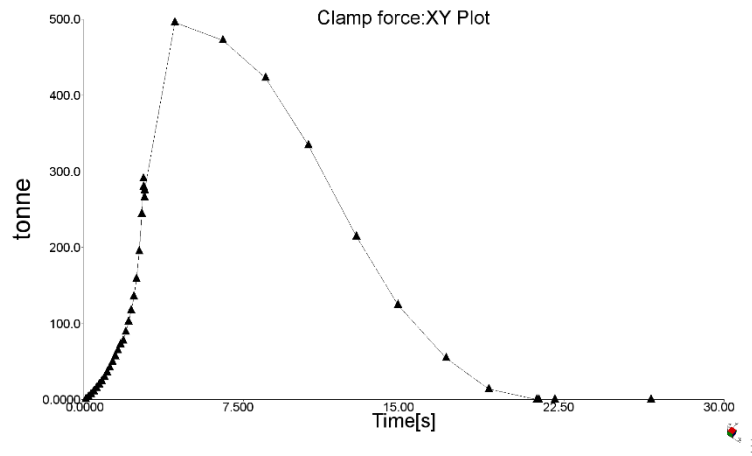


Figure 30 – Minimum injection machine clamping force.

2.4 Injection mold thermal concepts

When the injection mold mechanisms are properly designed, the mold cooling system is ready to be designed. This cooling system usually consists of drilled holes in the steel through which cold water flows, in critical areas of the mold, alloy components with a higher thermal conductivity can be used to help dissipate heat from the part. Here the goal is to obtain as uniform temperatures as possible throughout the engraving region.

Here the thermal study is important because it helps to detect areas where heat removal is less efficient and so can be solved by testing various configurations until the most suitable one is found.

Now will be presented the results that are analyzed in the thermal study in the Moldflow software and what considerations are made. Figure 31 shows an example of a numerical model prepared for a thermal study in the Moldflow software. The modeling of the cooling circuits can be performed in two ways, either the 3D design of the channels is considered, and they are meshed with tetrahedral elements or the centerlines of the circuits are provided and for each line a property is given and after meshing process, will be converted into beam elements (1D elements) as represented in Figure 31.

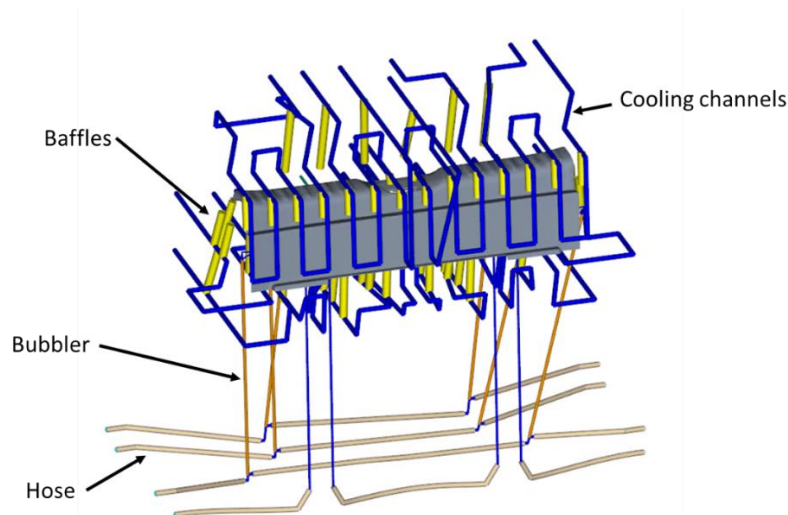


Figure 31 – Elements used in thermal simulations in Moldflow.

Channels are the most common elements that have the function of cooling the part, transporting the coolant along the injection mold or, in certain injection mold concepts, are also used to homogenize the temperature of the injection mold mass. The hoses are defined to consider the external connections between circuits, where for calculation purposes only the pressure loss caused by them is considered, i.e., it is assumed that they have perfect insulation.

The baffles and bubblers are used to cool regions where access with channels is difficult. The bubbler can also be used as a water transport element when there is only one shaft in lifters or extraction sliders, which have to transport the coolant in both ways. The bubblers are generally used when the size of the region to be cooled does not allow the use of baffles. The way how of the baffles and bubblers work can be observed in Figure 25, where it is possible to see the differences in the functioning between both cooling elements.

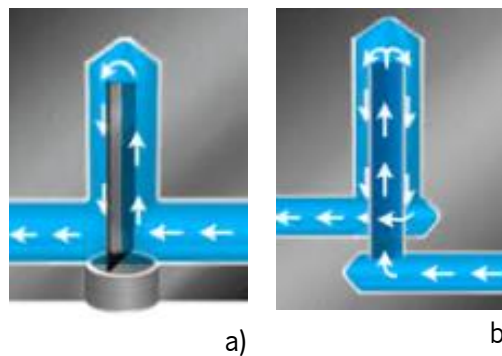


Figure 32 – Representation of the functioning a) a baffle and b) a bubbler [42].

In the first instance, the cooling circuits are analyzed. Here attention must be paid to a number of results in order to obtain an overall perception of the cooling system. Figure 33 shows the results for the cooling circuits that are analyzed. There are 4 points where their analysis is indispensable, which are the flow per circuit (Figure 33 a)), the pressure per circuit (Figure 33 b)), the Reynolds value (Figure 33 c)), and the temperature variation (Figure 33 d)). The flow rate must be analyzed because there is a limited amount of flow that is able to reach the mold, which will be divided by each circuit so that the pressures are balanced between the inlet and outlet. As with the flow rate, the maximum pressure is also a limitation that exists, but in the Moldflow software is only possible input one condition, so if is used the flow rate as input the maximum pressure obtained in each circuit must be analyzed so that it does not exceed the network limit. If the pressure exceeds the network limit, the circuit must be resized to reduce pressure drop. The same analysis needs to be made if the pressure is used as input.

To increase the efficiency of the refrigerant circuit, the flow must be in a turbulent regime. Generally, the coolant used is water, and in this case, it should attempt to have a Reynolds value greater than 10000. As stated, a turbulent flow ensures an higher heat removal rate, as is known there is a direct relationship between the Nusselt number and the amount of heat removed by the cooling channels and

as is possible to observe on the Figure 34, the best range starts when the Reynolds is higher than 10000 since this type of cooling can be compared to a tubular heat exchangers. Finally, the temperature variation of the circuits, as it is obvious that as the temperature of the coolant increases, its ability to remove heat decreases. Generally, by convention, the temperature variation between the inlet of the circuit and the outlet should not exceed 3 to 5°C, if this happens the circuit should be divided, or the diameter of the hole increased to allow an increased coolant flow if possible.

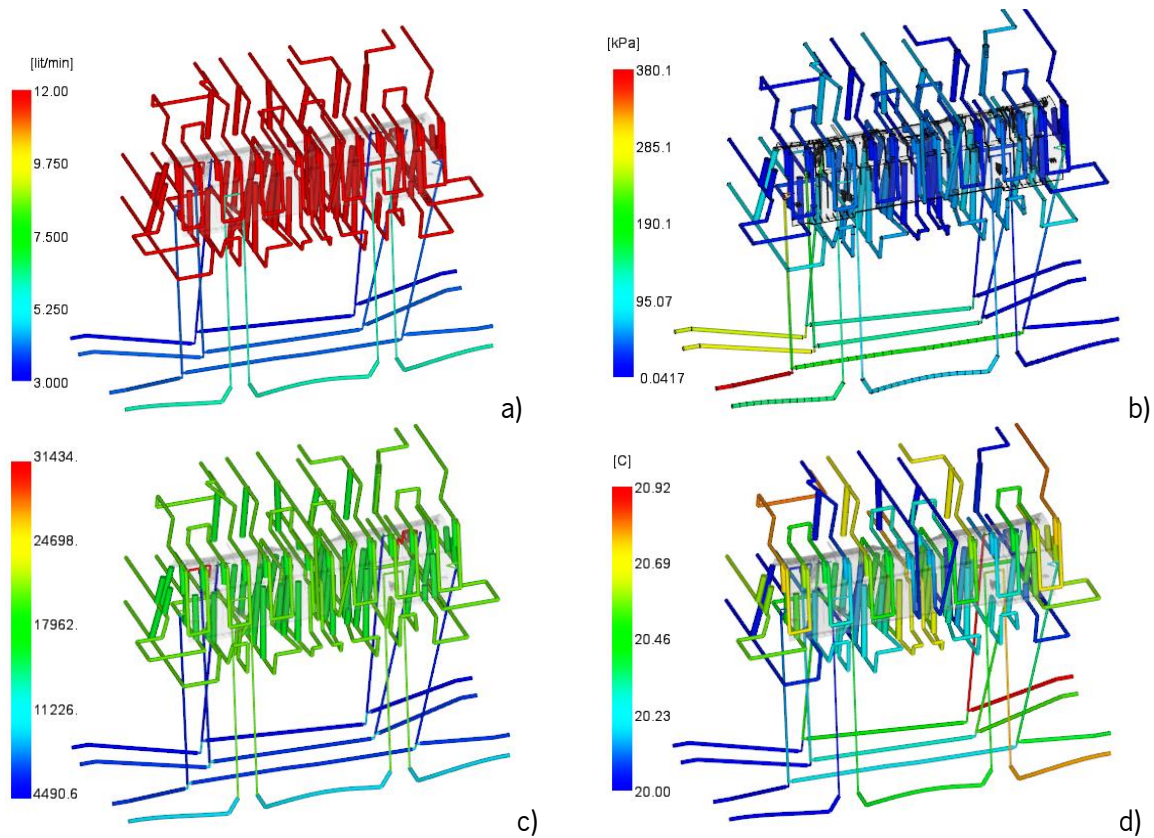


Figure 33 – Cooling channels analysis results, where a) is the coolant flow rate, b) is the circuits pressure, c) the Reynolds number at each location and c) is the coolant temperature.

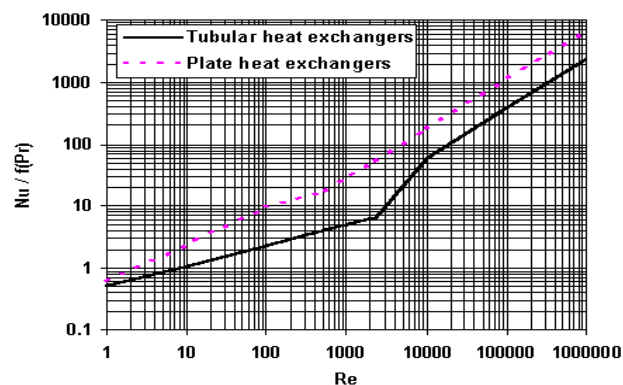


Figure 34 – Relationship between the Nusselt number in function of the Prandtl and Reynolds numbers [149].

After the analysis of the cooling circuits, is possible to move on to the analysis of the part and the molding region of the injection mold. Figure 35 shows in matrix form the temperature results of the part and the molding region on the cavity and core sides.

In the analysis of the part and the molding region, is checked if the temperature variations are not very significant. The visible areas of the part are the first to be analyzed, because here significant temperature variations can lead to the appearance of aesthetic defects. In many parts, due to their shape it is not possible to obtain a very homogeneous temperature, as can be seen both in the part and in the molding surface on the core side in Figure 35. With a more recent technique, that is the construction of the components using metal printing, allows to create circuits with almost no limitation in their shape, because here they are built as the component is printed and not by drilling. Obtaining homogeneous temperatures is important because the volumetric shrinkage of the part will be influenced by temperature variations, that is, with a more homogeneous temperature, the factor that will have more influence on the part's shrinkage will be its geometry. Since the cooling of the injected part usually represents the largest part of the cycle time, working to have a homogeneous part temperature can improve and reduce cycle time.

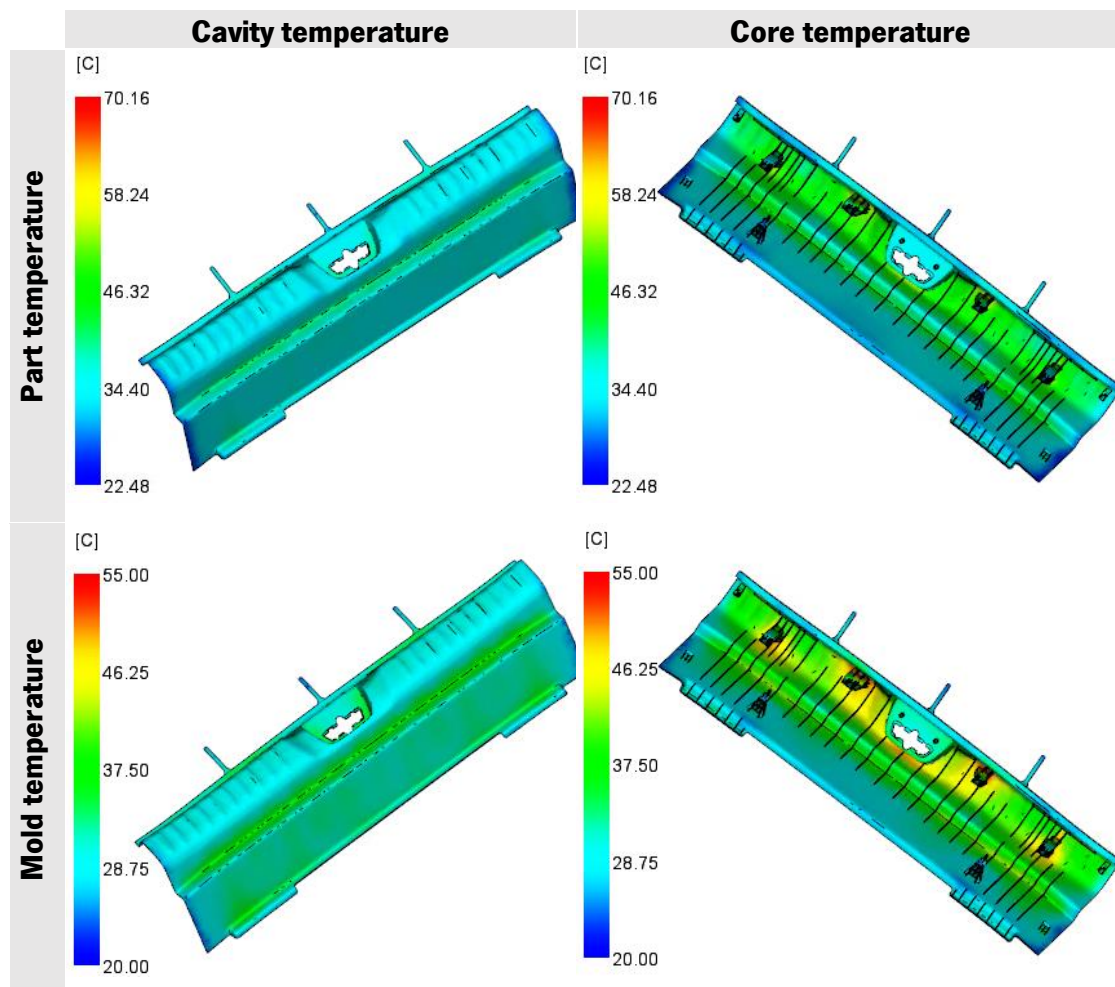


Figure 35 – Temperature distribution in the part and molding region of the cavity side and core sides.

Finally, a comparison is made at various points on the part between the temperature variation between the tap and cavity side as shown in Figure 36. Because if the temperature variations on the part

are significant between the tap side and the cavity side, there may be very different contractions locally, which can lead to local deformation of the part or the appearance of aesthetic defects.

As with the analysis of part filling, the thermal study of the mold involves a whole set of factors that cannot be analyzed separately. In order to obtain efficient cooling, it is necessary to determine which combination of factors can lead to the best possible outcome.

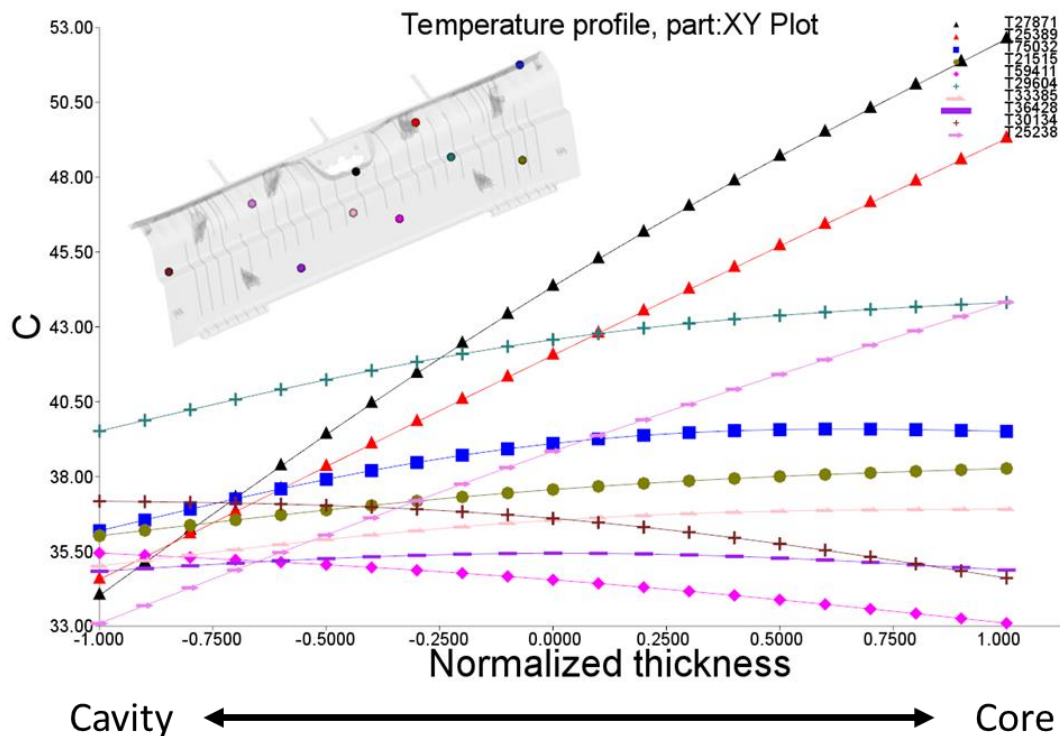


Figure 36 – Temperature profile between the core and cavity side of the part.

3 Fatigue design theoretical concepts

The definition of fatigue which is stated by ASTM [150] is the following: “The process of progressive localized permanent structural change occurring in a material subjected to conditions that produce fluctuating stresses and strains at some point or points and that may culminate in cracks or complete fracture after a sufficient number of fluctuations”. Since the focus of this work is to determine the fatigue strength of injection molds, and for the reader to follow the theoretical assumptions that will be used throughout this work, this section will present theoretical concepts about metal alloys fatigue. Where the main focus will be on the formulations of curves that characterize the material behavior, the factors that affect the fatigue life and some methodologies for the calculation of the fatigue endurance limit of components. With this it is intended that the designer can have a handbook that he can consult during the design phase to obtain the information he needs in his analysis.

For injection tools the fatigue design is very important since the tools are subjected to cyclic loads through all its work-life, for this case the loads tend to be constant through each cycle as shown by the Figure 37. Where is possible to observe that the applied pressure load is equal for every cycle.

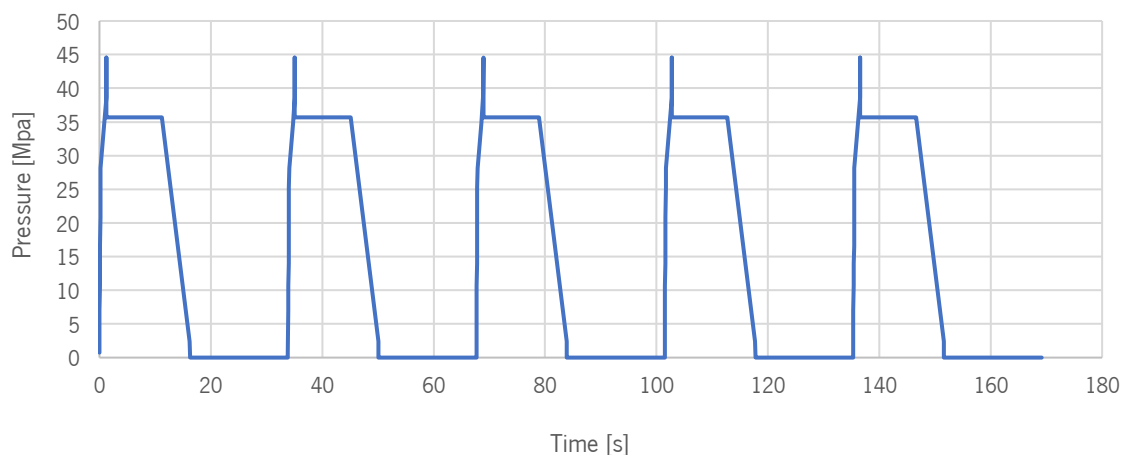


Figure 37 - Pressure at the injection mold entrance during five consecutive cycles.

The fatigue failure many times looks like sudden but usually has been developing from the beginning of the part usage or somewhere along its work-life, and this phenomenon has been noticed and studied since the XIX century where the German engineer Wohler [151] initiated the proper studies which had as objective the comprehension of failures on the axles of locomotives, caused by the metal fatigue. By these studies there are some points that stand until our days which are:

- Development of materials and treatment that could lead into an increase of the mechanical resistance to the fatigue and be economically viable.
- Development of calculation algorithms for structures or components which are subjected to fatigue.

With the evolution of the manufacturing technologies and with the constant pressure from the economical side, there was a need to manufacture smaller components and/or with more complex geometries. Forcing a more detailed study of the fatigue behavior of those components. With it, more detailed studies of all the factors that have an impact on the components fatigue endurance limits.

With this evolution sometimes the number of failures due to fatigue of the components although may increase, does not mean that it is due to lack of studies on the fatigue of the materials, but often related to the reduction of safety margins during the project and with this higher allowable stresses are used and hence, when there are deviations in the use of the components off the margins for which they were designed, they eventually give way to fatigue. But the opposite could also happen, as it will be presented on this chapter. Sometimes the fatigue failures could also be planned at the design stage and the studies to make this happen could be more complex than the ones made for infinite life.

As it will be possible to understand it is necessary to have under account some base points such as:

- There are several models to design parts to fatigue life and they should be used every time those parts are subjected to cyclic loads.
- The use of higher static safety factors isn't a synonymous of having a higher fatigue security coefficient since there are many factors that influence the fatigue limit of a component.
- To have a good design of a part to the fatigue even using CAE software's there is always a careful analysis of all the considerations made for the design and should be tested to ensure that those considerations are valid. These tests aren't always economically feasible, for those cases an extra care should be taken for all considerations made during the analysis since they can't be tested to the fatigue and with it be validated.
- The trials should always be considered as a validation mechanism, not as a design mechanism.
- The synergies caused by the interaction between loads, geometry, residual stresses, duration of loads and many other factors should be always considered during the design since these interactions penalize the fatigue strength endurance limit of the components and if not considered there is a huge risk of underestimation of the stresses.

From all the researches made on the field of the metal failure modes is possible to resume the mechanical failure modes (or non-mechanical factor which have influence on mechanical failures) for notched or unnotched parts in several categories which are presented on Table 1. As is possible to observe there are many ways for a mechanical part to fail so to reduce the probability of appearing such failure proper design should be made, with proper design undesirable losses can be reduced. In Figure 38 is shown a scheme proposed by the Society of Automotive Engineers Fatigue Design and Evaluation that presents a methodology for the fatigue design of components.

Table 1 - Failure modes of metals [120].

I	Excess deformation – elastic, yielding or unset of plasticity
II	Ductile fracture – substantial plasticity and high-energy absorption
III	Brittle fracture – little plasticity and low energy absorptions
IV	Impact on dynamic loading – excess deformation or fracture
V	Creep – excess deformation or fracture
VI	Relaxation – loss of residual stress or external loading
VII	Thermal shock – cracking and/or fracture
VIII	Wear – many possible fracture mechanisms
IX	Buckling – elastic or plastic
X	Corrosion, hydrogen embrittlement, neutron irradiation
XI	Stress corrosion cracking
XII	Esthetic aspects
XIII	Fatigue – repeated loading: <ul style="list-style-type: none"> a) Fatigue crack nucleation. b) Fatigue crack growth. c) Constant or variable amplitude loading. d) Uniaxial/multiaxial loading. e) Corrosion fatigue. f) Fretting fatigue. g) Creep fatigue. • Isothermal. • Thermomechanical. h) Combination of a) to g).

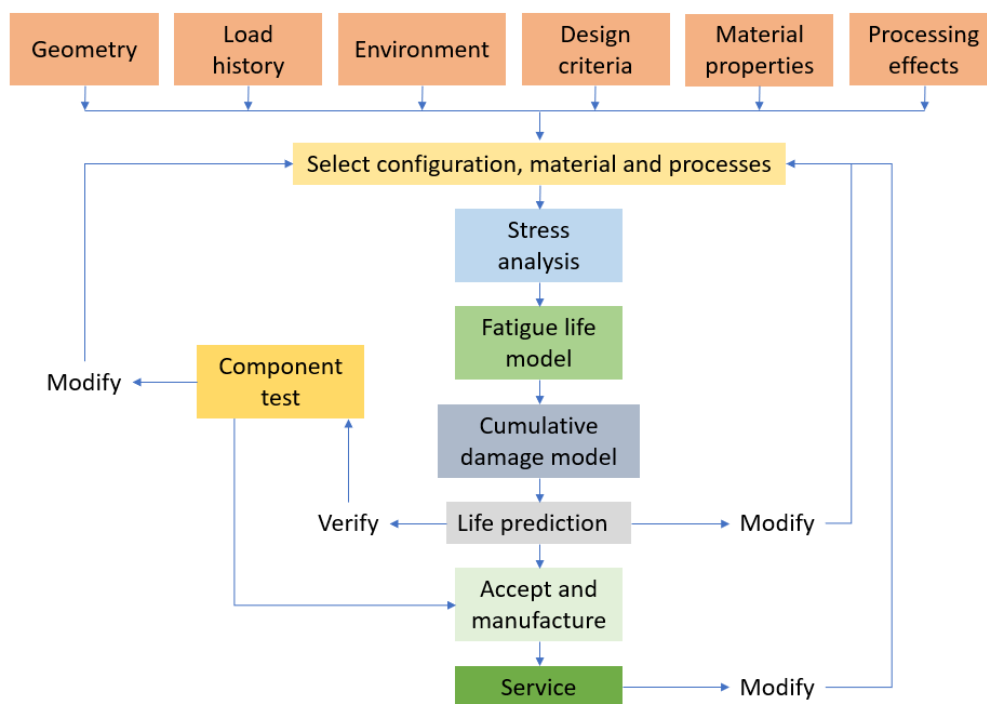


Figure 38 - Fatigue design flow chart created for use by the Society of Automotive Engineers Fatigue Design and Evaluation [151].

The work that design engineer can do over the fatigue life of mechanical components could have different purposes and each one is treated differently so, this studies can be grouped in four different possible situations [151]:

- The in-house tool, where the designer knows the load history of the part to be designed and with it starts by avoiding the stress concentrations as much as possible and will choose the materials, surface finishing and treatments, the stresses, shape and weight so the it can fit the machine perfectly. These studies can have a good safety margin that could lead into more iterations to improve the relation weight or space, expected lifetime and cost and if the loads aren't uniformly repeated the design has to study the cumulative damage.
- The new model, when there is a change in size, material, shape or connections of the part. Here the designer can use data provided by the previous models to improve the model but there is always the need to conduct tests due the changes. Because doesn't mater how small they are, they can lead to different results than the ones provided by the previous subjects.
- The new product, here is where the designer must put most effort during the part design since all the information must be validated. Despite nowadays the use of numerical simulations can reduce the work load by making easier the study of more complex models, is still needed a good inquiry of the working conditions and loads. Is still needed the recourse of fatigue trial tests to verify if all the assumption made under the design process are valid or if there is a need the review the design.
- The design to code, this is one type of design which is usually conservative but for many industries is indispensable since their products must comply with the country codes and with it have an insurance if fatigue failure happens. But some countries don't accept this as sufficient to take out the responsibilities of the designer in case of failure by fatigue.

The responsible mechanisms for the mechanical failures by fatigue of metals can be observed at a wide range of scales, these mechanisms when studied at a larger scale they look simply to understand but when they are analyzed at a smaller scale, they are very complex and can vary with different metals.

The fatigue crack nucleation for ductile materials is usually caused by the cyclic loadings which cause plasticity within a grain scale, on the order of 10^{-3} mm. To have a better understanding of the scale of the mechanisms related to metal fatigue which will be presented on this chapter the Table 2 shows the scale of some metallurgical features.

For the design of parts or structures to fatigue there are four criterions which depend of the application of the studied part/structure. These criterions are:

- Infinite-life design, is one of the oldest criterions, where the design is made so the stresses or strains are at the elastic curve and are inferior to the material fatigue endurance. This design criterion can be too expensive and create parts that will have a higher resistance to cyclic loading than the needed and also could create new problems for its work-life.
- Safe-life design, is where the parts are designed to last for a finite amount of time or cycles. For instance, the automotive industry tests components at their maximum regime to have a mark for the maximum endurance of the component and here they can design so the component lasts the expected life-time instead of making a design where the component lasts for much more time than the needed one. But, of course with this kind of design there still is a need to keep a safety margin to account for the unknown variables. The safe-life design has under its basis the stress/strain - number of cycles curves and if needed the crack growth relations, which can be used in trials or numerical simulations or both. As will be observed later in this work this is also the design that must be used in the injection molds.
- Fail-safe design, when parts or structures are designed with this criterion, they are very conditioned to their designed life-time, *i.e.*, when it is reached the designed average lifetime the part/structure has to be removed from service. This design criteria as can be understood by its name, if one part fails by fatigue the structure can support this damage without total failure but, this comes with a very high cost and as other design criterions is very difficult to have into account all the external factors during the design. This is mostly used on the aircraft industry where some parts are designed so if one member fails the full system can overcome this failure and keep out of the total failure (for instance the fuselage, wings...).
- Damage tolerant design, this criterion is an evolution from the fail-safe design where is known that cracks are most likely to occur at some stage of the component lifetime. These cracks can be caused by the manufacturing process or due to cyclic loading so this criterion uses fracture mechanics to predict the crack behavior and with it can have an estimation if the crack will grow to a stage that total failure is most likely to occur, and with this design comes the periodic inspections that are needed to check if the crack evolution is like predicted.

Table 2 - Metallurgical feature scales [152].

Scale [mm]	Metallurgical feature
10^{-6}	Ions, electron cloud
10^{-5}	Dislocations
10^{-4}	Sub-grain boundary precipitates

Scale [mm]	Metallurgical feature
10^{-3}	Sub-grain slip band
10^{-2}	Grains, inclusions, voids

To implement any of the designs presented above there are three different approaches that can be used which are the analytical approach, the service loading testing and satisfactory experience with previous similar designs where the last one is the most commonly used in the tool making industry.

3.1 Nucleation of fatigue fractures

For parts subjected to cyclic loads the fatigue failure is a possibility. The fatigue cracks nucleate always in surfaces which combines the maximum stresses with minimum strength. So, the fatigue failure is caused by the nucleation and propagation of cracks that appear on parts subjected to dynamic stresses, so the fatigue process can be separated by four consecutive stages as presented in Figure 39.



Figure 39 – Fatigue cracks stages.

When the part ruptures the fracture can occupy an area of 1 up to almost 100% of the fractured surface, this depends on the applied stresses and material metallurgy. Very often the final fracture can be distinguished from the region where the crack propagated because the latter tends to present friction marks, smoothness or corrosion but, there are exceptions. The microscopic fatigue cracks usually nucleate and grow at the plane of maximum shear stress and the macroscopic fatigue cracks propagate through the plane of maximum tensile stress. But, for the macroscopic fatigue cracks there are cases like for torsional and multiaxial loads where was observed that these cracks propagated through the plane of maximum shear. In Table 3 is represented schematically fatigue cracks for round specimens subjected to axial and bending loads where the hashed regions represent the final fracture regions and the lighter regions with arrows the fatigue crack regions.

If the normal stress is elevated the region of fatigue stress crack propagation is small and vice versa and consequently, the region of unstable final rupture will be bigger the greater are the applied stresses.

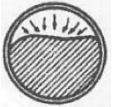
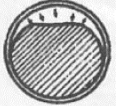
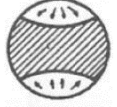
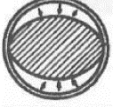
3.2 Principal mechanisms of nucleation and microscopic growth of fatigue cracks

The fatigue cracks initiate at regions of maximum stress and minimum strength and depending on the material and the stage of the fatigue process, crack growth can proceed by a number of mechanisms such as striation formation, microvoid coalescence and cleavage.

Cyclic applications of inelastic strain to a metal can cause continuous changes until stability is reached. This means that the material becomes either more or less resistant to the applied strain, i.e.,

the material either cyclic hardens or cyclic softens. Based on the monotonic stress-strain curve, sometimes is possible to determine if the alloy will harden or soften during the cyclic loading, since the hardening and softening is related to the density and arrangement of the dislocated structure and substructure of the material. The dislocations are a very important factor on the metal fatigue and are also a basic way to understand the mechanical metallurgy and response during the cyclic loading, and with that is possible to have an overview of the mechanisms present during cyclic loading deformations and the development of fatigue damage. A typical used term to describe the inelastic deformation between deformations is the “slip” which happen in metals when there are dislocations between individual grains along crystallographic planes, that is the result of stress or strain cycling. If the dislocations are constrained so that they can only move or slide, the material is said to harden cyclically which ultimately causes the yield stress to increase in that region to a value greater than the monotonic yielding of steel. For steels that already have a high hardness or have been hardened, the inelastic deformation caused by alternating loads causes the region where the steel structure is sliding to rearrange itself in a configuration that offers less resistance to sliding.

Table 3 - Schematic fatigue fractures for axial and bending loads for round specimens [153].

High normal stress			Low normal stress		
Smooth	Mild notch	Sharp notch	Smooth	Mild notch	Sharp notch
					
Tension and tension-compression					
					
Unidirectional bending					
					
Reversed bending					
					
Rotating bending					
					

The material hardening, or softening could occur at a localized scale or at a global scale, for instance the presence of surface discontinuities could lead to localized scale softening due to the cyclic stress or strain present in the region and where the inelastic deformation is very localized, and the bulk of the material will experience elastic/reversible deformation.

Metals have a crystalline structure which makes their atomic structure arranged with an ordered manner, which usually this ordered arrangement makes them susceptible to have preferable planes for slip. This planes for most metals are usually orientated on the plane of maximum shear stress so, the crystallography of the material has an influence on the degree of slip. For ductile materials when applied cyclic stresses or strains the slip lines, due to their lower restriction on dislocation movement, can be present since the beginning of the part life and their dislocation movement is higher and can become persistent and irreversible due to preferential stacking of displaced material. As for brittle materials the slip is limited since the dislocations are limited by the material mechanical properties, and for the material which show an intermediate ductile behavior they have more slip than brittle materials, but it is limited when compared with ductile materials.

The nucleation of fatigue cracks is associated to the slip under cyclic fatigue loading conditions, the Figure 40 a) presents a scheme of fatigue cracks under monotonic load and cyclic loads, as is possible to observe in Figure 40 b) the slip lines are finer when the part is solicited to cyclic loading. The slip lines observed in the Figure 40 b) appear as parallel lines within a grain when they are observed parallel to the free surface with a scanning electron microscope, the free surface of the studied parts is polished to ensure that the crack nucleation isn't caused by surface defects.

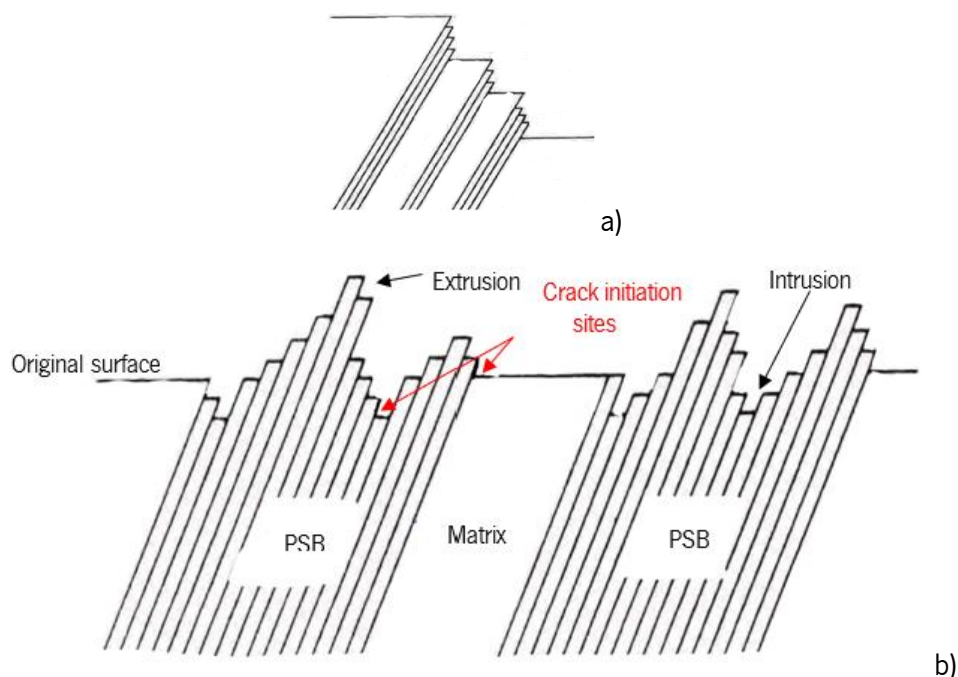


Figure 40 - Schematic of slip due to external loads. a) static load and b) cyclic load [154] (adapted).

The crack nucleation appears at a microscopic scale, so with the appearance of the scanning electron microscope it was also possible to study with more detail this phenomenon because the microscopes could magnify up to 1000 times at the time of its appearance but, with this new device it was possible to magnify the sample in 10 times more than the microscope.

The appearance of the extrusions or inclusions occur at the surface of metals and is related to the effects of cyclic loadings which when there isn't elastic recuperation of the dislocation, these extrusions and intrusions will create persistent slip bands (PSB). The intrusions lead to stress concentrations that could be the origin of the crack nucleation but, even if the crack doesn't nucleate at an intrusion the persistent slip band are always directly or indirectly related to the crack nucleation.

On the Figure 41 is represented schematically the appearance of inclusions and extrusions on a metal surface under cyclic loading, where the vertical arrows represent the loading direction, and the horizontal arrows represent the progression of the slip lines. The representations of the fatigue cracks presented above show the propagation on a plane but, the fatigue cracks are three-dimensional since they propagate through the surface (which isn't represented on the Figure 40 and Figure 41) and depth. Some slip bands can be removed by polishing the surface, but on the surface more distinct slip bands could be present which won't be removed by the polishing which are called persistent slip bands. since the slip bands are directly or indirectly responsible by the crack nucleation, the removal of the slip bands especially before the creation of PSB's could improve substantially the fatigue life, using for instance the electropolishing since this process can remove several microns and with it delaying the formation of PSB's. Of course, these techniques for delaying the crack nucleation are feasible if it is expected that the crack appears at the visible surface, for instance in the injection molds there are cases where the fatigue starts at the cooling/heating channels [141], and these channels aren't easily accessible or even if the crack nucleation starts due to metal defects, here the delaying of the fatigue nucleation could be more difficult or impossible.

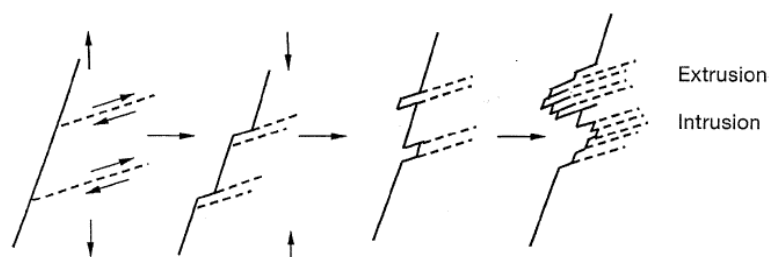


Figure 41 - Fatigue progression and detail of the appearance of inclusions and extrusions under cyclic loading [151].

The crack nucleation at the surface can be easily explained. The region where usually the stress concentration appears is at the surface and also the region of less resistance to the appearance of slip bands and with-it inelastic deformations.

The crack nucleation at slip bands tend to grow on the maximum shear stress plane since this is the main factor that leads to the slip bands appearance, here the grow of the microcrack is strongly influenced by the grains size and organization and the extent of the plasticity of the crack tip. Also, at this stage the crack nucleates at the maximum shear stress plane so is possible to admit that this is a two-dimensional crack as represented in Figure 42. The crack growth by the shear plane doesn't mean that it will grow through the same plane because the crack can change its direction, but always growing oriented near or at the maximum shear plane. At this point the crack can assume a three-dimensional shape.

As stated, not all fatigue cracks nucleate at the slip bands, they usually occur at a material discontinuity, and if there is a material discontinuity that is more suitable for the crack nucleation than the slip bands won't be the initiation location of the crack. The cracks can also sometimes occur just below the surface, but this is less common.

For brittle metals or high strength metals when the microcracks usually appear at inclusions or voids and they tend to grow along the maximum tensile stress planes, so for these materials the propagation has a different behavior than for ductile materials where crack grows along the maximum shear stress plane.

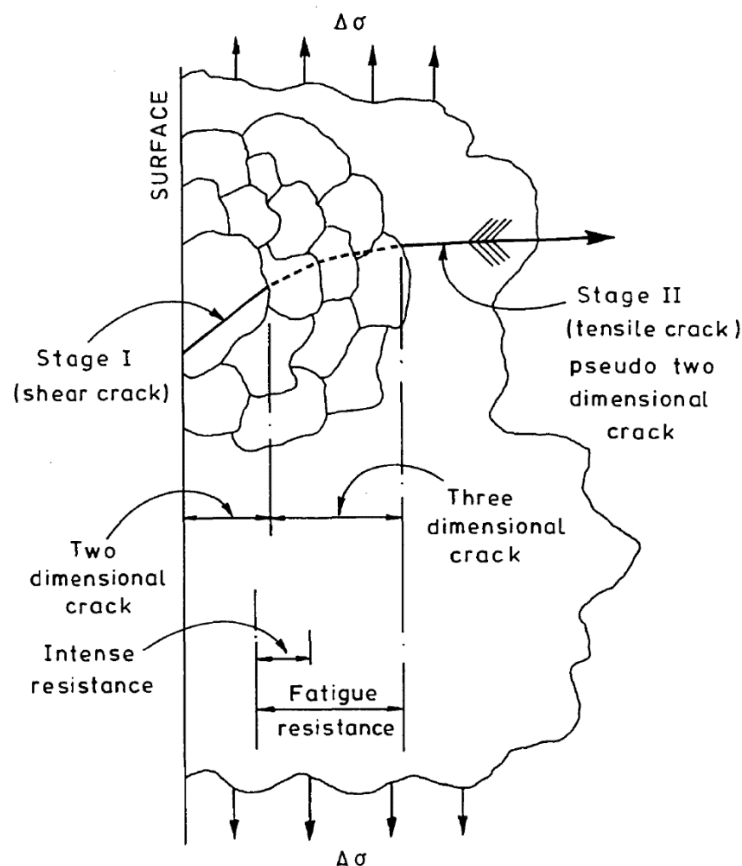


Figure 42 - Schematic of stages I and II of the propagation stages for transcrystalline microscopic fatigue crack, and the indication of the fatigue resistance in terms of difficulty of stage I crack pass to a stage II crack [155].

The effects of temperature during cyclic loading could have a significant influence on the fatigue life, when the structure is at moderate temperature the grain boundary precipitates and with it there is a strengthen of the metal. But, when metals are subjected to higher temperatures the reduction of its fatigue strength can be observed since the same precipitates that at moderate temperatures can increase its strength, here they have an opposite influence. The effects of the temperature in the fatigue endurance limit will be addressed again in section 3.6.5 with more detail.

A factor that can have a big impact on the metal fatigue-life is the presence of impurities, this is a factor that designer engineers can't control in advance and these impurities segregated to the boundaries of the grains can lead to material embrittlement and with it the crack nucleation at a grain boundary could appear, forward by its growth by the grain boundary.

Up to this point was presented factors that could lead no crack nucleation due to stresses and strains but, microcracks can be present before the first load cycle due to the material manufacturing process or surface treatment and here the nucleation phase could not exist if the stressed region already has a microcrack.

When the microcracks appear and cyclic loads are applied they tend to coalesce and grow along the plane of maximum tensile stress. For the fatigue crack growth there are two stages to define it, which are called stage I and stage II that represent the shear mode and tensile mode respectively, these stages are represented schematically on the Figure 42 where is possible to demonstrate how the crack grows on each stage.

As stated before, usually the crack nucleation happens at regions where due to cyclic loads, intrusions and extrusions start to form up to the point that persistent slip bands appear. Which due to the continued application of cyclic loads, then the microcracks are formed that can only be seen using high magnification. If the stress and strains are high enough these cracks tend to merge and grow and then become visible to the naked eye and if they reach a critical size sudden fracture occurs. But cracks may also stop and present no more growth if they reach into compressive residual stress regions or if the crack grows up to a region with stresses or strains not high enough to promote the growth. As shown in Figure 42 there is a region where there is a greater resistance to crack nucleation, this is due to the boundaries between grains that tend to be a barrier to slip.

For the design of structures, to outstand the applied loads during its lifetime, most of the times there is a compromise that must be made to achieve the best results. Because for metal alloys, the factors that can improve the metal resistance to the crack nucleation usually don't favor the macrocrack growth resistance. For instance, the grain size, if the size of the grain is small there is some resistance to the crack nucleation but, when this resistance is overcome, the crack can grow faster on this finer grain. But

if the grain presents a coarser structure it is less resistant to the crack nucleation but more resistant to the crack growth since the grain boundaries act as deflectors or stoppers. However, it must be taken into account that cracks may already be present prior to the beginning of the alternating loadings.

As soon as a micro-crack or micro-cracks begin to appear, because not all of them eventually propagate, as the alternating loads continues, the crack tends to appear and later, if conditions allow it to grow. This propagation occurs starting in stage I and as the crack continues it progresses to stage II as shown in Figure 42. Figure 43 present two ways in which these two stages can occur. As the cracks can propagate along the free surface as is illustrated in Figure 43 a) or into its core as can be seen in Figure 43 b).

Stage I always occurs along the maximum shear plane until conditions exist for normal stresses to become more relevant for crack propagation. Once the maximum normal stresses are primarily responsible for crack propagation, stage II of crack propagation begins, and the crack will propagate perpendicular to the maximum principal stress.

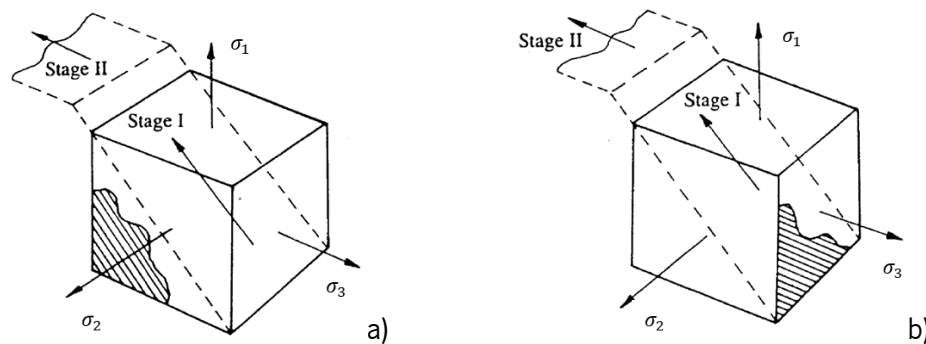


Figure 43 – Two different fatigue crack propagations and representation of the first and second stages for each one. Where a) is the representation of crack that grow parallel to the free surface and b) is for cracks that propagate away from the free surface [155] (adapted).

Cracks propagating along the surface (Figure 43 a)) tend to be less dangerous than those propagating away from the free surface to the core. For the latter, they tend to have less material to support the stress caused by alternating loading which can lead to brittle fracture being reached more quickly.

3.3 S-N curves

The S-N curves also known by Wöhler curves, are created by the results of several trials at different amplitude stresses (S_a). The number of cycles is usually plotted on a logarithmic scale, as for the stress amplitude can be presented on a logarithmic or linear scale, but the use of a logarithmic scale is preferred to get a linear scale between the S_a and the N axis for a high range of the N values.

The region where the probes fail between 1 and 10^4 cycles is known as the low cycle fatigue region (LCF), between this region and 10^7 to 10^{10} cycles is called the high cycle fatigue region (HCF) and for higher number of cycles is known as giga cycle fatigue region (GCF). Even if the border between the LCF and HCF is a matter of discussion and being not well defined, is important to understand that at the low

cycle fatigue region the fatigue is associated to the microplastic deformation on each cycle and the high cycle fatigue region the fatigue is connected to the elastic behavior on a macro scale, even if as known plastic deformation could exist at microscopic levels.

The S-N curves made from probes are usually made for an $R = -1$ ($S_m = 0$), as it will be addressed on the section 3.6.7 where the mean stresses effects are referred, there are methods to account for these effects on this curve. Of course, better results would be archived if conducted trials for the part working ratios but, the trials are very time consuming and expensive. As shown in the Figure 45 the mean stress effects have an impact on the S-N curve.

But, by general experience the stress amplitude (S_a) shows a higher impact on the fatigue than mean stress effects. Also, in terms of fatigue life, the reverse loading shows a significative impact on the crack tip plasticity because crack opening is very considerable for the crack extension, which makes the crack progression harder for negative mean stresses, that's one of the main reasons for the fatigue diagrams are usually made for mean stresses equal or higher than zero.

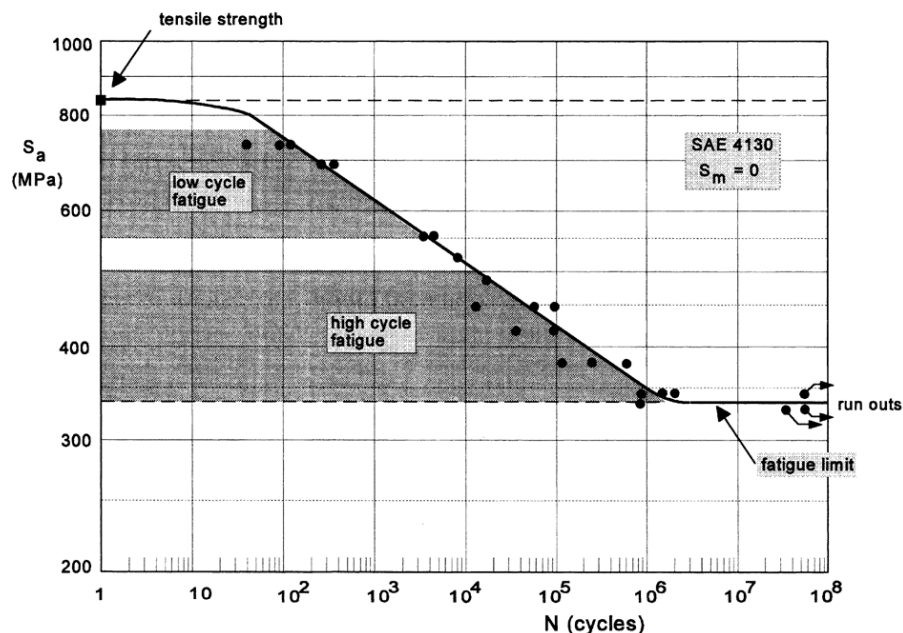


Figure 44 – Representation of a S-N curve of a low-alloy steel SAE 4130 [156] (adapted).

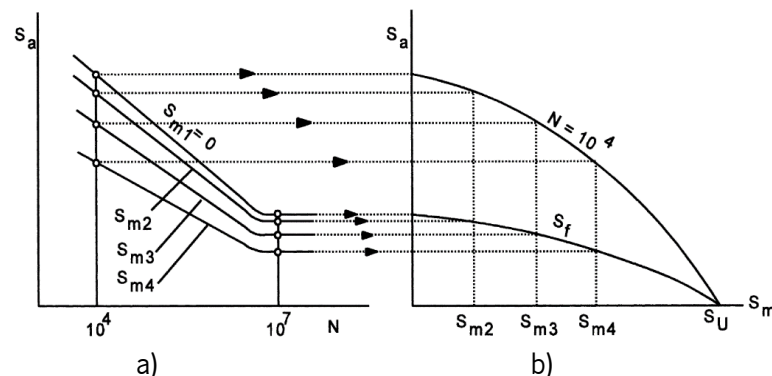


Figure 45 – Fatigue diagram for different mean stresses where a) is the representation of the S-N curves for different S_m values and b) is the representation of the fatigue diagram obtained from the S-N curves [156].

When trials data are not available there are several theoretical models which could generate a S-N curve based on easily obtained properties, some of these models will now be presented.

As stated before, the trials to obtain the S-N curve for a given material are very time consuming so, the use of monotonic properties are a good approach for their construction. As mentioned in the work of Bandera, et al [157] the use of the of Basquin and Manson-Coffin equations for the LCF and HCF regions and the Murakami formulation for the GCF region are very used on steels since they are based on the steels monotonic properties. Due to the lack of mathematical formulations to develop a theoretical S-N curve, on their work it was used a model for the LCF and HCF region and other for the GCF region. In the Figure 46 is possible to observe the different cycle regions and also where the Basquin formulation is valid.

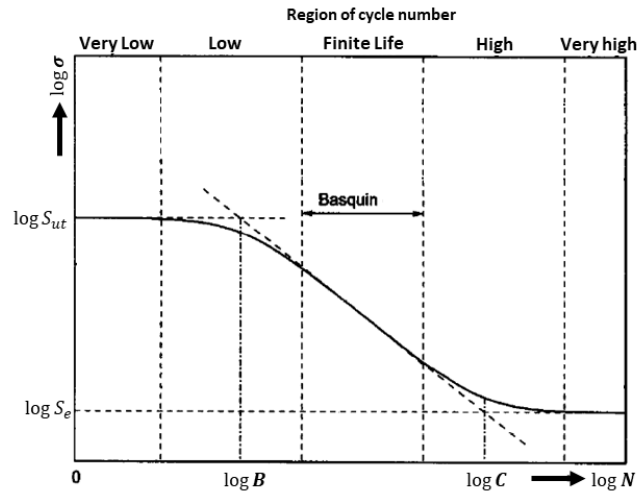


Figure 46 – S-N curve representation where is shown the regions of the curve [135] (adapted).

3.3.1 Basquin equation

The Basquin equation is a simple function that describes the fatigue limit dependence on the number of cycles as presented in the Equation 1.

$$S(N) = aN^b \quad \text{Equation 1}$$

Which can be extended to the low-cycle region by the Equation 2:

$$S(N) = a(N + B)^b \quad \text{Equation 2}$$

For the high cycle-region the Equation 3 can be used:

$$S(N) = aN^b + S_\infty \quad \text{Equation 3}$$

The previous equations parameters will be described below when the Palmgren function is addressed, except for the S_∞ which represents the permanent fatigue limit.

3.3.2 Palmgren, Kohout and Vechet equations

From the Basquin equation other methods have been developed like the Palmgren function or the Kohout and Vechet which are expressed by the Equation 4 and Equation 5 respectively:

$$S(N) = a(N + B)^b + c \quad \text{Equation 4}$$

$$S(N) = a \left(\frac{N + B}{N + N_k} \right)^b \quad \text{Equation 5}$$

Where:

$S(N)$ – is the stress amplitude (S_a) for a certain number of cycles (N).

a , b and c – are the parameters of the function that are related to the material.

B and N_k – are the number of cycles at the first and second knee points respectively.

The knee points are represented in the Figure 47 and as it will be demonstrated, are important points for all the formulations to generate theoretical S-N curves.

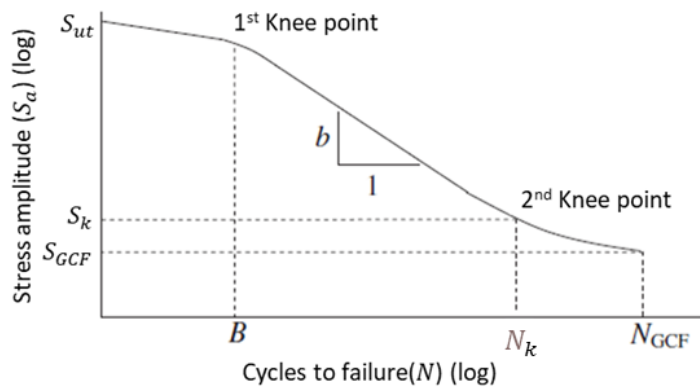


Figure 47 – Representation of the full range S-N curve and some related terms [157] (adapted).

The Equation 4 and Equation 5 are able to represent the S-N curve from the first cycle (S_{ut}) up to the GCF region, where the main difference between both curves is at the slope in the GCF region. The Palmgren function presents a continuing slope in the GCF region as for the Kohout and Vechet function admits that the curve is horizontal at the GCF region.

3.3.3 Simplified full range S-N curve

The method proposed by Bandara, et. al [157] is based on the Palmgren formulation presented in the Equation 4, next the procedure used to develop this simplified model will be described.

For the Palmgren formulation is necessary to determine the parameters a , b , c and B . to do so, it must be considered first the variables a , c and B , where in order to determine these values, three coordinates of the S-N curve will be determined from a set of principles, which will then be used in the development of the formulation.

The first coordinate to be used is for $N = 1/4$, where is considered that $S(1/4) \cong S_{ut}$. For the value of B is considered that this value is much larger than $1/4$ cycles so $B \cong 1/4 + B$ so, having this into account:

$$S_{ut} = aB^b + c \quad \text{Equation 6}$$

As for the second coordinate which is the knee point at the HCF region and GCF region interface, as shown in the Figure 47 assume de coordinate $(N_k; S_k)$, being N_k the number of cycles and S_k the respective stress amplitude which is represented by:

$$S_k = aN_k^b + c \quad \text{Equation 7}$$

For the third coordinate which is in the GCF region represented in Figure 47 which the coordinates $(N_{GCF}; S_{GCF})$, where is usually assumed that the value for the N_{GCF} is between $10^7 - 10^{10}$ and like for the points presented before the stress amplitude is represented by:

$$S_{GCF} = aN_{GCF}^b + c \quad \text{Equation 8}$$

By solving the above is obtained the following equations:

$$a = \frac{S_{GCF} - S_k}{N_{GCF}^b - N_k^b} \quad \text{Equation 9}$$

$$c = \frac{1}{2} \left(\frac{(S_{GCF} + S_k)(N_{GCF}^b - N_k^b) - (S_{GCF} - S_k)(N_{GCF}^b + N_k^b)}{N_{GCF}^b - N_k^b} \right) \quad \text{Equation 10}$$

$$B = \left(\frac{S_{ut} - c}{a} \right)^{\frac{1}{b}} \quad \text{Equation 11}$$

To obtain the values for a, c and B, can be gathered by solving the equation presented above by replacing the values of S_{ut} , S_{GCF} , S_k , and N_k . Of course, the only property known by the material monotonic curve is the ultimate strength (S_{ut}). All the other values can be determined by using the formulations which will be presented in section 3.3.3.

By Means of the model proposed by Murakami with Wang's modifications which is presented on the paper of Bandara, et all [157], where is shown a simplified fatigue strength prediction formula for HCF-GCF region, which the authors consider that this region appears for values of $N_{GCF} > 10^7$ cycles. And for a ration of R= -1, so they propose the following equation:

$$S_{GCF} = \frac{S_{ut}^{1/3}}{1000} (Hv + 120)(155 - 7 \log N_{GCF}) \quad \text{Equation 12}$$

Where:

S_{GCF} – Is the fatigue strength at GCF range in N/mm^2 .

Hv – Vickers hardness in $kg.f/mm^2$.

For this formulation is considered that the value of S_k can be obtained by:

$$S_k = 0,5S_{ut} \quad \text{Equation 13}$$

And substituting S_{GCF} by S_k and N_{GCF} by N_k on the Equation 12 and since the value of S_k is obtained by the Equation 13, is possible to know the value of N_k by:

$$N_k = 10^{\left(\frac{0.155(Hv+120)S_{ut}^{1/3} - 0.5S_{ut}}{0.007(Hv+120)S_{ut}^{1/3}} \right)} \quad \text{Equation 14}$$

So, by the modified Palmgren function the S-N curve can be created by the following equation:

$$\begin{aligned} S(N) &= \left(\frac{S_{GCF} - S_k}{N_{GCF}^b - N_k^b} \right) (N + B)^b \\ &+ \frac{1}{2} \left(\frac{(S_{GCF} + S_k)(N_{GCF}^b - N_k^b) - (S_{GCF} - S_k)(N_{GCF}^b + N_k^b)}{N_{GCF}^b - N_k^b} \right) \end{aligned} \quad \text{Equation 15}$$

This model presents a limitation in terms of range of steels, being only valid for steels with a ultimate strength (S_{ut}) equal or inferior to 1400 MPa and $Hv > 70 \text{ kg.f/mm}^2$ [157].

To obtain all the needed parameters for the Equation 15 is still needed to obtain the value of the exponent b, which is the slope of the S-N curve between the first and second knee. So, empirically is possible to state that the fatigue strength at the first knee varies between the range of $0,76S_{ut}$ up to $0,9S_{ut}$. On the work made by Bandara, et al. [157], is stated that the value of $0,76S_{ut}$ is more suitable for steels with $S_{ut} < 400 \text{ MPa}$ and $0,9S_{ut}$ for steels with $400 < S_{ut} < 1400 \text{ MPa}$. But they also present a global value for b of $-0,2$. On their work it was presented some results for several steels where is compared the theoretical curves with trials results which is presented on the Figure 48, where was possible to observe that there is a good relationship between the predicted results and the trials. Having a special interest for high strength steels which have the most interest for this work as it will be possible to observe later it is possible to observe that the modified Palmgren formulation present promising results for high strength tool steels.

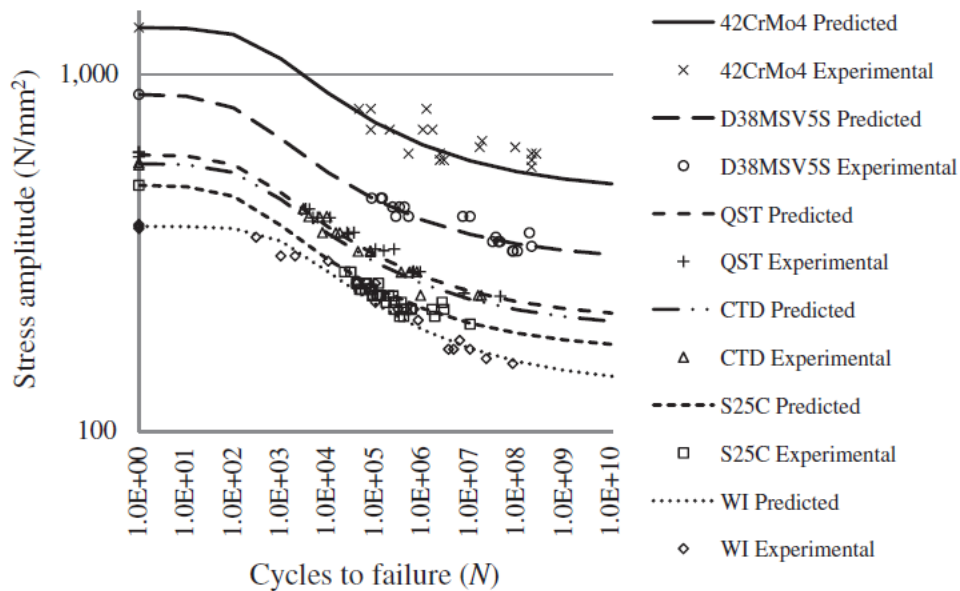


Figure 48 – Predicted S-N curves using the proposed formula using the global value for b and experimental S-N data [157].

3.3.4 Strzelecki model

Now it will be presented another model which also presents a relationship between the hardness and the fatigue life known as Strzelecki model [158]. The modified Palmgren formulation was based on the ultimate tensile strength and material hardness, as for the Strzelecki model also uses the tensile strength and material hardness but uses too the material yield strength. For these models the following relationships are used:

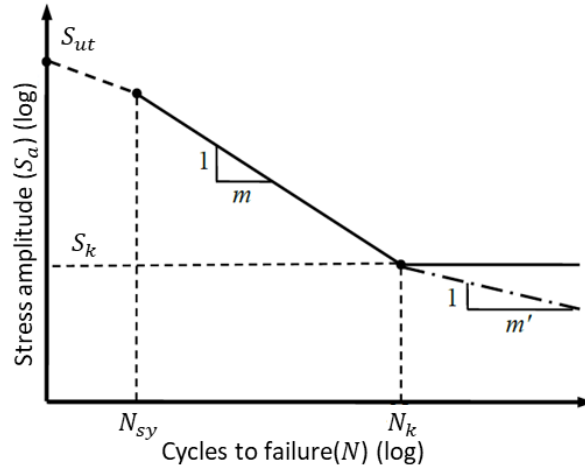


Figure 49 – Illustration of the S-N curve with accordingly with the Strzelecki formulation [158]

$$m = \frac{\log\left(\frac{10^6}{N_{sy}}\right)}{\log\left(\frac{0,9S_y}{S_k}\right)} \quad \text{Equation 16}$$

$$N_{sy} = 400 \left(\frac{S_y}{S_{ut}}\right)^{-10} \quad \text{Equation 17}$$

$$A = m \cdot \log(S_k) + \log(N_0) \quad \text{Equation 18}$$

Where:

S_y – Yield strength of the material.

N_{sy} – Number of cycles to failure at stress amplitude equal to $0,9S_y$.

A – Constant from Basquin equation.

m – Slope of the S-N curve accordingly with the Figure 49 .

The value of N_0 , which represents the inflection point is assumed to be 10^6 cycles [158]. If the variables m , N_{sy} , A from the Equation 16, Equation 17 and Equation 18 respectively are replaced on the Basquin Weibull equation (Equation 19), is obtained the expression for the S-N curve for the low cycle region up to the high cycle region.

$$\log(N) = A + m \log(S_a) \quad \text{Equation 19}$$

$$S_a = 10^{\left(\frac{\log(N) \log\left(\frac{0,9S_y}{S_e}\right) - \log\left(\frac{10^6}{N S_y}\right) \log(S_e) - \frac{\log(N_0) \log\left(\frac{0,9S_y}{S_e}\right)}{\log\left(\frac{10^6}{S_y}\right)} \right)} \quad \text{Equation 20}$$

This model doesn't present any formulation for the GCF region, it considers by other work made by Strzelecki, et al. [159] a value of m' (Figure 49) equal to 15 for normal stresses and 25 for shear stresses and the curve is valid $10^6 < N < 10^8$, for higher number of cycles is admitted that the curve is horizontal.

3.3.5 Hardness method

Another method to develop an S-N curve based on the hardness which is proposed by Bandara, et al. [160] that is based on the Palmgren formulation (Equation 2). For this method there is a key point to its formulation which is the relation between the metal tensile stress and the hardness:

$$S_{ut} \cong 3,40HB \quad \text{Equation 21}$$

This Equation 21 is only valid for steels as stated by Roessle and Fatemi [139], where they also present a similar expression (Equation 22) and presents its limitations such as that is only valid for low and medium strength carbon alloy steels. On their paper they present other formulation to express the steel ultimate strength which is presented on Equation 23. Form the analysis of Figure 50 where the Equation 22 and Equation 23 are compared is possible to observe that for higher values of hardness the differences start between both equation start to increase where the Equation 23 shows a better fit.

$$S_{ut} \cong 3,45HB \quad \text{Equation 22}$$

$$S_{ut} \cong 0,0012(HB)^2 + 3,3(HB) \quad \text{Equation 23}$$

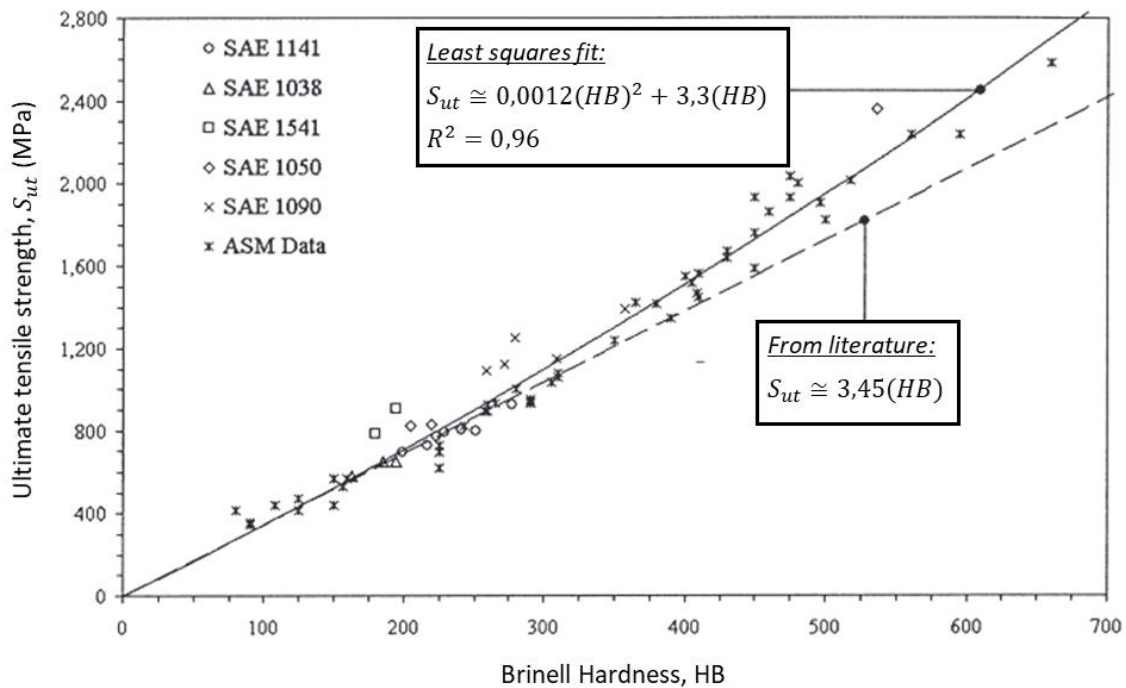


Figure 50 – Plot of the relations between the ultimate strength and steel hardness [139].

So, having under account the Equation 4, Equation 9, Equation 10 and Equation 11 which are the full range equation, the a , c and B parameters respectively. The formulation proposed by Bandara, et al. [160] focus on the S_{GCF} , S_k and N_k values which are determined by the following equations:

$$S_{GCF} = 0,0015(HB)^{1/3}(HB + 120)(155 - 7 \log(N_{GCF})) \quad \text{Equation 24}$$

Where, the value of N_{GCF} should be equal or higher than 10^3 . Now the determination of the number of cycles at the point which the 2nd knee point occurs (see Figure 47), can be determined by:

$$S_k = 1,7HB \quad \text{Equation 25}$$

To determine the number of cycles for the stress amplitude S_k the authors propose the Equation 26:

$$N_k = 10^{\left(22,14 - \frac{162(HB)^{2/3}}{HB+120}\right)} \quad \text{Equation 26}$$

To determine the value of the slope b the authors use the same empirical principal as for the Palmgren formulation to obtain the values for the stress amplitude for the first knee but, instead of determining the value of the number of cycles where the first knee occurs here is assumed that the first knee happens at 1000 cycles. As for the Palmgren formulation having the values of the stress amplitude and number of cycles of the first knee and second knee is possible to determine the value of the slope but, here as for the Palmgren formulation the authors also present a global value which is 0,21.

3.4 ε -N curves

The S-N curves are a good approach for the fatigue analysis at the HCF region but, the results obtained by trials present a higher data scatter at LCF regions, this is related to the presence of plastic deformation which on the ε -N curves is better defined as it will be shown on this chapter.

3.4.1 Coffin-Manson relationship

This approach relates the number of cycles to reach failure ($2N_f$) to the strain amplitude ($\Delta\varepsilon/2$). This relation is commonly called by the Coffin-Manson relationship and can be written as:

$$\frac{\Delta\varepsilon}{2} = \frac{\Delta\varepsilon_e}{2} + \frac{\Delta\varepsilon_p}{2} \quad \text{Equation 27}$$

Where:

$\frac{\Delta\varepsilon}{2}$ – represents the total deformation.

$\frac{\Delta\varepsilon_e}{2}$ – represents the elastic deformation.

$\frac{\Delta\varepsilon_p}{2}$ – represents the plastic deformation.

At the HCF the fatigue mechanism at a macro scale shows to be mostly an elastic phenomenon but, at LCF the plastic deformation occurs at every cycle. As is possible to observe on the Figure 51 where is represented the stress-strain loops and how the stress amplitude changes for the same condition at LCF,

during each cycle for the same strain amplitude the stress amplitude changes, now the stress amplitude decrease, or increase is related to the material hardening or softening.

The evaluation of the rate and extension of the cyclic hardening or softening is possible by recording the stress variation in function of number of cycles for a determined extension. The Figure 51 a) shows the increase of the stress amplitude for each cycle which means the material is hardening, the opposite is shown in Figure 51 b) where is shown the stress effects of material softening. This effects are more evident at the first cycles then it tends to stabilize, when the material reaches its cyclic transient behavior, where the cycle stabilization represents about 10 to 40% of the total fatigue life [156].

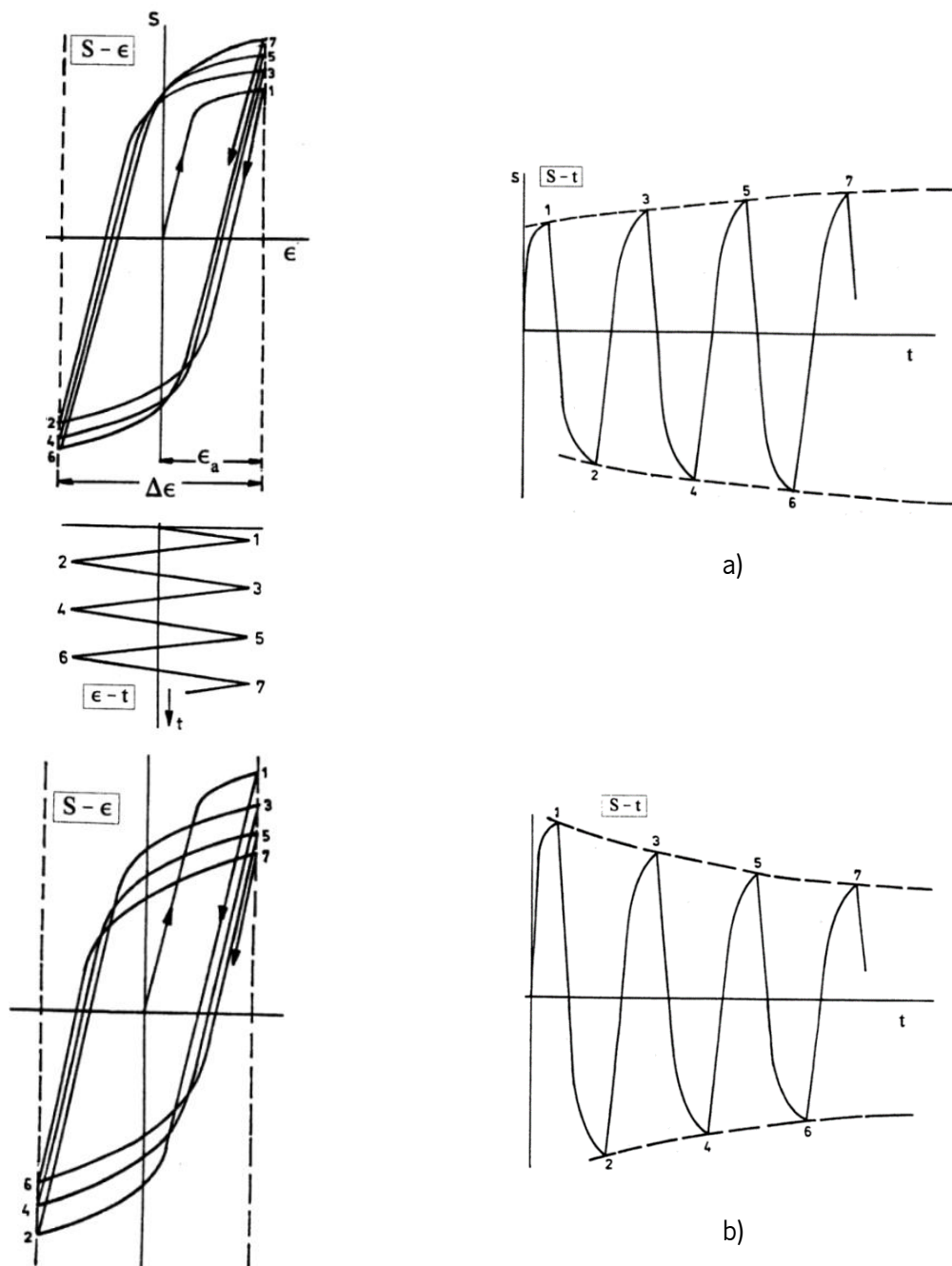


Figure 51 – Stress-strain loops during constant strain amplitude cycles and stress history for cyclic a) hardening and b) softening [156].

Of course, the ε - N curves aren't exclusive for the design at LCF, as it was presented above on the Equation 27 it is possible to observe that it has terms to account for the elastic and plastic phenomenon's and as it will be presented through this work this method to characterize a material is useful for fatigue design under multiaxial stresses independently if it is for LCF or HCF.

The Coffin-Manson relationship presented above on Equation 27 and as stated before it has a term to account for the elastic and for the plastic behavior, these terms can be determined by:

$$\frac{\Delta\varepsilon_e}{2} = \frac{\sigma'_f}{E} (2N)^{bf} \quad \text{Equation 28}$$

Where:

σ'_f - fatigue strength.

E - Young's Modulus.

N - Number of cycles.

bf - fatigue strength exponent.

$$\frac{\Delta\varepsilon_p}{2} = \varepsilon'_f (2N)^{cf} \quad \text{Equation 29}$$

Where:

ε'_f - fatigue ductility coefficient.

cf - fatigue ductility exponent.

By substituting the Equation 28 and Equation 29 in Equation 27 is obtained:

$$\frac{\Delta\varepsilon}{2} = \frac{\sigma'_f}{E} (2N)^{bf} + \varepsilon'_f (2N)^{cf} \quad \text{Equation 30}$$

As presented for the S-N curves, now it will be shown theoretical approaches to obtain the needed parameters for the Coffin-Manson equation.

3.4.2 Four-point correlation method

The first formulation to be presented is the four-point correlation method. This method proposes the use of four points to obtain the elastic and plastic curves, as is possible to understand by the Coffin-Manson equation the curve is the sum of the elastic and plastic deformations, this is possible to comprehend from the Figure 52.

By the Figure 52 is possible to observe that the elastic curve and the plastic curve present themselves as straight lines which each line can be determined by 2 points. On this work won't be displayed the procedure presented by Manson [161] to obtain the parameters presented below, it will only be presented the final approach to obtain the needed parameters. This method uses the monotonic tensile properties to estimate all the necessary parameters, so to determine the needed properties for the Coffin-Manson

equation by this method is used the following equations:

$$\sigma'_f = \frac{E}{2} \times 10^{bf \log 2 + \log \left[\frac{2,5 S_{ut}(1+\varepsilon_f)}{E} \right]} \quad \text{Equation 31}$$

$$bf = \frac{\log \left(\frac{2,5(1+\varepsilon_f)}{0,9} \right)}{\log \left(\frac{1}{4 \times 10^5} \right)} \quad \text{Equation 32}$$

$$\varepsilon'_f = \frac{1}{2} \times 10^{cf \log \left(\frac{1}{20} \right) + \log \left(\frac{1}{4} \varepsilon_f^{3/4} \right)} \quad \text{Equation 33}$$

$$cf = \frac{1}{3} \log \left(\frac{0,0132 - \Delta\varepsilon^*}{1,91} \right) - \frac{1}{3} \log \left(\frac{1}{4} \varepsilon_f^{3/4} \right) \quad \text{Equation 34}$$

Where $\Delta\varepsilon^*$ represents the elastic strain range at 10^4 cycles and is obtained by the Equation 35:

$$\Delta\varepsilon^* = 10^{bf \log(4 \times 10^4) + \log \left(\frac{2,5 S_{ut}(1+\varepsilon_f)}{E} \right)} \quad \text{Equation 35}$$

Where:

ε_f – is the true fracture ductility.

S_{ut} – is the ultimate tensile strength.

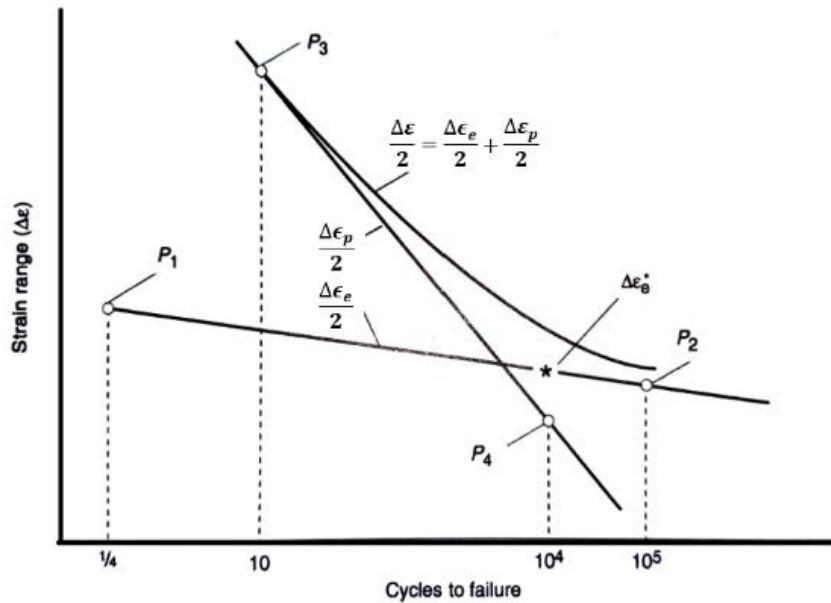


Figure 52 – Schematic of fatigue life curve with the four points highlighted for the elastic and plastic strain lines [162] (adapted).

Other method to obtain the fatigue parameters is the universal slopes method, here is assumed that the slopes of the elastic and plastic-life curves don't present significant changes for different materials and, the fatigue properties are obtained by:

$$\sigma'_f = 1,9018 S_{ut} \quad \text{Equation 36}$$

$$\varepsilon'_f = 0,7579 \varepsilon_f^{0,6} \quad \text{Equation 37}$$

As for the parameters b and c is assumed the values -0,12 and -0,6 respectively.

3.4.3 Modified Four-point correlation method

Having the four-point correlation method as its base, a new proposal was presented and known by modified four-point correlation method. To estimate its fatigue properties the following expressions were proposed:

$$\sigma'_f = S_{ut}(1 + \varepsilon_f) \quad \text{Equation 38}$$

$$bf = \frac{1}{6} \left(\log \left(0,16 \left(\frac{S_{ut}}{E} \right)^{0,81} \right) - \log \left(\frac{S_{ut}}{E} \right) \right) \quad \text{Equation 39}$$

$$\varepsilon'_f = \varepsilon_f \quad \text{Equation 40}$$

$$cf = \frac{1}{4} \log \left(\frac{0,00737 - \frac{\Delta \varepsilon^*}{2}}{2,074} \right) - \log(\varepsilon_f) \quad \text{Equation 41}$$

Where:

$$\frac{\Delta \varepsilon^*}{2} = \frac{S_f}{E} \left(10^{\frac{2}{3} \left(\log 0,16 \left(\frac{S_{ut}}{E} \right)^{0,81} \right) - \log \left(\frac{S_f}{E} \right)} \right) \quad \text{Equation 42}$$

And,

$$S_f \cong \sigma'_f \quad \text{Equation 43}$$

3.4.4 Universal slopes method

For the Universal slopes method, it was also proposed a modification which is known as the modified universal slopes method, where the fatigue properties are obtained by:

$$\sigma'_f = E \times 0,623 \left(\frac{S_{ut}}{E} \right)^{0,832} \quad \text{Equation 44}$$

$$\varepsilon'_f = 0,0196 \varepsilon_f^{0,155} \left(\frac{S_{ut}}{E} \right)^{-0,53} \quad \text{Equation 45}$$

And for this method it was also presented global values for the parameters bf and cf which are - 0.09 and -0.56 respectively.

Another method to obtain the needed fatigue properties is the uniform material law, which presents similarities to the universal slopes method but, in this method for the slopes of the elastic and plastic curves is presented different values for different types of materials such as steels, aluminum and titanium. For this work only be presented the formulation to low-alloy or unalloyed steels so, the fatigue properties are determined by:

$$\sigma'_f = 1,5 S_{ut} \quad \text{Equation 46}$$

$$\varepsilon'_f = 0,59 \psi \quad \text{Equation 47}$$

Where:

$$\psi = 1 \text{ for } \frac{S_{ut}}{E} \leq 0,003;$$

$$\psi = 1.375 - 125 \frac{S_{ut}}{E} \text{ for } \frac{S_{ut}}{E} > 0,003.$$

As for the values of b and c they assume the values of -0,087 and -0,58 respectively.

3.4.5 Hardness method

The last method to be presented is based on the material hardness, the Roessle and Fatemi method is valid for materials with a hardness values comprehended between 150HB and 700HB. To obtain the fatigue properties by this method the following expressions are used:

$$\sigma'_f = 4,25(HB) + 225 \quad \text{Equation 48}$$

$$\epsilon'_f = \frac{1}{E} [0,32(HB)^2 - 487(HB) + 191000] \quad \text{Equation 49}$$

As for the values of b and c this method proposes the same values as the modified universal slopes method. But, for this same method Roessle and Fatemi [139] proposed some improvements in terms of accuracy for steels. As for the fatigue strength coefficient (σ'_f) instead of the Equation 48 is presented the following equation:

$$\sigma'_f = 1,04(HB) + 345 \quad \text{Equation 50}$$

For the fatigue ductility coefficient (ϵ'_f) there are no modifications so Equation 49 still valid but, the fatigue strength exponent (bf) is assumed a global value of -0,09. As for the value of the fatigue ductility exponent (cf) is assumed a global value of -0,56.

3.5 Comparison between S-N and ϵ -N curves

The use of S-N curves is common since the work presented by Wöhler in the 1850s, although now they can present a more complex approach. The S-N method is more useful when applied to actual components or component-like test members [162] because the use of those components has the advantage to include factors like surface finish, notches and other factors on the curve. But, when the S-N curve is obtained from a probe it could be difficult to account for the penalty factors such as surface finish, notches, residual stresses, and others. Another disadvantage is the account of mean stress effects in notches based on normal stresses due to the load sequences, by the Figure 53 where is present a σ - ϵ diagram of a notched member, is possible to observe that the stress-strain response is different depending on the load sequence.

The strain-based models have the advantage of better predicting the mean stress effects on notches but, the finite element method analysis can be a powerful tool to overcome this problem for S-N prediction models. But, for low cycle fatigue the ϵ -N models present better results than the S-N models. The S-N models for low cycle fatigue use factors that aren't necessary on the ϵ -N models so, it's safe to say that

S-N models shouldn't be used for LCF, they present better results for HCF.

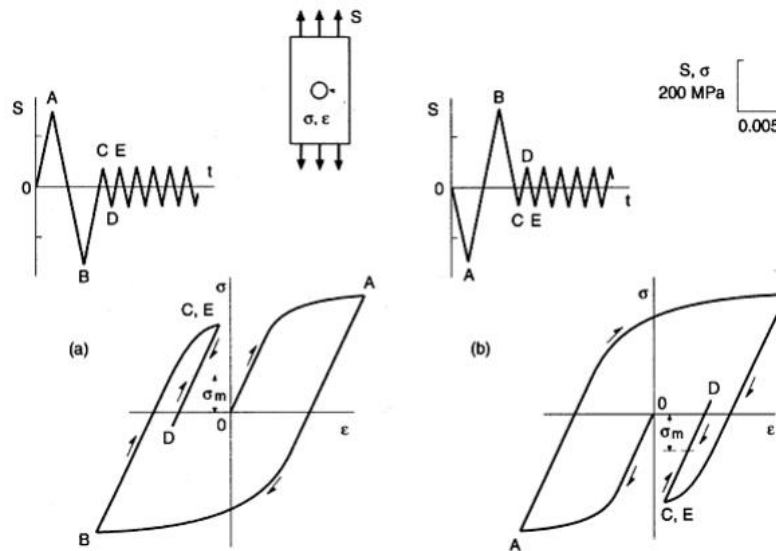


Figure 53 – Two similar load histories diverging only in the first cycle (OABC) applied to a notch member [163] (adapted).

The multiaxial fatigue models can be categorized as stress based, strain based, energy based and fracture mechanics-based model being the last one not addressed because it is outside the goal of this work. But, it is commonly known that stress-based models are limited to regimes where the plastic deformations are negligible as for the strain-based models, can be used for LCF and HCF but the range of application should always be well analyzed for each method.

3.6 Factors affecting the fatigue life

To perform the fatigue analysis of a component for LCF or HCF regions, it is indispensable to account for the many factors which can have an impact on the fatigue behavior of the material, some of the major factors which can have an impact on the fatigue behavior are [164]:

- Loading type (k_L).
- The component size and stress or strain distribution (k_b).
- Surface finish (k_s).
- Effects of notches (k_t).
- Temperature effects (k_d).
- Mean stress or strain effects.
- Environmental effects.
- Metallurgical factors and material properties.
- Strain rate and frequency effects.

Obtaining the curves that characterize the fatigue behavior of a material are usually done for a specimen free of factors that contribute to crack formation, that is, without notches, with good surface finish and at a controlled environment. Often, when it is not possible for the designer to test specimens

at or near working conditions, it is necessary to use existing data and apply correction factors that take the above effects into account. But, for the application of these effects, there are methods that have been studied and that can currently be found in the literature available on the subject, that will be presented briefly below.

Now the correction of the curve, so it can approximate to the conditions under which the component will work, is done by correction factors that are related to the metallic material and its behavior when subjected to cyclic loads. Figure 54 shows how the correction factors are applied to fit the material curve, with the example given here being for an S-N curve. With this example is possible to verify which are the factors that have an influence from the first load cycle to the factors that start to be more influent with the approach to the gigacycle regime.

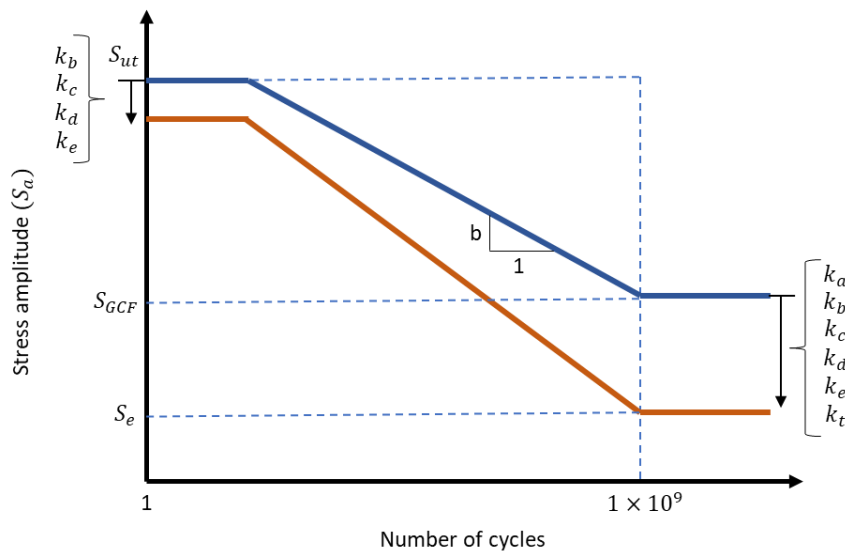


Figure 54 – Correction factors applied to a S-N curve obtained from a smooth test probe.

And then the consideration of mean stress or strain effects which is an adjustment of the value of cyclic loading to the existing curve for the material as will be explained in section 3.6.6.

3.6.1 Loading type (k.)

When materials are studied to evaluate their fatigue endurance to variable amplitude loading, there are three major loading types that can be used which are bending, axial and torsional loadings. Since most material testing trials are made for one kind of loading, and in the real world the applied loads could be from different kinds or even a combination of loadings. So, to account for these effects is used a load correction factor (k_c).

Where:

$$k_c = \begin{cases} 1 & \text{Bending} \\ 0,85 & \text{Axial} \\ 0,59 & \text{Torsion} \end{cases}$$

3.6.2 The component size and stress or strain distribution (k_s)

By the trials made with unnotched probes was possible to observe that the fatigue strength varies with the probe size. By the research made on this subject the theory is that this is caused by the stress gradient [164].

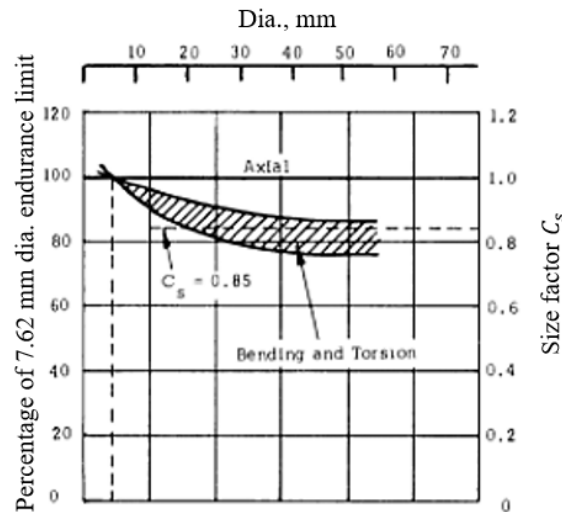


Figure 55 – Size factor (C_s) for steels [164] (adapted).

As can be comprehended from previous statements and research, the fatigue limit is determined by the recourse to trials of probes, which will tend to crack from the weakest stressed region at the surface so, with the increase of the surface area the probability of appearing such points will also increase. So, it is logical that probes with higher stressed surfaces will tend to have a lower fatigue limit.

By the research made by Lipson and Juvinal [165] is possible to obtain a correction factor for unnotched specimens to get a first approximation but, for more precise results experimental trials are indispensable. So, to specimens up to 13 mm in diameter the probe fatigue endurance limit will decrease around 15%, from 13 to 50 mm in diameter the variance is low so for probes in this diameter range is considered a C_s value of 0.85 for bending and torsion as for axial loadings is considered that the C_s value is 1 because the size effects are almost negligible.

But, as mentioned above, these values are only valid for notched parts, considering that generally the parts have their most critical locations at the notches, the fatigue strength limit provided for this situation is very important, and will be addressed again in chapter 3.6.4.

3.6.3 Surface finish (k_s)

The probes used in trials to obtain the fatigue curves have a polished surface finish to reduce the influence of the surface finishing defects on the results. But many times, in practice the surface finish is very different, depending on the part requirements.

Since the fatigue failure usually starts at the surface, it is logical that any surface defects or changes on its topology or hardness will have a great impact on the part fatigue life under alternated loadings.

And for this factor there is some kind of correlation between the stress amplitude, where for higher loads the surface aspect has a higher influence on the part endurance to alternated loads. This factor could be related to the morphology of high strength steels, since they have finer grains than low strength steels, so they tend to present a higher notch sensitivity.

Usually, the surfaces are characterized as polished, milled, cold rolled, hot rolled and as forged which helps to divide in groups and with it the identification of the types of surface finishing's.

So, to account the surface effects in the S-N curve it is usual to use a surface finish factor k_s , where the value obtained for this factor is a relationship between the surface finish and the material tensile strength. Using the regression analysis from Mischke is obtained the following expression:

$$k_s = a_s \cdot S_{ut}^{b_s}$$

Where:

S_{ut} is the steel tensile strength.

a_s and b_s are constants obtained from the Table 4.

Table 4 - Surface finish parameters for steels [112].

Surface finish	a (for S_{ut} in MPa)	b
<i>Polished</i>	1.58	-0.085
<i>Milled or cold rolled</i>	4.51	-0.265
<i>Hot rolled</i>	57.7	-0.718
<i>Forged</i>	272	-0.995

It is also possible to obtain the values for the reduction coefficient that take into account the surface finish from graphs as the one shown in Figure 56. Where there are more curves representing different surface finishes that can lead to obtaining a value closer to the one required.

Due to the importance of the surface condition on the fatigue endurance limit, developments have been made to increase the surface resistance to alternated loadings and with it increasing the component lifetime. Some techniques to improve the surface resistance to the crack nucleation which could be from mechanical surface treatments, thermal or chemical surface treatments:

- Shot peening.
- Surface rolling.
- Surface hardening by carburizing, nitriding, flame hardening or induction hardening.

But is always advisable to have some precautions due to stress concentrations caused by the surface treatment, especially in processes such as flame hardening and induction hardening. In order to have a starting point when it is not possible to use experimental tests to determine the surface treatment factor, designers can use values from other researches, such as the ones presented in Table 5.

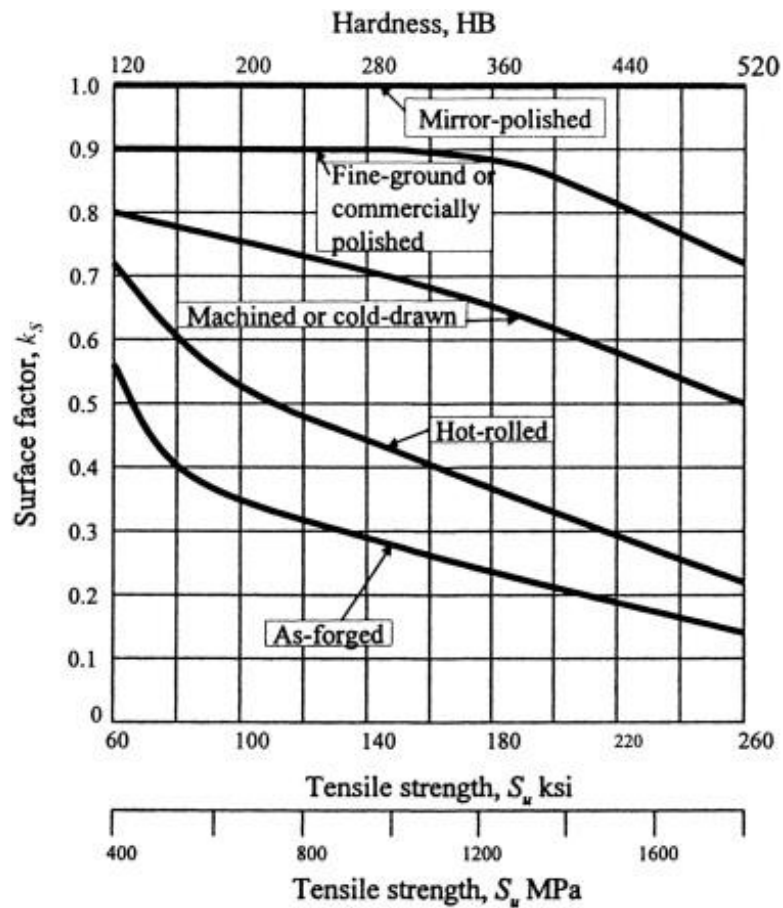


Figure 56 – Surface factor that accounts for the surface finish on the steel fatigue endurance [151].

Table 5 – Surface treatment factors for steels with surface treatments [166] (adapted).

Surface treatment	Unnotched components	Notched components
Steel		
Chemo-thermal treatment		
Nitriding		
Depth of case 0,1 – 0,4 mm	1,1 – 1,15	1,3 – 2
Surface hardness 700-1000 HV10	1,15 – 1,25	1,9 – 3
Case hardening		
Depth of case 0,2 – 0,8 mm	1,1 – 1,5	1,2 – 2
Surface hardness 670 – 750 HV10	1,2 – 2	1,5 – 2,5
Carbo-nitriding		
Depth of case 0,2 – 0,8 mm	1,8	
Surface hardness 670 – 750 HV10		
Mechanical treatment		
Cold rolling	1,1 – 1,25	1,3 – 1,8
Shot peening	1,1 – 1,2	1,1 – 1,5
Thermal treatment		
Inductive hardening		
Flame-hardening		
Depth of case 0,9 – 1,5 mm	1,2 – 1,5	1,5 – 2,5
Surface hardness 51 – 64 HRC	1,3 – 1,6	1,6 – 2,8

As can be observed in Table 5, there are surface treatments that have the ability to increase the resistance of the surface to crack nucleation. Generally, the correction values are penalizing, that is, smaller than 1, but as it can be observed in Table 5 when some surface treatments are applied this value gets higher than 1.

3.6.4 Effects of notches (k.)

On most of the mechanical structures the notches can't be avoided, and their behavior must be understood, so proper design can be performed to reduce or eliminate the risk of mechanical fracture during the structure lifetime. To do so there are five parameters that should be understood how they behave in addition to the behavior of the smooth specimen of the material to be studied, which are:

- Concentrations of stresses and of strain.
- Stress gradients.
- Mean stress effects and residual stresses.
- Local yielding.
- Nucleation and growth of cracks.

On structures where some kind of discontinuities are present, such as holes or sharp angles which causes inhomogeneous stress distributions, induce the appearance of concentration of stresses and strains as shown in Figure 57 where a bar is loaded on its longitudinal direction and with the recourse to photo-elastic fringe photography and to finite elements is possible to observe the effects of the notches on the stresses distribution.

In Figure 58 is possible to see the effects on the stress's distribution caused by the stress concentration at the boundary of the hole. Through this chapter this theme will be addressed to present some important concepts.

The degree of concentration is measured by the elastic stress concentration factor, K_t , this factor is defined by the ratio of maximum stress or strain at a notch and the nominal stress or strain.

The σ_{nom} is the stress value considering that the stress concentration doesn't occur. Where the shape of the notch (geometrical discontinuity) has a big impact on the value of the stress concentration. The stress concentration plays a big role on the fatigue resistance of a component or structure so this factor should be addressed with care during the design.

To obtain the stress concentration factor there are two major approaches, being them:

- By calculation using analytical models or by using numerical models being the last one recommended for complex geometries.
- By measurements using strain gages or photo-elastic methods to measure these effects.

$$K_t = \frac{\sigma_{max}}{\sigma_{nom}} = \frac{\varepsilon}{e} \text{ as long as } \frac{\sigma}{\varepsilon} = E$$

Equation 51

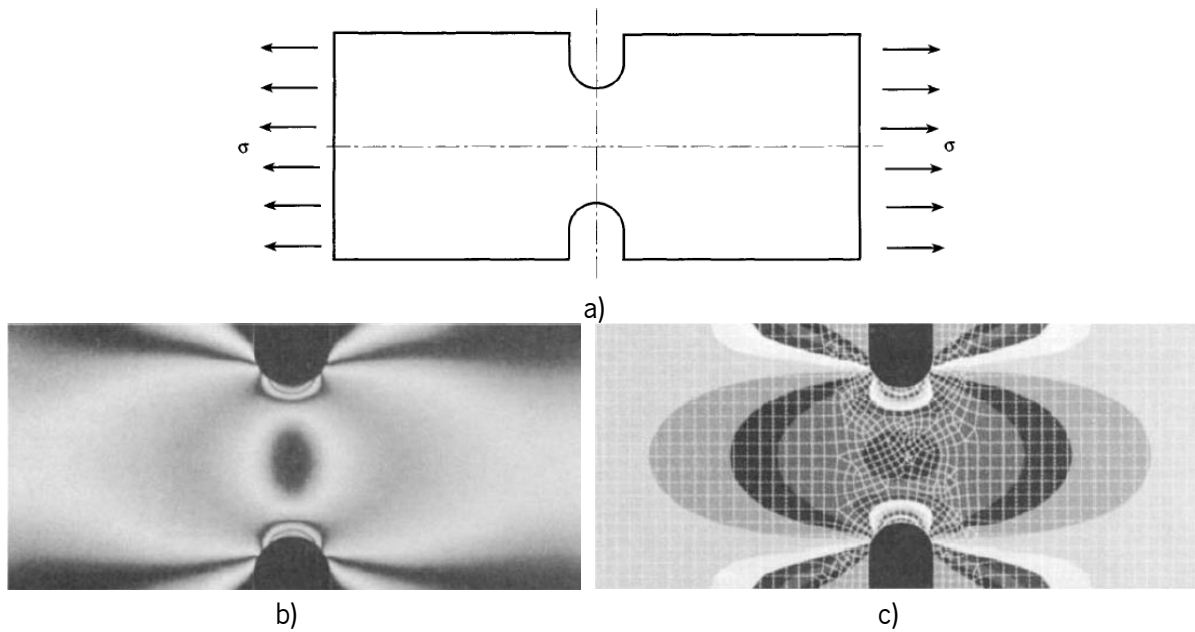


Figure 57 – Stresses distribution in a bar with symmetric notches: a) bar shape and loading direction; b) photo-elastic fringe photography; c) Stresses distribution obtained from a finite element solution [167].

On this theme there is an important concept that must be considered, which is the stress gradient, because the gradient at the notch root shows or at least gives an idea of the volume of material highly stressed. For instance, in Figure 59 is represented an example of a section of an injection mold where is present a notch, on the figure is shown the isochrone lines related to the maximum principal stress at the notch (σ_1). From where is possible to understand that the surface tangential stresses tend to decrease slowly when compared to the stresses away from the surface. If it is recalled previous statements, is stated that the crack nucleation tends to start at the surface and this phenomenon is understood because the region with higher stresses is closer to the surface. So, is possible to conclude that the gradient along the surface has a bigger importance for the decrease of the fatigue resistance.

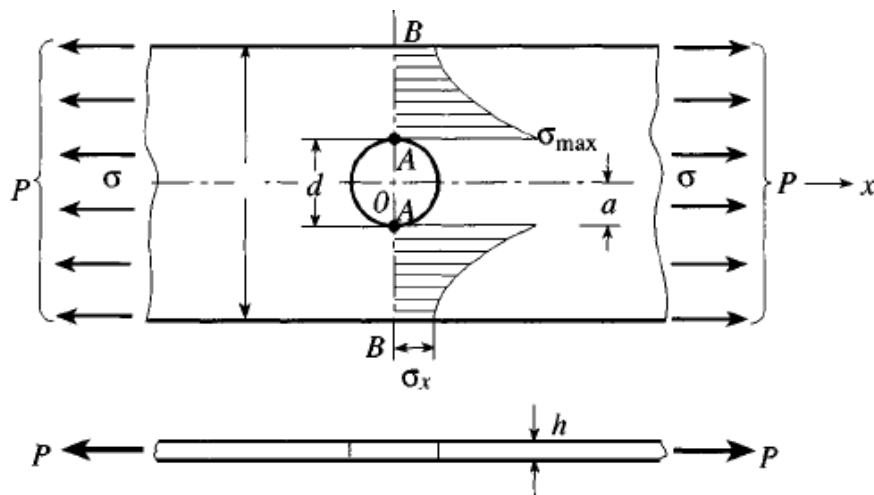


Figure 58 – Example of a bar with a centered hole under a longitudinal load with the respective stress distribution at the plane which crosses the center of the hole [167].

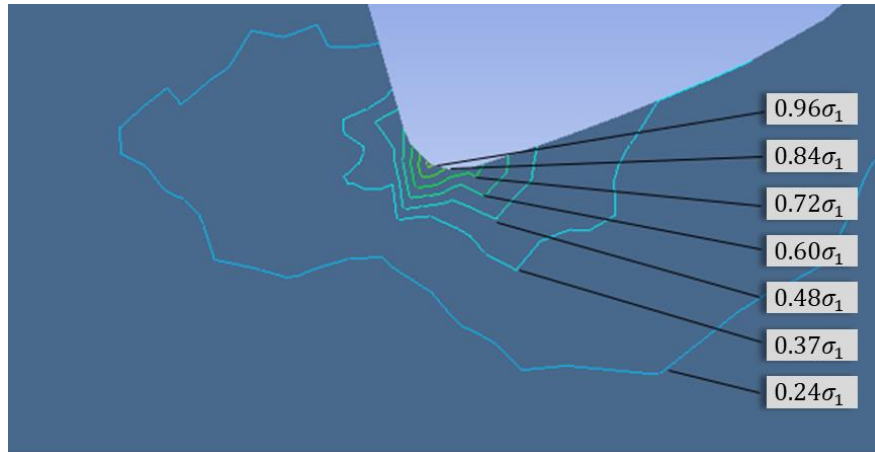


Figure 59 – Isochrone lines representing the stress concentration at a notch from a finite element model.

By the earlier conclusions is safe to affirm that the surface quality plays a big role at these regions since it can reduce even further the fatigue resistance at these locations.

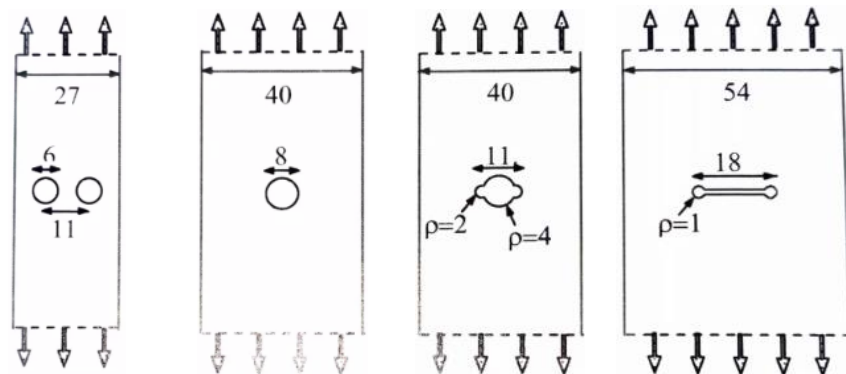
To obtain the stress concentration factors there are three major techniques which are photo-elasticity, FEM calculations and estimation. The last one can be made by the recourse of several tables like the ones that can be found on Peterson's stress concentration factor book [167]. The use of extensometers was excluded due to the difficulty of placing them on the regions more solicited.

On Table 6 is presented the K_t results for four plates with different geometric discontinuities where the stress concentration factor increases from the case 1 to 4. The values of K_t for each notch at each case are compared for the three calculation methods mentioned above, i.e., the photo-elasticity method, the finite element calculation and by estimation. By the photo-elasticity, since this is a method of measuring the stress intensity it leaves no doubts that the K_t is increasing from the plate 1 up to the plate 4. The estimation is a good method for simple geometries and specially to shapes which can be simplified to a 2D problems so, it will be used as a term of comparison from where is possible to observe that the finite element calculations can retrieve a good approximation, better than the photo-elasticity method as is possible to observe in Table 6.

Through this work the use of the finite element method approach will be used, since gathering the stress or strain values for complex models could be an impossible challenge to do by analytical methods. So, the use of numerical methods is very common and handy, and this is a good approach to obtain the needed data but at the regions where is intended to study. A relatively fine mesh must be applied to ensure a good accuracy of the results. Studies such as the one presented by Cichański [168] where is studied the mesh influence on the results, shows the problems and the capabilities of the mesh in numerical simulations. In this work was shown that mapped meshes present results with an higher precision but, the use of free mesh with an higher number of degrees of freedom can present results with a slightly lower level of accuracy. This factor is important because in complex geometries and when

the shape simplification isn't an option the use of a mapped mesh could be problematic, and the free mesh can give a better approach in terms of computation time versus results and model discretization.

Table 6 – Comparison of K_t between four different specimens obtained by three different techniques [156].



Case	1	2	3	4
K_t obtained by:				
Photo-elasticity	1.8	2.5	3.1	4.5
FE calculation	2.0	2.5	3.6	5.2
Estimation	2.17	2.51	3.47	5.36

As stated by Hodhigere, et al. [169] the use of numerical methods can present close results to the ones obtained by analytical calculations but, as shown in Table 6 when the complexity increases the numerical approach tends to give better results. As is presented on the works made by Gunwant and Rawat and the work of Parle, et al. [170], [171] the use of software's for numerical simulations gives the designer the possibility to have a visual perspective of the stress distribution. With it, the designer can, if possible, make design adjustments to improve the component, and so, create a more durable component. So, the use of numerical methods nowadays, is an indispensable tool for structural design optimization.

Of course, that experimental techniques can retrieve better results, but the cost-effectivity and time efficiency aren't on the side of this approach. So, the use of numerical methods to obtain the stress concentration factor shows to be the best approach for complex geometries but, must be done with some care.

Also, the designers must have into account that during the production process some uncertainties might appear like the change of the notch radius, which if it was possible to measure some results scatter would be expected.

3.6.5 Temperature effects (k_t)

Temperature has a major effect on the fatigue strength of metallic alloys, because there is a tendency for metallic alloys to have higher endurance limits at low temperatures. However, at extremely low

temperatures the metallic alloys become brittle and thus less likely to withstand alternating loads without reaching the brittle failure point. At very high temperatures, the strength limit of steels disappears.

When the temperatures are higher than approximately half of the melting point of the material, creep becomes an extremely important factor which makes the study of fatigue strength not feasible in terms of the structural behavior of alloy metals.

Another point that is relevant and should be considered is that high temperatures can lead to annealing of the alloy, which will lead to the elimination of residual compressive stresses, and as already discussed these stresses are beneficial in the way that they contribute to an increase in fatigue fracture toughness. But as can be seen in Figure 60, for specimens tested at room temperature and exposed to air, the fatigue strength progressively decreases with increasing temperature.

In some literature it is possible to find equations that allow the obtaining of reference values for the correction factor. For example, the FKM-Guideline [172] presents the following equations dependent on material types to obtain correction factors:

- Correction factor for aluminum alloys where $T > 50^{\circ}\text{C}$:

$$K_d = 1 - 1,2 \times 10^{-3}(T - 50)^2 \quad \text{Equation 52}$$

- Correction factor for fine-grained steel where $T > 60^{\circ}\text{C}$:

$$K_d = 1 - 10^{-3}T \quad \text{Equation 53}$$

- Correction factor for all steels except fine-grained steel where $T > 100^{\circ}\text{C}$:

$$K_d = 1 - 1,4 \times 10^{-3}(T - 100) \quad \text{Equation 54}$$

- Correction factor for steel castings where $T > 100^{\circ}\text{C}$:

$$K_d = 1 - 1,2 \times 10^{-3}(T - 100) \quad \text{Equation 55}$$

- Correction factor for ductile irons where $T > 100^{\circ}\text{C}$:

$$K_d = 1 - 1,6 (10^{-3} \times T)^2 \quad \text{Equation 56}$$

- Correction factor for malleable cast iron where $T > 50^{\circ}\text{C}$:

$$K_d = 1 - 1,3 (10^{-3} \times T)^2 \quad \text{Equation 57}$$

- Correction factor for grey cast iron where $T > 50^{\circ}\text{C}$:

$$K_d = 1 - 1 (10^{-3} \times T)^2 \quad \text{Equation 58}$$

But the use of these factors does not mean that tests under conditions closer to those of work are dispensed with, because there are several factors that are important and may not be accounted for in these equations.

3.6.6 Environmental effects

The effects of environment on reducing the fatigue strength limit may be more relevant than those of stress concentration or surface finish, for example. But just as they can be much more influential, they are also the most complex to determine, although the effect of temperature has already been referred to in section 3.6.5, it does not take into account that depending on the environment in which the component

is inserted the temperature can serve as a catalyst for the oxidation of the component which in turn will further decrease the life of the component subjected to alternating loads.

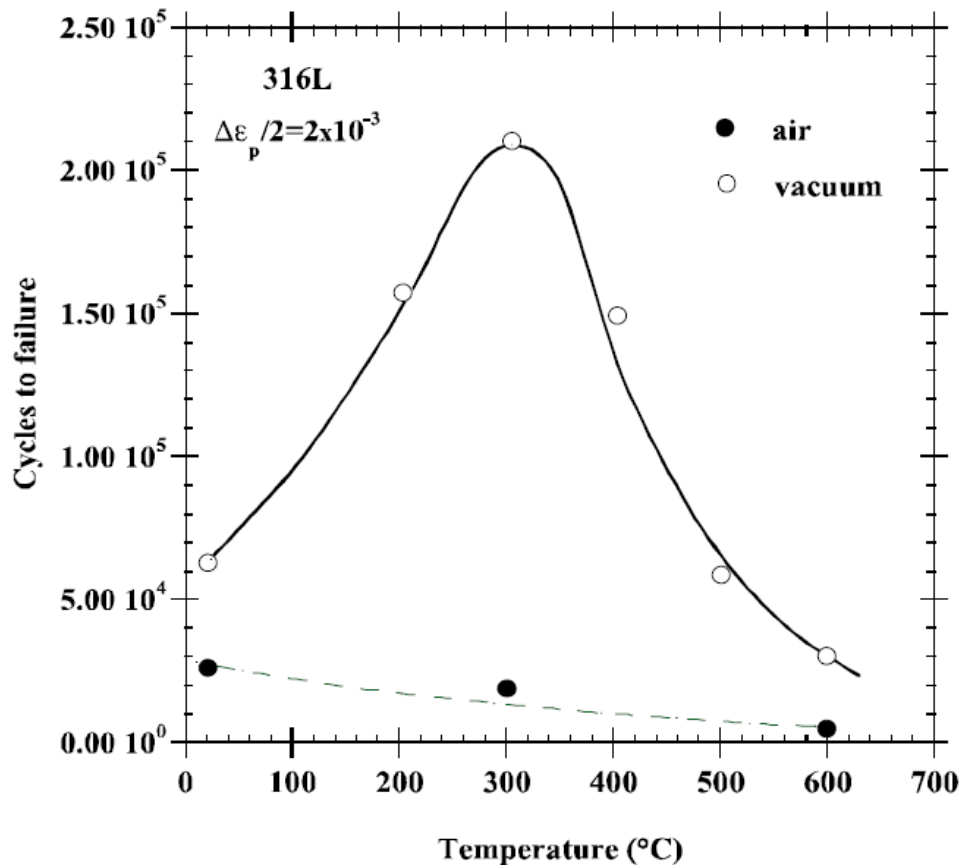


Figure 60 – Curves of lifetime of a 316L stainless steel probes tested at 1Hz under cyclic loading in air and vacuum at different temperatures [173]

Even in the study of the specimen from which the results are shown in Figure 60, a comparison is made for a specimen exposed to air and in vacuum. Where it can be seen that when it is in vacuum it behaves better than the specimen exposed to air. And it can also be seen that unlike the specimen exposed to air, it has an increase in fatigue strength until approximately 300°C, which then decreases sharply. The oxidation of the component is critical, because when it is subjected to a static load, an oxide layer is created on the surface that protects the core from the atmosphere, but if the component is subjected to alternating loads, in addition to the corrosion creating pitches on the surface, the appearance of slip bands will lead to the core no longer being protected from corrosion and thus the fatigue strength decreases significantly.

As can be seen in Table 7, where the difference between the fatigue endurance strength of two steels when exposed to air and to a corrosive environment is presented. The difference in the fatigue endurance strength of steels in corrosive environments is notorious. It can also be seen that there are steels that tend to have greater resistance to the effects of corrosion, as plain carbon steels tend to be less resistant to the effects of oxidation when compared to alloyed steels with high chromium content.

Table 7 – Comparison of the fatigue endurance strength of steels exposed to air and under a corrosive environment of complete immersion in 6,8% salt water [174] (adapted)

Material	S_{ut} [MPa]	Endurance limit [MPa]		Percent reduction
		In air	In salt water	
SAE 1050	799.79	370.94	155.82	≈58%
5% Cr steel	799.79	455.05	325.43	≈28%

There are surface treatments that can be applied in order to reduce the effects caused by oxidation such as cold rolling, plating's and paint for example. But for components that will work in oxidizing or corrosive environments, tests must be performed under these conditions to determine the reduction factor to be used to adjust the fatigue curve to these conditions.

3.6.7 Reliability factor (k_r)

When it is not possible to perform tests to determine the fatigue strength limit of a component, it is not possible to perform a statistical analysis that is able to account for the variability of the obtained results. Therefore, in order to have a factor that takes this point into account, the reliability is used, which is presented for several levels that have as assumption a normal distribution of 8%.

Table 8 shows values for the reliability factor (k_r) that range from a reliability on the order of 50% to a reliability on of 99.9999%.

Table 8 – Reliability factors for a standard deviation at 8% deviation from the strength endurance limit.

Reliability [%]	Reliability factor (k_r)
50	1
90	0,879
95	0,868
99	0,814
99,9	0,753
99,99	0,702
99,999	0,659
99,9999	0,620

3.6.8 Effect of mean stress

From what was presented up to this moment it could give the wrong idea that for an alternating stress which presents a value lower than the material fatigue limit or endurance limit it won't reach failure independent of the number of cycles. But when analyzing, for instance, the S-N curve it must be took under advisement that these values are related for a give R ratio and if the alternating stresses have a

different ratio, these values must be corrected to be able to comparison with the S-N curve.

The crack nucleation for ductile materials is caused by the shear stress for ductile materials as stated before, and the normal stresses are responsible for the crack propagation so, is possible to extrapolate that the normal mean stresses are responsible for the opening and closing of the microcracks. The opening and closing of the cracks lead to the crack propagation or retardation respectively, where the effects of the mean stresses have a higher impact for HCF or GCF regime than for LCF regime where the effects are minimum or neglectable due to the high plastic deformations. The mean stress can be represented by S_m which are defined by:

$$S_m = \frac{\sigma_1 + \sigma_3}{2} \quad \text{Equation 59}$$

Where the values of σ_1 and σ_3 are the maximum and minimum principal stresses in a stress cycle.

Since for many cyclic loaded structures the R is not equal to -1 which means that S_m is equal to 0 and usually the S-N curves are created using these relations, and as stated before the mean stress has a significant impact on the fatigue life. As is possible to observe in Figure 61, for a given material is clearly presented that for normal alternating stresses acting in compression the fatigue life tends to be higher than normal alternating stresses acting in tension so, there is a need to account for the mean stresses on the fatigue life calculations.

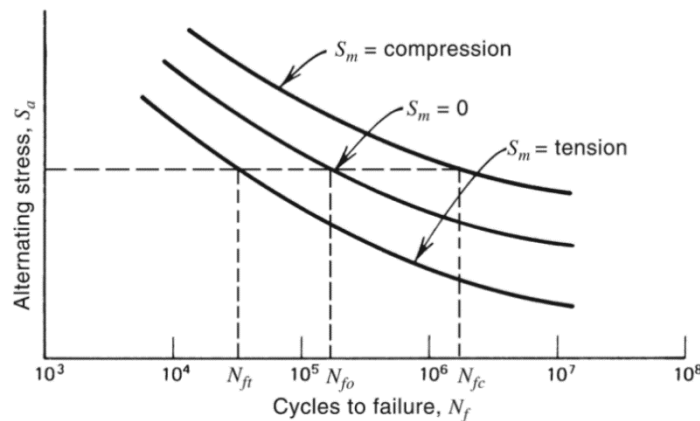


Figure 61 – Effect of mean stress on fatigue life [151].

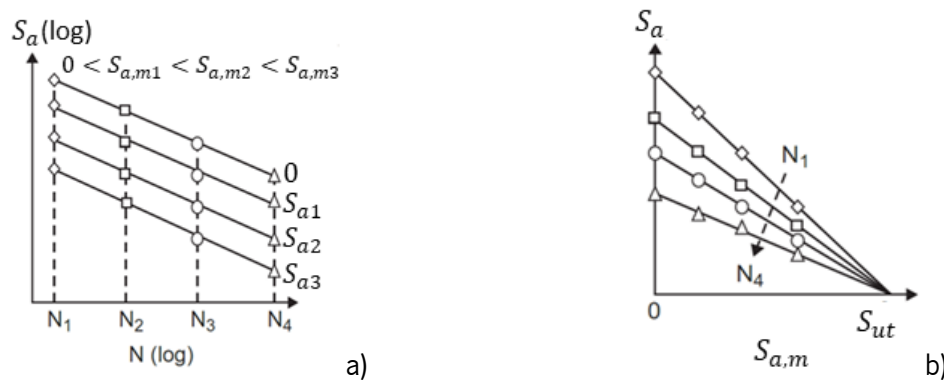


Figure 62 -Constant life plots on normal stress in S_a and S_m coordinates [166] (adapted).

Over the years there have been several works on the effects of average stresses in metallic alloys, where it has been possible to observe how this factor is linked to fatigue life. In this section we will present the considerations made for the effect of average stress on fatigue life, which will later be considered in the cumulative fatigue damage analysis.

The mean stress data are generally presented in terms of constant-life diagrams that are plots of all combination of alternating and mean stresses resulting in the same finite life to failure as shown on the Figure 63.

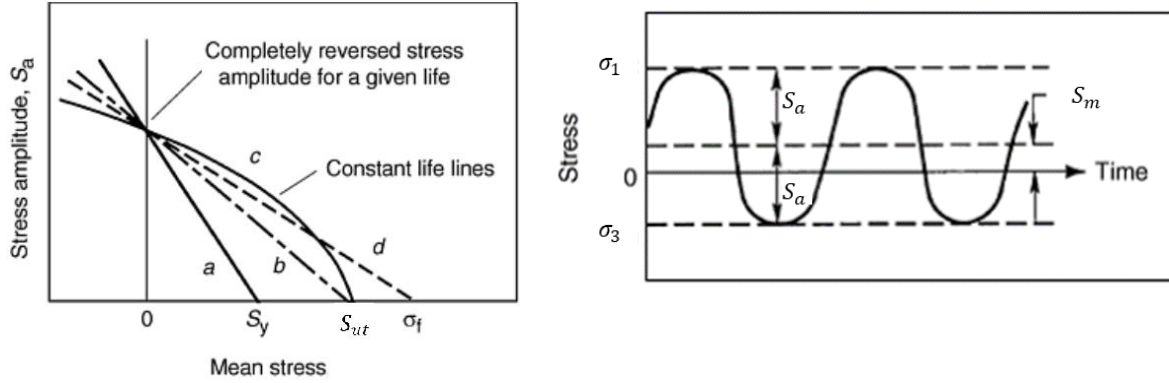


Figure 63 – various forms of presenting mean-stress data, where line “a” represents the Soderberg curve, line “b” the Goodman curve, line “c” the Gerber parabola and line “d” the Morrow curve [163] (adapted).

The straight line with dashes presented on the Figure 63 (line b) is the representation of the Goodman line which translates on the following equation:

$$\frac{S_a}{S_f} + \frac{S_m}{S_{ut}} = 1 \Leftrightarrow S_a = S_f \left(1 - \frac{S_m}{S_{ut}} \right) \quad \text{Equation 60}$$

The solid straight line represents the Soderberg equation which is represented by the Equation 61:

$$\frac{S_a}{S_f} + \frac{S_m}{S_y} = 1 \Leftrightarrow S_a = S_f \left(1 - \frac{S_m}{S_y} \right) \quad \text{Equation 61}$$

The line c is a representation of the Gerber parabola which can be obtained by the Equation 62:

$$\frac{S_a}{S_f} + \left(\frac{S_m}{S_{ut}} \right)^2 = 1 \Leftrightarrow S_a = S_f \left[1 - \left(\frac{S_m}{S_{ut}} \right)^2 \right] \quad \text{Equation 62}$$

As for the straight line with dashes represented on the Figure 63 (line d) which is known as the Morrow curve that can be obtained by the Equation 63:

$$\frac{S_a}{S_f} + \frac{S_m}{\sigma_f} = 1 \Leftrightarrow S_a = S_f \left(1 - \frac{S_m}{\sigma_f} \right) \quad \text{Equation 63}$$

Where:

S_a – stands for the alternating stress.

S_m – mean stress.

σ_f – Fatigue strength.

S_{ut} – Ultimate tensile stress.

S_y – Yield tensile strength.

S_f – completely reversed stress amplitude for a given life.

As a guideline for the case of tensile mean stress:

- The Soderberg relation is very conservative for most cases.
- The Goodman relation presents good results for brittle materials, as for ductile materials shows to be very conservative.
- As for the Gerber equation it tends to present good results for ductile materials.

Despite the previous statements, when yielding is considered as undesirable, the Soderberg method is commonly favored. As stated, the compressive loads are known to be beneficial, the Goodman or the Morrow equation can be suitably extrapolated to account for the compressive mean stress region.

The previous methods only present good results for tensile mean stresses. But other methods can be used to account for the means stresses in those cases. And even for the evaluation of tensile mean stresses there are other methods that can be used but the ones presented above are some of the most common theories.

3.7 Variable amplitude loading

The loads on components or structures usually do not have a constant amplitude. While, for example, to obtain the S-N curves the trials are performed for constant amplitude loads, which although it is a method of characterizing the materials to their fatigue strength for a given alternating load, as the loads always have the same amplitude the damage caused by each cycle is approximately equal. In the case of variable amplitude loads, such as those shown in Figure 64, it is necessary to determine the damage that each alternating cycle causes to the component.

This section will discuss the concept of damage, and how it can be determined. Another concept that will be discussed in this chapter is cycle counting. When dealing with cycles with alternating loads with variable amplitude, it is necessary to determine its amplitude to compare it with the S-N curves for example. In addition, it is also necessary to know if a certain amplitude repeats itself in the analyzed spectrum and how many times it repeats itself in order to facilitate the analysis and understand how the components will behave.

3.7.1 Cumulative damage

To account the effects of cycle variable loads is needed a method to measure the effect of each cycle, this concept is called “Damage”. This concept “measures” the damage of each cycle in the total life of the component, i.e., it defines a fraction of life or cycle ratio which represents one event or series of

events. The sum of this fractions must be equal or inferior to 1 or critical damage will occur, and it will be expected the failure by fatigue of the component. For the case of tolerant damage design where cracks are expected and are managed so they don't reach a critical size, this method can be applied but this theme won't be addressed on this work.

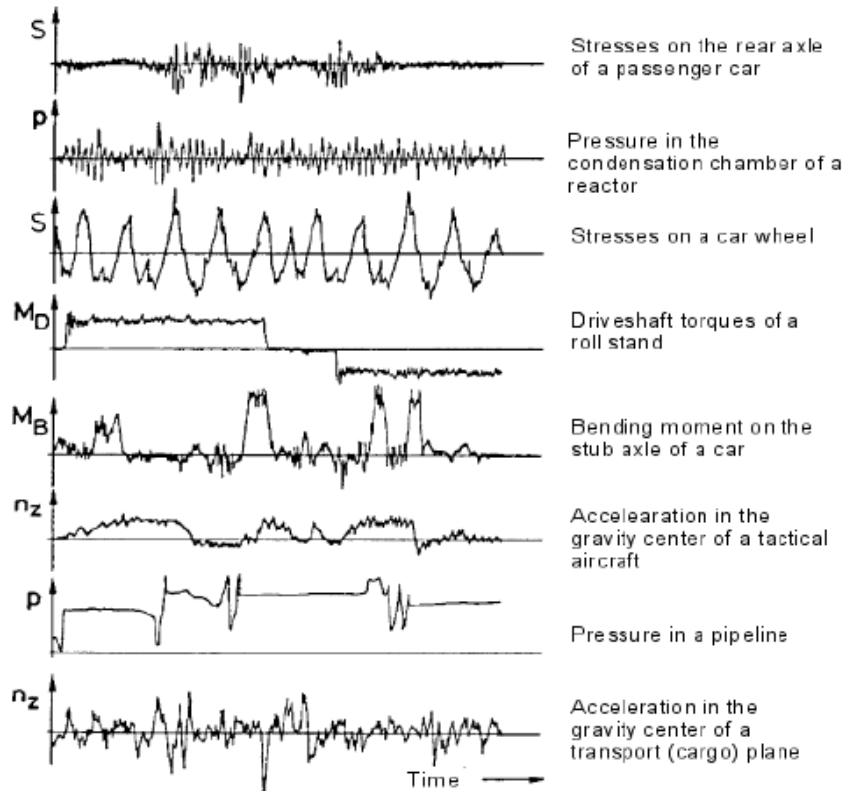


Figure 64 – Examples of variable loading spectrums [173].

Besides the life fraction, there are other ways to evaluate the cumulative damage. These methods use metallurgical parameters, mechanical parameters and physical measures. But, since the most common method to measure the damage is the life fraction or cycle ratio, so this will be the method discussed on this work.

For theories to measure the cumulative damage it will be discussed the Palmgren-Miner linear damage rule for the stress-based analysis. The Palmgren-Miner linear damage rule states that the damage caused by one cycle is defined by:

$$D_{PM} = \frac{1}{N_f} \quad \text{Equation 64}$$

Where:

N_f – is the number of repetitions of the same cycle that equals the median life to failure.

On the Figure 65 is possible to observe an amplitude loading with two blocks of constant amplitude loading with two blocks of constant amplitude stress cycles and the respective S-N curve where is represented the stress amplitudes.

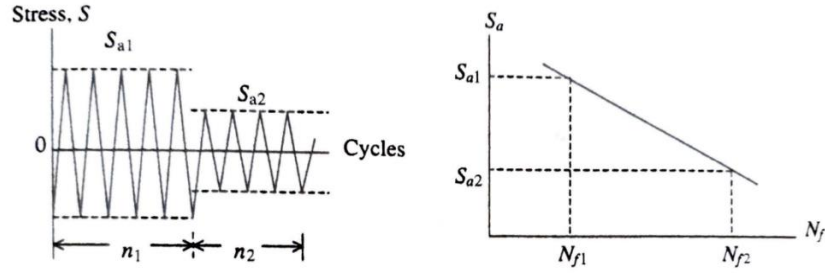


Figure 65 – Example of a block with bilevel loading sequence [151].

So, by the Figure 65 the damage effects caused by the first block are determined by $n_1.D_1 = n_1/N_{f1}$, and for the second block the damage will be determined using the same relationship so, $n_2.D_2 = n_2/N_{f2}$. If more cycles with different load amplitudes appear, the process is repeated. The Palmgren-Miner linear damage rule accounts the damage by all the variable amplitude loadings by adding all the damage ratios where their sum should be less than 1 or failure is likely to occur. So, the previous statement can be shown by Equation 65:

$$\sum \frac{n_i}{N_{fi}} = 1 \quad \text{Equation 65}$$

If it is considered the figure above and if the two blocks repeat themselves a certain number of times the following equation can be applied:

$$B_f \left[\sum \frac{n_i}{N_{fi}} \right] = 1 \quad \text{Equation 66}$$

Being B_f the number of repetitions of the loading blocks up to failure, so by using this model is possible to predict the number of blocks repetitions to reach failure.

For alternating stresses bellow the fatigue limit is considered that these cycles won't cause any damage so they are disregarded on the cumulative damage, but their effect on the residual stresses at notches might not be so neglectable as this method assumes. The Palmgren-Miner rule assumes that the fatigue damage is represented by a single damage parameter which starts at zero up to one which represents failure. But the Palmgren-Miner rule has some limitations, as it admits that alternating stresses lower than the fatigue limit stress cause zero or negligible damage, because it is based on the principle that these alternating loads do not have the ability to generate microcracks. But if a component subjected to the alternating variable loading shown in Figure 65 is analyzed in which the alternating stress S_{a2} is below the fatigue limit stress but where the alternating stress S_{a1} has already caused microcracking, the alternating stress S_{a2} is capable of leading to crack propagation and thus to fatigue failure.

In the example shown in Figure 66 the number of cycles to failure for two types of alternating load, one with a flat spectrum and one with a steep spectrum, is presented. In both loads the alternating

stresses start above the fatigue limit and decrease to a value below the fatigue limit. In both spectrums rupture is reached, but in the case of the flat spectrum it is verified that when it is under alternating loads below the fatigue limit stress it has a lower durability. With this it can be concluded that loads below the fatigue limit stress can contribute to crack propagation but the time to failure is strongly conditioned by the loading history.

The damage caused by each load cycle is directly related to the value of the applied alternated stress but is also related to the damage caused by previous cycles and this interaction is a complex process which makes difficult to reach into a method with high reliability for predicting fatigue life when variable amplitude loading is present. But, at the moment the non-linear damage methods did not present any significant changes in terms of reliability for the exception of some loading cases [175], [176]. So, without a more reliable model the Palmgren-Miner rule still one of the most used methods to determine the damage parameter for fatigue life and have the advantage of covering practically all kinds of structural elements [136].

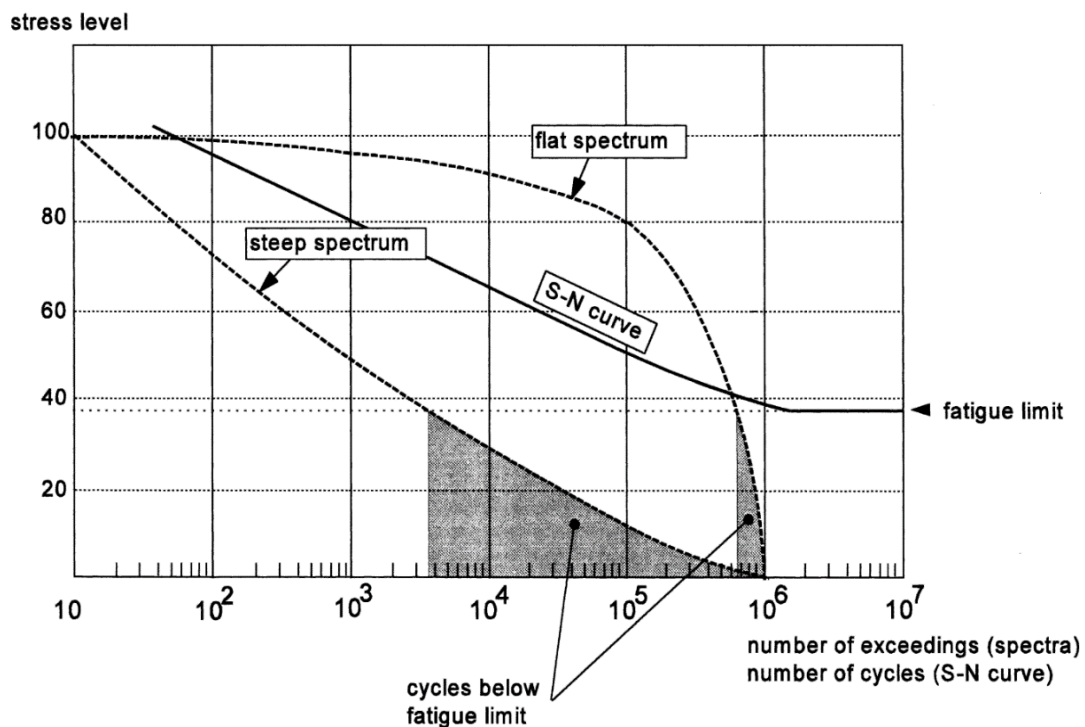


Figure 66 – Influence of the load spectrum on the fatigue limit [177].

3.7.2 Cycle counting methods

When faced with variable amplitude loading, it can be extremely complex to perform a fatigue analysis of the components. Therefore, in order to be able to compare with S–N or ϵ –N curves, it is necessary to be able to convert the variable amplitudes to constant ones in order to make the comparison possible. So, the process of changing from a history of variable loadings to a number of loadings of constant amplitude is what is called the cycle counting method.

For this there are different methods, methods that can also lead to different results because of the way in which the cycle counting is carried out varies depending on the method, that can lead to differences between the amplitudes of the cycles obtained from the load history. In this section three cycle counting methods will be presented so that some of their differences can be demonstrated.

The **peak counting method** will be the first to be addressed, in this method the reference load level is the first step in starting to cycle count the load history. From there the maximum and minimum peak values are determined as shown in Figure 67 a).

The cycle that causes the most damage is determined by joining the peak with the highest value and the peak with the lowest value, thus obtaining the pair that corresponds to the cycle with the greatest amplitude. This process is repeated until the pairs of peaks previously determined are exhausted obtaining a result as presented in Figure 67 b).

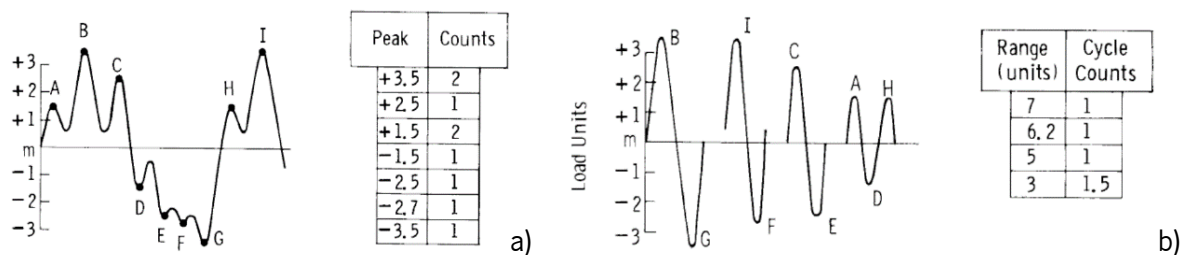


Figure 67 – Peak counting method where a) is the load cycling history and the peak count table and b) is the resulting cycling counting [178].

As for the **level crossing method**, in this method the axis of stresses or strains is divided into a certain number of increments, where the smaller the increment the higher the resolution of the method.

Then each time a portion of the curve with the loading history crosses a level is counted, where, as can be seen in Figure 68 a) there is an intermediate load value that should be chosen at the very beginning. And, for value increments greater than the intermediate point m is counted each time the curve passes a level in the upward direction and the opposite for increments lower than m . Once the counting of the intersections between the loading curve and the increment levels is done, the combination is done in order to obtain complete cycles. The combination process is done by combining the levels that were most often crossed above the value of m with those that were most often crossed below the value of m , and so on for the remaining cycles as shown in Figure 68 b).

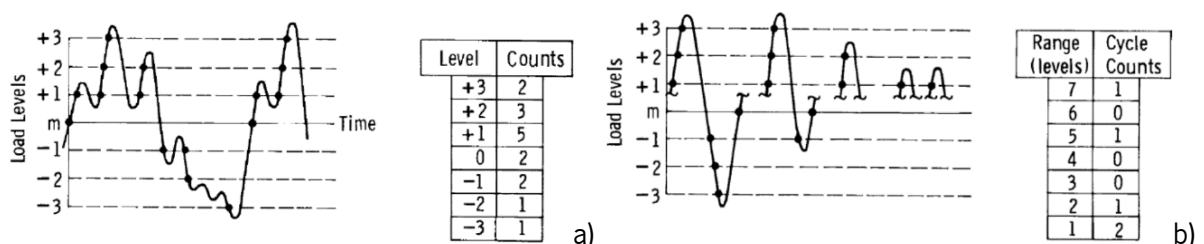


Figure 68 – Level crossing counting method where a) is the load cycling history with the levels and the crossing counting table and b) is the resulting cycling counting [178].

Finally, the **Rainflow counting method** is one of the most used methods for either load-time, stress-time or strain-time cycle counting. This method got its name from its comparison to the rainwater running off the roofs of a "pagoda", which falls from the edge of each roof onto the roof below. This method has the following operation:

- First it is turned the loading history 90° so that the time axis is pointing downward as shown in Figure 69.
- The half cycle is set that allows each Rainflow to fall on the next roofs until:
 - Fall into a section corresponding to a higher amplitude.
 - Find a section that has already been found by a previous fall.
 - Do not fall on any section.

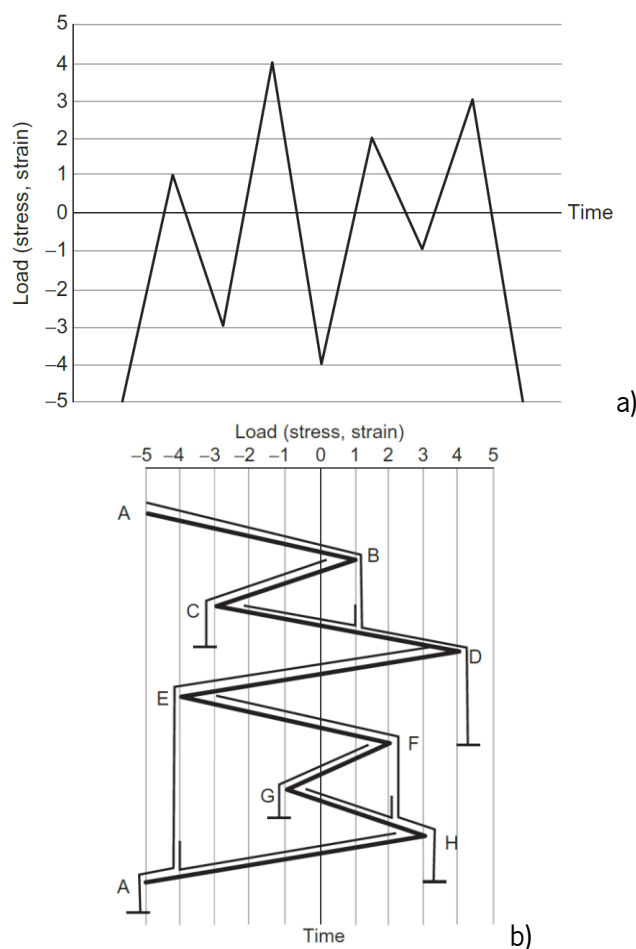


Figure 69 – Rainflow counting method where a) is the load cycling history and b) is the Rainflow counting [166].

Given the loading history presented in Figure 69, the number of reversals determined according to the Rainflow counting method is presented in Table 9, while the cycles obtained by the method are presented in Table 10.

Given what has been presented for cycle counting methods it is possible to get an idea of the differences that exist between them. For example, in the work done by R. Stelzer, et al [178] where they

study the differences between different cycle counting methods, they find that the peak counting method tends to present cycles with higher amplitudes. But the Rainflow counting method brings the advantage of being able to take into account the sequence in which alternating loads are applied, which does not happen with the other two methods presented in this work.

Table 9 – Reversal counts using the Rainflow counting method [166].

Number of reversals	From	To	From	To	Range	Mean
1	A	D	-5	4	9	-0,5
1	B	A	4	-5	9	-0,5
1	B	C	1	-3	4	-1
1	C	B	-3	1	4	-1
1	E	H	-4	3	7	-0,5
1	H	E	3	-4	7	-0,5
1	F	G	4	-1	3	0,5
1	G	F	-1	2	3	0,5

Table 10 – Cycles obtained from the Rainflow counting method [166].

Number of cycles	From	To	From	To	Range	Mean
1	A	D	-5	4	9	-0,5
1	B	C	1	-3	4	-1
1	E	H	-4	3	7	-0,5
1	F	G	2	-1	3	0,5

3.8 Multiaxial stresses

So far, models have been described for the creation of curves that characterize the behavior of metallic alloys subjected to alternating loads, ways to adjust the curve to the various factors that can influence the fatigue life of a component, methods for counting cycles in the case of loads with varying amplitude cycles, and how to consider the damage done by each alternating cycle in order to be able to determine the fatigue life of a component. But the most common application in industrial terms is in a situation where components are subjected to multiaxial loading, and the curves that are obtained for the fatigue behavior of materials are for non-multiaxial loading. Therefore, in this section we will present models used to be able to contemplate multi-axial loading by taking into account the S-N or ε -N curves that are obtained from specimen tests or obtained from empirical models.

For the present work it will be accepted that there are two methods for making fatigue predictions, the first being the scientific method where:

- The sequence and magnitude of the loads can be accurately determined so the entire sequence of loads is well defined.
- The parameters such as material characteristic curves, stress concentrations, and other factors influencing fatigue life are known.

- The specimen has a tendentially simple geometry where all critical regions are known right from the start.
- The nominal stresses and stress concentrations in the critical regions are known.
- The predictions made are only valid for the batch of material tested.
- Since scientific predictions are generally made to understand the limitations and capabilities of different methods and methodologies, the economics are not relevant even when the predictions are not as reliable as expected.
- The method should be able to be used to make predictions for a number of materials, loading sequences, and geometries with reliable results.

The second method is fatigue prediction by the industrial method, which is significantly different from the scientific method in several respects:

- Usually not all parameters are known, from the loading spectrum to factors that can influence the fatigue life of components.
- Often the fracture locations, the fracture mode as well as the initial fracture size and shape are unknown.
- Determining the critical cross sections and obtaining the corresponding nominal stresses is difficult even with numerical models such as the finite element method.
- It is usually necessary to use high reliability factors for the prediction of the fatigue strength of the components.
- When models cannot be tested due to economic reasons, specimens are often made to perform the tests where it is necessary to scale the results to the real model where size effects for example have to be taken into account as well as other effects depending on the case.
- Predictions for fatigue strength must be made quickly and economically.
- Validation with a full-scale model may not be performed, depending on the specifics of the industry.
- One of the major problems of the industrial method is the potentially catastrophic consequences if the designer does not correctly assume all parameters that can lead to fatigue failures.

In order to consider the models that can be used, it is necessary to take into account some concepts. One of them is proportional and non-proportional loads. These two types of loading have a great influence on the models that can be used for fatigue calculations for multiaxial loading, and the difference between them lies in the way the normal stresses and shear stresses behave over the loading cycle.

For proportional loading, the normal stresses and shear stresses are in phase, so the instant when the peak normal stress and shear stress are reached occur simultaneously, as can be seen in Figure 70.

For non-proportional loading, unlike proportional loading, the peak normal stress and shear stress occur at different times, as shown in Figure 71.

Fatigue calculation methods for multiaxial stress loading can be classified as stress-based, strain-based, and energy-based approaches, or also by fracture mechanics approach when a damage-tolerant design is desired.

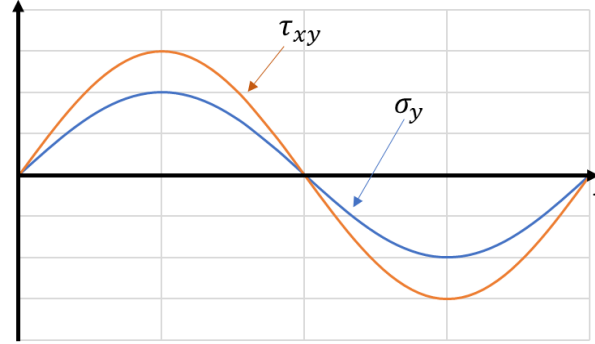


Figure 70 – Example of a proportional loading with the normal and shear stresses in-phase.

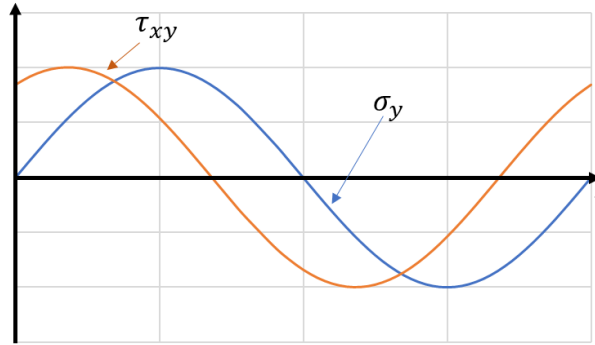


Figure 71 – Example of a non-proportional loading with the normal and shear stresses out-of-phase.

3.8.1 Stress-based criteria

For the stress-based criteria two models will be presented, which will be the Octahedral shear stress theory and the Sines Method.

3.8.1.1 Octahedral shear stress theory

For octahedral shear stress theory, an amplitude of the nominal equivalent stress (S_{qm}) is determined, which can be obtained from the Equation 67.

$$S_{qm} = \frac{1}{\sqrt{2}} \sqrt{(\sigma_{a1} - \sigma_{a2})^2 + (\sigma_{a2} - \sigma_{a3})^2 + (\sigma_{a3} - \sigma_{a1})^2} \quad \text{Equation 67}$$

Where σ_{a1} , σ_{a2} and σ_{a3} represent the alternated main stresses and by convention $\sigma_{a1} > \sigma_{a2} > \sigma_{a3}$.

By determining the S_{qm} value, the multi-axial alternating stresses are converted in equivalent uniaxial alternating stresses so it is possible to perform the comparison with the S-N curves of the materials obtained from uniaxial alternating loads. This method is valid for ductile materials, for brittle materials the principal stress method is more appropriate, where $S_{qm} = \sigma_{a1}$.

The octahedral shear stress theory has the drawback of being insensitive to hydrostatic stresses, and if the nominal equivalent stress is zero, which can happen in alternating hydrostatic loading, the damage caused by the loading will be disregarded by this formulation. For these cases it is advisable to use the Equation 68.

$$S_{qm} = \underbrace{\sigma_{m1} + \sigma_{m2} + \sigma_{m3}}_{\text{Sum of the mean nominal stresses}} = \sigma_{mx} + \sigma_{my} + \sigma_{mz} \quad \text{Equation 68}$$

This method not only better represents S_{qm} for hydrostatic loads, but also has the advantage that it can assume negative values unlike the Equation 67, which leads to a better representation of compressive stresses.

Once the equivalent nominal stress has been determined, the expected fatigue life can be determined from the formulations for uniaxial loading.

The advantage of this method lies in its simplicity and not requiring more material parameters than those obtained from the monotonic and S-N curve. However, this method should only be used for proportional loading conditions.

3.8.1.2 Sines method

The Sines method, which is based on the octahedral shear stress method, but in this method the average hydrostatic stresses are also taken into account, and is represented mathematically by the Equation 69:

$$\sqrt{2}S_{Nf} = \sqrt{(\sigma_{a1} - \sigma_{a2})^2 + (\sigma_{a2} - \sigma_{a3})^2 + (\sigma_{a3} - \sigma_{a1})^2} + m_s(\sigma_{m1} + \sigma_{m2} + \sigma_{m3}) \quad \text{Equation 69}$$

Where,

- m_s , is the coefficient of mean stress influence.
- S_{Nf} , is the uniaxial stress influence.

When the value of m_s is not known for the region under study it can be assumed the value of 0,5. The value of m is determined experimentally from tests from which the fatigue strength is obtained but for an R different than -1.

This method has the advantage that no additional calculation is required to determine the expected fatigue life as in the octahedral shear stress theory. But as with the octahedral shear stress method it should not be used for non-proportional loading and has the disadvantage of having a parameter that requires further experimental testing of the material.

3.8.2 Strain-based approach

As with the stress-based approach, the strain-based approach has similar calculation methodologies,

one of them being octahedral shear strain theory which can be described by the Equation 70.

$$\varepsilon_{qa} = \frac{\sqrt{(\varepsilon_{a1} - \varepsilon_{a2})^2 + (\varepsilon_{a2} - \varepsilon_{a3})^2 + (\varepsilon_{a3} - \varepsilon_{a1})^2}}{\sqrt{2}(1 + \nu)} \quad \text{Equation 70}$$

Where $\varepsilon_{a i}$ are the principal alternating strains, with $\varepsilon_{a 1} > \varepsilon_{a 2} > \varepsilon_{a 3}$.

The equivalent strain determined by the equation above can be used on the ε -N curve equation to obtain the supported number of cycles. As the equivalent stress approach, this method isn't suitable for non-proportional multiaxial loading.

The energy strain-based approaches use the products of strain to quantify the fatigue damage, where some methods have been presented to quantify the multiaxial fatigue damage. One of them is the plastic work per cycle as the parameter for crack nucleation, where this plastic work is determined by calculating the integration of the products of the stresses and plastic strain increments for each of the six components of stress (the sum of the six integrals is the plastic work for each cycle).

But the determination of the hysteresis loops require a special attention to the considerations made for the hardening rules and flow rules, especially when non-proportions loads are present since the shape of the hysteresis loops are very dependent of the loadings.

The application of this models to high-cycle fatigue is difficult due to the low plastic deformations but, for these situations some models have been present such as the ones presented by Ellyin, et. al., and Park and Nelson [179].

The use of this methods for non-proportional loading is not recommended but, if used, is needed to have under consideration that this method does not reflect fatigue damage nucleation and growth observed on specific planes.

3.8.3 Critical plane approach

Through the observation of experiments was possible to conclude that crack nucleation and growth by specific planes which are the maximum tensile plane or stress planes and maximum shear plane.

The models that correlate the cumulative damage with the stress or strain are called by "critical plane models", where these models could be an asset to predict the orientation of the crack and failure plane.

For high cycle fatigue, Findley developed a models based on the stress quantities, where it was identified the cycle shear stress and the normal stress at the shear stress plane as the parameters governing the fatigue damage.

$$\frac{\Delta\tau}{2} + K\sigma_n = C_1 \quad \text{Equation 71}$$

Where:

$\frac{\Delta\tau}{2}$, is the shear stress amplitude.

σ_n , is the normal stress acting on the $\Delta\tau$ plane.

K and C_1 are constants.

By these models crack nucleation occurs on the plane where is present the largest value of $\frac{\Delta\tau}{2} + K\sigma_n$. Brown and Miller formulated an approach in terms of strains, where a simple formulation for this approach is:

$$\frac{\Delta\gamma_{max}}{2} + S\Delta\epsilon_n = C_1 \quad \text{Equation 72}$$

Where:

$\frac{\Delta\gamma_{max}}{2}$, is the maximum shear strain amplitude.

$\Delta\epsilon_n$, is the normal strain range on the $\Delta\gamma_{max}$ plane.

S is the material dependent constant.

The critical plane approach attempts to describe the physical nature of the fatigue damage, where the formulation based on the physical interpretation of fatigue damage is called Fatemi-Socie model.

The Fatemi-Socie model uses the maximum shear strain amplitude and the maximum normal stress which act on the maximum shear strain plane to analyze the fatigue damage for each cycle. The cracks usually present an irregular shape at a microscopic level, since they grow through the materials grain structures and this will result on frictional forces between the cracks surfaces during the loading cycles, which will work in favor of the material life when subjected to shear stress. Because during the cyclic loads the crack tip will divide the applied force, and this will work in favor of the fatigue life. But, during this process the tensile stresses are perpendicular to the crack plane, if they exist, they will separate the crack surfaces and with it the friction forces will decrease and consequently the fatigue life will be reduced. So, having this phenomenon's into account the Fatemi-Socie model is expressed by the Equation 73:

$$\frac{\Delta\gamma_{max}}{2} \left(1 + K \frac{\sigma_{n,max}}{S_y} \right) = C_1 \quad \text{Equation 73}$$

Where:

S_y , is the material yield strength.

K , material constant.

The value of K can be obtained by fitting the fatigue data from a uniaxial test to fatigue data from torsion tests. If fatigue data is not available, the value of K can be assumed to be equal to 1. The mean stresses or the residual stresses are accounted by these models since the maximum normal stresses are represented by Equation 74:

$$\sigma_{n,max} = \sigma_{n,a} + \sigma_{n,m} \quad \text{Equation 74}$$

Where:

$\sigma_{n,a}$, is the alternated normal stress.

$\sigma_{n,m}$, is the residual normal stresses.

Here the effects of the material hardening are accounted by the maximum normal stress term which results from the non-proportional loading, this happens because the hardening will result on the increase of the alternating normal stresses.

Since for most cases under study there is some difficulty in obtaining all the needed properties to fill in the equation of the Fatemi-Socie model, so it can be expressed by:

$$\begin{aligned} \frac{\Delta\gamma_{max}}{2} \left(1 + K \frac{\sigma_{n,max}}{S_y} \right) \\ = \left[(1 + \nu_e) \frac{\sigma'_f}{E} (2N_f)^b + (1 + \nu_p) \varepsilon'_f (2N_f)^c \right] \times \left[1 + K \frac{\sigma'_f}{2S_y} (2N_f)^b \right] \end{aligned} \quad \text{Equation 75}$$

Where:

ν_e , is the elastic Poisson ratio.

ν_p , is the plastic Poisson ratio.

Once the equation above is calculated for multiaxial loading (or the left side term) the fatigue life can be estimated. The Fatemi-Socie model assumes that cyclic shear strain must exist to occur fatigue damage, so this model is suitable for most metals since most of the energy is dissipated by shear on the maximum shear plane where the crack will nucleate. The damage model of Smith-Watson-Topper is suitable for the analysis since the maximum principal strain amplitude, the maximum normal stress and the maximum principal strain amplitude are accounted.

For this model the critical plane is where the value of $\varepsilon_{a1}\sigma_{n,max}$ are higher. Like the Fatemi-Socie model, the account of the cyclic hardening is made by the maximum normal stress term.

Complex loading could result in non-proportional loading type in which it is difficult or impossible to determine the plane with maximum shear stress/strain amplitude, but some works have been performed to overcome these difficulties such as the one presented by Kumarswamy Karpanan [180].

For most ductile materials, fatigue crack tends to initiate on the plane with maximum shear stress amplitude and at the free surface. Therefore, for non-proportional loading case, stress or strained-based critical plane search methods (CPS) are commonly used for component fatigue analysis. In the CPS method, all possible planes at a point are analysed for the maximum shear stress range.

For proportional loadings the fatigue cracks tend to initiate on one of the shear stress planes. Additionally, the mean stress on these planes is also calculated to account for the mean stress effects

on the fatigue stress amplitude.

The critical plane search method was first developed by Findley [181]. By this method the effects of shear stress and maximum normal stresses are considered on the maximum shear stress plane to identify the critical plane. Where it was shown that the combined action of shear and normal stress is responsible for fatigue damage as is possible to observe by the Equation 76:

$$\left(\frac{\Delta\tau}{2} + k\sigma_n\right)_{max} = f \quad \text{Equation 76}$$

Where:

$\Delta\tau/2$, is the shear stress amplitude.

k , is the material constant.

σ_n , is the normal stress amplitude.

Another proposed method is the **McDiarmid method** where is defined the critical plane as the plane where the maximum shear stress amplitude is reached, this is a logic step since for ductile metals this is the plane where the cracks tend to nucleate, and this method can be expressed by:

$$\frac{\Delta\tau_{max}}{2t_{A,B}} + \frac{\sigma_{n,max}}{2 \cdot S_{ut}} = 1 \quad \text{Equation 77}$$

Where:

$\Delta\tau_{max}$, is the maximum shear stress range.

$\sigma_{n,max}$, is the normal stress on the maximum shear stress range.

S_{ut} , is the ultimate stress.

The **Fatemi-Socie model** uses the combination of shear strain and normal stress to search for the critical plane as is possible to observe on the Equation 78:

$$\frac{\Delta\gamma_{max}}{2} \left(1 + \frac{\sigma_{N,max}}{S_y}\right) = \frac{\tau'_f}{G} (2N_f)^{bf} + \gamma'_f (2N_f)^{cf} \quad \text{Equation 78}$$

Where:

$\Delta\gamma_{max}/2$, is the maximum shear strain amplitude.

S_y , is the Yield stress.

τ'_f , is the shear fatigue strength coefficient $\approx \sigma'_f/\sqrt{3}$.

G , is the shear modulus.

$2N_f$, is the number of reversals to failure.

bf , is the fatigue strength exponent.

cf , is the fatigue ductility exponent.

σ'_f , fatigue strength coefficient.

As can be seen, this method is able to the fatigue strength in multiaxial loading situations with a good resolution, but it needs much more information about the materials that will be studied. For situations in which there are non-proportional loads it is an advisable method because it is able to deal with this factor. Although the calculations can be complex when the non-proportionality of the load is present in the three main directions and respective shear planes.

It also brings the advantage of determining the critical plane, this together with the location of the least fatigue resistant region can lead to act in the design phase with more data on that same region and improve the design in order to minimize the damage caused by alternating loads in that same plane.

4 Computer applications and methodology to conduct the fatigue analysis

The objective of this work is to develop an algorithm/procedure to perform the fatigue calculation of injection tools using numerical methods. To do so two software's are used where one is used to study the injection of the plastic part and the other is used to study the structural behavior of the injection tool. But, to perform an accurate structural simulation is needed data with rigor. So, through this chapter will be presented applications to generate the material fatigue properties. Other application that will be responsible to pick up the data from Moldflow Insight (rheology numerical simulation software) and arrange it so it can be understood by ANSYS Mechanical (structural numerical simulation software).

Therefore, this section will be divided into four parts. The first one is related to the presentation of an application that using the empirical models presented in section 3 is able to generate the necessary data for the construction of S-N and ε -N curves so that these can be incorporated in commercial finite element software that have the ability to perform fatigue calculations. With this it is intended to have one more tool to help the designer at a preliminary stage of the project to have a quick perspective of the points that may be critical in terms of fatigue resistance. Also in this part, experimental results will be presented in order to demonstrate the choices made for the used empirical model.

The second part is related to fundamental application for this work. Because it will be presented an applications to pass the loads that are calculated in Moldflow during the rheological simulation to the structural numerical simulation software.

The third part is the presentation of the application that encompasses the research done and that will be responsible for the fatigue resistance calculations of injection molds will be presented. Studies performed with this application will be demonstrated later in this work, so that the reader of this work can understand how this application will generate relevant results for the design phase.

Finally, a proposal for the design of a mold incorporating the numerical and fatigue studies will be presented. This is intended to begin to standardize the design work to optimize resources and obtain comparable information.

4.1 Application for fatigue data creation

The basis of this application is the empirical models presented in the chapters 3.3 and 3.4. From this application made in Python will be generated the needed data to create the S-N and ε -N curves. The S-N curve will be done from the hardness method (model 2) shown in the section 3.3.5 and validated in section 4.1.1. As for the ε -N curve, it was chosen an empirical model which presents conservative values

like the model for the S-N curve and also can be generated by the steel monotonic data. The generation of the ε -N curve is made so it can be also an input on the structural simulation software to help in the analysis of the critical locations.

To construct this application the algorithm presented in Figure 72 was employed, from which is possible to generate the S-N curve and ε -N curve from empirical models which have been studied on the chapters 3.3 and 3.4.

For the S-N curve it was chosen the model 2 as will be presented in the chapter 4.1.1.2 as the most conservative model. Since this work is made for injection tools, where there is little room to test the tools to its fatigue endurance due to the high cost and the low time to market. But the use of these models could lead to a better understanding of the tool's limits with a safe margin of error.

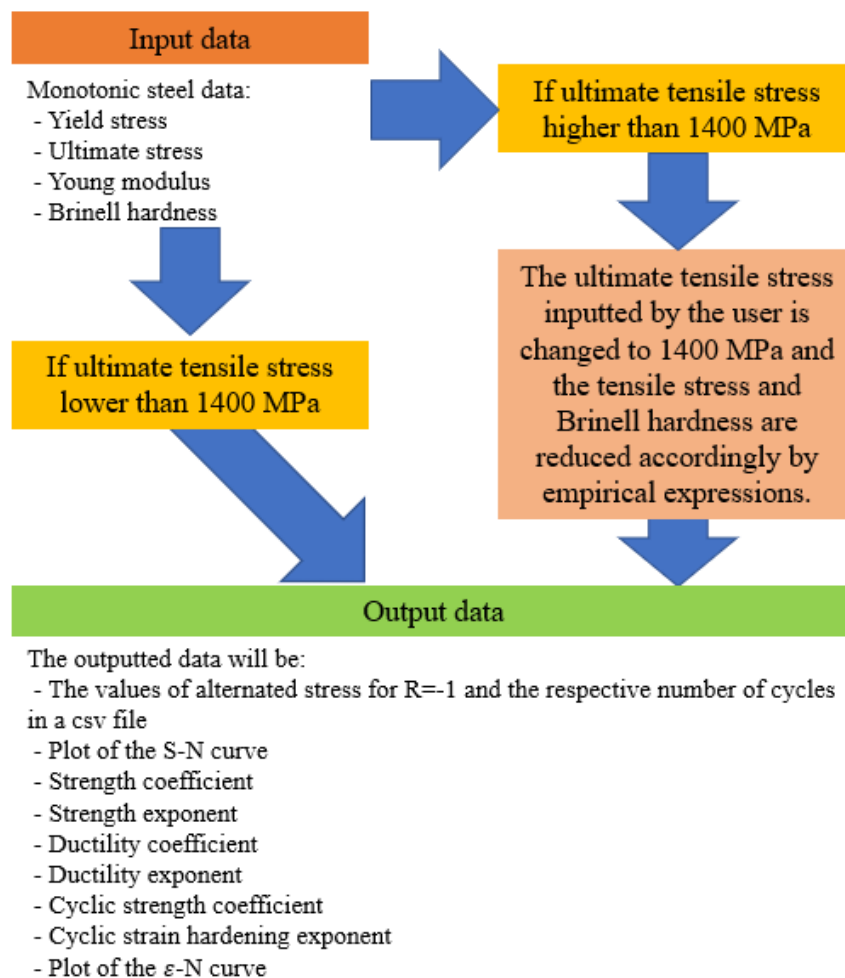


Figure 72 – Algorithm of the data creation application.

As for the ε -N curve it was chosen the Roessle and Fatemi method, it was chosen due to the needed parameters which are very similar to the ones used to generate the S-N curve and as is possible to observe on the Figure 73 where is presented a comparison between the Model 1, model 2 and the data based on the Roessle and Fatemi method, where from the applied stress was determined the equivalent strain and from it was determined the predicted number of cycle to failure by the method.

From the Figure 73 is possible to understand that for higher stress amplitudes the ε -N model shows less conservative values but, as stated before these models are better suited for low cycle fatigue. When is analyzed the curves at the region of 1×10^6 cycles, the ε -N curve shows to be more conservative than the S-N curves, that comes in line with the literature that shows that the S-N curves are better suited for the high and giga-cycle regions. The reason to have this model present on this application and this work is the need of the parameters for certain multiaxial stresses methodologies to determine the fatigue life of parts subjected to multiaxial stresses.

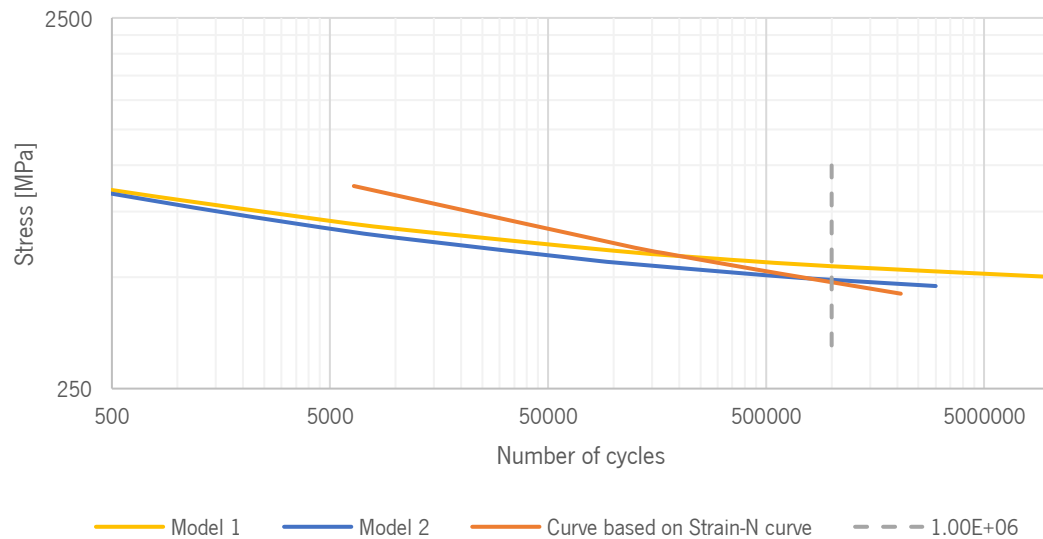


Figure 73 – Comparison between the S-N models and the Roessle and Fatemi ε -N model, using the equivalent stress as the term of comparison.

With the considerations made above, the application was made using the Python coding language from which was created the interface presented in the Figure 74.

The application window titled "Steel fatigue properties" includes the following elements:

- Instructions:** "The following fatigue data is valid for a probe as displayed on the image bellow. All values are obtained by empirical models only valid for steels with an ultimate strength lower than 1400 MPa."
- Input Fields:**
 - Yield stress (MPa):
 - Ultimate stress (MPa):
 - Young modulus (GPa):
 - Brinell Hardness:
- File Handling:**
 - A "Directory" button.
 - A label: "Write the file with the file extension .txt or .csv."
 - A "Type file name:" input field.
 - "Run" and "Quit" buttons.
- Technical Drawing:** A diagram of a tensile specimen with dimensions: total length 160, grip diameter $\varnothing 12 (+0.03, +0.03)$, gauge length 50, gauge diameter $\varnothing 24$, and base diameter $\varnothing 30$. It also shows fillet radii R1 and R32.2.
- Disclaimer:** "This APP can't be commercialized. The obtained data should always be validated by experimental trials."

Figure 74 - Application interface window to input the steel monotonic data.

The application will generate the data for steels with an ultimate tensile stress equal or lower than 1400 MPa, if the steel presents properties where the ultimate tensile strength is higher than 1400 MPa the application automatically changes the values, for the S-N fatigue curve generation, of the Brinell hardness and ultimate strength for HB=411 and $S_{ut}=1400$ MPa. This simplification is made having under account that the fatigue endurance limit doesn't increase proportionally for ultimate tensile strength higher than 1400 MPa as it does for lower values, in the Figure 75 is possible to observe that for values higher than the imposed limit the tested steels present a higher scatter of values which makes the conservative imposed limit an good approach for the design process of injection tools.

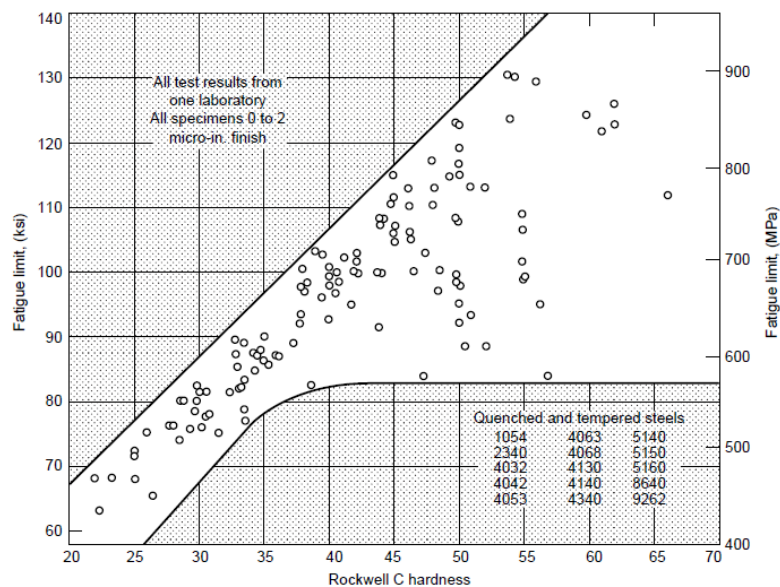


Figure 75 – Relationship between the steel hardness and the fatigue endurance limit [182].

This application generates a .csv or .txt file, depending on the user preference for the S-N values so it can be imported to the structural software such as ANSYS as shown in the Figure 76.

With the use of this application is possible to get the needed information for the fatigue curves to feed the numerical simulation software so it can produce the fatigue life information so the designers can work on the solution to improve the component if needed.

In order to validate the empirical models used to create the S-N curves, experimental tests were performed on a steel common in injection molding. With this we intend to present the reasons for the empirical model chosen for the application and demonstrate how with this application we intend to present values that are conservative taking into account the specificities of this industry.

4.1.1 Empirical S-N curve validation

For the evaluation and prediction of fatigue lifetime of mechanical components the use of mathematical and statistical models is indispensable since this field is very complex and can be influenced by many factors besides the materials properties, where the stress range, stress level and the

size effect are one of the most challenging problems in this field and one of the most controversial. Being these factors target of so many discussions and with the need of reliable material characterizations for design mechanical parts there is a necessity of using statistical models to the fatigue analysis. There are many problems which could appear from the experiments made to gather data for the fatigue analysis and being these trials so expensive and time consuming could lead to the extrapolation of values. Which isn't a good approach due to the many factors involved in this field.

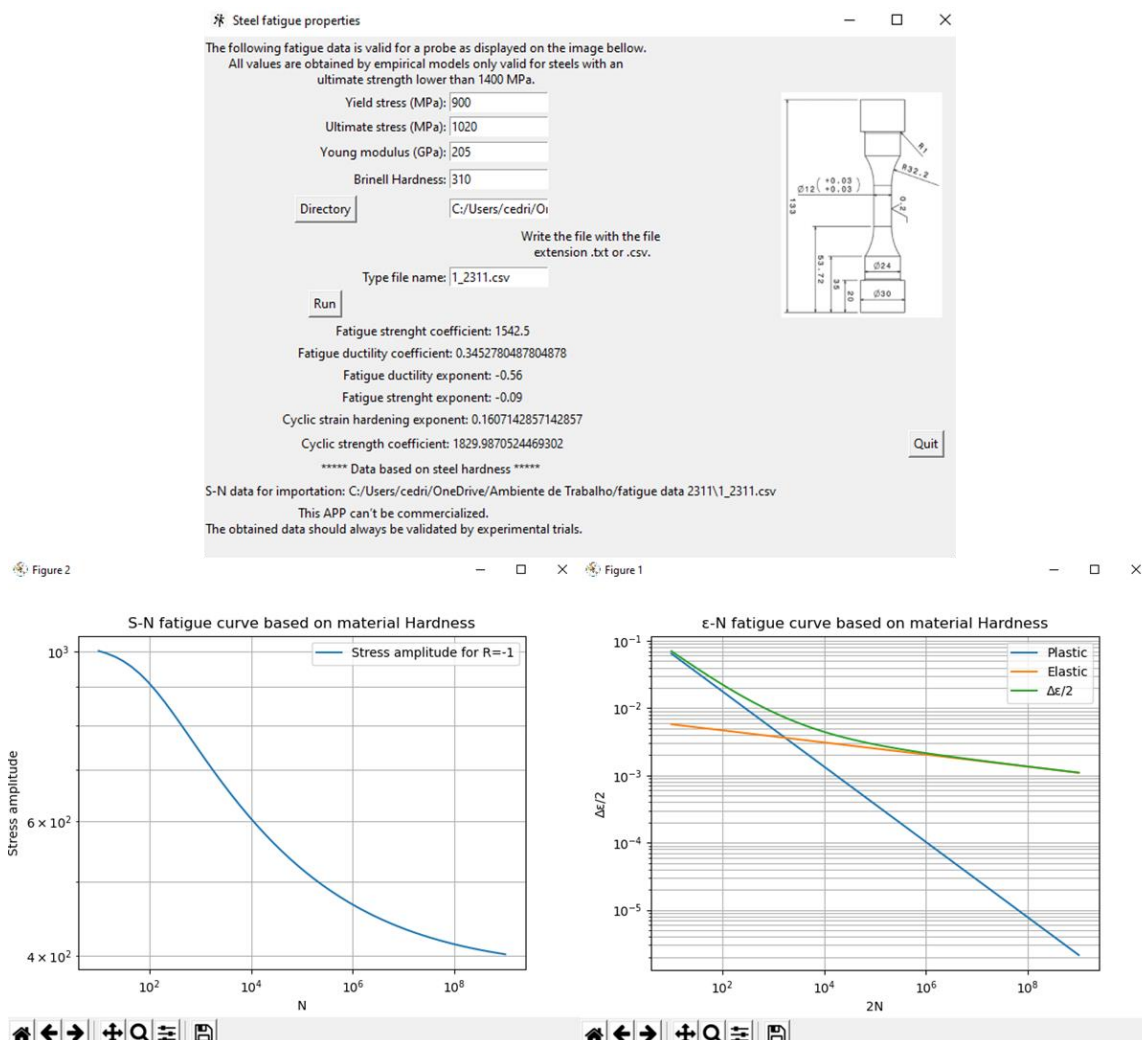


Figure 76 – Application interface output windows of the fatigue data.

The injection molds have specific steels to resist the work load during its work-life, these steels need an high hardness due to the erosion caused by the passage of the polymer during the injection and also high strength since these tools work with high pressures and high thermal variation which combined together can cause high stresses. The geometries of the molded parts also can lead to stresses concentrations where the high yield strength is an indispensable factor so the tool can resist all the work loads, but these factors have a negative impact on the steel fatigue resistance so, the study of its properties is indispensable and in this chapter two common tools steels are studied for its strength and fatigue limits in order to create theoretical numerical models. In this chapter is also presented a

theoretical model for the fatigue S-N curves so they can be used in numerical simulations to predict fatigue failure.

4.1.1.1 Tensile trials results of tool steel's

Since the goal of this study is to obtain the empirical fatigue curves using the monotonic steel properties, it was chosen a common tool steel used to manufacture injection tools for plastic parts for the automotive industry. The steel used for this study was the DIN 40CrMnMo7 from UDDEHOLM commonly known in the tool makers industry as EN 1.2311, which is used to make injection tools due to its properties such as good polishing, photo-etching properties, good machinability, high purity, good homogeneity and uniform hardness. The steel datasheet is presented in Appendix A, which will be used as reference for the trials made to obtain the monotonic data.

To obtain the monotonic properties, seven probes were made following the European standard EN ISO 6892-1, from which was obtained the dimensions presented in the Figure 77.

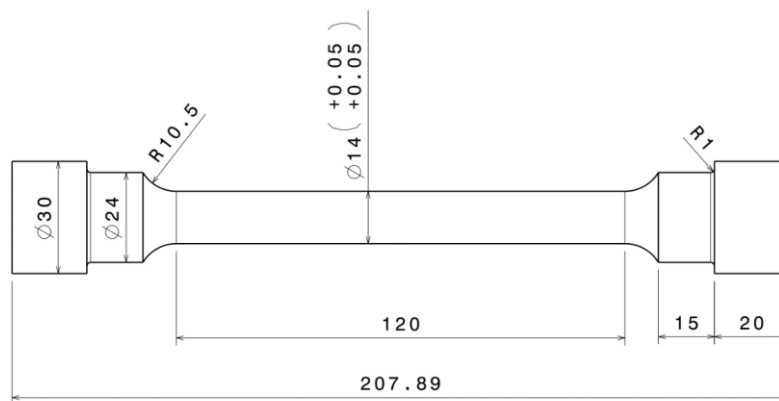


Figure 77 – Dimensions of the steel probe used to obtain the monotonic properties following the EN ISO 6892-1 standard.

From the probes will be measured the hardness, and from a destructive tensile trial will be obtained the Young's Modulus (E), Yield stress (S_y) and ultimate tensile stress (S_{ut}). From these results a comparison will be made with the steel datasheet to gather an idea of the security margin of the provided data, the probes were made from two different steel rods (five of each) from different lots to observe if there are any significant differences from the obtained results between lots.

The tensile tests were performed on a DARTEC testing machine with the layout shown in Figure 78 a), and a Sandner Messtechnik mechanical strain gauge - Serie: Dartec was used to measure the desired properties. The strain gauge is represented in Figure 78 b), this mechanical strain gauge has a measuring length of 100 mm.

Table 11 shows the values obtained from the monotonic tests performed on the 7 specimens. To obtain the hardness values of the material, a BAQ dynaROCK II portable hardness measuring device shown in Figure 79 was used, where ten measurements were taken for each specimen and the median

was considered as the value to be considered.

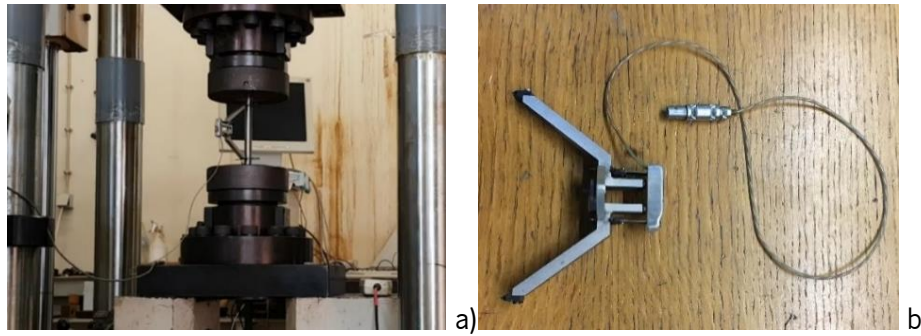


Figure 78 – Trial set-up where a) shows the probe in the trial machine during tensile trial with mechanical strain gauge and b) the mechanical strain gauge.

Table 11 – Monotonic data from the steel provider and from tensile trials from EN 1.2311.

	Young's Modulus [MPa]	Yield stress [MPa]	Ultimate tensile stress [MPa]	Brinell Hardness [HB]
Steel datasheet	205000	900	1020	310
Trial 1	210558.4	1014.3	1135.4	336
Trial 2	Miss trial			
Trial 3	215728.4	1014.1	1136.9	337
Trial 4	210647.4	1020.7	1150.5	341
Trial 5	209404.8	913.2	1142.3	337
Trial 6	213878.5	1026.6	1149.3	339
Trial 7	205243.0	1024.5	1142.5	336
Average from trials	210910.1	1002.2	1142.8	≈338

From the tensile trials was possible to observe that the tested probes presented higher values for the all the measured parameters. So, since the material to produce the probes came from two different lots and for the exception of the trial 5 which showed values close to the steel datasheet, is possible to conclude that the data provided by the datasheet is conservative compared to the results obtained from the tensile trials.

4.1.1.2 Results from fatigue trials of steel probes

The fatigue trials made during this work has the purpose of validating an empirical S-N fatigue curve and not to create the fatigue curve. To do so thirty trials were made using the same steel as the one used for the tensile trial tests, the UDDEHOLM EN 1.2311.

The probes were manufactured having under account the standard from E606/E606M – 12, which resulted in the probe with the shape presented in Figure 80. As for the tensile trials the probes were made from the same two material lots as the ones used for the tensile trials.



Figure 79 – BAQ dynaROCK II portable hardness measuring device.

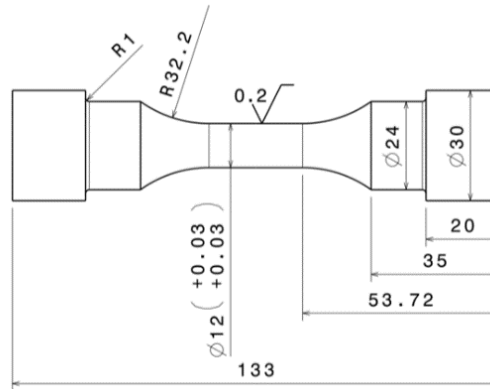


Figure 80 – Dimensions of the steel probe used to verify the S-N curve following the E606/E606M – 12 standard.

To conduct the trials it was used as basis, two curves presented by Charminda S. Bandera, et al. [157], [160] presented in the section 3.3. From which was possible to obtain the two plots for the S-N curves, using the simplified full range S-N curve presented in section 3.3.3, which will be addressed from now on as model 1. The second curve used as a basis is obtained from the Hardness method presented in section 3.3.5, which will be addressed as model 2. The curves are presented in Figure 81 for the steel EN 1.2311, which will then be compared individually with the tests on the steel probes in order to verify which of them can give the best predictions.

During the trials, model 2 is used as reference since this model is the most conservative, to determine the point to stop the trials in the case the probes doesn't crack by fatigue. There are two criterions, the first is if it passes 1×10^7 cycles because it is outside the normal cycle range for injection tools. The other criterion is if it presents an endurance higher than 10% of the model 1 prediction.

Figure 82 and Figure 83 show the results obtained from the experimental tests with the empirical curves obtained from models 1 and 2 respectively. When the results are compared with the curve of model 1 it can be observed that the results tend to converge with the curve especially as the alternating stresses are lower and therefore closer to the high cycle and giga cycle values. When closer to the region of oligocyclic fatigue one finds that model 1 tends to show values that are too optimistic when compared to the experimental results. However, the S-N curves tend to present better results in the high cycle and giga cycle regions.

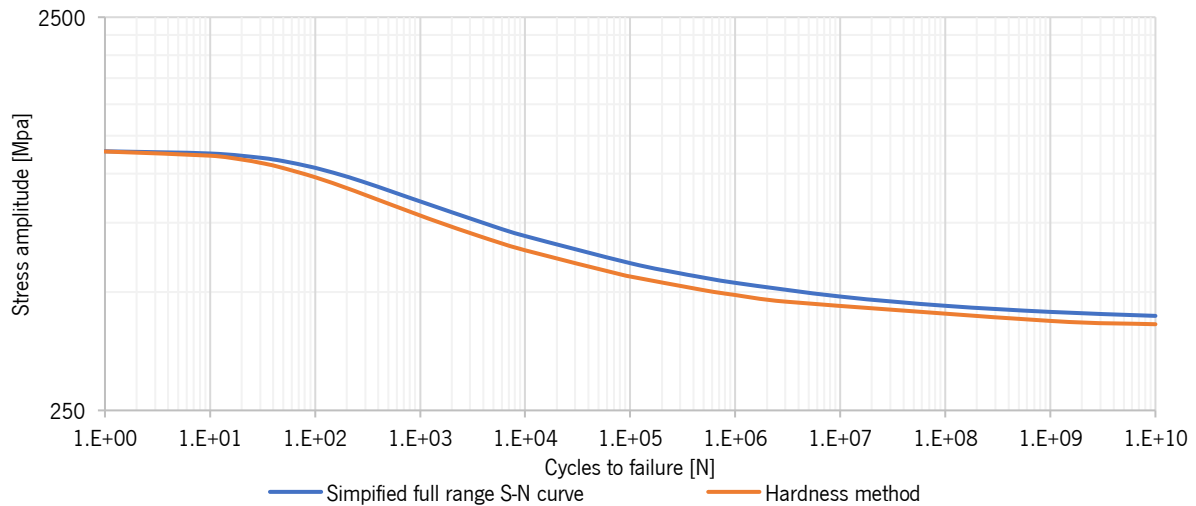


Figure 81 – Full range S-N curves obtained by the Simplified full range S-N curve (model 1) and the hardness method (model 2) for the steel EN 1.2311.

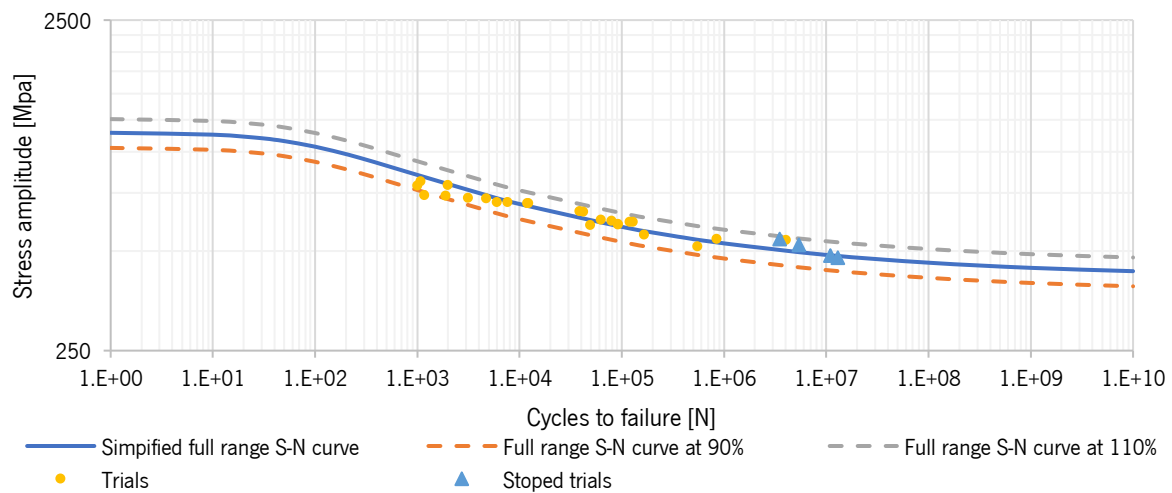


Figure 82 – Plot with the simplified full range S-N curve (Model 1) and the trials results.

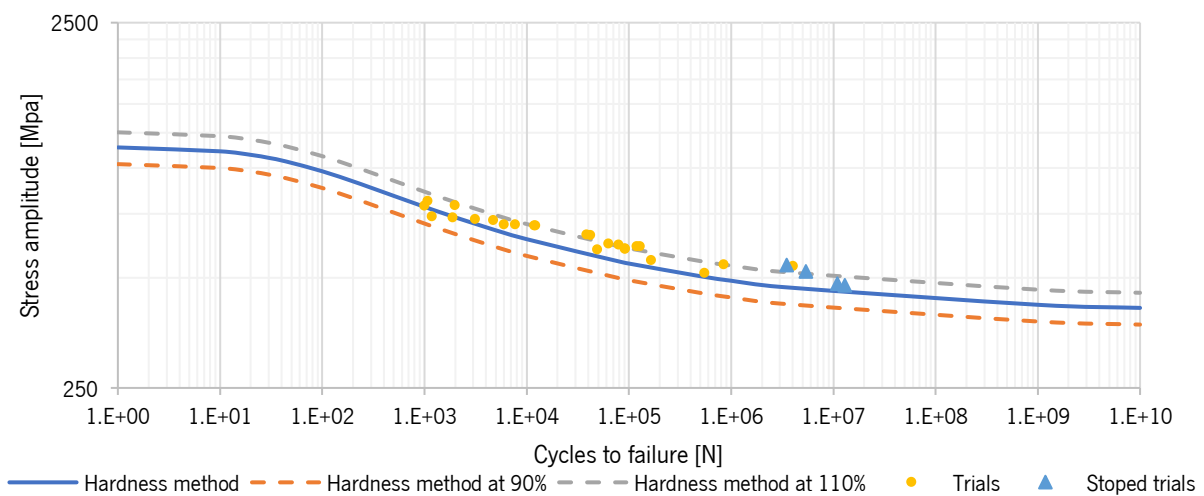


Figure 83 – Plot with the hardness method curve (Model 2) and the trials results.

The model 2, proves to be more conservative when compared to the experimental results. In this model the results obtained experimentally tend to align with the curve plotted at 110% of the value of the empirical model. Therefore, the results obtained align with what was presented in the analyzed papers.

It can also be concluded that for this steel, these empirical curves present a good response, however model 2 is more conservative which may be more interesting for use in design phases in which there is no experimental information about the steel due to safety margin it presents.

4.2 Link between Moldflow and structural software

In order to study the tool is needed to have the input loads for the numerical software. In the case of injection tools these loads can be thermal loads and pressure loads. In the section 5 this theme will be addressed, to explain why and when these loads should be combined and when only the pressure loads are needed.

The consideration of the loads can be done by three methods. The first is to consider a homogeneous load through the entire engraving region which can lead to erroneous results, the second is the manual mapping of the loads for the instant where the pressures are higher. Which could lead to miss some instants where the hydrostatic pressure caused by the polymer can result in deformations of the tool that could generate more damage than in the instant of maximum pressure, and this process could be complex and time consuming. The third method is to import the loads from the rheological numerical simulation and map these loads in the tool on the structural numerical simulation software. With this process is possible to use the data already obtained from the rheology numerical simulation and also with a multistep numerical simulation study with all the needed timesteps to obtain the results needed to have a good understanding of the stresses on the tool through all the filling and packing phases.

In this work will be presented an application to convert the data from Moldflow into a generic datafile which will have the Moldflow node number, node position in space related to a global coordinate system, and the load data. To convert the data exported from Moldflow, the created application coded in Python follows the algorithm presented in the Figure 84.

For this algorithm to work there are some rules which must be followed such as the global system of coordinates should be the same in the Moldflow simulation, in the CAD design and in the FEM software's. Is very important to use the same global system of coordinates on all software's to avoid mistakes and to have a more fluid workflow. The files containing the load data from Moldflow must be numbered from 1 to n , n being the number of timesteps exported from Moldflow.

The last box to be filled by the user is related to the value to be admitted for the unfilled nodes, this is needed because Moldflow doesn't write values on the nodes at the unfilled region during the filling stage. For the pressure loads this is not problematic since the pressure at the unfilled nodes is 0 MPa, but for thermal loads this is not true. For these loads can be considered the tool working temperature, which is a good approximation, because even if there are variations of temperature at the tool surface globally they should be on average equal to the working temperature.

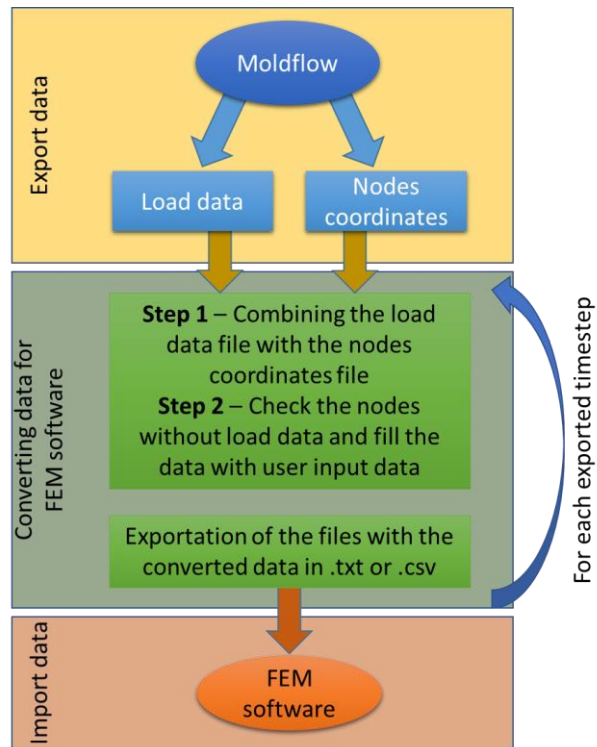


Figure 84 – Algorithm for the link interface between Moldflow and FEM software.

Data generator Moldflow 2 ANSYS

askdirectory Point the location of the files to import.

For the file name, the file numeration and extension should not be used. See the example below where only the orange section should be written.

-> Type data file name: pressure_data.txt

For the node data file it should be typed without the extension.

-> Type node file name:

-> Number of data files to import:

-> Insert value for unfilled nodes: (For example: 0 for pressure)

Run Quit

Figure 85 – Application interface window to generate the input file for the structural numerical simulation.

For the output files they are named as Time=" timestep instant"[s].txt, this notation is used to simplify the identification of each file, since the timestep instant is identified in the file name there is no need to check the correspondence between the file and the timestep instant from the injection pressure curve. Which helps on the structural numerical simulation since not all timesteps are needed and with this identification is easier to choose the needed files to import.

With this application it will be possible to map the loads obtained from the commercial software Moldflow onto a finite element software for structural calculation. This will allow the designer to have insight into the mold behavior throughout the injection cycle and not only at the instant of maximum load. For the quality of the input data will largely define the quality of the results that are obtained. For example,

Figure 86 shows a case in which the pressures obtained from the commercial software Moldflow were mapped. It is also noticeable that there is a table that presents the different imported time-steps in order to understand how the pressures will affect the mold behavior.

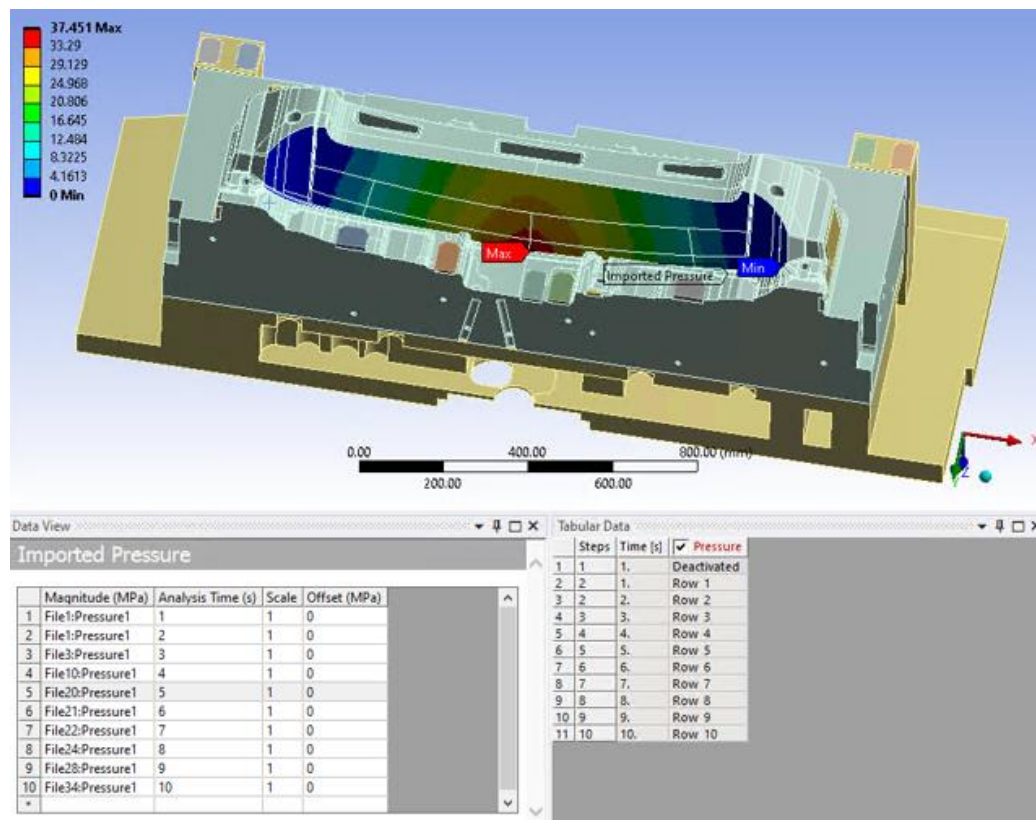


Figure 86 – Mapped pressures of one time-step in a mold engraving region using the application presented in this chapter and the table presenting the other mapped time-steps in ANSYS Mechanical.

This will give a better insight into the mold behavior when compared to using a single time-step with the maximum pressure that can be applied inside the mold. For example, using the maximum pressure across the entire mold region may result in oversizing the mold, as the maximum pressure value may not be felt across the entire mold region, or it may be felt in different regions at different time instants. Therefore, being the value of the loads felt inside the mold variable over time, it is important to study how they affect the mold behavior over time, in order to correctly identify points that may give rise to defects in the part or damage in the mold, as well as to have the necessary information to solve these problems in the design phase.

Also, this structural study contemplating several time instants will be fundamental for a better fatigue analysis, because as we will see next in section 4.3, the use of several time instants is fundamental for the presented application to have purpose and to be more reliable.

4.3 Fatigue endurance limit calculation algorithm

To conduct the fatigue calculations, it was developed one application dedicated to injection tools. This

is a stand-alone application, so it will work using data easily obtained from software's such as ANSYS and then will perform the calculations where will calculate the safety coefficient, number of injection cycles to crack nucleation, damage per cycle and maximum alternated stress.

This application is built to be ready to be implemented directly as a standalone application developed in Python. As is possible to observe in the diagram of Figure 87 the application is made in three modules where the modules related to the input data and the results are ready to be modified so the algorithm can work as an embebbed application in a chosen software.

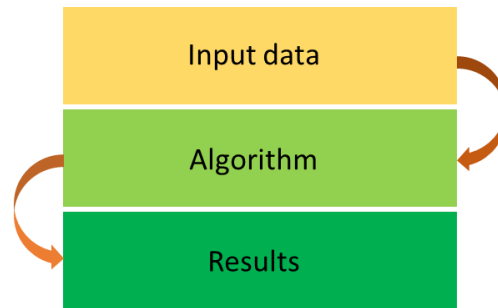


Figure 87 – Three phases of the application for fatigue calculation.

In the Figure 89 is presented the algorithm followed to code the application to calculate the tool fatigue resistance. The three phases are highlighted, as it was stated above this code is modular so it can be adapted to be used embebbed in a FEM software.

For the input data there are four items that are required to conduct the calculations which are:

- Number of cycles from project: this is where the user inputs the number of injection cycles that the tool should endure without failures by fatigue.
- Principal stresses: This data is the principal stresses exported from the FEM software. On the case of this application is formatted to import the data exported from ANSYS Mechanical in csv files with the formatting shown in Figure 88. The file has for all nodes the stresses for each time-step made on the numerical simulation. With these files must be imported a csv file with the nodes and their coordinates for the output of the results.

Node	Minimum Principal Stress (MPa) for Minimum	Minimum Principal Stress (MPa) for Minimum	Minimum Principal Stress (MPa) for Minimum	Minimum Principal Stress (MPa) for Minimum	Minimum Principal Stress (MPa) for Minimum	Minimum Principal Stress (MPa) for Minimum	Minimum Principal Stress (MPa) for Minimum	Minimum Principal Stress (MPa) for Minimum	Minimum Principal Stress (MPa) for Minimum	Minimum Principal Stress (MPa) for Minimum
Number	Principal Stress	Principal Stress 2	Principal Stress 3	Principal Stress 4	Principal Stress 5	Principal Stress 6	Principal Stress 7	Principal Stress 8	Principal Stress 9	Principal Stress 10
9046882	-24.721	-24.798	-24.873	-24.769	-22.516	-18.829	-15.465	-12.67	-11.657	-35.644
9046883	-109.34	-108.94	-108.88	-108.71	-106.39	-102.51	-99.083	-87.814	-78.243	-56.32
9046884	-25.086	-25.241	-25.377	-25.285	-23.26	-19.993	-16.733	-14.225	-13.063	-34.643
9046885	-25.191	-25.417	-25.616	-25.533	-23.67	-20.707	-17.609	-15.365	-14.247	-34.47
9046886	-25.422	-25.713	-25.967	-25.893	-24.189	-21.449	-18.501	-16.537	-15.594	-34.271
9046887	-25.623	-25.952	-26.238	-26.17	-24.576	-22.093	-19.396	-17.837	-17.024	-30.456
9046888	-25.557	-25.893	-26.18	-26.117	-24.651	-22.428	-20.255	-19.341	-18.617	-27.078

Figure 88 – Example of the maximum principal stress exported from ANSYS Mechanical.

- Material monotonic properties: these values are needed to generate the S-N curve which will be used to determine the number of cycles that the tool is predicted to endure.
- Fatigue endurance strength reduction factor: as presented in chapter 3.6 there are several

factors which will have a direct impact on the S-N curve, they need to be accounted in the calculations.

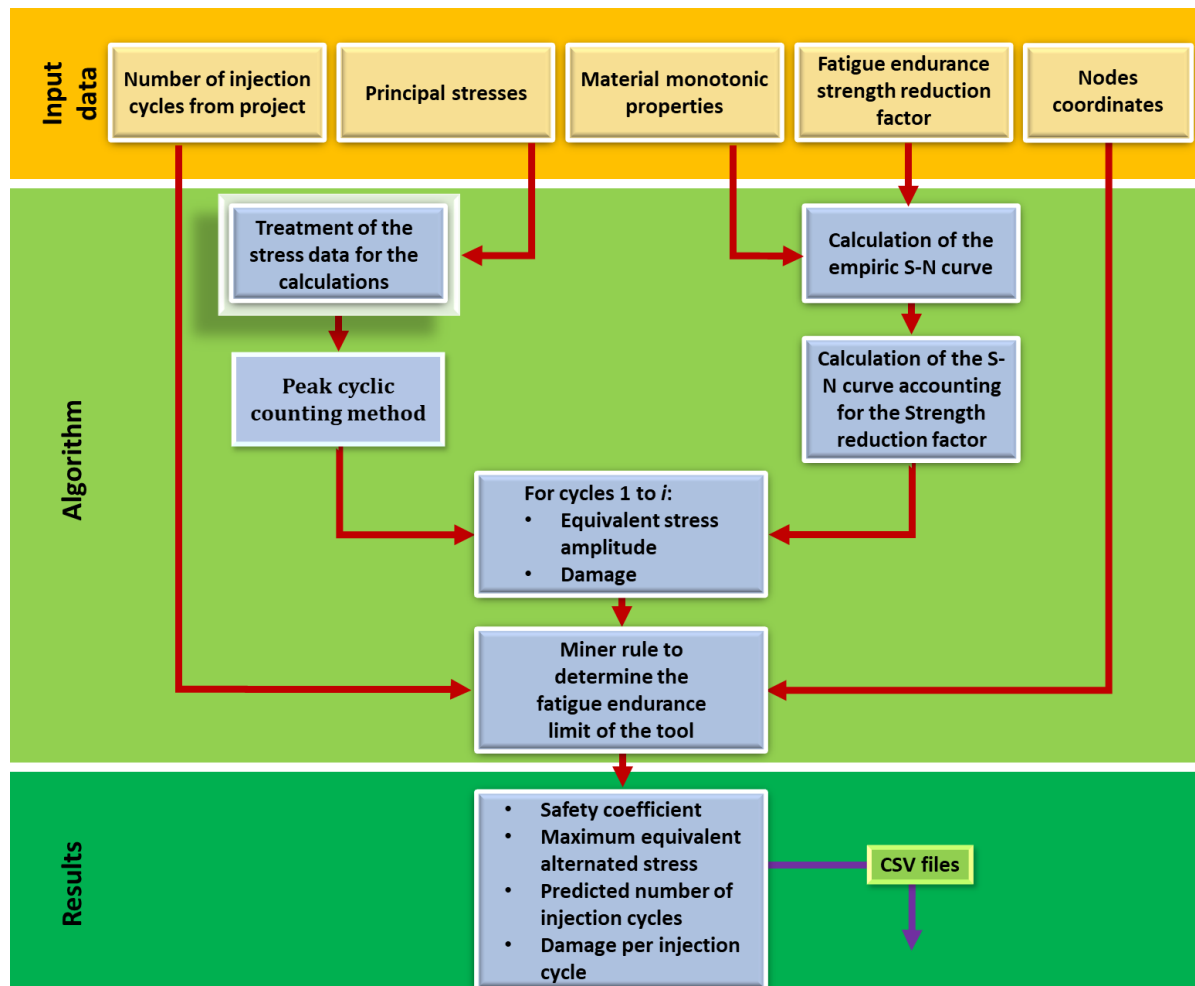


Figure 89 – Scheme of the procedure for the fatigue calculation application and its outputs.

The algorithm is composed by 6 blocs, at the beginning as presented in Figure 89 there are two operations made in parallel which will provide the needed data to proceed with the calculation of the required results. In Figure 90 is presented how the application manages the data imported, where it splits the array by node to several lists, so it can conduct the next steps. So, this module is highlighted because here the imported data, if comes with a different format than the one presented in Figure 88 needs to be converted to this format, this is one of the reasons why the code is composed by blocks of code, so these operations can be adapted more easily to different FEM software's.

Figure 91 shows the algorithm of the application that will perform the calculation. Here we show the steps the application takes to perform the calculations. As can be seen, there is only interaction with the user at the very beginning, where the user has to enter the necessary data so that the calculations can be performed. Therefore, after entering the files with the principal stresses, the nodes and their coordinates, steel data and the fatigue reduction factors of the region under study, the application is

ready to perform the calculations and generate the files with the results autonomously. In the first step the input data will be prepared so that it can be analyzed, where in parallel the principal stresses are treated and the curve that characterizes the steel to fatigue is generated. From there a number of loops will be executed where for each node the necessary results will be calculated so that the analyst can perform the necessary analysis.

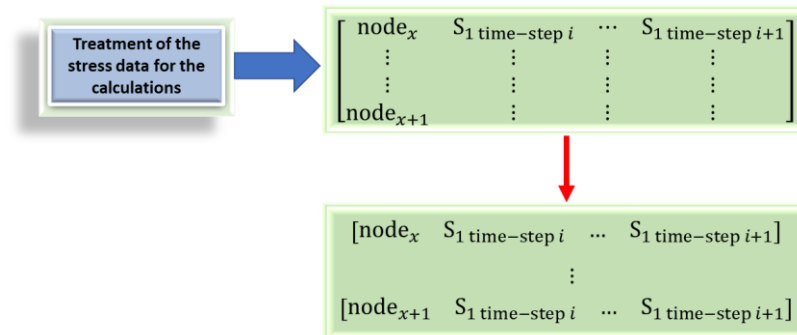


Figure 90 – Management of the maximum principal stress data.

In Figure 92 is presented the procedure used on the application to calculate the proposed results. The stresses files are imported and placed in arrays, from where the array containing the maximum principal stresses is considered the “master” array, so the order that the nodes will be selected is determined by the order which they are present on this array.

After extracting from the arrays of the principal stresses the necessary values for the calculations, which, as shown in Figure 90, are ordered from the first time-step to the last, then the values pass on to the module that will determine the number of cycles that is based on the peak counting method. Here the values of the maximum principal stresses are used to determine the load cycles. In order to determine the inflections, a comparison is made between three consecutive points to verify whether the intermediate point is a peak or not. In the Figure 93 is presented how this algorithm works where three cases are presented, two with inflections (Sel. 1 and Sel. 3) and one where the three consecutive points don't show any inflection. In the Sel. 1 and Sel. 3 the intermediate point corresponds to a maximum and a minimum respectively, so the values are added to the list of values of maximums and minimums and, for the values of the time-step, determined as an inflection in the maximum principal stresses, in the minimum and middle principal stresses values are also added to their respective lists as it is done for the maximum principal stresses. Here it is in this diagram also more noticeable how the calculations are performed and how the loops are executed in order to go through all the nodes and their alternating load cycles. it is also understood how important the normalization of the input data is, i.e. it is imperative that the arrangement of the nodes is the same in all arrays. Therefore, in the software used to perform the structural calculations it is important that it has the ability to export the ordered nodes as well as the different time-steps. If this does not happen the files will need to be treated to meet these requirements

before being inserted within this application.

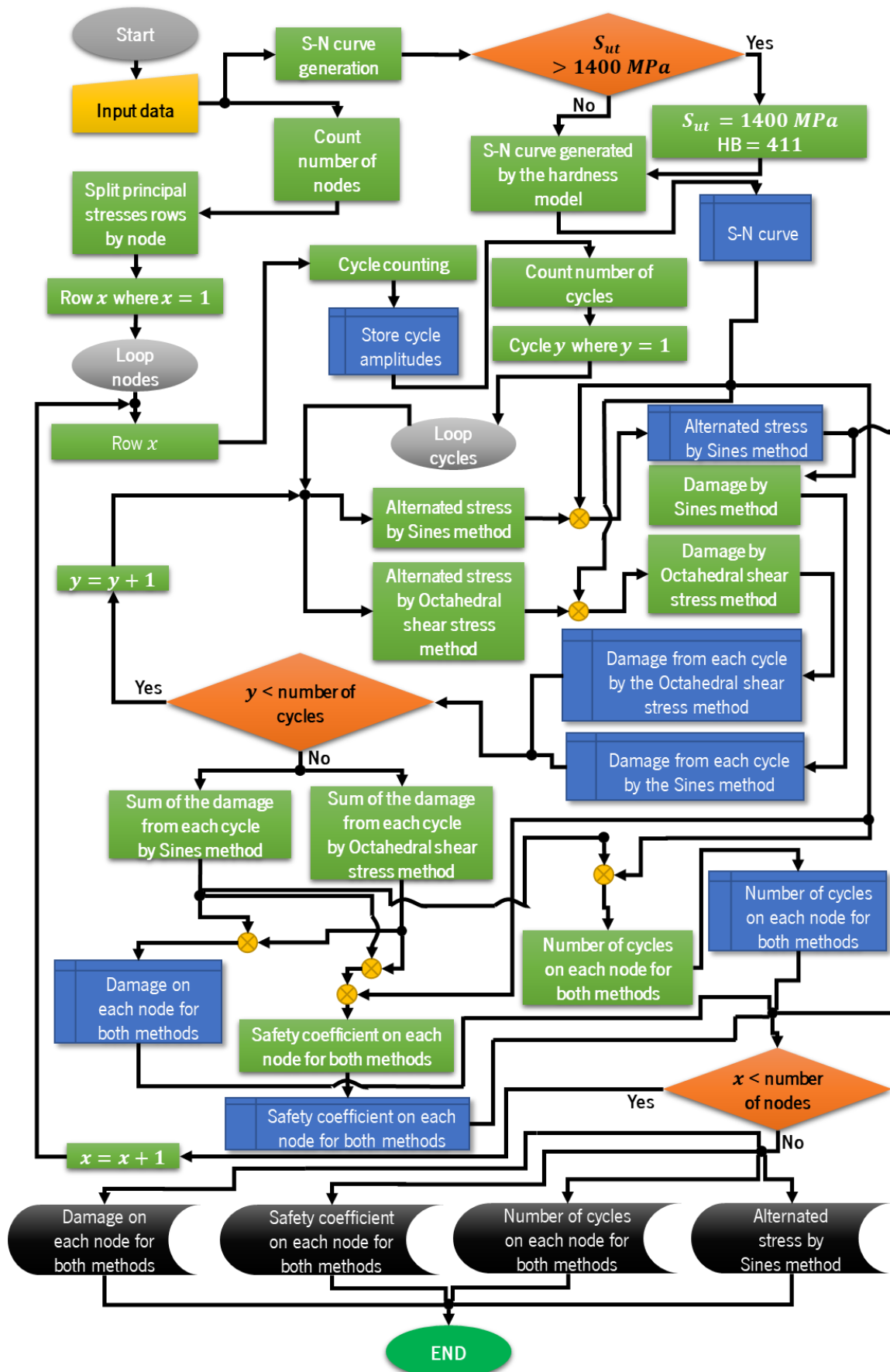


Figure 91 – Algorithm of the fatigue endurance limit application.

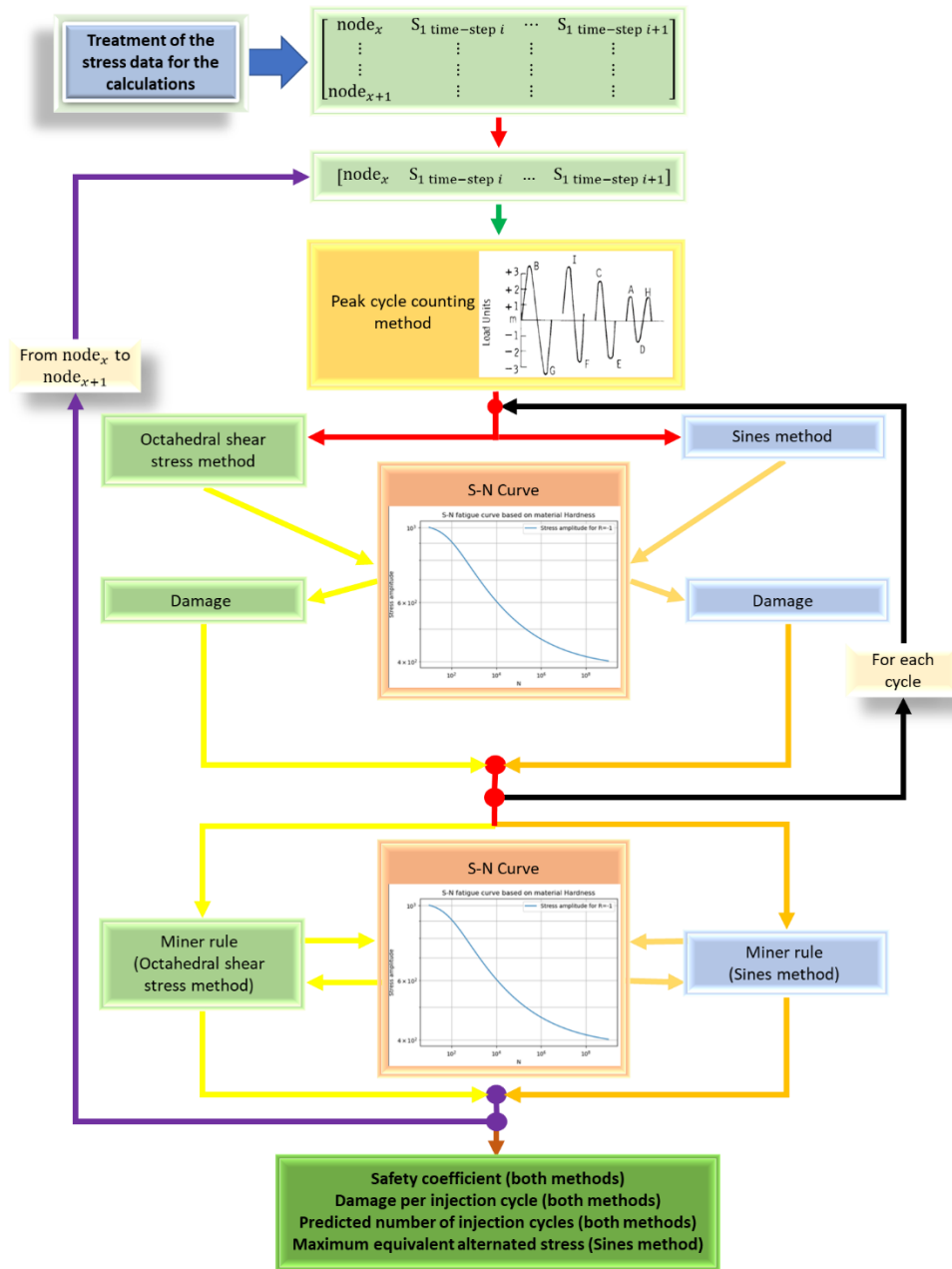


Figure 92 – Application procedure to calculate the tool fatigue endurance.

For the case of having three consecutive points without any inflection as presented by Sel. 2 in Figure 93, the algorithm discards this point and moves on, in this case being the points 4, 5 and 6 the next three consecutive points to analyze. This procedure is done for all time-steps, but as it is a three-point technique, the first and last time-steps are not analyzed. Therefore, it is necessary to check if the first and last time-step correspond to a minimum or maximum or if it can be discarded because it does not correspond to a peak. If they are a minimum or a maximum, the values are saved by the process already described for the three-point technique.

As the results of the numerical simulations disregard the initial stress state, where it is admitted that they are null, that is, at the end of each injection cycle it is admitted that there are no residual stresses caused by the previous cycle. Therefore, it is admitted that there is a peak where the stresses are zero

which can correspond to a maximum or minimum depending on whether the region is under tension or compression. So, an analysis is performed to the main stresses and if the region is under tension, one more point is added to the lists of minimums, otherwise it is added to the lists of maximums.

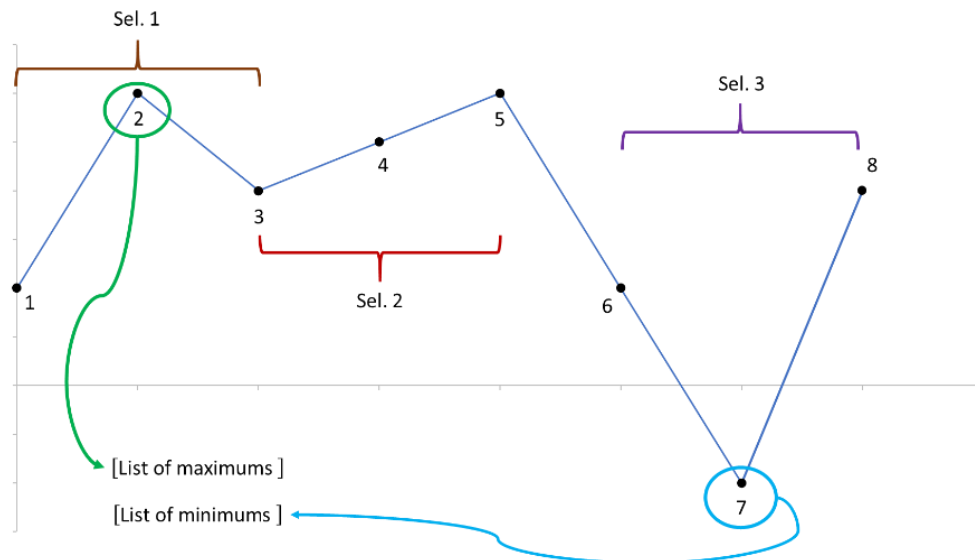


Figure 93 – Example of the counting cycles mechanism.

After finding the peaks, it is checked whether the value lists have the same dimension, that is, whether the number of peaks in the maximums and minimums lists are the same. If they are, values are removed from the list that has the most values in order to privilege the highest amplitudes in the cycle counter. Always removing the same time-step to each list of maximums or minimums from where is obtained 6 lists of values as presented in Figure 94 that will be used for the peak counting method.

Having obtained the load cycles, it is possible to proceed to the calculation of the equivalent alternated stress. Here the process is done by two methods such as presented in Figure 92, where the calculations are made by the Sines method and octahedral shear stress theory. This is done for each load cycle and where the damage caused by each load cycle is added in order to determine the damage of each injection cycle by the two methods.

From there it is possible to determine the number of cycles supported using the Miner rule along with the S-N curve. The S-N curve used already includes the reduction factor that is provided as an input, this curve is based on the empirical model also used in the application presented in the chapter 4.1.

The last operation to be performed by the application is the generation of the csv files with the results. This part is highlighted as it is necessary because the application works as a standalone application and hence need to create files that can be read by other software's. To do this, it will combine the results that come in an array, where the first column identifies the node and the second column has the result corresponding to the node, with another array that contains the nodes in the first column and in the rest the coordinates of the nodes in the space. as exemplified in Figure 95.

$[S_1 \text{ time-step } i \text{ max} \quad \dots \quad S_1 \text{ time-step } i \text{ max}+1]$
$[S_2 \text{ time-step } i \text{ max} \quad \dots \quad S_2 \text{ time-step } i \text{ max}+1]$
$[S_3 \text{ time-step } i \text{ max} \quad \dots \quad S_3 \text{ time-step } i \text{ max}+1]$
$[S_1 \text{ time-step } i \text{ min} \quad \dots \quad S_1 \text{ time-step } i \text{ min}+1]$
$[S_2 \text{ time-step } i \text{ min} \quad \dots \quad S_2 \text{ time-step } i \text{ min}+1]$
$[S_3 \text{ time-step } i \text{ min} \quad \dots \quad S_3 \text{ time-step } i \text{ min}+1]$

Figure 94 – Output from the algorithm to detect the peaks.

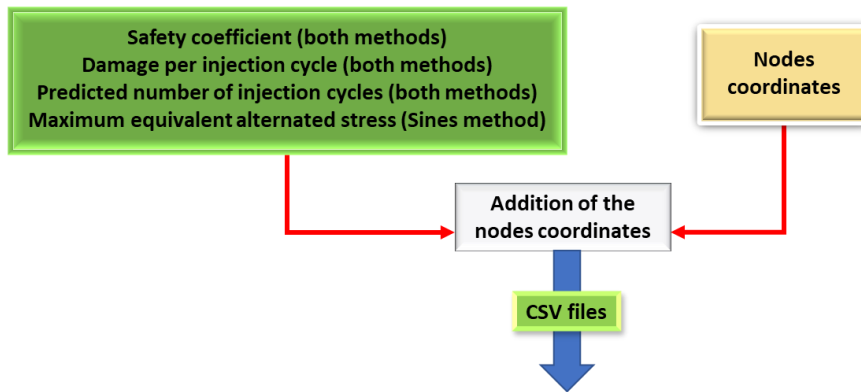


Figure 95 – Generation of the output files.

In the Figure 96 is presented the graphical interface of the developed application to generate the data discussed above. For this application as already stated, is necessary to input the files obtained from the FEM software which are the principal stresses data and the respective nodes coordinates, this last file is needed to map the values in an external visualization software. Then the user must input the number of steps exported, the value of m for the Sines method, the metal monotonic data, the fatigue endurance limit reduction factor and the predicted number of cycles that the injection mold is required to endure. The value of the coefficient m , by default is considered 0.5 but if the user has obtained a different value by experimental trials or literature, can use a more appropriate value for the calculations. The fatigue endurance limit reduction factor must be determined by the user to properly correct the S-N curve for the metal surface conditions of the region under analysis. As mentioned in section 3.6, the factors that influence fatigue strength can have an effect on the entire S-N curve as well as cumulative effects, i.e., their effect is felt as alternate cycles accumulate. Therefore, in the first cell should be placed the product of the factors that have cumulative effects, and in the second cell the product of the remaining factors. Finally, the user can click the button “Check data” to activate the button “Run”, this step is to ensure that the user have filled all the needed data before running the calculations. Before starting the calculations, the user must choose a location to save the generated files, if not, the application will save

at a default location in the computer.

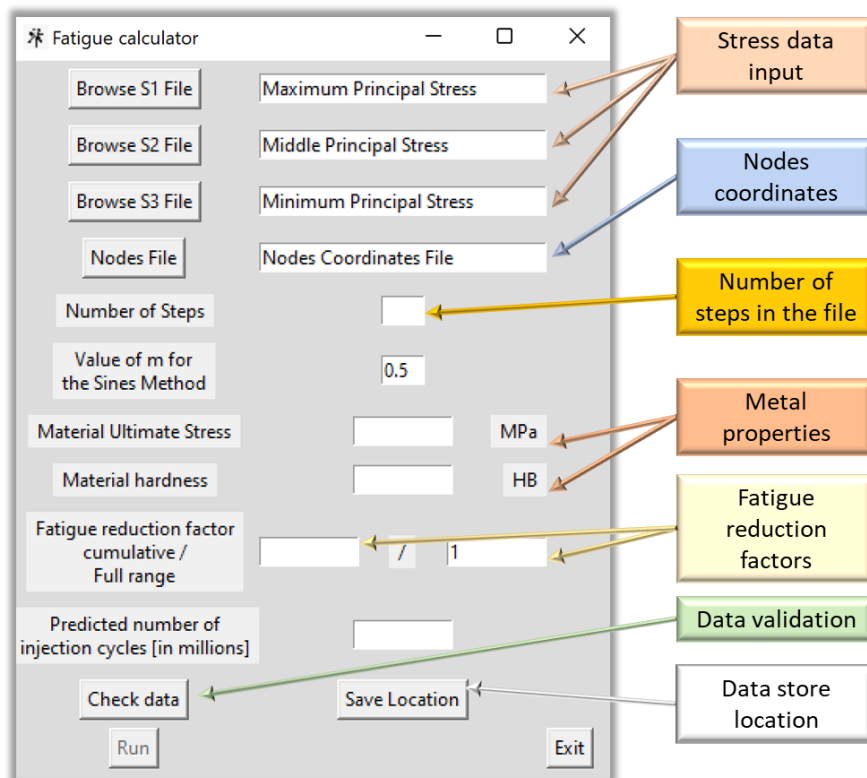


Figure 96 – Application graphical user interface.

With this we will have an output with the results recorded in csv files where the first column corresponds to the nodes, the next three contain the coordinates of the nodes in the space and the last one the corresponding result as shown in the example presented in Figure 97.

# Node	x [m]	y [m]	z [m]	Alternated stress [MPa]
9046882	5.66E-01	-8.47E-02	3.32E-02	1.65E+01
9046883	6.17E-01	-5.20E-02	-3.97E-02	-2.41E+01
9046884	5.71E-01	-8.28E-02	2.87E-02	1.47E+01

Figure 97 – Example of the csv file output for the equivalent alternated stress.

Having the results, it is necessary to be able to analyze them, so for this there are two options, one of which is to import the results into the FEM software if this possibility is provided by the software. As the results have all the necessary information for mapping the results into a surface because the files have the results linked to the nodes and the position of the nodes in space, it is possible to import them and use the method that is most convenient to map the results.

In case the FEM software is not able to import the results, it is proposed to use an open-source software, based on the Python language known as ParaView. For that, it is necessary to export a STL file with the components of which the surfaces were analyzed by the application. Having the STL file, it is imported into ParaView along with the csv files which contain the results as shown in the diagram in

Figure 98. This way it is possible to analyze the results without the need to use FEM software. Which can be advantageous considering that in the case of large models this process can be faster although at the same time it is a little less user friendly.

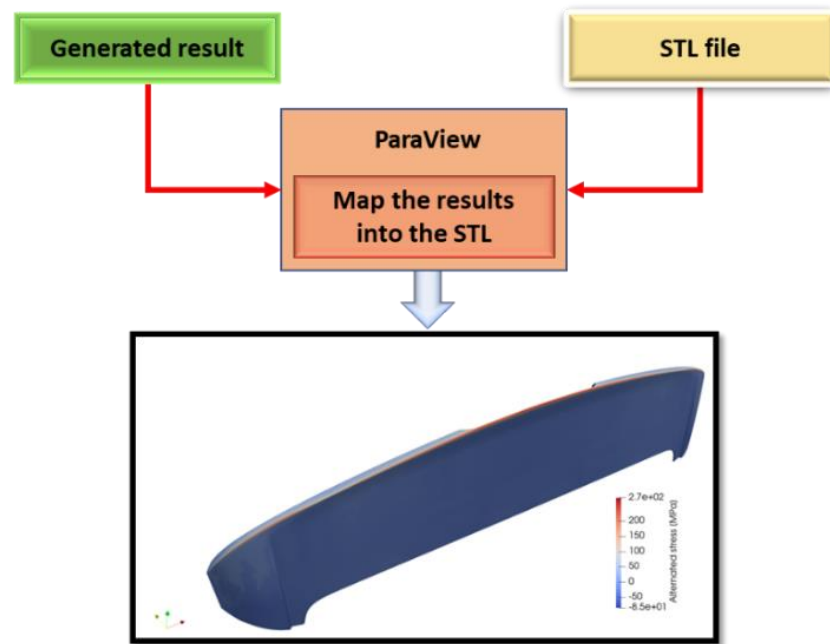


Figure 98 – Scheme to analyze the results outside the FEM software.

Therefore, this application will be used in the last phase of the study of mold resistance to fatigue caused by successive alternating loadings. That is, after the calculations performed using numerical simulation software for structural calculations, the data will be extracted so that this application can perform the necessary calculations. In the case of injection molds, different regions have different surface finishes, that is, as many data sets should be exported as the number of regions with different surface finishes or surface treatments. In case points are detected where fatigue crack nucleation is likely within the lifetime of the injection mold, the process should be repeated until the implemented improvements lead to no fatigue failure problems being found.

4.4 Proposed methodology for fatigue calculation of injection molds

In addition to creating tools capable of generating data for modeling curves that describe the fatigue behavior of steel alloys, creating bridges between numerical simulation software, and finally being able to perform semi-automatic fatigue calculations for injection molds, this work also aims to create a methodology for performing fatigue strength calculations.

Therefore, as it is obvious having tools does not mean that they are being properly applied. Therefore, in this section is intended to give an overview of how they should be used in order to get the most out of the created tools. And one should keep in mind that in the use of CAE tools, being methodic is very important to obtain reliable results. Therefore, taking into account the scheme presented in Figure 1, the

proposal for the execution of the numerical studies on injection molds to determine its fatigue endurance limit will be presented in Figure 99. With this proposal we intend to standardize the way injection molds are studied so that there can be a term of comparison between studies. For example, it is not possible to have a reliable comparison between numerical studies of injection molds where only the instant of maximum load is considered in one structural study and another where several time-steps are considered which will give a better perception of the mold's behavior throughout the cycle.

Therefore, for this model proposed at the beginning there is no difference between what is currently done in this industry. That is, as soon as the CAD of the plastic part for which the injection mold is to be built is received, the specifications of the plant where the mold will produce the parts and the limitations on the injection of the part are obtained, the design of the mechanics and dimensions of the injection mold, as well as the rheology studies of the part, can be carried out. The rheological study of the part once completed will give the positions of the injection point or points to the mold design team. Here the CAE team and the design team tend to work in partnership in order to build the most optimized tool taking into account all the limitations that may exist. Once this part of the process is complete, one can move on to structural numerical analysis.

Here the CAD model of the injection mold will be output and the rheological numerical study will provide the data needed to interface the application with the structural numerical calculation software. For the numerical structural calculation, it should be taken into account which time-steps should be used in order to have a reliable study and at the same time with the shortest possible calculation time. With this structural study the relative displacements between the cavity side and the core side of the mold must be analyzed as well as the stresses present in order to verify if the different regions of the mold are below the yield strength of the steel used. Also, if the used numerical software allows it, as in the case of the commercial software ANSYS Mechanical which has a simple fatigue calculation module, that can be used for a first approach to the results.

To perform this analysis, the application presented for generating the curves that characterize the materials empirically should be used to enter this information into the materials database. If all the requirements are met regarding the mold behavior in terms of structure and operation, one can move on to the fatigue analysis of the mold.

Then, if no points are detected in the structural numerical simulation that would require modifications to the mold design, we move on to the fatigue strength analysis of the injection mold, from which we have to extract the principal stresses from the structural numerical simulation. This procedure will have to be done as many times as there are regions with different finishes or different steel alloys.

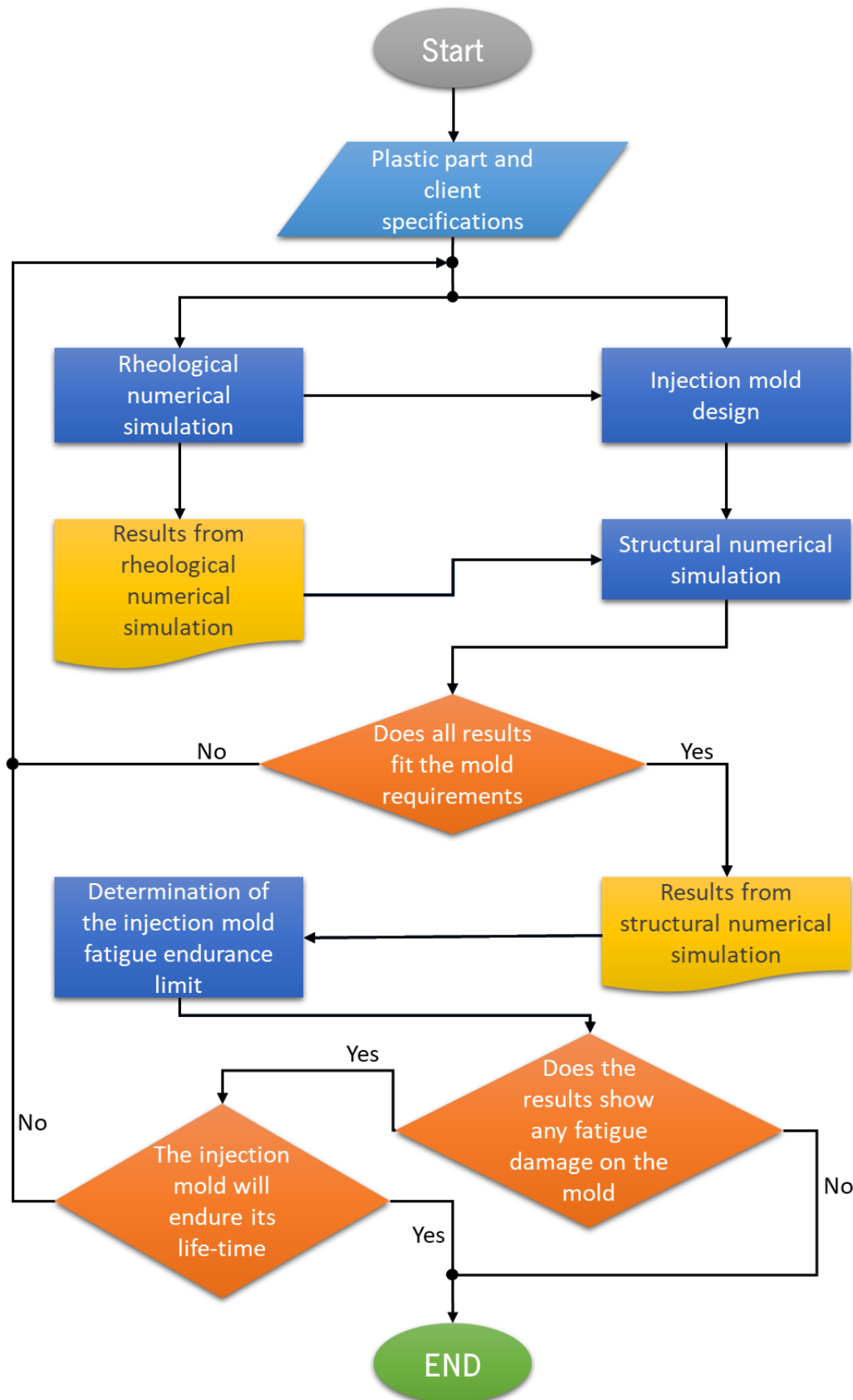


Figure 99 – Diagram of the proposed methodology for the determination of the fatigue endurance limit of injection molds.

With the principal stresses extracted from the numerical simulation software, and having determined the alloys and finishes that will be applied to the injection mold, all the necessary information has already

been collected for the application that will calculate the fatigue resistance of the molds. After entering the data into the application as presented in section 4.3 and the application performs the calculations, the CAE analyst can proceed with the data analysis. If there is no possibility of significant damage caused by alternating loads, the studies can be considered complete and the injection mold can be manufactured. If significant damage is detected in any region, it must be analyzed if this damage can lead to crack nucleation during the mold's life time. If this does not happen, the studies are considered finished and the injection mold can be manufactured; if crack nucleation during the mold's life time is probable, it is necessary to go back to the beginning of the design phase.

With the application of this methodology, it is expected that a good level of normalization will occur and that this will facilitate the analysis between different molds, since analysts will have to analyze information based on identical assumptions. These applications should also facilitate the work of preparing the models by generating in a semi-automatic way all the necessary information for the realization of the numerical simulations and further on the application for the fatigue calculation of injection molds facilitates by compiling the various models necessary for the calculations.

5 Injection mold structural analysis

On this chapter will be presented the study of two injection tools and respective plastic parts with numerical analysis. For this study will be used the CAE software's Moldflow and ANSYS to conduct the study of the injection process and the tool structural behavior, respectively.

The main goal is to understand how the simulations can give an approximate result when comparing with the experimental results. To do so, the injection tools will be equipped with industrial sensors so it can be possible to obtain information to be compared with the numerical simulations. As for the numerical simulations, the study of the injection process will be performed having under account the filling parameters from the injection trials which later will be used to obtain the loads for the structural numerical simulation using the application presented in section 4.1.1.

For the structural analysis numerical simulations, for each tool will be made four different analyses as presented in the schematic on Figure 100. By this analysis it is expected to understand the disparities between the numerical simulations and the trials, also is expected to understand the level of simplifications which can be made on the tool geometry so the computation time can be reduced without losing the results accuracy. This approach is necessary because in terms of using quasi-static or transient simulations there is a difference since the quasi-static simulation only works with the system stiffness which could lead to overconservative results, because the injection process can present significant changes of pressure loads during the injection process in small amounts of time so, in these simulations is expected to observe the major differences and how they can affect the results.

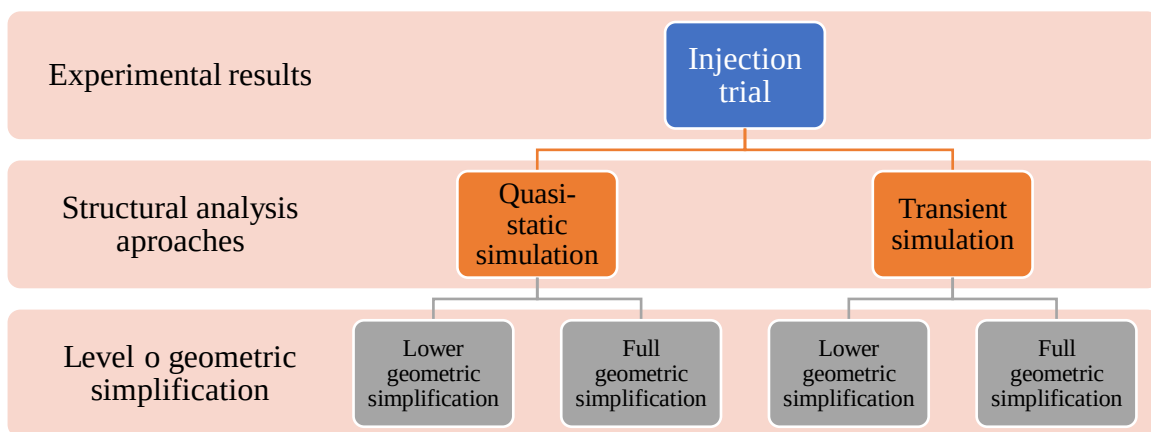


Figure 100 – Diagram for all the structural simulations made for the structural analysis.

To perform this study has stated before two tools will be studied. They differ in their complexity and in the injection profile. From now on the injection tools will be addressed as tool 1 and tool 2 as presented in the Figure 101, where the tool 1 injects two plastic parts with two injection points for each part and only one hot runner for both parts from which will be shared the molten plastic to fill both parts as shown in Figure 102 a).

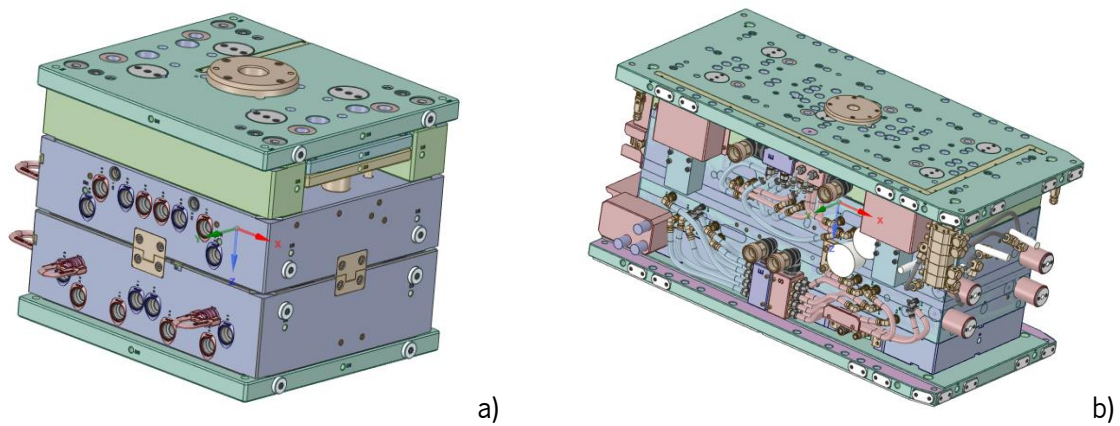


Figure 101 – Full model of the injection tools under study for study. a) is the tool 1 and b) the tool 2.

The tool 2, as shown in Figure 102 b), is more complex both in the part shape as in the injection since this tool has 7 injection points where 5 are cold gates and 2 are hot gates. The hot gates are the ones where the hot runner injects directly into the plastic part. In this part all the hot runners are equipped with valve gates so they can open only when the molten plastic reaches a certain location.

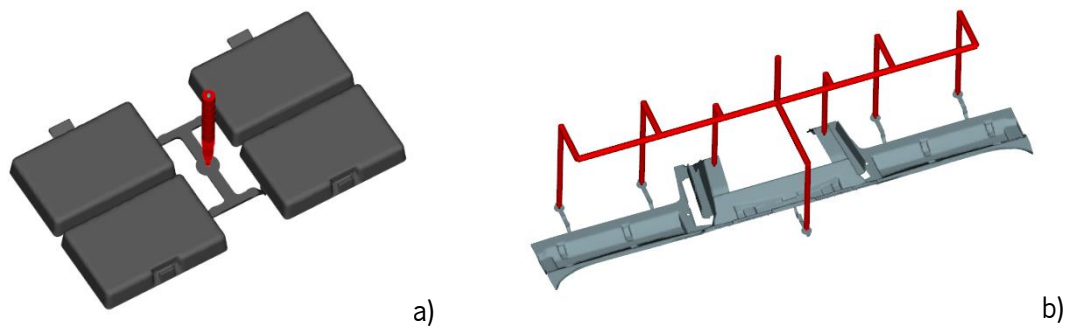


Figure 102 – Moldflow models from the plastic parts produced a) by tool 1 and b) by tool 2.

In the Figure 101 are represented the full models of both tools which will be simplified for the simulations where all the accessories will be removed, such as the cooling accessories, extraction plates, bolts, hoses, interlocks, electrical devices and other non-structural members. For the structural members such as interlocks, bolts and extraction plates their removal to simplify the model is based on the constraints made in the numerical model.

The function of the interlocks is to ensure the positioning of the component during the assembly of the injection tool and to ensure that the component during its service life won't slip due to shear forces at the connection, the bolts ensure that the components don't separate from themselves during their service life. In the Figure 103 is represented how the interlocks (in this case they are also sliding components to other fitting components responsible to ensure the positioning of the tool core and cavity when the tool opens and closes for the part ejection) works by imposing a restriction to movement of the components by the shear forces. Also is also represented the bolts where the orange arrows represent the working direction of the bolts.

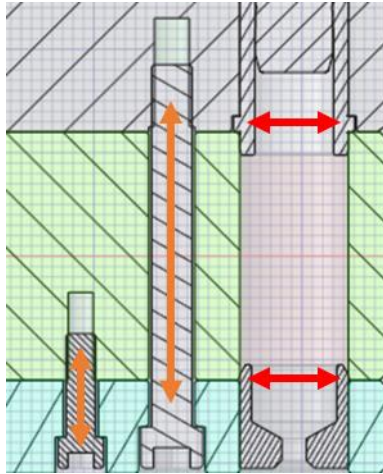


Figure 103 – Section of tool 1 where is represented the fixation between the bottom clamp plate, the rails and the core. The orange arrows represent the locked directions by the components.

So, having into account the locking directions caused by the interlocks and bolts is safe to assume that no sliding and no separation will occur. Having this into account these components will be removed from the numerical model to reduce the model size and will be replaced by a contact restraint which will represent the no separation and no sliding conditions through the connections interface.

The injection tools have components responsible for the plastic part ejection and demolding, but through all the instants under study on those components are locked in their position. Besides these components there are others called inserts which are made independently in order to reduce the tool manufacturing complexity or because they are made from a different material than the rest of the tool. They are grouped because they will be restrained in the numerical model in the same way. In the Figure 104 is represented one insert and one extraction component to represent graphically the reason of the constraint which will be applied. The extraction component in terms of mechanical behavior is similar to the lifter, the main difference between them is their function for the part ejection where the lifter only pulls the part away from the core, the extraction component helps to demold a complex region of the plastic part which couldn't demold without the spreading of the molding regions.

In Figure 104 is possible to observe that during the injection cycle both components are fitted in a lodge on the core where some surfaces are in contact with the molten plastic and the others are fitted to the core steel. In the Figure 104 b) is possible to observe the region where the molten plastic will apply the pressure on the components (pressure direction represented by red arrows), and the regions of the extraction component that are fitted to the core are represented with an orange line. For the regions in contact with the core steel the following assumptions are made:

- The bottom surface of the insert/slider/extraction components is bounded to the core since this surface doesn't have the freedom to separate from the core and can't move laterally since there are bolts or a guiding shaft locking it into its position. And to help the locking of

the components in terms of sliding there are the side walls of the core which also keeps the components on its position.

- For the other walls is considered that the only restriction to their displacement is the core wall, but they can slide and spread apart depending on the pressure load, their geometry and core displacement during the injection cycle.

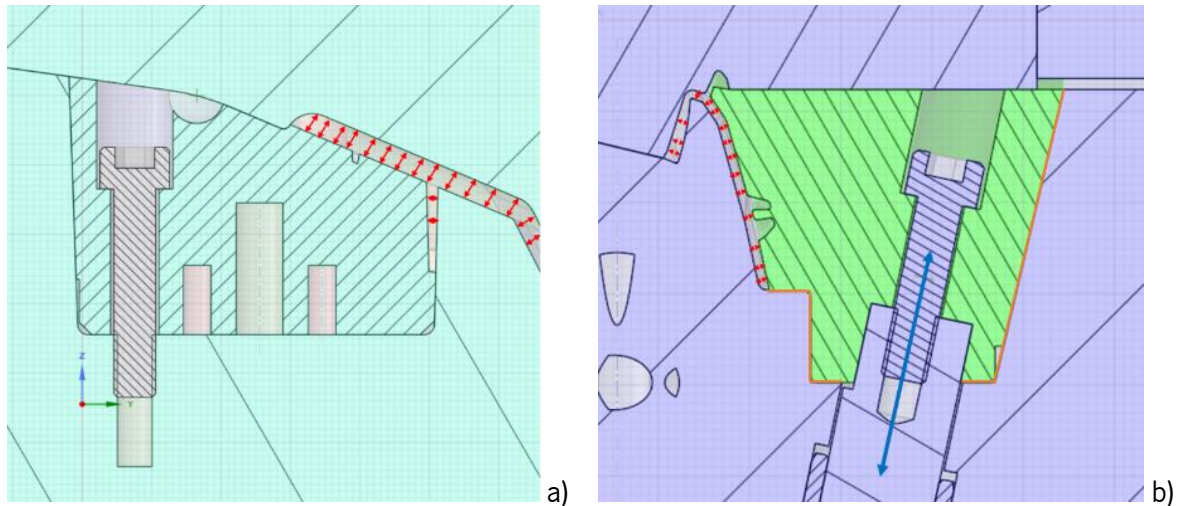


Figure 104 – Section view of one example of a) an insert and b) a slider. Where the red arrows represent the pressure load caused by the molten polymer, the blue arrow represents the movement made by the extraction component during the ejection of the plastic part and the orange line represents the walls in contact between the core/extraction component.

With these simplifications is expected to reduce the computation time without any significant loss of results accuracy.

The other simplification made is the removal of all chamfers because they force the generation of small elements at their location increasing the computation time without any significant impact on the results. Another simplification is related to the bolts, since the bolts are removed, the holes related to them are also removed also to reduce the size of the numerical model without any expected impact on the results.

These simplifications are made through all the simulations discussed on the diagram presented in Figure 100. As for the simplifications pointed in the diagram in Figure 100 are related to the adding of components, for instance the pressure and adjustment plates which are made from a different steel grade than the rest of the tool. Also, the structure on the core side where the bottom clamp plate, the rails and the supports are joined to be transformed in one component. Furthermore, all the holes and lodgments on the outer border of the injection tool and the lodgments for pressure and adjustment plates are filled also with the goal to reduce the computation time by reducing the number of elements and the number of connections.

As shown in the Figure 105 the main difference is related to the number of components and the simplifications by the removal of some lodgments made for sensors and other components.

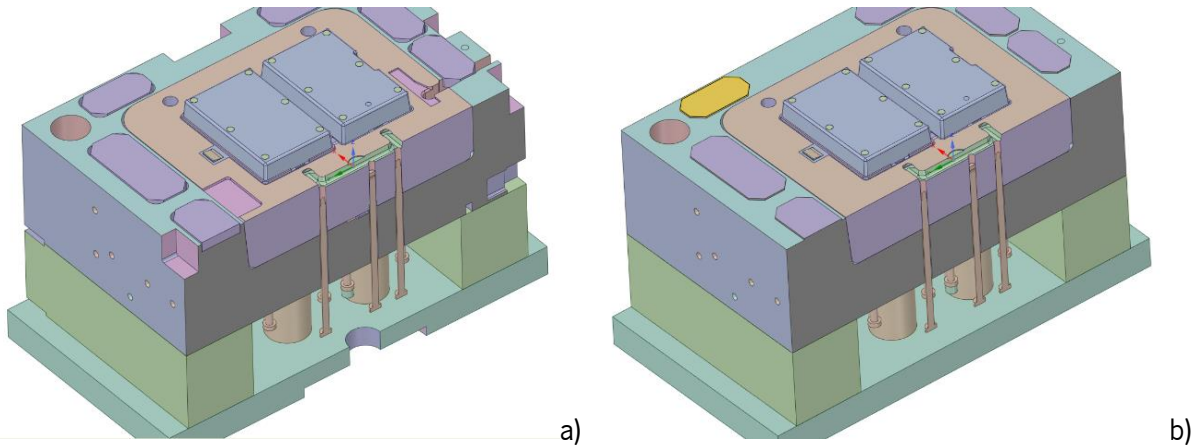


Figure 105 – Core side CAD from tool 1 prepared for the numerical analysis where a) is the model for the numerical analysis with lower simplification and b) for the numerical analysis fully simplified.

In the Table 12 is presented a summary for the differences of number of components between levels of simplification made for both tools. In section 5.2 will be presented the differences for the numerical model in terms of number of elements, connections and computation time.

Given Table 12, these simplifications will translate into a reduction in the number of elements and contacts. This can be observed in Table 13 where the number of contacts generated is presented. Here, contrary to what was presented in Table 12, the total number includes the contacts on the core side plus those on the cavity side plus those at the core-cavity interface.

This study aims to understand the effect of simplifications on the results obtained, and whether these simplifications lead to results that diverge from those obtained experimentally.

Table 12 – Number of components for each tool at each level of simplification.

		Core side	Cavity side	Total
Components in tool 1	Lower geometric simplification	34	3	37
	Full geometric simplification	21	3	24
Components in tool 2	Lower geometric simplification	46	28	74
	Full geometric simplification	11	6	17

Table 13 – Number of contacts for each tool at each level of simplification.

		Core side	Cavity side	Total
Contacts in tool 1	Lower geometric simplification	41	3	51
	Full geometric simplification	22	3	27
Contacts in tool 2	Lower geometric simplification	75	31	154
	Full geometric simplification	18	7	35

5.1 Plastic part rheology

As presented in the diagram on Figure 100, the simulations are made by reverse engineering. So for the inputs of the rheological numerical simulation, it was used the data collected from the injection trials.

In both tools an indirect pressure sensor was used to gather the pressure profile in one location of the tool. This sensor is useful to have an idea of the pressure profile but, for the pressure values on the engraving region caused by the molten plastic, the values obtained by indirect measuring will differ because of the way the sensor was assembled and the behavior of the surrounding components. As shown in Figure 106 the indirect pressure sensor is placed at the bottom of an ejection shaft and usually bolted to the extraction plate which is not represented in the figure.

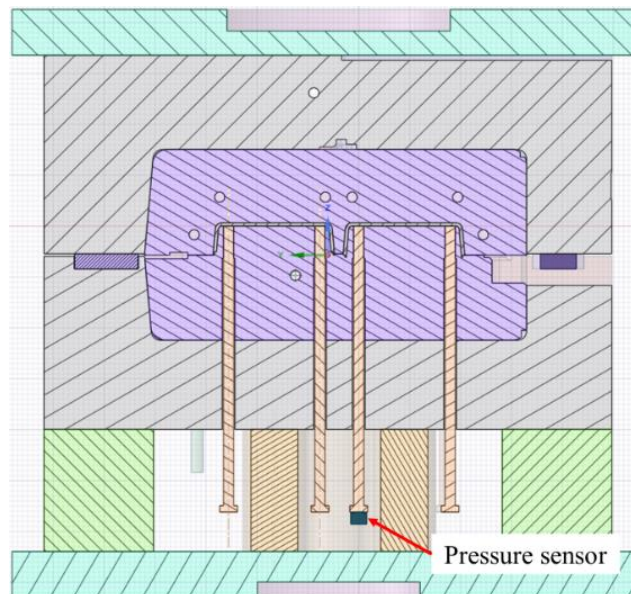


Figure 106 – Section of the simplified CAD of tool 1 detailing the position of the indirect pressure sensor.

5.1.1 Rheology numerical analysis from tool 1

As stated above the simulations are simple reverse engineering of the injection trial of the tool 1, from where the data presented in Table 14 was obtained.

Table 14 – Input data for numerical simulation obtained from trial.

Plastic material	ExxonMobil Chemical - Exxtral BNU011 In Appendix B is presented the polymer datasheet
Filling phase control	Flow rate [205 cm ³ /s]
Packing pressure	13 seconds – Packing pressure profile in Figure 107
Cooling time	20 seconds
Tool surface temperature	30 °C
Melt temperature	245°C

This data presented in Table 14 was used as input data for the rheological numerical analysis made in the software Moldflow, but there are some simplifications that must be addressed. The first is that the software doesn't consider for the numerical simulation the effects of the injection machine barrel, where exists some pressure drop which will lead into a higher pressure at the injection tool entrance than the one considered by Moldflow. Since a flow rate of material is the input data considered the pressure values will only consider the pressure loss at the part and injection channels.

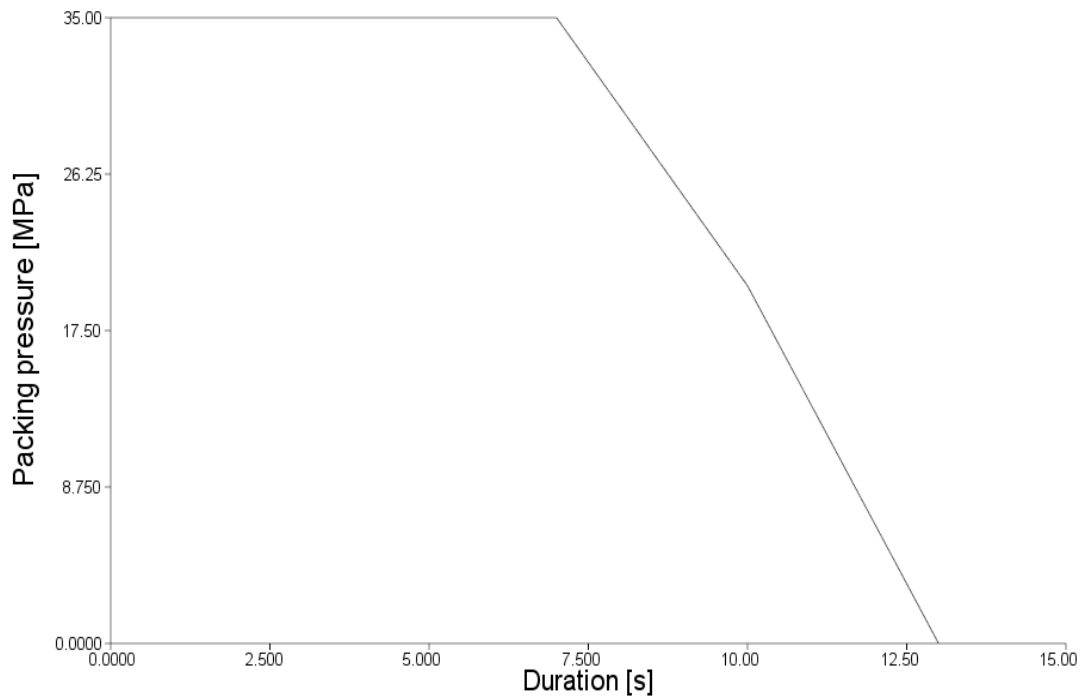


Figure 107 – Packing pressure profile of tool1.

The other simplification is related to the engraving surface temperature, which for this analysis was considered as homogeneous, but in reality, the temperature is variable though the surface depending on the geometry and proximity of cooling channels. However, since usually the temperature problem in injection tools is related with hot spots, that will help with the material viscosity by decreasing locally the polymer cooling rate and so reducing the injection pressure, is considered that this factor is beneficial since in the worst case it will slightly increase the injection pressure which will minimize the non-consideration of the injection machine barrel.

In Figure 108 it is possible to observe that there is some difficulty to fill the part, the last region to be filled is at the middle of the plastic flow range so is expected to have a spike in the pressure at the engraving region to force the molten plastic into the last region to be filled. This is caused by the change of the part thickness which will offer resistance to the plastic flow which as stated, will force the increase of the injection pressure to force the filling of the rest of the plastic part. In Figure 109 is presented the pressure distribution in the engraving region and is possible to observe that the pressure starts to increase

through the filled region for the exception of the last region to be filled, which means that there is high resistance to the molten plastic flow to finish the part filling.

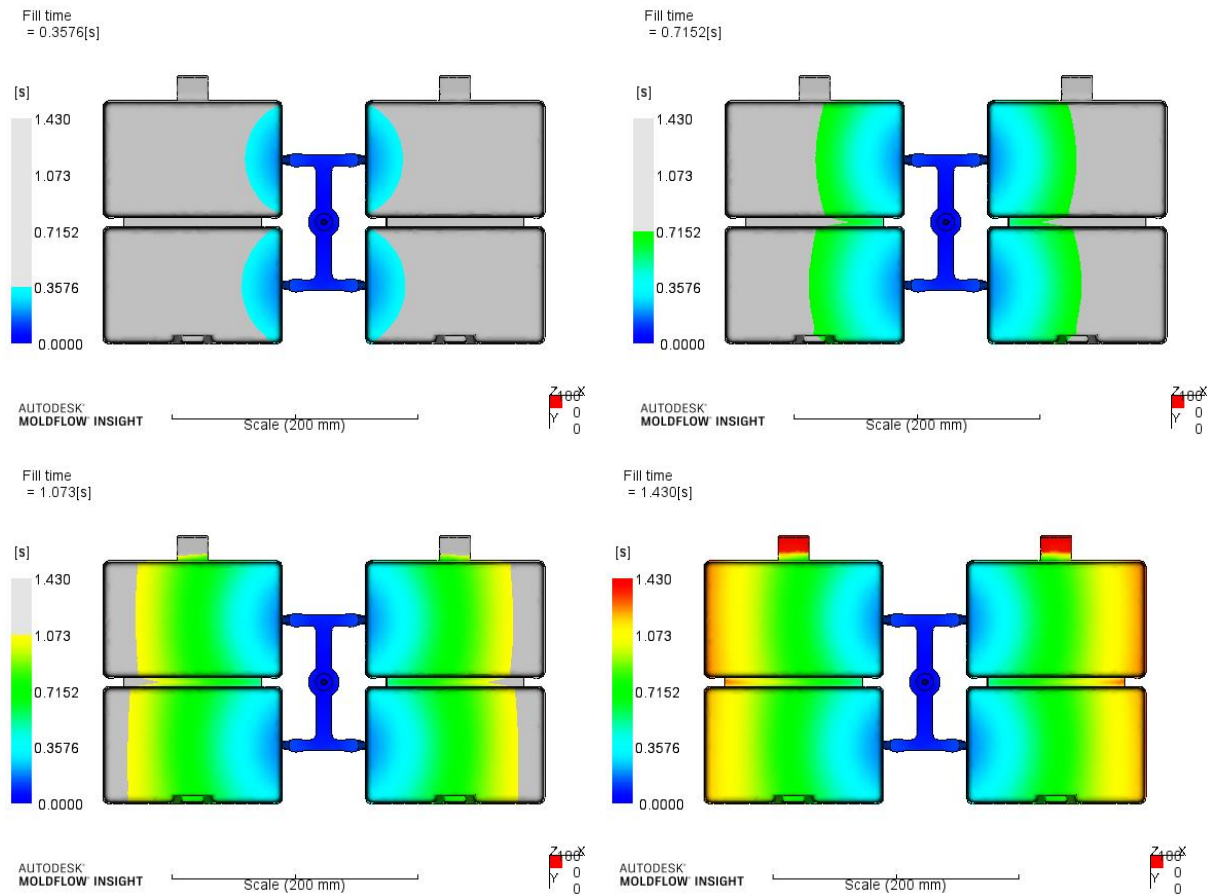


Figure 108 – Filling pattern of the plastic part from tool 1.

In Figure 110 is shown the pressure profile in several points where is possible to observe the pressure loss between the injection tool entrance and the engraving region. This pressure profile combined with the images of the pressure distribution, are useful to know the pressure instants needed for the structural simulation. On the curves shown in Figure 110 is possible to observe that for many intervals the pressure increases at an almost constant rate, so there is no need to increase the number of time steps on the structural simulation for these intervals unless there are significative changes on the polymer distribution pattern.

For the clamping force, the simulation has determined that at least 184.5 Ton will be required, i.e., 1.81×10^6 N to sustain the pressure inside the tool. Following the Moldflow documentation [42], this value should be multiplied by a factor of 1.2 to obtain the required clamping force for the injection molding machine. This factor is used to compensate for the pressure drop that is not covered in the numerical simulation due to the simplifications used by the software.

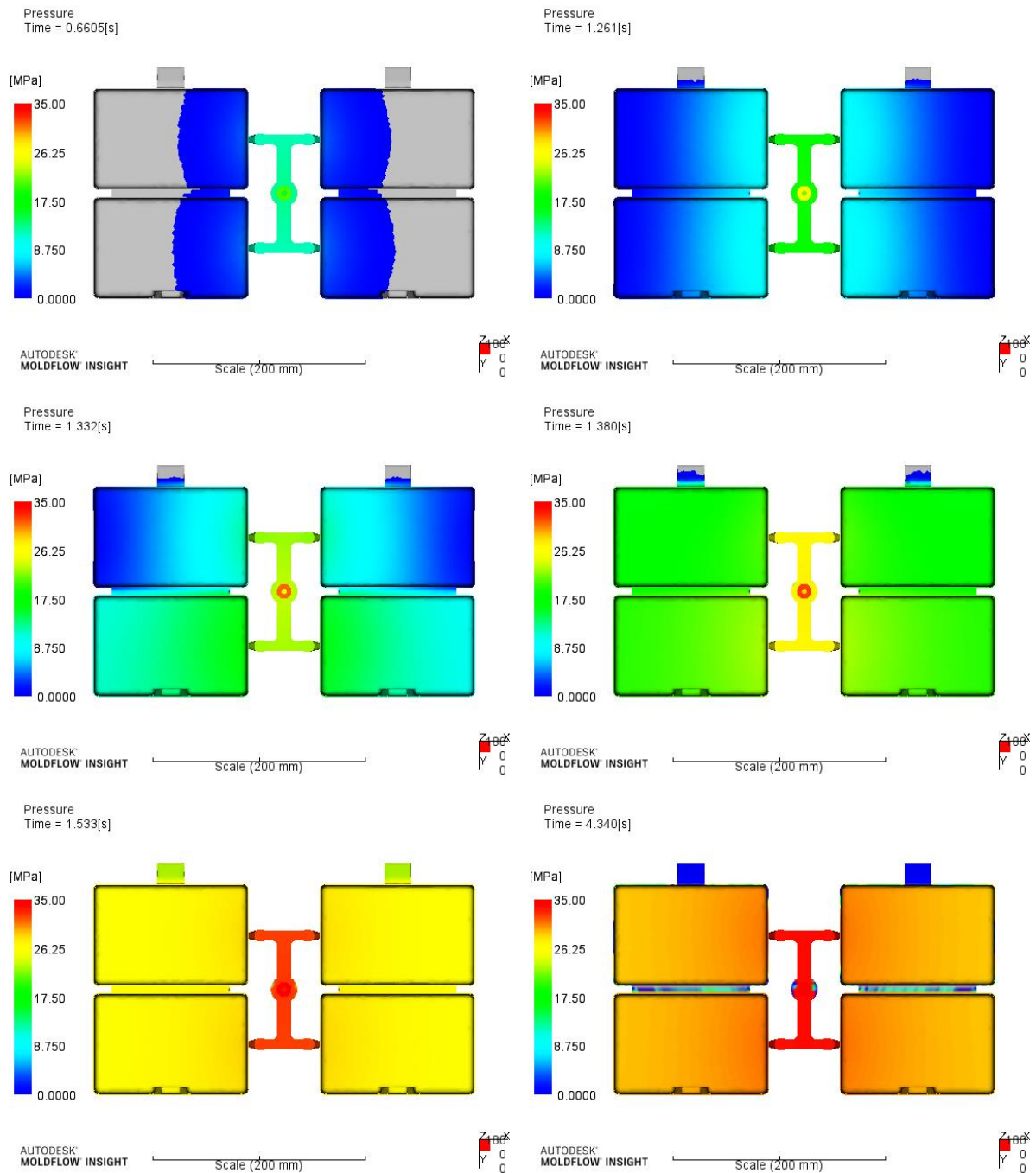


Figure 109 – Pressure distribution during the part injection cycle of tool 1.

From the analysis of the pressure distribution in the molding region, certain time instants were chosen that are relative to the instants where there are large variations in the volume filled by the polymer and where there are major pressure variations in the molding region as well as the instant of highest pressure in the recording region. These instants are represented in Figure 111 by the dashed vertical lines as well as the injection pressure, i.e., the pressure at the carburizer inlet, which is represented by the blue line.

These values of the pressure distribution will be exported from Moldflow, then the application presented on the chapter 4.1.1 will be used to generate the files needed to import on the structural software.

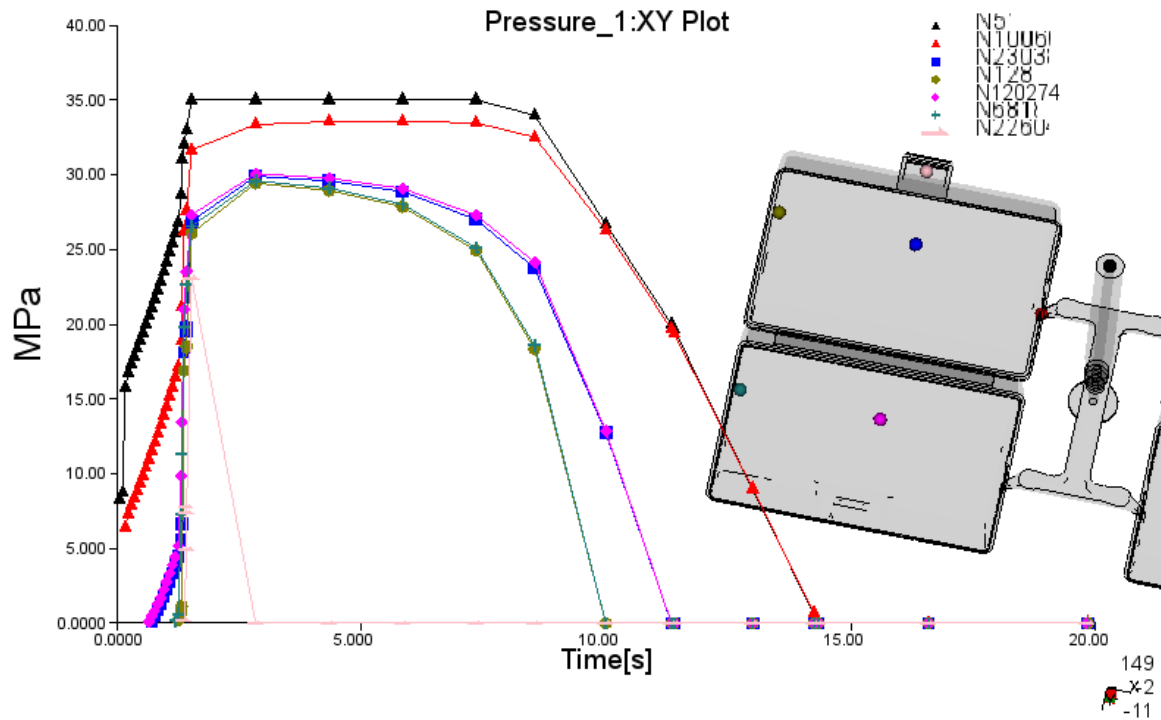


Figure 110 – Pressure at defined locations through time. The black line is the pressure at the injection tool entrance, the red line at the cold gate, the others are at important locations at the engraving region.

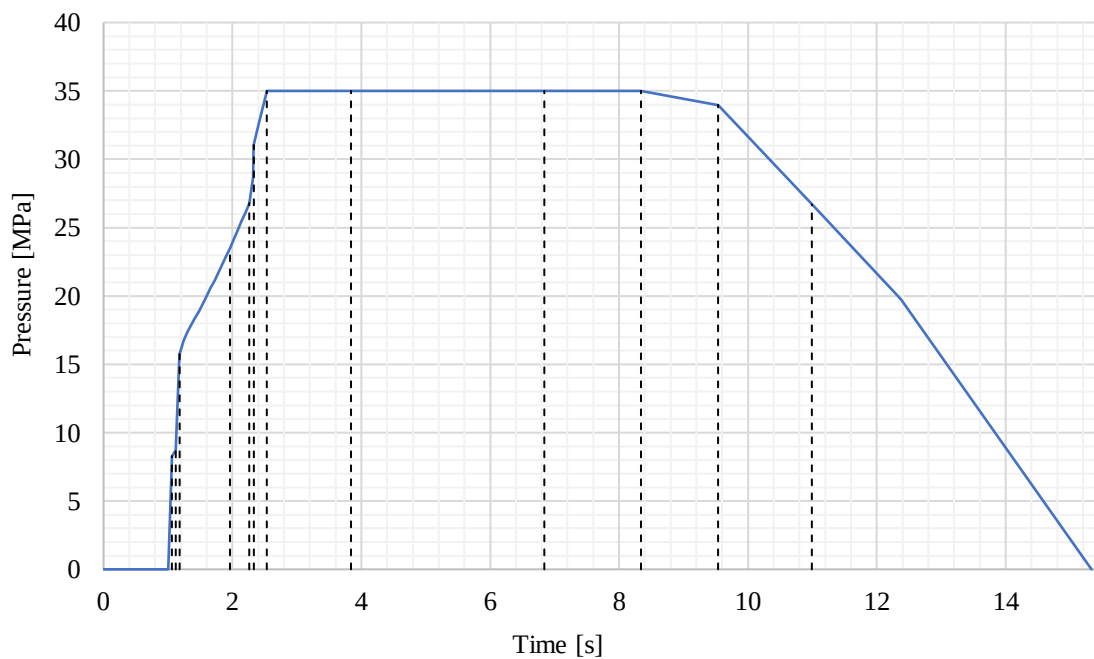


Figure 111 – Injection pressure curve and the dashed lines represent the time-steps used on the structural simulations.

5.1.2 Rheology numerical analysis from tool 2

As for the tool 1, in this tool it was performed the simulation with the parameters obtained from the trials. The parameters obtained from the trial are presented below.

Table 15 – Input data for numerical simulation obtained from trial.

Plastic material	ExxonMobil Chemical - Exxtral HMU202 In Appendix B is presented the polymer datasheet
Filling phase control	The filling control was made accordingly with the profile presented in Figure 112
Packing pressure	14.5 seconds – Packing pressure profile in Figure 113
Cooling time	28 seconds
Tool surface temperature	28 °C
Melt temperature	240°C

On this tool, the injection control was made by an injection velocity profile as presented in Figure 112, this is made to control the material flow and the pressure due to the influence of the part shape and the opening of valve gates.

The same assumptions mentioned in the chapter 5.1.2 are also valid for this numerical simulation, and also in this tool an indirect pressure sensor was used to gather the pressure profile.

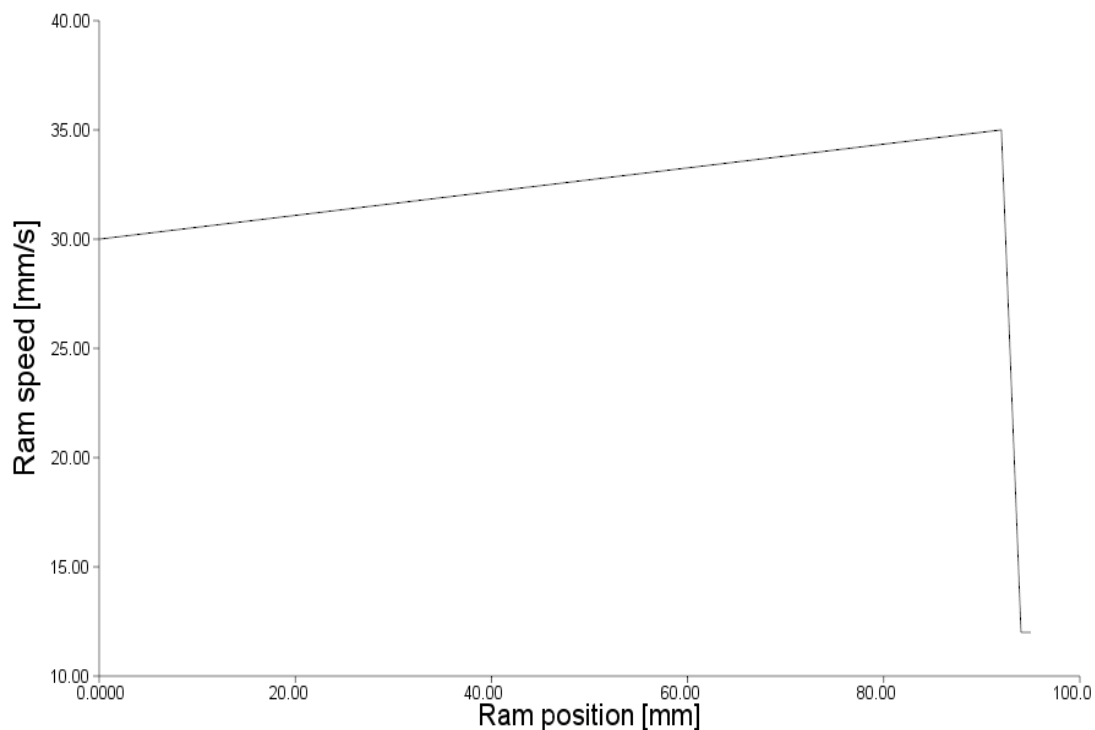


Figure 112 – Ram speed Vs ram position in tool 2.

In Figure 114 is presented the filling pattern of the plastic part and as is possible to observe the use of this system with hot runners with valve gates allows a more controlled filling of the plastic part with lower pressures and without the appearance of additional weld lines than the ones caused by the holes on the part.

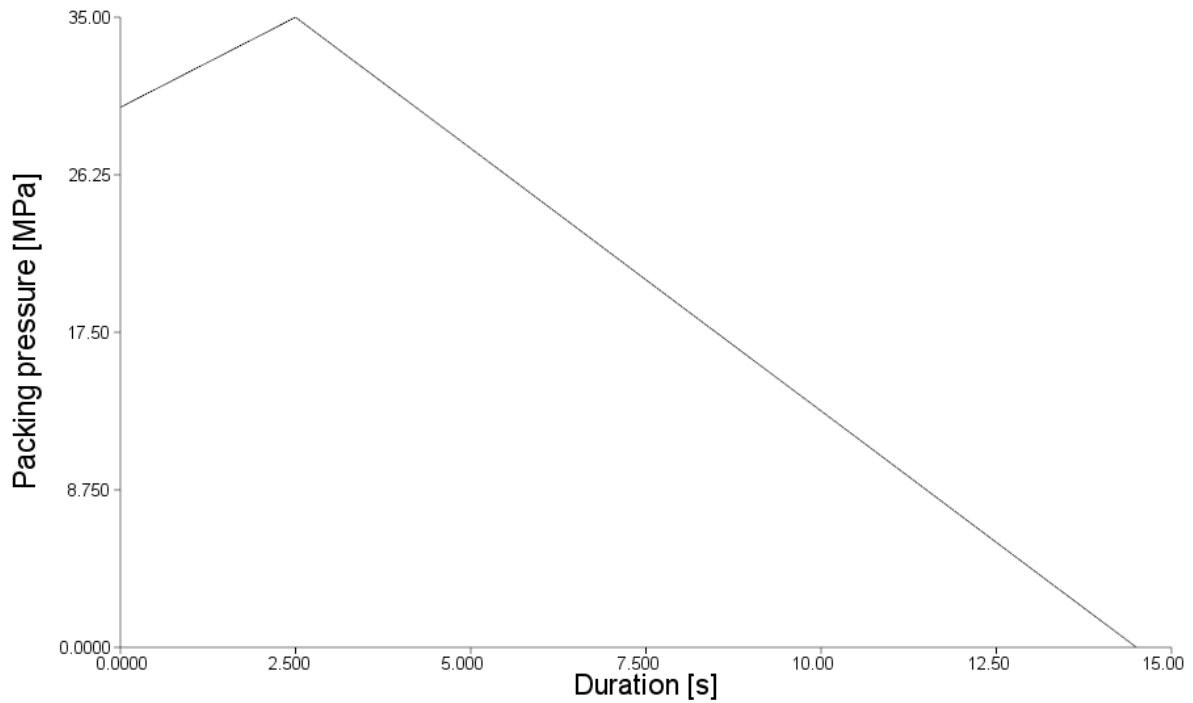


Figure 113 – Packing pressure profile of tool1.

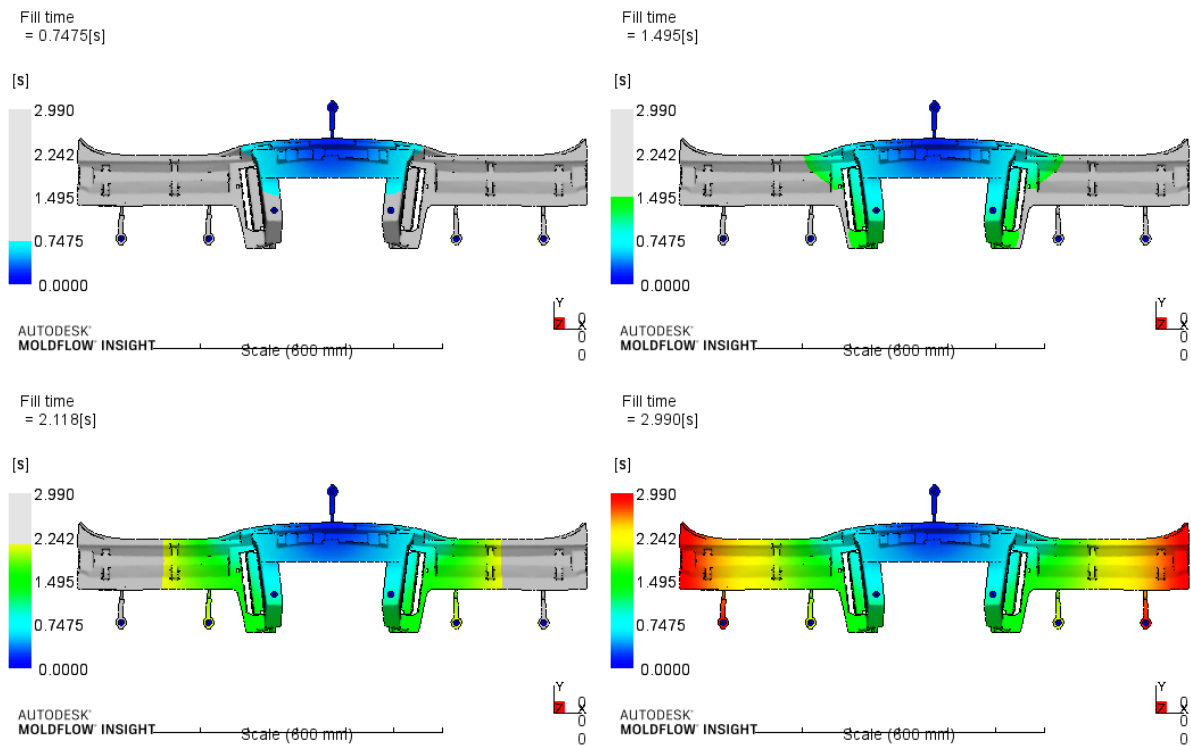


Figure 114 – Filling pattern of the plastic part from tool 2.

As for the pressure distribution which is presented in the Figure 115, is possible to observe that during the filling stage the higher pressures are present at the center of the part. This is normal since this is the first region to be filled and since the first gate to inject polymer in the tool is open since the beginning of the injection up to the end of the packing, so the pressure buildup is normal within the increase of polymer inside the tool. But when the plastic part is fully filled, the pressure tends to get more

homogenous through the part.

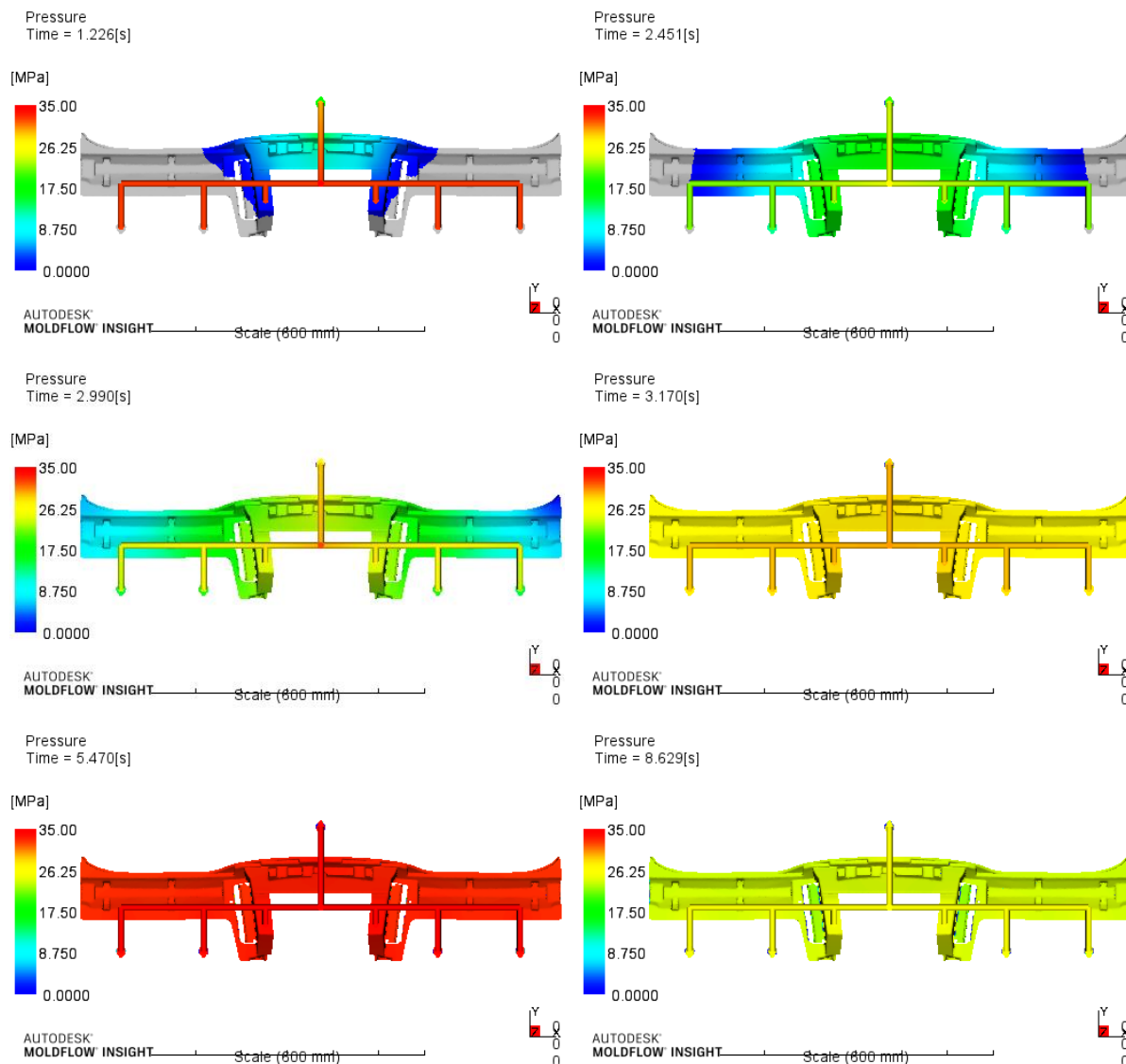


Figure 115 – Pressure distribution during the part injection cycle of tool 2.

At the packing stage is when the maximum pressure is reached. At this stage is when is expected the maximum stresses in the tool. But the frozen layer of the part is higher because at the moment that the polymer touches the steel the polymer starts to create a solid layer that will increase its thickness through time. So, having this into account this layer will also create resistance to the tool components displacement, and this is one of the reasons why the plastic part might not present defects caused by the theoretical increase of the needed clamp force, because due to the frozen layer the polymer can't infiltrate any gap caused by the injection pressure buildup. The only consequence is the increase of the part thickness.

When the frozen layer gets higher the pressure will decrease up the stage that the frozen layer reaches 100% of the part thickness and the pressure on the tool walls at those regions will disappear up to the point that the entire part is frozen. At that stage the pressure in the engraving region will reach 0 MPa.

In Figure 116 is possible to check how the pressure in some regions reaches 0 MPa even before the packing pressure is released. In this plot is only presented the pressures at some critical locations for half of the part, because is expected to have the similar values on the other half since the part is symmetric.

Other information of interest which can be obtained by the pressure plot is the pressure variations during the filling stage, the pressure drops through this stage are caused by the opening of the valve gates which decreases the effort to fill the part. This is the reason why the study of the tool stresses during the filling is so important because this can help to obtain data needed to reduce the stresses of the tool during the design phase.

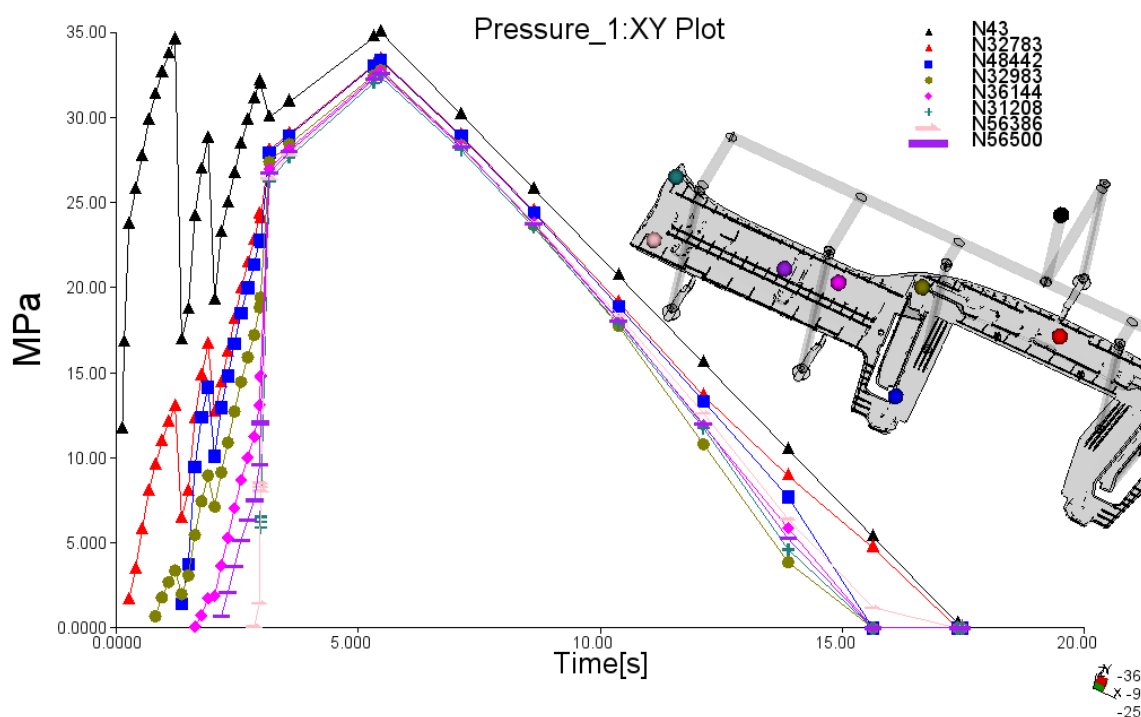


Figure 116 – Pressure at defined locations through time. The black line is the pressure at the injection tool entrance, the others are at important regions at the engraving region.

As for the tool 1, the time steps used for the structural simulation were chosen so it is possible to gather the maximum information on the tool deformations and stresses through the injection cycle as possible. These instants are represented in Figure 117 by the dashed vertical lines as well as the injection pressure, i.e., the pressure at the manifold inlet, which is represented by the blue line. For the clamping force, the simulation has determined that at least 500 Ton will be required, i.e., 4.91×10^6 N to sustain the pressure inside the tool, value that must be multiplied by 1.2 as previously mentioned to be used in the numerical simulation.

As mentioned, the application already presented in this work will be used to obtain the files with the pressures to be used in the structural simulations.

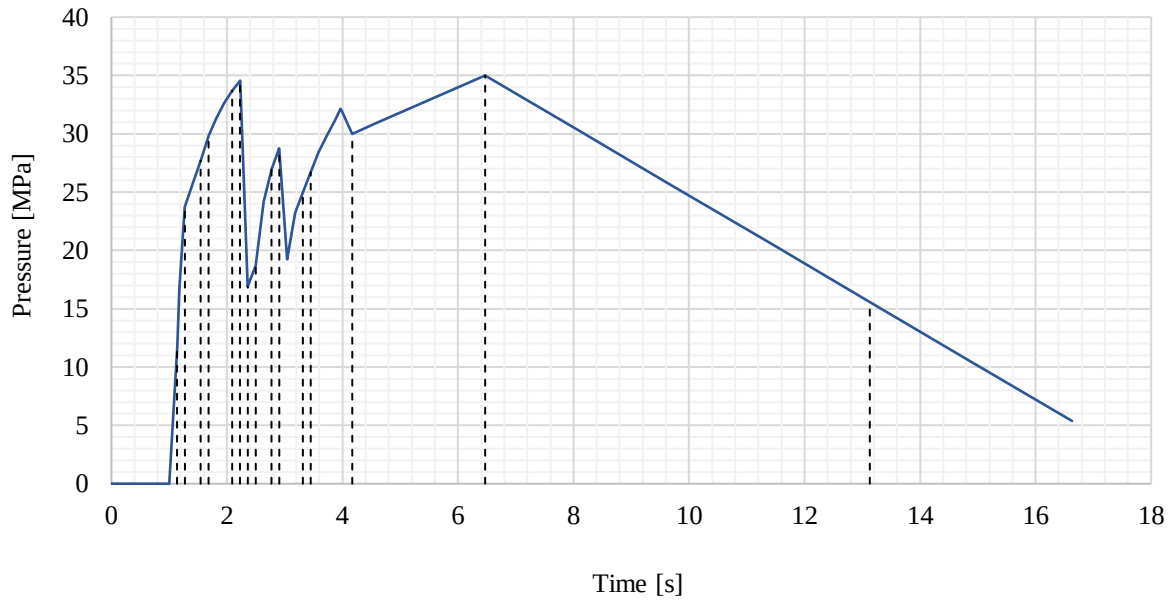


Figure 117 - Injection pressure curve and the dashed lines represent the time-steps used on the structural simulations of the tool 2.

5.2 Numerical models for the structural analysis

According to the diagram in Figure 100, four simulations are performed for each injection mold. As there is one model with few simplifications and another completely simplified, from this, besides understanding how simulations in quasi-static and transient regimes can affect the results. It is also possible to understand how geometric simplifications can influence the results and in what way. Furthermore, this approach aims to analyze to what extent the models can be simplified and in what situations transient simulations, which are more time-consuming, are necessary.

5.2.1 Structural numerical analysis from tool 1

In order to perform structural analysis of an injection mold, it is necessary to understand the boundary conditions that will be applied and the reasons for their use. In Figure 118 are presented the boundary conditions used in the analysis of tool 1. Firstly, for the study of this mold is only considered half of the model because the injection mold is symmetric in the plane displayed in the Figure 118, with this it is intended to reduce the computation time because being the model symmetric it is expected that the results are also in this plane. Next, for the analysis it is considered that the injection mold plates are undeformable, so in the conditions that are applied to the clamping plates of the injection mold this restriction is applied. Finally, on one side of the mold the clamping force that is applied by the injection machine is applied and on the other side a restriction is applied to the stay plate that prevents translation in all directions as well as rotation in all axes.

The application of pressure will be carried out in the molding regions of the injection mold, for which the pressure results obtained from the rheological study at the times defined in Figure 111 are used.

In this model the extractors will be considered for two reasons, the first is the molding area they have

which is significant when compared to the molding region. Secondly, a Kistler 9204B indirect pressure sensor is applied to the base of one of the extractors as shown in Figure 119. Since for the extractor plates is assumed that they don't to deform or move during injection, a restriction is applied at the base of the extractors that prevents their translation in all axes and rotation around all axes.

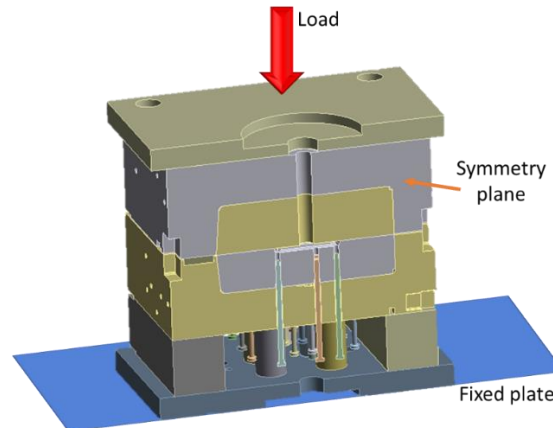


Figure 118 – Boundary conditions for the analysis of tool 1.

For the clamping force applied to the injection mold the value obtained by Moldflow with an increment of 20% is considered, this increment is a value recommended by the literature [42] in order to compensate factors that are disregarded in the rheological analysis. As for the steels used in the injection mold construction, the molding components were built in EN 1.2311, the structural components in EN 1.1730 and the pressure and adjustment plates in EN 1.2312. The data sheets for these steels can be found in Appendix A.

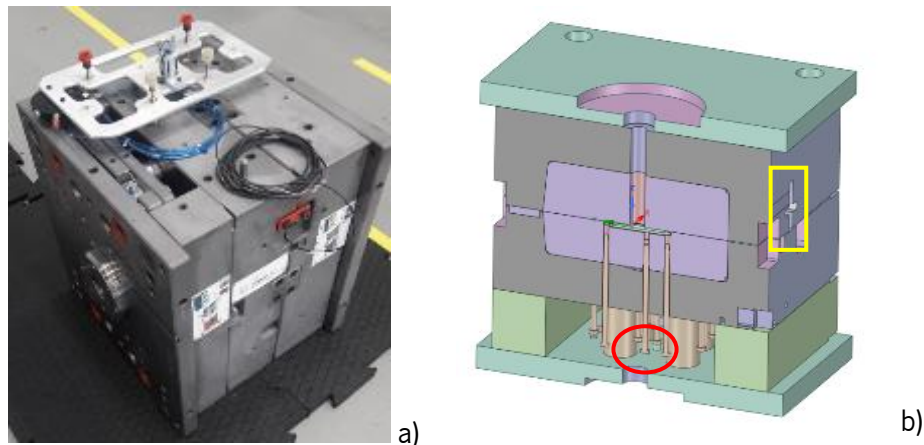


Figure 119 – Injection tool under study a) photo of the tool, b) model under study where the red highlighted region is the position of an indirect pressure sensor and the yellow highlighted region marks the position of the Inductive length measuring probe.

Taking into account all the factors described above, will be presented the results obtained from the performed structural numerical simulations. Here, for each analysis, the results obtained in the four simulations are presented in order to understand the influence of all the factors under study. In a first approach to the results, the injection mold deformations will be analyzed. These results are presented for the four simulations in Figure 120. From the analysis of the deformations, firstly it is noticeable that

the differences between the results obtained by the simulations in quasi-static regime and in transient regime are almost imperceptible. Also, in a first analysis it is quite noticeable the influence of the simplifications of the geometry in the results obtained, as can be observed in the geometry, the loads pass in a more uniform way that ends up having influence on the mold deformations. There is also another aspect that has influence and is related to the geometric simplifications that are the adjustment and pressure plates, because these are inside machined boxes, are also made in an alloy different from the others that besides having a greater hardness also have mechanical properties such as the young's modulus and Poisson's ratio. Although these differences are small, taking into account that deformations in the order of 10^{-2} mm can lead to quality problems in the injected part. In addition to the mesh size the proper characterization of the materials is important, because joining similar materials for simulations can translate into an oversimplification of the problem.

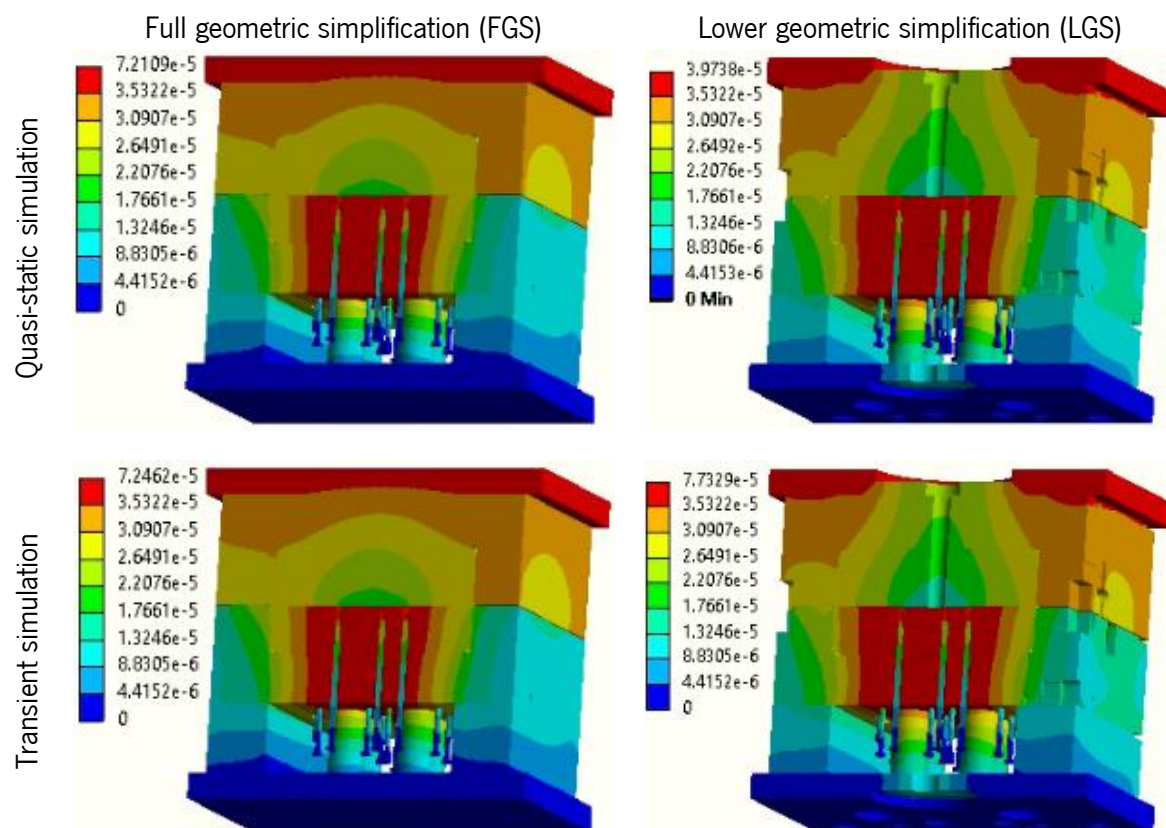


Figure 120 – Injection mold deformations in mm at 1.5329 seconds after the beginning of the injection.

If the results of the mold deformations are analyzed in detail, it is verified that the plate where the load is applied deforms uniformly as would be expected considering the restrictions placed where it is admitted that the injection machine plates are undeformable. it is also verified that on the core side, there is a large deformation due to the lack of support of the steel that is being loaded by the injection pressure, here the differences caused by the geometric simplifications are less significant because the

action of the injection pressure is significantly higher than the actions that are caused by the geometric discontinuities.

When Figure 121 is analyzed it can be observed that just as for the conclusions drawn in the analysis of Figure 120, the stress distribution and stress value is influenced more by the geometric simplifications than by the regime in which the simulation is performed. But analyzing Figure 121 one can easily understand how the stresses applied to the mold are distributed by the components and with this it is possible to have a notion of which regions should be further strengthened or not in order to better control the deformations of the mold. For example, the supports on the core side may not be properly distributed, and the combined stress and strain results help you understand how you can use these components to restrict the freedom of movement of the core components.

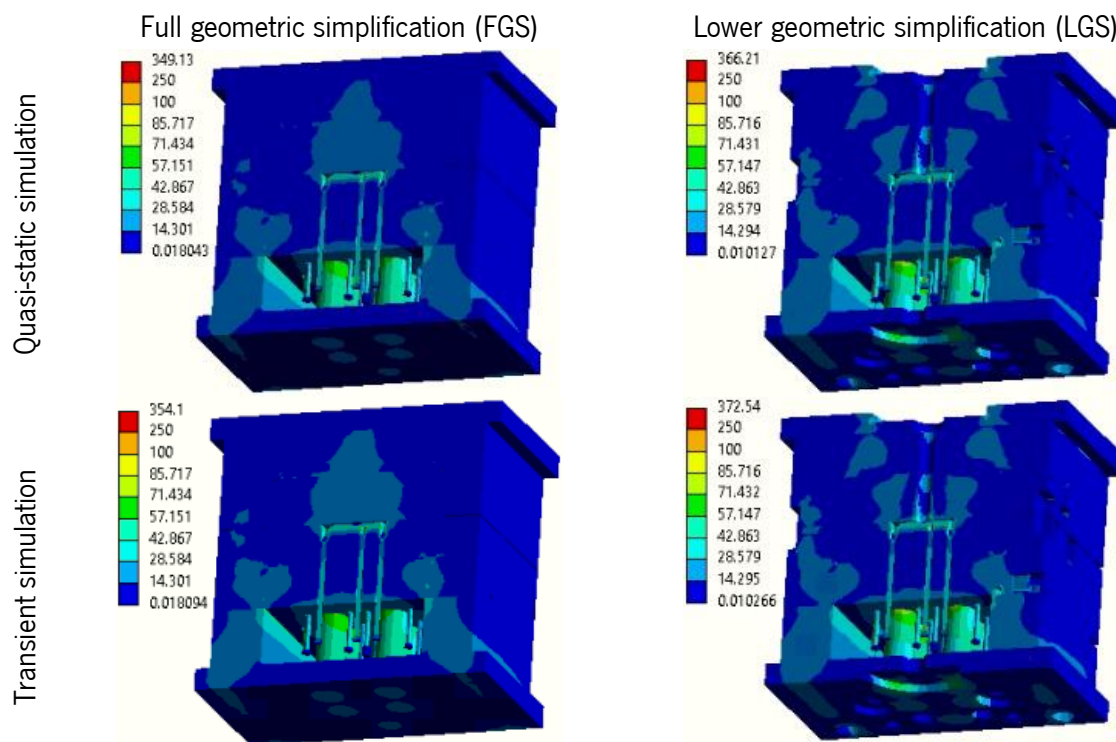


Figure 121 – Stresses [MPa] distribution at 1.5329 seconds after the beginning of the injection.

To get a further insight into the differences between the simulations performed at different regimes and with different levels of geometric simplification, four points were defined where two are located on the cavity insert and two on the core side, these points are represented in Figure 122.

In the graphics that show the evolution of the displacements in the demolding direction throughout the injection cycle, where the values of point N1 are represented in Figure 123, point N2 in Figure 124, in Figure 125 the point N3 and finally in Figure 126 point N4. By analyzing the curves, it can be seen that the conclusions drawn regarding the performance of simulations in transient or quasi-static regime for this injection mold remain valid, i.e., the results are very close throughout the injection cycle although

in the discharge phase of the injection pressure it is already possible to verify some differences. Therefore, taking into account that the unloading phase is not very relevant during the analysis because all stress peaks have already been passed. And since structural integrity is not compromised, and at this stage the part is partially or completely cooled, i.e., there is no risk of small deformations in the mold causing defects in the part. Also, with the unloading, the mold tends to its initial position so that no problems can be observed.

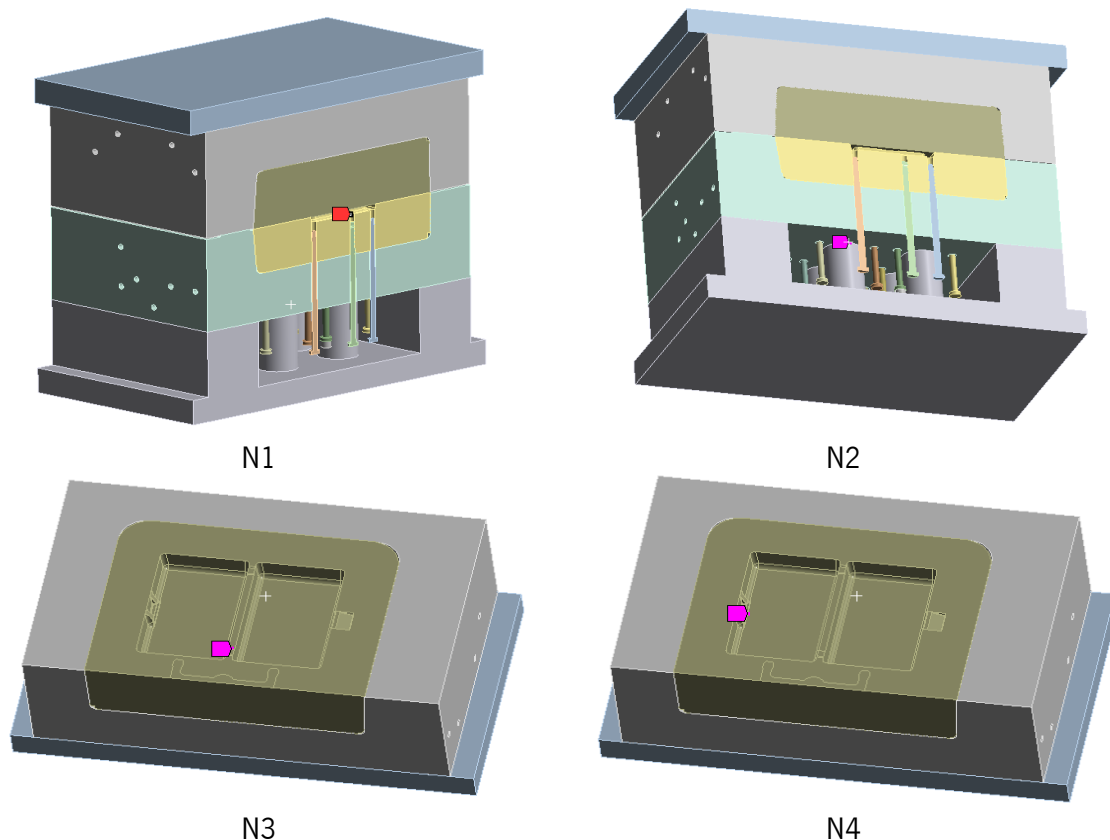


Figure 122 – Locations chosen to perform the comparison through the injection cycle.

When comparing the results obtained for the simulations with different levels of geometric simplification of the model, there is a significant difference in the results. With the exception of point N2 where the differences are already present in the first instants when only the clamping force is applied, for the other points that are located in the molding region it appears that when only the clamping force is applied the differences are not very significant, but as the injection pressure becomes more considerable in the efforts exerted in the mold the values obtained end up having significant differences.

Starting with point N2, since it has a slightly different behavior from the others that can be seen in Figure 124. This point that is located in a support of the mold structure, which has the function of supporting the most unsupported region of the mold due to the space needed for the extraction plates. The mold structure ends up being requested during the entire injection cycle, because at the beginning it directly supports the load applied by the injection machine, i.e., the clamping force and then it has to

support the efforts brought by the elastic deformation of the molding components caused by the injection pressure. For that reason, the LGS model shows values approximately 10% higher than for the FGS when the clamping force is applied, and 14% higher at the instant of highest pressure inside the injection mold.

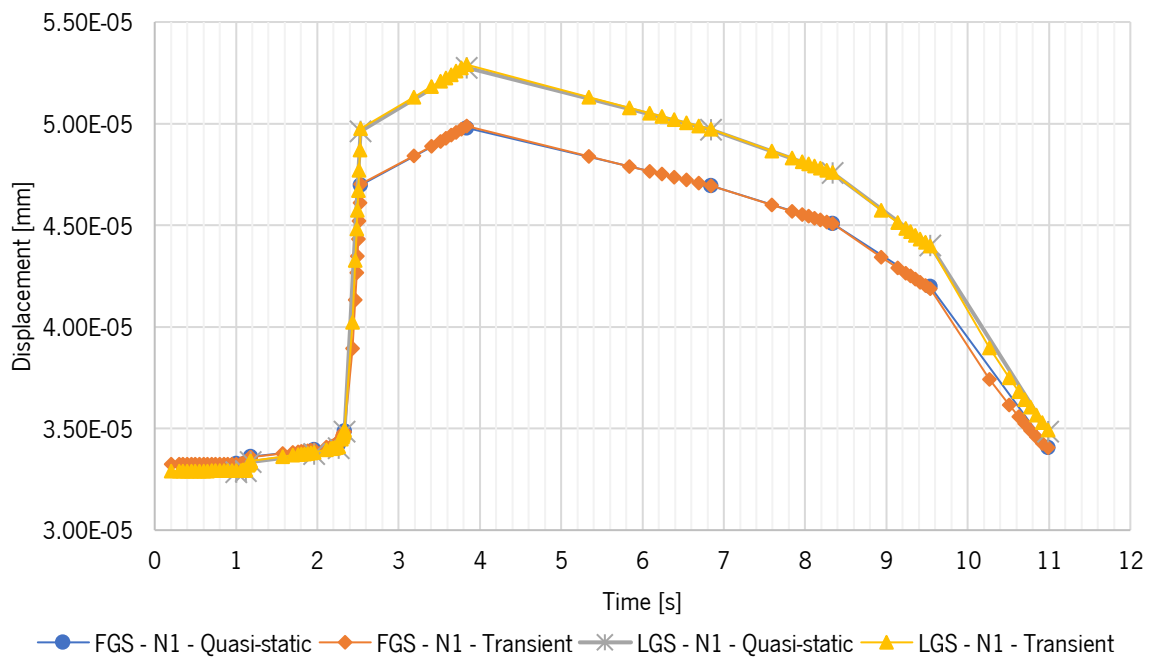


Figure 123 – Displacement through the injection cycle of the point N1.

For points N1 and N3 it can be observed that until the beginning of injection, i.e., when only the clamping force is applied, the values are very close because these points are in locations where the clamping force has less influence. But, as the injection pressure increases, the differences in values between the simulations with LGS and FGS increase. Therefore, at the instant when only the clamping force is applied the difference in results is less than 2%. However, as the pressure inside the molding zone increases, the differences become more evident although for point N3 the difference is significantly higher, since for point N1 the difference is on the order of 5% where the LGS simulation presents higher values, but for point N3 this difference is already on the order of 37%, where, as for point N1 the LGS simulation presents a greater displacement.

Point N4 was chosen because it is a very stressed region in terms of von Mises stresses, as can be observed in Figure 127, but at the same time it is far away from the partition plane which reduces the influence caused by the clamping force and also because in the molding region the simplifications are very reduced or even none. When observing the displacement curves shown in Figure 126 that correspond to point N4, it is verified that the differences along the filling cycle are less than 3.5%, which demonstrates that in a zone where the load with most influence is the injection pressure because the clamping force has very little influence in this region, leads the level of geometric simplification to be less relevant.

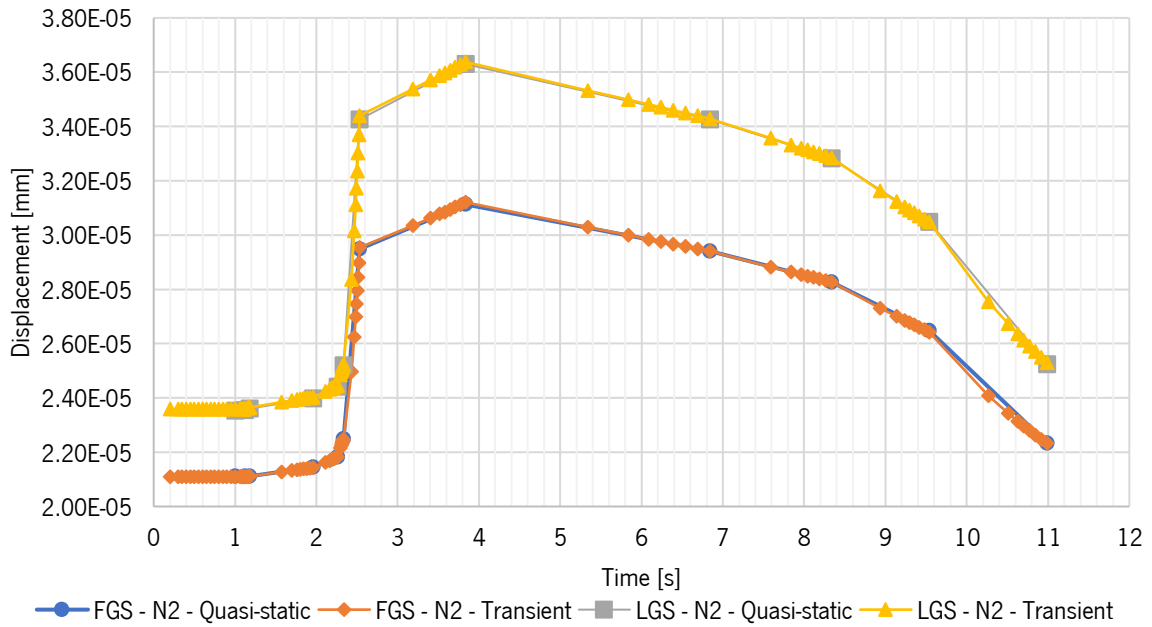


Figure 124 – Displacement through the injection cycle of the point N2.

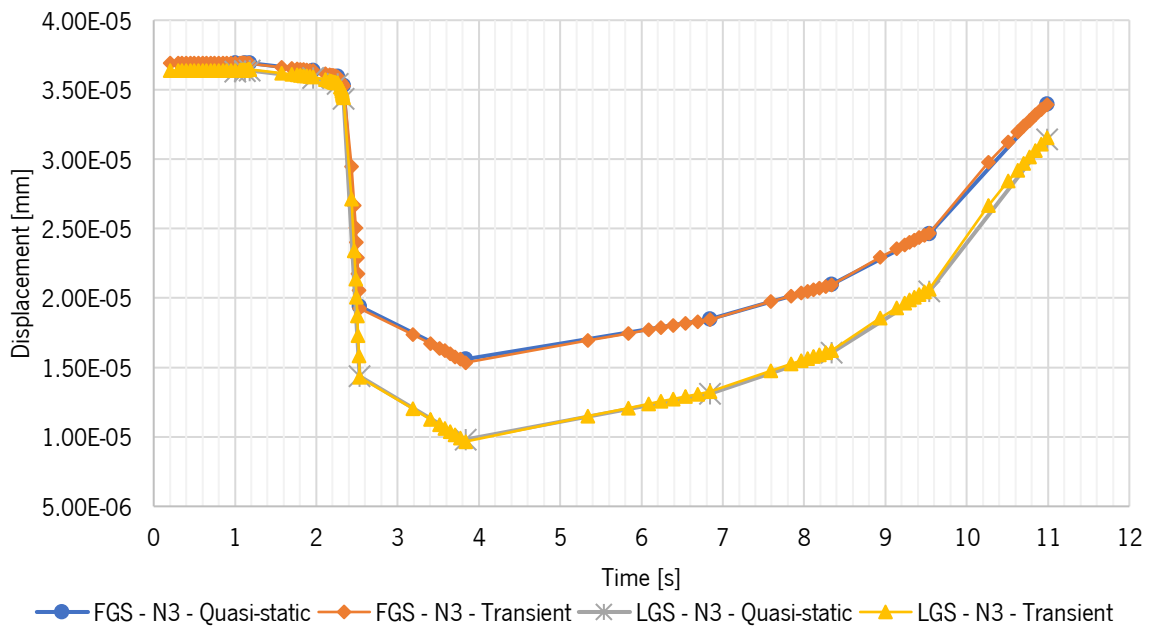


Figure 125 – Displacement through the injection cycle of the point N3.

Considering the comparative results between the simulations in transient and quasi-static regimes, for the results that will be presented only the quasi-static simulations will be considered, and also because the intention is to demonstrate differences in results caused by geometric simplifications.

As can be seen in Figure 127, the stress distribution in the simulations with FGS and LGS although globally it looks similar, in a more careful analysis is noticeable that there are several differences. Besides those already presented in the displacements, which are directly proportional to the stresses it is also verified that it cannot be admitted that for the simulation with LGS that the stresses are always

higher.

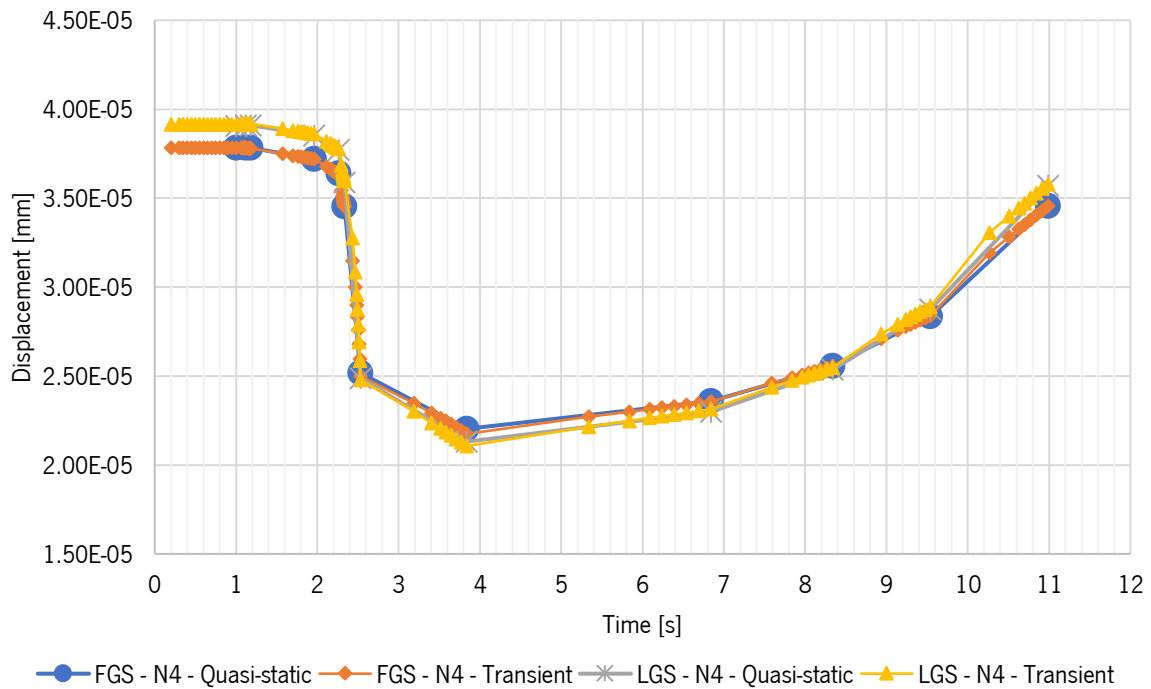


Figure 126 – Displacement through the injection cycle of the point N4.

As mentioned before, the clamping force has an influence on the results, and since the distribution of the clamping force is the same in all simulations, it is necessary to consider the effects of simplifications. For when making oversimplifications in the geometry, it is found that the passage of the pressure caused by the clamping force in the partition plane is greater and with a more homogeneous distribution. As shown in Figure 128, in the FGS model the pressure plate or the fitting plane is joined to the steel block, i.e., it is not taken into account that they are made from steels with slightly different properties, and in the LGS model the pressure plate has clearance all around it, therefore, with the tons of force not applied in the mold, it has freedom to make the barrel effect, which in the FGS model is highly restricted. The combination of these two factors ends up influencing the results and, together with the simplifications in the injection mold walls, the mold with FGS is slightly more robust than the LGS and when small deformations can lead to problems in the injected component, this type of simplification must be done very rigorously to reduce its influence on the results.

Because the pressure that is exerted during the injection process in the mold will counteract the effect of the clamping force and thereby push the molding surfaces so that the core and cavity sides tend to move apart. Even if the clamping force is sufficient to keep the injection mold globally closed, the elastic deformations of the steel can lead to a gap in the partition plane near the molding regions, and this gap should be studied because if it exceeds a limit value that depends on the injected material, it can lead to changes in the design.

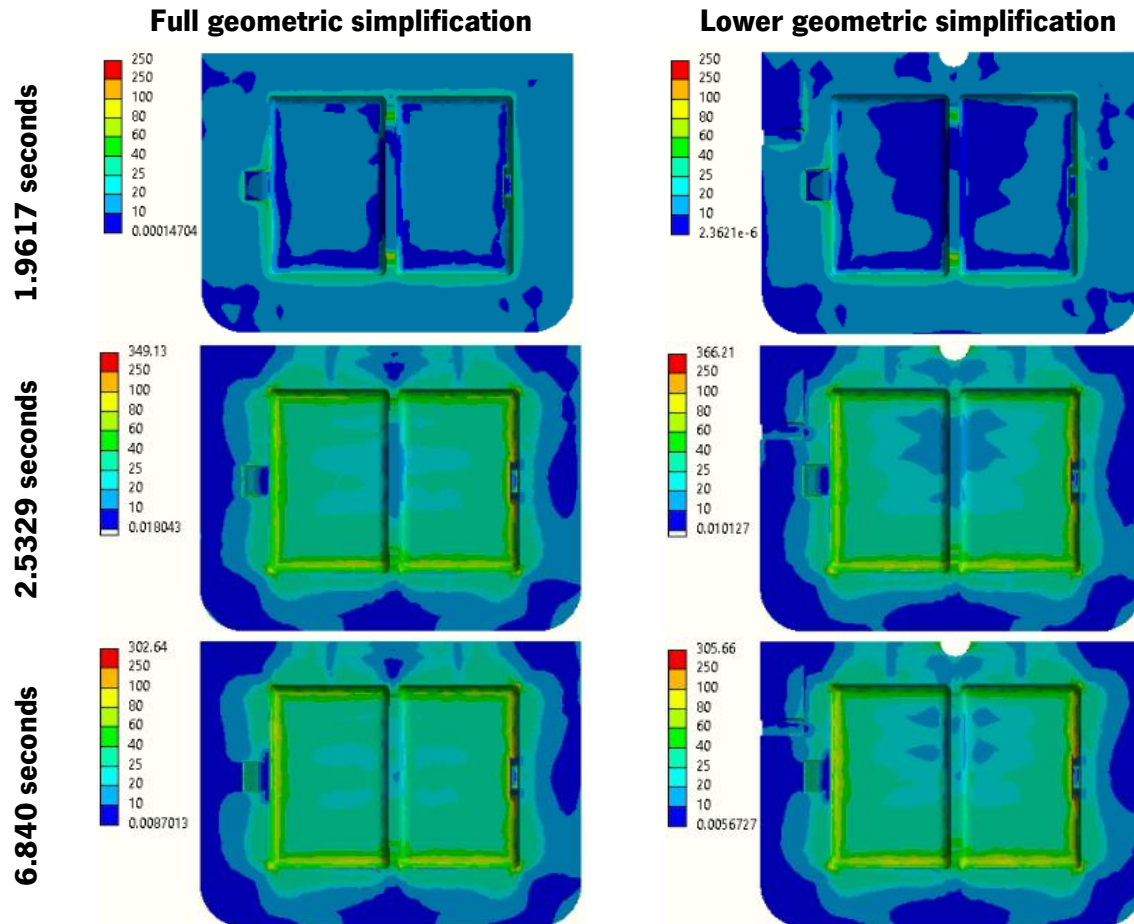


Figure 127 – von Mises stress distribution in the cavity insert through three different instants of the injection cycle.

Therefore, the simplifications in geometry also lead to significantly different values as can be seen in Figure 129, because when studying the gap between the side of the core and the cavity, one is looking for values on the order of 10^{-2} so the simplifications can have a significative impact. As can be observed in the simulation results for this mold shown in Figure 129, the numerical model with LGS ends up presenting globally higher gap values than with FGS with the increase of pressure in the molding region.

Therefore, considering all the results evaluated so far, it can be concluded that simplifications to the geometry are necessary to reduce computation times, but it is necessary to take into account that excessive simplifications can lead to a misrepresentation of the results and thus not observing existing problems.

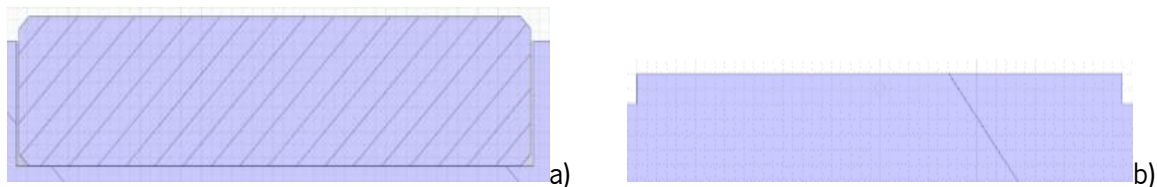


Figure 128 – Detail of a pressure plate with a) LGS where its pocket is present and with b) FGS where the pressure plate is merged to the core.

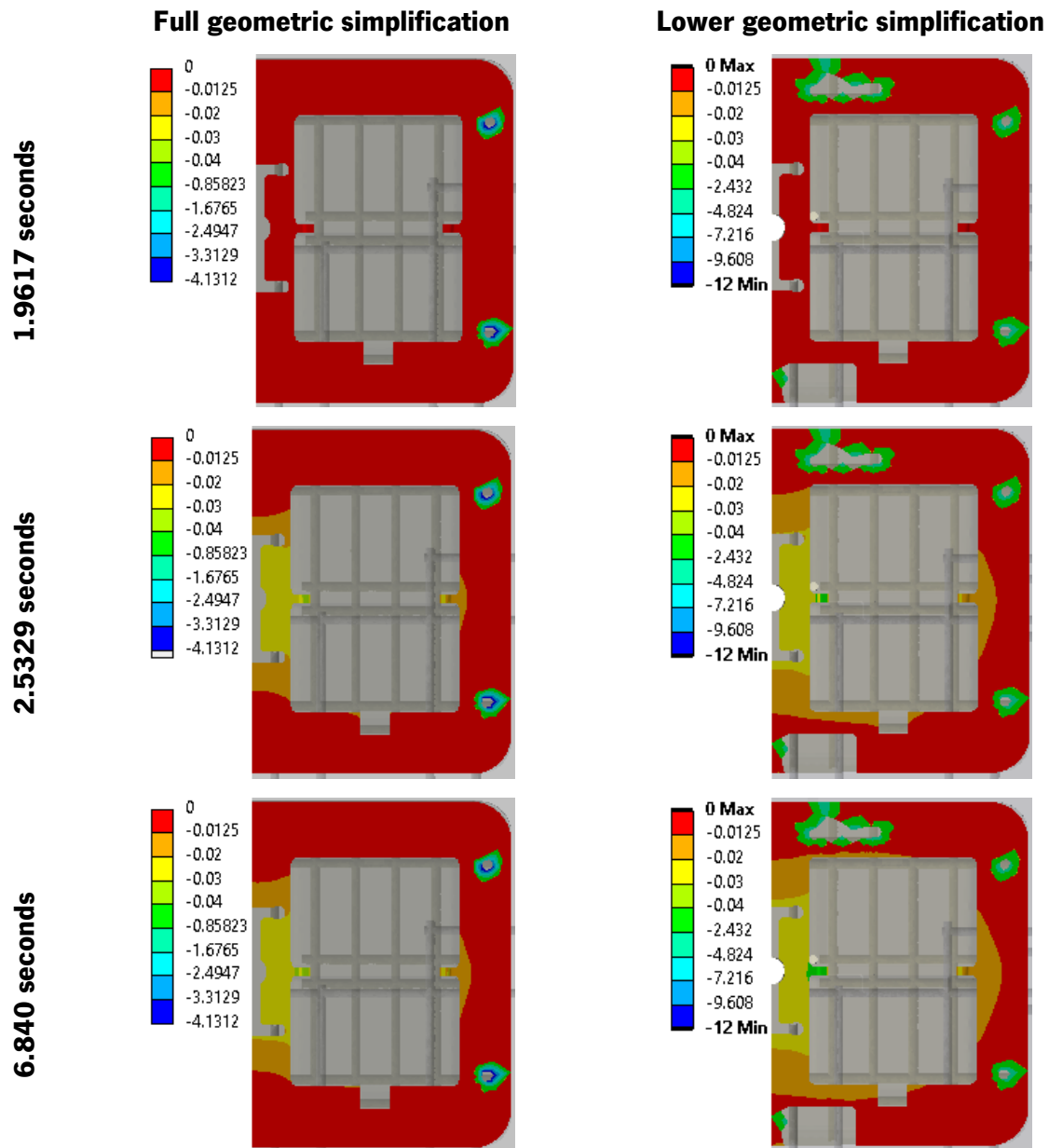


Figure 129 – Gap between the core and cavity in the adjustment zone.

5.2.2 Structural numerical analysis from tool 2

The structural study of tool 2 has some differences from that of tool 1. As can be observed in Figure 130, for this study no symmetry condition was considered, although the part is symmetrical. This is made to understand how the grooves milled in the manifold plate will influence the deformation of the injection mold.

Another point that differs from tool 1 is the application of the clamping force applied by the injection machine to the mold. Here this is applied on the core side, although the results are identical because the injection machine plates are considered undeformable. In this model only one ejector shaft is considered where the sensor is placed in contact with the mold clamping plate when the extraction plate is retracted. With this there are no convergence problems in the numerical model, something that tends

to happen in the transient simulations of tool 1.

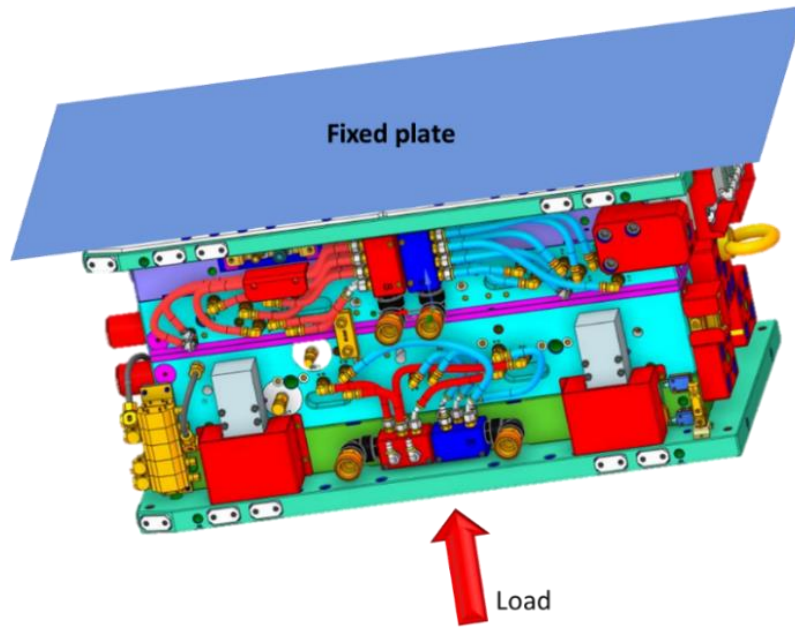


Figure 130 – Boundary conditions for the analysis of tool 2.

Also, in the mold clamping plate where the clamping force is applied, a restriction has been applied to the clamping plate whereby movements in the directions parallel to the plate are blocked and the rotation in all axes is also restrained. As for the pressure exerted by the polymer, the same conditions will be used as for tool 1. And for the value of the mold clamping force as for tool 1, 20% will be added to the value obtained in the rheological study of the plastic part. The same steels were used in the construction of this injection mold as were used in the construction of tool 1.

In this mold, sensors will also be placed in order to collect data that can be compared with that obtained from the studies using numerical simulation software. The same sensors will be used as in tool 1 and their location in the tool is shown in Figure 131. Although for the study of this injection mold more inductive length measuring probes will be placed as shown in Figure 131 b).

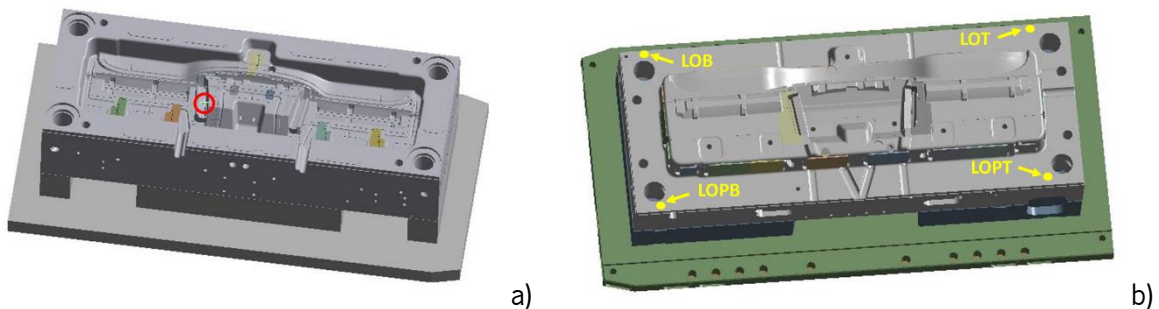


Figure 131 – Injection tool under study a) tool core side where the red highlighted region is the position of an indirect pressure sensor and b) is the tool cavity side where the yellow highlighted regions mark the position of the Inductive length measuring probes.

To compare the effects caused by the geometric simplifications and the regime used for the simulations, the injection mold deformations and von Mises stresses will be analyzed. In order to be able to verify the behavior at the middle of the injection mold, it will be splitted by the symmetry plane of the

molding region. Starting with the deformations plot, Figure 132 shows the injection mold strains at the instant when the highest pressure is applied to the molding region, that is, at 6.47 seconds after the start of injection. To facilitate the visualization of the most affected regions, the core side will always be pointed up, because it is expected that the most requested region is where the mold clamping force is applied.

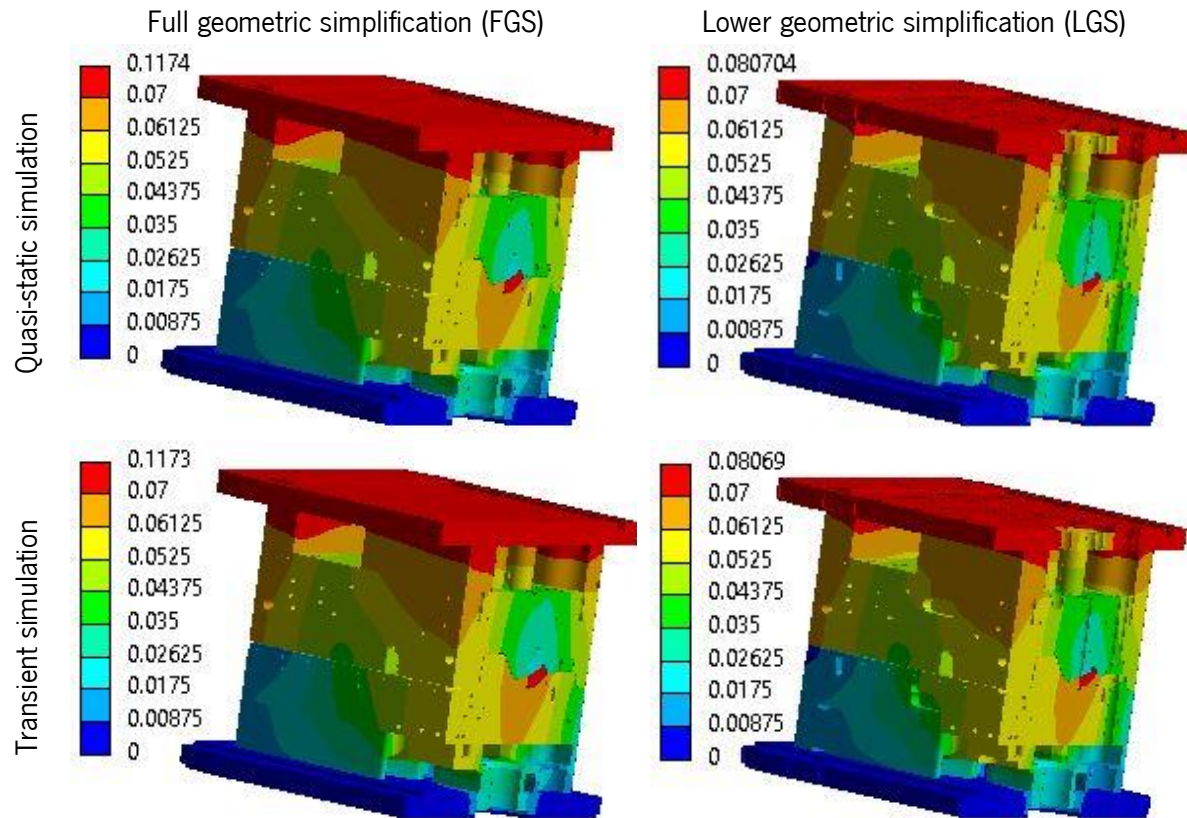


Figure 132 – Injection mold deformations in mm at 6,4701 seconds after the beginning of the injection.

By analyzing the deformations distribution, it is possible to verify that for the model where more geometrical simplifications were made, there are also more deformations of the components on the core side. It is noticeable that the region on the core side that suffers a deformation higher than 0.07 mm is larger in the model with FGS. But when a comparison is made between the numerical simulations in quasi-static and transient regimes, the observed deformations are very similar, i.e., when comparing the results obtained for the FGS and LGS models, it is verified that there are no significant differences. Therefore, the results that had been observed in tool 1 are observable again in tool 2 even though the molds are different in terms of size, applied forces and injection typology. On the cavity side, is observable that the differences between the four results obtained do not present very different values. Although with a more careful analysis, it is visible that the FGS model presents slightly higher deformations through a slightly larger region. Much of this is due to the fact that the simplifications made, increased the area where the loads are distributed. With this, the regions that by themselves are already more susceptible to presenting higher deformation values end up being even more stressed. Also, not removing all the

grooves and holes present in the model, leads to more deformations in the model, but distributed throughout all the model. For example, at the mold clamping plate on the core side there is a significant variation in deformations in the LGS model when compared to the FGS model, this is entirely due to the non-simplification of the model. Because on the LGS model the clamping plate shows a lower support at the central region and when loads are applied this region tends to deform. More examples like this are easy to find when analyzing the mold in detail.

Figure 133 shows the distribution of von Mises stresses in the mold, also at the instant when more pressure is exerted by the polymer in the molding region. Although the deformations in the mold are directly linked to the stresses, with this result it is possible to have a better view of the differences that exist between the FGS and LGS models.

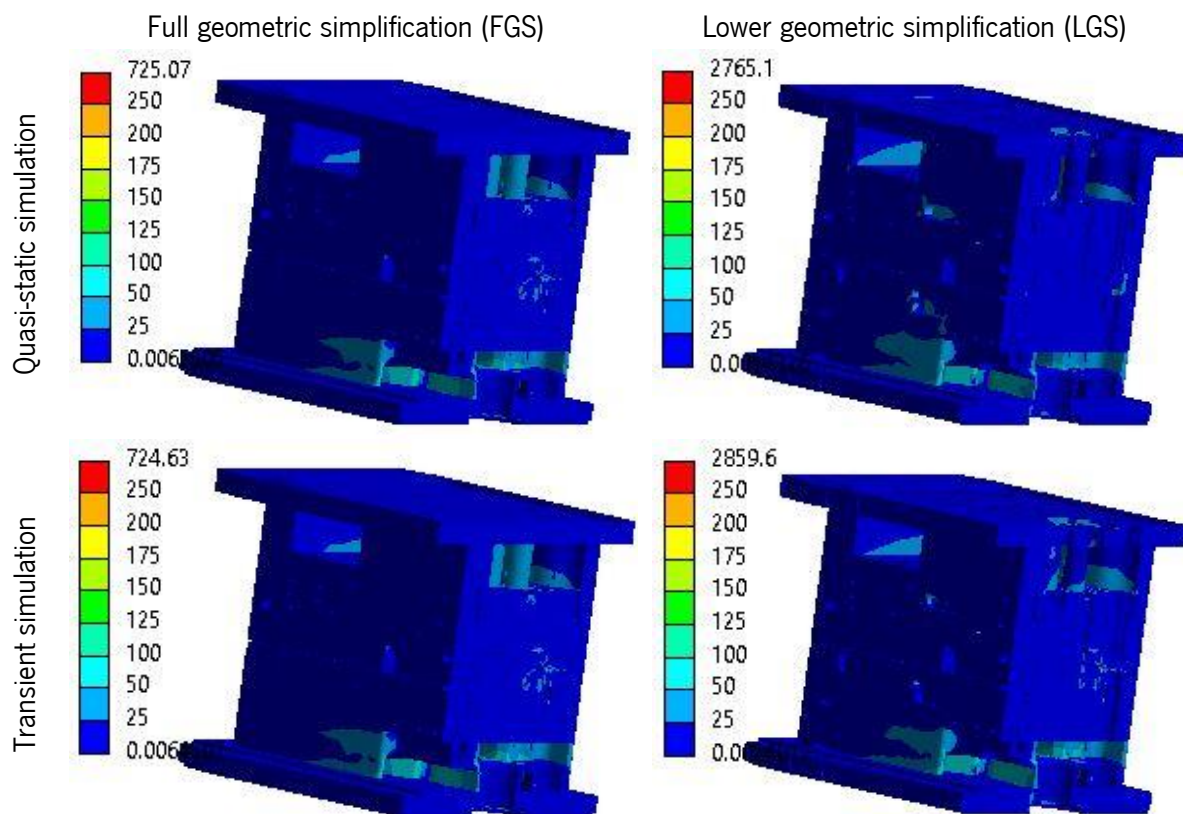


Figure 133 – Stresses [MPa] distribution at 6,4701 seconds after the beginning of the injection.

The differences in the distribution of von Mises stresses between the FGS and LGS models are noticeable. The point that stands out the most is the stresses in the supports in the central region of the mold on the core side, where the FGS model presents higher stresses. A force distributed over the entire surface is applied to the clamping plate on the core side, but since in the FGS model the clamping plate has no holes or grooves, the force is passed on to the following components more evenly than in the LGS model. Because in the LGS models there is a groove in the central area for centering the mold in the injection machine, where the clamping force is not applied, therefore the supports that are underneath

this slot are little solicited. Besides that, on the side of the cavity in the manifold plate, there is a significant slot to accommodate the injection system, so, when the injection pressure is applied, the steel of the molding components is free to move in both directions and thus distribute better the efforts suffered. Although this is beneficial at the structural level, the decrease in localized stiffness can lead to quality problems in the injected part. On the cavity side, the stress distribution is very similar between both models, although they are slightly higher in the FGS, as already observed in the distribution of deformations.

As for the differences between the models calculated in the quasi-static regime and in the transient regime, in this model it was possible to observe two distinct features. While for the model with FGS the differences between the calculations performed in quasi-static and transient regime are not negligible, for the model with LGS the same does not happen. Firstly, the way the loads are applied, when compared to tool 1, in addition to the filling time of the tool 2 part being approximately double, the pressure evolution is also different because in tool 2 there are many pressure drops during filling. Here the effects disregarded by the quasi-static simulations end up having an influence on the results mainly in the LGS model which is more shredded.

When analyzing the displacement at the points measured by the sensors, at the shown in Figure 131, but here making the comparison between the results obtained from the simulations performed at different regimes and with different levels of simplification of the model. The difference between the different curves is minimal in the first few instants, i.e., when only the clamping force is applied and at the beginning of filling the cavity with molten polymer. From the moment that the molding region begins to have enough molten polymer to exert a pressure in a sufficient large area to oppose the clamping force applied by the injection machine, the largest differences already observed between the FGS and LGS models begin to appear. It should be noted that although the differences are less than 2% at these points, this is not representative of all relevant regions of the injection mold. Furthermore, it is intended to understand how each calculation method differs from the others. By analyzing in detail, the simulations performed in transient and quasi-static regimes, it is noticeable that these present the highest differences when the time-steps of the quasi-static simulations are farthest apart. That is, as the transient simulations were programmed to ramp interpolate the loads, is possible to obtain the stresses in the mold for intermediate times. It can also be seen that these differences and this behavior is constant at all measured points, as can be seen in Figure 134 to Figure 137.

Taking into account that the differences in the time-steps chosen to perform the calculations are less than 1% and that generally simulations in the quasi-static regime tend to obtain higher and therefore more conservative values. In order to compare the differences obtained between the 4 measured points,

the curves obtained for this regime will be compared. As already mentioned, when oversimplifying the models leads to certain phenomena being disregarded, the comparison will be made with the LGS model.

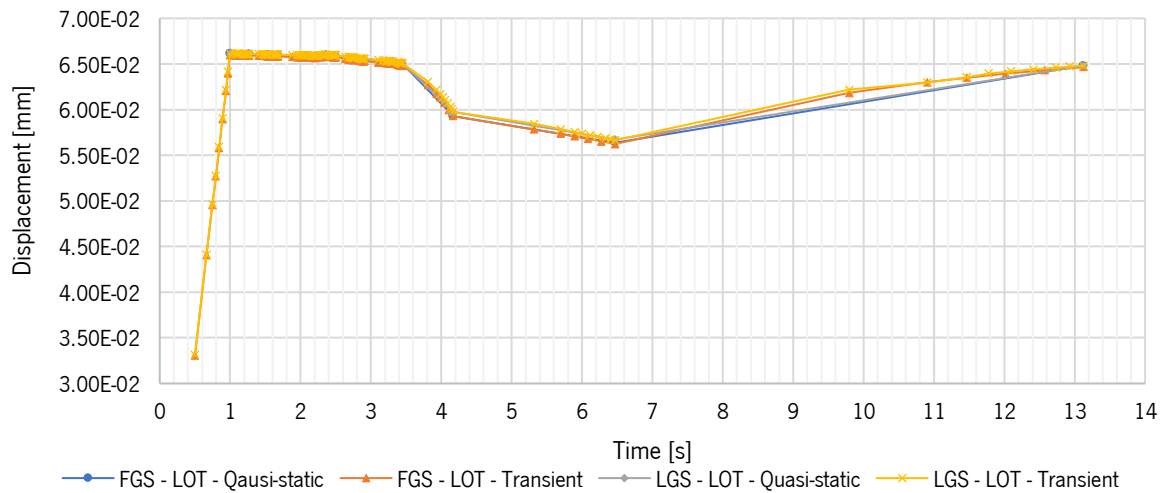


Figure 134 – Displacement through the injection cycle of the point LOT.

When analyzing Figure 138, can be observed that there is a difference of about 11% between the values obtained. That is, the four sides do not move uniformly during the applied loads. This was expected since the mold is not bi-symmetrical. But since the part is symmetrical, in the partition plane one could expect that there would be a symmetry between the points that are opposed by the symmetry plane of the part. But this does not happen, in fact the two points that show a similar deformation are the points LOT and LOPB, which leads to verify that there is no symmetrical behavior at these points. This, as already mentioned, is due to the grooves in the manifold plate that are not symmetrical and have a significant size to the point of making it impossible to consider the symmetry of the mold for study purposes.

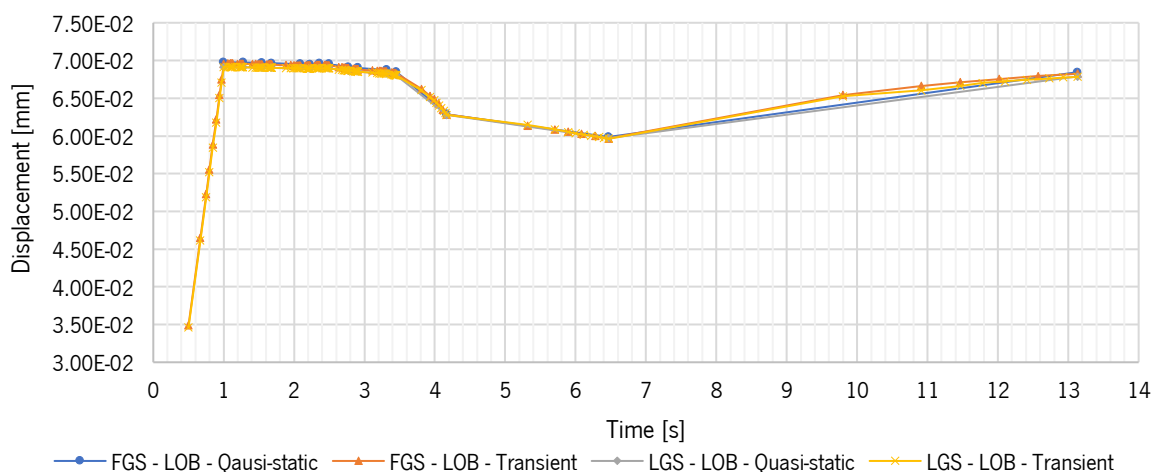


Figure 135 – Displacement through the injection cycle of the point LOB.

Therefore, at the stage when the designer is preparing the model to perform the numerical calculation he must take into account, in case he wants to simplify the model by only considering half of the mold,

that the symmetry has to be in all the mold components and not only in the molding components that are responsible for supporting most of the applied forces.

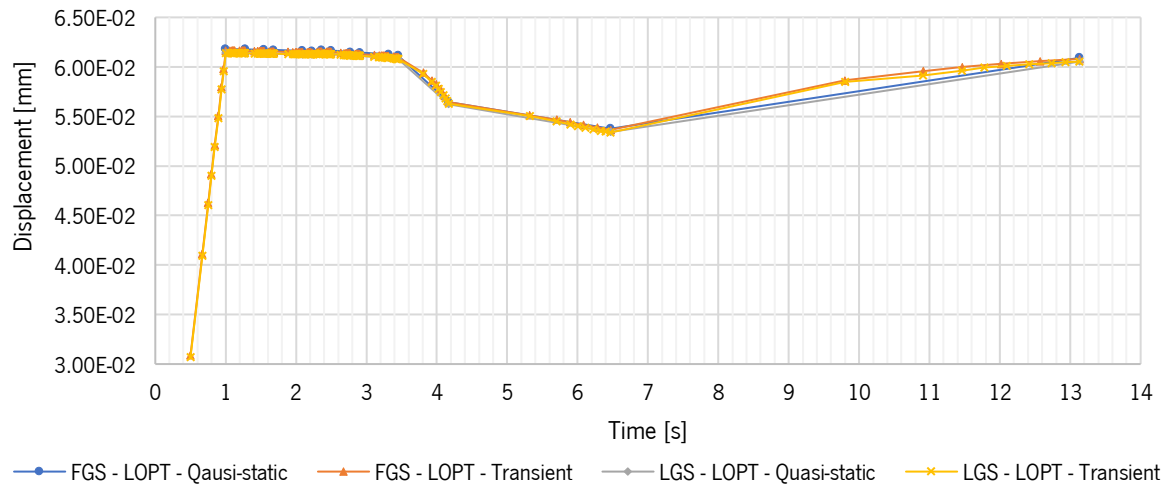


Figure 136 – Displacement through the injection cycle of the point LOPT.

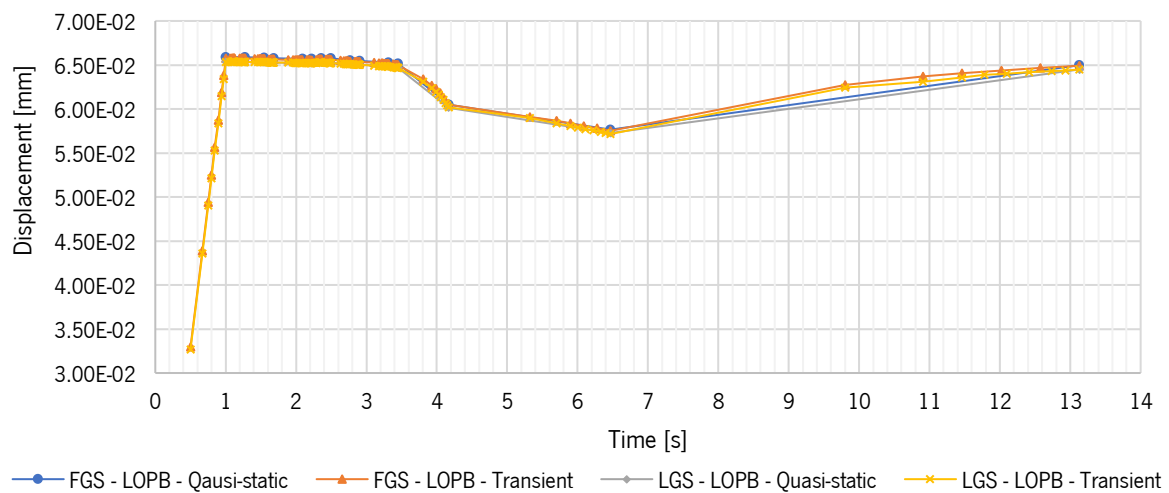


Figure 137 - Displacement through the injection cycle of the point LOPB.

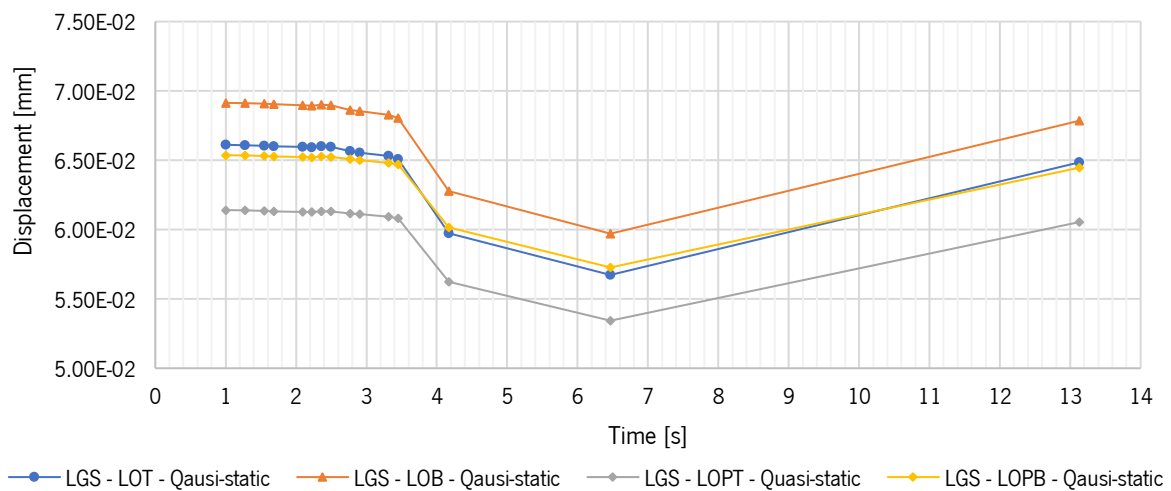


Figure 138 – Comparison of the displacement between the four locations.

5.3 Experimental results

In order to understand how the numerical simulations can approximate the injection molds behavior, this section will compare the values from numerical models with the values obtained from measurements performed on the molds. The input data for the rheological simulation was obtained from the injection mold test parameters to get the numerical results closer to the experimental ones. With this it is expected to understand how the results can differ and what the causes are. And what precautions should be considered when performing numerical studies during the injection mold design phase.

5.3.1 Experimental results from tool 1

As already mentioned, sensors have been placed in the molds to get data that is possible to compare. For tool 1, the sensors used were placed in the mold at the locations shown in Figure 119.

To perform this analysis, comparing the pressure values obtained at the base of the ejector shaft will be the first step. Which in the mold is measured from a pressure sensor Kistler 9204B. For the numerical simulation, the results are obtained from the contact pressure between the ejector base and the sensor. The results obtained are presented in Figure 139. In order to help understand how the injection pressure evolves over the cycle, the injection pressure curve obtained from Moldflow was also added. Only the quasi-static numerical model curves are being used for the analysis, because it had already been concluded that the differences between the numerical models in the quasi-static and transient regime are very small for this injection mold. In a first analysis a significant difference is noticeable between the pressure results obtained numerically and experimentally.

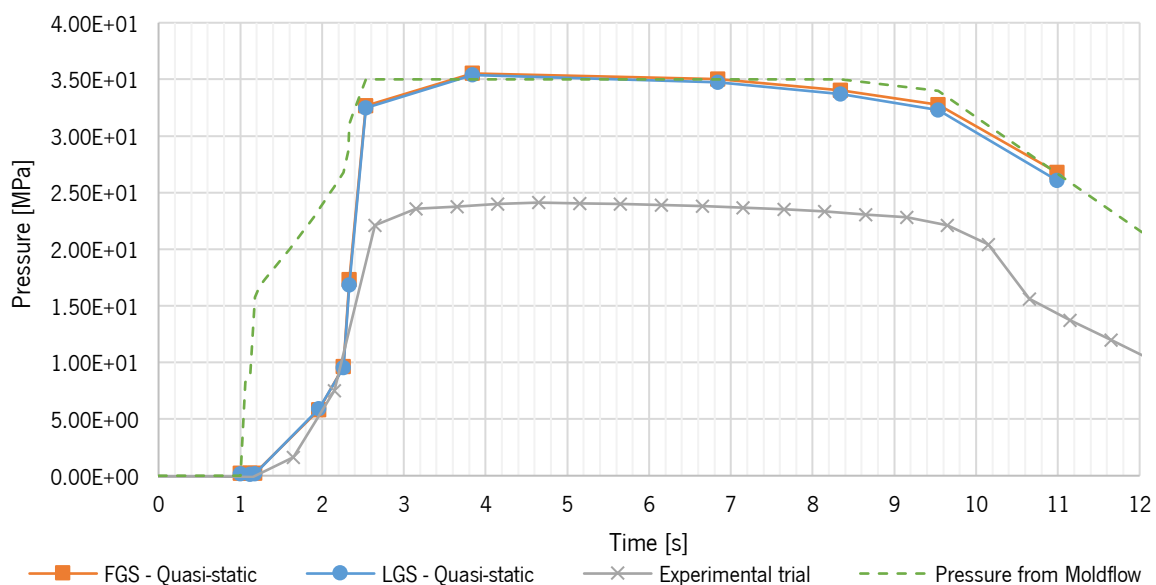


Figure 139 – Pressure at the indirect sensor, comparison between numerical and experimental data.

Although in the first instants, that is, during the filling phase of the part the discrepancies are minimal, as the pressure inside the cavity increases the discrepancies between the numerical and the experimental

models become more significant. The experimental model tends to present lower values, in fact, in the packing phase the differences are up to 35%. For this mold, the differences can be explained by two distinct reasons, where the first is related to the type of sensor and how it is assembled, while the second is related to the cooling of the injected polymer. The Kistler 9204B sensor, is an indirect pressure sensor. And shown in Figure 119, it is mounted at the base of the ejector shaft, which immediately leads to two things that must be considered. One is that being an indirect sensor, is not measuring the pressure that the polymer is applying at a point in the molding region, but rather the pressure that a molding component is exerting on its support due to the molten polymer pressure. This leads to the fact that there are factors such as shaft bending, friction caused by shaft adjustment, and the centricity of the sensor relative to the shaft that can lead to deviations from the numerical models.

As for the cooling of the polymer, it is a fact that when the molten polymer touches the steel in the molding region it immediately starts to cool. Moreover, as the polymer is at a temperature around 245°C and the steel at a temperature around 30°C, when the contact between the molten polymer and the steel happens, a solidified polymer layer is created almost immediately. This solidified polymer layer will increase over time with the heat transfer between the polymer and the cold source that will be the steel of the molding region. With the increase of the cooled polymer layer, or frozen layer, even if there is a significant pressure in the molten polymer, it will not be fully passed to the ejection shaft because of the cooled layer that will absorb some of the stress caused by the pressure.

Looking at the results for the cavity displacement relative to the core presented in Figure 140, were to measure the displacement of the cavity relative to the core in the mold is used a linear inductive position sensor (TESA GT22). The results are apparently somewhat contradictory when compared to those for pressure.

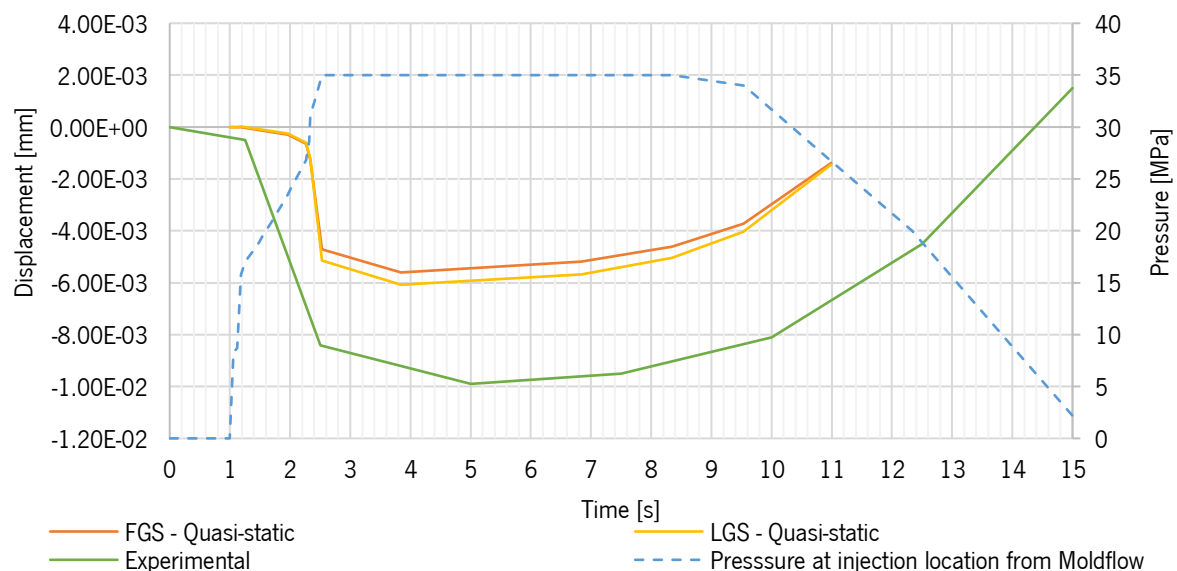


Figure 140 – Displacement at the partition plane.

When analyzing the curves with the numerical and experimental results one immediately sees that here the experimentally measured displacement is greater than the numerical one. What in a first instance seems contradictory with the results obtained for the pressure, because this was higher in the numerical model. But one must keep in mind that steel has a certain elasticity and ductility associated with it, and since pressure is not applied across the entire partition plane, it is to be expected that there is greater deformation in the region directly below and above the molding region due to the shape of the part.

Therefore, since the injection mold on the core side is not massive it ends up being more susceptible to greater deformation in the central region. When analyzing Figure 141, which shows the deformation of the mold at an instant of higher pressure within the molding region with a scale factor of $1,6 \times 10^3$. From there it is more noticeable that on the core side the mold tends to bend, so that the region supported by the rails tends to rise while the central region where the circular supports are placed, has the greatest deformation and opposite to the region supported by the rails. This happens for a very simple reason, the stiffness given to the region supported by the rails is significantly higher than the stiffness given by the cylindrical supports. Considering this, it is normal that with the pressure being essentially exerted in the region supported by elements that confer less rigidity to the core side, there is a greater tendency for the core to present a higher deformation in that region. As the rails have greater rigidity and are placed in the area further from the center of the mold, they end up working as joints and thus the mold tends to bend in the central region. And with that, the region over the rails tends to make the opposite movement. The cavity side, which tends to be massive, tends to have a more homogeneous deformation.

But the profile of the curves presented in Figure 140 for the deformation, is very identical among all, where again the LGS model continues to present a result closer to the experimental one as expected.

As for the rheological simulation, that has a direct impact on the structural simulation since it serves as the object for obtaining the loads for the structural numerical simulation. For this mold showed that the filling of the part occurs approximately 0.2 seconds faster than in the test, this represents a difference of approximately 8%. This can be derived from several factors, as in rheological simulation there are numerous factors that lead to differences between the numerical results and the experimental ones. Firstly, the characterization of the polymer, even if this is done with great accuracy in preparing it for injection, can lead to deviations from the predictions made from the numerical model. Then, the Moldflow software does not consider the effect that the injection machine can have on the injection of the casting, i.e., the input data are at the entrance of the mold. And the injection behavior can be affected by what happens in the injection machine because this is where the plasticization of the polymer takes place, which is done by the rotation of the screw. In addition, the spindle size and shape can also lead to the

fact that not all predictions made during the rheological study do not translate during the injection process. The viscoelastic behavior of polymeric material is very complex to model, and software such as Moldflow resort to simplifications that must be known by designers, simplifications that make CAE studies at this level economically feasible.

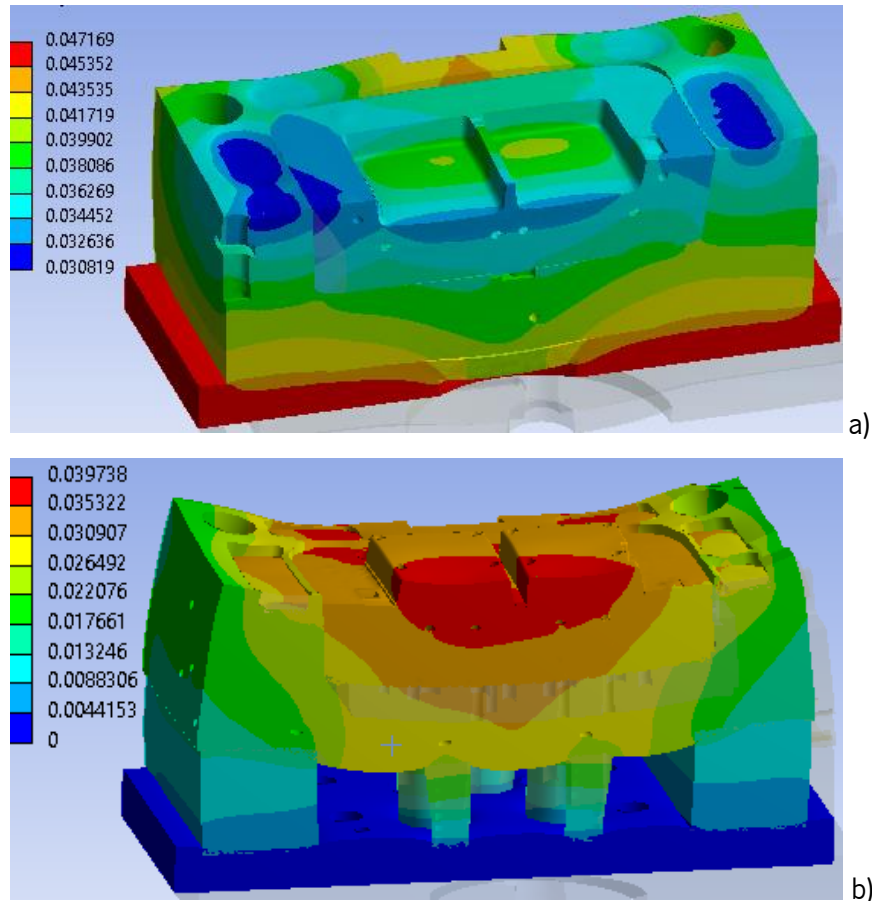


Figure 141 – Displacement in mm relative to the global coordinate system where a) is the cavity and b) the core.

As for the results obtained for the clamping force, it was verified that the value predicted in the simulation with the correction factor was able to maintain the clamped mold during the injection cycle. It was not possible to perform conclusive tests without the correction factor or with a value between 0 and 1,2.

5.3.2 Experimental results from tool 2

For tool 2, data was collected on the displacement between the core and cavity blocks in the parting plane, the value of the pressure at the base of an ejector shaft was also obtained. Therefore, the same measurements were made as for the tool 1 with the difference that for this injection mold the displacement was measured at four points, shown in Figure 131.

Figure 142 show the dislocations between the core and cavity side in the partition plane from the experimental measurements. By analyzing the experimental curves, it can be observed that the measured values differ at all measured points as observed in Figure 138 with the numerical results. Although the

LOPT point shows a significantly higher displacement than the others in the experimental results. After analysis at the test site, it was found that the movable plate of the injection machine presented a problem in the locking system, which led to a higher deformation of the injection mold in this quadrant. Therefore, the experimental results for this point will not be compared with the numerical ones because the effects of the movable plate are not considered in the simulation.

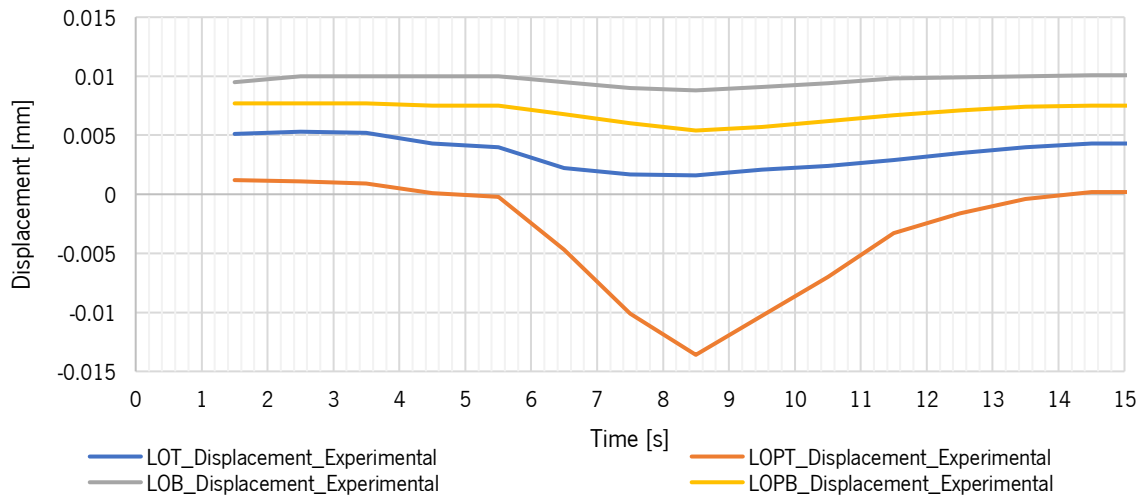


Figure 142 – Displacement at the partition plane for one injection cycle.

It is also possible to verify that for the experimental test the measured displacements at each point are distinct as predicted in the numerical results. But here the results are significantly lower than those obtained from the numerical models. It should also be noted that in the numerical model the curves have a very similar profile, only varying in value. In the experimental curves this is not so evident, although it is noted that the largest amplitude occurs at the instant of highest pressure within the molding region just as in the numerical model. Here the curves have a slightly different profile between them. For example, it is evident that at the LOT point the tendency for the deformation to increase begins earlier than at the other points (discarding the results from the LOPT point).

The deformation of the injection molding machine plates is a factor that will influence the results, as can be seen in studies conducted, such as the work presented by Carpenter, et. al. [24] where it is possible to verify how the injection machine can influence the mold deformations. But in this work, it is intended to perform the numerical studies as if it were in a design phase. Therefore, at this stage this information is not available, so it is assumed that the plates of the machine are undeformable. To compare the deformations, and as already done for tool 1, the deformation value will be zeroed. Where it is assumed that at the moment the clamping force is fully applied the deformation is zero. With this it is intended to understand how the injection mold deforms during the injection and packing phases.

By analyzing Figure 143, it is noticeable that in the numerically obtained results there is a faster response than in the experimental test. As already mentioned, the structural simulation relies on the

rheological simulation to obtain the loads. Therefore, as for tool 1, it can be seen that the rheological simulation tends to predict cycle times shorter than those obtained experimentally.

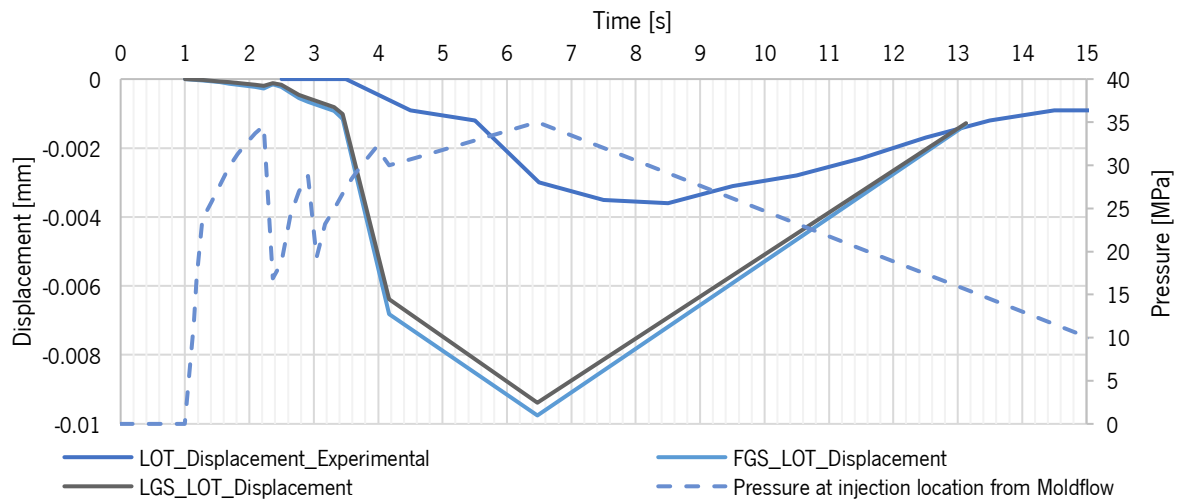


Figure 143 – Comparison of the displacement at the partition plane in the LOT position between the numerical and experimental results.

So, with this mold that has a larger volume of material to be injected, and with an injection system with hydraulic valve gates, the response times of the whole system tend to be longer. And with that the appearance of a significant discrepancy between the values obtained experimentally and numerically, even using the input parameters of the experimental test in the numerical simulation. For this mold there is a lag of approximately 2.02 seconds between the deformation peak obtained from the numerical simulation and the experimental one. In terms of the difference in displacement amplitude, the numerical models show values approximately 2.6 times higher than the experimental ones.

For the LOB point, the displacement curves are shown in Figure 144. And as for the LOT point the delay between the instant when the numerical and the experimental model reach the peak displacement is identical. But at the LOB point, the curve obtained numerically does not present a profile identical to the one obtained experimentally. At the LOT point, it is verified that the curve obtained numerically ends up presenting as many plateaus as the experimental one, although they have a greater amplitude, and with this also a greater slope. At the LOB point the experimental curve appears to have a parabola-shaped profile, while the numerical model shows that, as for the LOT point, it has stages according to the mold loading. At the LOB point the difference between the results obtained numerically and experimentally is significantly higher than for the LOT point. Here the displacement amplitude obtained experimentally is approximately 7.86 times smaller than the amplitude obtained numerically.

At the LOPB point, the peak-lag between the numerically and experimentally obtained curves is identical to the previous points as expected, as can be seen in Figure 145. At this point the numerical model presents a response to loading with a profile identical to the experimental one, but with a larger

amplitude and also earlier. This factor, as already explained is related to the results from the rheological simulation. Here the numerical model, at the peak displacement presented an amplitude 3.5 times higher than the experimental one.

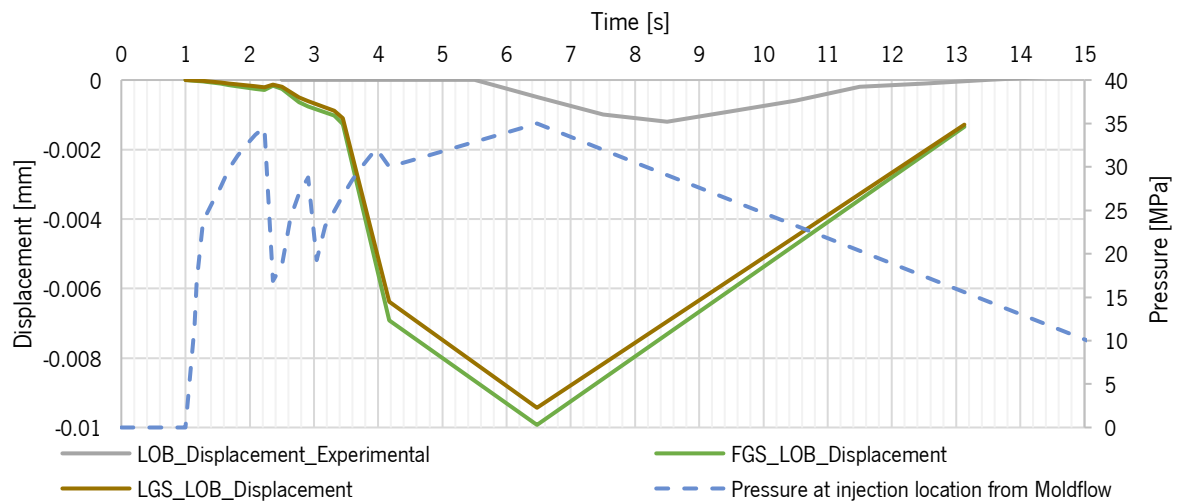


Figure 144 – Comparison of the displacement at the partition plane in the LOB position between the numerical and experimental results.

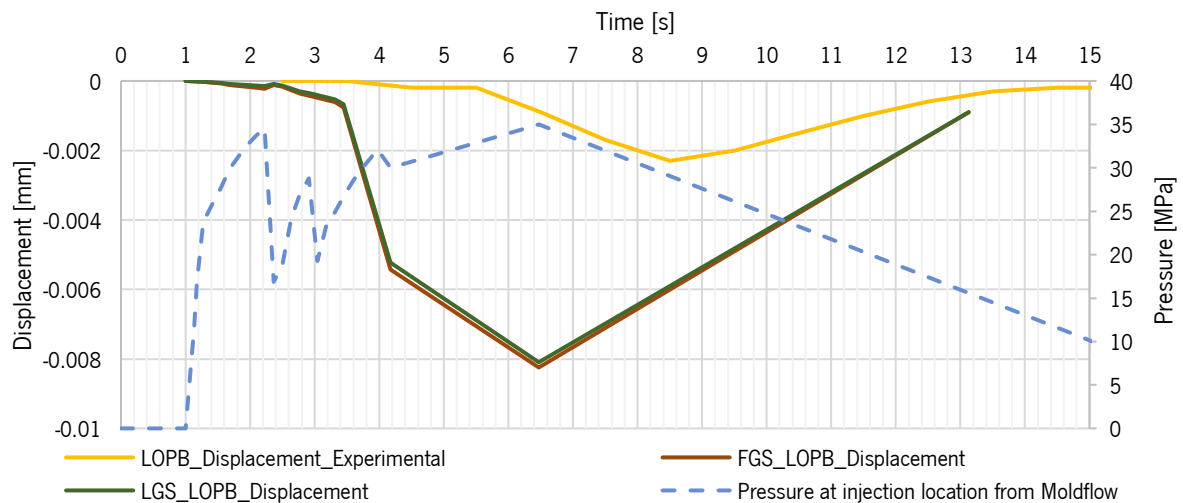


Figure 145 – Comparison of the displacement at the partition plane in the LOPB position between the numerical and experimental results.

With this analysis it is understood that the numerical simulations tend to present more conservative results for the amplitude of the displacements. But the analysis of the curves of the numerical models, can lead to a good understanding of the behavior of the injection mold. But of course, there are factors that are difficult to model, such as the behavior of the clamping plates of the injection machine that have a direct influence on the result. In this study the clamping plates of the injection molding machine were considered as rigid elements, which does not correspond exactly to reality. And the way the movable plate of the injection molding machine works, and its size can lead to deviations of the results that are difficult to predict. So, for mold sizing, working towards conservative results turns out to be the best procedure considering the time that exists for designing and manufacturing them. Although the LGS

model always shows results closer to the experimental ones, the difference between the experimental and the numerical results is very significant. Therefore, for tool 2, the simplifications are not the main cause for the difference between the numerical and experimental results.

As was done for tool 1, here too the pressure at the base of an ejector shaft was measured. The comparison of the measured results with those obtained from the numerical simulation are shown in Figure 146. As can be seen, both the curve profiles and the values are significantly different.

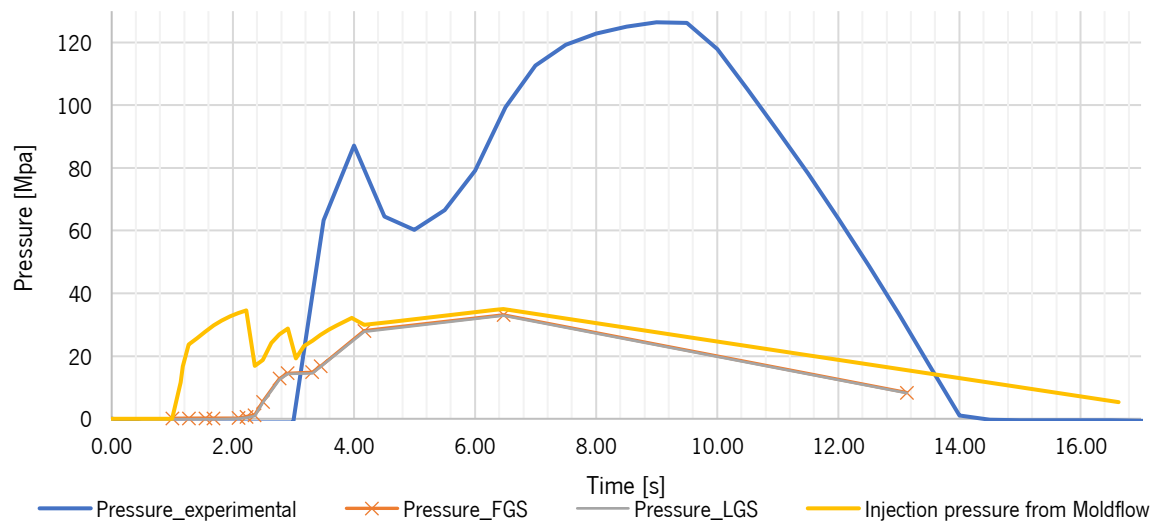


Figure 146 – Pressure at the indirect sensor, comparison between numerical and experimental data.

Starting by analyzing the measured pressure peaks, it can be seen that the lag is very identical with that already obtained for the displacement results. But with one difference, in the experimental results there is a first peak of approximately 87 MPa, followed by a second peak of approximately 127 MPa. This is due to the way the filling of the part is done. When analyzing the injection pressure curve obtained from Moldflow, it can be seen that there are four pressure peaks, the first three during the filling phase, where at each opening of the nozzle valves there is a drop in injection pressure. The last peak occurs in the compaction phase with all nozzles already open. The pressure sensor placed at the base of the ejection shaft, should be able to measure the pressure variation caused by the opening of a sequence of injection nozzles plus the variation during the compaction phase. And this is noticeable in the experimental curve, but for the pressure curve obtained numerically this is not noticeable. By analyzing Figure 117, it is verified that one of the injection pressure peaks was not selected for the execution of the structural numerical simulation, because the criterion used was the pressure variation within the molding region and this peak did not present a significant variation along the mold to be considered. But this led to the fact that for the numerical pressure curve one of the pressure peaks was missed.

But the biggest problem in the results obtained for the pressure curves is the discrepancy between the experimental and theoretical results. It would be expected to have pressure results obtained from the

numerical models to be significantly higher than the experimental ones, as had already happened in this injection mold for the displacement results. But the results obtained presented just the opposite. The experimental results presented significantly higher values than the numerical ones. These results are a consequence of two major factors that are difficult to be quantified in order to be included in the simulations, the first being the sensor assembly and the second being the behavior of the moving plate of the injection machine. The mounting of the sensor and ejection shaft will have a direct impact on the measured results, because the contact between the base of the ejection shaft and the sensor is very important and at the same time difficult to execute and replicate. As for the machine plate, as already mentioned, it is not rigid as considered in the numerical simulation. And at the corners of the machine's movable platen the locking is done, while in the central region is where a hydraulic will apply the clamping force that will keep the mold closed during the injection cycle. Therefore, the application of the clamping force causes the platen of the injection machine to slightly deform in the central region and consequently the clamping plate of the injection mold. As the pressure sensor is placed between the base of the ejector and the injection mold clamping plate, it is also placed in a region close to the region of application of the clamping force. Therefore, the pressure sensor is not only being solicited by the injection pressure, but it will also end up measuring the pressure exerted by the deformation of the mold clamp plate and the movable plate of the injection machine. Therefore, experimentally the measured pressure from the pressure sensor, will encompass the injection pressure and the pressure exerted by the deformation of the injection machine movable plate.

It is also necessary to consider that friction-related effects are not considered in the simulation, which can also lead to differences between the experimental and the numerical results. However, when admitting that there is no friction, it is admitted that in the numerical model the displacements between components can be higher, and thus the results tend to be more on the safe side.

5.4 Chapter conclusions

This chapter was aimed at understanding how numerical simulations could translate the behavior of injection molds. And it was noticeable that the simulations tend to present conservative results regarding the behavior of the injection mold. But that they tend to demonstrate the mold behavior, although some care must be taken in the preparation of the models.

For geometry preparation, oversimplifications can lead to the simulation not correctly modeling the problem. It was found that models with few geometric simplifications turn out to be closer to the experimental results. For example, the joining of the clamping and adjusting plates to the remaining steel of the mold, leads to their behavior being disregarded at the stage when the clamping force is applied.

Therefore, simplifications should be restricted to elements that are of little relevance to mold behavior, such as chamfers or fillets. Consideration should also be given to how the mold components work, so that the interactions between components are correctly applied in the numerical model, i.e., which contacts should be treated as bonded or frictional/frictionless. Friction was not considered in these simulations, but when there is a correct definition of this factor, it should be considered as it may lead to less conservative predictions.

The use of data from rheological simulations leads to a better understanding of the behavior of the injection mold during the injection cycle. But it is important that the choice of time-steps to be calculated is made with criteria so as not to disregard moments that are important for the analysis.

Boundary conditions can be a factor that leads to deviations between reality and the numerical model, so it must be taken into account that the behavior of the injection molding machine is difficult to be properly modeled and therefore tends to be disregarded. With this, one can verify whether certain mold behaviors during testing are intrinsic to the injection mold or whether they are derived from external factors. In other words, this can be another tool to detect the cause of problems during the injection phase.

Finally, it was also possible to understand that the simulations although they can translate the mold behavior, fail to predict the cycle time. That is, even using the parameters used in the test the rheological simulations showed cycle times lower than those obtained experimentally. With this note, it is verified that the use of calculations in transient regime ends up not bringing any advantage because one of the most relevant factors in this type of regime ends up not being very realistic. Furthermore, from the results obtained it was verified that the differences between the results obtained from models in quasi-static or transient regimes are not very relevant. The models calculated in quasi-static regime can give a good prediction with the advantage of having computation times significantly lower than those in transient regime as can be observed in Appendix C. Therefore, from these analyses it was possible to understand that by making a correct identification of the time instants to be studied in the structural simulation, having a models prepared with some care and with a correct definition of the boundary conditions it is possible to predict the behavior of the injection mold through the injection cycle. Although these predictions tend to be conservative.

6 Fatigue theoretical model analysis

Through this chapter will be presented 5 tools which showed fatigue cracks before reaching their projected lifetime. For each tool is made the structural analysis accordingly to the methodology discussed in the chapter 5.4. But to obtain the fatigue results, an application will be presented on this chapter. This application was designed to study injection tools and from which is expected to retrieve the expected safety factor, maximum alternated stress, expected number of injections for the tool.

First will be presented the tools under study and the part filling simulations to obtain the pressure loads for the fatigue analysis, where like for the simulations presented in section 5, they will be realized taking into account the injection trials information to replicate as close as possible the tool working conditions.

After the presentation of the inputs, for the injection tools by obtaining the pressures, will be presented the results from the structural simulation which will be used for the computation of the fatigue results.

Then, will be presented an application made to perform the fatigue calculations. Here will be presented the used models and the assumptions made to perform the calculations and the needed inputs. Then the fatigue calculations will be presented for the studied tools and compared to their work life up to the appearance of visible fatigue cracks.

6.1 Tools under study

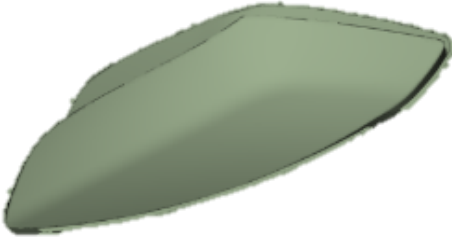
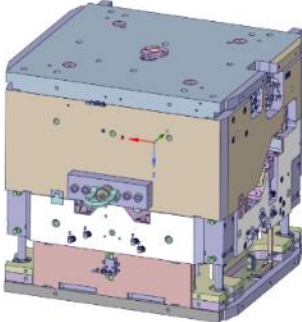
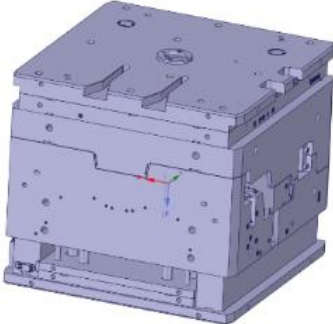
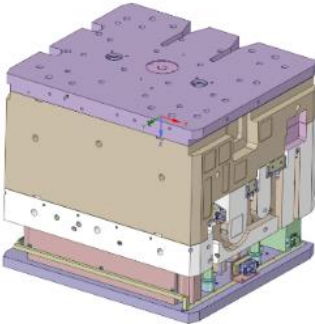
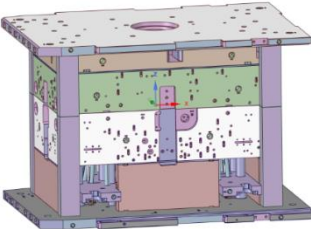
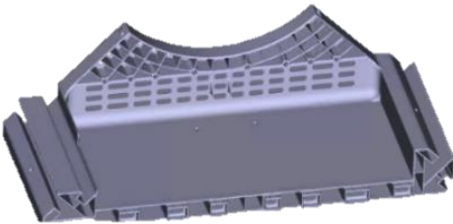
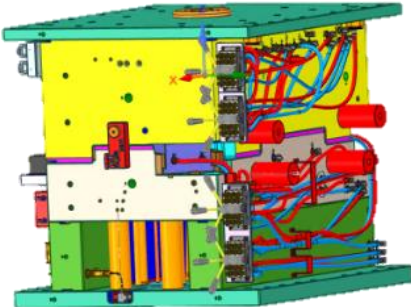
To conduct the studies to validate the procedure and algorithms to evaluate the fatigue endurance of the tools, the first step is the determination of the loads which will be used for the structural analysis.

In this chapter will be presented the parts and respective tools and the simulation conditions for each tool. To do so, a set of rheological simulations will be performed to gather the pressure loads for the structural analysis where for all analysis is took under account the tool working conditions. In the Table 16 is presented all the parts and tools that will be studied where each one of them presented fatigue cracks during its work life. To make easier the differentiation between the injection tools, each one will have an identification which can be encountered on the first column of Table 16. For each tool a numerical study was made in Moldflow to gather the pressure loads to use on the structural simulation. As for the structural numerical simulations presented in the previous chapter, from these simulations will be analyzed the results so only the time-steps relevant for the structural numerical simulations are used.

6.1.1 Rheological numerical simulation results

As stated above, these simulations are performed having into account the filling parameters gathered during the trials. Now will be presented the filling conditions, the filling results and the pressure detail for critical locations.

Table 16 – Polymer parts and respective tools.

Tool ID	Molded part	Injection tool
Fat_1		
Fat_2		
Fat_3		
Fat_4		
Fat_5		

6.1.1.1 Results for the tool Fat_1

In the Table 17 is presented the inputs for the rheological simulation. For the simulation the filling phase and the first instants of the packing phase had a special attention, because it is the time interval where the pressures are higher so it is also the time interval where more damage can be made to the tool in terms of fatigue endurance.

Table 17 – Input data for numerical simulation of Fat_1.

Plastic material	Mitsubishi Chemical Corporation - Durabio D7340R In Appendix B is presented the polymer datasheet
Filling phase control	Flow rate [438 cm ³ /s]
Packing pressure	10 seconds at 45 MPa more 5 seconds of decompression.
Cooling time	20 seconds
Tool surface temperature	80 °C
Melt temperature	240°C

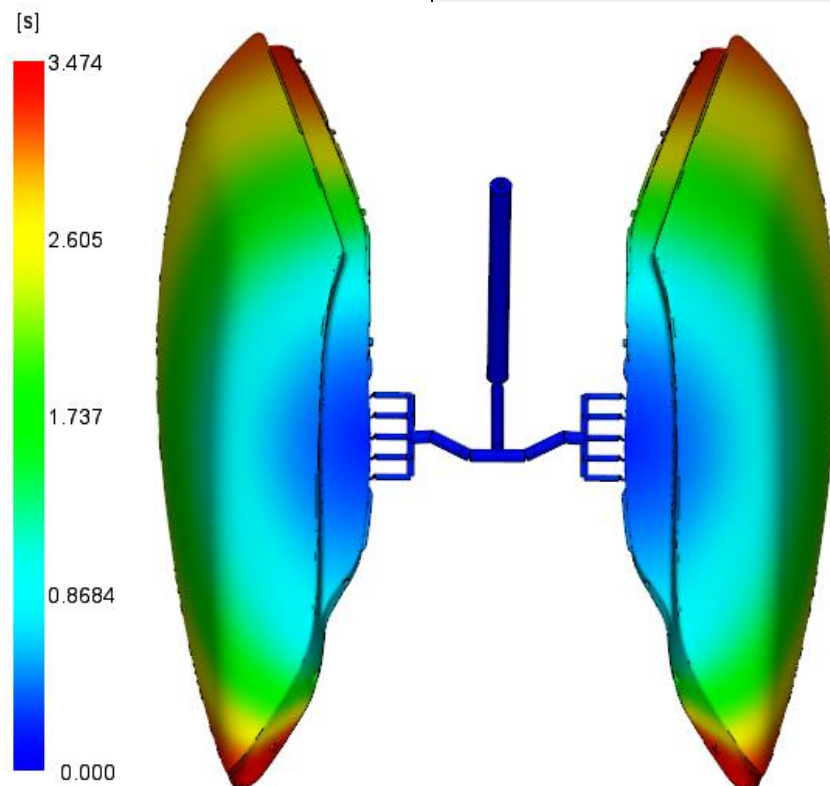


Figure 147 – Filling pattern of the part.

From the Figure 147 is possible to observe that the filling is globally balanced, but since the parts aren't symmetric one of the sides fills before the other. The difference is small but enough to have the pressure unbalanced inside the tool at the end of filling as is possible to observe in Figure 148. When the machine commutes from the velocity controlled filling to the pressure controlled filling one side is

already filled that is why the pressure is higher at that side (Figure 148 – a) which will reflect at the end of the part filling which can be observed in Figure 148 – b.

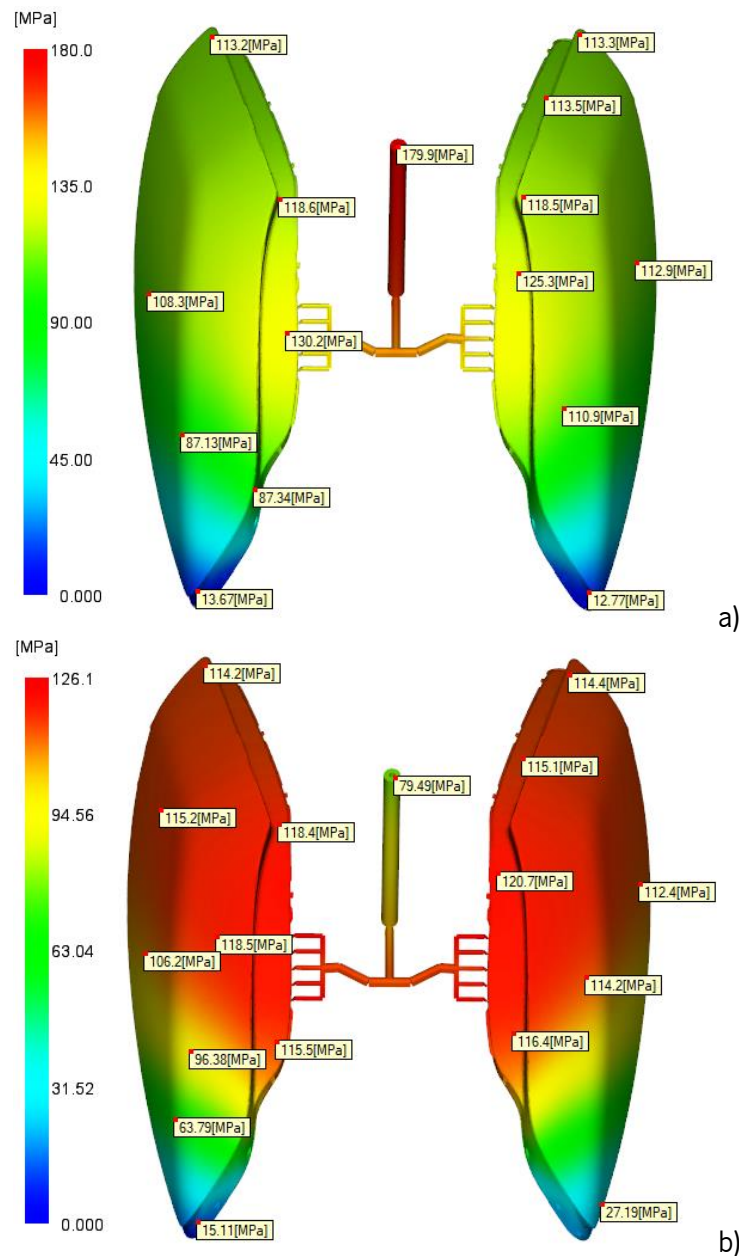


Figure 148 – Pressure distribution a) at the velocity/pressure switchover and b) at the end of filling.

In Figure 149 is presented the pressure at some locations which are critical for the tool fatigue endurance. From this plot is also possible to understand the pressure loss existent between the hot runner entrance the part, which in the case of this tool is higher than the pressure loss from in the part filling.

The tool will need a minimum clamping force of 2639 Ton to ensure there are no separation between the core and cavity during the injection cycle. The plot presented in Figure 150 shows the needed for each pressure instant, but in reality, the clamp force is equal through all the injection cycle, this plot only helps to understand the needed clamp force for the tool at each instant.

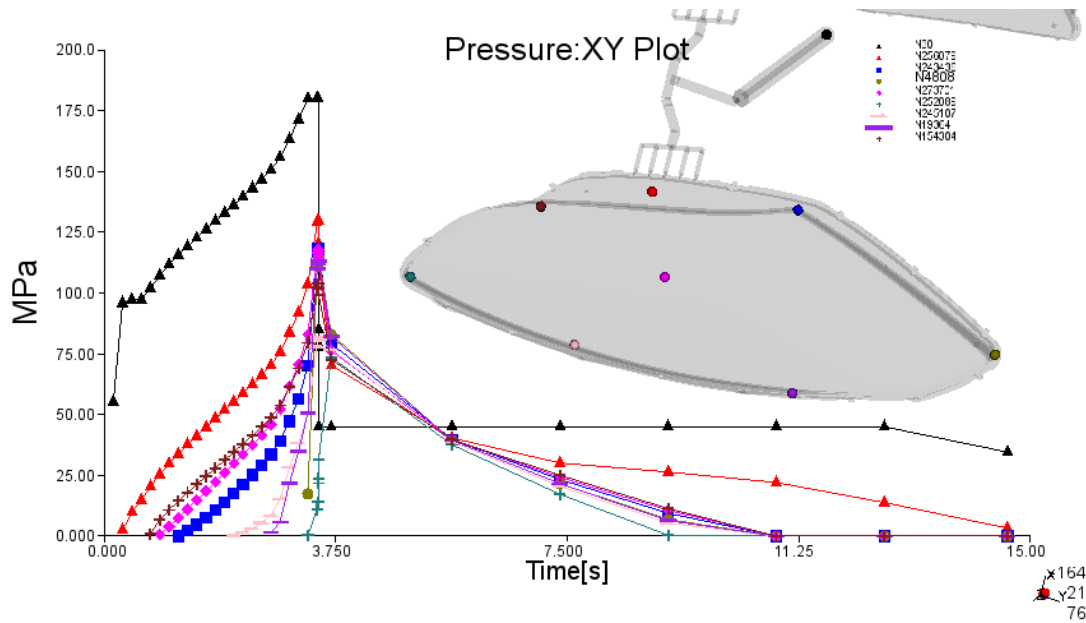


Figure 149 – Pressure through time in critical locations of the engraving region.

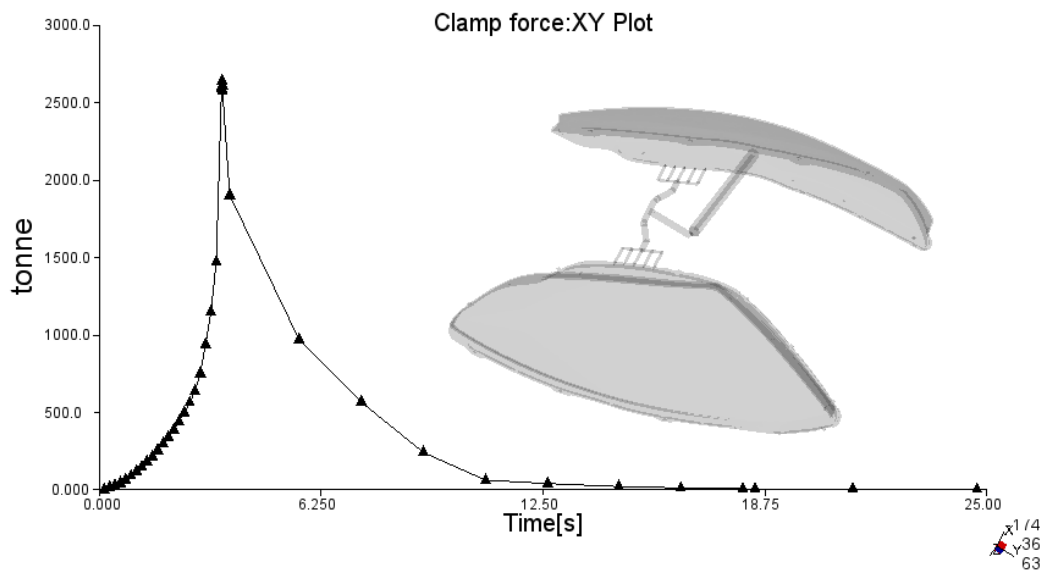


Figure 150 – Needed force to keep the injection tool closed through the injection cycle.

6.1.1.2 Results for the tool Fat_2

In the Table 18 is presented the inputs for the rheological simulation for the plastic part of Fat_2. Like for Fat_1 will be paid a higher attention to the filling phase due to the high pressure needed to fill the cavity.

By the comparison of the inputs shown in Table 18 and Table 17 is possible to verify that the control of the part filling is similar. This is related to the injected material that must be injected with a high flow rate to prevent the front flow of the molten plastic to cool down to avoid aesthetical defects. Is also possible to observe that the packing phase is equal to the tool Fat_1.

Unlike the Fat_1 part, in this part is not expected to have the molten plastic reaching both extremities

at the same time since its shape forces the flow to divide itself to contour the hole in the part. As is possible to observe in Figure 151 the flow fronts will end against each other.

Table 18 – Input data for numerical simulation of Fat_2.

Plastic material	Mitsubishi Chemical Corporation - Durabio D7340R In Appendix B is presented the polymer datasheet
Filling phase control	Flow rate [225 cm ³ /s]
Packing pressure	10 seconds at 45 MPa more 5 seconds of decompression.
Cooling time	20 seconds
Tool surface temperature	80 °C
Melt temperature	240°C

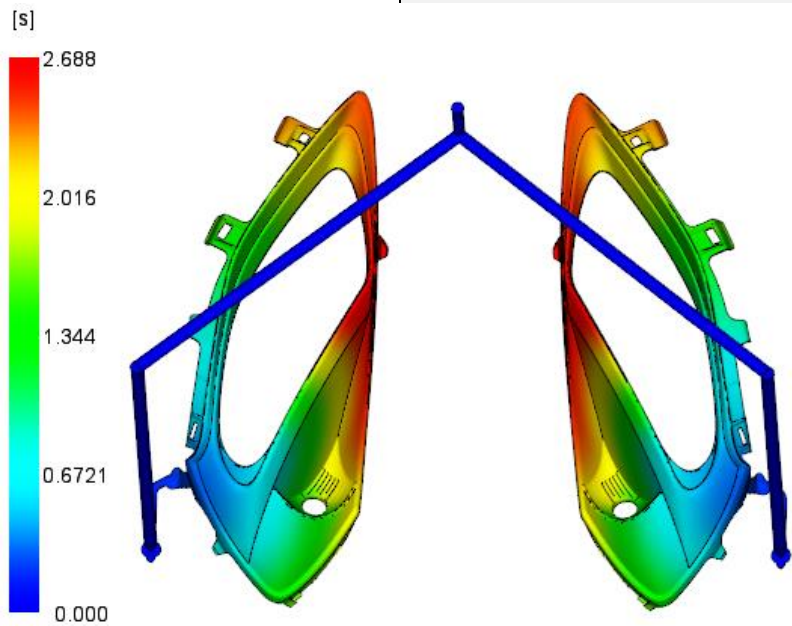


Figure 151 – Filling pattern of the part.

From the Figure 152 is visible how the flow pattern has a significative impact on the part pressure distribution. Of course, the higher pressures are at the injection location, but at the rest of the part when pressures are very unbalanced besides the possible impact on the part quality, the tool can be placed under a higher stress during the filling stage. From Figure 152 b) is possible to observe that the region in front of the gate will be subjected to much higher pressures.

From Figure 153 where is presented a pressure plot through time at specific locations is even more visible how the pressure is unbalanced through the part. Only at one instant the pressure is equilibrated on the part at the packing stage but rapidly this equilibrium fades away.

As for the clamp force, this injection tool will require at least 796 Ton of clamp force to keep the cavity and core together during the injection cycle.

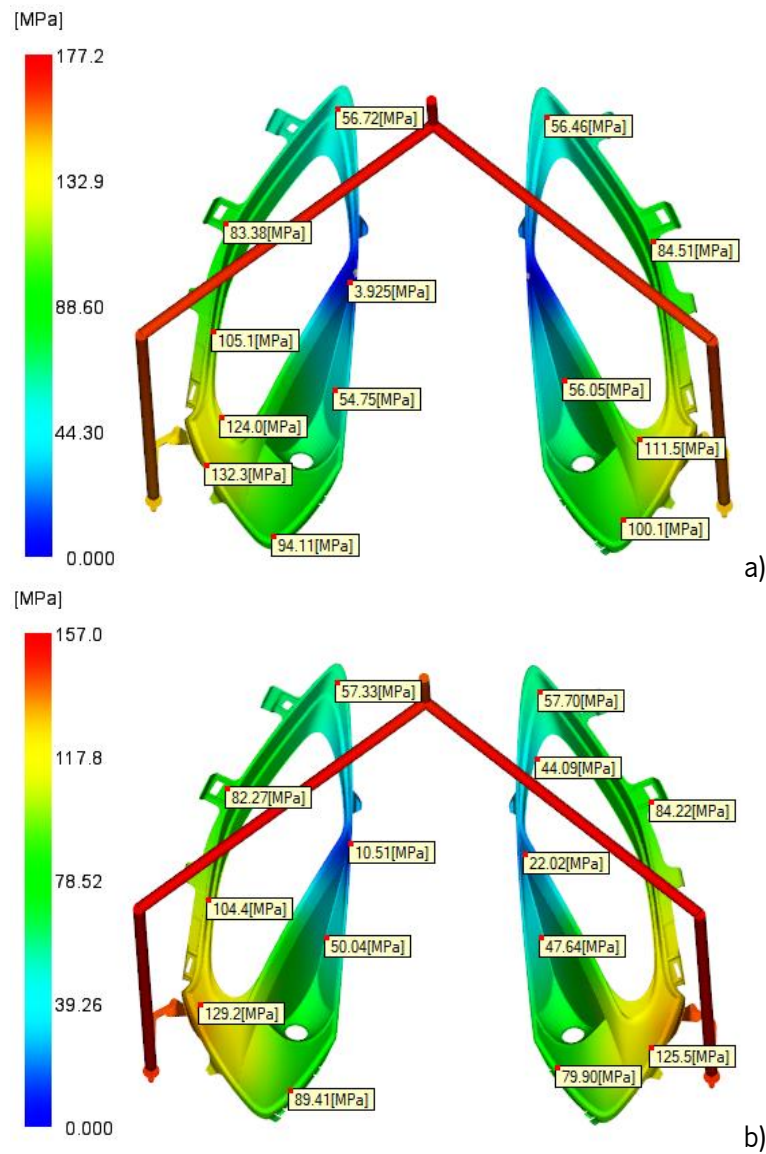


Figure 152 – Pressure distribution a) at the velocity/pressure switchover and b) at the end of filling.

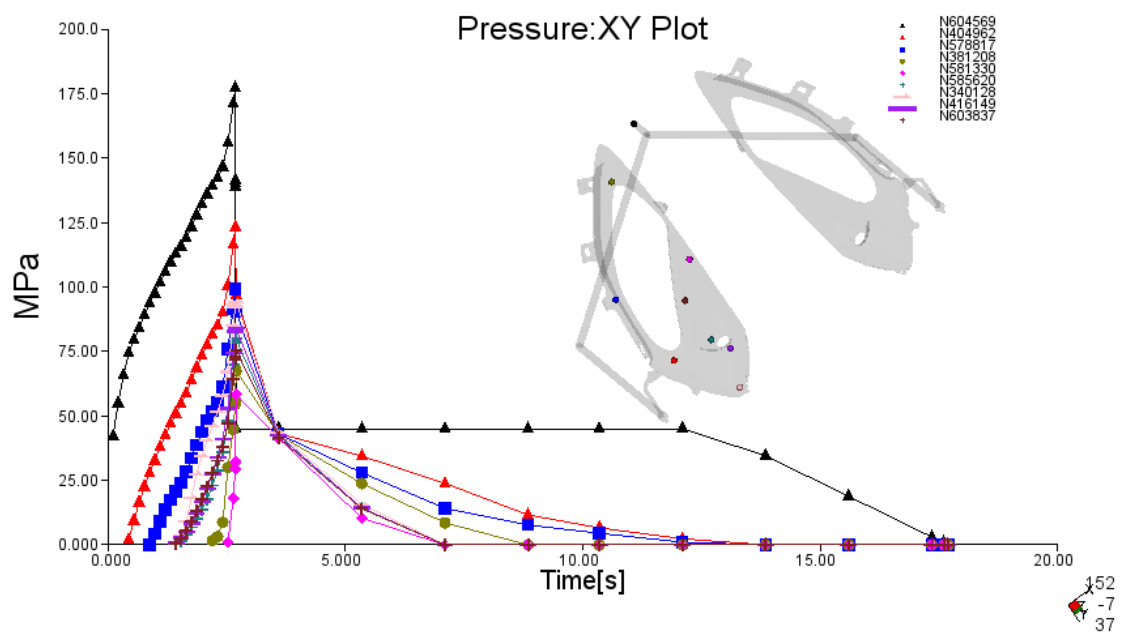


Figure 153 – Pressure through time in critical locations of the engraving region.

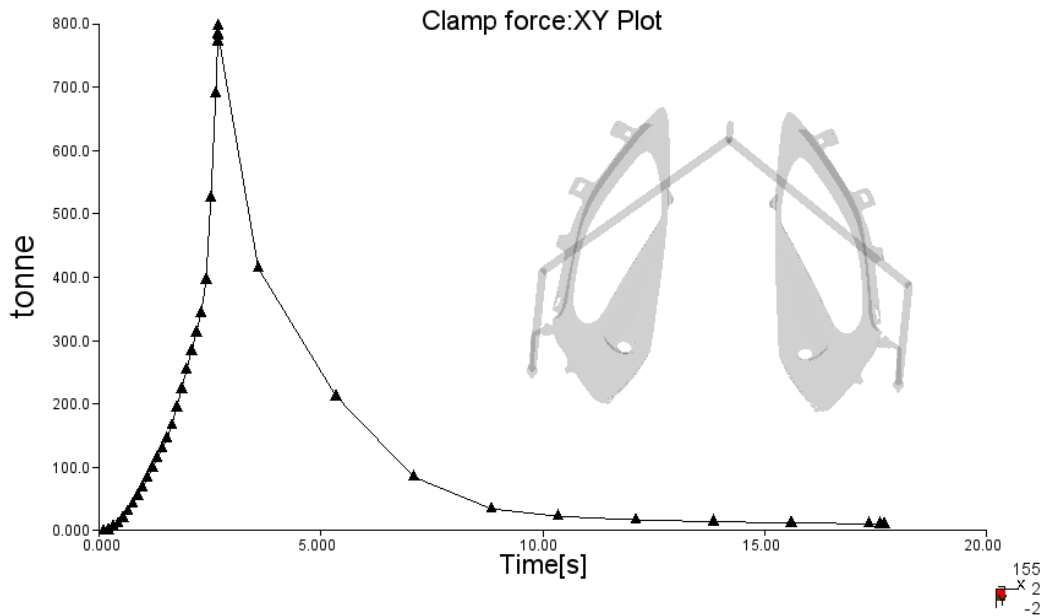


Figure 154 – Needed force to keep the injection tool closed through the injection cycle.

6.1.1.3 Results for the tool Fat_3

The input parameters for the rheological simulation of Fat_3 are present in the Table 19, which like for the part of Fat_1 and Fat_2 the part filling and packing is similar. These similarities are normal since the parts are filled with the same polymer and have similar filling problems.

Table 19 – Input data for numerical simulation of Fat_3.

Plastic material	Mitsubishi Chemical Corporation - Durabio D7340R
	In Appendix B is presented the polymer datasheet
Filling phase control	Flow rate [217 cm ³ /s]
Packing pressure	10 seconds at 45 MPa more 5 seconds of decompression.
Cooling time	20 seconds
Tool surface temperature	80 °C
Melt temperature	240°C

This part has two big obstructions to the flow pattern, but unlike the Fat_2 part the filling pattern tends to go from one side of the part up to the other as presented in Figure 155. For this part the pressures will be also unbalanced, due to the part shape is normal to have this kind of unbalancing as is possible to observe in Figure 156, but for this tool there is a difference when compared with the tool Fat_2 since when is compared the plot of pressure through time (Figure 157) is possible to observe that during the packing stage the pressure tends to be homogeneous through the part. So, it is expected to have the unbalanced pressure through a lower time, this will happen during the filling stage but though the packing stage the pressures will be more balanced though the engraving region.

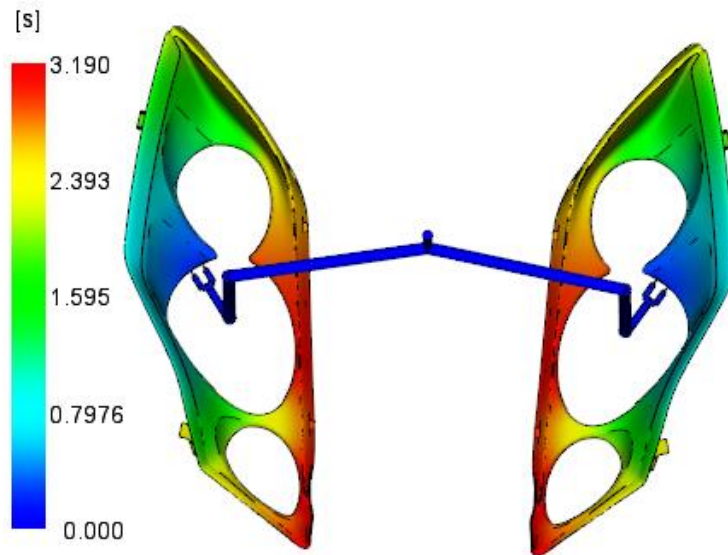


Figure 155 – Filling pattern of the part.

The needed minimum clamp force to keep the tool closed during the injection cycle is 828 Ton as is possible to understand from the plot presented in Figure 158.

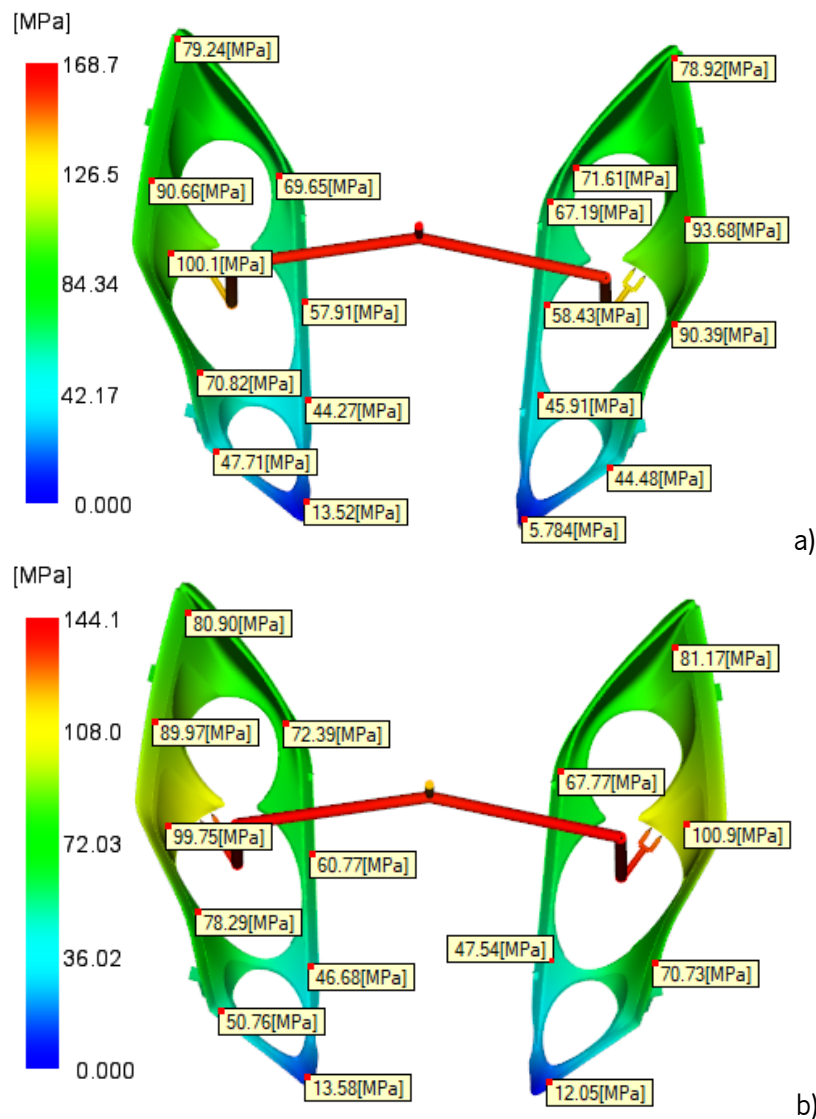


Figure 156 – Pressure distribution a) at the velocity/pressure switchover and b) at the end of filling.

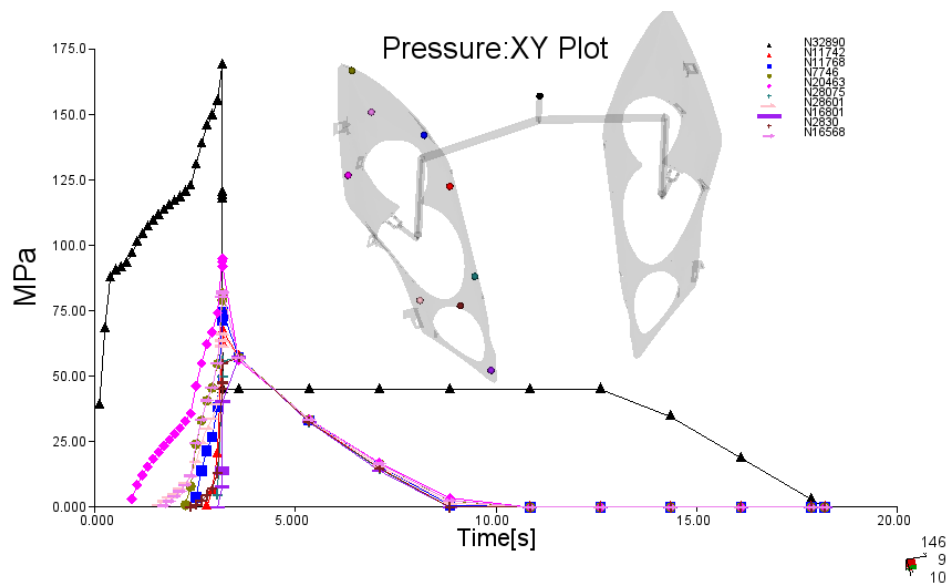


Figure 157 – Pressure through time in critical locations of the engraving region.

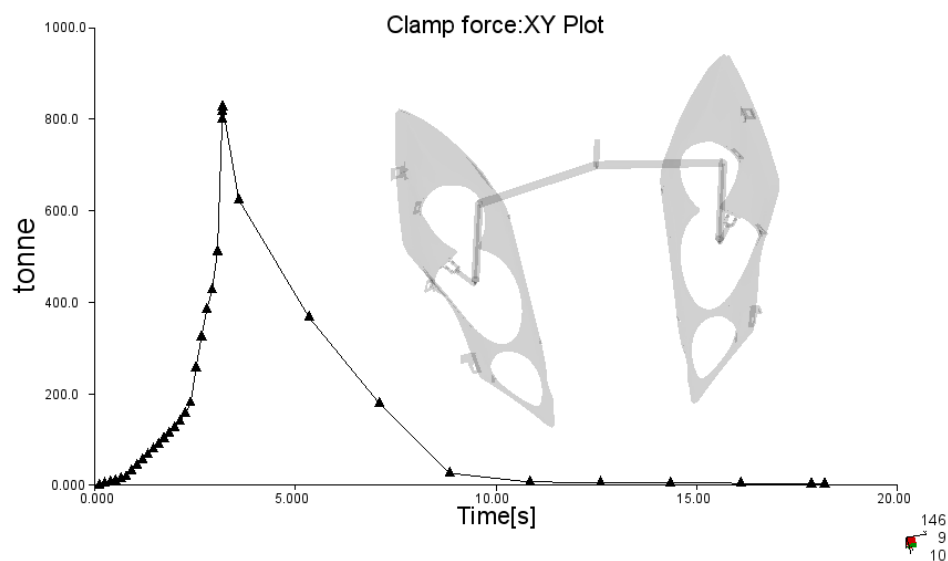


Figure 158 – Needed force to keep the injection tool closed through the injection cycle.

6.1.1.4 Results for the tool Fat_4

The obtained input parameters for the rheological simulation of Fat_4 are present in the Table 20.

Table 20 – Input data for numerical simulation of Fat_4.

Plastic material	SunAllomer Ltd - TRC104N In Appendix B is presented the polymer datasheet
Filling phase control	Flow rate [594 cm ³ /s]
Packing pressure	10 seconds at 60 MPa more 5 seconds of decompression
Cooling time	20 seconds
Tool surface temperature	20 °C
Melt temperature	225°C

For this tool is possible to observe that the rate of injected material is higher, mostly because the parts have a higher volume than the ones presented before and due to the materials characteristics is needed to inject the parts with enough flow rate to reduce aesthetic defects caused by the front flow temperature variations.

This part has a different mechanism to ensure the proper filling without aesthetical defects, to do so this tool has valve gate injection system where the central valve gate (VG_1 from Figure 159) is the first to open then the other two valve gates open when the cold channels are fully filled (VG_2 from Figure 159).

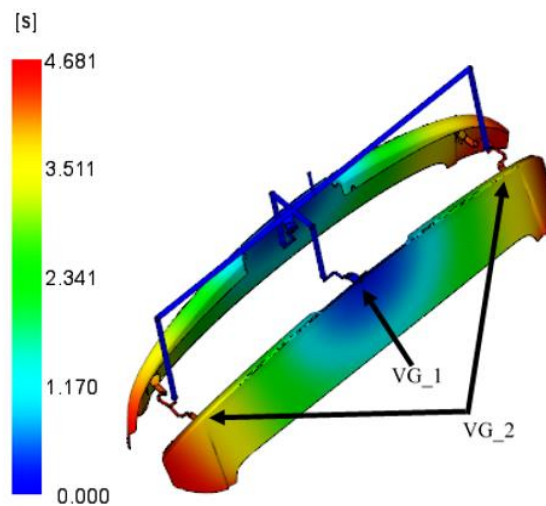


Figure 159 – Filling pattern of the part.

Since these parts are symmetrical is expected to have a balanced filling of the parts and the pressure distribution should also be balanced during the filling and packing stage. This can be corroborated by the pressure distribution showed in the Figure 160, where there is a symmetry of the pressure values by the center of the part. Of course, the central region of the parts will have an higher pressure during the filling stage since this region will have molten plastic entering under pressure during the entire filling stage.

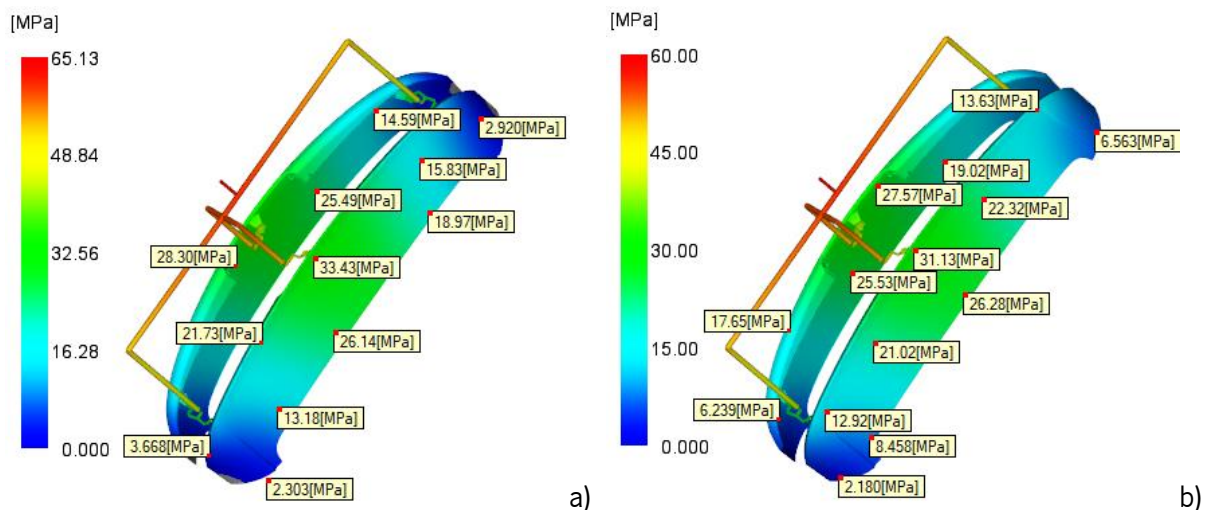


Figure 160 – Pressure distribution a) at the velocity/pressure switchover and b) at the end of filling.

From Figure 161 is possible to understand that the pressure is higher at the center of the part and during the filling phase it decreases up to the part extremity but, at the packing phase the pressure tends to be more homogeneous through the part until most of the part is molten because, when the part starts to freeze up there are no molten material to apply pressure and with it the pressure at those locations starts to decrease.

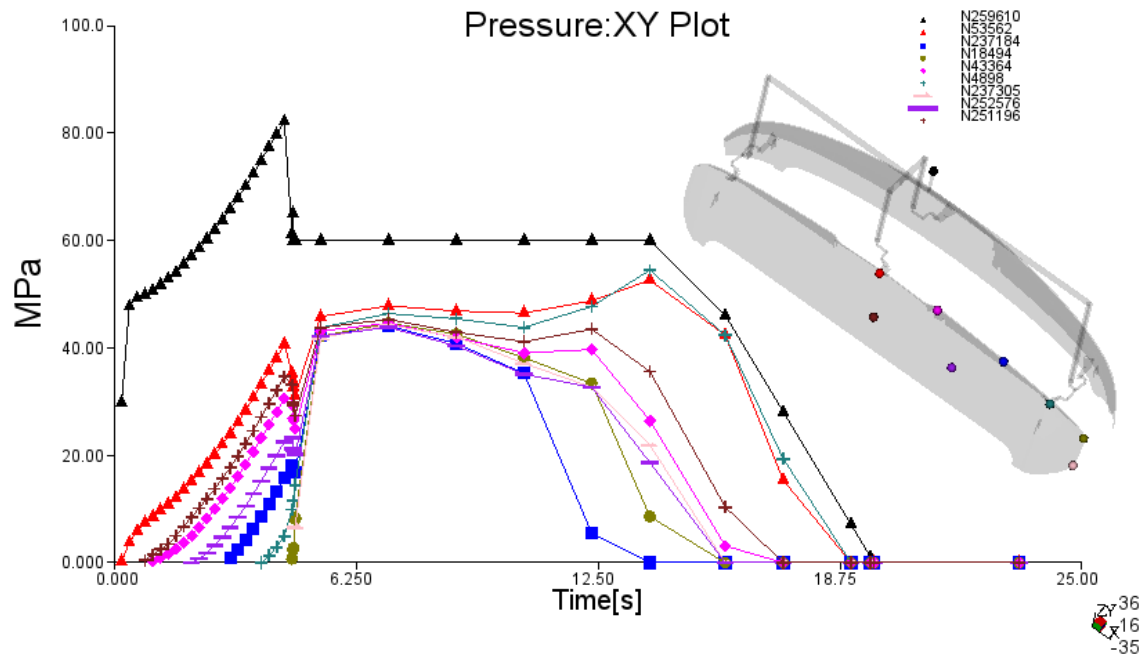


Figure 161 – Pressure through time in critical locations of the engraving region.

The minimum clamp force to ensure that the cavity and core doesn't separate during the injection cycle is 802 Ton.

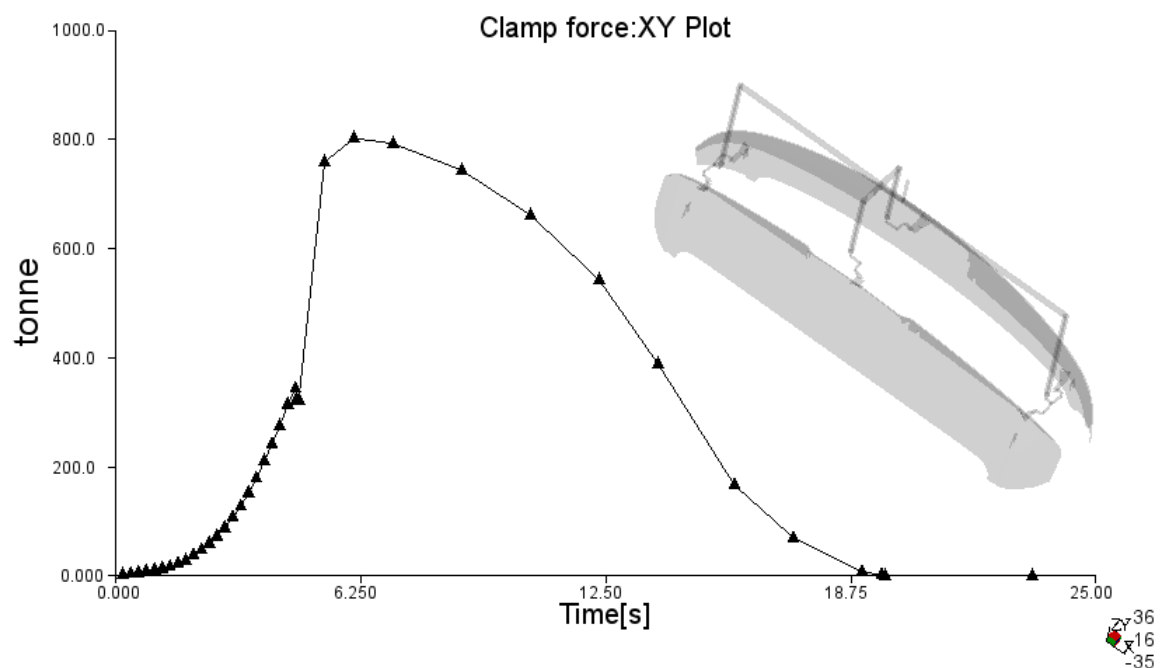


Figure 162 – Needed force to keep the injection tool closed through the injection cycle.

6.1.1.5 Results for the tool Fat_5

The obtained input parameters for the rheological simulation of Fat_5 are present in the Table 21. For this tool is possible to observe that is the one from all the previous injection molds studied on this section that have the highest injection flow rate, this is due to the injection used on this tool which requires a high amount of material to fill the part fast and with good conditions to produce a structurally stable plastic part.

Table 21 – Input data for numerical simulation of Fat_5.

Plastic material	Borealis Europe – BD950M0 In Appendix B is presented the polymer datasheet
Filling phase control	Flow rate [$1075 \text{ cm}^3/\text{s}$]
Packing pressure	18 seconds at 25 MPa more 10 seconds of decompression.
Cooling time	10 seconds
Tool surface temperature	40 °C
Melt temperature	240°C

This plastic part, unlike all the others that have been presented, has four injection points direct to the part, i.e., the hot channels are placed directly on the part. As can be seen in Figure 163, the filling starts with the four hot runners open simultaneously, because if it were done in sequence, it would be difficult to keep the pressures within manageable limits for the mold, and furthermore, in this part visible joint lines are not a problem. With all four hot runners open it is also possible to fill the part faster and thereby reduce cycle time.

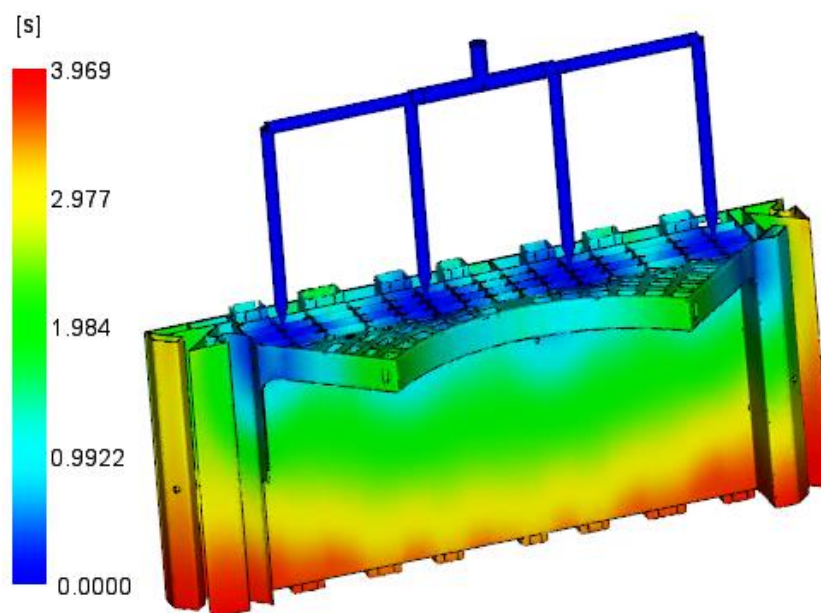


Figure 163 – Filling pattern of the part.

From Figure 164 a) it can be seen that the filling is not balanced, i.e. at the V/P switchover instant it is easily noticeable that the remaining part to be filled is not symmetrical, this is due to the shape of the part that is also not symmetrical. This pressure imbalance will obviously be reflected in the pressure in the molding zone at the end of filling as shown in Figure 164 b). These unbalances always bring two associated problems, one is a significant increase in injection pressure and the other is an unbalanced loading in the mold that in turn will be passed to the injection machine. As a rule, we try to keep the efforts as balanced as possible so as not to cause permanent damage to both the injection mold and the injection machine.

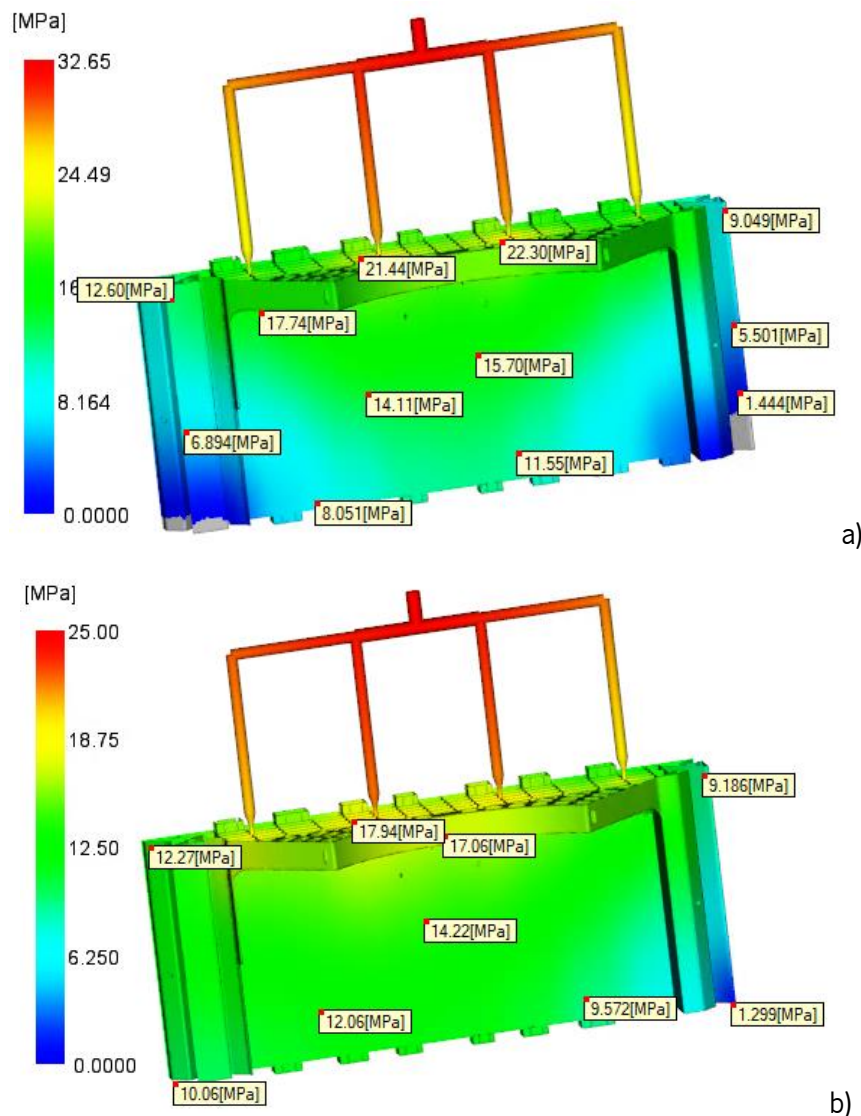


Figure 164 – Pressure distribution a) at the velocity/pressure switchover and b) at the end of filling.

By analyzing Figure 165 it can be observed that depending on the location, the pressure curve profile is different. For example, unlike the previous parts, a point near the injection locations, such as the one represented by the dark blue curve in Figure 165 would be expected to maintain higher pressures for a longer period of time than others more distant, something that does not occur. Because, besides the

proximity to the injection points it is necessary to take into account the frozen layer of polymer, i.e., in addition to the distance to the injection point there are other factors that influence the pressure at each location, hence rheological studies assume great importance in obtaining the loads to be used in structural studies.

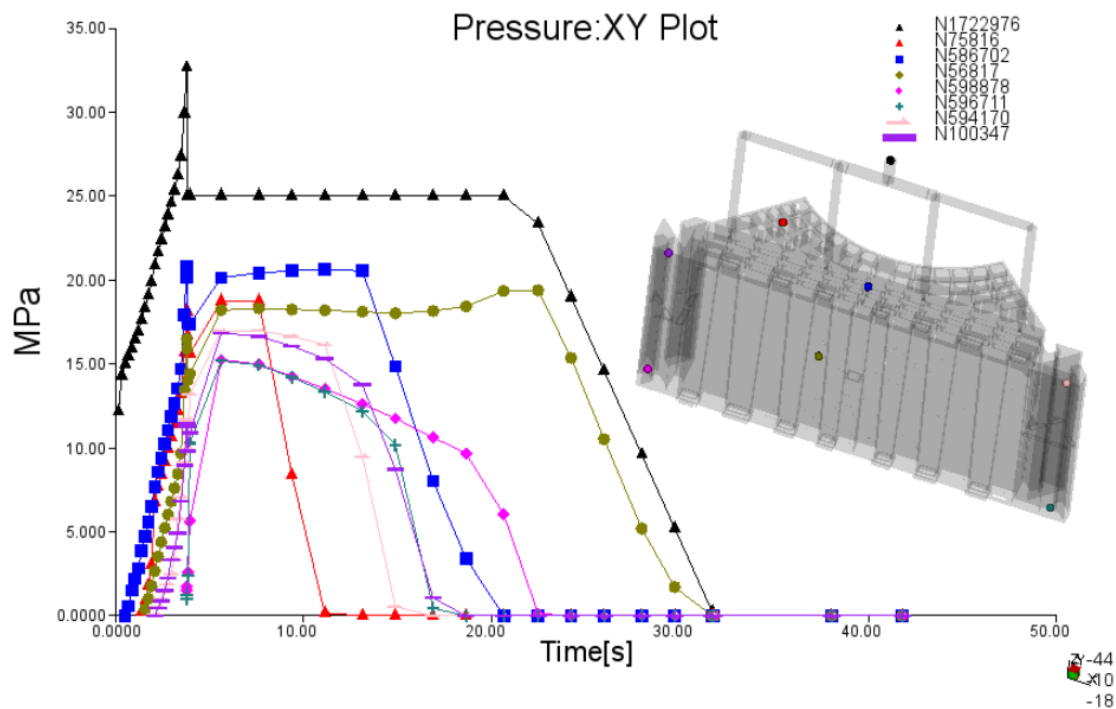


Figure 165 – Pressure through time in critical locations of the engraving region.

The minimum clamp force to ensure that the cavity and core does not separate during the injection cycle is 171 Ton.

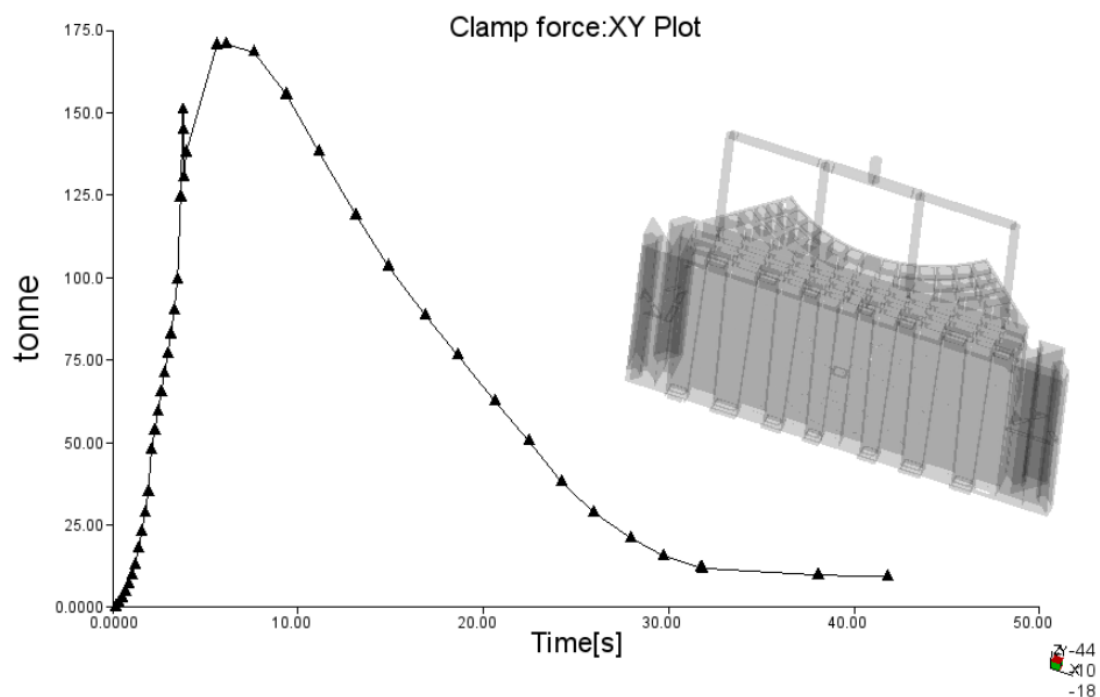


Figure 166 – Needed force to keep the injection tool closed through the injection cycle.

6.1.2 Structural simulations

In contrast to what was presented in chapter 5.2, in this chapter won't be made an extensive evaluation to the results of the structural study. Important results for the calculation of the fatigue strength of the injection molds under study will be presented. For this, the stresses in the most stressed regions during the injection cycle at the instant when the clamping force is applied and then at the most relevant instants will be presented. With this, it is intended to demonstrate the region's most subject to fatigue failure.

During this work it is considered that at the end of 1 second only the clamping force is applied and that only from that moment on the filling phase starts, whereas the injection cycle includes these two phases plus the cooling phase, which will not be evaluated because it is not relevant to this work.

As in the structural studies presented in section 5.2, the thermal loads in the injection mold are not considered because these molds work at temperatures below 40 °C and only the molding region will suffer a temperature variation during a short period of time, i.e., when the flow front of the molten polymer passes and after the passage of this flow front the temperature tends to decrease drastically to the nominal working temperature of the injection mold.

6.1.2.1 *Results for the tool Fat_1*

For this mold, since it produces two symmetrical parts it will only be considered half a model having a symmetry condition applied on the plane of symmetry. Since only half a model is used, the clamping force applied is half of that foreseen in section 7.1.1.1.

As presented in Figure 167 the instant when the clamping force is applied to both the cavity and the tap, there are relevant stresses. In regions that have higher stresses, i.e., close to or higher than the yield stress at the instant when only the clamping force is applied is relative to small geometric interferences. In this model no geometric simplifications were performed in the partition plane, hence small interferences may occur, interferences that exist in the physical model, but that are adjusted when the mold is tuned. It is found that these interferences can have an impact on the quality of the results. Also, due to the pressures that are exerted in the mold during the injection process a significant clamping force is required, this is reflected in the distribution of stresses in the mold, as can be seen in the structure on the side of the core, the supports already have stresses around 100 to 200 MPa that do not increase significantly at the time of highest pressure inside the mold.

When the highest pressures are reached in the injection mold, the stresses in the molding region on the cavity side increase significantly as shown in Figure 167, as well as the stresses in the partition plane adjustment region on the core side. Although these are the regions with the best surface finish, where the molding zone has the best finish, they are also the most stressed during the injection cycle, hence

the detailed analysis performed to these regions.

Figure 168 shows the von Mises stresses in the molding regions where it can be observed that while only the clamping force is applied to the injection mold, it can be seen that stresses on the order of 150 to 200 MPa are already visible in the cavity. These stress values before applying any pressure in the molding region are critical because besides already presenting high values before the polymer injection starts, it is also located on a notch that tends to be more susceptible to the appearance of cracks. But in the cavity, as the pressure increases, the most requested region when the clamping force is applied, ends up not presenting a significant change in terms of von Mises stresses. On the other hand, the stresses along the molding region increase significantly. Special attention should be given to a notch present in the cavity where the stresses increase significantly to values higher than 200 MPa.

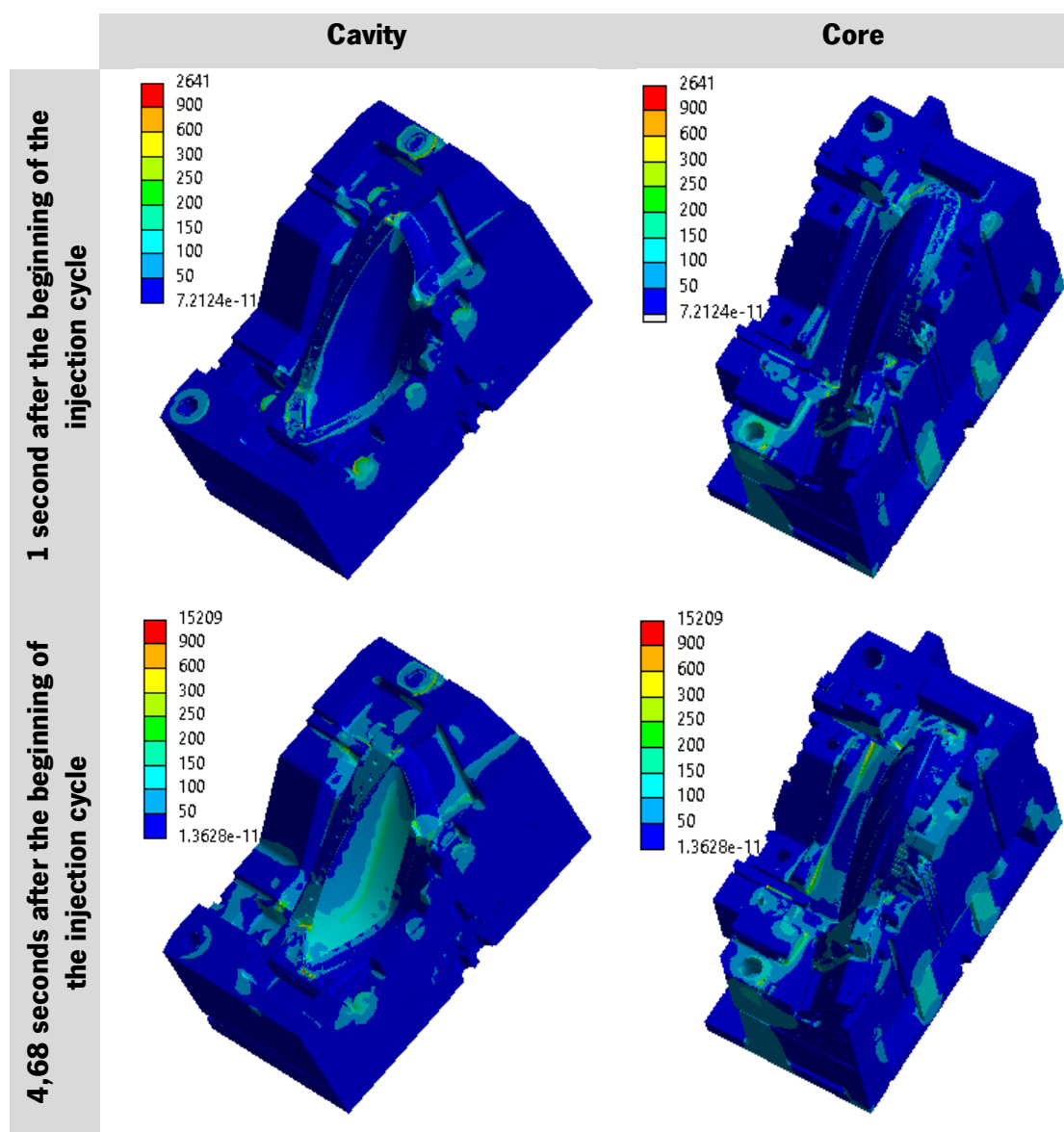


Figure 167 – von Mises stress distribution through the core and cavity sides of Fat_1 at the instant where only the clamp force is applied and at the instant of highest pressure through the engraving region.

On the core side there are two distinct points that require some attention can be observed in Figure

168: one is when the clamping force is applied where the stresses increase locally. The other is in an area near the partition plane adjustment zone where stresses increase significantly with the increase of the injection pressure.

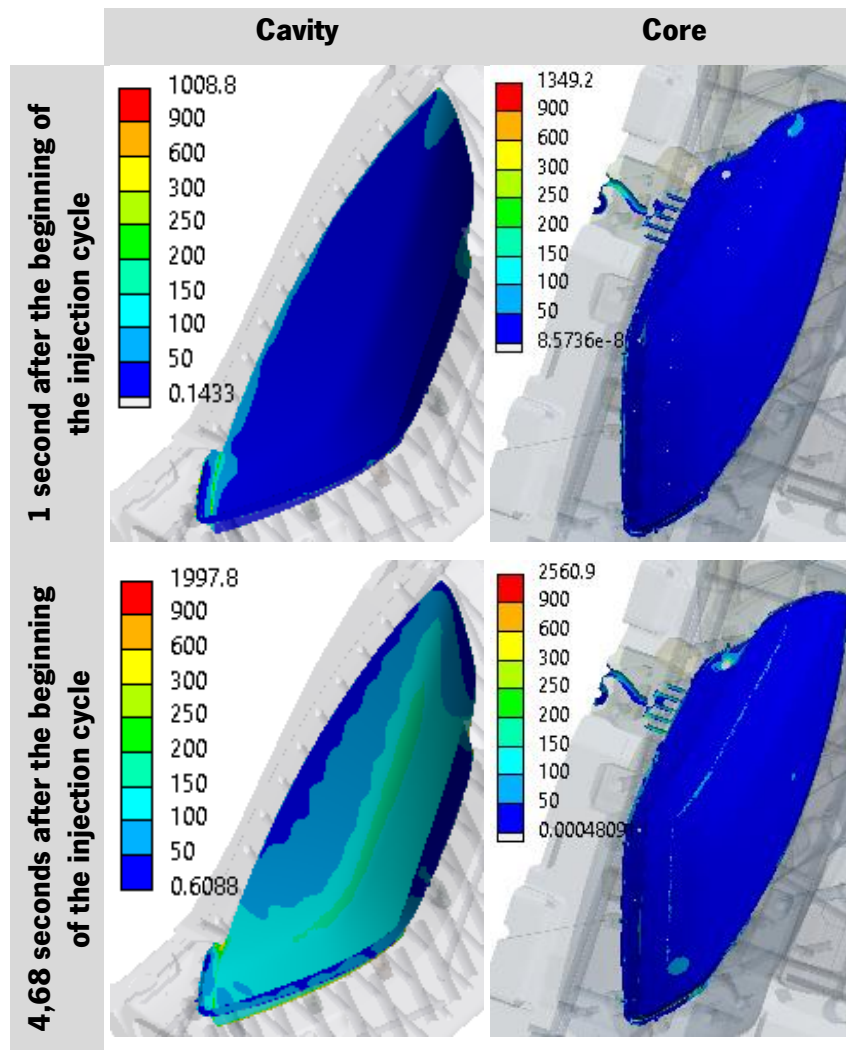


Figure 168 – von Mises stress distribution through the core and cavity engraving regions of Fat_1 at the instant where only the clamp force is applied and at the instant of highest pressure through the engraving region.

In the regions of adjustment in the partition plane shown in Figure 169 it can be seen that either on the cavity side and core side, high stresses are already observed in regions where there are geometric discontinuities, although in the region closest to the symmetry plane used in numerical simulation they appear in a wider extension. The application of the clamping force will lead to the core side being pressed against the cavity side, although in many regions the increase in stresses is related to the compression of the steel, where there are geometric discontinuities such as curved surfaces it will lead to bending moments being applied in those regions which consequently lead to them being mechanically more stressed and more liable to fatigue failure.

When the peak injection pressure is reached in the cavity, there are higher stresses in the partition plane fit region where there are some regions that have von Mises stresses greater than 300 MPa,

regions that are subject to fatigue failure.

It is also found that some regions that were being mechanically stressed when the clamping force was applied, that the stresses eventually became zero or residual with the application of the injection pressure. This happens because the injection pressure will counteract the clamping force applied to the mold until at the very limit the pressure exerted by the core side on the cavity side is residual.

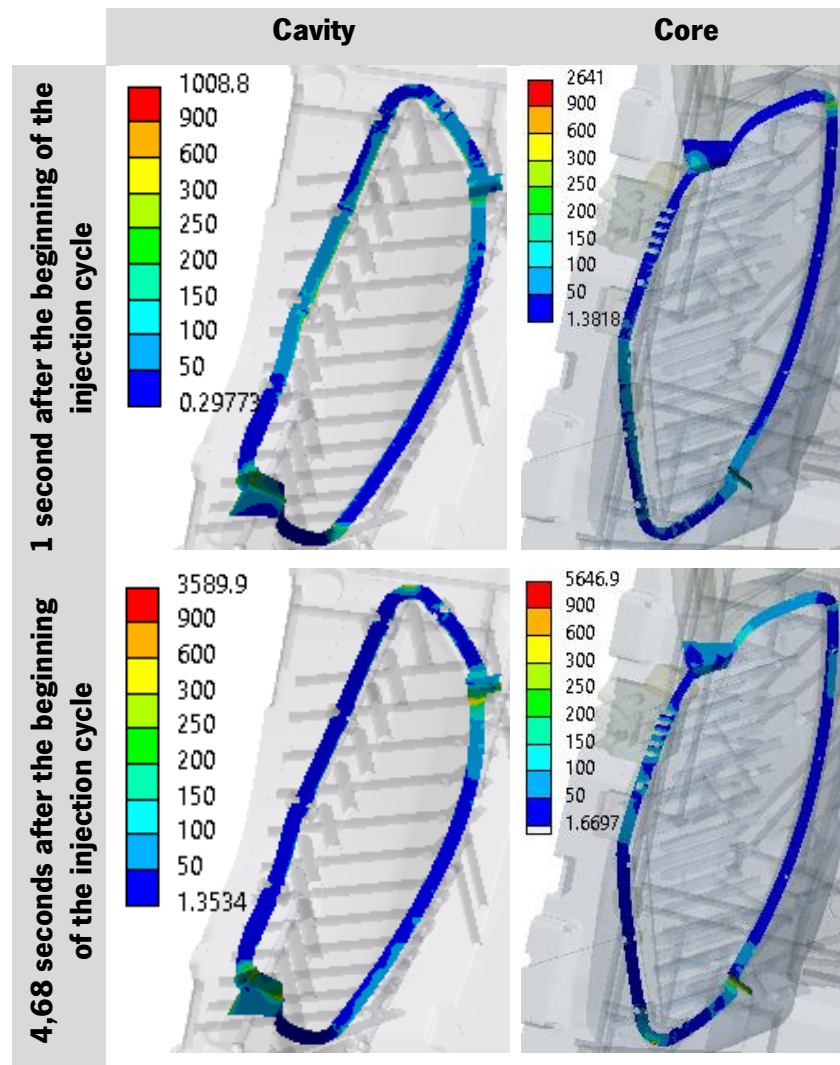


Figure 169 – von Mises stress distribution through the core and cavity fitting regions of the partition plane of Fat_1 at the instant where only the clamp force is applied and at the instant of highest pressure through the engraving region.

6.1.2.2 Results for the tool Fat_2

For the study of this injection mold, as it was performed for Fat_1, only half of the mold will be analyzed because here the injected parts are also symmetric and so is the mold.

Figure 170 shows the von Mises stress distribution in the mold on the cavity and core side, and unlike Fat_1, here when the clamping force is applied no very high stresses appear, although they are already felt in regions near the surfaces where there is contact between the core and cavity side.

When the moment when the greatest pressure is reached inside the injection mold, as can be seen in Figure 170, the stresses increase significantly both on the side of the core and the cavity. But on the

side of the core there are regions with stresses exceeding 300 MPa that in terms of fatigue resistance may be critical. On the cavity side they are also present, but in regions far from the molding zones and are essentially linked to singularities.

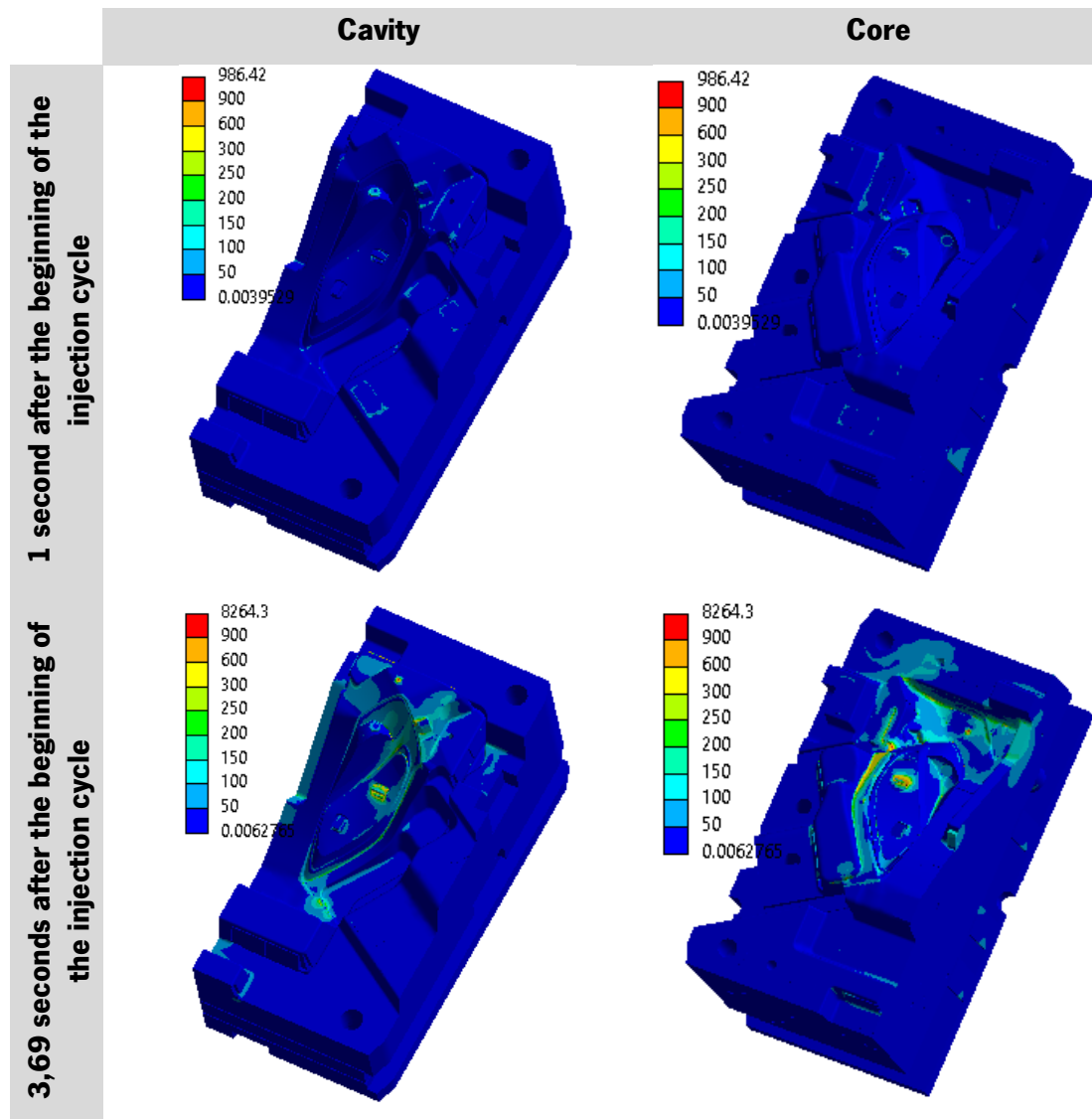


Figure 170 – von Mises stress distribution through the core and cavity sides of Fat_2 at the instant where only the clamp force is applied and at the instant of highest pressure through the engraving region.

As already mentioned, and as can be seen in Figure 171 and Figure 172, when the clamping force is applied the von Mises stresses are not significant, and as can be seen they are globally lower than 50 MPa.

But in Figure 171, can be observed that when the maximum pressure is reached in the molding region, the von Mises stresses increase substantially in the molding region, both on the core and the cavity sides. Due to the shape of the part, which in several regions has a V-shaped format, the stresses end up being significant, although the cavity side has a complex surface and is more prone to bending moments, which results in the extension of the regions with higher stresses.

In Figure 172, at the moment of maximum pressure inside the mold, the stresses rise significantly.

But here, unlike what happens in the molding region, the core side is where the highest stresses are located, and there are large regions with von Mises stress values above 300 MPa, becoming relevant regions for fatigue resistance analysis.

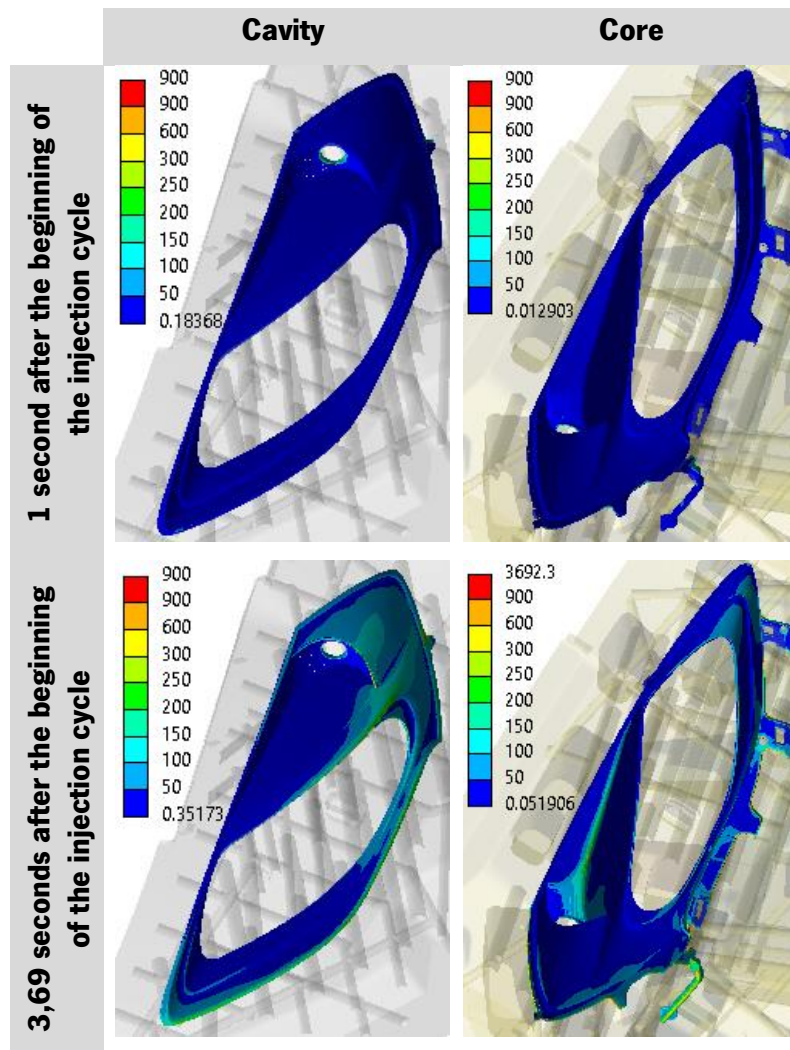


Figure 171 – von Mises stress distribution through the core and cavity engraving regions of Fat_2 at the instant where only the clamp force is applied and at the instant of highest pressure through the engraving region.

6.1.2.3 Results for the tool Fat_3

For the injection mold Fat_3 which although it has an identical injection concept to the Fat_1 and Fat_2 molds it ended up having a slightly different behavior. For the construction of the numerical model of this mold we also only considered half a mold for the same reasons as the previous molds. As can be seen in Figure 173, although it doesn't present significant stresses in the core and cavity blocks and respective molding components, as it also happens in the Fat_2 injection mold when the clamping force is applied, but at the same instant the structure of the mold already presents stresses around 150 MPa, that is, as it occurred in the Fat_1 mold. But, also as in Fat_1 mold, even when the maximum pressure is reached inside the molding region, the stress distribution in the mold structure does not vary in a relevant way.

The molding region and its partition plane, as shown in Figure 173 when the pressures inside the mold reach maximum values the stresses both in the molding region and in several regions of the partition plane increase significantly. In addition to the molding regions and partition plane, the locking cones placed all around the injection mold on the parting plane are heavily stressed. Taking into account the level of stress on the locking cones it can be concluded that in this mold the cavity is contributing for the components on the core side to be able to withstand the pressure caused by the molten polymer on them without moving from its position in order to avoid causing any kind of aesthetic defect on the part.

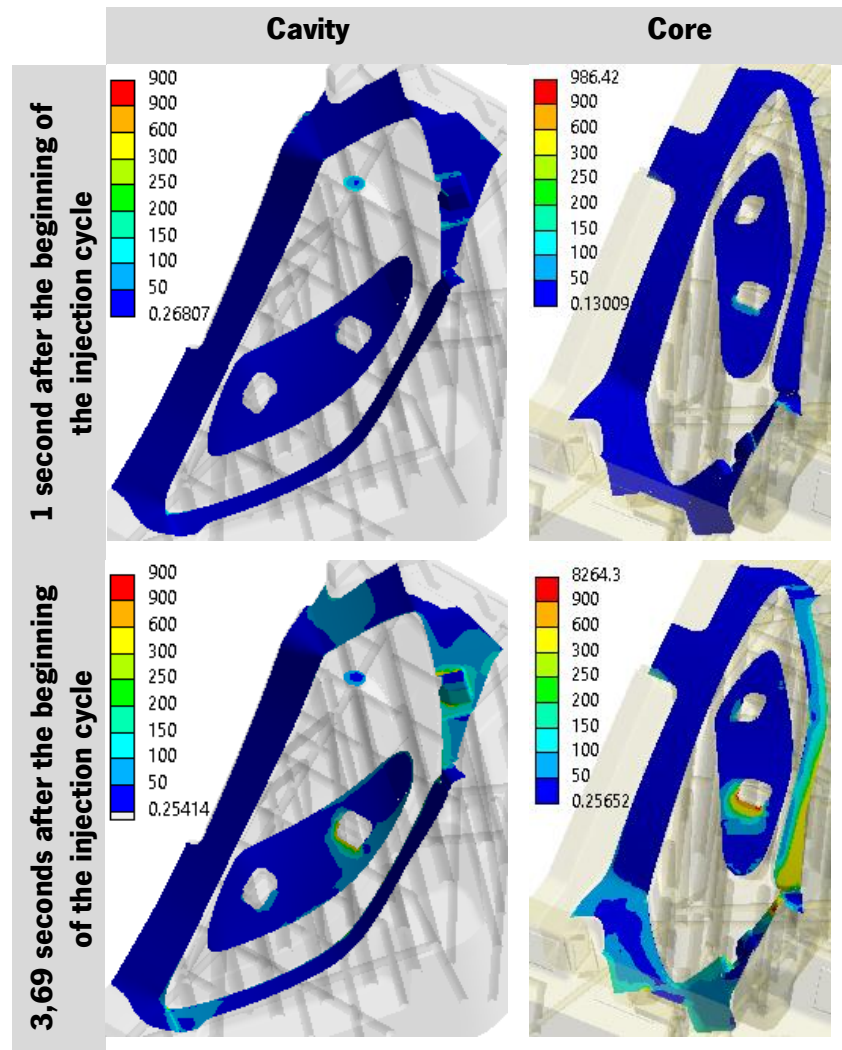


Figure 172 – von Mises stress distribution through the core and cavity fitting regions of the partition plane of Fat_2 at the instant where only the clamp force is applied and at the instant of highest pressure through the engraving region.

By analyzing the molding regions of the cavity and of the Fat_3 mold in Figure 174 it can be observed in more detail that when the clamping force is applied there are some regions on the cavity side that are slightly stressed, these regions are located near the adjustment zone in the partition plane but as the injection pressure increases the stresses eventually decrease. Although it is visible that both on the cavity side and on the core side the injection pressure has a great impact on the notches, the V-shape of the surfaces leads to a higher stress on the root of the notches and bordering regions.

However, on the cavity side, taking into account that it has a more pronounced notch, the higher stresses end up being located in the notch, while on the tap side where the notch on the molding surface is less pronounced, i.e., the V-shape has a greater angle between surfaces leads to regions with higher stresses having a greater extension. Taking into account that there are sections in the molding regions on the cavity and core side with stresses higher than 250 MPa and 300 MPa respectively, these regions are susceptible to the appearance of fatigue cracks and therefore must be properly analyzed.

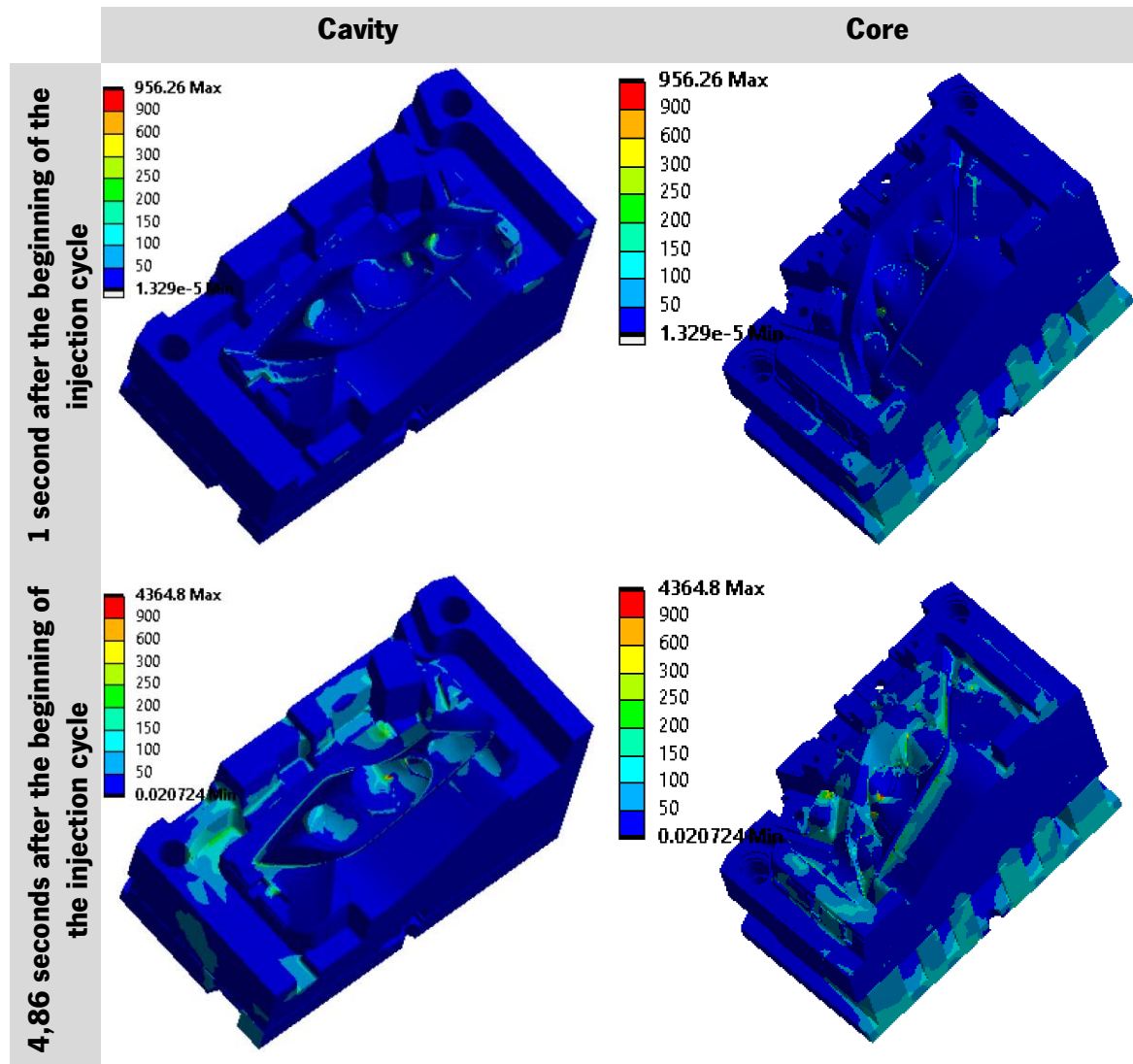


Figure 173 – von Mises stress distribution through the core and cavity sides of Fat_3 at the instant where only the clamp force is applied and at the instant of highest pressure through the engraving region.

For the fitting region of the partition plane where can be seen the von Mises stresses at the instants where only the clamping force is applied and then at the instant of maximum pressure in the molding zone in Figure 175. Here as already mentioned it can be observed that when only the clamping force is applied there are some regions where the stresses end up being a bit more solicited, when a more careful analysis is made it can be noted that these regions, both on the core and the cavity side, are locations where there are notches that lead to bending moments appearing when the clamping force is applied.

However, when the maximum pressure is reached inside the injection mold, although some of the regions that present higher stresses when only the clamping force is applied will eventually recover to residual stresses. It can also be seen, especially on the fitting area on the core side, that the fitting area near the molding region is highly stressed. So here also these regions should undergo a fatigue resistance analysis in order to understand their endurance.

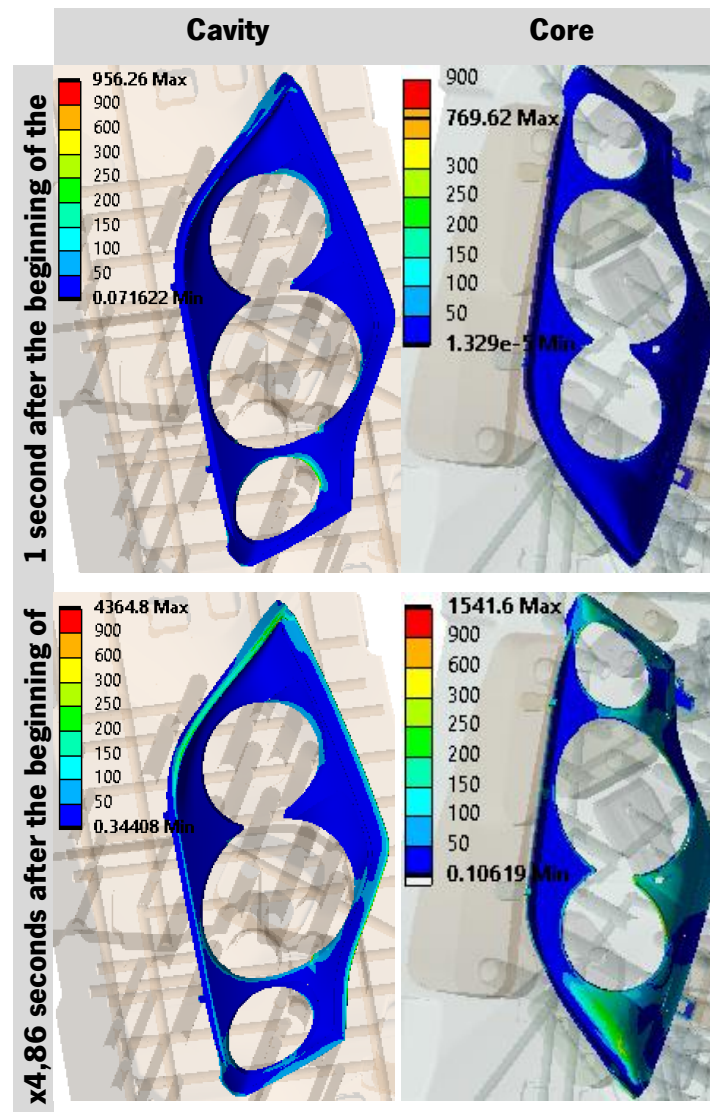


Figure 174 – von Mises stress distribution through the core and cavity engraving regions of Fat_3 at the instant where only the clamp force is applied and at the instant of highest pressure through the engraving region.

6.1.2.4 Results for the tool Fat_4

As already mentioned in section 6.1.1.4, the Fat_4 injection mold has a different injection design from the previous injection molds, and this means that unlike the previous molds in which the pressure inside the mold would increase until it reached a maximum and then return to residual, here it will increase until the second sequence of nozzles is opened, at which point the pressure will decrease and then increase again until a maximum pressure inside the mold is reached. With this there will be regions that will present more than one load cycle, although for the results that will be presented in this section

only the results for the instant when only the clamping force is applied and at the instant of greatest pressure inside the mold will be presented. For the Fat_4 mold, as for the previous ones, only half of the model is considered because this injection mold is producing two equal parts, but considering their distribution in the mold, they end up making the mold symmetrical.

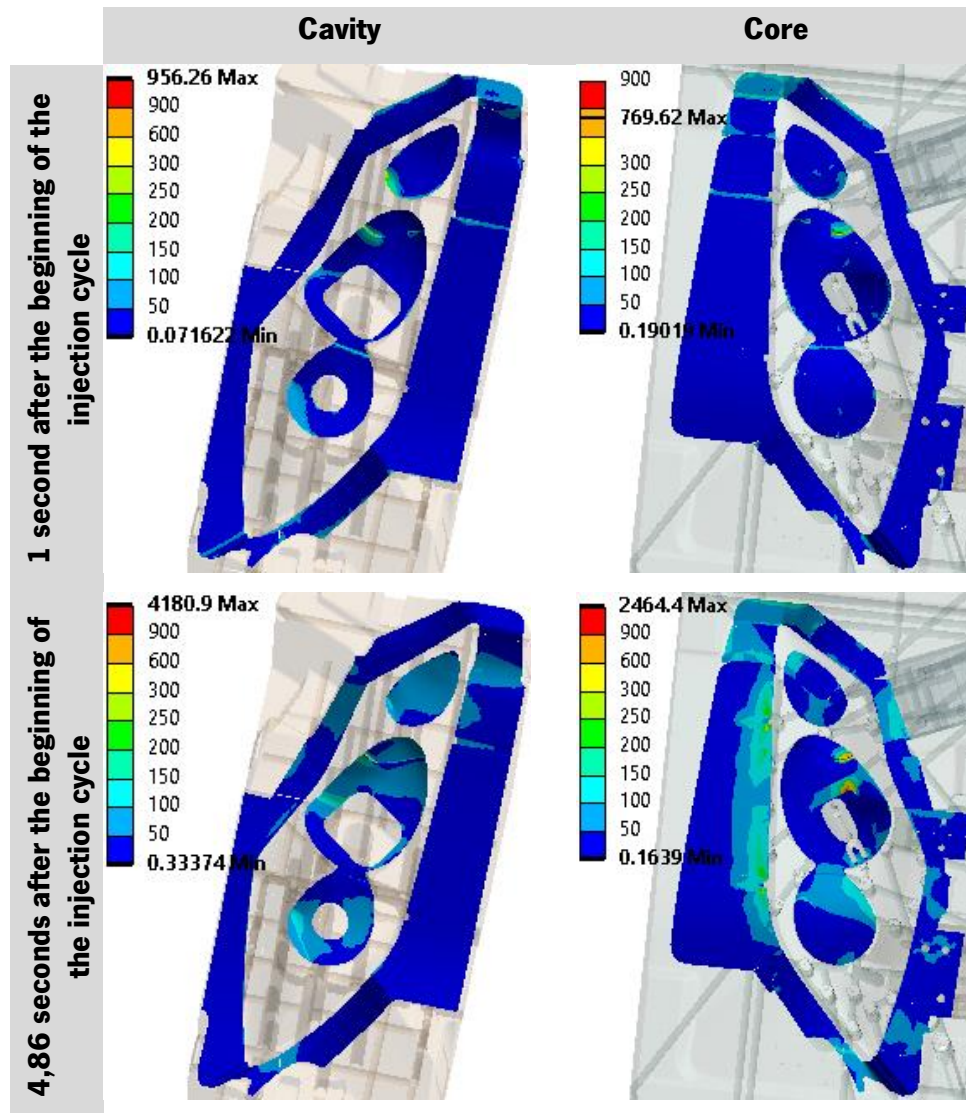


Figure 175 – von Mises stress distribution through the core and cavity fitting regions of the partition plane of Fat_3 at the instant where only the clamp force is applied and at the instant of highest pressure through the engraving region.

Figure 176 shows the distribution of von Mises stresses in the Fat_4 mold, where it can be seen from the start that at the instant when only the clamping force is applied, both on the core and cavity sides, there is already an increase in stresses. For the cavity side these are more visible in the engraving near the partition plane and near the locking cones. On the core side there is an almost uniform increase of stresses in the pressure plates, which is normal because they are subject to compression due to the clamping force, but it should be noted that the stress distribution is very homogeneous between the pressure plates, which leads to the conclusion that the forces are well distributed throughout the mold. Also, on the core side, it can be noticed that the mold structure is being very stressed only with the

clamping force, but there is no significant change at the instant when the maximum pressure inside the mold is reached. It should also be noted that there is a region where the stresses are higher than 300 MPa, this is due to small geometric interferences which having in mind that they are located away from the most mechanically solicited areas it can be disregarded because this will not influence the quality of the study within the relevant regions.

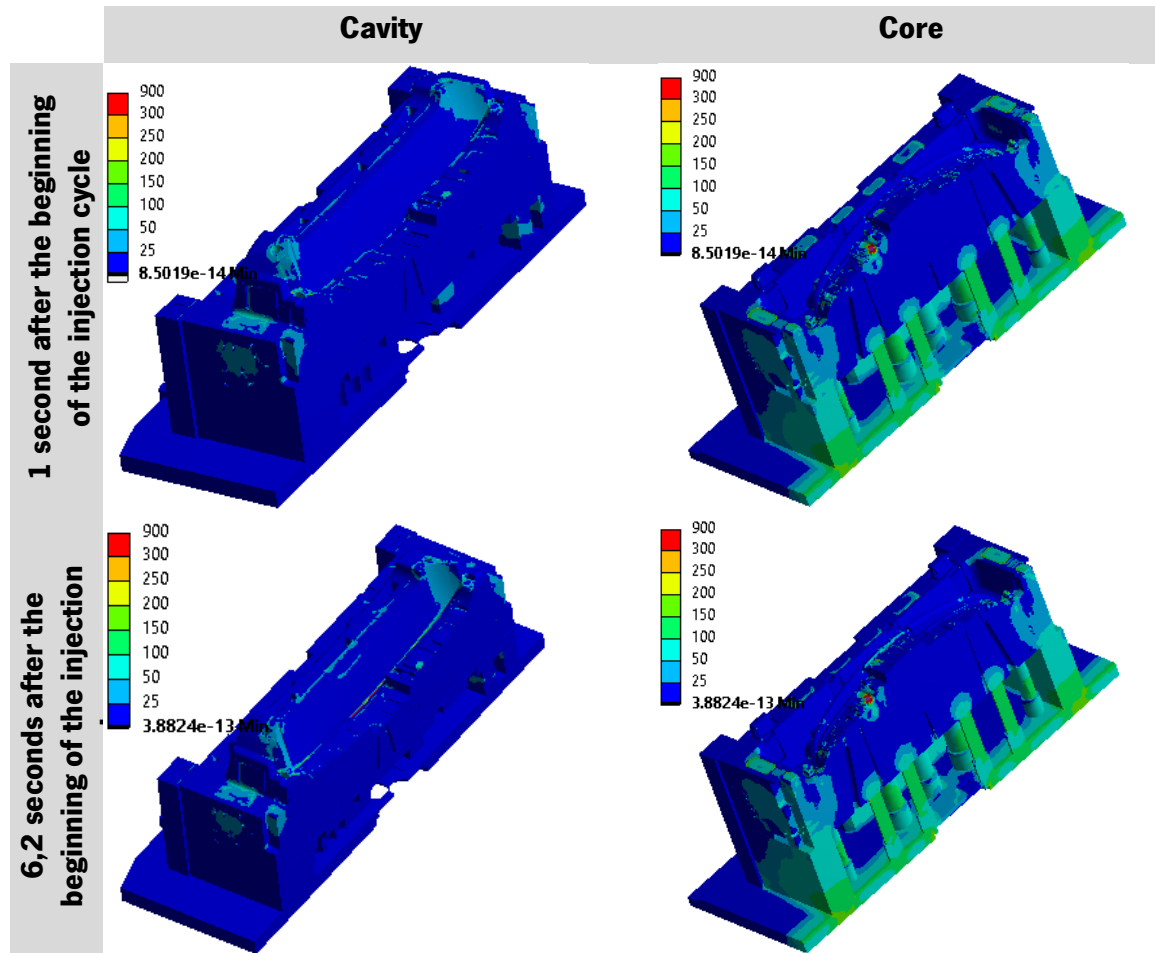


Figure 176 – von Mises stress distribution through the core and cavity sides of Fat_4 at the instant where only the clamp force is applied and at the instant of highest pressure through the engraving region.

For the cavity side it is found that between the instant when only the clamping force is applied to the mold the global von Mises stresses do not vary much to the instant of maximum pressure in the mold. Although in the molding region some differences are notorious, because due to the notch in the molding region when the pressure is applied to the engraving regions it ends up being significantly solicited and for this reason it is a relevant region for analysis of its fatigue endurance. Since the part is symmetrical, the stresses are also symmetrical throughout the mold, i.e. this mold turns out to be bi-symmetrical. Therefore, to facilitate inspection of the results presented in Figure 177 and Figure 178 only one quarter of the mold will be presented.

By analyzing Figure 177 it can be concluded that right at the application of the clamping force the cavity already shows stresses although globally below 75 MPa but already located close to the geometric

discontinuities. Because the V-shape of the part to be injected will translate into a shape of the cavity in which when the clamping force is applied bending moments already began to be applied in the notches of the part.

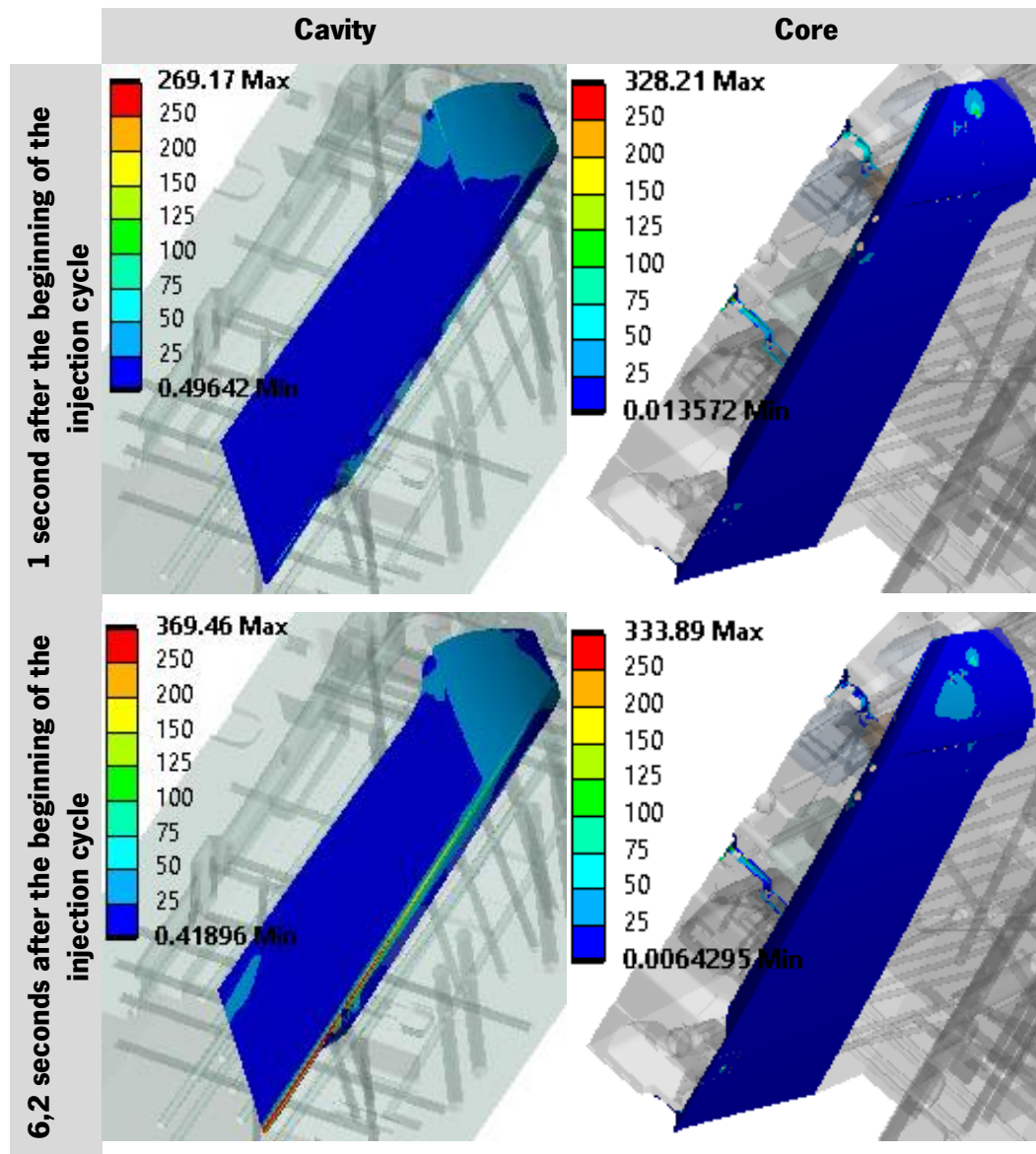


Figure 177 – von Mises stress distribution through the core and cavity engraving regions of Fat_4 at the instant where only the clamp force is applied and at the instant of highest pressure through the engraving region.

But when the maximum pressure is reached the stresses in the cavity increase to their maximum in the notch that runs the cavity longitudinally, and the values are highest in the plane of symmetry of the engraving region because this is the least resistant area of the mold due to the shape of the molding area of the cavity. For the core side there are no significant von Mises stresses even when the maximum pressure instant is reached.

For the parting plane adjustment region either at the instant when only the clamping force is applied or at the instant of highest pressure in the mold the von Mises stresses are not very significant, as can be seen in Figure 178.

Although, due to the shape of the part there are many notches that can be susceptible to the appearance of cracks even for low alternating stresses because the surface finish is an important factor for their fatigue resistance.

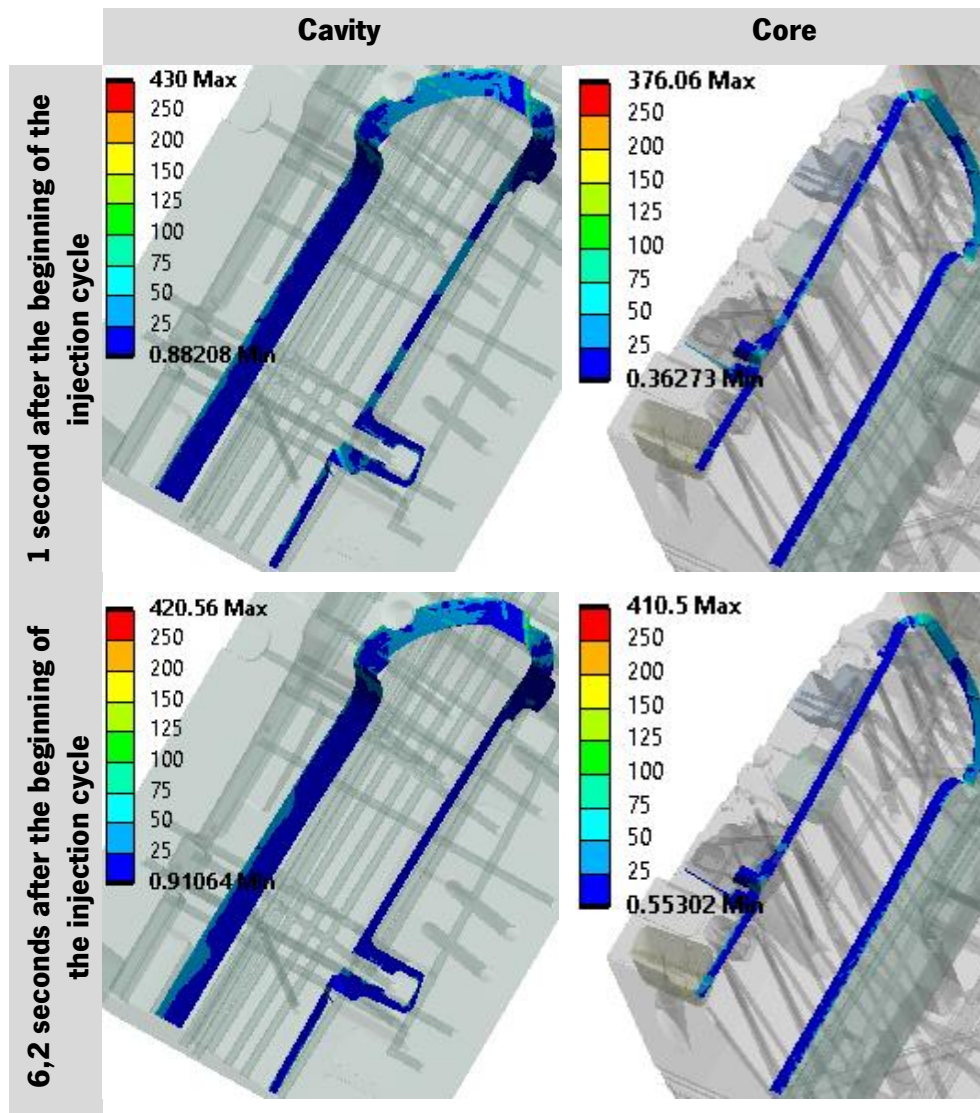


Figure 178 – von Mises stress distribution through the core and cavity fitting regions of the partition plane of Fat_4 at the instant where only the clamp force is applied and at the instant of highest pressure through the engraving region.

6.1.2.5 Results for the tool Fat_5

In contrast to what was presented for the previous molds, for the Fat_5 injection mold, the results are presented for three time-steps, with the first still being the instant when only the clamping force is being applied to the mold, the next is when the maximum stresses at the most fragile components of the injection mold are reached, and the third is when the maximum pressure inside the injection mold is reached. As already presented in section 6.1.1.5 for the injection of this part four hot nozzles are used to inject the molten plastic directly into the molding region.

This will lead the injection mold to present a different behavior from the previous ones, as can be observed in Figure 179, globally the stresses in the injection mold are residual, only at the instant of

highest pressure in the molding region that occurs 8.51 seconds after the start of the injection cycle it is possible to observe some points where the von Mises stresses are between 150 and 200 MPa. But considering the complexity of the part, the following figures will be more elucidative since they show the stresses at the most critical points of the injection mold.

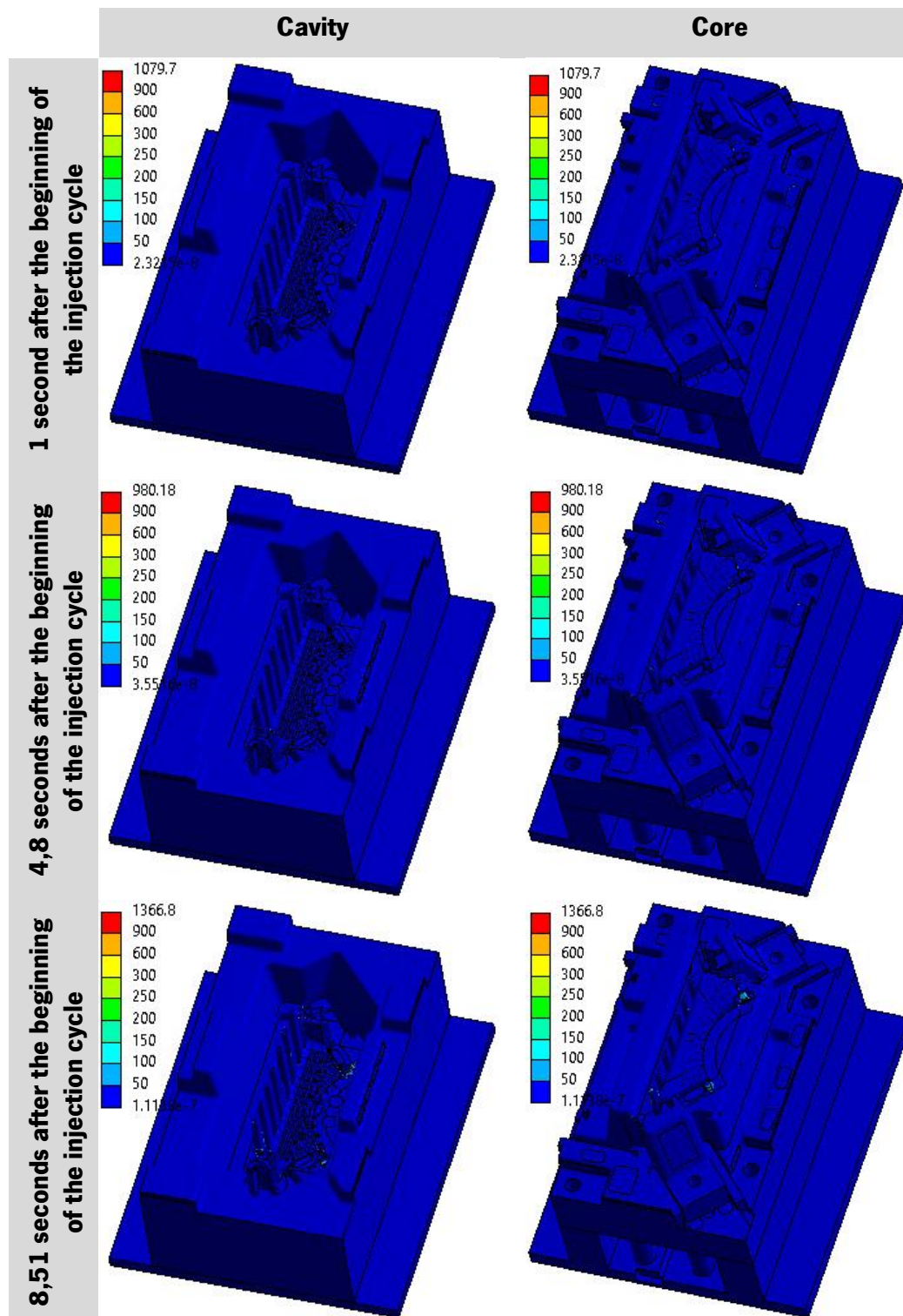


Figure 179 – von Mises stress distribution through the core and cavity sides of Fat_5 at the instant where only the clamp force is applied and at the instant of highest pressure through the engraving region.

When analyzing the results of the von Mises stresses of the molding region and the partition plane

adjustment region that are presented in Figure 180 and Figure 181 respectively, it can be observed that for the instant when only the clamping force is applied, there are no regions with relevant stresses that could lead to damage.

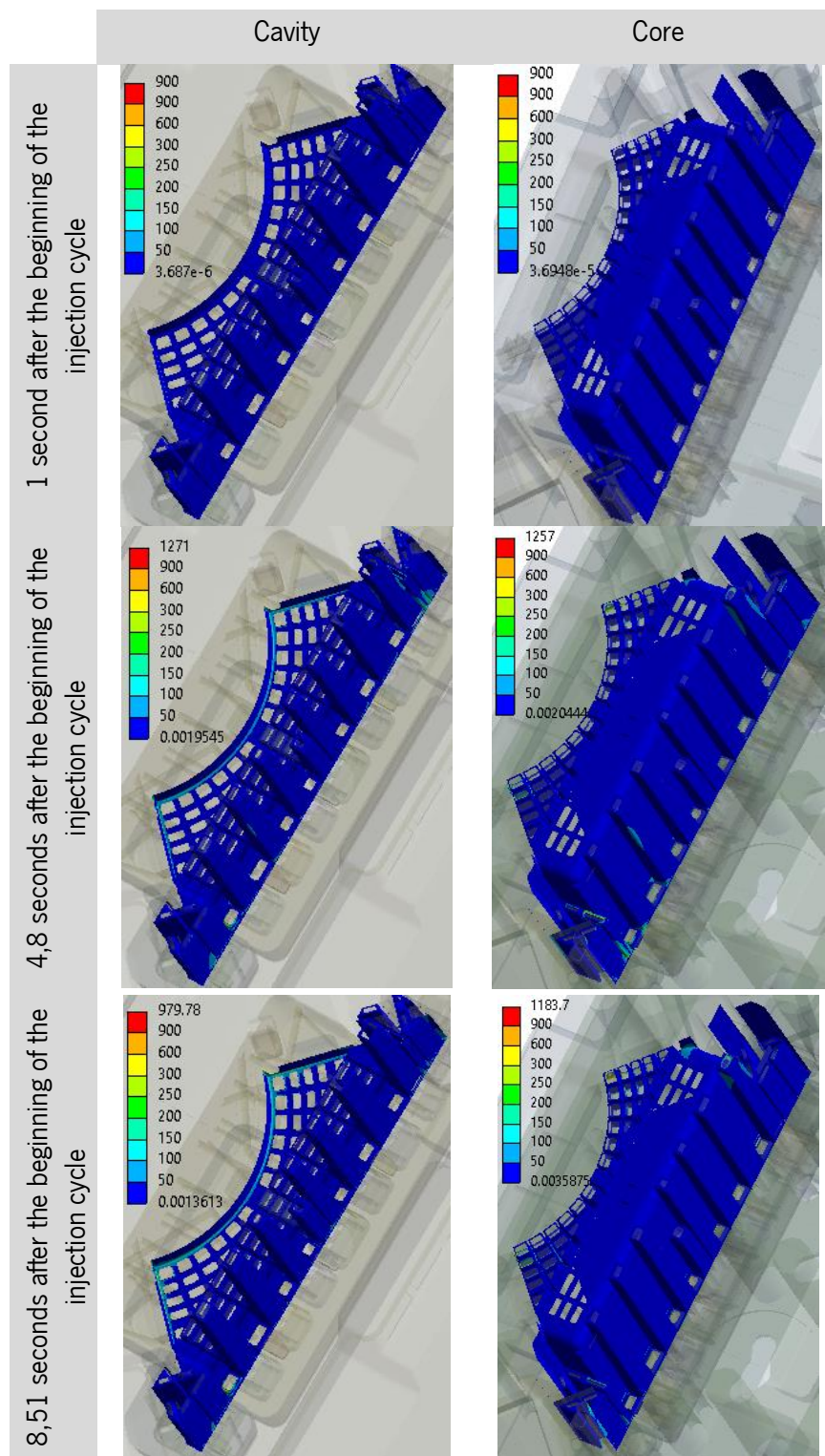


Figure 180 – von Mises stress distribution through the core and cavity engraving regions of Fat_5 at the instant where only the clamp force is applied and at the instant of highest pressure through the engraving region.

However, in the molding region shown in Figure 180, with the pressure exerted by the molten polymer it can be observed that both on the side of the core and on the side of the cavity in several notches,

stresses in the range of 150 to 200 MPa are already visible. Another detail that can be observed is that the location of the most stressed regions on the core side varies significantly between the 4,8 to 8,51 seconds after the start of the cycle, this is due to the filling pattern and taking into account that the geometry is very susceptible to deformations in the steel components during the filling phase this should be analyzed with care.

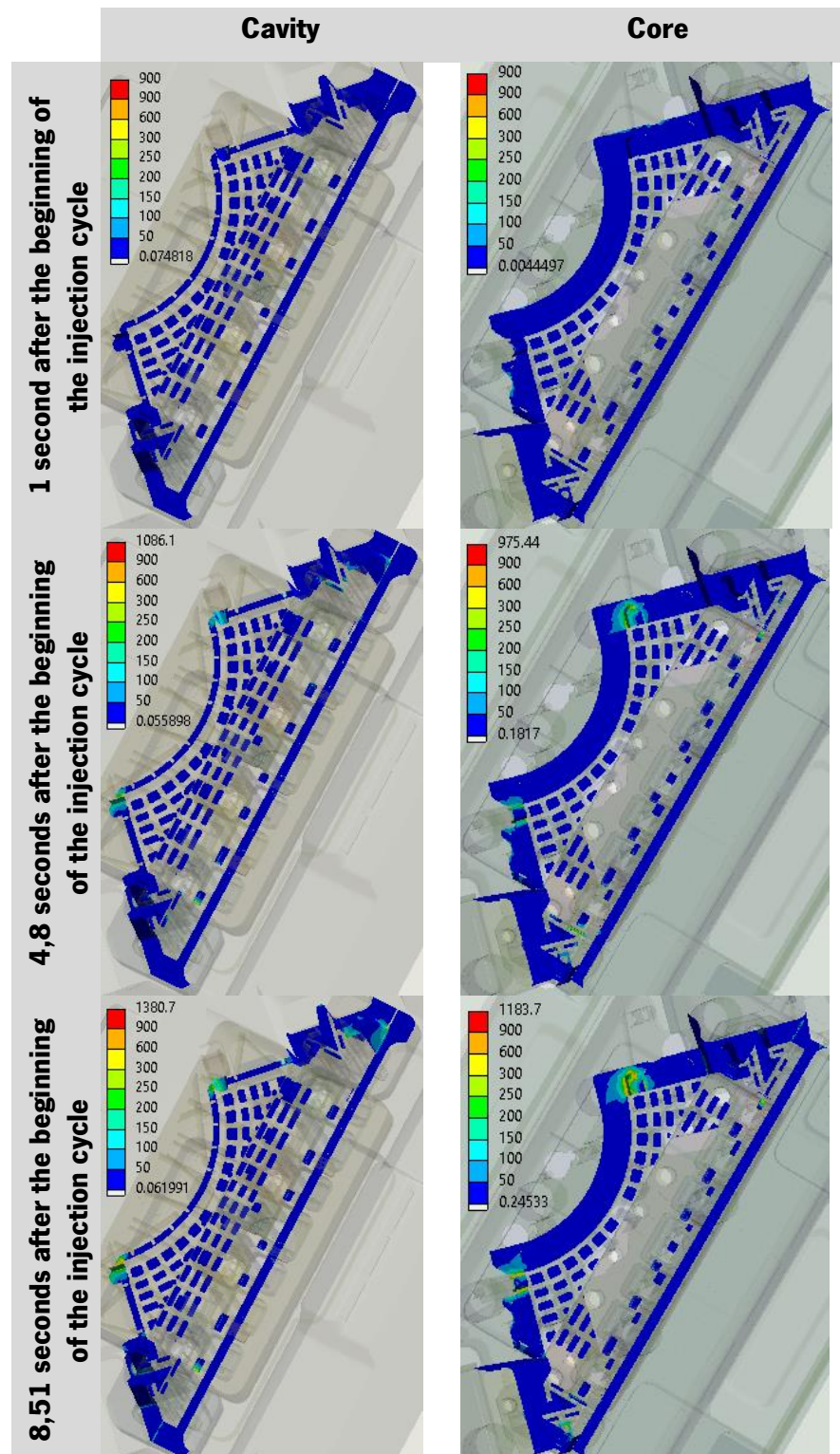


Figure 181 – von Mises stress distribution through the core and cavity fitting regions of the partition plane of Fat_5 at the instant where only the clamp force is applied and at the instant of highest pressure through the engraving region.

On the cavity side it is observable that at the instant of highest pressure in the mold there are fewer stressed regions, i.e., as it is the first region to be completely filled by polymer and consequently with more balanced pressures on all surfaces of the molding region, the components most susceptible to displacements during the filling phase eventually return to their initial positions and thus the stresses become zero or residual.

Both for the core and cavity sides, as described, there are components that end up deforming during the injection phase and with this may occur the appearance of cracks. Therefore, these components should be properly studied for their fatigue endurance.

For the partition plane adjustment region, as can be observed in Figure 139, when the clamping force is applied there are no locations with relevant stresses. But, as the molten polymer fills the molding region it can be seen that both on the cavity and core sides there are some regions where the von Mises stresses increase significantly, and always close to notches where the geometric variations of the partition plane are significant, i.e., the influence of the partition plane shape is preponderant for the evolution of stresses in this mold. As can be observed in Figure 181, where the two moments of time in which there is pressure applied inside the mold are presented, and the presence and evolution of stresses with the increase of internal pressure is notorious, always in the same regions, while the remaining regions continue with residual or zero values, which has not always been the case in the injection molds presented so far.

As mentioned, this mold has a different design than the ones presented before, and from the analysis performed it was found that there is a set of inserts that are located on the cavity and core side that are very stressed during the injection process. In Figure 182 can be observed the full set of inserts where the pressure distribution is presented in each time-step, the distribution of von Mises stresses in the inserts for the respective time-steps and finally the total deformations in the most stressed inserts are presented. At first glance it is noticeable that at the instant when the pressures are globally higher throughout the molding region, at 8,51 seconds after the start of the injection cycle, it is also at the instant when both the von Mises stresses and the total deformations are globally lower.

When analyzing the first instant presented, that is, at 4,8 seconds after the start of the injection cycle, it is noticeable that there is an imbalance in the distribution of pressures in the inserts, i.e., ideally the pressure should be symmetrical in the plane perpendicular to the partition plane, these imbalances lead to an accumulation of pressures on the side where the polymer has already filled the entire molding region.

But, in the case of this injection mold, where there are slim inserts that meet other slim inserts in the middle of the mold, during the injection phase, when pressure is applied to the insert on the cavity side,

it deforms and causes the deformation of the insert that is in contact from the core side. As while there is only polymer on the cavity side, the inserts on the core side end up not having any force opposing their bending around their base and thus end up suffering a great deal of stress on the notches present at its base.

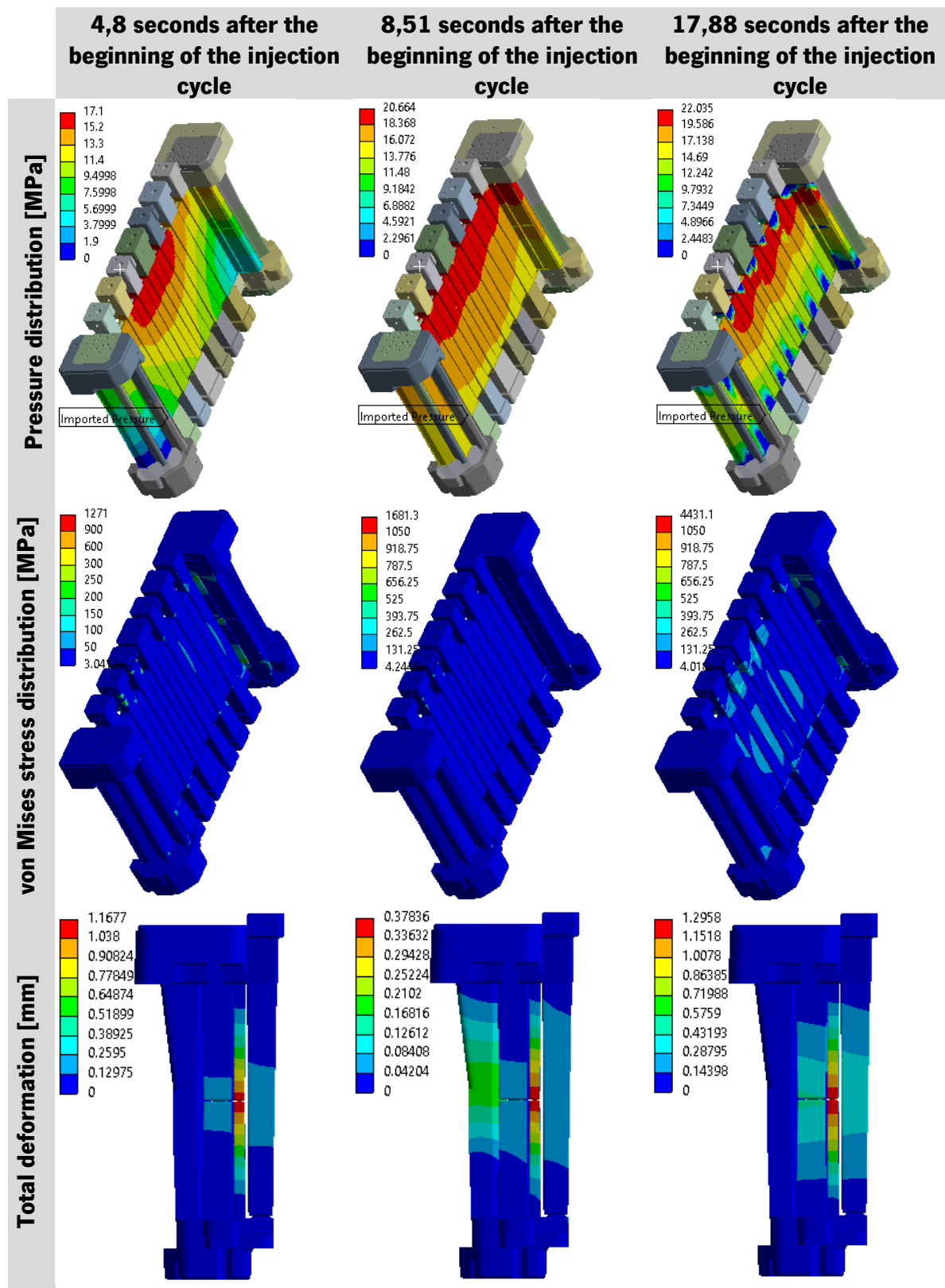


Figure 182 – Pressure load, von Mises stress distribution and total deformation of the most stressed components of Fat_5.

At 17,88 seconds after the start of the injection cycle, both the von Mises stresses and the total deformations increase again, i.e., as can be observed in the pressure distribution shown in Figure 182, at 17,88 seconds the pressures are again very unbalanced between the cavity side and the core side, being the cavity side where the highest pressures are observed. Considering this unbalance of pressures, the inserts that end up being the most fragile components of this injection mold present again higher stresses.

6.1.3 Fatigue endurance limit calculations

From the numerical simulations presented in section 6.1.2 the data regarding the principal stresses and nodes of the regions under study will be extracted. With these data it will be possible to use the application presented in section 4.3 to obtain the necessary data to perform the fatigue study of the five injection molds. Since ANSYS is able to import the calculated data the analysis will be done in the ANSYS Mechanical environment, which makes it easier to compare the results calculated by the application with those already calculated in the numerical simulation.

During this section, an analysis of the fatigue strength of the five molds under study will be performed, where it will be possible to present all the results obtained, as well as the care that should be taken with the use of this application.

For the presentation of the results, for the first injection mold under study all the results that the application exports will be presented, but for the following ones only the results obtained by the Sines method will be presented. The choice of the Sines method is due to this method's better ability to account for mean stresses.

In order to study the injection molds, the following regions will be analyzed: molding region on the cavity side, partition plane fitting region on the cavity side, molding region on the core side, and partition plane fitting region on the core side.

6.1.3.1 Results for the tool Fat_1

In a first stage, besides obtaining the structural simulation data, it is also necessary to analyze the conditions under which the injection mold will work to know which S-N curve correction factors should be used for each region under study.

Figure 183 shows the graph from which the reduction coefficients that account for the effects of surface finishing will be taken. Where the vertical red line is determined from the ultimate tensile strength of the steel and the horizontal lines are used to obtain the reduction factor for each type of surface finish. For this it is necessary to take into account that the steel used for all the molding regions is EN 1.2343, where the information regarding the steel can be consulted in Appendix A.

Considering that the molding region on the cavity side is mirror polished the value of k_s is 1, i.e., the surface finish is considered to have no influence on the S-N curve. The molding area on the core side, on the other hand, is only polished, so as can be observed from the orange line in Figure 183, the k_s value should be approximately equal to 0,84. For the fitting regions in the partition plane, either on the tap or cavity side, the same type of finish is considered, i.e. machined where the blue line of Figure 183 shows a correction factor of about 0,6.

For this calculation only one other reduction factor will be considered, and that is reliability. Considering that injection molds are unique tools and where fatigue failures cannot occur during the expected lifetime of the injection mold. Therefore, a reliability factor of 99,9% was considered, which translates into a k_r value of approximately 0,753.

This mold had an expected life span of 1.5 million cycles. This is important data to determine the coefficient of safety. Having collected all this information, can now enter all the data into the application so that the calculations can be performed.

Figure 184 shows the maximum alternating stresses in each region. This result serves to get a good idea of the most stressed regions since it summarizes in one image the highest alternating stresses values regardless of the instant in which the alternating cycle occurs. By analyzing the maximum alternating stresses, it can be observed the regions that already proved to be the most stressed in the von Mises stress analysis of Figure 168, are also the ones that present the highest alternating stresses. Although here it is easier to distinguish the most relevant regions for fatigue analysis. Considering the mentioned reduction factors, the fatigue endurance limit is approximately 398 MPa. A more extensive analysis of the molding region of the cavity shows that only one region exceeds this value.

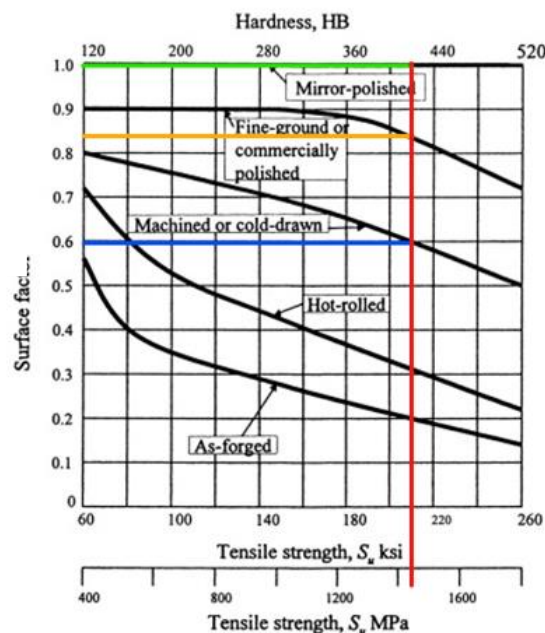


Figure 183 – Fatigue reduction coefficients for the Fat_1 tool.

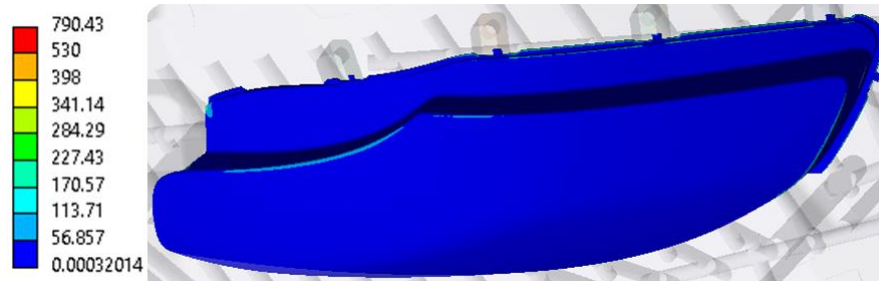


Figure 184 – Maximum alternated stress in MPa at each location by the Sines method at the cavity engraving region of Fat_1.

In Figure 185 and Figure 186 can be seen the region where the alternating stresses exceed the fatigue strength limit. Here the damage calculated by the octahedral shear stress method and the Sines method respectively is shown. Here it is seen that the damage at each injection cycle calculated by the Sines method is significantly lower than that obtained by the octahedral shear stress method.

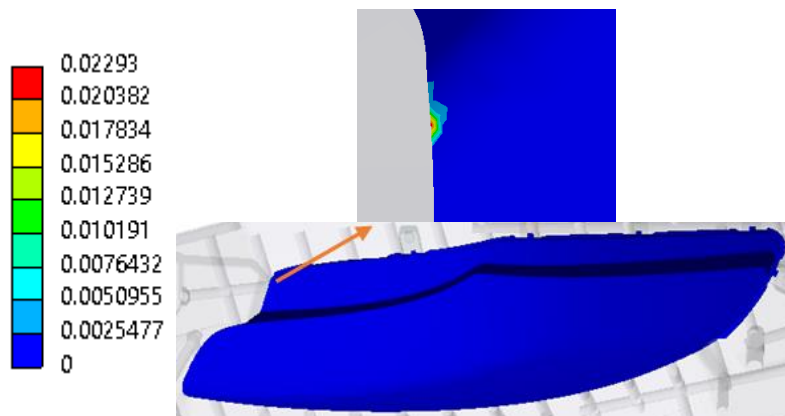


Figure 185 – Damage determined by the octahedral shear stress method at the cavity engraving region of Fat_1.

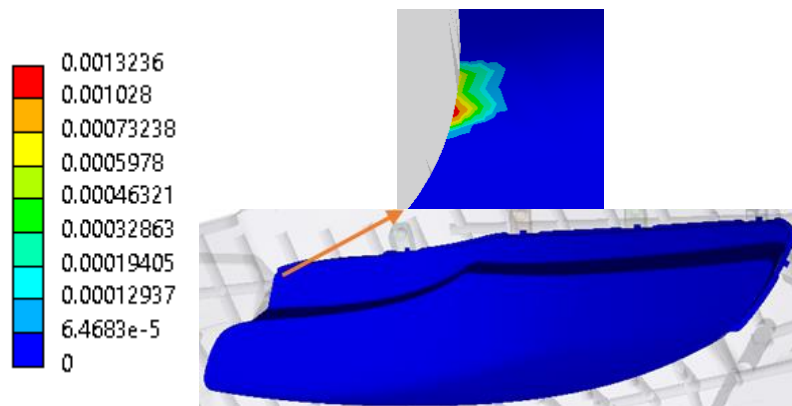


Figure 186 – Damage determined by the Sines method at the cavity engraving region of Fat_1.

However, when analyzing Figure 187, where the number of cycles that the region under analysis endures until the crack initiation determined by the octahedral shear stress and Sines methods is presented. Here it is noticeable, as expected, that the number of cycles supported until crack nucleation determined by the Sines method is higher.

Figure 188 shows the coefficient of safety determined by the two methods used in the previous calculations. Here it is visible that the region with a coefficient of safety less than 1 is higher when this is determined by the Sines method.

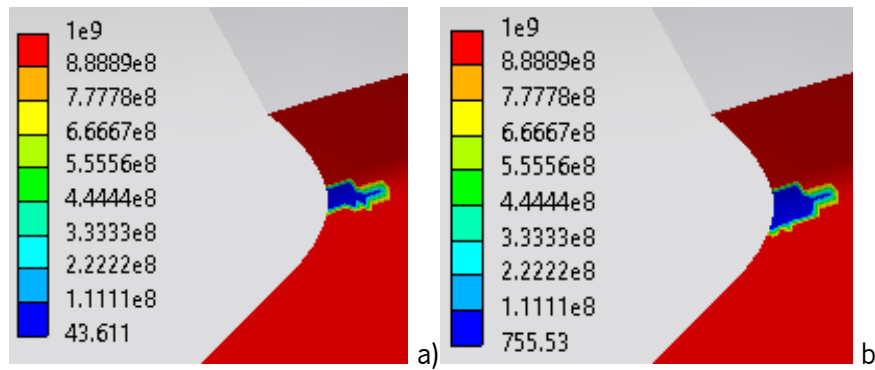


Figure 187 – Number of cycles before crack nucleation determined by a) the octahedral shear stress and by b) the Sines method at the cavity engraving region of Fat_1.

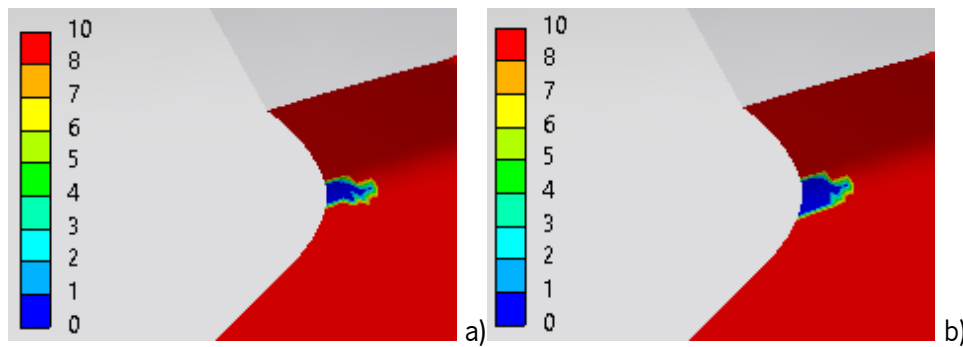


Figure 188 – Safety factor determined by a) the octahedral shear stress and by b) the Sines method at the cavity engraving region of Fat_1.

But these results do not mean that this region is critical in terms of fatigue strength, because due to the geometry a singularity may be present. Therefore, this analysis should be combined with the results obtained for the partition plane fit region on the cavity side.

In Figure 189 it can be observed that the peak of the maximum alternating stress occurs in the same region, although it can be seen that there are other regions that present high values. Where considering the reduction factors, the fatigue endurance limit is approximately 239 MPa

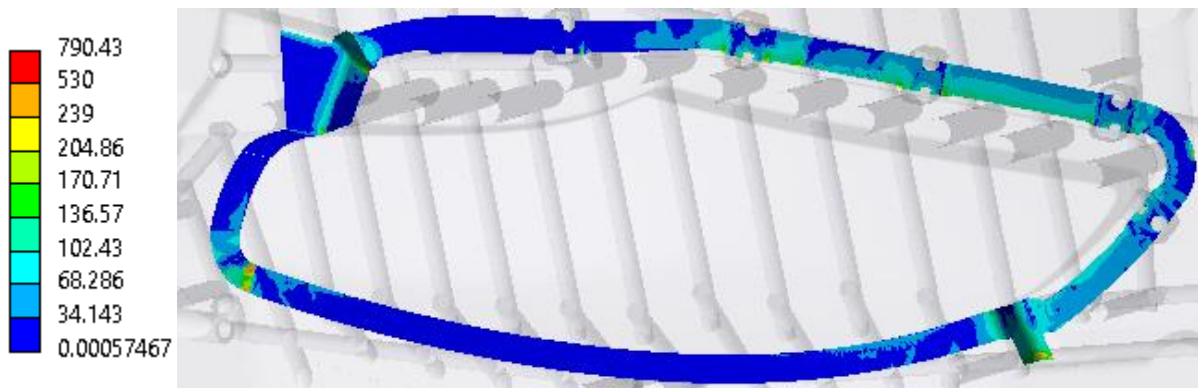


Figure 189 – Maximum alternated stress in MPa at each location by the Sines method at the fitting region on the cavity side of Fat_1.

Moving on to the damage analysis per cycle that is presented in Figure 190 and Figure 191 which correspond to the damage calculated by the octahedral shear stress method and the Sines method respectively. It is again verified that the octahedral shear stress method presents a higher value than the one obtained by the Sines method.

In Figure 192, where the number of cycles until crack nucleation is presented, it can be seen that besides being located in the region adjacent to the one detected in the molding region on the cavity side, it is also very concentrated near the radius where the division between the molding region and the cavity-side adjustment region is made. Adding this to the results of von Mises stresses, it can be observed that these values are the result of a singularity in the numerical model.

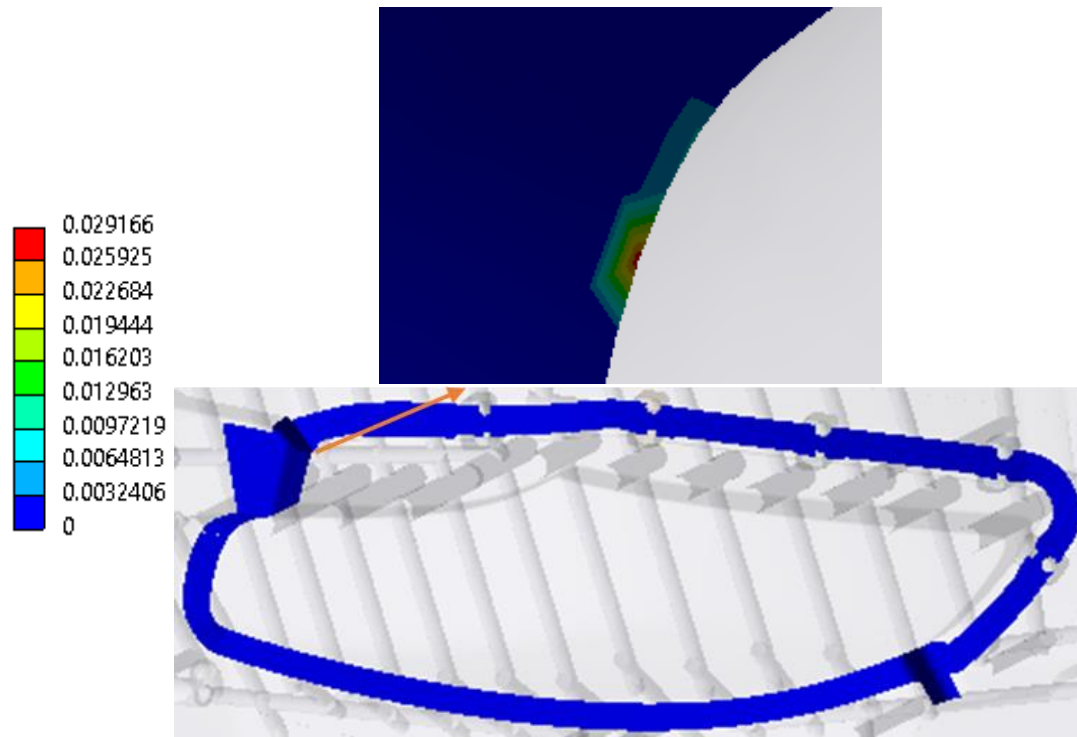


Figure 190 – Damage determined by the octahedral shear stress method at the fitting region on the cavity side of Fat_1.

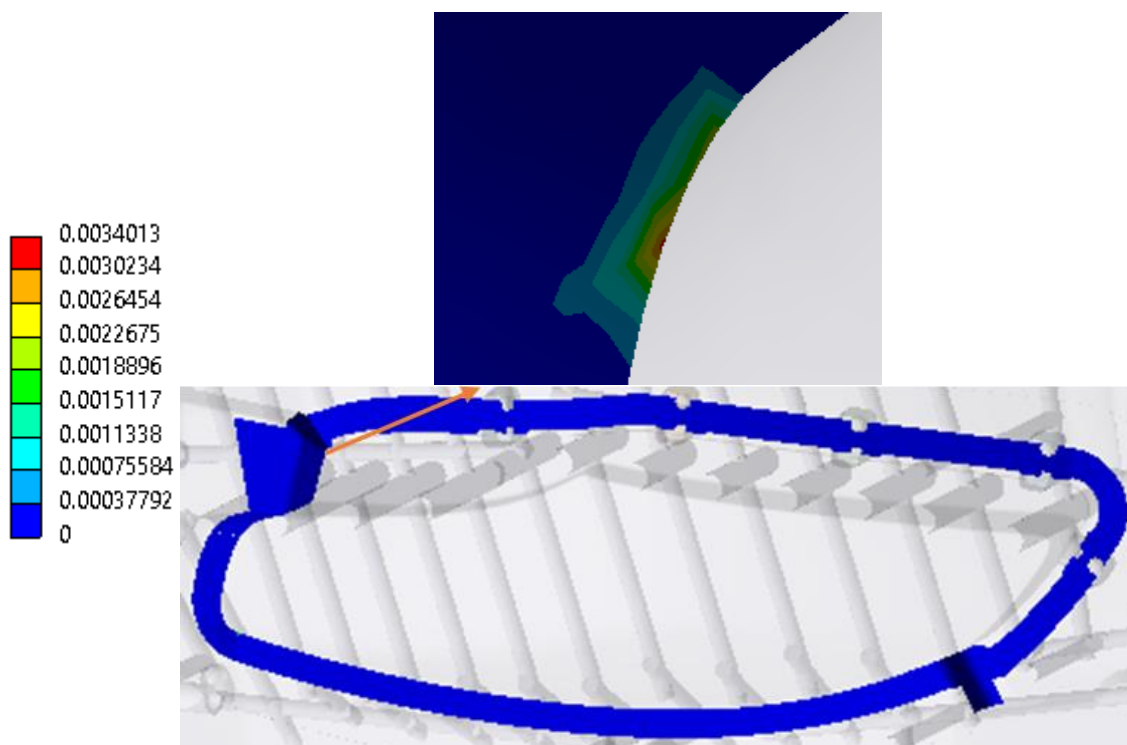


Figure 191 – Damage determined by the Sines method at the fitting region on the cavity side of Fat_1.

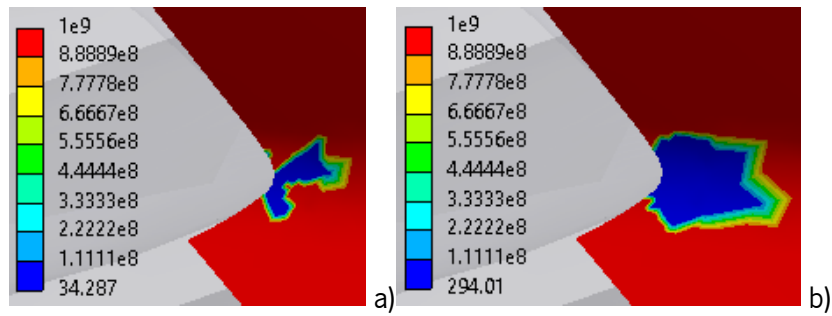


Figure 192 – Number of cycles before crack nucleation determined by a) the octahedral shear stress and by b) the Sines method at the fitting region on the cavity side of Fat_1.

The results obtained for the fatigue safety coefficient shown in Figure 193, are in line with those obtained for the molding region where the Sines model proves to be a relatively more conservative.

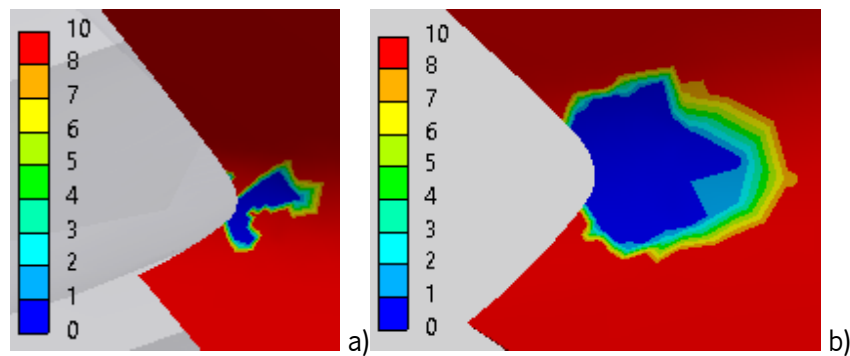


Figure 193 – Safety factor determined by a) the octahedral shear stress and by b) the Sines method at the fitting region on the cavity side of Fat_1.

The developed application works with .csv files, since this format is used by many numerical software's, but this format has a limitation on the allowed number of rows and columns. As the number of nodes of the numerical model of the molding region of the male side exceeds this maximum number of lines it was necessary to divide this region into three parts that are presented in Figure 194.

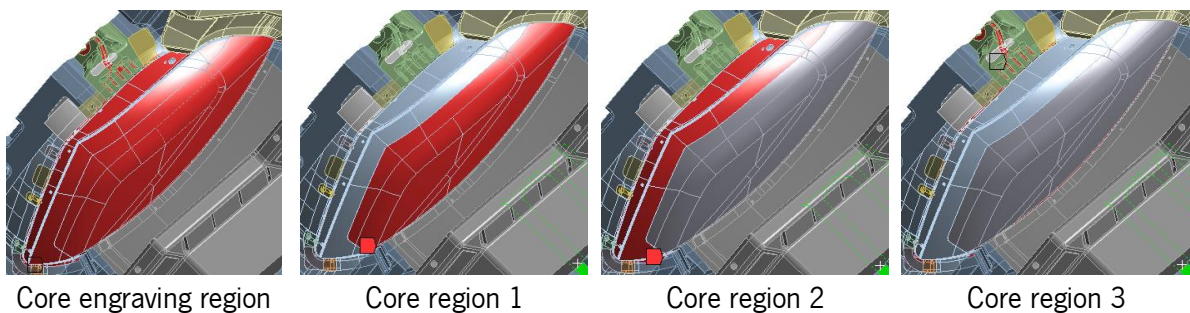


Figure 194 – Core engraving region and the sections under individual analysis.

The molding region on the tap side has a lower fatigue endurance limit than the molding region on the cavity side, since the reduction coefficients are different. Here the value of the fatigue endurance limit is 334 MPa. From the analysis of the maximum alternated stresses that is presented in Figure 195 it is possible to observe that these are essentially located in the regions with blends with small radius. Although it is clear that there is a region that has a high alternated maximum stress value that is next to a face that limits the component.

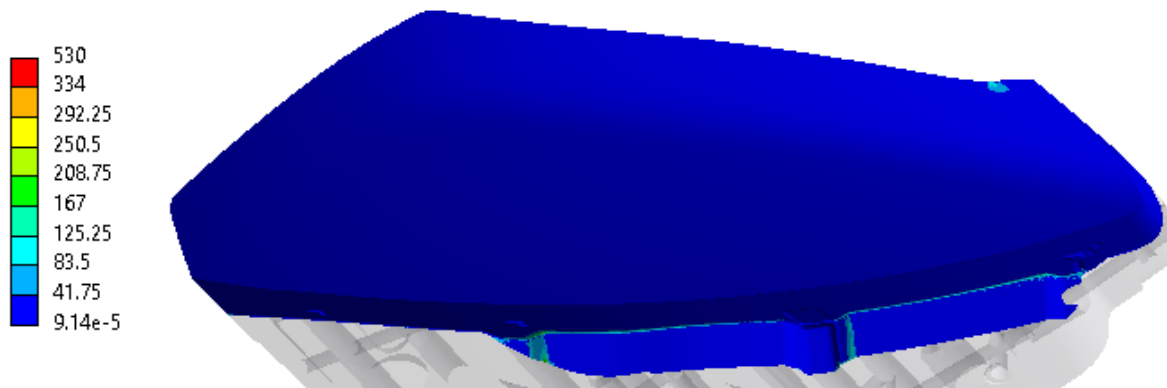


Figure 195 – Maximum alternated stress in MPa at each location by the Sines method on the core engraving region 1 of Fat_1.

In this region it can be observed from the results presented for damage per cycle in Figure 196 and Figure 197 that again the Sines method tends to be more sensitive. Here by the octahedral shear stress method no damage was detected, while for the Sines method damage was detected in a region, although when analyzing Figure 198 where the number of cycles until crack nucleation is presented it is verified that the number of cycles supported is higher than the 1.5 million cycles that the component will have to support.

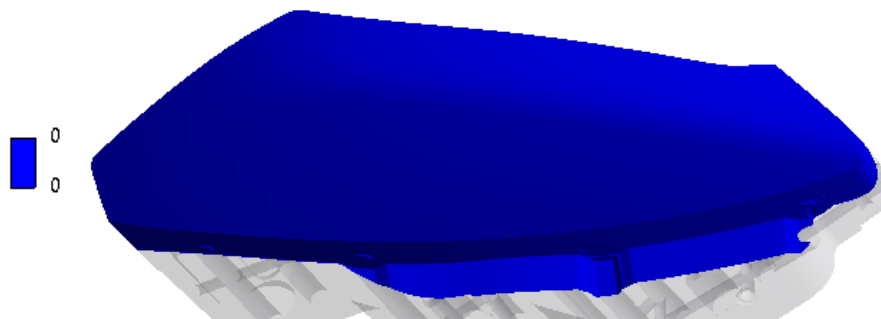


Figure 196 – Damage determined by the octahedral shear stress method on the core engraving region 1 of Fat_1.

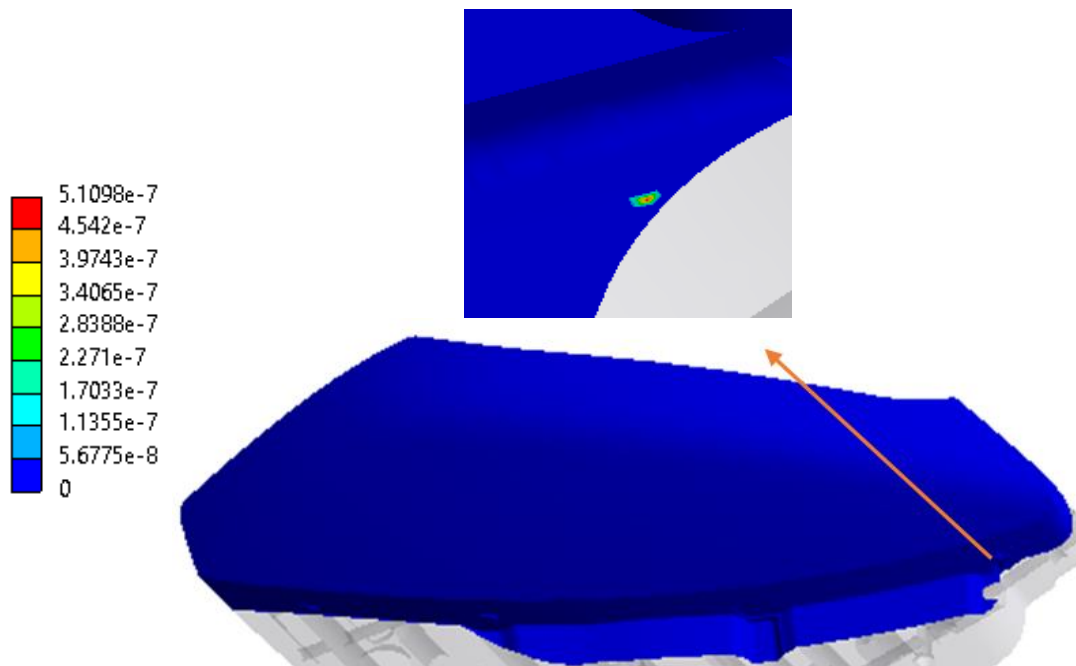


Figure 197 – Damage determined by the Sines method on the core engraving region 1 of Fat_1.

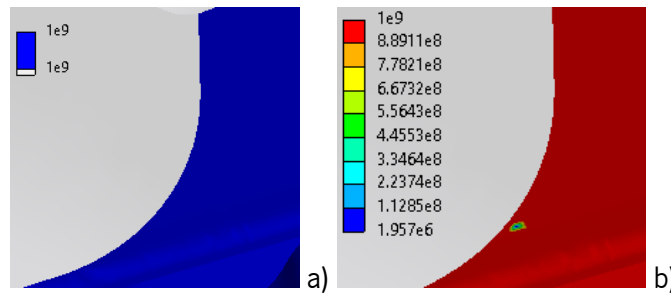


Figure 198 – Number of cycles before crack nucleation determined by a) the octahedral shear stress and by b) the Sines method on the core engraving region 1 of Fat_1.

As would be expected from the results obtained for damage, the fatigue safety coefficient for octahedral shear stress is equal to or greater than 10. While for the Sines method, even in the region where alternating stresses exceeding the fatigue endurance limit were detected, the value of the coefficient of safety is greater than 1 because as had already been verified the component is capable of withstanding more than 1.5 million cycles as can be observed in Figure 199.

However, from a more careful analysis of the results it appears that this alternated stress peak is related to a singularity present in the model.

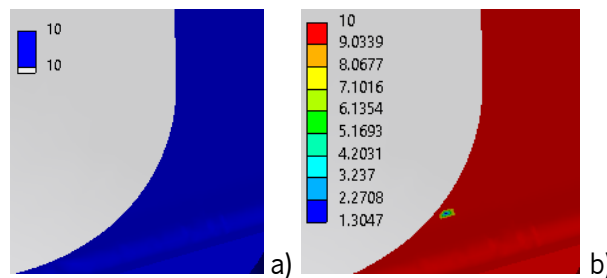


Figure 199 – Safety factor determined by a) the octahedral shear stress and by b) the Sines method on the core engraving region 1 of Fat_1.

For region 2 it can be seen that the highest alternating stresses are also found near the sharpest radius, as presented in Figure 200. But it is verified that the highest values are near the edge of the region under study, as it will be possible to observe in more detail when the damage analysis is done.

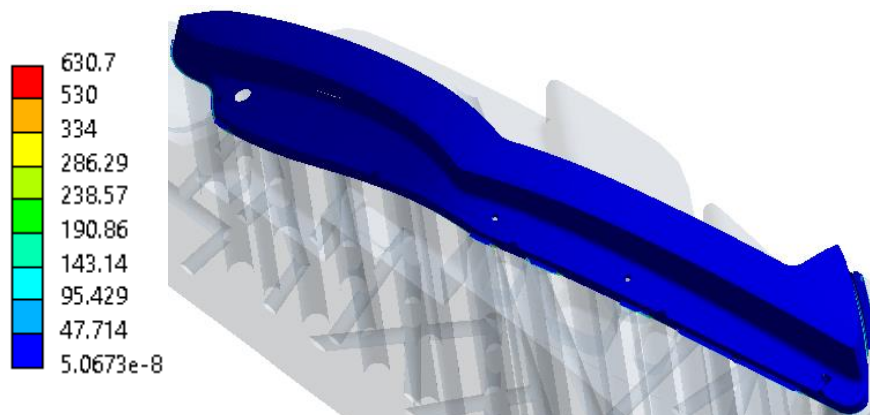


Figure 200 – Maximum alternated stress at each location by the Sines method on the core engraving region 2 of Fat_1.

Here two zones show the highest damage, as shown in Figure 201 and Figure 202 for the damage results determined by the octahedral shear stress method and the Sines method. Although for the Sines

method, it seems that in one of the zones the damage is much lower, when the results of the number of cycles until the crack nucleation are presented, it is verified that what happens is the opposite.

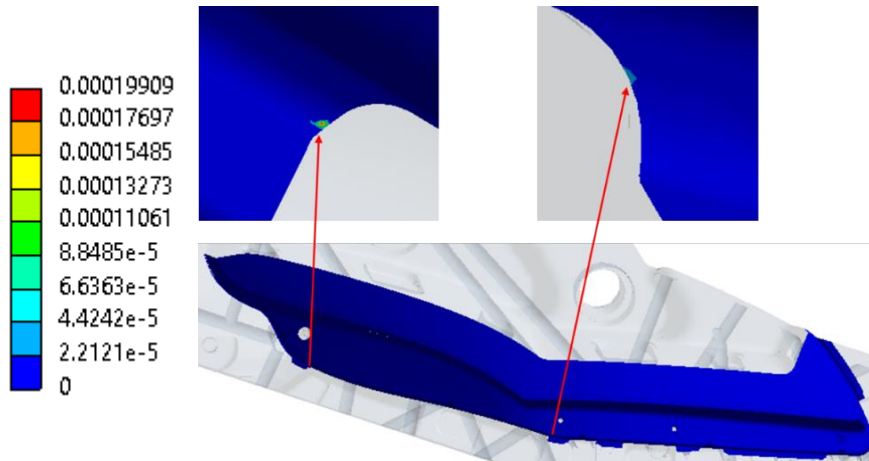


Figure 201 – Damage determined by the octahedral shear stress method on the core engraving region 2 of Fat_1.

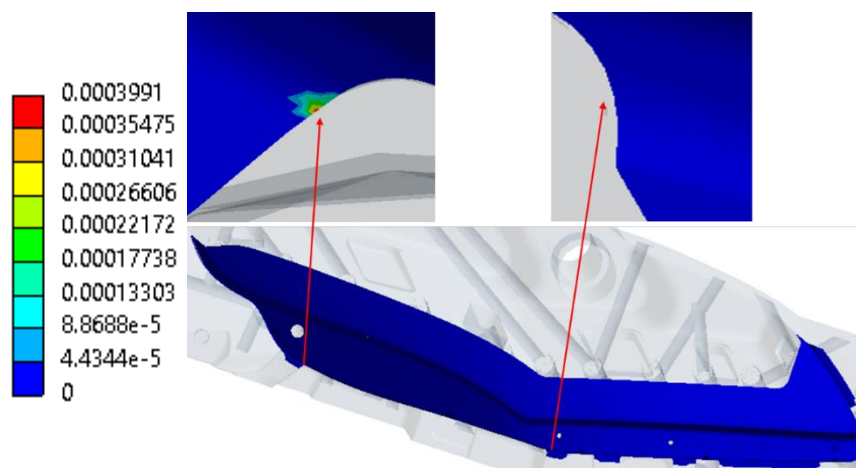


Figure 202 – Damage determined by the Sines method on the core engraving region 2 of Fat_1.

With the analysis of the number of cycles until crack nucleation it again appears that the Sines model tends to be more sensitive than the octahedral shear stress, as can be observed in Figure 203 and Figure 204.

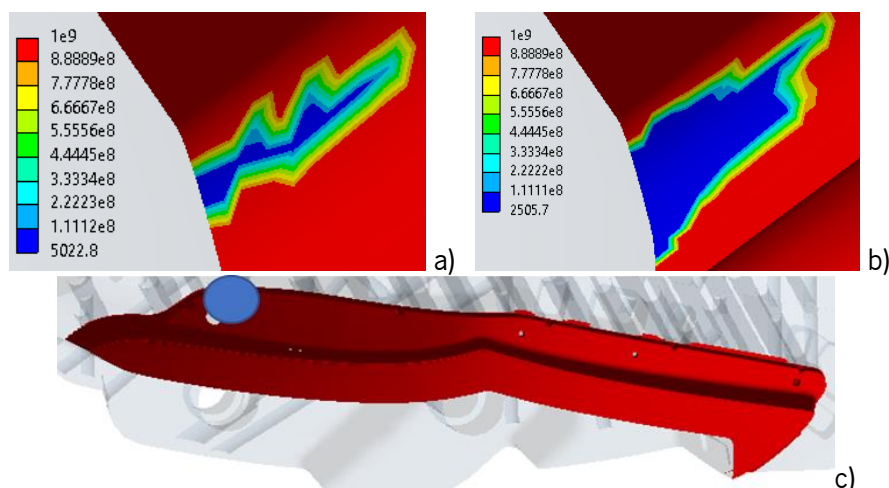


Figure 203 – Number of cycles before crack nucleation determined by a) the octahedral shear stress and by b) the Sines method and c) is the region under analysis on the core engraving region 2 of Fat_1.

Here it is verified that in both zones under analysis there is a significant region along the radius that is quite stressed. When joining this result with the von Mises stress analysis these two regions are points of interest and should be treated as potential locations where crack nucleation can occur because here one is not dealing with high stresses resulting from singularities in the numerical model.

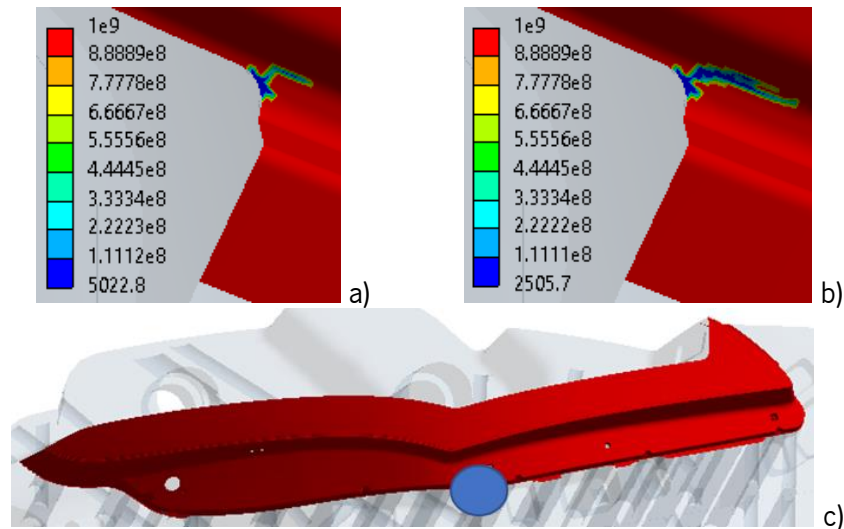


Figure 204 – Number of cycles before crack nucleation determined by a) the octahedral shear stress and by b) the Sines method and c) is the region under analysis on the core engraving region 2 of Fat_1.

By analyzing the fatigue safety coefficient, it can be seen that the area shown in Figure 205 is more susceptible to damage due to alternating loads than the region shown in Figure 206. For in the zone represented in Figure 206 the damage is very concentrated at the boundary between components, which may be the result of contact between them although it propagates a little along the radius when the result determined by the Sines model is analyzed. In the zone represented in Figure 205 it is more noticeable regardless of the calculation model, that the damage expands along the radius and according to the Sines model the critical region has a larger area than by the octahedral shear stress model.

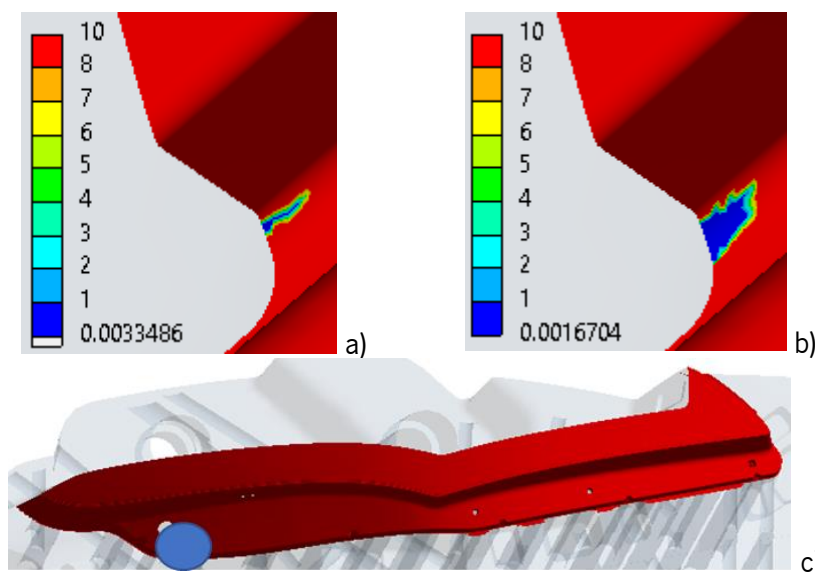


Figure 205 – Safety factor determined by a) the octahedral shear stress and by b) the Sines method and c) is the region under analysis on the core engraving region 2 of Fat_1.

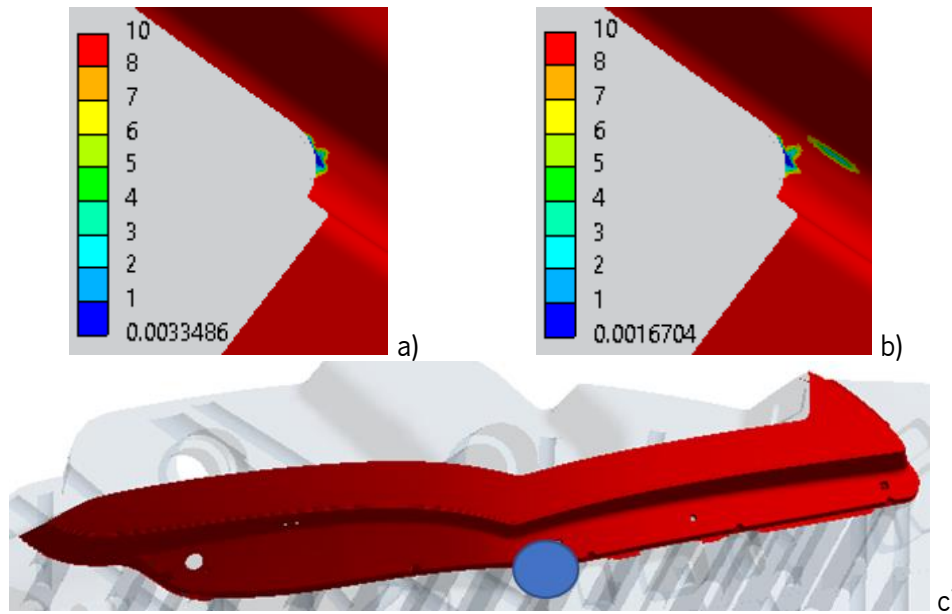


Figure 206 – Safety factor determined by a) the octahedral shear stress and by b) the Sines method and c) is the region under analysis on the core engraving region 2 of Fat_1.

Finally, the last molding region under analysis from the FAT_1 injection mold. Here by the analysis of Figure 208, it can be seen that the overall maximum alternating stresses are lower than the fatigue endurance limit. Even in the region where the pressures are higher, the stresses do not exceed 334 MPa. Therefore, this region helps to demonstrate that the loads exerted on the mold, although they are the factor that leads to the occurrence of cracks, they have to be combined with other factors such as the geometry of the region for there to be the risk of cracking in the molding region.

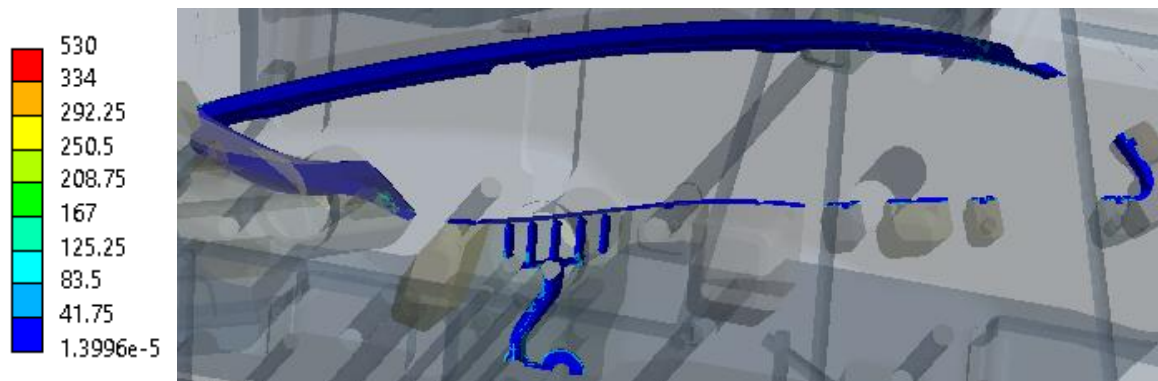


Figure 207 – Maximum alternated stress in MPa at each location by the Sines method on the core engraving region 3 of Fat_1.

When performing the damage analysis, which can be observed in Figure 208 and Figure 209, it can be perceived that only one region presents values above 0, which is duly evidenced in the image. This occurs at an interface between two mold components and being too centralized on one edge of the component it can be discarded. For even when analyzing the von Mises stresses in section 6.1.2.1, it turns out that this result is caused by yet another singularity in the numerical model.

Here, it turns out that both the octahedral shear stress method and the Sines method identified the same region. The Sines method had lower values than the octahedral shear stress method. This is even

more visible in Figure 210 where the number of cycles predicted until the crack nucleation is presented. Where the Sines model presents evidently a higher number of cycles until the crack nucleation.

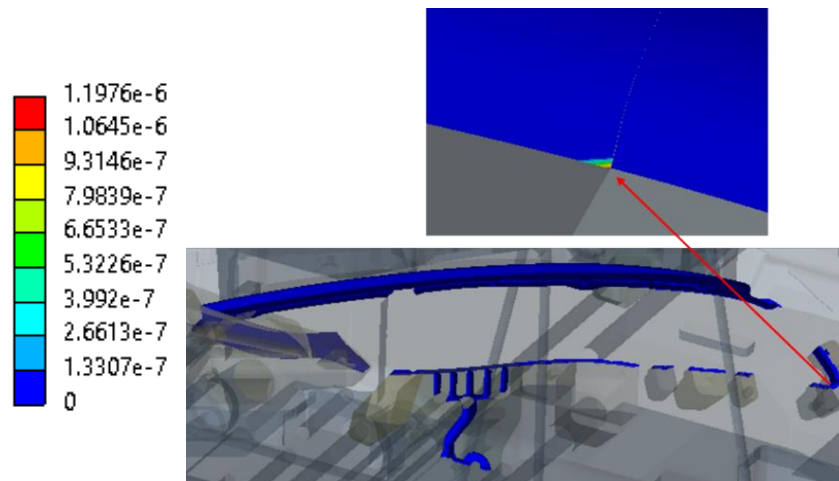


Figure 208 – Damage determined by the octahedral shear stress method on the core engraving region 3 of Fat_1.

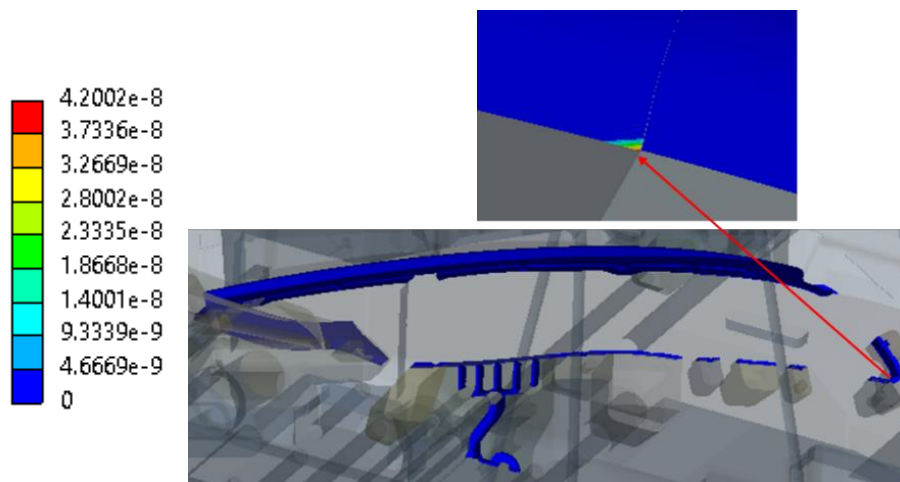


Figure 209 – Damage determined by the Sines method on the core engraving region 3 of Fat_1.

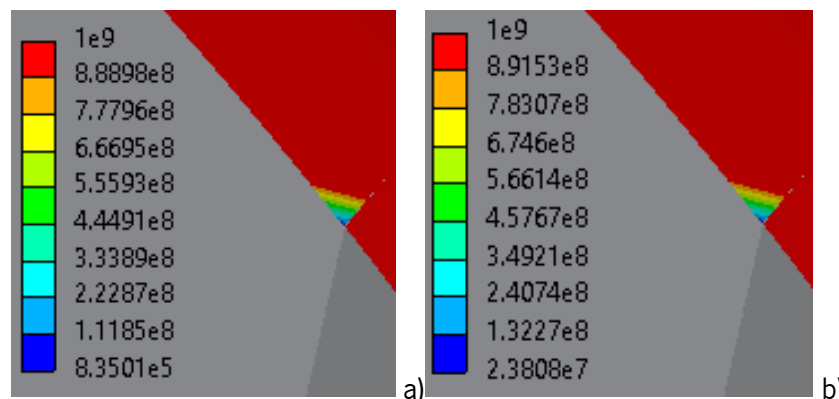


Figure 210 – Number of cycles before crack nucleation determined by a) the octahedral shear stress and by b) the Sines method on the core engraving region 3 of Fat_1.

This variation between the expected number of cycles until crack nucleation has repercussions on the fatigue safety coefficient, because as can be seen in Figure 211 according to the Sines method there is no possibility of failure, while for the octahedral shear stress method there is the possibility of crack nucleation before 1,5 million cycles.

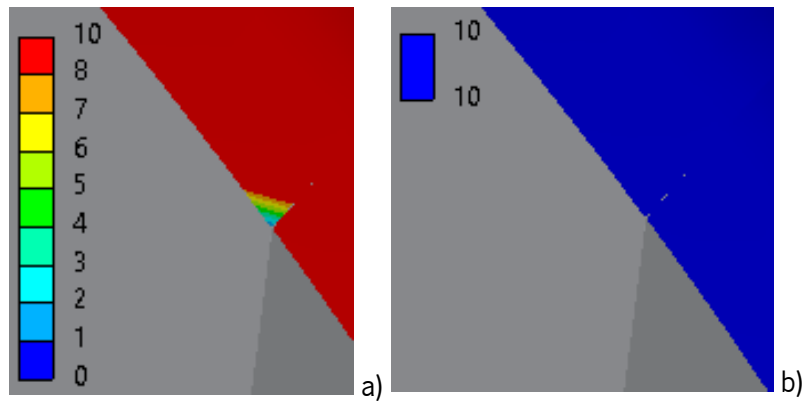


Figure 211 – Safety factor determined by a) the octahedral shear stress and by b) the Sines method on the core engraving region 3 of Fat_1.

Finally, the analysis of the fitting region in the partition plane on the core side. Here as shares the same reduction coefficients as the cavity side adjustment region and therefore the same value for the endurance fatigue limit, i.e., 239 MPa.

By analyzing Figure 212, where the maximum alternating stresses are presented, it can be seen that a region presents values above 239 MPa, a region that will be analyzed next in order to verify if it presents problems for the mold in question.

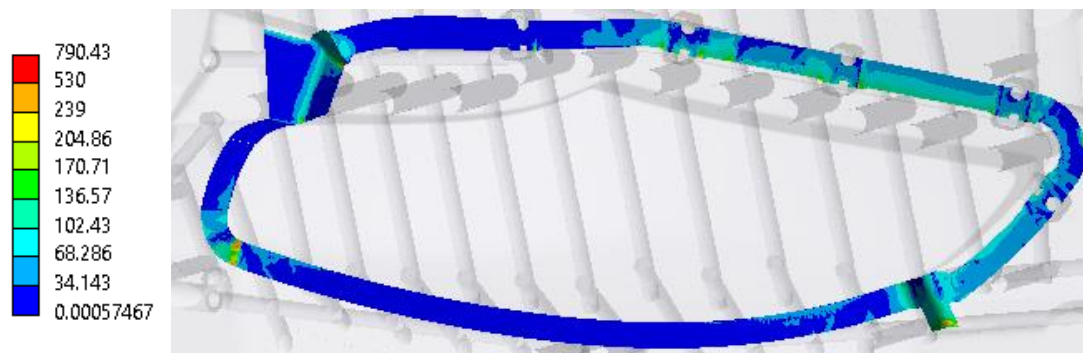


Figure 212 – Maximum alternated stress in MPa at each location by the Sines method at the fitting region on the core side of Fat_1.

For the adjustment region there is only one point that effectively presents a significant damage value to the point of being studied. Figure 213 and Figure 214 show the results obtained by the octahedral shear stress method and the Sines method.

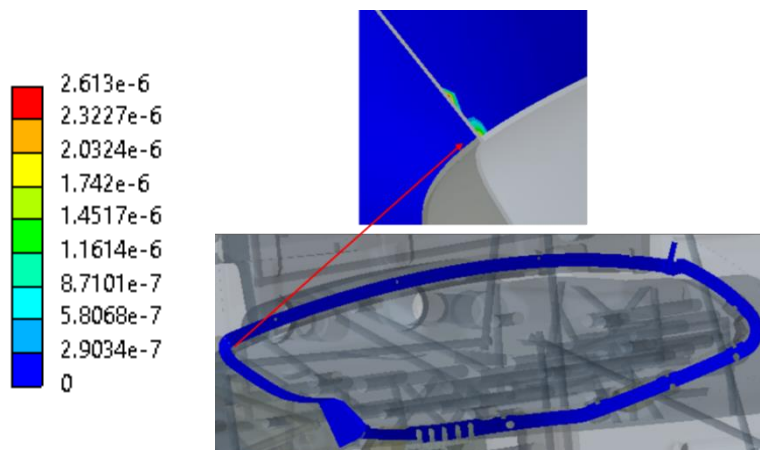


Figure 213 – Damage determined by the octahedral shear stress method at the fitting region on the core side of Fat_1.

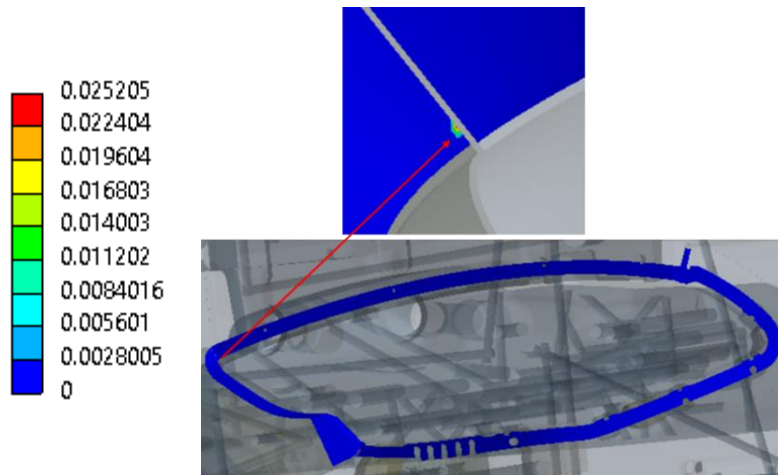


Figure 214 – Damage determined by the Sines method at the fitting region on the core side of Fat_1.

Here the data detected by the Sines method is significantly higher than that determined by the octahedral shear stress method. Although it seems to be in a more confined region, when analyzing the number of cycles until the crack nucleation, it is verified that in reality the affected region is higher as it is perceptible in Figure 215.

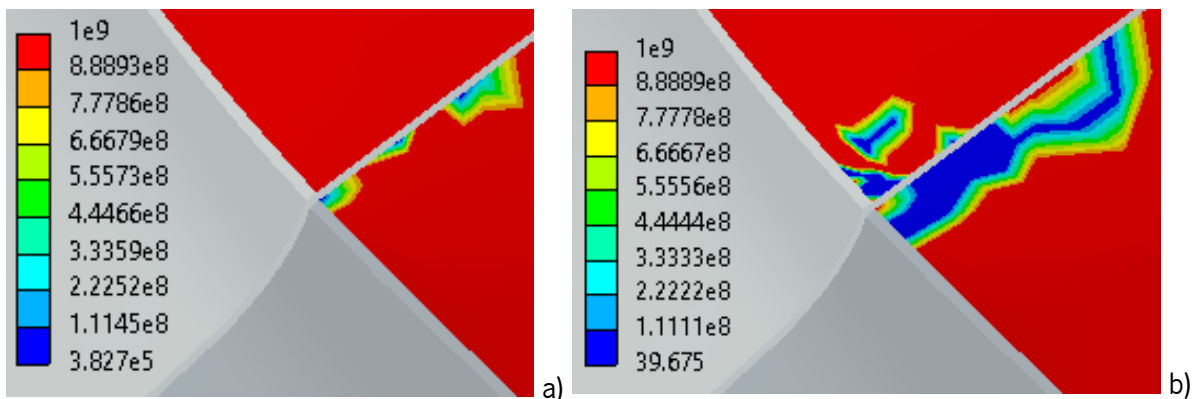


Figure 215 – Number of cycles before crack nucleation determined by a) the octahedral shear stress and by b) the Sines method at the fitting region on the core side of Fat_1.

The results are shown in Figure 216, where the fatigue safety coefficient is demonstrated. Here, as expected, it brings what had already been observed in Figure 210. And taking into account all the results obtained, this region should also be taken into account as a propitious region for crack nucleation.

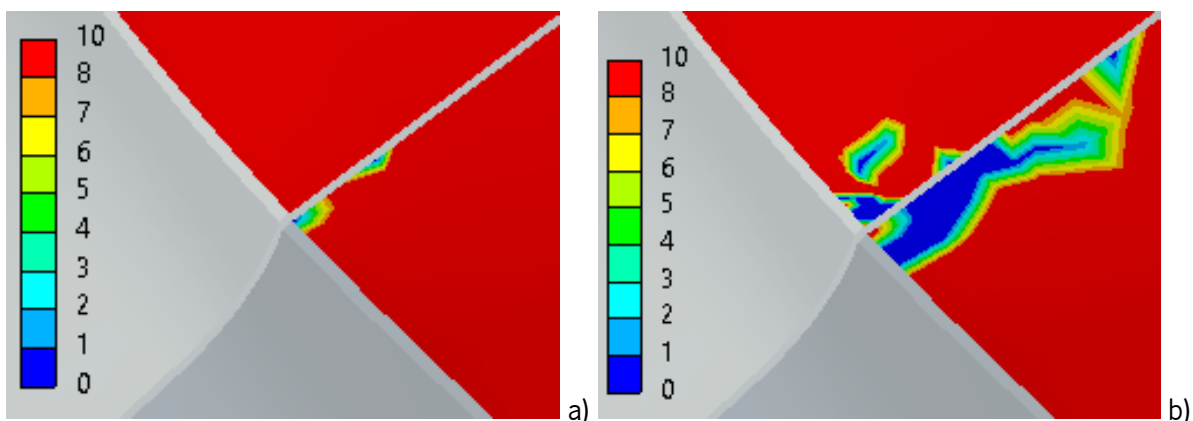


Figure 216 – Safety factor determined by a) the octahedral shear stress and by b) the Sines method at the fitting region on the core side of Fat_1.

Therefore, as already mentioned, in this work are being used molds manufactured by Group Simoldes. And according to their records this mold presented a crack in a region that both models used detected as a very probable area for crack nucleation. Although there is no photographic record or the number of cycles performed by the mold until the crack became visible, the crack location was duly registered. The place was located in the region 2 of the core side, in the same zone that is represented in Figure 205.

Although there is a region in the fitting region in the partition plane on the core side that shows significant damage caused by fatigue, it ends up being able to differ in the injection mold in relation to the CAD model. Because the adjustment process is a manual work and during this process the fitting region may have been relieved enough so that the effort was not so high and thus the crack ended up not nucleate. Since according to the calculations performed, it would have a significantly shorter lifetime than the region where the crack was detected.

Therefore, for this mold the models used would be able to predict the region where the crack nucleate but also presented other locations, so it can be concluded that for this injection mold in terms of possible locations for crack nucleation presented conservative results but without any conclusive results in terms of number of cycles to crack nucleation.

6.1.3.2 Results for the tool Fat_2

As already mentioned from this section only the results calculated from the Sines method will be presented. Where the objective is the same, that is to determine the points where fatigue failure can occur and then compare with the mold in order to verify if it would be possible to predict the failure using this method.

For the study of this mold, it will be started by the engraving region on the cavity side. Where the maximum alternating stresses are represented in Figure 217. This mold was manufactured with the same type of steel as the Fat_1 mold and with the same types of surface finishing, therefore the fatigue endurance limit for this region will be in the order of 398 MPa.

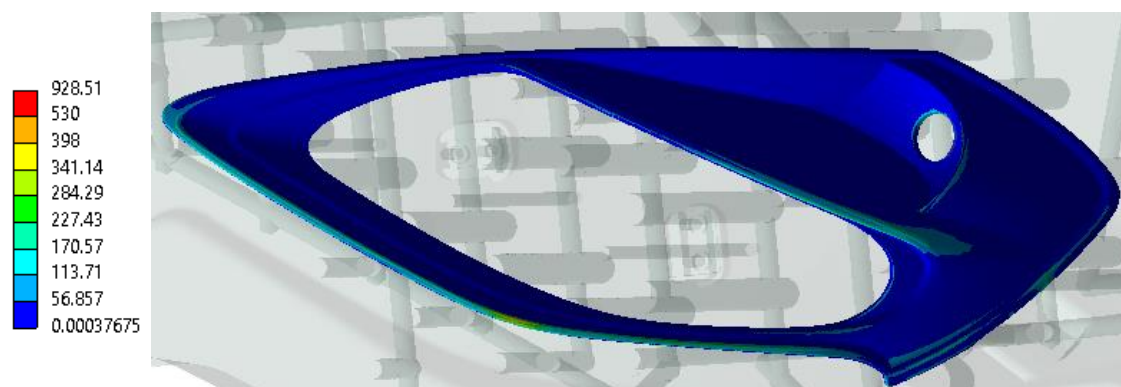


Figure 217 – Maximum alternated stress in MPa at each location by the Sines method at the cavity engraving region of Fat_2.

By analyzing the results of the maximum alternating stress, it is possible to verify that there is a zone where the alternating stresses exceed the fatigue endurance limit, so it will be necessary to understand if this zone is problematic considering the lifetime of the injection mold.

When performing the damage analysis, which can be done from Figure 218, it can be observed that the region where the alternating stresses are highest has some points where the damage is significant. That when compared with the result regarding the number of cycles supported until the crack nucleation represented in Figure 219. Where it is possible to observe that at the root of the radius as well as in the region closest to the boundary with the fitting region in the partition plane can be cause for concern in terms of fatigue resistance.

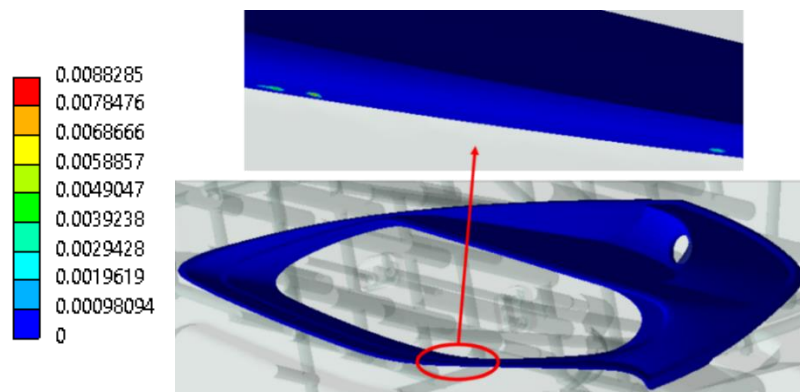


Figure 218 – Damage determined by the Sines method at the cavity engraving region of Fat_2.

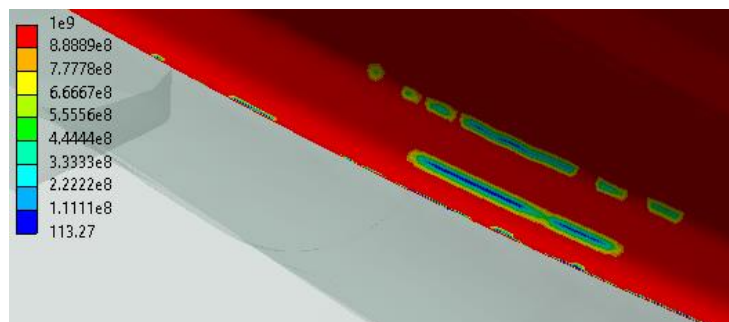


Figure 219 – Number of cycles before crack nucleation determined the Sines method at the cavity engraving region of Fat_2.

Now, moving on to the safety coefficient that is shown in Figure 220, it can be seen that the most solicited region is the one that is located in the engraving region closest to the fitting region in the partition plane.

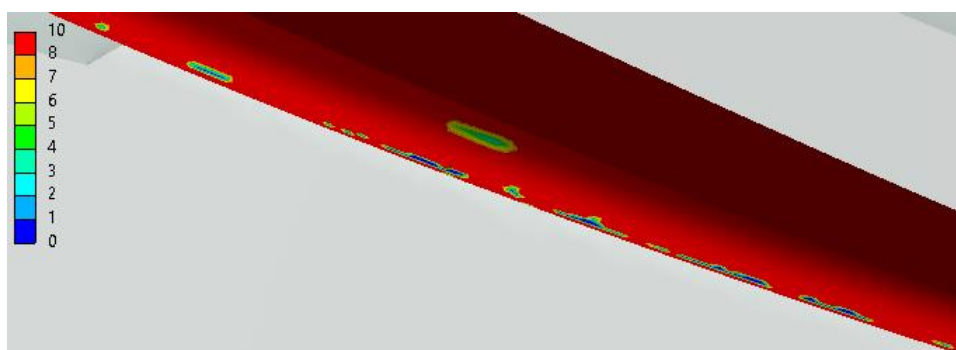


Figure 220 – Safety factor determined by the Sines method at the cavity engraving region of Fat_2.

For the fitting region in the partition plane on the cavity side, which has a calculated fatigue endurance limit in the order of 239 MPa. In Figure 221 where the maximum alternating stresses are presented it can be seen that there are some regions with highly stressed, especially those closest to the injection point. And it should also be noted that these regions which have higher alternating stresses or are near the boundary between the adjustment region in the partition plane and the engraving region. Or next to notches where stress concentrations are more likely to occur.

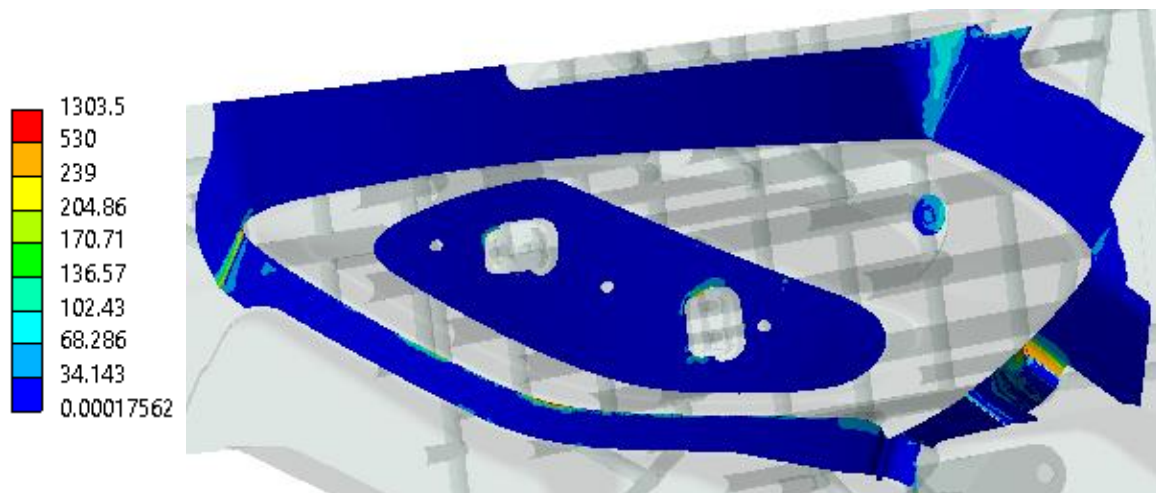


Figure 221 – Maximum alternated stress in MPa at each location by the Sines method at the fitting region on the cavity side of Fat_2.

When analyzing the results of damage per cycle and the number of cycles until crack nucleation, which are represented in Figure 222 and Figure 223 respectively, it is noticeable that there are three regions where the appearance of cracks is more likely.

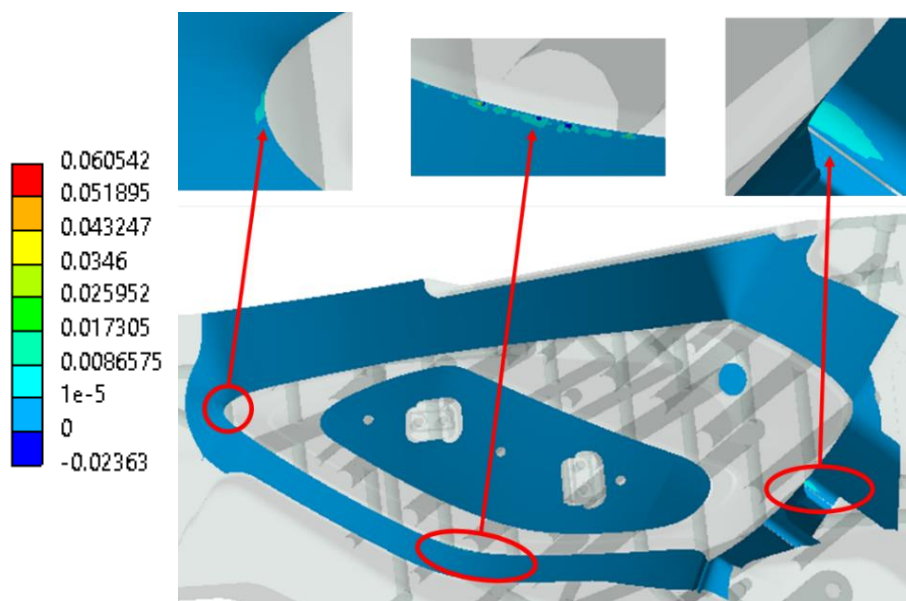


Figure 222 – Damage determined by the Sines method at the fitting region on the cavity side of Fat_2.

It is necessary to take into account that there are some points where the von Mises stresses, already analyzed in section 6.1.2.2, reach values higher than the failure stress. This is a result of intersections caused by the way the discretization of the model is done and that leads to the fact that the analysis of

the results must be made with great criterion and caution. Here it is noticeable that both for the damage and the result of the number of cycles until crack nucleation showed results that are not possible. This is the result of having stresses too high, and obviously if they existed in the injection mold, it would be facing the appearance of cracks due to the reach of the material ultimate tensile stress and not by fatigue.

As for the critical points, here they become more evident than in the result of the maximum alternating stresses, however as mentioned before, the occurrence of damage does not mean that it is enough for crack nucleation in the mold's lifetime.

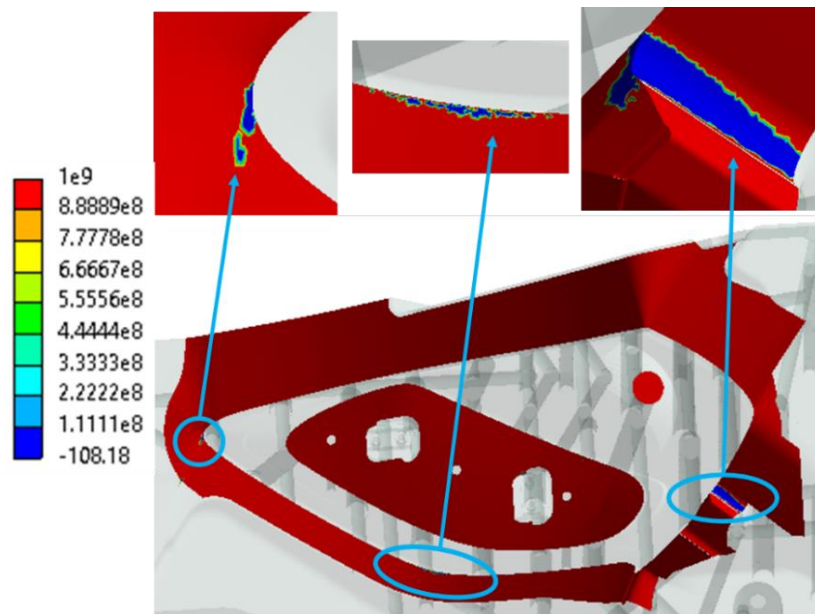


Figure 223 – Number of cycles before crack nucleation determined the Sines method at the fitting region on the cavity side of Fat_2.

But from the safety coefficient analysis, where the number of cycles that the injection mold must endure is taken into account, it is possible to see that these three points are susceptible to the appearance of cracks. In Figure 224 where the results of the safety coefficient are presented for the adjustment region in the partition plane on the cavity side, it is verified that the three locations already mentioned are very propitious for crack nucleation.

Now, looking at the core side of the injection mold Fat_2, can also be seen that there are highly stressed regions. As presented in Figure 225 there are regions that well exceed the reduced endurance fatigue limit. Here, as on the cavity side, the most critical alternating stress values are found near the injection point. Although there are other more distant regions with alternating stress values close to the reduced fatigue endurance limit, but lower therefore not giving rise to any further analysis.

By performing the damage and number of cycles to crack nucleation analysis, that the results can be analyzed in Figure 226 and Figure 227 respectively. It is possible to verify that there are five regions that are very relevant to analyze in order to realize if they are able to resist the mold's lifetime without the occurrence of crack nucleation.

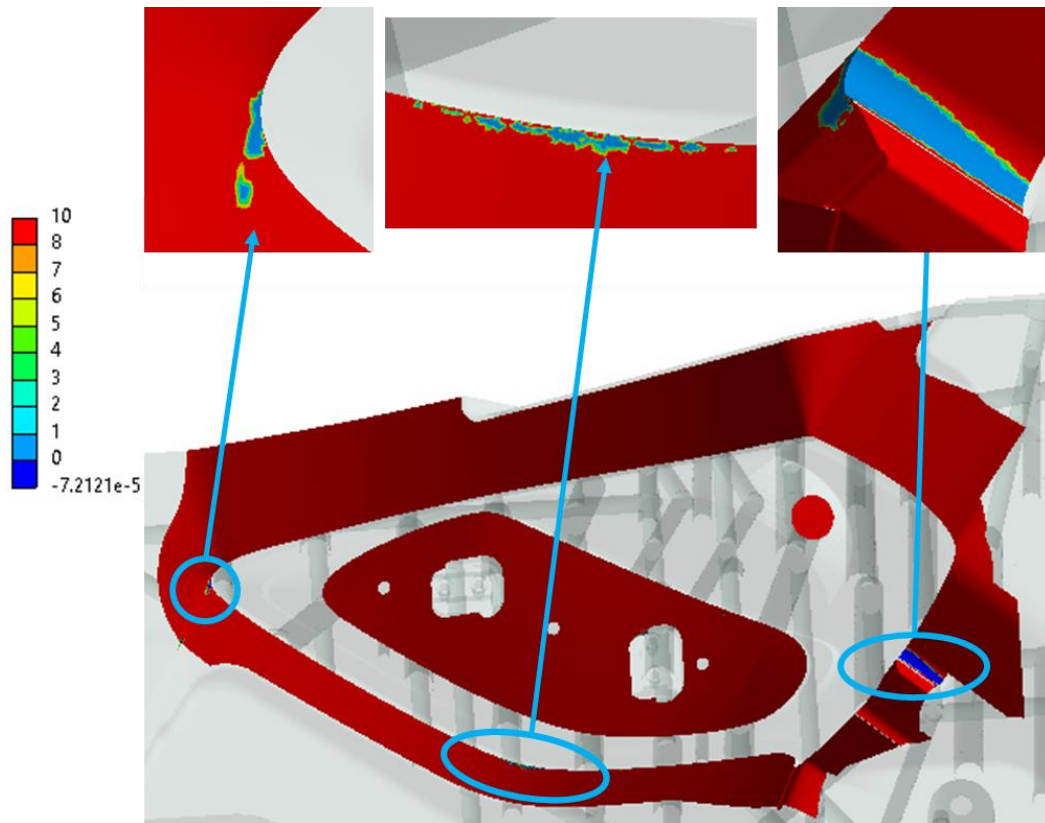


Figure 224 – Safety factor determined by the Sines method at the fitting region on the cavity side of Fat_2.

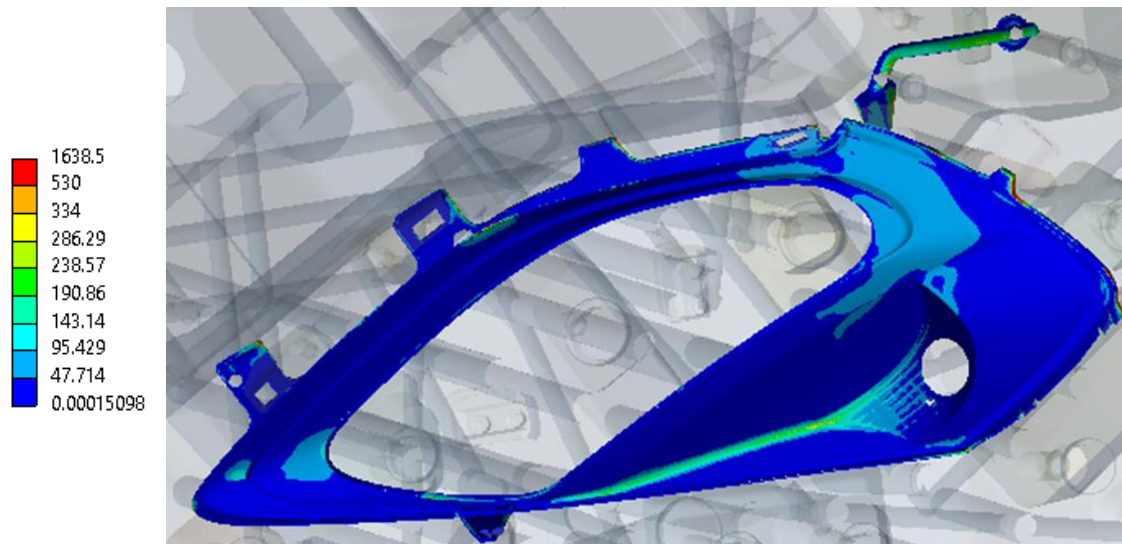


Figure 225 – Maximum alternated stress in MPa at each location by the Sines method at the core engraving region of Fat_2.

From a more careful analysis and comparing with the results already presented in section 6.1.2.2, it is verified that the regions which present higher damage to be considered for fatigue analysis besides being in the regions that support the highest loads also present sharp notches.

When analyzing the result of the fatigue safety coefficient, in Figure 228, it is verified that all regions under study are propitious to crack nucleation. Something that was already perceptible in Figure 227, where the number of cycles predicted until crack nucleation is represented. Because it already showed large regions with values below 1.07 million cycles and this mold has a lifetime of 1.5 million cycles.

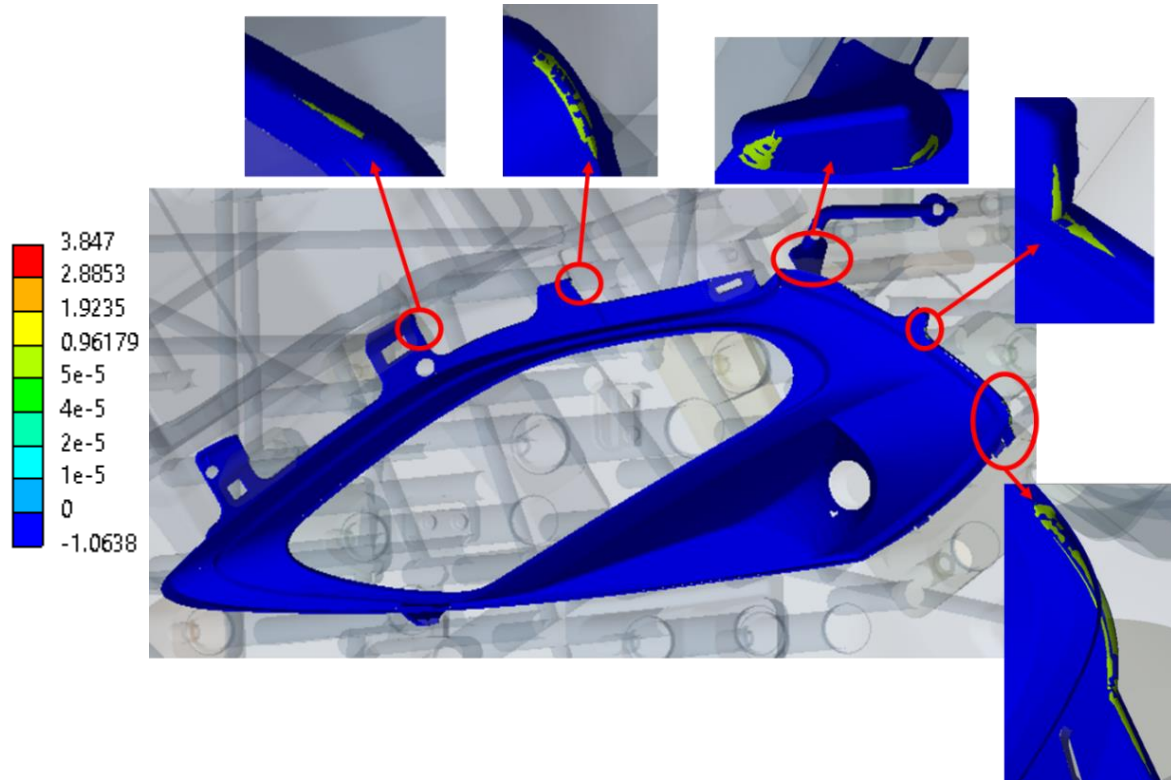


Figure 226 – Damage determined by the Sines method at the core engraving region of Fat_2.

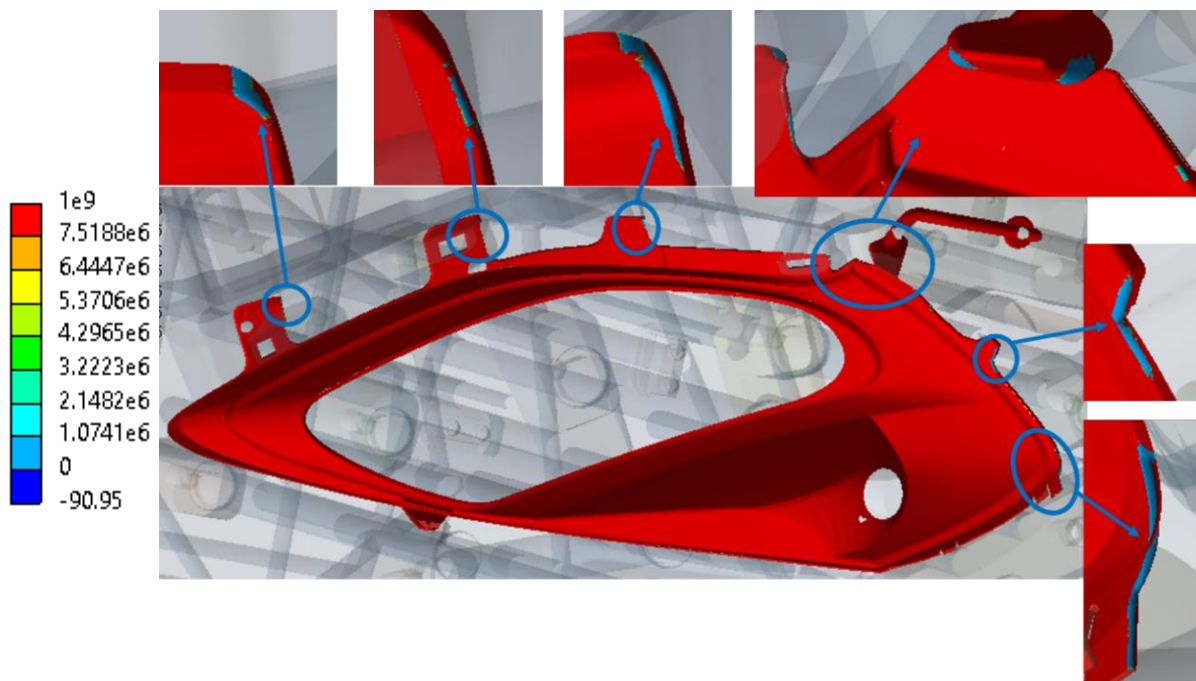


Figure 227 – Number of cycles before crack nucleation determined the Sines method at the core engraving region of Fat_2.

Therefore, the engraving region on the core side of this injection mold would be one of the critical locations. However, the nucleation of the crack can occur without it being perceptible, so, in the regions where the pressures are higher it is likely that small defects can start to appear in the part. Because the polymer that is being injected, being an amorphous material that when is the melt state has a good capacity to penetrate in very small lodges when combined with high pressures.

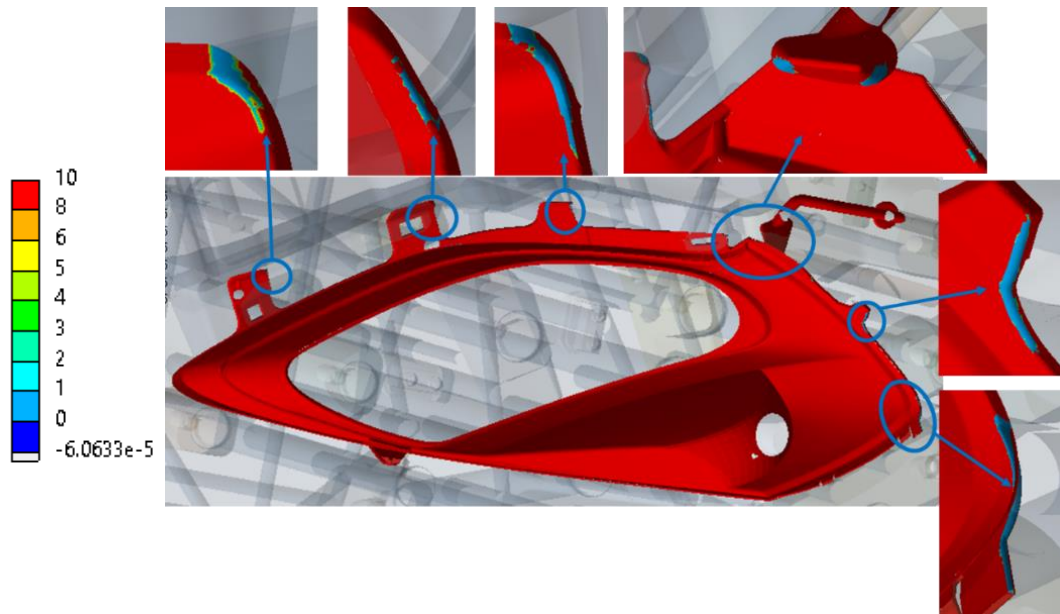


Figure 228 – Safety factor determined by the Sines method at the core engraving region of Fat_2.

The fitting region on the core side is the last one left to analyze for this injection mold. Figure 229 shows the maximum alternating stresses in the fitting region, it is at first glance is noticeable that some of the regions where the stresses exceed the reduced fatigue endurance limit are adjacent to the same regions that were identified as critical regions in the core engraving region.

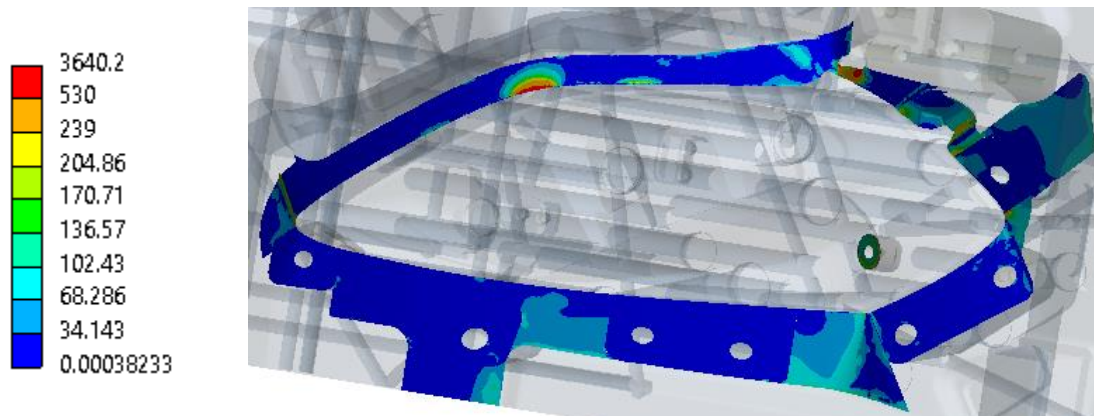


Figure 229 – Maximum alternated stress in MPa at each location by the Sines method at the fitting region on the core side of Fat_2.

By analyzing Figure 230 and Figure 231 which show the results obtained for damage and number of cycles to crack nucleation respectively. It is possible to verify that there are five regions that are significantly stressed to the point of being considered critical for the resistance to the loading cycles that the injection mold will have to support during its lifetime. Considering that the most stressed areas are near the edge separating the fitting region from the engraving region and combining with the results already obtained for the engraving region, these areas will be extremely critical and very susceptible to the appearance of cracks.

In Figure 232 where the fatigue safety coefficient is presented, it is clearly seen that all five regions are potential points for crack initiation. By combining the results obtained here with those presented in

section 6.1.2.2, it can be seen that the notches present in the etch region are largely responsible for the reduction in fatigue resistance of the fitting zone.

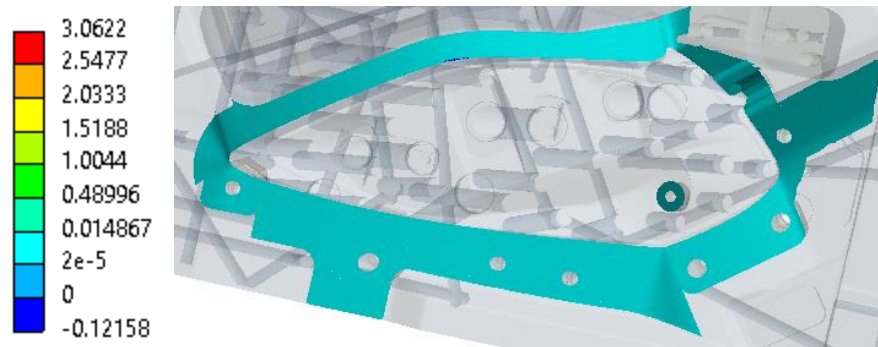


Figure 230 – Damage determined by the Sines method at the fitting region on the core side of Fat_2.

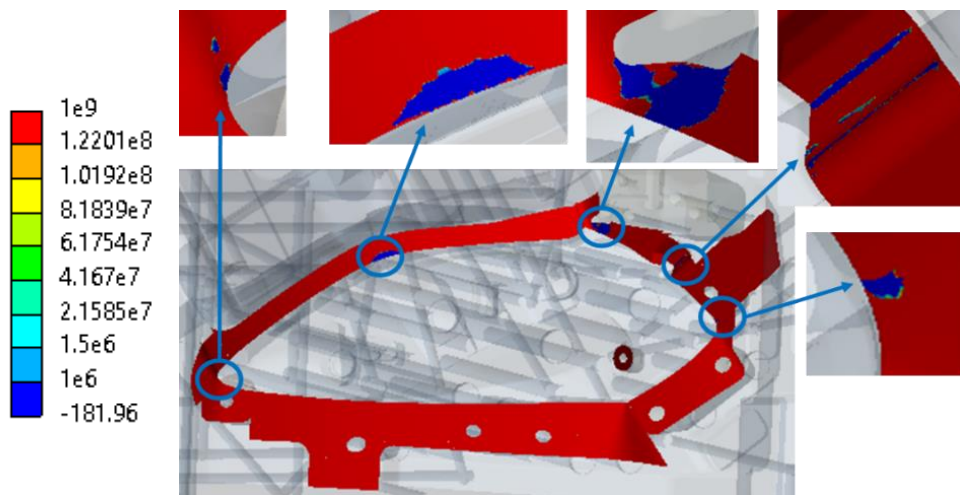


Figure 231 – Number of cycles before crack nucleation determined the Sines method at the fitting region on the core side of Fat_2.

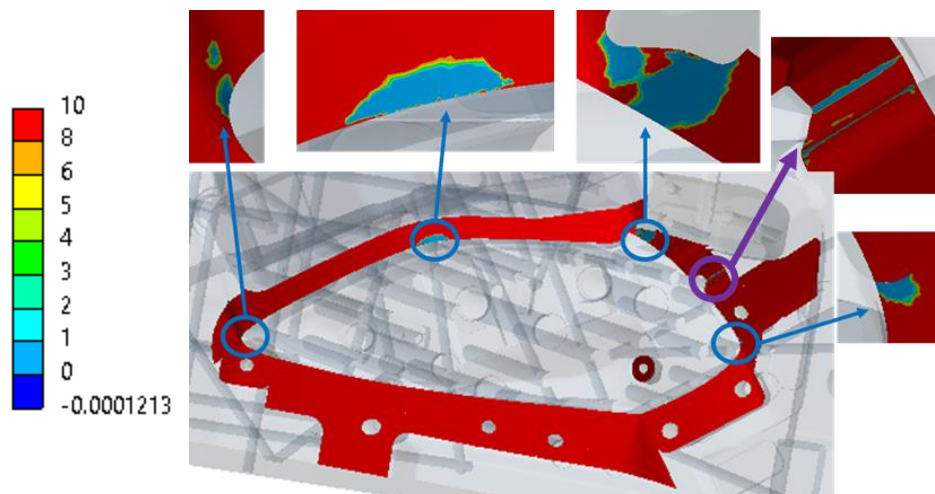


Figure 232 – Safety factor determined by the Sines method at the fitting region on the core side of Fat_2.

Although in the engraving region of the cavity side there is a region that is susceptible to fatigue crack initiation, on the core side of the injection mold, it is found that the most affected regions start in the molding region and continue into the fitting region in the partition plane which tend to be more susceptible to the crack initiation due to the way the loading is applied and its geometry. Therefore, these regions by the analysis of the results are the most probable for crack nucleation.

When comparing with the records of the injection mold, it is verified that a region highlighted in Figure 232 in purple was the one that presented cracks caused by fatigue. In this mold there is also no record of the number of cycles until the crack was detected. So, for this tool the model also presented conservative results considering that besides the region that effectively presented a crack other four regions were also detected as regions with a high probability for the appearance of cracks caused by fatigue.

6.1.3.3 Results for the tool Fat_3

The injection mold has an injection concept identical to the molds Fat_1 and Fat_2, but as it was possible to observe in the two injection molds previously studied, although the injection concept is identical, there were differences in the regions where fatigue failures would be more likely to occur. Therefore, in the analysis of this mold it will be possible to verify that here too the crack analysis must be performed with great care and criteria.

It should be noted that the reduction coefficients are the same that were determined for the Fat_1 mold. As the same types of steel were used the reduced fatigue endurance coefficients will be the same as for the Fat_1 mold both for the engraving regions and the fitting regions in the partition plane.

Starting with the analysis of the maximum alternating stresses in the engraving region on the cavity side, which can be seen in Figure 233. By observing the result, it can be observed that there are two more stressed areas during the injection cycle, one of them is in a notch on the engraving region while the other is near the fitting region. It will be possible to observe the reason for its appearance when analyzing the partition plane adjustment region on the cavity side.

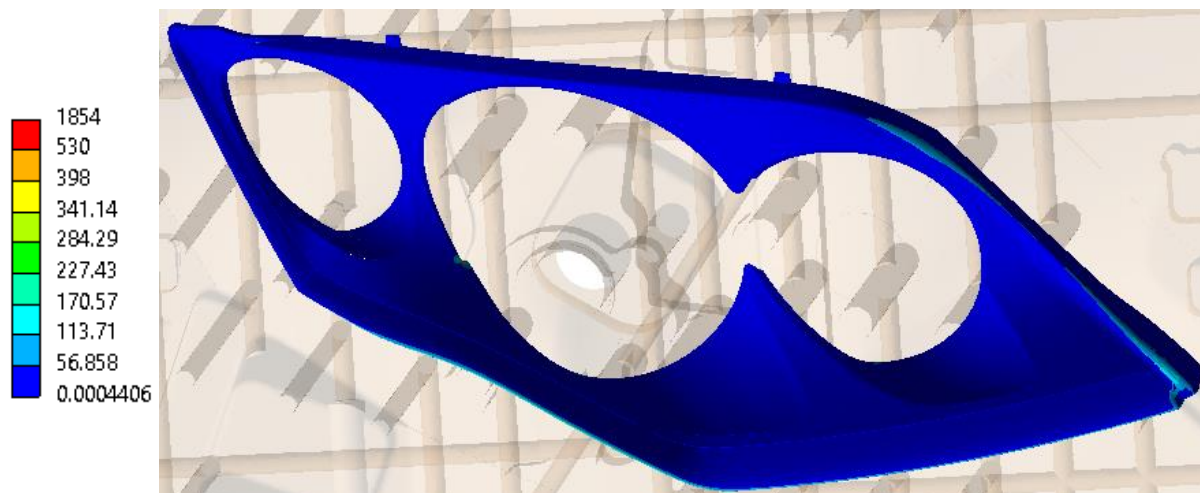


Figure 233 – Maximum alternated stress in MPa at each location by the Sines method at the cavity engraving region of Fat_3.

When one moves to the analysis of the damage and the number of cycles until crack nucleation, presented in Figure 234 and Figure 235 it is found that the region most likely to fail by fatigue is the one being stressed by the loads present in the fitting region in the partition plane.

And consequently, as it would already be perceptible either in the analysis of the maximum alternating stresses or when presenting the results of the von Mises stresses in section 6.1.2.3, the stresses are higher than the ultimate tensile stress. So, taking into account how the code built for this application leads to the appearance of impossible values such as safety coefficients or number of cycles until the crack nucleation with negative values. Those must be analyzed individually to check the cause for this values.

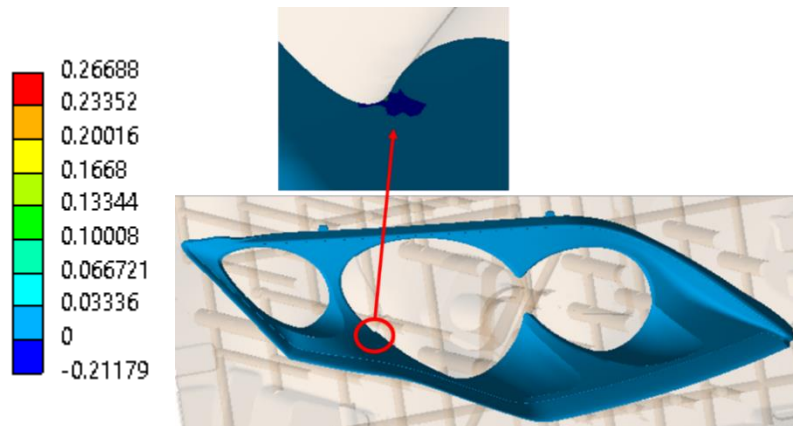


Figure 234 – Damage determined by the Sines method at the cavity engraving region of Fat_3.

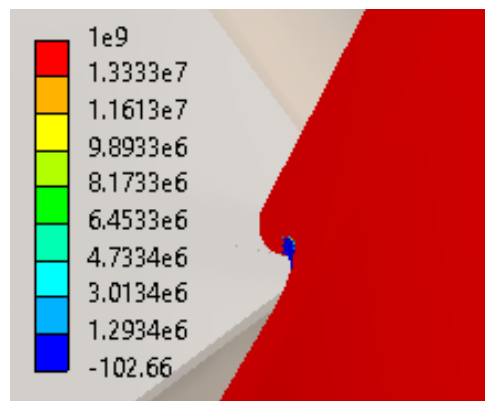


Figure 235 – Number of cycles before crack nucleation determined the Sines method at the cavity engraving region of Fat_3.

As noted, the only region with a coefficient of safety less than 1 is located in a region adjacent to the fitting region in the mold partition plane on the cavity side. The values for the coefficient of safety can be observed in Figure 236, but a more detailed analysis of the region will have to be done in conjunction with the analysis of the fitting region.

For the analysis of the tuning region in the partition plane on the cavity side, as done in all analyses up to this point, one will start with the distribution of the maximum alternating stresses, which is presented in Figure 237.

By the analysis of the maximum alternating stresses, it is verified that effectively the fitting region in the partition plane adjacent to the region with lower fatigue endurance of the engraving region is the only one that presents values above the reduced fatigue endurance limit. Therefore, this region will be subject to further analysis in order to understand to what level the resistance is reduced and why.

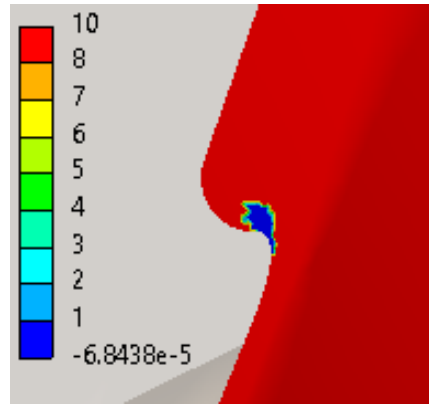


Figure 236 – Safety factor determined by the Sines method at the cavity engraving region of Fat_3.



Figure 237 – Maximum alternated stress in MPa at each location by the Sines method at the fitting region on the cavity side of Fat_3.

By comparing the results obtained for the maximum alternating stress with the results for the damage and the number of cycles to crack nucleation that are presented in Figure 238 and Figure 239 respectively, it becomes clear that the most stressed region is the one detected earlier. It should be noted that this most stressed region starts at the root of a notch and taking into account the results obtained for the von Mises stress distribution presented in Figure 175 it is verified that right at the instant when the clamping force is applied, that this region is highly stressed. The shape of the notch and the shape of the surfaces around it where the forces are applied lead to the notch root being heavily stressed when the clamping force is applied, although with the injection pressure these stresses are relieved, the load employed by the clamping force leads to the notch root being severely stressed.

When analyzing the fatigue safety coefficient shown in Figure 240, it can be noticed that there is a significant region with a coefficient less than 1. Therefore, crack nucleation is very likely in this region. But it should be kept in mind that this region is also problematic in terms of discretization of the surfaces due to its complexity. This can lead to the results obtained for the stresses being exaggerated due to this error. Furthermore, the fitting regions are areas where operators during the mold tuning process when assembling the mold, make some manual adjustments to the surfaces by polishing them. This can lead

to differences between the CAD model and the physical model that can consequently lead to divergences between the numerical results and those obtained from the physical models.

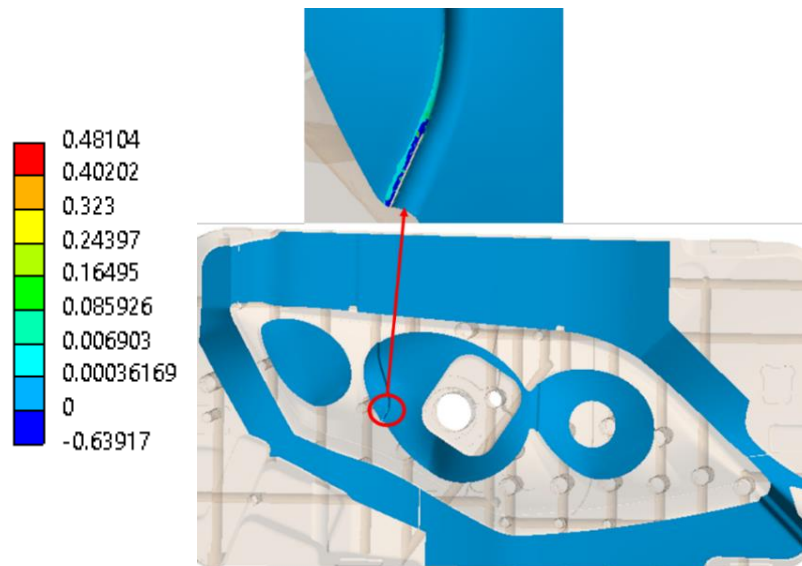


Figure 238 – Damage determined by the Sines method at the fitting region on the cavity side of Fat_3.

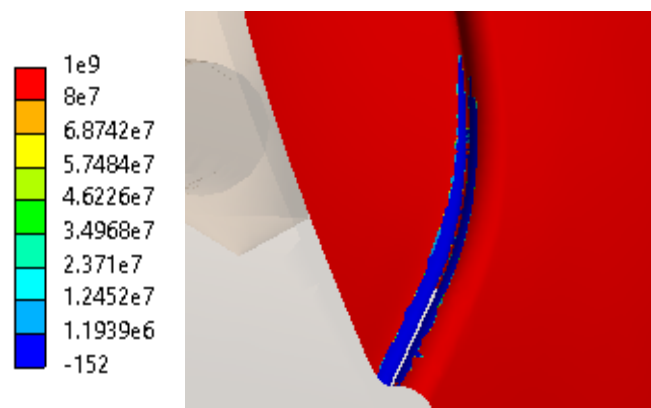


Figure 239 – Number of cycles before crack nucleation determined by the Sines method at the fitting region on the cavity side of Fat_3.



Figure 240 – Safety factor determined by the Sines method at the fitting region on the cavity side of Fat_3.

Now, for the engraving area on the core side of this injection mold, when analyzing the maximum alternating stresses presented in Figure 241, it is possible to verify that although there are stress values worth mentioning in several areas of the region, only in one place the stresses exceed the reduced fatigue

endurance limit. As will be demonstrated when presenting the remaining results for the engraving region on the side of the mold core.



Figure 241 – Maximum alternated stress in MPa at each location by the Sines method at the core engraving region of Fat_3.

The mentioned region can be more easily located from the results obtained for the damage or those obtained for the number of cycles to crack nucleation, which are presented in Figure 242 and Figure 243 respectively. In Figure 242 it is pointed out the region where the alternating loads are enough to lead the alternating stresses to exceed the reduced fatigue endurance limit, but it is more noticeable the fatigue resistance reduction when looking at the result for the number of cycles until crack nucleation. Here it is noticeable that the area concerned is located at the root of a notch that is being loaded both during the application of the clamping force and then during the filling and packing phase during the injection cycle.

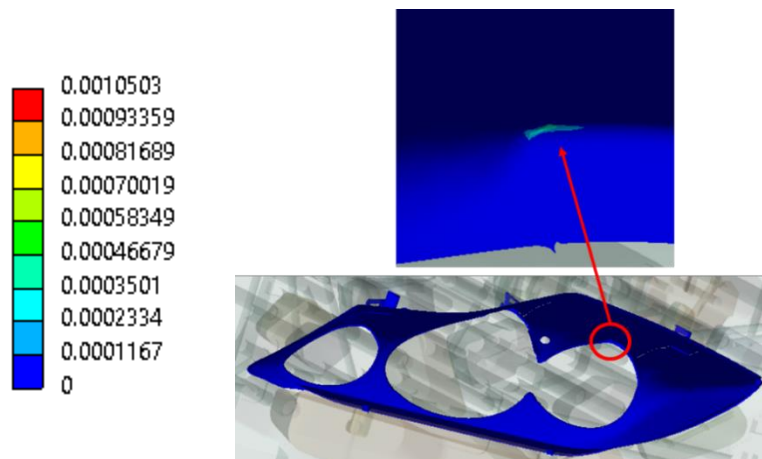


Figure 242 – Damage determined by the Sines method at the core engraving region of Fat_3.

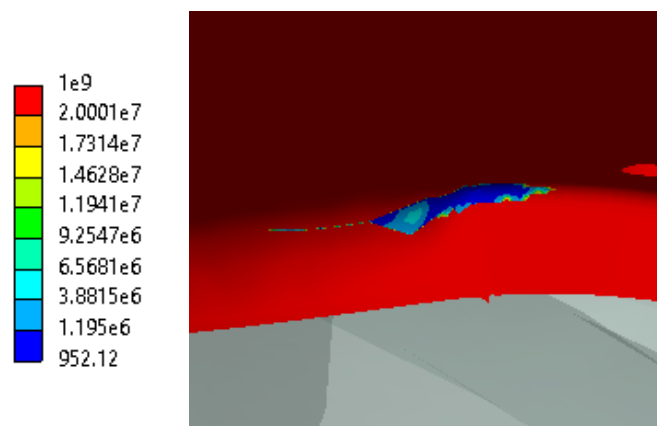


Figure 243 – Number of cycles before crack nucleation determined the Sines method at the core engraving region of Fat_3.

In order to better understand which region effectively presents a high risk of fatigue failure, the analysis of Figure 244 helps to delimit this same region. And as it is perceivable, the region where the fatigue safety factor is lower than 1, besides being concentrated at the root of a notch, this notch does not follow a straight line, but follows an accentuated curvature that ends up making this region even more susceptible to fatigue crack nucleation.

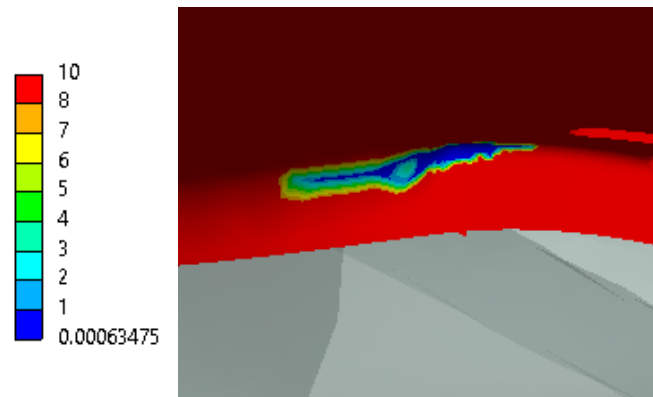


Figure 244 – Safety factor determined by the Sines method at the core engraving region of Fat_3.

Finally, for the Fat_3 mold, the partition plane fitting region on the core side remains to be analyzed. From Figure 245 where the result of the maximum alternated stresses is presented, it can be noticed that there are two regions that stand out. However, one of them is more relevant, besides being the one with the highest values, also because it is the one that will contact the most stressed region of the partition plane fitting region on the cavity side.

As with the adjacent region on the cavity side, this region is more stressed when applying the clamping force due to its geometry. However, when analyzing the remaining results, it will be possible to understand that the values obtained do not demonstrate a damage as high as that obtained for the adjacent region on the cavity side.

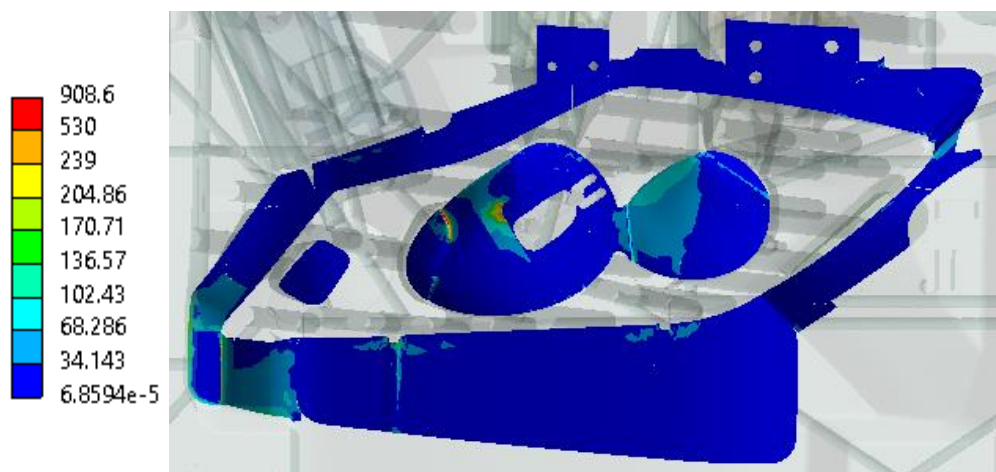


Figure 245 – Maximum alternated stress in MPa at each location by the Sines method at the fitting region on the core side of Fat_3.

By the analysis of Figure 246 and Figure 247 that present the results of damage and number of cycles until crack nucleation respectively, it is verified that only one region presents values that can lead

to the occurrence of a crack. Exactly the region adjacent to the most requested region of the cavity side adjustment area.

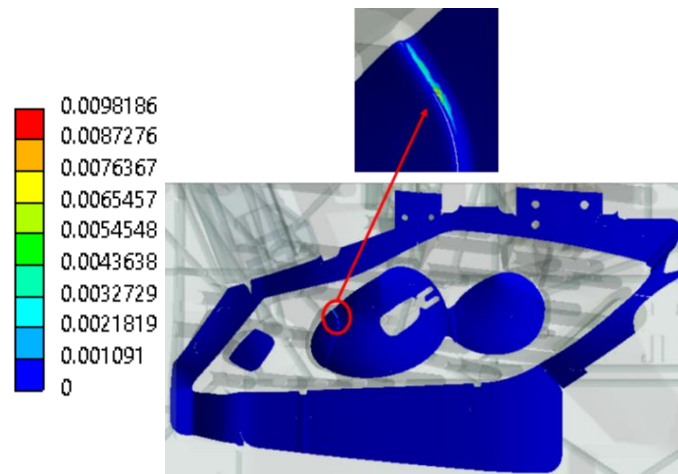


Figure 246 – Damage determined by the Sines method at the fitting region on the core side of Fat_3.

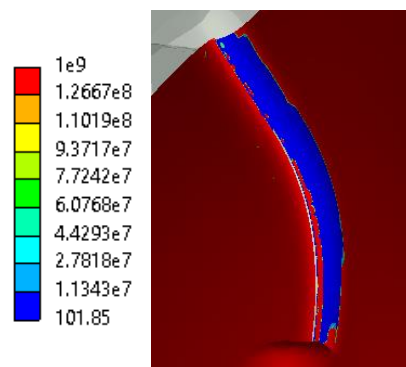


Figure 247 – Number of cycles before crack nucleation determined the Sines method at the fitting region on the core side of Fat_3.

As has been shown for all the regions studied, the coefficient for the most requested area, which is noticeable here that also along the entire notch the coefficient of safety is found to be less than one, as shown in Figure 248. Therefore, this region is also a good candidate for crack nucleation during the lifetime of the mold.

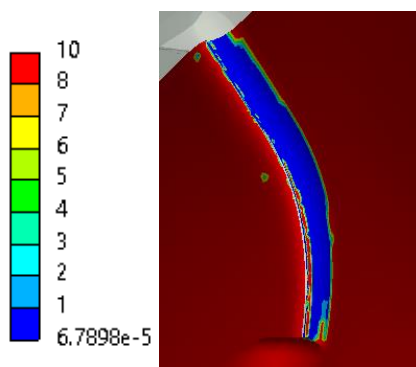


Figure 248 – Safety factor determined by the Sines method at the fitting region on the core side of Fat_3.

For this mold, there are 3 regions where crack nucleation is most likely, these are as follows: one region of the fitting area in the partition plane on the cavity side and the adjacent one on the core side which are illustrated in Figure 238 and Figure 247 respectively; The other is a notch in the engraving

region on the core side as presented in Figure 242.

Taking into account the results obtained and adding these results to those already obtained for the structural analysis of the mold, it can be observed that the most likely regions for crack nucleation are those in the fitting area on both the core and cavity sides. Because the surface in this region tends to have a poor surface finishing than in the engraving regions, which facilitates the crack nucleation, and this associated to the high stresses that are felt in these areas makes it become the main candidate for the crack nucleation and appearance of a visible crack.

When comparing the theoretical results with the ones obtained from the physical model, it is verified that effectively the crack became visible in the predicted region, that is, in that region in the adjustment area but only on the cavity side. What could be expected considering that the numerical models pointed out that region as being the one with the highest stresses. Again, for this mold it is only known that the crack became visible before the end of the mold lifetime, there is no record of the number of cycles performed by the mold until the crack became visible.

6.1.3.4 Results for the tool Fat_4

As it was already possible to verify by the analyses already presented in section 6.1.1 and 6.1.2, this injection mold has a different injection design from those presented so far, which in turn will have an impact on the results obtained for the distribution of stresses in the mold. Molds in which the parts are injected with sequential injection systems tend to have greater variations in pressure throughout the injection cycles and consequently greater variations in stresses in the mold.

Also, the type of steel used in the construction of this injection mold is different from the previous ones, for this mold the EN 1.2311 was used, where its datasheet can be found in Appendix A. For the determination of the reduction coefficients to be used on the different mold surfaces the following was considered: The engraving region on the cavity side was considered to be polished with a good finish; the engraving region on the core side was considered to be machined with a good surface finish as well; as for the adjustment regions in the partition plane, both for the core side and cavity sides, which after machining have been laser hardened, but without any surface treatment to eliminate micro-cracks a reduction coefficient equal to that admitted for the electrical discharge machining (EDM) process was considered.

Therefore, based on the previous considerations, the reduction coefficients relative to the surface finish of the regions to be studied were obtained. Therefore, for the region of the cavity side engraving region the k_s factor of 0,9, while for the region of the core side engraving is obtained a reduction factor of 0,69. The obtaining of these values can be seen in Figure 249 where the red vertical line represents the ultimate tensile stress of the material, with the green horizontal line the value for the polished region

and with the blue horizontal line the value for the machined region. According to the research done, it is admitted that the reduction factor for surfaces machined by the EDM process that the factor is 35% lower than that obtained for machined surfaces, so we have a value of k_s equal to 0,45 [183]–[185].

For the reliability factor (k_e), the same as in the previous molds was used, that is for 99,9% which is approximately 0,753. To calculate the fatigue safety coefficient of the injection mold, it is considered that the mold should support 1.25 million cycles.

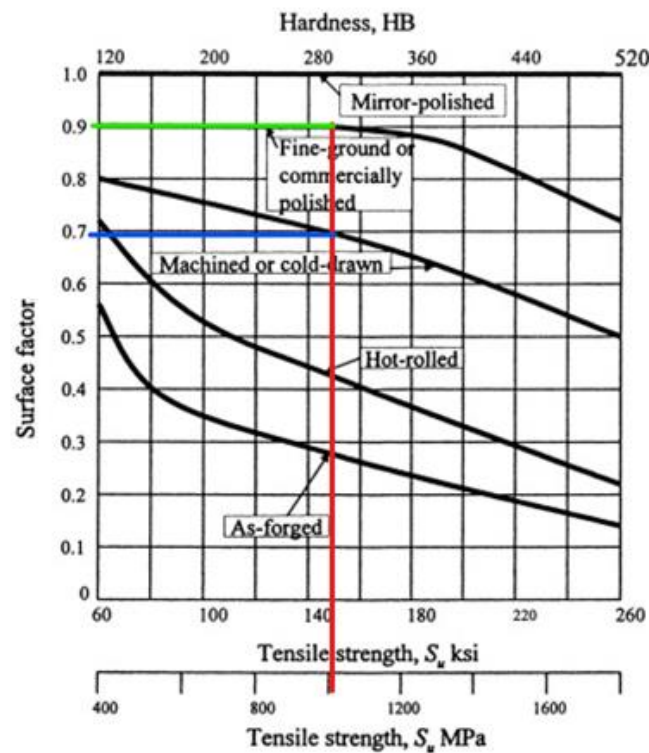


Figure 249 – Fatigue reduction coefficients for the Fat_4 tool.

Therefore, for the engraving region on the cavity side a value of 285 MPa is obtained for the reduced fatigue endurance limit. Taking this into account and analyzing the values of maximum alternated stress presented in Figure 250, it is verified that no region appears to exceed this limit. It is noticeable that in the image only half of the piece is represented because it is assumed that since the piece is symmetrical, the values obtained will also be approximately symmetrical.

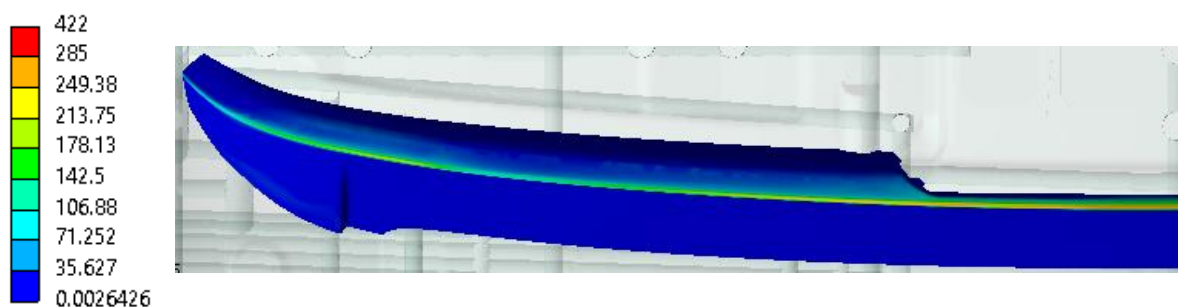


Figure 250 – Maximum alternated stress in MPa at each location by the Sines method at the cavity engraving region of Fat_4.

As to be expected and given that the reduced fatigue endurance limit is not exceeded at any location, it would also not be expected that the damage result would show a value different from 0 as is noticeable in Figure 251. And consequently, the fatigue safety coefficient for this region is maximum throughout, as shown in Figure 252.

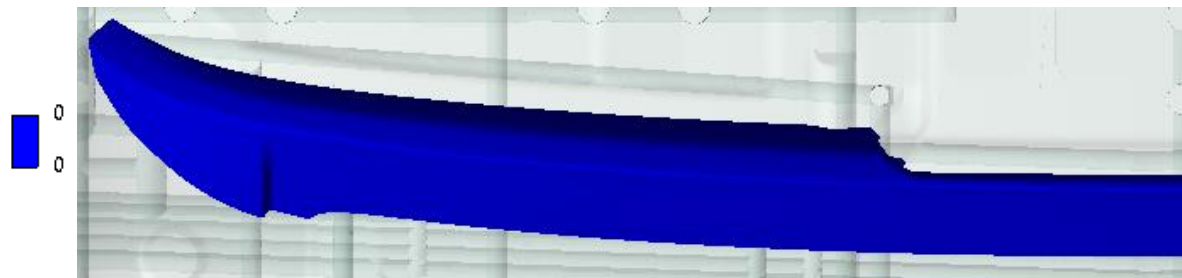


Figure 251 – Damage determined by the Sines method at the cavity engraving region of Fat_4.

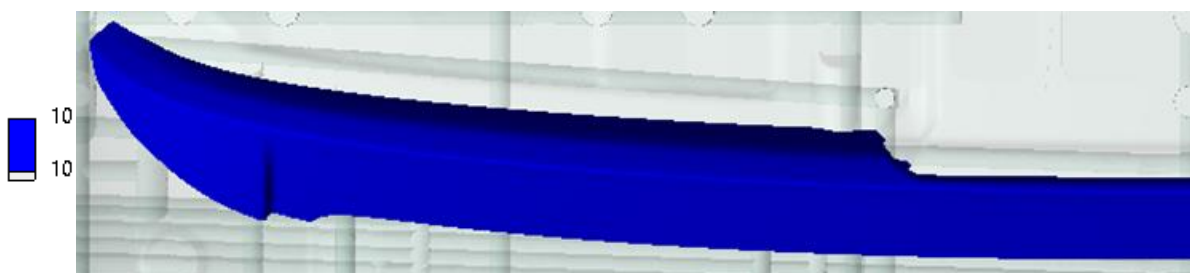


Figure 252 – Safety factor determined by the Sines method at the cavity engraving region of Fat_4.

For the fitting region in the partition plane on the cavity side, it can already be seen that there are places where the maximum alternated stresses exceed the 142 MPa of reduced fatigue endurance limit obtained for this region. Figure 253 shows the maximum alternating stresses for the fitting region in the partition plane on the cavity side, where it is clear that especially near some notches the stresses cannot be disregarded, since besides being high they are very focused in those areas.

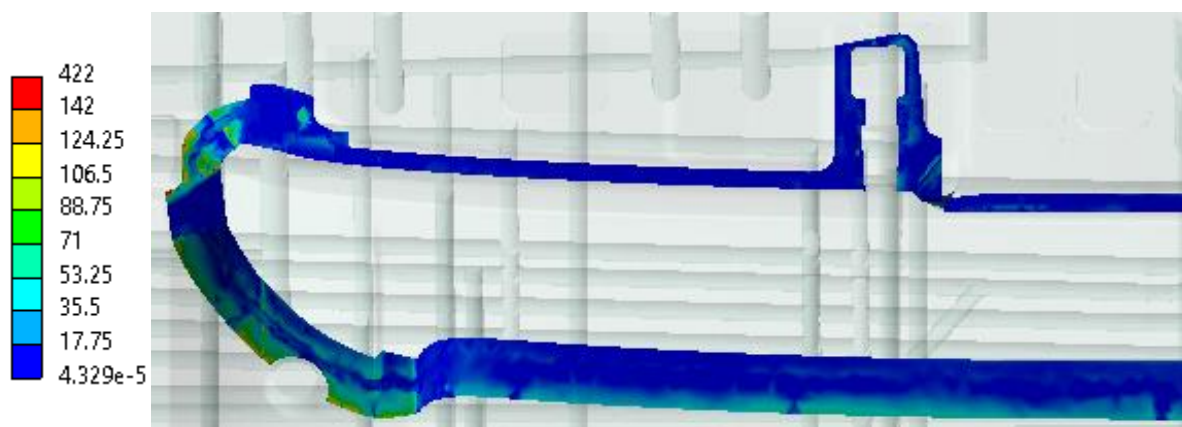


Figure 253 – Maximum alternated stress in MPa at each location by the Sines method at the fitting region on the cavity side of Fat_4.

When analyzing Figure 254 and Figure 255 where the results for damage and number of cycles to crack nucleation respectively are presented, it can be observed that there is a location near a notch that shows to have low fatigue resistance. When observing the number of cycles until the crack nucleation it is verified that there are a set of points along the notch that present low values.

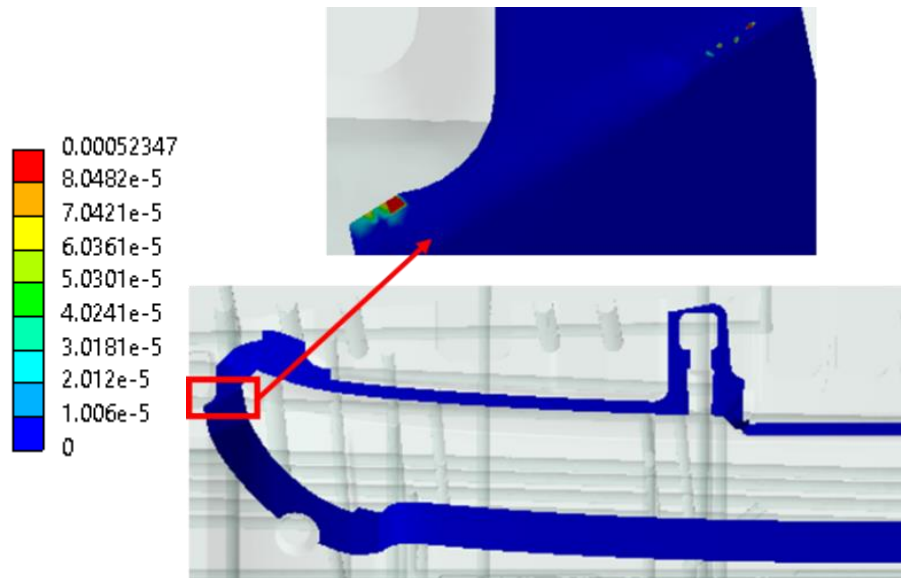


Figure 254 – Damage determined by the Sines method at the fitting region on the cavity side of Fat_4.

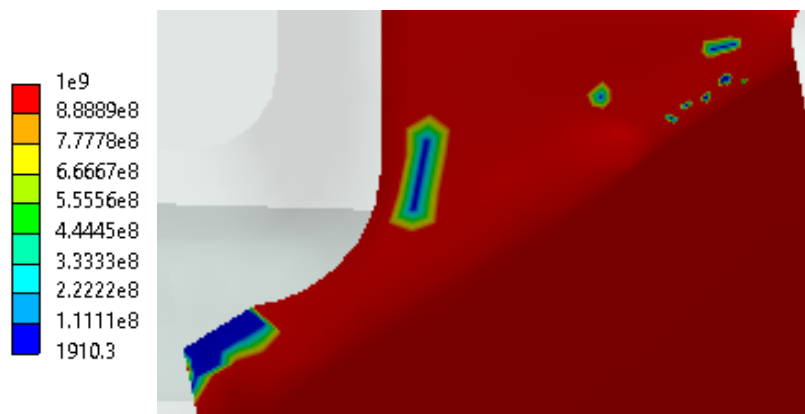


Figure 255 – Number of cycles before crack nucleation determined the Sines method at the fitting region on the cavity side of Fat_4.

The results obtained for the fatigue safety coefficient confirm that the region detected present a high probability for crack nucleation within the lifetime of the injection mold, as can be observed in Figure 256.

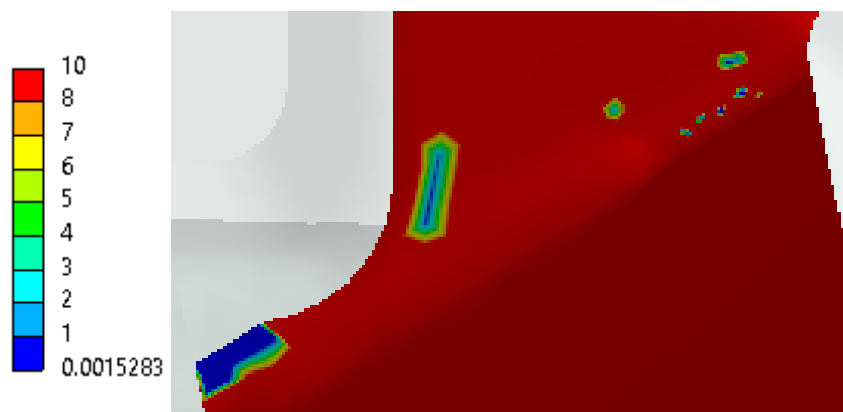


Figure 256 – Safety factor determined by the Sines method at the fitting region on the cavity side of Fat_4.

For the engraving region on the core side it can be seen that the maximum alternating stresses reach the highest values at the most central region of the mold, as shown in Figure 257. It is noticeable that the highest stresses start at the root of the radius and then dissipate along the part.

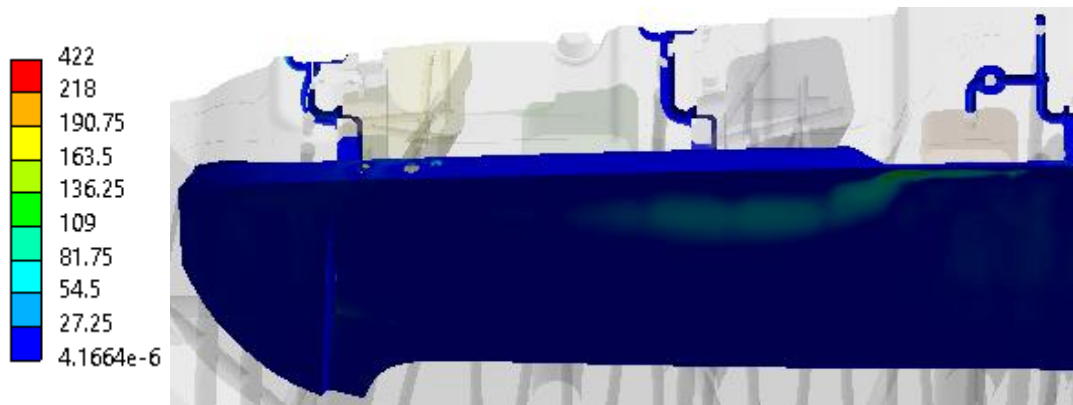


Figure 257 – Maximum alternated stress at each location by the Sines method at the core engraving region of Fat_4.

When moving on to the analysis of the damage and number of cycles to crack nucleation, presented in Figure 258 and Figure 259 respectively, it can be seen that the most susceptible region for crack nucleation is in an notch near the edge of the engraving region.

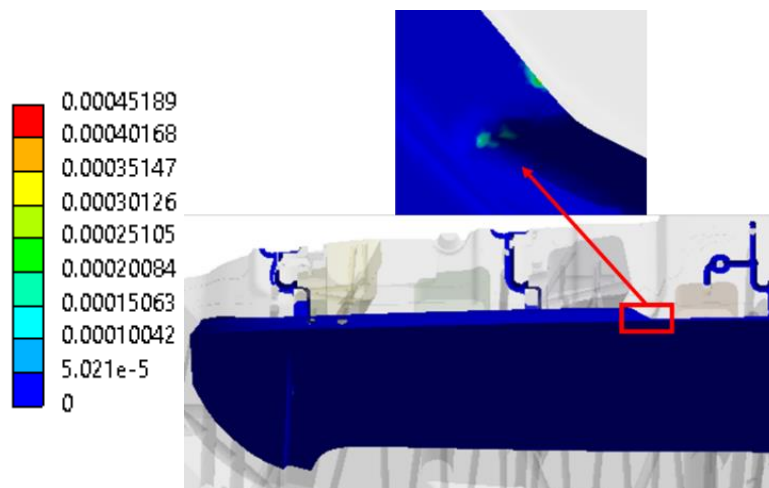


Figure 258 – Damage determined by the Sines method at the core engraving region of Fat_4.

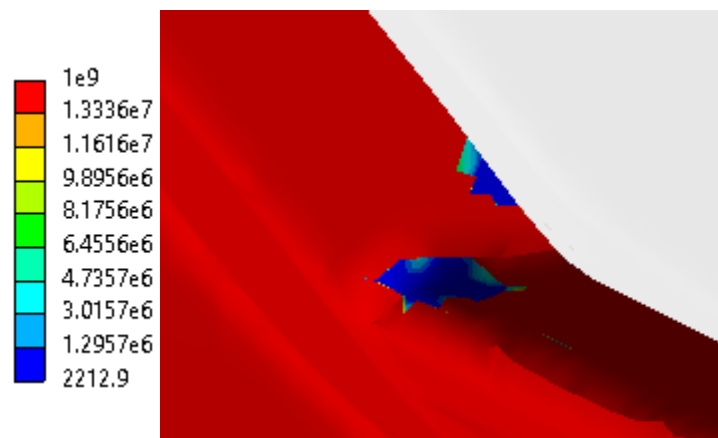


Figure 259 – Number of cycles before crack nucleation determined the Sines method at the core engraving region of Fat_4.

When the fatigue safety coefficient shown in Figure 260 is analyzed, it can be seen that this region is effectively one of the critical points in the injection mold. And as it will be seen forward that the adjacent region is also a critical point, so this point should also be taken into great consideration as a potential region for failure by fatigue.

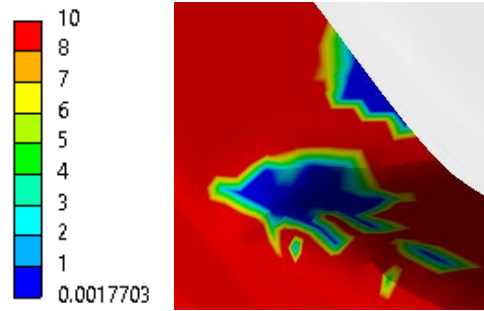


Figure 260 – Safety factor determined by the Sines method at the core engraving region of Fat_4.

Finally moving on to the analysis of the fitting zone in the partition plane on the core side. Where it is possible to verify that there are some regions where the maximum alternating stresses are more significant, as presented in Figure 261. But only one of these regions presents significant damage as will be seen below.



Figure 261 – Maximum alternated stress at each location by the Sines method at the fitting region on the core side of Fat_4.

As already mentioned, only one area presents damage, as can be noticed in Figure 262. And even when analyzing Figure 263, where the number of cycles until crack nucleation is presented, it can be observed that the whole region indicated is propitious to crack nucleation.

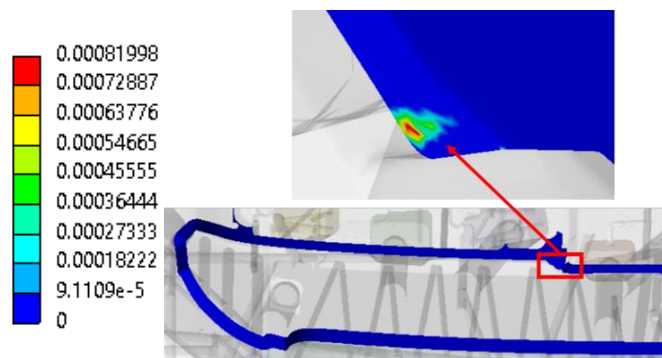


Figure 262 – Damage determined by the Sines method at the fitting region on the core side of Fat_4.

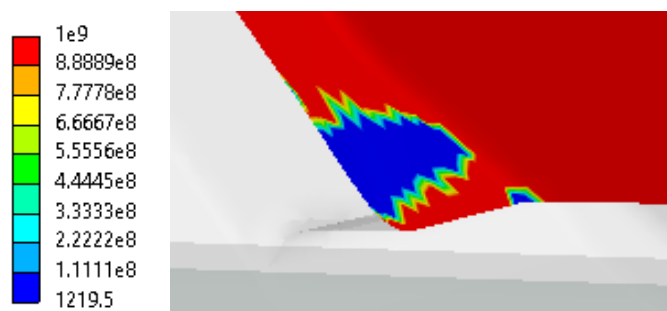


Figure 263 – Number of cycles before crack nucleation determined the Sines method at the fitting region on the core side of Fat_4.

From the analysis of the fatigue safety coefficient shown in Figure 264 for the critical region, it is easier to see the regions that will not be able to support the alternated loads during the mold's lifetime. Therefore, there is a region that starts in the engraving region on the core side and propagates to the fitting zone in the partition plane.

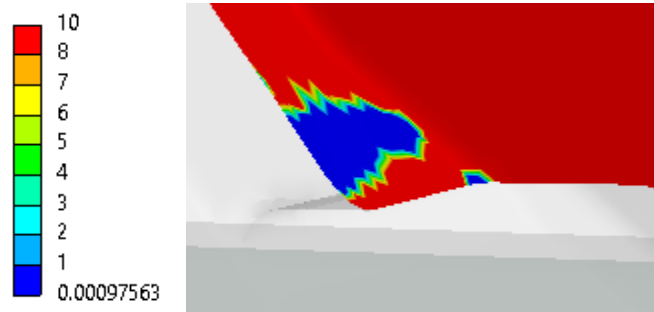


Figure 264 – Safety factor determined by the Sines method at the fitting region on the core side of Fat_4.

From the analysis performed, it can be seen that the region where cracks appear is located in the adjustment region in the partition plane that is identical in Figure 254. For it is the one that theoretically needs fewer cycles until fatigue crack initiation. And next will be the region shown in Figure 258 and Figure 262, which are adjacent and both on the core side. This region extends from the molding region on the core side to the adjustment region in the partition plane of the core side.

When compared with the results obtained in the physical model, shown in Figure 265, where the cracks marked with dye infiltration are shown. The cracks are present in the adjustment region in the partition plane on the cavity side, which is in line with what had already been predicted. However, the cracks were only detected after approximately 148,000 injection cycles. Although it is a value significantly higher than predicted, this represents the number of cycles until the crack is detected. That is, the calculated value is for the crack nucleation, and after the crack nucleation it is necessary that it reaches a sufficient dimension to be detectable.



Figure 265 – Images of the locations where fatigue cracks appeared in the tool.

6.1.3.5 Results for the tool Fat_5

Finally, the last mold under study is quite different from the others, as was already presented in section 6.1.2.5. This mold has the particularity that the engraving regions on both the core and the cavity sides are built from two different manufacturing processes. For, due to the geometry of the part not all regions are possible to be milled so they were manufactured by EDM.

Therefore, in the end, different reduction factors will be obtained for the surface finish for each of the zones. Figure 266 shows the recording regions of the cavity that are manufactured by each of the processes. Given that the molding components of this injection mold are made in EN 1.2311, the reduction factors for the surface finish are as follows: Machined regions on the cavity and tap side with good surface finish is considered a reduction coefficient R_s equal to 0,69; for the cavity and tap regions manufactured by EDM that are also polished to facilitate demolding is considered a reduction coefficient equal to 0,57; The adjustment regions as they are also machined with good finish is also admitted a reduction coefficient R_s equal to 0,69. As for the reduction factor concerning reliability, the same one that was applied for the other molds studied will be used, that is, a factor of 0.753 for a reliability of 99,9%. This tool should be able to endure 1 million cycles which is the predicted number of plastic parts to be manufactured.

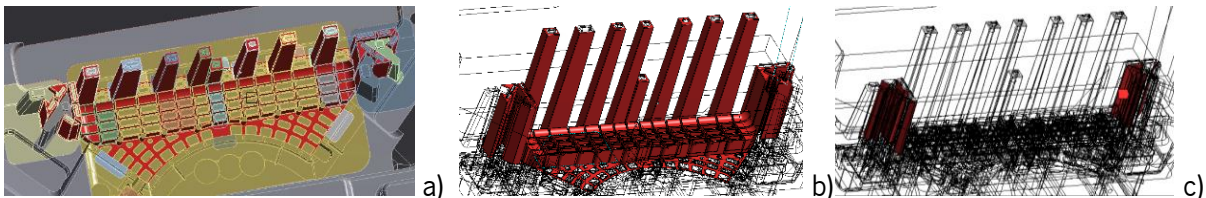


Figure 266 – The a) engraving region is made by two different manufacturing processes where the region b) is milled and the region c) is made by EDM.

In the course of this section the results for the engraving region on both the cavity and the core side will be presented separately for the milled regions and for the regions manufactured by EDM. Starting with the analysis of the maximum alternated stresses in the engraving region on the cavity side of the injection mold that can be seen in Figure 267.

When analyzing the machined region shown in Figure 267 a) it is verified that approximately in the middle of each of the "towers" the alternating stresses are already noteworthy although they are below the reduced fatigue endurance limit of 218 MPa. But as will be presented later, the critical point is at the base of one of the "towers", and its geometry will prove to be favorable to crack nucleation. When analyzing Figure 267 b), where the maximum alternating stresses in the region manufactured by EDM are presented, it is verified that the most stressed regions are near the base of each of the "towers". The reduced fatigue endurance limit is 180 MPa, which is approximately 40 MPa lower than the machined region. Further on it will be possible to verify that again the critical region is located at the base of the

"towers", although here in some places it is much more expressive.

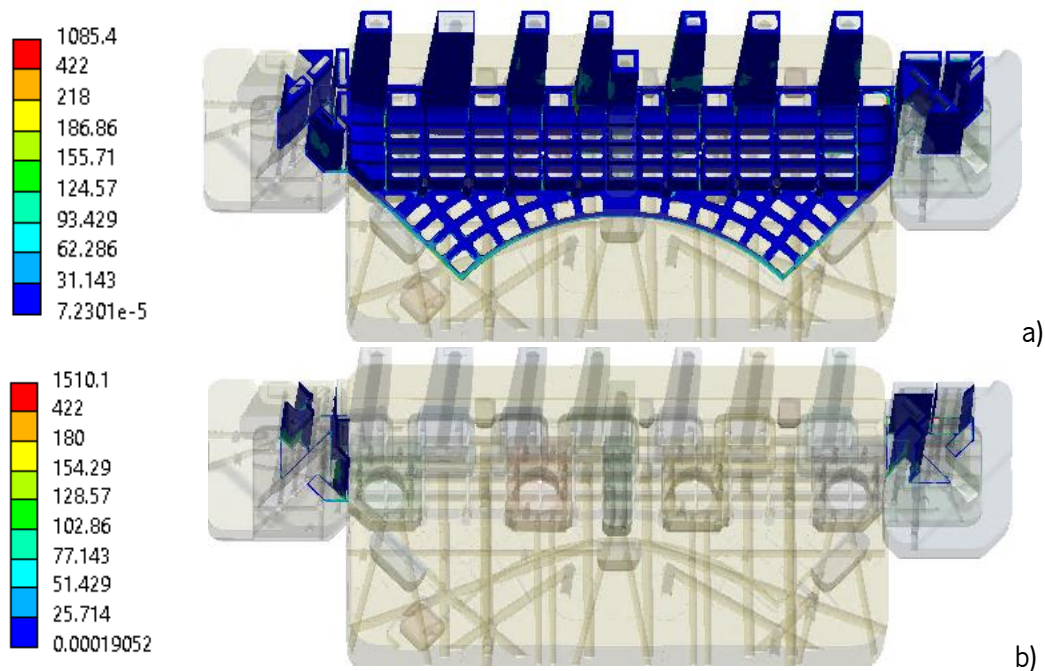


Figure 267 – Maximum alternated stress in MPa at each location by the Sines method at the cavity engraving region of Fat_5 a) in the milled region and b) on the EDM region.

Figure 268 and Figure 269 show the results for damage and number of cycles to crack nucleation respectively. Here it is possible to observe the regions that are more favorable for crack nucleation. Where in Figure 268 a) a critical point in the milled region, situated in the root of a radius, is identical, then by the analysis of Figure 269 a) it is easier to verify that this critical region is very concentrated along the radius. For the region fabricated by EDM shown in figure 263 b) it can be seen that there are two zones where the damage is visible.

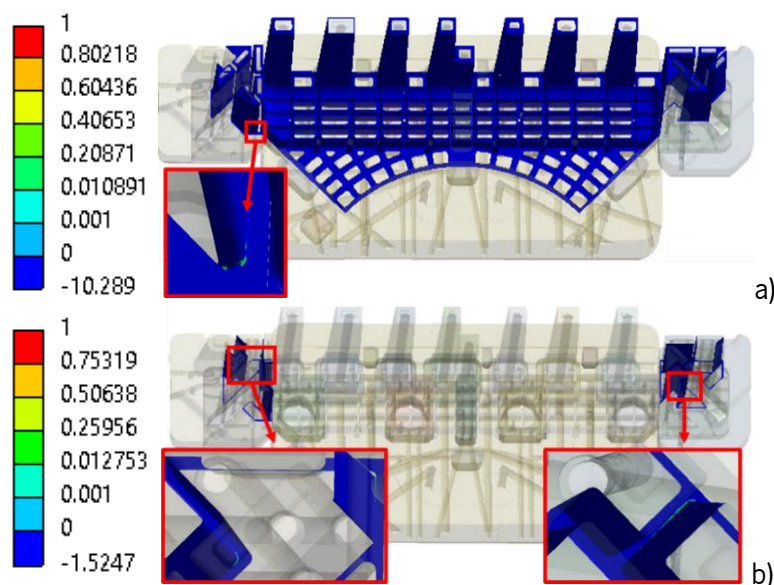


Figure 268 – Damage determined by the Sines method at the cavity engraving region of Fat_5 a) in the milled region and b) on the EDM region.

Also, in the regions manufactured by EDM it is verified that the radii present at the base of the "towers"

are a stress concentration factor that makes them critical regions. In Figure 269 b) it is quite noticeable the fragility of these regions to the alternating loads to which they will be subjected.

But taking into account the number of cycles that the injection mold will have to endure, it can be seen in Figure 270 that the most critical region is in the machined region on the cavity side. In this region it is evident that the fatigue safety coefficient is less than one along the entire radius.

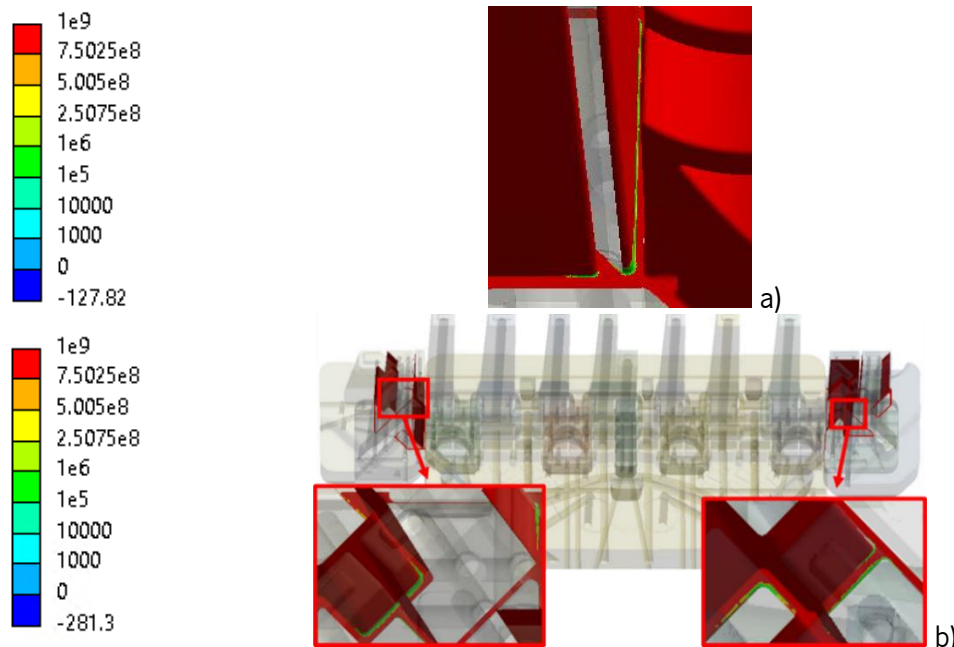


Figure 269 – Number of cycles before crack nucleation determined the Sines method at the cavity engraving region of Fat_5 a) in the milled region and b) on the EDM region.

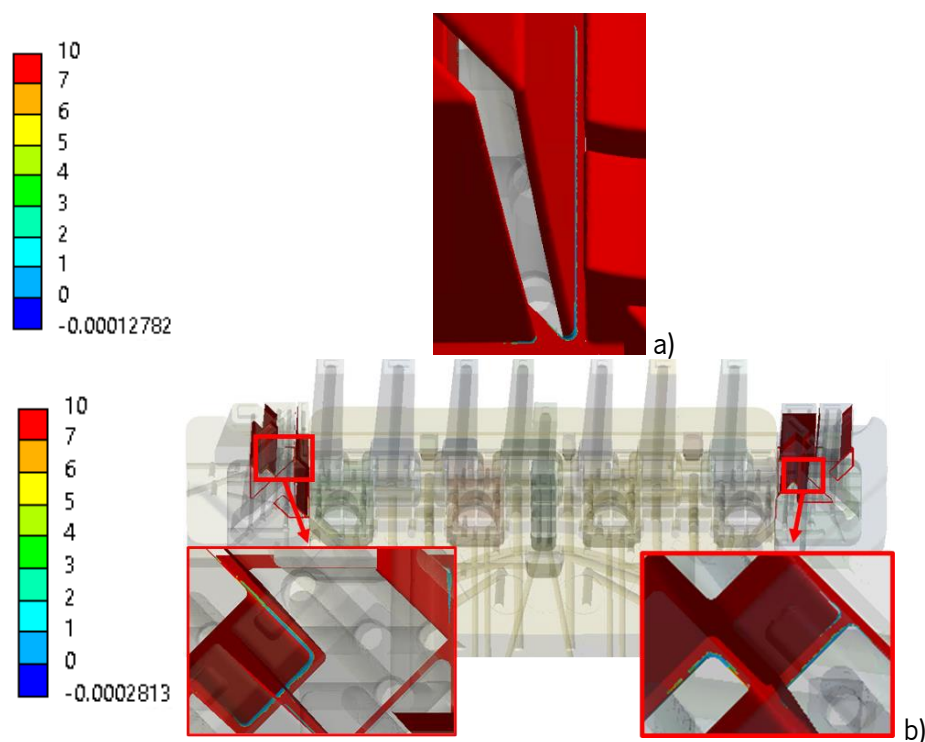


Figure 270 – Safety factor determined by the Sines method at the cavity engraving region of Fat_5 a) in the milled region and b) on the EDM region.

When the Figure 270 b) is analyzed, it can be seen that along the most stressed surfaces the safety

coefficient is globally superior to one. Although there are some nodes that present a result for the coefficient of safety lower than one, this is due to singularities in the model that are typical in this type of geometric shapes.

Now for the tuning region in the partition plane on the cavity side, it turns out that the maximum alternating stresses are globally lower than the reduced fatigue endurance limit as can be seen in Figure 271.

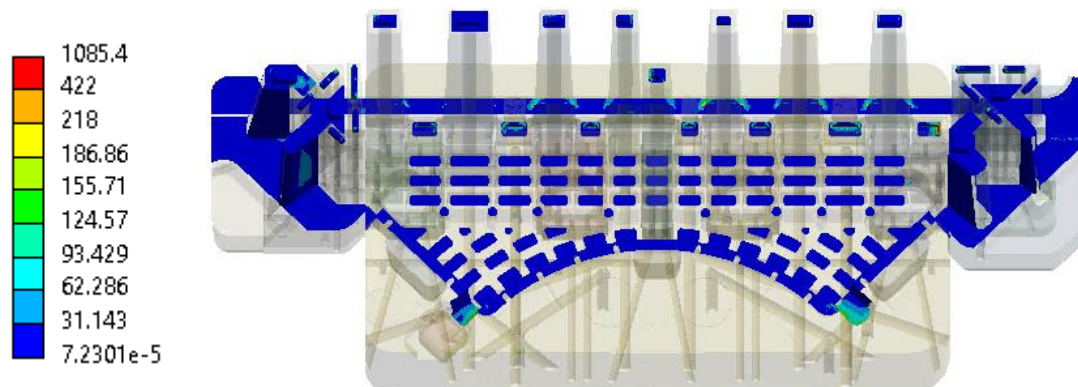


Figure 271 – Maximum alternated stress at each location by the Sines method at the fitting region on the cavity side of Fat_5.

The same conclusion can be drawn when analyzing the result obtained for the damage by injection cycle that is presented in Figure 272, however there is a small region that presents high values that may go unnoticed in the analysis of this fault. But as could already be observed in section 6.1.2.5 in the von Mises stress analysis, there is an intersection between two surfaces that leads to very high stresses in the region marked in Figure 273.

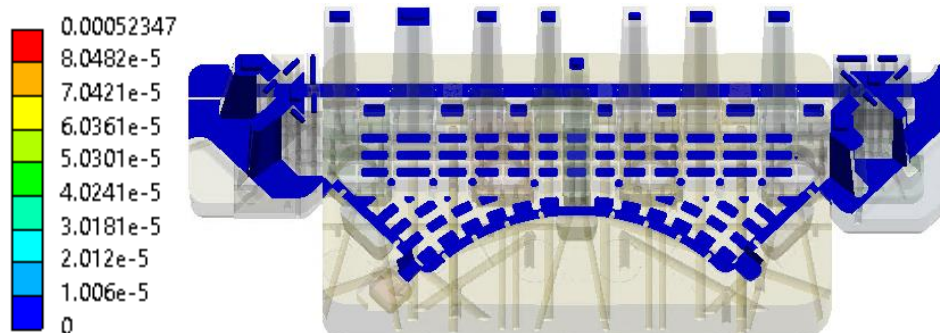


Figure 272 – Damage determined by the Sines method at the fitting region on the cavity side of Fat_5.

As the only location where the fatigue safety coefficient is less than one, which is duly noted in Figure 273, is caused by an intercept problem in the numerical model, it can be considered that there is no point in this region that is susceptible to fatigue failure. Therefore, by the analyses performed on the cavity side there is a location where crack nucleation is likely for the life cycle of the injection mold.

For the core side of the injection mold, Figure 274 shows the maximum alternating stresses for the engraving region. In Figure 274 a) where the maximum alternating stresses in the milled region can be observed, can be seen that there are several regions that present stresses worthy of being analyzed. But

the places where the stresses present themselves as being most critical is at the base of two of the "towers" as will be demonstrated later as being the critical points.

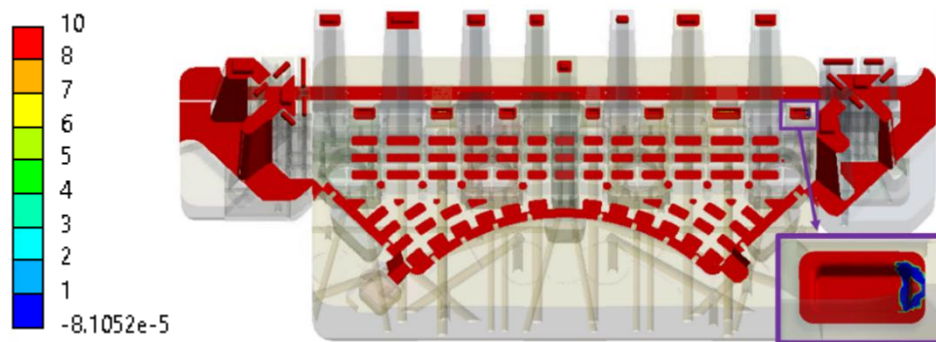


Figure 273 – Safety factor determined by the Sines method at the fitting region on the cavity side of Fat_5.

Again, the concentration of stresses near the radius of a fillet leads to the need for further analysis. The same can be observed in the region fabricated by EDM shown in Figure 274 b), however as there are several places with fillets where the stress concentrations are visible, and they are close to each other this leads to this region having to be properly analyzed.

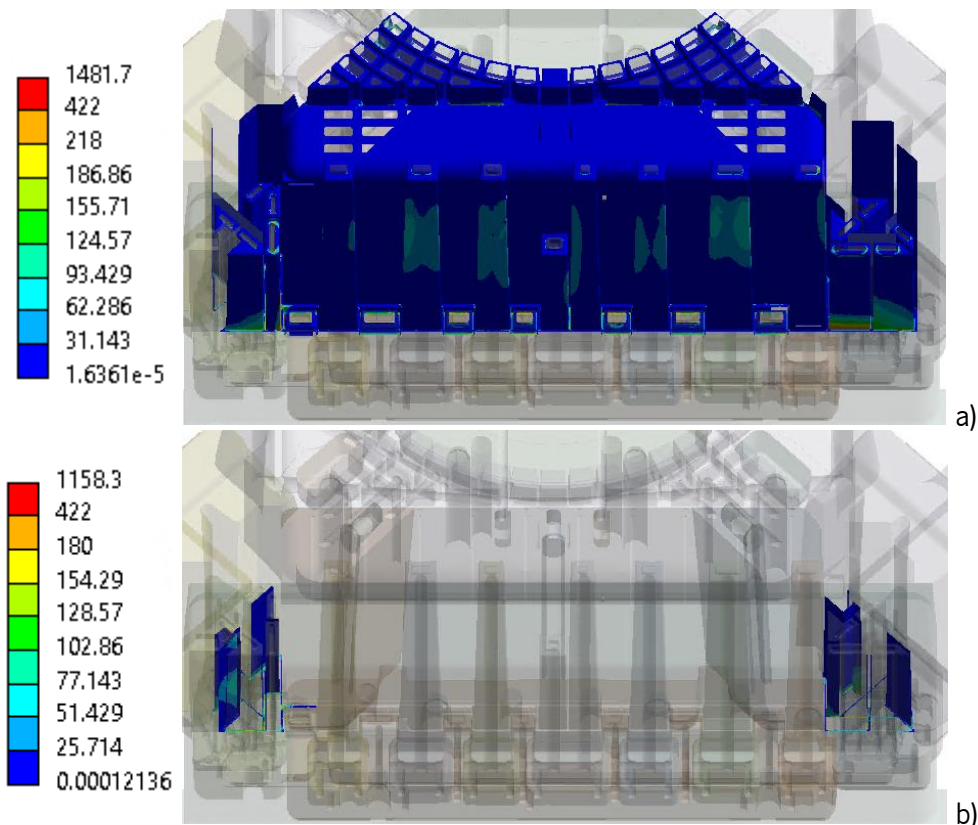


Figure 274 – Maximum alternated stress at each location by the Sines method at the core engraving region of Fat_5 a) in the milled region and b) on the EDM region.

By analyzing Figure 275 and Figure 276 which represent the damage and number of cycles to crack nucleation respectively, it is possible to observe in greater detail the most critical regions in terms of fatigue resistance. In Figure 275 a) can be seen that there are two locations where the damage is considerable in the milled region. And when passing on to the analysis of the same region but now using

the result obtained for the number of cycles until the crack nucleation presented in Figure 276 a) it is perceptible that in the two marked regions, there is a lower fatigue endurance resistance region that starts at the root of the radius and ends up extending along the adjacent surfaces. Of course, that although the result presents an entire affected area, the cracks tend to nucleate in the most fragile region and propitious for its nucleation. Therefore, although can be observed surfaces that are affected by the concentration of stresses in the notch, the crack will tend to start in the notch and once the nucleation of the crack or cracks occurs in the notch region, it will lead to the stresses that will be felt in the region being different from the ones obtained in the original model.

When analyzing the safety coefficient for the milled region shown in Figure 277 a) it is noticeable that of the two regions detected as being critical, one of them shows to be more prone to crack nucleation. The region evidenced in light blue, presents along the radius a fatigue coefficient of safety lower than one. As it was already detected, this region is propitious to stress concentration and the unbalance already described in section 6.1.2.5 is central to the damage increase in this region. Even though the region marked in purple presents significant damage, it is capable of withstanding the number of cycles that the injection mold will have during its lifetime.

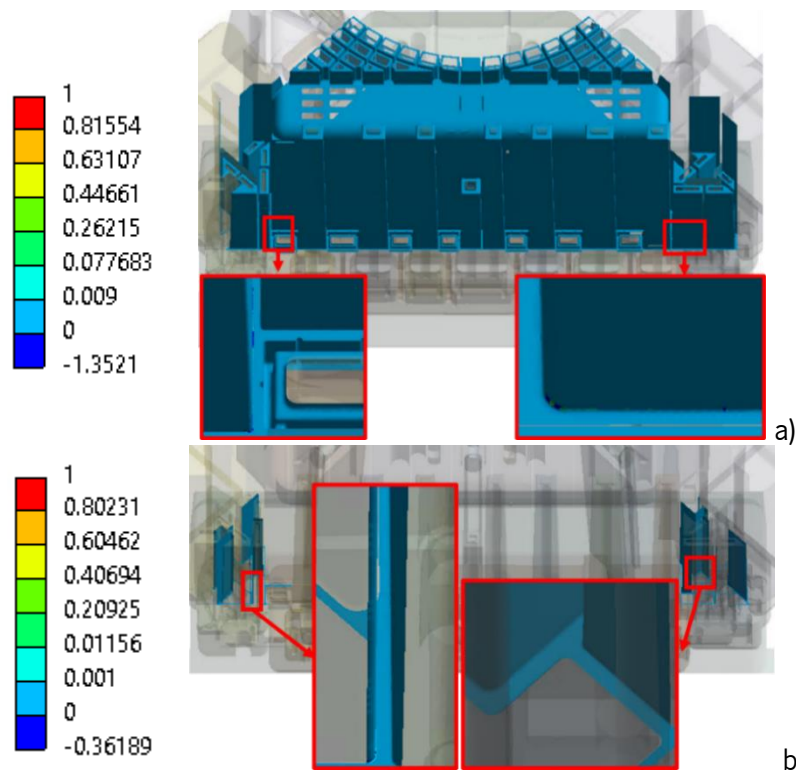


Figure 275 – Damage determined by the Sines method at the core engraving region of Fat_5 a) in the milled region and b) on the EDM region.

When the region manufactured by EDM is analyzed it can be seen in Figure 275 b) that two zones are subject to greater damage than the others, but in Figure 276 b), where the number of cycles that the zones in question should endure until crack nucleation is shown, the extent of the critical zones can be

seen. Basically, all the fillets where the stress concentration is visible end up being critical zones. For here it should be considered that the resistance to fatigue failure is lower based on the manufacturing process, but it should also be taken into account that the type of polishing and polishing quality of the surfaces is fundamental for this analysis to be reliable.

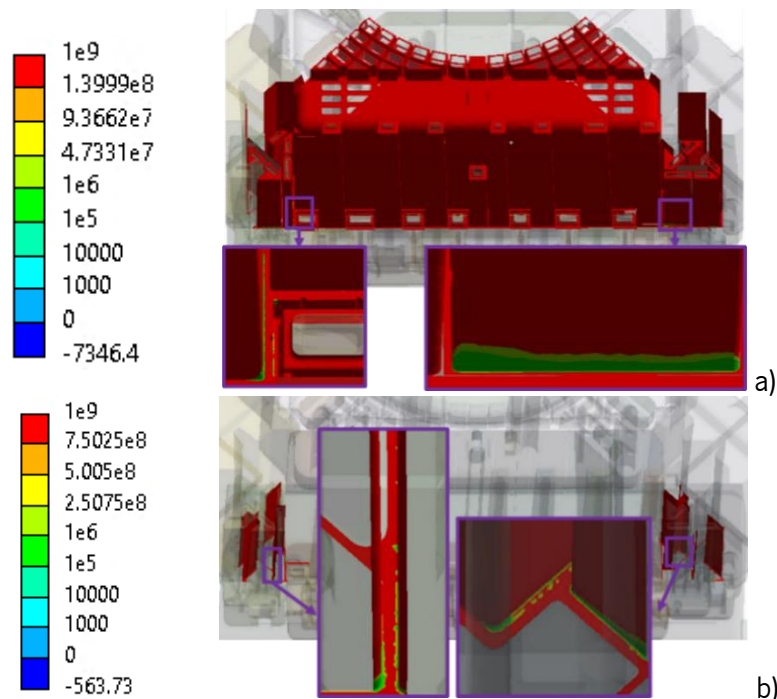


Figure 276 – Number of cycles before crack nucleation determined the Sines method at the core engraving region of Fat_5 a) in the milled region and b) on the EDM region.

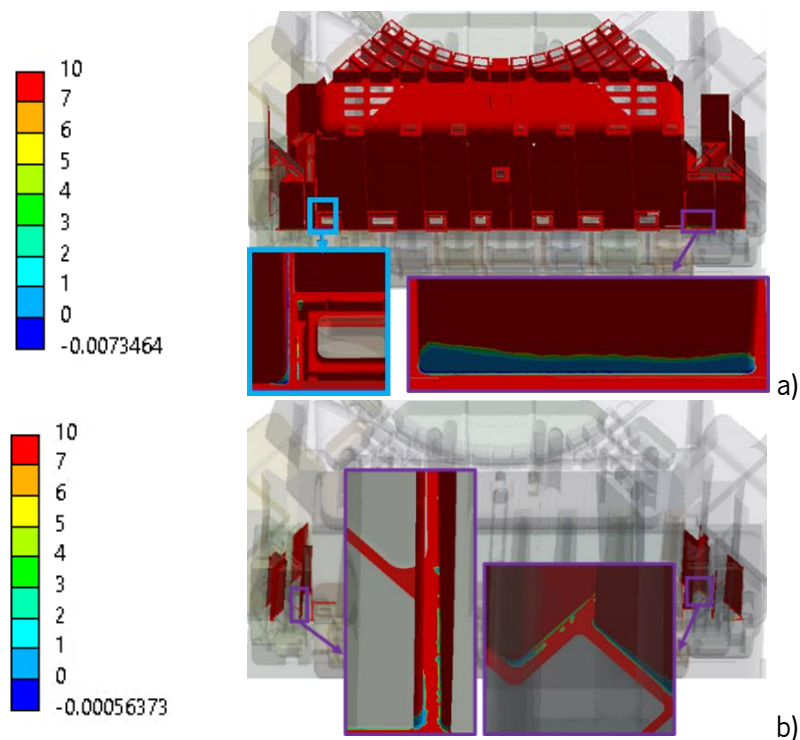


Figure 277 – Safety factor determined by the Sines method at the core engraving region of Fat_5 a) in the milled region and b) on the EDM region.

However, when analyzing the fatigue safety coefficient for this region, which is presented in Figure

277 b) it is noticeable that the lowest values are greater than one. Therefore, for the lifetime of the injection mold the appearance of cracks in this region should not be expected.

Finally, to conclude the fatigue analysis of this injection mold, the fitting region in the partition plane on the core side of the injection mold will be analyzed. As it will be demonstrated from the results obtained, there is no probability of crack nucleation in this region, but it will be possible to demonstrate the problem of oversimplification of the geometries.

In Figure 278, where the maximum alternating stresses are presented, it can be seen that there are two places where the maximum alternating stresses exceed the reduced fatigue endurance limit. This is due to a simplification performed in the CAD model and consequently passed on to the numerical model, which causes the results obtained to be significantly different from those that could be measured from the physical model.

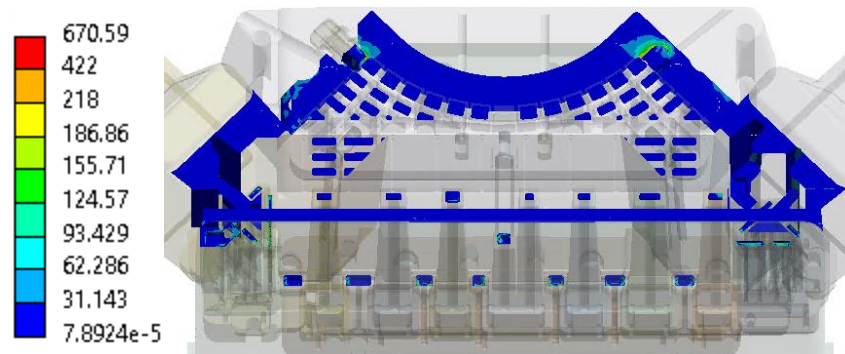


Figure 278 – Maximum alternated stress at each location by the Sines method at the fitting region on the core side of Fat_5.

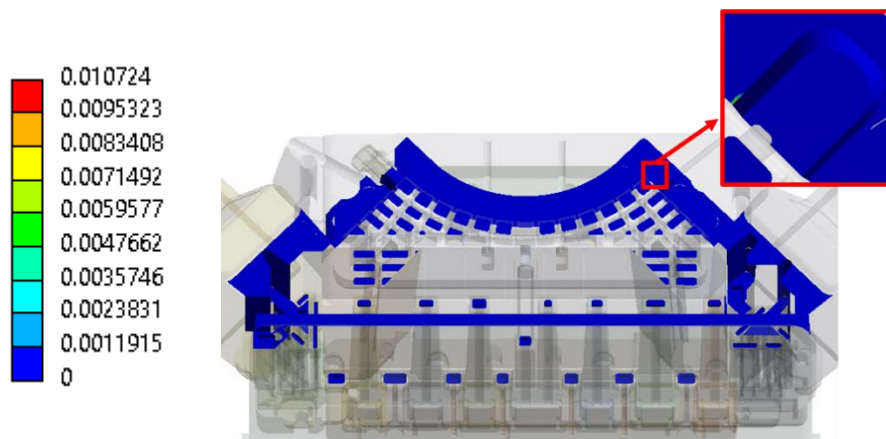


Figure 279 – Damage determined by the Sines method at the fitting region on the core side of Fat_5.

As can be seen in more detail in the highlighted region of Figure 274 the fillets have been removed, this improves the computation times because the number of model elements is reduced. But this brings the disadvantage of increasing the effects of stress concentration and consequently the increase of the alternating stress amplitude in the region. By doing an analysis based only on results without trying to understand the causes, this region would soon be considered a critical point.

Because the results obtained for the number of cycles until the crack nucleation presented in Figure

280 or even the fatigue safety coefficient presented in Figure 281, show that the crack is very probable, however the intersection of two milled surfaces is not in a sharp angle. This zone is always with the radius of the tool that will be milled and being a locking zone of the mold is machined with a significant radius in order to facilitate the machining, the adjustment and its resistance.

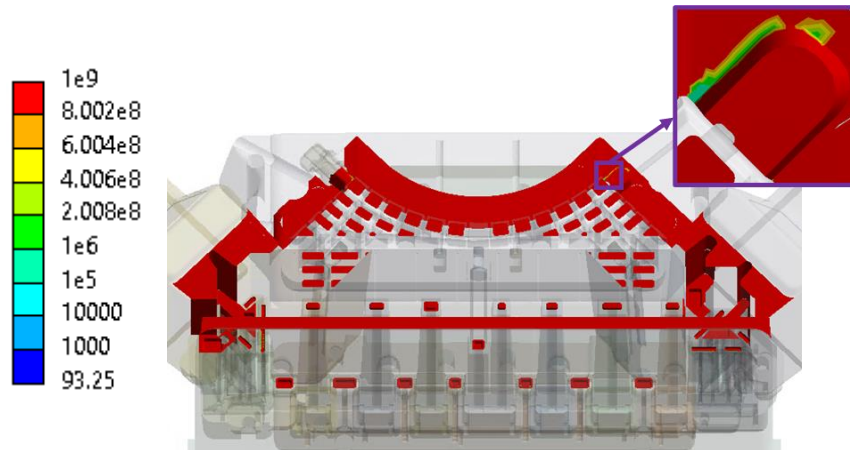


Figure 280 – Number of cycles before crack nucleation determined the Sines method at the fitting region on the core side of Fat_5.

Therefore, it is not expected that there are any problems related to fatigue failures in the tuning region on the partition plane on the core side for the Fat_5 injection mold.

Therefore, from the analysis of this injection mold it is possible to conclude that there are two zones of the injection mold, one on the cavity side and another on the core side that are susceptible to crack nucleation as a consequence of the alternated loads to which the mold will be subjected.

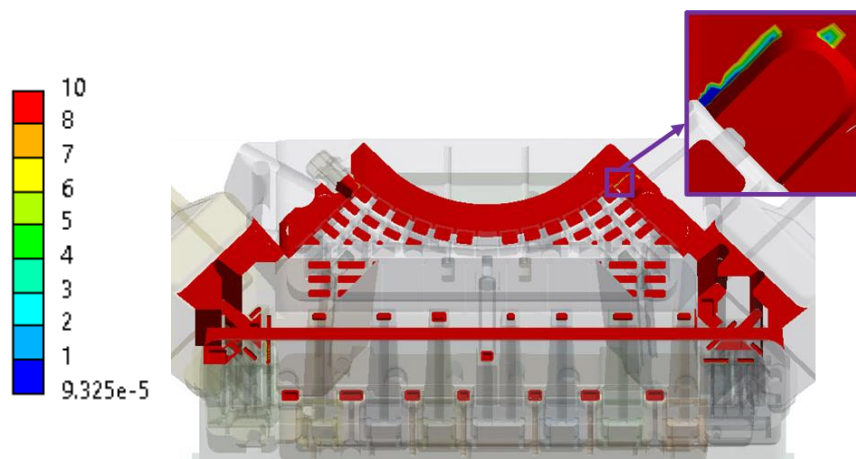


Figure 281 – Safety factor determined by the Sines method at the fitting region on the core side of Fat_5.

The regions detected as being the most propitious for crack nucleation are presented in Figure 282, which have already been identified in Figure 270 a) and Figure 277 a). As it is visible the locations most susceptible to crack nucleation are at the base of the "towers" that are in contact in the mold partition plane, i.e., when bending occurs in one of them the other ends up following the displacement. Considering that this region is located on the side where there is the unbalancing of the injection, the component on the side of the mold ends up being more stressed. This is because the side of the tap is

the last to fill with molten polymer, the pressure exerted on the walls of the components ends up being more unbalanced than the side of the cavity. As in this side of the mold there is an imbalance in the filling which ends up stressing this region even more, that leads to be prone to the nucleation of cracks.

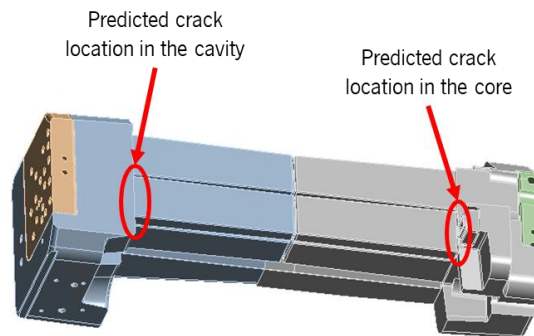


Figure 282 – Predicted locations for fatigue crack nucleation in the injection mold Fat_5.

When comparing the results obtained from the physical model with the calculated ones, it was possible to verify that the region where the crack was detected corresponds to the predicted location on the tongue side. As previously referred, the region where it was predicted the nucleation of the crack on the core side would be the most probable of appearing a crack because it was the one that presented higher alternating stresses for a larger region. Figure 283 shows the mold component where the crack was detected, which corresponds to the insert on the male side where the crack nucleation was predicted. As for the predicted lifetime, at the root of the radius it would be on the order of 1000 cycles. However, the crack in the component in the physical model was detected at the end of approximately 9000 injection cycles. However, it is necessary to take into account that the prediction is made for the crack nucleation, and the values of the physical model are for when the crack is detected. Probably the difference between the predicted and the actual is much lower.



Figure 283 – Image of the cracked component where the region with the visible crack is highlighted.

6.2 Chapter conclusions

This section aimed to apply a methodology for fatigue calculation of injection molds from numerical models. To this end, rheological simulations were performed, from which the loads to be applied to the injection molds were taken. In the structural simulations the criteria mentioned in section 5 were

considered, but in the Fat_5 mold a simplification was made in a region that was not critical in the mold in terms of operation to demonstrate one more problem that can occur from geometric oversimplifications. From the structural simulations, the necessary information was gathered so that an application external to the finite element software could perform the necessary calculations to be able to make a prediction of the fatigue resistance of the five molds studied.

Therefore, to obtain the loads from the rheological simulations are very important, because this is where the pressure values in the molding region will be exported from and where the clamping force of the injection mold is determined. Although in this study the machine parameters were used as input data for the rheological simulations at the design stage, these simulations should be performed as rigorously as possible, not only to predict the injection quality, but also to obtain the loads for the structural simulations with a good approximation to the conditions under which the injection mold will work.

For structural simulations of injection molds that work without large temperature variations, that is, without external systems that drastically vary the temperature of the steel, this variable can be neglected because the variations tend to be small and with little influence in terms of reducing the value of the fatigue endurance limit. As for potential problems of stress accumulation caused by thermal expansion, one should bear in mind that an injection mold must be tuned to the working temperature so that all its moving components are not too stuck due to this factor. The stress concentrations due to thermal expansion are eventually relieved during the process of tuning of the injection mold.

In structural simulations, the preparation of the numerical models begins with the preparation of the geometry. That is, it is necessary to clean up the geometry that will be calculated in order to reduce computation time, but at the same time care should be taken not to oversimplify because this can lead to slight deviations in the results that can be crucial to the decision about whether the product is properly sized or not. And as demonstrated in the Fat_5 mold where a simplification of the geometry was made in a region that would have to support high loads, it was soon verified that this led to it becoming a critical prompt.

Therefore, it is verified that the rigor and care in the simplification of the geometry leads to results that can be obtained with less computation time but without a loss in the quality of the results obtained. This is extremely important because as can be seen in Appendix C the computation times are significant, but the care in preserving them is critically important because molds are custom tools and component failures can take weeks before they can be fixed.

Finally, after rigorous analysis of the structural simulations and validation of the results, i.e., if no structural problems or deformations problems inherent to the elastic deformation of the mold when

subjected to loading are detected, one can move on to the exportation of data for further fatigue analysis of the critical regions.

In the structural simulation validation analysis, it is essential to verify if the number of time instants used in the simulation are sufficient to describe the mold behavior throughout the injection cycles. It is normal not to use all the time-steps exported from the rheological simulation software for the structural simulation because many of them will not bring relevant information and would significantly increase the computation time. But using too few time-steps can lead that problems occurring over time may not be detected. In the Fat_4 and Fat5 injection molds this is evident, in one because of the injection concept which leads to significant pressure variations within the engraving region and in the other because the injection is not balanced which leads to fatigue resistance problems.

For the calculation of the most critical regions for the occurrence of fatigue failure, the application developed to perform the calculation from two different methods tends to be an added value for the design phase. First of all, except for the reduction coefficients related to the factors that affect fatigue life, it does not require specialized knowledge to perform the calculations and still brings the added value of being able to calculate all the nodes of a given region in a fast and methodical way, which reduces the possibility of operator errors. However, the analysis must be done with great criteria, for example, when you get negative damage values or a negative safety coefficient, must understand the cause of these values that are associated to stresses in a region that exceed the ultimate tensile strength of the material. Because they may have various causes such as singularities in the numerical model or intersections in the mesh of different components and this should be analyzed to understand if these results compromise the analysis or if they are isolated without much influence on what is around them.

Therefore, the use of the application developed associated with the ability of this application to be associated with numerical simulation software in order to present the results obtained is an added value in the design phase. However, the definition of the reduction coefficients associated with the factors that affect the S-N curve is very important and is a work that should be developed to use coefficients that are less conservative but still able to lead to safe results in the design phase. This work should consider the way the application generates the S-N curve used for the fatigue steel characterization that is based on the models already studied in section 4 of this paper.

Finally, the proposed methodology for performing fatigue strength verification calculations made in this section, which starts in the rheological studies until culminating in the calculation of the critical regions for its fatigue strength proves to be an added value. Considering that in a very visual way it is possible to demonstrate if the region will be able to withstand the alternating loads that it will be subject to during its lifetime. With this, a designer can have a tool dedicated to the injection molding industry

that can easily be used during the design phase and thus avoid the huge costs associated with fatigue failures.

7 Conclusions

This work had as a major objective the creation of a methodology and tools to determine the fatigue resistance of injection molds. Objective that it was possible to meet by working in several areas related to the mold design process.

To accomplish all the work it was necessary to study the following areas: Characterization of steel alloys to its fatigue properties from empirical models; Creation of links between a rheological numerical simulation software and structural simulations software's; In structural simulations understand what kind of simplifications can be used and what kind of numerical simulation regimes can be used; And finally, use the points studied previously to perform structural simulations for subsequent fatigue analysis of critical regions with the help of an application developed for this purpose.

Starting with the fatigue characterization of tool steels, in this work literature research was conducted in order to find out what proposals existed for the generation of S-N and ε -N curves that were able to characterize the types of steels that are commonly used for injection molding. But for these empirical models, the search was restricted to models that were able to characterize the material from properties that could be obtained from data sheets from suppliers, without the need to perform tests on the steels.

After obtaining formulations capable of complying with the imposed restrictions, an experimental trial of a common tool steel in the injection mold industry was performed, in order to understand how the empirical formulations would be able to generate curves that would describe the fatigue behavior of the steel. From this trial it was possible to choose an empirical curve, which presented conservative results when compared to the experimental trials.

The choice of a conservative curve was made due to the need to have a safety margin, since fatigue testing of the steels used in each injection mold is not feasible considering the time available from the start of the project until the mold is delivered for the first tests. And since injection molds are unique tools, using data from other injection molds turns out to be somewhat irrelevant for the characterization of the steels, since they can come from different suppliers and not to mention the problem that the steel blocks even coming from the same supplier can come from different castings, so it is very likely that they have different behaviors. For example, when one looks at the results obtained for the tensile test of the steel specimens studied in this paper, it can be seen that both the ultimate tensile strength and hardness in most trials are higher than those stated in the supplier's data sheet. This happens because in the data sheet the supplier describes an average value, that is, it can be conservative or not and for this reason a model that produces more conservative curves was chosen. Furthermore, although this work is based more on the generation of empirical S-N curves, the developed application also gives the coefficients for

the construction of ε -N curves, which according to the literature research carried out, are also values that will lead the curve to present conservative values. The advantage of these models is that they are based on monotonic steel properties that can be easily obtained without the need for time consuming, complex and expensive experimental trials.

The purpose of creating this application was to feed the materials database of the structural simulation software. For example, Ansys Mechanical allows to make a very simple fatigue analysis that can be useful for a first approach in a preliminary stage of the project. But to perform this analysis it is necessary to add to the materials database the information needed to generate the curves.

In structural analysis there was always the problem of how to consider the loads because the distribution of loads along the region of engraving is not homogeneous, especially during the filling phase. Therefore, either an average pressure was applied to the entire molding region was considered or boundaries were created in the CAD model of the engraving region to be used later in the structural calculation on the finite element analysis software as boundaries for the pressure intervals that were admitted for each region, but this, besides being a complex process, tended to be not very rigorous. Therefore, since there are rheological simulations from which values for pressure or/and temperature can be obtained, it would be more interesting to use this information in the structural simulation. Thus, by maintaining a rule where the molding regions in the structural simulation and the part in the rheological simulation have the same location relative to the global coordinate system, one can pass the desired information from the rheological simulation to the structural one. The application created uses data exported from the Moldflow software to convert it into a format that is understandable by structural numerical simulation software's. With this it is possible to map the engraving surfaces with the pressure and/or temperature values obtained from the rheological simulation. With this tool it is possible to improve the quality of the structural simulation input data and thus have a better understanding of the phenomena that are occurring in the injection mold during the injection cycle. It is also possible to better predict the problems that may occur during the phases in which there is a greater imbalance of pressure distribution within the mold or even when the molding region is not balanced. As for example in the Fat_5 injection mold, if a homogeneous pressure was used throughout the entire molding region, it would overlook the instant when the greatest damage is caused in the mold. Also, for the mold fatigue analysis the use of several time-steps that account for the pressure variation throughout the filling phase is indispensable. Because in molds with only one injection point, it can have one or two alternating loads per cycle, but for injection molds with sequential injection this may not be true. Therefore, the application developed besides being useful for calculating the fatigue resistance of the injection mold, it also helps

to understand phenomena that can give rise to defects in the injected part that using a homogeneous pressure throughout the molding region could be overlooked.

For the work done in the analysis of structural numerical simulations where it was possible to compare with the physical models from sensors that were embedded in the injection molds, it was possible to understand some important aspects for the preparation and analysis of numerical models. Firstly, the preparation of numerical models is an important point because these models are usually large, since the size of the injection molds is high and with many small details, with the particularity of having to detect displacements in the order of tenths of a millimeter. So, simplifications must be made to try to save as much computation time as possible, but they must be made in such a manner that their influence on the results is negligible. From the comparisons it was possible to verify that the results obtained from the structural numerical simulations tend to be conservative, however they are very dependent on the values obtained from the rheological numerical simulations and the rheological simulations tend to present shorter cycle times when compared to the real cycle times, even when trying to replicate the machine parameters. Therefore, this leads the choice of using structural simulations in transient regime to be questioned, since this type of simulation considers factors directly related to the load acting time. Through the analyses performed, it was verified that even if it was known that the time intervals of load action, mainly during the filling phase of the molding region, were smaller than the ones obtained experimentally. It was also verified that the differences were not significant when compared to the simulations in quasi-static regime. Therefore, for this type of molds there is no need to use simulations in transient regime, because the increase in computation time is not justified when compared to the slight differences between the results obtained.

Five injection molds were studied having under account the methodologies and procedures learned by the study of the previous injection molds and which according to the data that was provided they had presented fatigue related cracks. All these injection molds had not been subject to changes that would lead to the addition of material by a welding process, because this type of intervention would compromise the results of the fatigue study that was intended to be performed. And using the procedures for obtaining information to perform the structural numerical simulations and the care to be taken with the preparation of the models, the simulations were performed based on the information provided about the operating conditions of the injection molds. From these structural simulations, calculations were performed to find out if the built application would be able to predict the occurrence of these cracks in a design phase and with this could have avoided these failures.

By the analysis performed it was verified that the application was able to predict the locations in which the cracks appeared in the corresponding injection mold. But it also predicted the appearance of cracks

in other regions that were not detected in the physical models. While the intention of the application is to predict where there is a risk of crack nucleation occurring, it does not perform a crack propagation study. Hence, it is possible that some of the locations where crack nucleation was predicted in the physical model don't present a visible crack, because if it is still in stage I it is possible that it hasn't reached a dimension that could be detected with the means at disposal. As in the injection molds there is the objective of not having cracks, detecting the points where crack nucleation can occur is important to be able to prevent this kind of failure.

The use of this application to calculate the fatigue strength as well as to analyze the results of numerical simulations requires the ability to understand the results that are being presented. That is, as it was possible to observe many times results like the number of cycles until crack nucleation with negative values, or the damage value per cycle that are often related to singularities in the models. Although these singularities could be minimized with further mesh refinement, this would drastically increase the computation time. Therefore, the designer will have to analyze the results in order to understand if it is necessary to refine the model or if the singularity can be discarded. Obviously, this leads to it being complex to determine the number of cycles until crack nucleation, but at the same time, as observed in this work, when the extent of the region with significant damage greatly exceeds the region affected by the singularity the probability of nucleation of a crack or several cracks occurring in that region is high. And, since the appearance of cracks cannot occur, determining the number of cycles until the crack nucleation in a more detailed way ends up being irrelevant. Also, in all the analyses performed it was verified that the fatigue safety factor analysis ends up facilitating the analysis of the problem because it becomes easier to detect the critical points. Therefore, using this application developed specifically for performing fatigue strength calculations for injection molds proves to be very useful, as it is relatively easy to use and is able to quickly present results to a wide range of finite element software, as long as they are able to export csv files with the necessary information. And if the software is able to import csv files to analyze the results, the analysis can be done directly in the software the designer uses, otherwise can be used open-source software like the one presented in section 7.1.3.

Furthermore, with this work it was possible to create tools and methodologies that can be directly applied in the mold industry in order to help designers to detect potential problems related to fatigue damage, and thus be able to act on these potential problems still at the design stage.

8 Future research

The work developed should be considered as a first approach to the problem, since there are still several points that need to be developed. For the characterization of materials, although for the most common steel alloys the empirical model presented is able to answer the need in a conservative way, there are new alloys that may not be well characterized by this formulation. With the constant need to respond to market requirements, several steel alloys with significantly improved thermal properties than traditional steels have begun to appear, and these alloys should be studied in order to understand their fatigue behavior. Also, with the introduction of additive manufacturing in the mold industry, the possibility to build more complex geometries has increased, but at the same time all the formulation used in this work may not be able to give the answer needed for the determination of fatigue strength from this process. Therefore, it is crucial to analyze all the variables that can influence the fatigue strength of components built by additive manufacturing. Still within the injection molding processes that are used nowadays, there is a technology that is based on varying the mold temperature drastically in order to have the mold hot during the injection phase and then decreasing the mold temperature drastically in order to reduce the cycle time without compromising the part's appearance. Studies need to be conducted in order to understand if other care needs to be taken in the construction of the numerical model in addition to understanding if the methods considered for determining fatigue strength in the application developed in this work remain valid.

Finally, in addition to finding out the fatigue strength of the components it would also be interesting to know more information about the regions most likely to suffer fatigue damage. That is, it would be very interesting to know in which planes the crack nucleation is more likely to occur, or to know if the crack will tend to grow along the surface or if it will tend to grow in another direction, because with this information the designer would have more information to be able to create solutions that avoid the appearance of cracks in the mold. But of course, models that allow performing this type of analysis would have to start from the same principle as the one used in this work, i.e., it would have to be possible to perform the calculations from the monotonic properties of the steel because this is the only information that designers have in the design phase and performing steel tests is not feasible due to economic and/or time constraints available for the project.

References

- [1] “Uma breve história da injeção plástica, e como tudo começou. | Blog | Stark Ferramentaria.” [Online]. Available: <https://www.starkferramentaria.com.br/uma-breve-historia-da-injecao-plastica-e-como-tudo-comecou>. [Accessed: 11-Dec-2021].
- [2] I. Ferreira, J. A. Cabral, P. Saraiva, and M. C. Oliveira, “A multidisciplinary framework to support the design of injection mold tools,” *Struct. Multidiscip. Optim.*, vol. 49, no. 3, pp. 501–521, 2014.
- [3] S. H. Lee, “Feature-based non-manifold modeling system to integrate design and analysis of injection molding products,” *J. Mech. Sci. Technol.*, vol. 23, no. 5, pp. 1331–1341, May 2009.
- [4] I. Ferreira, O. De Weck, P. Saraiva, and J. Cabral, “Multidisciplinary optimization of injection molding systems,” *Struct. Multidiscip. Optim.*, vol. 41, no. 4, pp. 621–635, 2010.
- [5] C. K. Mok, K. S. Chin, and J. K. L. Ho, “An Interactive Knowledge-Based CAD System For Mould Design in Injection Moulding Processes,” *Int. J. Adv. Manuf. Technol.*, vol. 17, no. 1, pp. 27–38, Jan. 2001.
- [6] W. HE, Y. F. ZHANG, K. S. LEE, J. Y. H. FUH, and A. Y. C. NEE, “Automated process parameter resetting for injection moulding: a fuzzy-neuro approach,” *J. Intell. Manuf.*, vol. 9, no. 1, pp. 17–27, 1998.
- [7] I. Matin, M. Hadzistevic, J. Hodolic, D. Vukelic, and D. Lukic, “A CAD/CAE-integrated injection mold design system for plastic products,” *Int. J. Adv. Manuf. Technol.*, vol. 63, no. 5–8, pp. 595–607, Nov. 2012.
- [8] R. ALEXANDRA, “Cad/Cam/Cae Technology in Injection Mold Design,” *Constanta Marit. Univ. Anu.*, vol. 16, pp. 173–176, 2011.
- [9] Y.-M. Deng, Y. C. Lam, S. B. Tor, and G. A. Britton, “A CAD-CAE Integrated Injection Molding Design System,” *Eng. Comput.*, vol. 18, no. 1, pp. 80–92, Apr. 2002.
- [10] M. L. H. Low and K. S. Lee, “Application of standardization for initial design of plastic injection moulds,” *Int. J. Prod. Res.*, vol. 41, no. 10, pp. 2301–2324, Jan. 2003.
- [11] Z. Lou, H. Jiang, and X. Ruan, “Development of an integrated knowledge-based system for mold-base design,” *J. Mater. Process. Technol.*, vol. 150, no. 1–2, pp. 194–199, Jul. 2004.
- [12] W.-R. Jong, C.-H. Wu, H. H. Liu, and M.-Y. Li, “A collaborative navigation system for concurrent mold design,” *Int. J. Adv. Manuf. Technol.*, vol. 40, no. 3–4, pp. 215–225, Jan. 2009.
- [13] H. Zhou, S. Shi, and B. Ma, “A virtual injection molding system based on numerical simulation,” *Int. J. Adv. Manuf. Technol.*, vol. 40, no. 3–4, pp. 297–306, Jan. 2009.
- [14] K.-S. Chin and T. N. Wong, “Knowledge-based evaluation for the conceptual design development

- of injection molding parts," *Eng. Appl. Artif. Intell.*, vol. 9, no. 4, pp. 359–376, Aug. 1996.
- [15] H.-S. Mok, C.-H. Kim, and C.-B. Kim, "Automation of mold designs with the reuse of standard parts," *Expert Syst. Appl.*, vol. 38, no. 10, pp. 12537–12547, Sep. 2011.
- [16] Y.-S. Ma, S. B. Tor, and G. A. Britton, "The development of a standard component library for plastic injection mould design using an object-oriented approach," *Int. J. Adv. Manuf. Technol.*, vol. 22, no. 9–10, pp. 611–618, Nov. 2003.
- [17] C. K. Mok, K. S. Chin, and H. Lan, "An Internet-based intelligent design system for injection moulds," *Robot. Comput. Integr. Manuf.*, vol. 24, no. 1, pp. 1–15, Feb. 2008.
- [18] J.-M. Huang, Y.-T. Jou, L.-C. Zhang, S.-T. Wang, and C.-X. Huang, "A web-based model for developing: A mold base design system," *Expert Syst. Appl.*, vol. 36, no. 4, pp. 8356–8367, May 2009.
- [19] A. Bakharev and D. Astbury, "Simulation of ejection of moldings using combination of mold filling and thermo-mechanical analyses," in *The 11th International Conference on Numerical Methods in Industrial Forming Processes*, 2013, pp. 555–560.
- [20] A. J. Giacomini *et al.*, "Core deflection in injection molding," *J. Nonnewton. Fluid Mech.*, vol. 166, no. 16, pp. 908–914, 2011.
- [21] Y.-C. Chen, Y.-J. Liao, S.-C. Tseng, and A. J. Giacomini, "Core Deflection in Plastics Injection Molding: Direct Measurement, Flow Visualization and 3D Simulation," *Polym. Plast. Technol. Eng.*, vol. 50, no. 9, pp. 863–872, Jun. 2011.
- [22] "Auto Bumper Moulds, automobile mould maker, automotive mould maker, bumper mold, instrument panel mould, plastic injection mould maker| Products." [Online]. Available: http://www.jindianmold.com/english/automotive_bumper.htm. [Accessed: 03-Jul-2017].
- [23] Y.-M. Deng, G. A. Britton, Y. C. Lam, S. B. Tor, and Y. S. Ma, "Feature-based CAD-CAE integration model for injection-moulded product design," *Int. J. Prod. Res.*, vol. 40, no. 15, pp. 3737–3750, Jan. 2002.
- [24] B. Carpenter, S. Patil, R. Hoffman, B. Lilly, and J. Castro, "Effect of machine compliance on mold deflection during injection and packing of thermoplastic parts," *Polym. Eng. Sci.*, vol. 46, no. 7, pp. 844–852, Jul. 2006.
- [25] C. H. Wu and Y. J. Huang, "The influence of cavity deformation on the shrinkage and warpage of an injection-molded part," *Int. J. Adv. Manuf. Technol.*, vol. 32, no. 11–12, pp. 1144–1154, May 2007.
- [26] B. Nardin, K. Kuzman, and Z. Kampus, "Injection moulding simulation results as an input to the injection moulding process," *J. Mater. Process. Technol.*, vol. 130–131, pp. 310–314, Dec.

- 2002.
- [27] Y.-M. Deng, G. A. Britton, and Y. C. Lam, "Specification Model: A First Step Towards Automatic Interpretation of Injection Moulding CAE Results," *Int. J. Adv. Manuf. Technol.*, vol. 20, no. 11, pp. 833–843, Nov. 2002.
 - [28] K. Nijenhuis *et al.*, "Non-Newtonian Flows," in *Springer Handbook of Experimental Fluid Mechanics*, Berlin, Heidelberg: Springer Berlin Heidelberg, 2007, pp. 619–743.
 - [29] K. Sethuraman and K. Palanisamy, "Material Characterization of Injection Molded Thermo Plastic Materials for Finite element analysis - Practical Challenges," *Int. Conf. Smart Struct. Syst.*, pp. 46–52, 2013.
 - [30] G. Gou, P. Xie, W. Yang, and Y. Ding, "Online measurement of rheological properties of polypropylene based on an injection molding machine to simulate the injection-molding process," *Polym. Test.*, vol. 30, no. 8, pp. 826–832, 2011.
 - [31] A. I. Isayev and C. A. Hieber, "Toward a viscoelastic modelling of the injection molding of polymers," *Rheol. Acta*, vol. 19, no. 2, pp. 168–182, Mar. 1980.
 - [32] W. M. Chan, L. Yan, W. Xiang, and B. T. Cheok, "A 3D CAD knowledge-based assisted injection mould design system," *Int. J. Adv. Manuf. Technol.*, vol. 22, no. 5–6, pp. 387–395, Oct. 2003.
 - [33] V. Leo and C. Cuvellez, "The effect of the packing parameters, gate geometry, and mold elasticity on the final dimensions of a molded part," *Polym. Eng. Sci.*, vol. 36, no. 15, pp. 1961–1971, Aug. 1996.
 - [34] M. Lakkanna, G. C. Mohan Kumar, and R. Kadoli, "Computational design of mould sprue for injection moulding thermoplastics," *J. Comput. Des. Eng.*, vol. 3, no. 1, pp. 37–52, 2016.
 - [35] A. M. Grillet, A. C. B. Bogaerds, M. Bulters, G. W. M. Peters, and F. P. T. Baaijens, "An Investigation of Flow Mark Surface Defects in Injection Molding of Polymer Melts," *Mate Poster*, 2000.
 - [36] R. Spina, "Injection moulding of automotive components: comparison between hot runner systems for a case study," *J. Mater. Process. Technol.*, vol. 155–156, pp. 1497–1504, Nov. 2004.
 - [37] H. Hassan, "An experimental work on the effect of injection molding parameters on the cavity pressure and product weight," *Int. J. Adv. Manuf. Technol.*, vol. 67, no. 1–4, pp. 675–686, 2013.
 - [38] "hot runner system, hot runner systems produced by Yichuan in china." [Online]. Available: http://www.capmold.com/hot_runner_system.htm. [Accessed: 11-Jun-2017].
 - [39] E. Bociaga and P. Palutkiewicz, "The influence of injection molding parameters and blowing agent addition on selected properties, surface state, and structure of HDPE parts," *Polym. Eng. Sci.*,

- vol. 53, no. 4, pp. 780–791, Apr. 2013.
- [40] X. Yuwen, L. Chen, and Y. Han, “Numerical Simulation of Casting Filling Process Based on FLUENT,” *Energy Procedia*, vol. 17, pp. 1864–1871, 2012.
 - [41] S. Shin and W. Il Lee, “Finite element analysis of incompressible viscous flow with moving free surface by selective volume of fluid method,” *Int. J. Heat Fluid Flow*, vol. 21, no. 2, pp. 197–206, 2000.
 - [42] “Overview | Moldflow Insight | Autodesk Knowledge Network.” [Online]. Available: <https://knowledge.autodesk.com/support/moldflow-insight?sort=score>. [Accessed: 11-Jun-2017].
 - [43] S. J. Liu, C. H. Lin, and Y. C. Wu, “Minimizing the sinkmarks in injection-molded thermoplastics,” *Adv. Polym. Technol.*, vol. 20, no. 3, pp. 202–215, 2001.
 - [44] S. H. Tang, Y. M. Kong, S. M. Sapuan, R. Samin, and S. Sulaiman, “Design and thermal analysis of plastic injection mould,” *J. Mater. Process. Technol.*, vol. 171, no. 2, pp. 259–267, 2006.
 - [45] M. S. Rusdi *et al.*, “Numerical Investigation on the Effect of Pressure and Temperature on the Melt Filling During Injection Molding Process,” *Arab. J. Sci. Eng.*, vol. 41, no. 5, pp. 1907–1919, May 2016.
 - [46] J. Aisa, C. Javierre, and J. A. De la Serna, “An example of simulation tools use for large injection moulds design: The CONTENUR™ 2400 I solid waste container,” *J. Mater. Process. Technol.*, vol. 175, no. 1–3, pp. 15–19, 2006.
 - [47] H. Oktem, T. Erzurumlu, and I. Uzman, “Application of Taguchi optimization technique in determining plastic injection molding process parameters for a thin-shell part,” *Mater. Des.*, vol. 28, no. 4, pp. 1271–1278, 2007.
 - [48] C. P. Chen, M. T. Chuang, Y. H. Hsiao, Y. K. Yang, and C. H. Tsai, “Simulation and experimental study in determining injection molding process parameters for thin-shell plastic parts via design of experiments analysis,” *Expert Syst. Appl.*, vol. 36, no. 7, pp. 10752–10759, 2009.
 - [49] Y. Gao and X. Wang, “An effective warpage optimization method in injection molding based on the Kriging model,” *Int. J. Adv. Manuf. Technol.*, vol. 37, no. 9–10, pp. 953–960, Jun. 2008.
 - [50] Y. C. Lam, G. A. Britton, and D. S. Liu, “Optimisation of gate location with design constraints,” *Int. J. Adv. Manuf. Technol.*, vol. 24, no. 7–8, pp. 560–566, Oct. 2004.
 - [51] C. Shen and M. Zhai, “Algorithm to optimise number of gates and locations in an injection mould,” *Plast. Rubber Compos.*, vol. 33, no. 8, pp. 330–334, Oct. 2004.
 - [52] M. Zhai, Y. C. Lam*, C. K. Au, and D. S. Liu, “Automated Selection of Gate Location for Plastic Injection Molding Processing,” *Polym. Plast. Technol. Eng.*, vol. 44, no. 2, pp. 229–242, Feb.

- 2005.
- [53] M. Zhai and Y. Xie, "A study of gate location optimization of plastic injection molding using sequential linear programming," *Int. J. Adv. Manuf. Technol.*, vol. 49, no. 1–4, pp. 97–103, Jul. 2010.
 - [54] S. L. Mok, C. K. Kwong, and W. S. Lau, "An intelligent hybrid system for initial process parameter setting of injection moulding," *Int. J. Prod. Res.*, vol. 38, no. 17, pp. 4565–4576, Nov. 2000.
 - [55] Y.-M. Deng, D. Zheng, and X.-J. Lu, "Injection moulding optimisation of multi-class design variables using a PSO algorithm," *Int. J. Adv. Manuf. Technol.*, vol. 39, no. 7–8, pp. 690–698, Nov. 2008.
 - [56] Y. C. Lam, G. A. Britton, and Y.-M. Deng, "A computer-aided system for an optimal moulding conditions design using a simulation-based approach," *Int. J. Adv. Manuf. Technol.*, vol. 22, no. 7–8, pp. 574–586, Nov. 2003.
 - [57] K. S. Lee and J. C. Lin, "Design of the runner and gating system parameters for a multi-cavity injection mould using FEM and neural network," *Int. J. Adv. Manuf. Technol.*, vol. 27, no. 11–12, pp. 1089–1096, Feb. 2006.
 - [58] Y. C. Lam, L. Y. Zhai, K. Tai, and S. C. Fok, "An evolutionary approach for cooling system optimization in plastic injection moulding," *Int. J. Prod. Res.*, vol. 42, no. 10, pp. 2047–2061, May 2004.
 - [59] H. Zhou, Y. Zhang, J. Wen, and D. Li, "An acceleration method for the BEM-based cooling simulation of injection molding," *Eng. Anal. Bound. Elem.*, vol. 33, no. 8–9, pp. 1022–1030, Aug. 2009.
 - [60] L. Q. TANG, K. POCHIRAJU, C. CHASSAPIS, and S. MANOOCHEHRI, "THREE-DIMENSIONAL TRANSIENT MOLD COOLING ANALYSIS BASED ON GALERKIN FINITE ELEMENT FORMULATION WITH A MATRIX-FREE CONJUGATE GRADIENT TECHNIQUE," *Int. J. Numer. Methods Eng.*, vol. 39, no. 18, pp. 3049–3064, Sep. 1996.
 - [61] C. L. Li, C. G. Li, and A. C. K. Mok, "Automatic layout design of plastic injection mould cooling system," *CAD Comput. Aided Des.*, vol. 37, no. 7, pp. 645–662, 2005.
 - [62] C. G. Li and C. L. Li, "Plastic injection mould cooling system design by the configuration space method," *CAD Comput. Aided Des.*, vol. 40, no. 3, pp. 334–349, 2008.
 - [63] H. Qiao, "A systematic computer-aided approach to cooling system optimal design in plastic injection molding," *Int. J. Mech. Sci.*, vol. 48, no. 4, pp. 430–439, Apr. 2006.
 - [64] Y. F. Sun, K. S. Lee, and A. Y. C. Nee, "Design and FEM analysis of the milled groove insert method for cooling of plastic injection moulds," *Int. J. Adv. Manuf. Technol.*, vol. 24, no. 9–10,

- pp. 715–726, Nov. 2004.
- [65] A. L. Kelly, L. Mulvaney-Johnson, R. Beechey, and P. D. Coates, “The effect of copper alloy mold tooling on the performance of the injection molding process,” *Polym. Eng. Sci.*, vol. 51, no. 9, pp. 1837–1847, Sep. 2011.
 - [66] J. G. Kovács *et al.*, “Thermal simulations and measurements for rapid tool inserts in injection molding applications,” *Appl. Therm. Eng.*, vol. 85, 2015.
 - [67] A. B. M. Saifullah, S. H. Masood, and I. Sbarski, “Thermal–structural analysis of bi-metallic conformal cooling for injection moulds,” *Int. J. Adv. Manuf. Technol.*, vol. 62, no. 1–4, pp. 123–133, Sep. 2012.
 - [68] X. Xu, E. Sachs, and S. Allen, “The design of conformal cooling channels in injection molding tooling,” *Polym. Eng. Sci.*, vol. 41, no. 7, pp. 1265–1279, Jul. 2001.
 - [69] S. A. Jahan and H. El-Mounayri, “Optimal Conformal Cooling Channels in 3D Printed Dies for Plastic Injection Molding,” *Procedia Manuf.*, vol. 5, pp. 888–900, 2016.
 - [70] D. Modi, P. Simáček, and S. Advani, “Influence of injection gate definition on the flow-front approximation in numerical simulations of mold-filling processes,” *Control*, vol. 1248, no. October 2002, pp. 1237–1248, 2003.
 - [71] X. Zhuang, J. Ouyang, W. Zhou, Q. Liu, and B. Yang, “Numerical simulation of flow-induced stresses in mold filling process using a three-dimensional two-phase flow model,” *Arch. Appl. Mech.*, vol. 86, no. 7, pp. 1179–1201, Jul. 2016.
 - [72] K. . Pillai, C. . Tucker, and F. . Phelan, “Numerical simulation of injection/compression liquid composite molding. Part 1. Mesh generation,” *Compos. Part A Appl. Sci. Manuf.*, vol. 31, no. 1, pp. 87–94, Jan. 2000.
 - [73] K. M. Pillai, C. L. Tucker, and F. R. Phelan, “Numerical simulation of injection/compression liquid composite molding. Part 2: preform compression,” *Compos. Part A Appl. Sci. Manuf.*, vol. 32, no. 2, pp. 207–220, Feb. 2001.
 - [74] K. Han, J. Ni, J. Toth, L. J. Lee, and J. P. Greene, “Analysis of an injection/compression liquid composite molding process,” *Polym. Compos.*, vol. 19, no. 4, pp. 487–496, Aug. 1998.
 - [75] Y. C. Lam, S. C. Joshi, and X. L. Liu, “Numerical simulation of the mould-filling process in resin-transfer moulding,” *Compos. Sci. Technol.*, vol. 60, no. 6, pp. 845–855, May 2000.
 - [76] V. R. Voller and S. Peng, “An algorithm for analysis of polymer filling of molds,” *Polym. Eng. Sci.*, vol. 35, no. 22, pp. 1758–1765, Nov. 1995.
 - [77] Y.-E. Yoo and W. Il Lee, “Numerical simulation of the resin transfer mold filling process using the boundary element method,” *Polym. Compos.*, vol. 17, no. 3, pp. 368–374, Jun. 1996.

- [78] P. Simacek and S. G. Advani, "A numerical model to predict fiber tow saturation during liquid composite molding," *Compos. Sci. Technol.*, vol. 63, no. 12, pp. 1725–1736, Sep. 2003.
- [79] H. Tan and K. M. Pillai, "Multiscale modeling of unsaturated flow in dual-scale fiber preforms of liquid composite molding I: Isothermal flows," *Compos. Part A Appl. Sci. Manuf.*, vol. 43, no. 1, pp. 1–13, Jan. 2012.
- [80] A. I. Leonov, "Nonequilibrium thermodynamics and rheology of viscoelastic polymer media," *Rheol. Acta*, vol. 15, no. 2, pp. 85–98, Feb. 1976.
- [81] J. M. O'Brien and W. F. Hosford, "Spheroidization cycles for medium carbon steels," *Metall. Mater. Trans. A*, vol. 33, no. April, pp. 1255–1261, 2002.
- [82] J.-B. Kim, W.-S. Seo, and K. Park, "Damage prediction in the multistep forging process of subminiature screws," *Int. J. Precis. Eng. Manuf.*, vol. 13, no. 9, pp. 1619–1624, Sep. 2012.
- [83] C. Cao, D. Zhang, Z. He, L. Zhuang, and J. Zhang, "Enhanced and accelerated age hardening response of Al-5.2Mg-0.45Cu (wt%) alloy with Zn addition," *Mater. Sci. Eng. A*, vol. 666, pp. 34–42, 2016.
- [84] J. Brezinová and A. Guzanová, *THE WEAR OF INJECTION MOULD FUNCTIONAL PARTS IN CONTACT WITH POLYMER COMPOSITES*, vol. 16, no. 1. Strojnická fakulta Žilinskej univerzity, 2009.
- [85] G. Lindwall, J. Flyg, K. Frisk, and O. Sandberg, "Experimental and theoretical investigations of hot isostatically pressed-produced stainless steel/high alloy tool steel compound materials," *Metall. Mater. Trans. A Phys. Metall. Mater. Sci.*, vol. 42, no. 5, pp. 1165–1172, 2011.
- [86] T. A. Polyanskaya, "Wear of alloy tool steels under impact conditions," *Met. Sci. Heat Treat.*, vol. 15, no. 1, pp. 85–86, Jan. 1973.
- [87] J. Schneider, A. Kienzler, and V. Schulze, "Influence of surface condition on wear behavior of μ PIM mould inserts made of tool steel and cemented carbide," *Microsyst. Technol.*, vol. 44, pp. 1–8, 2011.
- [88] J. Schneider *et al.*, "Mechanical structuring, surface treatment and tribological characterization of steel mould inserts for micro powder injection moulding," *Microsyst. Technol.*, vol. 14, no. 12, pp. 1797–1803, 2008.
- [89] K. Fukaura, Y. Yokoyama, D. Yokoi, N. Tsujii, and K. Ono, "Fatigue of cold-work tool steels: Effect of heat treatment and carbide morphology on fatigue crack formation, life, and fracture surface observations," *Metall. Mater. Trans. A*, vol. 35, no. 4, pp. 1289–1300, Apr. 2004.
- [90] V. A. Krokha, "Relationship between yield stress in compression and mechanical properties of steel, copper, aluminum and their alloys," *Strength Mater.*, vol. 4, no. 6, pp. 708–715, Jun.

1972.

- [91] B. Karp, A. Dorogoy, and D. Rittel, "A Shear Compression Disk Specimen with Controlled Stress Triaxiality Under Dynamic Loading," *Exp. Mech.*, vol. 53, no. 2, pp. 243–253, 2013.
- [92] J. P. Fuertes, R. Luri, C. J. Luis, D. Salcedo, J. León, and I. Puertas, "Comparative Study of the Damage Attained with Different Specimens by FEM," *Procedia Eng.*, vol. 132, pp. 319–325, 2015.
- [93] K. Toda, H. Gondoh, H. Takechi, and M. Usuda, "Absorbed energy during compression and crushing of high strength steel sheets," *Metall. Trans. A*, vol. 7, no. 11, pp. 1619–1624, Nov. 1976.
- [94] H. Liu *et al.*, "Hot deformation behavior of low carbon steel during compression at elevated temperature," *J. Wuhan Univ. Technol. Sci. Ed.*, vol. 29, no. 3, pp. 601–605, 2014.
- [95] A. Dorogoy and D. Rittel, "Numerical Validation of the Shear Compression Specimen. Part I: Quasi-static Large Strain Testing," *Exp. Mech.*, vol. 45, no. 2, pp. 167–177, 2005.
- [96] O. Vosikovskiy, "The effect of stress ratio on fatigue crack growth rates in steels," *Eng. Fract. Mech.*, vol. 11, no. 3, pp. 595–602, Jan. 1979.
- [97] Y. MURAKAMI and T. ENDO, "Effects of small defects on fatigue strength of metals," *Int. J. Fatigue*, vol. 2, no. 1, pp. 23–30, Jan. 1980.
- [98] M. Knoerr, K. Lange, and T. Altan, "Fatigue failure of cold forging tooling: causes and possible solutions through fatigue analysis," *J. Mater. Process. Technol.*, vol. 46, no. 1–2, pp. 57–71, Oct. 1994.
- [99] N. M. A. Eid and P. F. Thomason, "The nucleation of fatigue cracks in a low-alloy steel under high-cycle fatigue conditions and uniaxial loading," *Acta Metall.*, vol. 27, no. 7, pp. 1239–1249, Jul. 1979.
- [100] X. Zheng, "On some basic problems of fatigue research in engineering," *Int. J. Fatigue*, vol. 23, no. 9, pp. 751–766, 2001.
- [101] W. Cui, "A state-of-the-art review on fatigue life prediction methods for metal structures," *J. Mar. Sci. Technol.*, vol. 7, no. 1, pp. 43–56, 2002.
- [102] J. COLIN and A. FATEMI, "Variable amplitude cyclic deformation and fatigue behaviour of stainless steel 304L including step, periodic, and random loadings," *Fatigue Fract. Eng. Mater. Struct.*, vol. 33, no. 4, pp. 205–220, Apr. 2010.
- [103] M. V. Borodii and S. M. Shukaev, "Additional cyclic strain hardening and its relation to material structure, mechanical characteristics, and lifetime," *Int. J. Fatigue*, vol. 29, no. 6, pp. 1184–1191, Jun. 2007.

- [104] D. McClafin and A. Fatemi, "Torsional deformation and fatigue of hardened steel including mean stress and stress gradient effects," *Int. J. Fatigue*, vol. 26, no. 7, pp. 773–784, Jul. 2004.
- [105] a Lynn and D. Duquesnay, "Computer simulation of variable amplitude fatigue crack initiation behaviour using a new strain-based cumulative damage model," *Int. J. Fatigue*, vol. 24, pp. 977–986, 2002.
- [106] M. Jono, "Fatigue damage and crack growth under variable amplitude loading with reference to the counting methods of stress-strain ranges," *Int. J. Fatigue*, vol. 27, no. 8, pp. 1006–1015, 2005.
- [107] S. S. Manson and G. R. Halford, "Practical implementation of the double linear damage rule and damage curve approach for treating cumulative fatigue damage," *Int. J. Fract.*, vol. 17, no. 2, pp. 169–192, Apr. 1981.
- [108] M. Zappalorto and D. Kujawski, "Neuber's rules and other solutions: Theoretical differences, formal analogies and energy interpretations," *Theor. Appl. Fract. Mech.*, vol. 79, pp. 2–13, Oct. 2015.
- [109] D. Kujawski and J. L. K. Teo, "A Generalization of Neuber's Rule for Numerical Applications," *Procedia Struct. Integr.*, vol. 5, pp. 883–888, Jan. 2017.
- [110] A. Fatemi and L. Yang, "Cumulative fatigue damage and life prediction theories: a survey of the state of the art for homogeneous materials," *Int. J. Fatigue*, vol. 20, no. 1, pp. 9–34, Jan. 1998.
- [111] S. Kilambi and S. Tipton, "Numerical evaluation of the original 'Neuber's Rule' for pure out-of-plane shear loading," *J. Strain Anal. Eng. Des.*, vol. Numerical, pp. 522–535, 2013.
- [112] J. E. Shigley, C. R. Mischke, R. G. Budynas, J. B. Aguiar, and J. M. Aguiar, *Projeto de engenharia mecânica*. Bookman, 2005.
- [113] M. Kuroda, "Extremely low cycle fatigue life prediction based on a new cumulative fatigue damage model," *Int. J. Fatigue*, vol. 24, no. 6, pp. 699–703, 2002.
- [114] J. Polák, R. Petráš, G. Chai, and V. Škorík, "Surface profile evolution and fatigue crack initiation in Sanicro 25 steel at room temperature," *Mater. Sci. Eng. A*, vol. 658, pp. 221–228, 2016.
- [115] X. Chen, Q. Gao, and X.-F. Sun, "LOW-CYCLE FATIGUE UNDER NON-PROPORTIONAL LOADING," *Fatigue Fract. Eng. Mater. Struct.*, vol. 19, no. 7, pp. 839–854, Jul. 1996.
- [116] T. Itoh and T. Yang, "Material dependence of multiaxial low cycle fatigue lives under non-proportional loading," *Int. J. Fatigue*, vol. 33, no. 8, pp. 1025–1031, Aug. 2011.
- [117] T. Itoh, M. Sakane, T. Hata, and N. Hamada, "A design procedure for assessing low cycle fatigue life under proportional and non-proportional loading," *Int. J. Fatigue*, vol. 28, no. 5–6, pp. 459–466, May 2006.

- [118] S. A. McKelvey, Y.-L. Lee, and M. E. Barkey, "Stress-Based Uniaxial Fatigue Analysis Using Methods Described in FKM-Guideline," *J. Fail. Anal. Prev.*, vol. 12, no. 5, pp. 445–484, Oct. 2012.
- [119] J. Li, Z. Yao, and Z. Zhang, "Multiaxial Fatigue Life Prediction for Steels Based on Some Simple Approximations," *J. Mater. Eng. Perform.*, vol. 24, no. 1, pp. 185–193, Jan. 2015.
- [120] I. V. Papadopoulos, P. Davoli, C. Gorla, M. Filippini, and A. Bernasconi, "A comparative study of multiaxial high-cycle fatigue criteria for metals," *Int. J. Fatigue*, vol. 19, no. 3, pp. 219–235, 1997.
- [121] P. Luo, W. Yao, L. Susmel, Y. Wang, and X. Ma, "A survey on multiaxial fatigue damage parameters under non-proportional loadings," *Fatigue Fract. Eng. Mater. Struct.*, vol. 40, 2017.
- [122] A. Cruces, P. Lopez-Crespo, S. Bressan, T. Itoh, and B. Moreno, "On the Behaviour of 316 and 304 Stainless Steel under Multiaxial Fatigue Loading: Application of the Critical Plane Approach," *Metals (Basel)*, vol. 9, p. 978, 2019.
- [123] C. M. Sonsino, "Multiaxial fatigue of welded joints under in-phase and out-of-phase local strains and stresses," *Int. J. Fatigue*, vol. 17, no. 1, pp. 55–70, Jan. 1995.
- [124] Y.-L. Lee, T. Tjhung, and A. Jordan, "A life prediction model for welded joints under multiaxial variable amplitude loading histories," *Int. J. Fatigue*, vol. 29, no. 6, pp. 1162–1173, Jun. 2007.
- [125] D. Pellinghelli, M. Riboli, and A. Spagnoli, "FULL-MODEL MULTIAXIAL FATIGUE LIFE CALCULATIONS WITH DIFFERENT CRITERIA," *Procedia Eng.*, vol. 213, pp. 126–136, Jan. 2018.
- [126] A. R. Khoei and M. Eghbalian, "Numerical simulation of cyclic behavior of ductile metals with a coupled damage-viscoplasticity model," *Comput. Mater. Sci.*, vol. 55, pp. 376–389, 2012.
- [127] K. A. Vansovich, I. P. Aistov, and V. I. Yadrov, "The Fatigue Surface Cracks Growth Model Taking into Account Plastic Deformations for a Cylinder of a Low-speed Long-stroke Compressor," *Procedia Eng.*, vol. 152, pp. 240–246, 2016.
- [128] J. C. Newman and J. C. Newman, "Validation of the Two-Parameter Fracture Criterion using finite-element analyses with the critical CTOA fracture criterion," *Eng. Fract. Mech.*, vol. 136, pp. 131–141, 2015.
- [129] J. Polák and J. Man, "Experimental evidence and physical models of fatigue crack initiation," *Int. J. Fatigue*, vol. 91, pp. 294–303, 2016.
- [130] P. Lopez-Crespo, B. Moreno, A. Lopez-Moreno, and J. Zapatero, "Study of crack orientation and fatigue life prediction in biaxial fatigue with critical plane models," *Eng. Fract. Mech.*, vol. 136, pp. 115–130, 2014.
- [131] R. Petráš, V. Škorík, and J. Polák, "Thermomechanical fatigue and damage mechanisms in Sanicro 25 steel," *Mater. Sci. Eng. A*, vol. 650, pp. 52–62, 2016.

- [132] M. Akita, Y. Uematsu, T. Kakiuchi, M. Nakajima, and R. Kawaguchi, "Defect-dominated fatigue behavior in type 630 stainless steel fabricated by selective laser melting," *Mater. Sci. Eng. A*, vol. 666, pp. 19–26, 2016.
- [133] M. R. Stoudt, L. E. Levine, and L. Ma, "Designing a Uniaxial Tension/Compression Test for Springback Analysis in High-Strength Steel Sheets," *Exp. Mech.*, pp. 1–9, 2016.
- [134] R. Basan, D. Rubeša, M. Franulović, and T. Marohnić, "Some Considerations on the Evaluation of Methods for the Estimation of Fatigue Parameters from Monotonic Properties," *Procedia Eng.*, vol. 101, pp. 18–25, Jan. 2015.
- [135] J. Kohout and S. Věchet, "New function for fatigue curves characterization and its multiple merits," *Int. J. Fatigue*, 2001.
- [136] R. Basan, M. Franulovic, D. Rubeša, and I. Prebil, "Implementation of strain-life fatigue parameters estimation methods in a web-based system," *Procedia Eng.*, vol. 10, pp. 2363–2368, Jan. 2011.
- [137] K. S. Kim, X. Chen, C. Han, and H. W. Lee, "Estimation methods for fatigue properties of steels under axial and torsional loading," *Int. J. Fatigue*, vol. 24, no. 7, pp. 783–793, Jul. 2002.
- [138] M. A. Meggiolaro and J. T. P. Castro, "Statistical evaluation of strain-life fatigue crack initiation predictions," *Int. J. Fatigue*, vol. 26, no. 5, pp. 463–476, May 2004.
- [139] M. L. Roessle and A. Fatemi, "Strain-controlled fatigue properties of steels and some simple approximations," *Int. J. Fatigue*, vol. 22, no. 6, pp. 495–511, Jul. 2000.
- [140] W.-S. Jeon and J.-H. Song, "An expert system for estimation of fatigue properties of metallic materials," *Int. J. Fatigue*, vol. 24, no. 6, pp. 685–698, Jun. 2002.
- [141] X. Li, G. Zhao, Y. Guan, and H. Li, "Research on thermal stress, deformation, and fatigue lifetime of the rapid heating cycle injection mold," *Int. J. Adv. Manuf. Technol.*, vol. 45, no. 3–4, pp. 261–275, 2009.
- [142] J. K. Kim and C. S. Lee, "Fatigue life estimation of injection mold core using simulation-based approach," *Int. J. Automot. Technol.*, vol. 14, no. 5, pp. 723–729, Oct. 2013.
- [143] Z. Zhang and Y. Chen, "A constrained intrinsic cohesive finite element method with little stiffness reduction for fracture simulation," *Eng. Fract. Mech.*, vol. 136, pp. 213–225, 2014.
- [144] I. Eshraghi and N. Soltani, "Stress intensity factor calculation for internal circumferential cracks in functionally graded cylinders using the weight function approach," *Eng. Fract. Mech.*, vol. 134, pp. 1–19, 2015.
- [145] B. COATES and S. A. ARGYROPOULOS, "The Effects of Surface Roughness and Metal Temperature on the Heat-Transfer Coefficient at the Metal Mold Interface," *Metall. Mater. Trans.*

- B*, vol. 38, no. 2, pp. 243–255, May 2007.
- [146] D. Masato, M. Sorgato, P. Parenti, M. Annoni, and G. Lucchetta, "Impact of deep cores surface topography generated by micro milling on the demolding force in micro injection molding," *J. Mater. Process. Technol.*, vol. 246, pp. 211–223, Aug. 2017.
 - [147] S.-C. Chen, Y. Chang, Y.-P. Chang, Y.-C. Chen, and C.-Y. Tseng, "Effect of cavity surface coating on mold temperature variation and the quality of injection molded parts," *Int. Commun. Heat Mass Transf.*, vol. 36, no. 10, pp. 1030–1035, Dec. 2009.
 - [148] H. Javadi, W. Jomaa, D. Texier, M. Brochu, and P. Bocher, "Surface Roughness Effects on the Fatigue Behavior of As-Machined Inconel718," *Solid State Phenom.*, vol. 258, pp. 306–309, 2016.
 - [149] "RheoHeat - Nu - Pr - Re relationship." [Online]. Available: http://www.rheoheat.se/b10_heat.html. [Accessed: 20-Nov-2017].
 - [150] *Standard Terminology Relating to Fatigue and Fracture Testing*, ASTM E1823., vol. 03.01. West Conshohocken, PA: ASTM International, 2000.
 - [151] R. I. Stephens and H. O. Fuchs, *Metal fatigue in engineering*. Wiley, 2001.
 - [152] "Why Metal Fatigue Matters," in *Metal Fatigue*, Dordrecht: Springer Netherlands, pp. 161–165.
 - [153] C. A. G. de M. Branco, *Mecânica dos materiais*. Fundação Calouste Gulbenkian. Serviço de Educação e Bolsas, 2006.
 - [154] S. Bhat and R. Patibandla, "Metal Fatigue and Basic Theoretical Models: A Review," in *Alloy Steel - Properties and Use*, 2011.
 - [155] K. J. Miller, "Materials science perspective of metal fatigue resistance," <http://dx.doi.org/10.1179/mst.1993.9.6.453>, vol. 9, no. 6, pp. 453–462, 2013.
 - [156] J. Schijve, *Fatigue of Structures and Materials*. Dordrecht: Springer Netherlands, 2009.
 - [157] C. Bandara, S. Siriwardane, U. I. Dissanayake, and R. Dissanayake, "Developing a full range S–N curve and estimating cumulative fatigue damage of steel elements," *Comput. Mater. Sci.*, vol. 96, pp. 96–101, 2015.
 - [158] P. Strzelecki and T. Tomaszewski, "Analytical models of the S-N curve based on the hardness of the material," *Procedia Struct. Integr.*, vol. 5, pp. 832–839, Jan. 2017.
 - [159] P. Strzelecki and J. Sempruch, "MODYFICATION OF SELECTED METHODS OF RAPID DETERMINATION OF FATIGUE CHARAKTERISTICS IN THE RANGE OF LIMITED FATIGUE LIFE."
 - [160] C. S. Bandara, S. C. Siriwardane, U. I. Dissanayake, and R. Dissanayake, "Hardness-Based Non-destructive Method for Developing Location Specific S-N Curves for Fatigue Life Evaluation," *J. Civ. Eng. Archit.*, vol. 10, pp. 183–191, 2016.

- [161] S. S. Manson, "Fatigue: A complex subject—Some simple approximations," *Exp. Mech.*, vol. 5, no. 4, pp. 193–226, 1965.
- [162] S. R. Lampman and N. DiMateo, *ASM handbook: Fatigue and fracture*, no. vol. 18. ASM International, 1996.
- [163] S. R. Lampman, "ASM Handbook: Volume 19, Fatigue and Fracture," *ASM Int.*, vol. 19, no. 9, pp. 557–565, 1996.
- [164] T. V. Duggan and J. Byrne, *Fatigue as a Design Criterion*. Macmillan Education UK, 1977.
- [165] C. Lipson, Juvinall, Robert C., *Handbook of stress and strength*. N.Y.: Macmillan, 1963.
- [166] Y.-L. Lee, M. E. Barkey, and H.-T. Kang, *Metal fatigue analysis handbook : practical problem-solving techniques for computer-aided engineering*. Butterworth-Heinemann, 2011.
- [167] P. D. Pilkey Walter, *Petersons Stress Concentration Factors*, no. Lyman 1989. 2008.
- [168] A. Cichański, "Influence of mesh morphology near the notch on precision of SCF determination," *J. Polish CIMAC*, vol. 5, pp. 65–71, 2010.
- [169] Y. Hodhigere, J. Jha, A. Tewari, and S. Mishra, "Finite Element Analysis-Based Approach for Stress Concentration Factor Calculation," in *Lecture Notes in Mechanical Engineering*, 2018, pp. 1–6.
- [170] D. Gunwant, R. Kshetri, and K. Rawat, "Determination of Stress Concentration factor in Linearly Elastic Structures with Different Stress-Raisers Using FEM," *Int. J. Eng. Res. Appl.*, vol. 6, pp. 2248–9622, 2016.
- [171] D. Parle, A. Patil, and A. Ambulgekar, "Evaluation of Stress Concentration Factor using ANSYS," 2011.
- [172] F. Maschinenbau, *Analytical strength assessment of components in mechanical engineering : FKM-guideline*, 5., rev. Ed. Frankfurt am Main: VDMA Verl., 2003.
- [173] C. Bathias and A. (André) Pineau, *Fatigue of materials and structures : fundamentals*. ISTE, 2010.
- [174] J. A. Bannantine, J. J. Comer, and J. L. Handrock, *Fundamentals of Metal Fatigue Analysis*. Prentice Hall, 1990.
- [175] P. Heuler and W. Schütz, "Fatigue life prediction in the crack initiation and crack propagation phases," 1985.
- [176] W. Schütz, "The prediction of fatigue life in the crack initiation and propagation stages-a state of the art survey," *Eng. Fract. Mech.*, vol. 11, no. 2, pp. 405–421, Jan. 1979.
- [177] J. Schijve, *Fatigue of structures and materials*. Springer Netherlands, 2009.
- [178] R. Stelzer, B. Carlton, and S. Mazzoni, "COMPARISON OF CYCLE COUNTING METHODS FOR POTENTIAL LIQUEFACTION AND STRUCTURAL FATIGUE ASSESSMENT," 2020.

- [179] F. Ellyin, "Fatigue Damage, Crack Growth and Life Prediction," *Fatigue Damage, Crack Growth Life Predict.*, 1996.
- [180] K. Karpanan, *Critical Plane Search Method for Biaxial and Multiaxial Fatigue Analysis*. 2016.
- [181] W. N. Findley, "A Theory for the Effect of Mean Stress on Fatigue of Metals Under Combined Torsion and Axial Load or Bending," *J. Eng. Ind.*, vol. 81, no. 4, pp. 301–305, Nov. 1959.
- [182] F.C. Campbell, *Elements of Metallurgy and Engineering Alloys - ASM International*. 2008.
- [183] H. Özerkan, "Theoretical determination of the fatigue strength limit of electrical discharge machined (EDM) AISI 8740 steel," *IOP Conf. Ser. Mater. Sci. Eng.*, vol. 393, p. 12103, 2018.
- [184] F. Ghanem, H. Sidhom, C. Braham, and M. Fitzpatrick, "Effect of Near-Surface Residual Stress and Microstructure Modification From Machining on the Fatigue Endurance of a Tool Steel," *J. Mater. Eng. Perform.*, vol. 11, 2002.
- [185] F. Ghanem, B. F. Nabil, H. Sidhom, and C. Braham, "Effects of finishing processes on the fatigue life improvements of electro-machined surfaces of tool steel," *Int. J. Adv. Manuf. Technol.*, vol. 52, pp. 583–595, 2011.

Appendix A

General

Uddeholm's tool steel *UHB 11* is an easily machinable carbon steel characterized by

- Good machinability
- Good mechanical strength.

UHB 11 is primarily intended to be used in the as-delivered condition. It is only to be heat treated for special applications.

Typical analysis	C 0,46	Si 0,2	Mn 0,7
Standard specification	W.-Nr. 1.1730		
Delivery condition	As rolled. Hardness approx. 200 HB		
Colour code	White		

Applications

- Punch holders
- Die holders
- Guide plates
- Backing plates
- Jigs
- Fixtures
- Simple bending dies
- Simple structural components.

Properties

Approximate values at room temperature.
Hardness 200 HB.

Tensile strength, R _m	640 N/mm ² 65 kp/mm ²
Yield strength, R _{p0,2}	340 N/mm ² 35 kp/mm ²
Reduction of area, Z	40 %
Elongation, A ₅	20 %

Note: The given figures are typical values and shall not be considered as guaranteed values.

Heat Treatment

UHB 11 is intended to be used untreated, i.e. in the as-delivered condition.

For applications where the material must be hardened to a higher hardness, the following instructions should be followed.

SOFT ANNEALING

Temperature approx. 700°C (1290°F).
Protect the steel and heat through to 700°C (1290°F). Cool in furnace 25°C/h (78°F/h) to 600°C (1110°F) and subsequently freely in air.
Hardness after soft annealing approx. 170 HB.

NORMALIZING

Normalizing temperature 840–870°C (1545–1066°F). Cooling in air.

STRESS RELIEVING

Temperature approx. 650°C (1200°F). After rough machining with chip-cutting tools, stress relieving may be advisable to minimize distortion if the tool is to be hardened. Holding time: 2 h after the entire piece has attained a temperature of approx. 650°C (1200°F). Cooling in furnace to approx. 500°C (930°F), followed by cooling in air.

HARDENING

Preheating temperature: 650°C (1200°F).

Austenitizing temperature: 820–870°C (1510–1600°F).

Holding time: 30 min. Protect against decarburization.

QUENCHING MEDIA

- Water
- Oil.

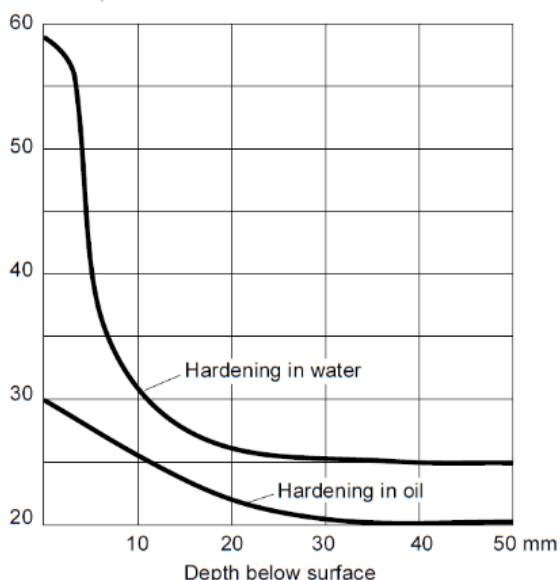


UHB 11 cross ground ready for delivery.

Hardness as a function of depth under surface

Dimension Ø 100 mm

Hardness, HRC



TEMPERING

Tempering temperature and hardness.

Quenching media	Hardness, HRC, after tempering Holdig time 1 h. Approx. value					
	200°C 390°F	250°C 480°F	300°C 570°F	400°C 750°F	500°C 930°F	600°C 1110°F
Water	58	56	53	45	37	27
Oil	31	30	29	27	25	20

Austenitizing temperature 830°C.

FLAME AND INDUCTION HARDENING

UHB 11 can be flame or induction hardened to a hardness of 57 ± 3 HRC. Water is normally used for quenching during continuous hardening. Temper immediately after hardening.

Cutting data recommendations

The cutting data below are to be considered as guiding values which must be adapted to existing local conditions. More detailed information can be found in Uddeholm "Cutting Data Recommendations".

TURNING

Cutting data parameters	Turning with carbide		Turning with high speed steel
	Rough turning	Fine turning	Fine turning
Cutting speed (v_c) m/min. f.p.m.	150–220 500–660	220–300 730–990	50 160
Feed (f) mm/r i.p.r.	0,3–0,6 0,012–0,023	–0,3 –0,012	–0,3 –0,012
Depth of cut (a_p) mm inch	2–6 0,08–0,20	–2 –0,08	–2 –0,08
Carbide designation ISO	P20–P30 Coated carbide	P10 Coated carbide or cement	–

DRILLING

High speed steel twist drills

Drill diameter Ø		Cutting speed (v_c)		Feed (f)	
mm	inch	m/min.	f.p.m.	mm/r	i.p.r.
–5	–3/16	25*	50*	0,08–0,20	0,003–0,008
5–10	3/16–3/8	25*	50*	0,20–0,30	0,008–0,012
10–15	3/8–5/8	25*	50*	0,30–0,35	0,012–0,014
15–20	5/8–3/4	25*	50*	0,35–0,40	0,014–0,016

* For coated HSS drill $v_c \sim 35$ m/min. (115 f.p.m.).

Carbide drills

Cutting data parameters	Type of drill		
	Indexable insert	Solid carbide	Brazed carbide ¹⁾
Cutting speed (v_c) m/min. f.p.m.	175–225 580–740	85 250	75 280
Feed (f) mm/r i.p.r.	0,05–0,25 ²⁾ 0,002–0,010 ²⁾	0,10–0,25 ²⁾ 0,004–0,010 ²⁾	0,15–0,25 ²⁾ 0,006–0,010 ²⁾

¹⁾ Drill with internal cooling channels and brazed carbide tip.

²⁾ Depending on drill diameter.

MILLING

Face and square shoulder face milling

Cutting data parameters	Milling with carbide		Milling with high speed steel
	Rough milling	Fine milling	Fine milling
Cutting speed (v_c) m/min. f.p.m.	160–200 530–660	200–300 660–990	35 115
Feed (f_z) mm/tooth inch/tooth	0,2–0,4 0,008–0,016	0,1–0,2 0,004–0,008	0,1 0,004
Depth of cut (a_p) mm inch	2–5 0,08–0,2	–2 –0,08	–2 –0,08
Carbide designation ISO	P20–P40 Coated carbide	P10–P20 Coated carbide	–

End milling

Cutting data parameters	Type of milling		
	Solid carbide	Carbide indexable insert	High speed steel
Cutting speed (v_c) m/min. f.p.m.	75 248	140–190 460–630	40 ¹⁾ 130
Feed (f_z) mm/tooth inch/tooth	0,03–0,20 ²⁾ 0,001–0,008 ²⁾	0,08–0,20 ²⁾ 0,003–0,008 ²⁾	0,05–0,35 ²⁾ 0,002–0,014 ²⁾
Carbide designation ISO	K10	P10–P20	–

¹⁾ For coated HSS end mill $v_c \approx 50$ m/min. (165 f.p.m.).

²⁾ Depending on radial depth of cut and cutter diameter.

GRINDING

General grinding wheel recommendation is given below. More information can be found in the Uddeholm publication "Grinding of Tool Steel".

Grinding operation	Soft annealed condition	Hardened condition
Face grinding straight wheel	A 46 HV	A 46 GV
Face grinding segment	A 24 GV	A 36 GV
Cylindrical grinding	A 46 LV	A 60 JV
Internal grinding	A 46 JV	A 60 IV
Profile grinding	A 100 LJ	A 120 JV

Welding

As is the case with most tool steels, the welding of UHB 11 is associated with a risk of cracking. In order to minimize the risk, welding should be carried out with preheating at 100–350°C (210–660°F).

Electrode: Unalloyed basic electrode for welding of unalloyed structural steel.

Note: Always use well dried basic electrodes. Welding can also be performed with an austenitic stainless electrode. The demand of preheating can be reduced but the filler metal does not reach the same hardness level as the base material when hardening.

Further information

Please contact your local Uddeholm office for further information on the selection, heat treatment, application and availability of Uddeholm tool steels.

EN 1.2311

General

Uddeholm Impax Supreme is a premium-quality vacuum-degassed Cr-Ni-Mo-alloyed steel which is supplied in the hardened and tempered condition, offering the following benefits:

- No hardening risks
- No hardening costs
- Time saving, e.g. no waiting for heat treatment
- Lower tool cost (e.g. no distortion to rectify)
- Modifications easily carried out
- Can be subsequently nitrided to increase surface wear resistance or locally flame hardened to reduce surface damage

Uddeholm Impax Supreme is manufactured to consistently high quality standards with a very low sulphur content, giving a steel with the following characteristics:

- Good polishing and photo-etching properties
- Good machinability
- High purity and good homogeneity
- Uniform hardness

Note: Uddeholm Impax Supreme is 100% ultrasonic tested.

Heavier sections are supplied premachined which offers the following advantages compared with un-machined material:

- Saving of weight
- Non-decarburized surface
- Exact nominal size (plus tolerance)
- Less machining
- Absence of scale minimizes machine and tool wear

Approx. analysis %	C	Si	Mn	Cr	Ni	Mo
	0.37	0.3	1.4	2.0	1.0	0.2
Standard spec.	AISI P20 modified					
Delivery condition	Hardened and tempered to 290–330 HB					
Colour code	Yellow/green					

Applications

- Injection moulds for thermoplastics
- Extrusion dies for thermoplastics
- Blow moulds
- Forming tools, press-brake dies (possibly flame hardened or nitrided)
- Aluminium die casting prototype dies
- Structural components, shafts

Properties

Physical data

Hardened and tempered to 310 HB.

Temperature	20°C (68°F)	200°C (390°F)
Density, kg/m ³ lbs/in ³	7 800 0.282	7 750 0.280
Coefficient of thermal expansion per °C from 20° per °F from 68°F	— —	12.7 × 10 ⁻⁶ 7.0 × 10 ⁻⁶
Thermal conductivity W/m °C Btu in/ft ² h °F	—	28 194
Modulus of elasticity N/mm ² tsi psi	205 000 13 280 29.7 × 10 ⁶	200 000 12 960 29.0 × 10 ⁶
Specific heat capacity J/kg °C Btu/lb °F	460 0.110	— —

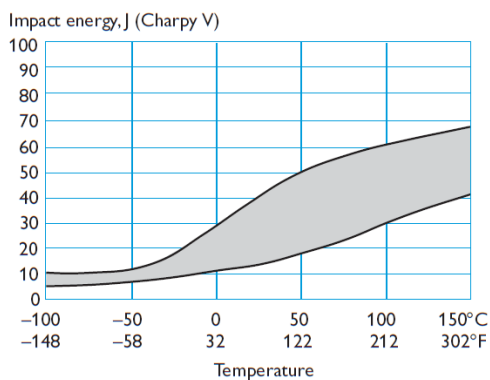
Mechanical properties

Impact strength, tensile strength and the compressive strength depends on the hardness in the delivered condition.

IMPACT STRENGTH

The energy absorption at impact testing depends on the test material (bar size and delivered hardness), testing temperature and the specimen (type, location, and orientation in the bar).

The graph below shows how the impact energy changes as a function of the test temperature and hardness variation within the delivery hardness range.



TENSILE STRENGTH

Approx. values. Samples were taken from a flat bar, 90 x 300 mm (3.5" x 11.8").

Hardness: 325 HB.

Testing temperature	20°C (68°F)	200°C (390°F)
Ultimate tensile strength R_m N/mm ²	1020	930
Yield strength $R_{p0.2}$ N/mm ²	900	800

COMPRESSIVE STRENGTH

Compressive yield strength $R_{c0.2}$ N/mm ²	850–1000
--	----------

Heat treatment

Uddeholm Impax Supreme is intended for use in the hardened and tempered condition, i.e. the delivery condition.

When, however, the steel is to be heat treated to a higher hardness or case hardened, the following instructions may be helpful.

Soft annealing

Protect the steel and heat through to 700°C (1300°F). Then cool in the furnace at 10°C (50°F) per hour to 600°C (1110°F), then freely in air.

Stress relieving

After rough machining the tool should be heated through to 550°C (1020°F), holding time 2 hours. Cool slowly to room temperature.

Hardening

Note: The steel should be fully soft annealed before hardening.

Preheating temperature: 500–600°C (930–1110°F).

Austenitizing temperature: 850°C (1560°F).

The steel should be heated through to the austenitizing temperature and held at temperature for 30 minutes.

Protect the tool against decarburization and oxidation during the hardening process.

Quenching media

- High speed gas/circulating atmosphere (Only suitable for small dimensions)
- Oil (60–80°C/140–175°F)
- Martempering bath 300°C (570°F) max. 4 minutes, then air

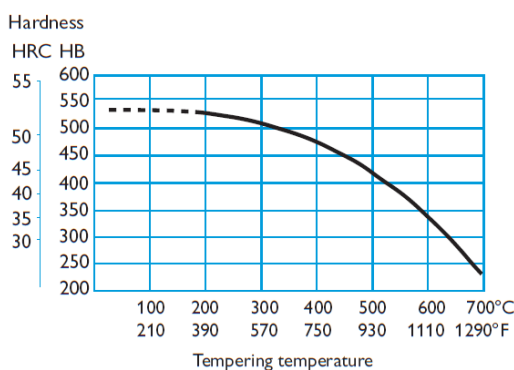
Note: Temper immediately tool reaches 50–70°C (120–160°F).

Tempering

Choose the tempering temperature according to the hardness required by reference to the tempering graph. Temper twice with intermediate cooling to room temperature. Lowest tempering temperature 180°C (360°F) for small inserts, but preferred minimum is 250°C (480°F). Holding time at temperature minimum 2 hours.

TEMPERING GRAPH

The diagram is valid for small samples 15 x 15 x 40 mm (0.6 x 0.6 x 1.6 in.) austenitized 30 min. at 850°C (1560°F), quenched in air and tempered 2 + 2 hours.



Flame and induction hardening

Uddeholm Impax Supreme can be flame or induction hardened to a hardness of approx. 50 HRC. Cooling in air is preferable.

Further information can be obtained from the Uddeholm Technical Services Report "Flame hardening of Uddeholm Impax Supreme".

Nitriding and nitrocarburizing

Nitriding gives a hard surface which is very resistant to wear and erosion. A nitrided surface also increases the corrosion resistance. For best result the following steps should be followed:

1. Rough machining
2. Stress tempering at 550°C (1020°F)
3. Grinding
4. Nitriding

Following surface hardness and nitriding depths will be achieved after nitriding:

	Temperature		Time h	Surface hardness HV ₁	Depth of case	
	°C	°F			mm	inch
Gas nitriding	525	977	20	650	0.30	0.012
	525	977	30	650	0.35	0.013
Ion- nitriding	480	896	24	700	0.30	0.012
	480	896	48	700	0.40	0.016
Nitrocar- burizing	570	1058	2	700	0.10	0.004

Cutting data recommendations

The cutting data below are to be considered as guiding values which must be adapted to existing local conditions.

More information can be found in our technical information "Cutting data recommendations".

Turning

Cutting data parameters	Turning with carbide		Turning with high speed steel Fine turning
	Rough turning	Fine turning	
Cutting speed, (v _c) m/min f.p.m.	120–170 394–558	170–220 558–722	15–20 49–66
Feed, (f) mm/r i.p.r.	0.2–0.4 0.008–0.016	0.05–0.2 0.002–0.008	0.05–0.3 0.002–0.012
Depth of cut, (a _p) mm inch	2–4 0.08–0.16	0.5–2 0.02–0.08	0.5–3 0.02–0.12
Carbide desig- nation, ISO	P20–P30 Coated carbide	P10 Coated carbide or Cemet	–

Drilling

HIGH SPEED STEEL TWIST DRILL

Drill diameter		Cutting speed, (v _c)		Feed, (f)	
mm	inch	m/min	f.p.m.	mm/r	i.p.r.
–5	–3/16	14–16*	46–52	0.08–0.15	0.003–0.006
5–10	3/16–3/8	14–16*	46–52	0.15–0.25	0.006–0.010
10–15	3/8–5/8	14–16*	46–52	0.25–0.30	0.010–0.012
15–20	5/8–3/4	14–16*	46–52	0.30–0.35	0.012–0.014

* For coated high speed steel drill v_c = 24–26 m/min (79–85 f.p.m.)

CARBIDE DRILL

Cutting data parameters	Type of drill		
	Indexable insert	Solid carbide	Carbide tip ¹⁾
Cutting speed, (v _c) m/min f.p.m.	180–200 600–656	120–150 394–492	60–80 197–262
Feed, (f) mm/r i.p.r.	0.05–0.15 ²⁾ 0.002–0.006 ²⁾	0.08–0.20 ³⁾ 0.003–0.008 ³⁾	0.15–0.25 ⁴⁾ 0.006–0.01 ⁴⁾

¹⁾ Drill with replaceable or brazed carbide tip

²⁾ Feed rate for drill diameter 20–40 mm (0.8"–1.6")

³⁾ Feed rate for drill diameter 5–20 mm (0.2"–0.8")

⁴⁾ Feed rate for drill diameter 10–20 mm (0.4"–0.8")

Milling

FACE AND SQUARE SHOULDER MILLING

Cutting data parameters	Milling with carbide	
	Rough milling	Fine milling
Cutting speed, (v _c) m/min f.p.m.	80–150 265–492	150–190 492–623
Feed, (f _z) mm/tooth inch/tooth	0.2–0.4 0.008–0.016	0.1–0.2 0.004–0.008
Depth of cut, (a _p) mm inch	2–4 0.08–0.16	–2 –0.08
Carbide designation ISO	P20–P40 Coated carbide	P10–P20 Coated carbide or Cemet

END MILLING

Cutting data parameters	Milling cutter		
	Solid carbide	Carbide indexable insert	High speed steel
Cutting speed, (v_c) m/min f.p.m.	70–110 230–361	80–120 262–394	15–20 ¹⁾ 49–66 ¹⁾
Feed, (f_z) mm/tooth inch/tooth	0.03–0.20 ²⁾ 0.001–0.008 ²⁾	0.08–0.20 ²⁾ 0.003–0.008 ²⁾	0.05–0.35 ²⁾ 0.002–0.014 ²⁾
Carbide designation, ISO	–	P20–P40	–

¹⁾ For coated high speed steel end mill $v_c = 35–40$ m/min (115–131 f.p.m.)

²⁾ Depending on radial depth of cut and cutter diameter

Grinding

A general grinding wheel recommendation is given below. More information can be found in the Uddeholm brochure “Grinding of Tool Steel”.

Type of grinding	Wheel recommendation
Face grinding straight wheel	A 46 HV
Face grinding segments	A 24 GV
Cylindrical grinding	A 60 KV
Internal grinding	A 46 JV
Profile grinding	A 100 KV

Welding

Good results when welding tool steel can be achieved if proper precautions are taken during welding (elevated working temperature, joint preparation, choice of consumables and welding procedure). If the tool is to be polished or photo-etched, it is necessary to work with an electrode type of matching composition.

Welding method	TIG	MMA (SMAW)
Working temperature	200–250°C (390–480°F)	200–250°C (390–480°F)
Consumables	IMPAX TIG-WELD	IMPAX WELD
Hardness after welding	300–330 HB	300–330 HB

Further information is given in the Uddeholm brochures “Welding of Tool Steel” and “Uddeholm Impax Weld/TIG-Weld”.

Electrical-discharge machining

If spark-erosion, EDM, is performed in the as delivered condition, the tool should then be given an additional temper at approx. 550°C (1020°F). If the steel has been rehardened, the additional tempering temperature should be 25°C (50°F) lower than the last tempering temperature used.

Further information can be obtained from the Uddeholm brochure “EDM of tool steel”.

Hard-chromium-plating

After hard-chromium-plating, the tool should be tempered for approx. 4 hours at 180°C (350°F) within 4 hours of plating in order to avoid hydrogen embrittlement.

Photo-etching

Uddeholm Impax Supreme is particularly suitable for texturing by the photo-etching process. Its very low sulphur content ensures accurate and consistent pattern reproduction.

For heavy sections an extra tempering at 550°C (1020°F) before photo-etching is recommended.

Polishing

Uddeholm Impax Supreme has good polishability in the hardened and tempered condition. After grinding, polishing is undertaken with aluminium oxide or diamond paste.

Note: Each steel grade has an optimum polishing time which largely depends on hardness and polishing technique. Overpolishing can lead to a poor surface finish (e.g. an “orange peel” effect).

Further information is given in the Uddeholm publication “Polishing of mould steel”.

Further information

Contact your local Uddeholm office for further information on the selection, heat treatment, application and availability of Uddeholm tool steels, including the publication “Steels for moulds”.

EN 1.2312

General

Uddeholm Holdax is a vacuum-degassed chromium-molybdenum-alloyed steel which is supplied in the hardened and tempered condition.

Uddeholm Holdax is characterized by:

- excellent machinability
- good resistance to indentation
- uniform hardness in all dimensions

Typical analysis %	C	Si	Mn	S	Cr	Mo
	0,40	0,4	1,5	0,07	1,9	0,2
Standard specification	AISI 4130–35 improved, W.-Nr. 1.2312					
Delivery condition	Hardened and tempered to 290–330 HB					
Colour code	Yellow/blue					

Applications

- Holders/bolsters for plastic moulds and die casting dies
- Plastics and rubber moulds with low requirements on polishability
- Support plates
- Constructional parts

Properties

Physical data

Hardened and tempered to 310 HB.

Temperature	20°C (68°F)	200°C (390°F)
Density, kg/m ³ lbs/in ³	7800 0,282	7750 0,280
Coefficient of thermal expansion per °C from 20°C per °F from 68°F	–	12,7 × 10 ⁻⁶ 6,1 × 10 ⁻⁶
Thermal conductivity W/m °C Btu in/ft ² h °F	–	33 225
Modulus of elasticity N/mm ² p.s.i.	200 000 29,0 × 10 ⁶	195 000 28,3 × 10 ⁶
Specific heat capacity J/kg °C Btu/lb °F	460 0,110	–

Mechanical properties

The tensile and compressive strength depends on the delivery hardness.

TENSILE STRENGTH

Tensile strength, R _m	800–1 100 N/mm ² 116 000–159 000 p.s.i.
Yield strength, R _{p0.2}	750–950 N/mm ² 109 000–138 000 p.s.i.

At 200°C (390°F) the tensile strength and the yield strength are approx. 100 N/mm² lower than at room temperature.

COMPRESSIVE STRENGTH

Yield strength, R _{c0.2}	850–1 000 N/mm ² 123 000–145 000 p.s.i.
-----------------------------------	---

The high sulphur content gives worse mechanical properties in the transverse direction compared with the longitudinal direction.

Heat treatment

Uddeholm Holdax is intended for use in the as-delivered condition. When the steel is to be heat treated to higher hardness, instructions below are to be followed.

Soft annealing

Protect the steel and heat through to 720°C (1330°F), holding time 2 hours. Cool in furnace at 10°C (50°F) per hour to 600°C (1110°F), then freely in air.

Stress tempering

After rough machining the tool should be heated through to 550°C (1020°F), holding time 2 hours, then cool freely in air.

Hardening

Note: The steel should be fully soft annealed before hardening.

Preheating temperature: 500–600°C (930–1110°F).

Austenitizing temperature: 850°C (1560°F).

The steel should be heated through to the austenitizing temperature and held at temperature for 30 minutes.

Protect the tool against decarburization and oxidation during the hardening process.

Quenching media

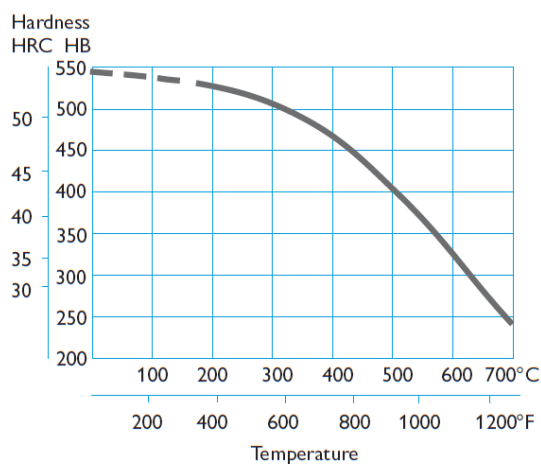
- Forced air/atmosphere (only for small tools)
- Oil
- Martempering bath 450–550°C (840–1020°F) max. 4 minutes, then cool in forced air

In order to obtain the optimum properties, the cooling rate should be as fast as is concomitant with acceptable distortion. Temper the tool as soon as its temperature reaches 50–70°C (120–160°F).

Tempering

Choose the tempering temperature according to the hardness required by reference to the tempering graph. Temper twice with intermediate cooling to room temperature. Lowest tempering temperature 180°C (360°F). Holding time at temperature minimum 2 hours.

The diagram below is valid for small samples (15 x 15 x 40 mm) quenched in air. Austenitizing temperature 850°C (1560°F), 30 minutes. Holding time 2 + 2 h.



Flame and induction hardening

Uddeholm Holdax can be flame or induction hardened to a hardness of approximate 50 HRC. Cooling in air is preferable. However, small tools sometimes require forced air cooling. Temper immediately after hardening.

Further information is given in the Uddeholm brochure "Flame hardening of Impax Supreme".

Nitriding and nitrocarburizing

Nitriding gives a hard surface layer which is very resistant to wear and erosion. A nitrided surface also increases the corrosion resistance.

For best result the following steps should be followed:

1. Rough machining
2. Stress tempering at 550°C (1020°F)
3. Fine machining
4. Nitriding

Following surface hardness and nitriding depths will be achieved after nitriding/nitrocarburizing.

	Temperature		Time h	Surface hardness HV	Depth of case approx.	
	°C	°F			mm	in.
Gas-nitriding	510	950	10	750	0.20	0.008
	510	950	30	750	0.30	0.012
	510	950	60	750	0.40	0.016
Ion-nitriding	480	895	10	750	0.20	0.008
	480	895	30	750	0.25	0.010
	480	895	60	750	0.35	0.014
Nitrocarburizing —gas —salt bath	580	1060	2.5	600	0.25	0.010
	580	1060	1	650	0.15	0.006

Machining recommendation

The cutting data below are to be considered as guiding values which must be adapted to existing local conditions.

Turning

Cutting data parameters	Turning with carbide		Turning with high speed steel Fine turning
	Rough turning	Fine turning	
Cutting speed (v_c) m/min. f.p.m.	140–190 460–620	190–240 620–790	20–25 65–80
Feed (f) mm/r i.p.r.	0,2–0,4 0,008–0,016	0,05–0,2 0,002–0,008	0,05–0,3 0,002–0,012
Dept of cut (a_p) mm inch	2–4 0,08–0,16	0,5–2 0,02–0,08	0,5–3 0,02–0,12
Carbide designation ISO	P20–P40 Coated carbide	P10–P20 Coated carbide or cermet	–

Drilling

HIGH SPEED STEEL TWIST DRILLS

Drill diameter		Cutting speed (v_c)		Feed (f)	
mm	inch	m/min	f.p.m.	mm/r	i.p.r.
– 5	– 3/16	18–20*	60–65*	0,08–0,20	0,003–0,008
5–10	3/16–3/8	18–20*	60–65*	0,20–0,30	0,008–0,012
10–15	3/8–5/8	18–20*	60–65*	0,30–0,35	0,012–0,014
15–20	5/8–3/4	18–20*	60–65*	0,35–0,40	0,014–0,016

* For coated HSS drills $v_c = 32–34$ m/min (105–112 f.p.m.)

CARBIDE DRILLS

Cutting data parameters	Type of drill		
	Indexable insert	Solid carbide	Brazed carbide ¹⁾
Cutting speed (v_c) m/min. f.p.m.	120–150 390–490	200–220 655–720	70–90 230–295
Feed (f) mm/r i.p.r.	0,05–0,15 ²⁾ 0,002–0,010 ⁶⁾	0,08–0,20 ³⁾ 0,003–0,008 ³⁾	0,15–0,25 ⁴⁾ 0,006–0,010 ⁴⁾

¹⁾ Drill with replaceable or brazed carbide tip

²⁾ Feed rate for drill diameter 20–40 mm (0.8"–1.6")

³⁾ Feed rate for drill diameter 5–20 mm (0.2"–0.8")

⁴⁾ Feed rate for drill diameter 10–20 mm (0.4"–0.8")

Milling

FACE AND SQUARE SHOULDER MILLING

Cutting data parameters	Milling with carbide	
	Rough milling	Fine milling
Cutting speed (v_c) m/min. f.p.m.	80–150 260–490	150–190 490–620
Feed (f_z) mm/tooth in/tooth	0,2–0,4 0,008–0,016	0,1–0,2 0,004–0,008
Dept of cut (a_p) mm inch	2–4 0,08–0,16	–2 –0,08
Carbide designation ISO	P20–P40 Coated carbide	P10–P20 Coated carbide or cermet

END MILLING

Cutting data parameters	Type of milling		
	Solid carbide	Carbide indexable insert	High speed steel
Cutting speed (v_c) m/min. f.p.m.	70–110 230–360	80–120 260–390	20–25 ¹⁾ 65–80 ¹⁾
Feed (f_z) mm/tooth in/tooth	0,03–0,20 ²⁾ 0,001–0,008	0,08–0,20 ²⁾ 0,003–0,008	0,05–0,35 ²⁾ 0,002–0,014
Carbide designation ISO	–	P20–P30	–

¹⁾ For coated HSS end mill $v_c = 40–45$ m/min. (130–148 f.p.m.)

²⁾ Depending on radial depth of cut and cutting diameter

Grinding

A general grinding wheel recommendation is given below. More information can be found in the Uddeholm publication "Grinding of Tool Steel".

Type of grinding	Prehardend condition
Face grinding straight wheel	A 46 HV
Face grinding segments	A 24 GV
Cylindrical grinding	A 60 KV
Internal grinding	A 60 JV
Profile grinding	A 100 LV

Welding

Good results when welding tool steel can be achieved if proper precautions are taken during welding (elevated working temperature, joint preparation, choice of consumables and welding procedure).

Welding method	TIG	MMA (SMAW)
Working temperature	200–250°C (390–480°F)	200–250°C (390–480°F)
Welding consumables	IMPAX TIG-WELD	IMPAX WELD
Hardness after welding	320–350 HB	320–350 HB

Uddeholm Holdax has a high sulphur content, which means an increased risk for hot cracking during welding. To minimize the risk, keep the dilution as low as possible.

Further information is given in the Uddeholm brochure “Welding of Tool Steel”.

Further information

Please contact your local Uddeholm office for further information on the selection, heat treatment and application of Uddeholm tool steel, including the publication “Steel for Moulds”.

EN 1.2343

General

Uddeholm Vidar Superior is a chromium-molybdenum-vanadium alloyed hot work tool steel which is characterized by:

- High level of resistance to thermal shock and thermal fatigue
- Good high-temperature strength
- Excellent toughness and ductility in all directions
- Excellent through-hardening properties
- Good dimensional stability during hardening

Typical analysis %	C	Si	Mn	Cr	Mo	V
	0.36	0.3	0.3	5.0	1.3	0.5
Standard specification	X36 CrMoV5 (CNOMO) X36 CrMoV5-1, W.-Nr. 1.2340 ~AISI H11, ~B H11, ~W.-Nr. 1.2343, ~AFNOR Z38 CDV 5, ~UNI X37 CrMoV 51 KU, ~UNE X37 CrMoV 5					
Delivery condition	Soft annealed to approx.180 HB					
Colour code	Red/orange with a white line across					

Improved tooling performance

The name "Superior" implies that by special processing techniques and close process control, the steel attains high purity and a very fine structure. Uddeholm Vidar Superior shows significant improvements in impact toughness compared to material of the H11 (W.-Nr. 1.2343) type.

The improved impact toughness is particularly valuable for tooling subjected to high mechanical and thermal stresses, e.g. die casting dies and forging tools. In practical terms, tools may be used at somewhat higher working hardness (2 HRC) without loss of toughness. Since increased service hardness limits the formation of thermal fatigue cracks, improved tool performance can be expected.

Applications

Tools for die casting

Part	Tin, lead, zinc alloys, HRC	Aluminium-, Magnesium alloys, HRC
Dies	46–50	42–48
Fixed inserts, cores	48–52	46–50
Sprue parts	(ORVAR)	(ORVAR)
Nozzles	(ORVAR)	(ORVAR)
Ejector pins (nitrided)	(ORVAR)	(ORVAR)
Plunger, shot-sleeve (normally nitrided)	(ORVAR)	(ORVAR)
Austenitizing temperature	980–1000°C (1795–1830°F)	

Tools for hot forging

Material	Austenitizing temperature (approx.)	HRC
Aluminium, magnesium	980–1000°C (1795–1830°F)	44–52
Copper alloys	980–1000°C (1795–1830°F)	44–52
Steel	980–1000°C (1795–1830°F)	40–50

Moulds for plastics

Part	Austenitizing temperature (approx.)	HRC
Injection moulds	980–1000°C (1795–1830°F)	46–52
Compression/transfer moulds		

Properties

Physical data

All specimens are taken from the centre of a 1000 x 200 mm (39,4" x 7,88") bar. Unless otherwise is indicated all specimens were hardened from 1000°C (1832°F), quenched in a vacuum furnace and tempered 2 + 2h at 600°C (1110°F) to 45 ±1 HRC.

Temperature	20°C (68°F)	200°C (392°F)	400°C (750°F)	600°C (1010°F)
Density, kg/m ³ lbs/in ³	7 800 0.282	7 750 0.280	7 700 0.278	7 600 0.275
Modulus of elasticity MPa psi	210 000 30.5 x 10 ⁶	200 000 29.1 x 10 ⁶	180 000 26.1 x 10 ⁶	140 000 20.3 x 10 ⁶
Coefficient of thermal expansion per °C from 20°C per °F from 68°F	– –	11.6 x 10 ⁻⁶ 6.4 x 10 ⁻⁶	12.4 x 10 ⁻⁶ 6.9 x 10 ⁻⁶	13.2 x 10 ⁻⁶ 7.3 x 10 ⁻⁶
Thermal conductivity W/m °C Btu in/ (ft ² h°F)	– –	30 211	30 211	31 218

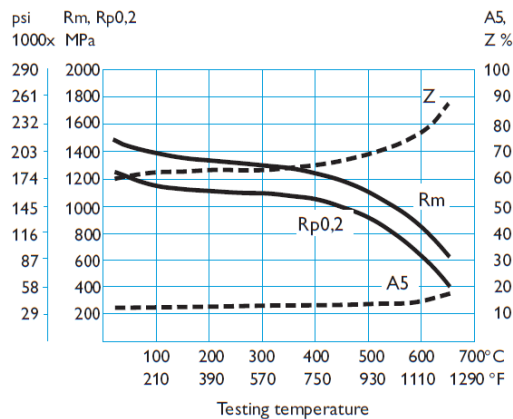
Mechanical properties

Approximate tensile strength at room temperature.

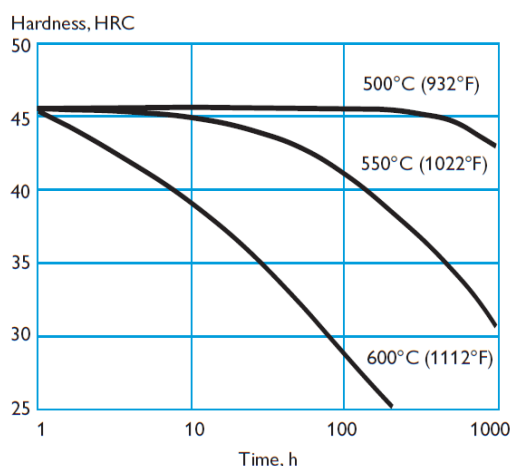
Hardness	45 HRC	46.5 HRC	48.5 HRC
Tensile strength R_m	1450 MPa 210 000 psi	1580 MPa 229 000 psi	1680 MPa 244 000 psi
Yield strength $R_{p0.2}$	1240 MPa 180 000 psi	1340 MPa 194 000 psi	1410 MPa 204 000 psi
Elongation A_5	13%	13%	12%
Reduction in area Z	65%	65%	64%

APPROXIMATE STRENGTH AT ELEVATED TEMPERATURES

Longitudinal direction.

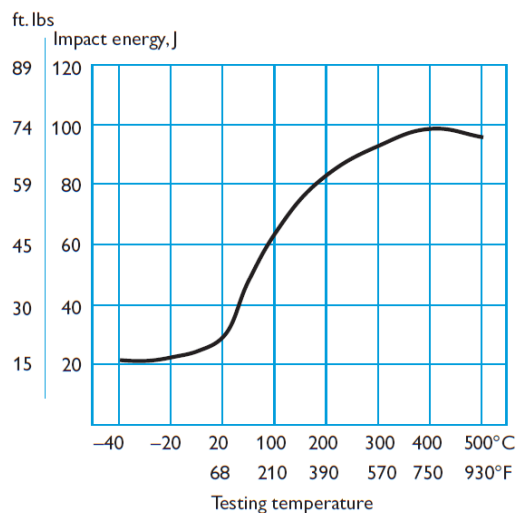


EFFECT OF TIME AT HIGH TEMPERATURES ON HARDNESS



EFFECT OF TESTING TEMPERATURES ON IMPACT ENERGY

Charpy V specimens, short transverse direction.



Heat treatment

Soft annealing

Protect the steel and heat through to 850°C (1562°F). Then cool in furnace at 10°C (20°F) per hour to 650°C (1202°F), then freely in air.

Stress relieving

After rough machining the tool should be heated through to 650°C (1202°F), holding time 2 hours. Cool slowly to 500°C (932°F), then freely in air.

Hardening

Preheating temperature: 600–900°C (1112–1652°F). Minimum two preheating steps at 600–650°C (1112–1202°F) and 820–850°C (1508–1562°F). When three preheats are used the second is carried out at 820°C (1508°F) and the third at 900°C (1652°F).

Austenitizing temperature: 980–1000°C (1796–1832°F).

Soaking time: 30–45 minutes.

Soaking time = time at austenitizing temperature after the tool is fully heated through.

Protect the tool against decarburization and oxidation during austenitizing.

Quenching media

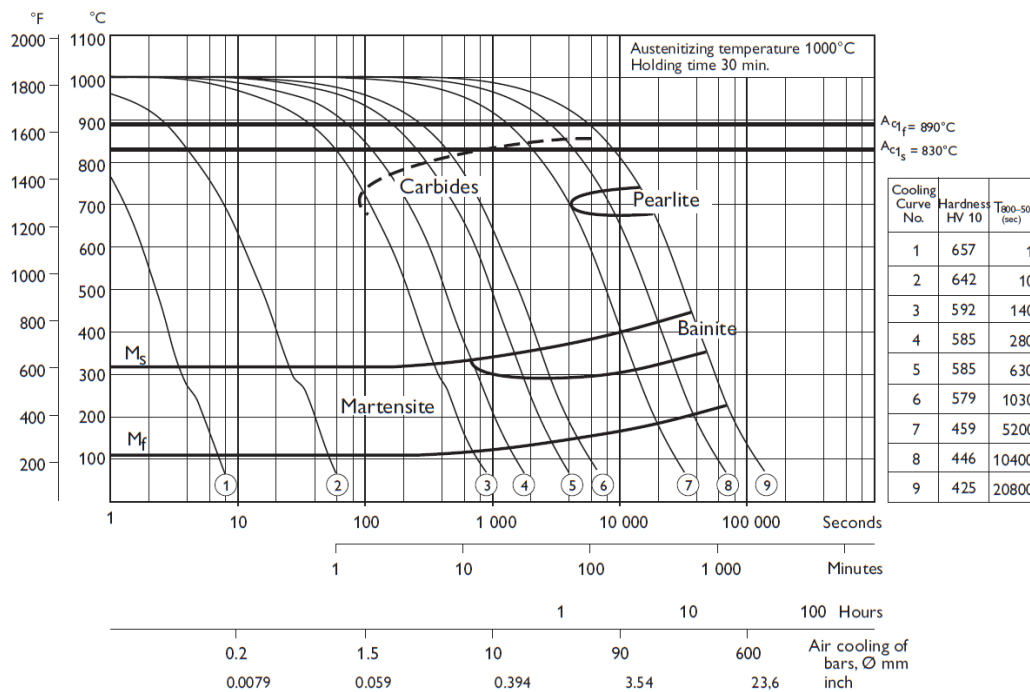
- High speed gas/circulating atmosphere.
- Vacuum (high speed gas with sufficient positive pressure). An interrupted quench at 350–450°C (662–842°F) is recommended where distortion control and quench cracking are a concern.
- Martempering bath (salt or fluidized bed) at 500–550°C (932–1022°F) or 180–220°C (356–428°F).
- Warm oil, approx. 80°C (176°F).

Note 1: Temper the tool as soon as its temperature reaches 50–70°C (122–158°F).

Note 2: In order to obtain the optimum properties for the tool, the cooling rate should be fast, but not at a level that gives excessive distortion or cracks.

CCT GRAPH

Austenitizing temperature 1000°C (1832°F). Holding time 30 minutes.



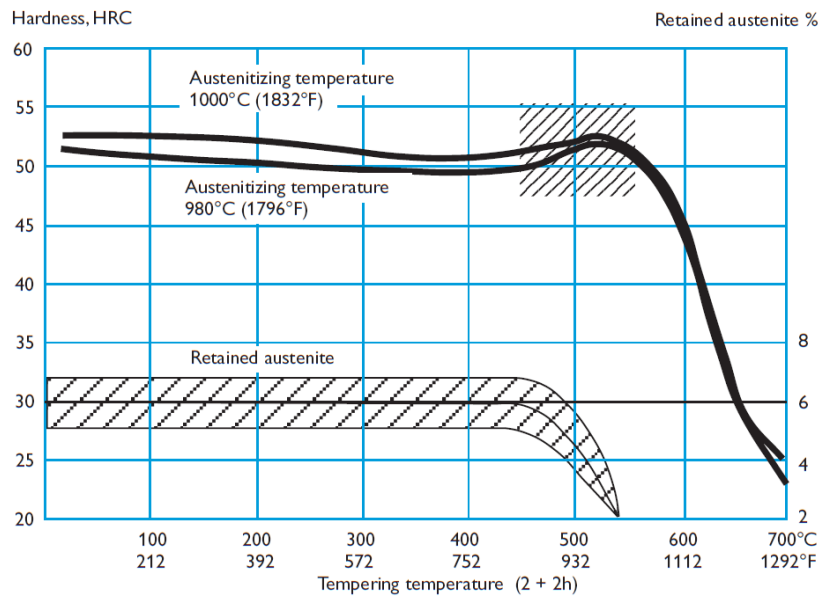
Tempering

Choose the tempering temperature according to the hardness required by reference to the tempering graph below. Temper minimum twice with intermediate cooling to room

temperature. Holding time at temperature is minimum 2 hours. Tempering in the temperature range 450–550°C (840–1020°F) for the intended final hardness will result in a lower toughness.

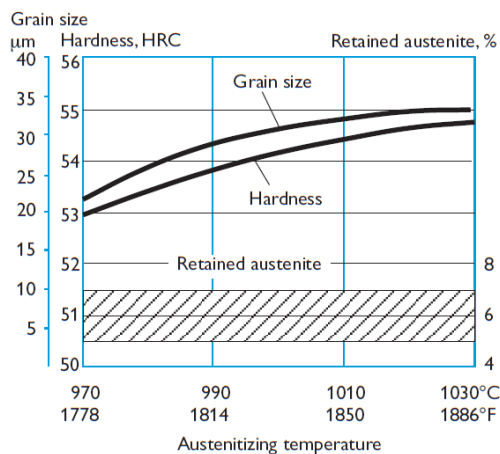
TEMPERING GRAPH

Air cooling of specimen 15 x 15 x 40 mm (0.6" x 0.6" x 1.6").



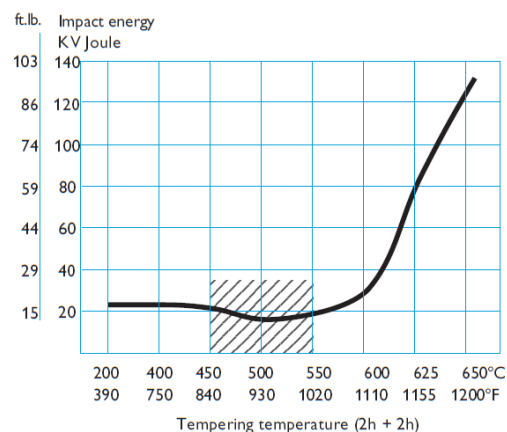
Above tempering curves are obtained after heat treatment of samples with a size of 15 x 15 x 40 mm, cooling in forced air. Lower hardness can be expected after heat treatment of tools and dies due to factors like actual tool size and heat treatment parameters.

HARDNESS, GRAIN SIZE AND RETAINED AUSTENITE AS A FUNCTION OF AUSTENIZING TEMPERATURE



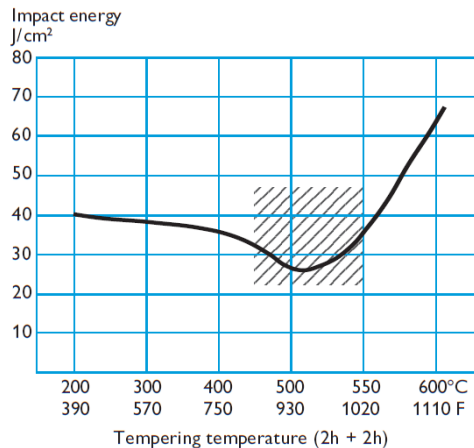
APPROXIMATE IMPACT ENERGY AT DIFFERENT TEMPERING TEMPERATURES

Charpy V specimens, short transverse direction.



Tempering in the temperature range 450–550°C (840–1020°F) for the intended final hardness will result in a lower toughness.

Charpy U specimens, short transverse direction.



Dimensional changes during hardening and tempering

During hardening and tempering the die is exposed to thermal as well as transformation stresses. This will inevitably result in dimensional changes and in the work case distortion. It is therefore recommended to always leave a machining allowance after machining before the die is hardened and tempered. Normally the size in the largest direction will shrink and the size in the smallest direction might increase but this is also a matter of the die size, the die design as well as the cooling rate after hardening.

For Uddeholm Vidar Superior it is recommended to leave a machining allowance of 0.2 per cent of the dimension in length, width and thickness.

Nitriding and nitrocarburizing

Nitriding and nitrocarburizing result in a hard surface layer which is very resistant to wear and erosion. The nitrided layer is, however, brittle and may crack or spall when exposed to mechanical or thermal shock, the risk increasing with layer thickness. Before nitriding, the tool should be hardened and tempered at a temperature at least 50°C (120°F) above the nitriding temperature.

Nitriding in ammonia gas at 510°C (950°F) or plasma nitriding in 75% hydrogen / 25% nitrogen at 480°C (896°F) both result in a surface hardness of ~1100 HV_{0.2}. In general, plasma nitriding is the preferred method because of better control over nitrogen potential; in particular, formation of the so-called “white layer”, which is not recommended for hot-work service, can readily be avoided.

However, careful gas nitriding can give perfectly acceptable results.

Uddeholm Vidar Superior can also be nitrocarburized in either gas or salt bath. The surface hardness after nitrocarburizing is 1000–1100 HV_{0.2}.

Depth of nitriding

Process	Time	Depth	
		mm	inch
Gas nitriding at 510°C (950°F)	10 h	0.12	0.0047
	30 h	0.21	0.0082
Plasma nitriding at 480°C (895°F)	10 h	0.10	0.0039
	30 h	0.19	0.0075
Nitrocarburizing – in gas at 580°C (1075°F) – in salt bath at 580°C (1075°F)	2.5 h	0.13	0.0051
	1 h	0.07	0.0028

* Depth of case = distance from surface where hardness is 50 HV_{0.2} over base hardness

Uddeholm Vidar Superior can also be nitrided in the soft annealed condition. The hardness and depth of case will, however, be reduced somewhat in this case.

Cutting data recommendations

The cutting data below are to be considered as guiding values which must be adapted to existing local conditions. More information can be found in the Uddeholm publication "Cutting data recommendation".

The recommendations, in following tables, are valid for Uddeholm Vidar Superior in soft annealed condition.

Turning

Cutting data parameters	Turning with carbide		Turning with HSS
	Rough turning	Fine turning	Fine turning
Cutting speed (v_c) m/min f.p.m.	170–220 558–722	220–270 722–886	25–30 82–98
Feed (f) mm/rev. i.p.r.	0.2–0.4 0.008–0.016	0.05–0.2 0.002–0.008	–0.3 –0.012
Depth of cut (a_p) mm inch	2–4 0.08–0.16	0.5–2 0.02–0.08	–2 –0.08
Carbide designation ISO US	P20–P30 C6–C5 Coated carbide	P10 C7 Coated carbide or cermet	–

HSS = High Speed Steel

Milling

FACE AND SQUARE SHOULDER MILLING

Cutting data parameters	Milling with carbide	
	Rough milling	Fine milling
Cutting speed (v_c) m/min f.p.m.	140–220 459–722	220–260 722–853
Feed (f_z) mm/tooth inch/tooth	0.2–0.4 0.008–0.016	0.1–0.2 0.004–0.008
Depth of cut (a_p) mm inch	2–4 0.08–0.16	–2 –0.08
Carbide designation ISO US	P20–P40 C6–C5 Coated carbide	P10 C7 Coated carbide or cermet

END MILLING

Cutting data parameters	Type of milling		
	Solid carbide	Carbide indexable insert	HSS
Cutting speed (v_c) m/min f.p.m.	145–185 476–607	150–190 492–623	30–35 ¹⁾ 98–115 ¹⁾
Feed (f_z) mm/tooth inch/tooth	0.03–0.2 ²⁾ 0.001–0.008 ²⁾	0.08–0.2 ²⁾ 0.003–0.008 ²⁾	0.05–0.35 ²⁾ 0.002–0.013 ²⁾
Carbide designation ISO US	–	P10–P20 C6–C5	–

¹⁾ For coated HSS end mill $v_c = 50–55$ m/min. (164–180 f.p.m.)

²⁾ Depending on radial depth of cut and cutter diameter

Drilling

HIGH SPEED STEEL TWIST DRILL

Drill diameter		Cutting speed (v_c)		Feed (f)	
mm	inch	m/min	f.p.m.	mm/rev.	i.p.r.
–5	–3/16	15–20*	49–66*	0.05–0.10	0.002–0.004
5–10	3/16–3/8	15–20*	49–66*	0.10–0.20	0.004–0.008
10–15	3/8–5/8	15–20*	49–66*	0.20–0.25	0.008–0.010
15–20	5/8–3/4	15–20*	49–66*	0.25–0.30	0.010–0.012

¹⁾ For coated HSS drill $v_c = 35–40$ m/min. (115–131 f.p.m.)

CARBIDE DRILL

Cutting data parameters	Type of drill		
	Indexable insert	Solid carbide	Carbide tip ¹⁾
Cutting speed (v_c) m/min f.p.m.	200–230 656–755	120–150 394–492	120–150 394–492
Feed (f) mm/rev. i.p.r.	0.05–0.15 ²⁾ 0.002–0.006 ²⁾	0.10–0.25 ³⁾ 0.004–0.01 ³⁾	0.15–0.25 ⁴⁾ 0.006–0.01 ⁴⁾

¹⁾ Drill with replaceable or brazed carbide tip

²⁾ Feed rate for drill diameter 20–40 mm (0.8"–1.6")

³⁾ Feed rate for drill diameter 5–20 mm (0.2"–0.8")

⁴⁾ Feed rate for drill diameter 10–20 mm (0.4"–0.8")

Grinding

A general grinding wheel recommendation is given below. More information can be found in the Uddeholm publication "Grinding of Tool Steel".

WHEEL RECOMMENDATION

Type of grinding	Soft annealed condition	Hardened condition
Face grinding straight wheel	A 46 HV	A 46 HV
Face grinding segments	A 24 GV	A 36 GV
Cylindrical grinding	A 46 LV	A 60 KV
Internal grinding	A 46 JV	A 60 IV
Profile grinding	A 100 IV	A 120 JV

Electrical Discharge Machining—EDM

Following the EDM process, the applicable die surfaces are covered with a resolidified layer (white layer) and a rehardened and untempered layer, both of which are very brittle and hence detrimental to die performance.

If EDM is used the white layer must be completely removed mechanically by grinding or stoning. After the finish machining the tool should also then be given an additional temper at approx. 25°C (50°F) below the highest previous tempering temperature.

Further information is given in the Uddeholm brochure "EDM of Tool Steel".

Polishing

Uddeholm Vidar Superior has good polishability in the hardened and tempered condition because of a very homogeneous structure. This coupled with a low level of non metallic inclusions, due to ESR process, ensures good surface finish after polishing.

Note: Each steel grade has an optimum polishing time which largely depends on hardness and polishing technique. Overpolishing can lead to a poor surface finish, "orange peel" or pitting.

Further information is given in the Uddeholm publication "Polishing of mould steel".

Photo-etching

Uddeholm Vidar Superior is particularly suitable for texturing by the photo-etching method. Its high level of homogeneity and low sulphur content ensures accurate and consistent pattern reproduction.

Welding

Welding of die components can be performed, with acceptable results, as long as the proper precautions are taken during the preparation of the joint, the filler material selection, the preheating of the die, the controlled cooling of the die and the post weld heat treatment processes. The following guidelines summarize the most important welding process parameters.

For more detailed information refer to Uddeholm's "Welding of Tool Steel" brochure.

Welding method	TIG	MMA
Preheating temperature*	Min. 325°C (620°F)	Min. 325°C (620°F)
Filler metals	DIEVAR TIG-WELD QRO 90 TIG-WELD	UTP 673 QRO 90 WELD
Maximum interpass temperature	475°C (880°F)	475°C (880°F)
Post welding cooling	20–40°C/h (35–70°F/h) for the first 2–3 hours and then freely in air.	
Hardness after welding	48–53 HRC	55–58 HRC (673) 48–53 HRC
<i>Heat treatment after welding</i>		
Hardened condition	Temper at 10–20°C (20–40°F) below the highest previous tempering temperature.	
Soft annealed condition	Soft-anneal the material at 850°C (1560°F) in protected atmosphere. Then cool in the furnace at 10°C (20°F) per hour to 650°C (1200°F) then freely in air.	

* Preheating temperature must be established throughout the die and must be maintained for the entirety of the welding process, to prevent weld cracking.

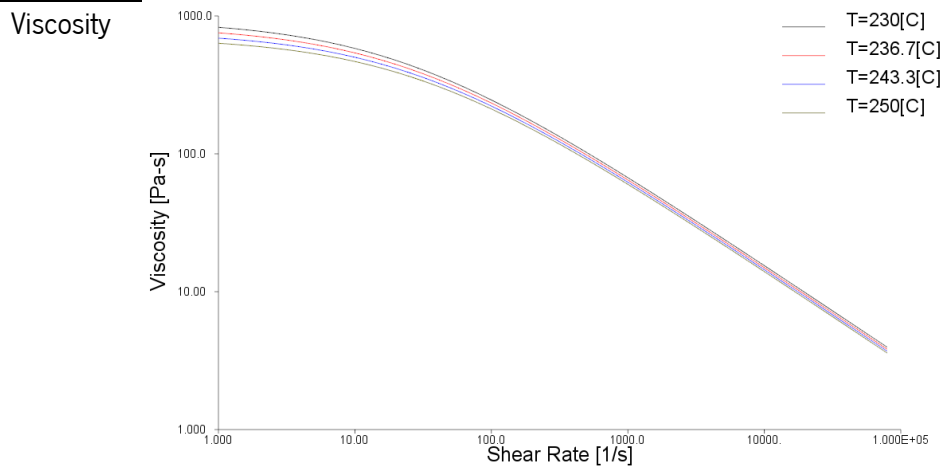
Further information

Please contact your local Uddeholm office for further information on the selection, heat treatment, application and availability of Uddeholm tool steel.

Appendix B

Table 22 – ExxonMobil Chemical - Exxtral BNU011 material properties.

Family name	POLYPROPYLENES (PP)	
Trade name	Exxtral BNU011	
Manufacturer	ExxonMobil Chemical	
Family abbreviation	PP	
Material structure	Crystalline	
Data source	Moldflow Corporation: pvT-Measured: mech-Supplemental	
Date last modified	25-JUL-06	
Date tested	08-OCT-02	
Grade code	BNU011	
Supplier code	EXXONMOB	
Fibers/fillers	Unfilled	
Mold surface temperature	60°C	
Melt temperature	240°C	
Mold temperature range (recommended)	Minimum	40°C
	Maximum	80°C
Melt temperature range (recommended)	Minimum	230°C
	Maximum	250°C
Absolute maximum melt temperature	300°C	
Ejection temperature	130°C	
Maximum shear stress	0.25 MPa	
Maximum shear rate	100000 1/s	



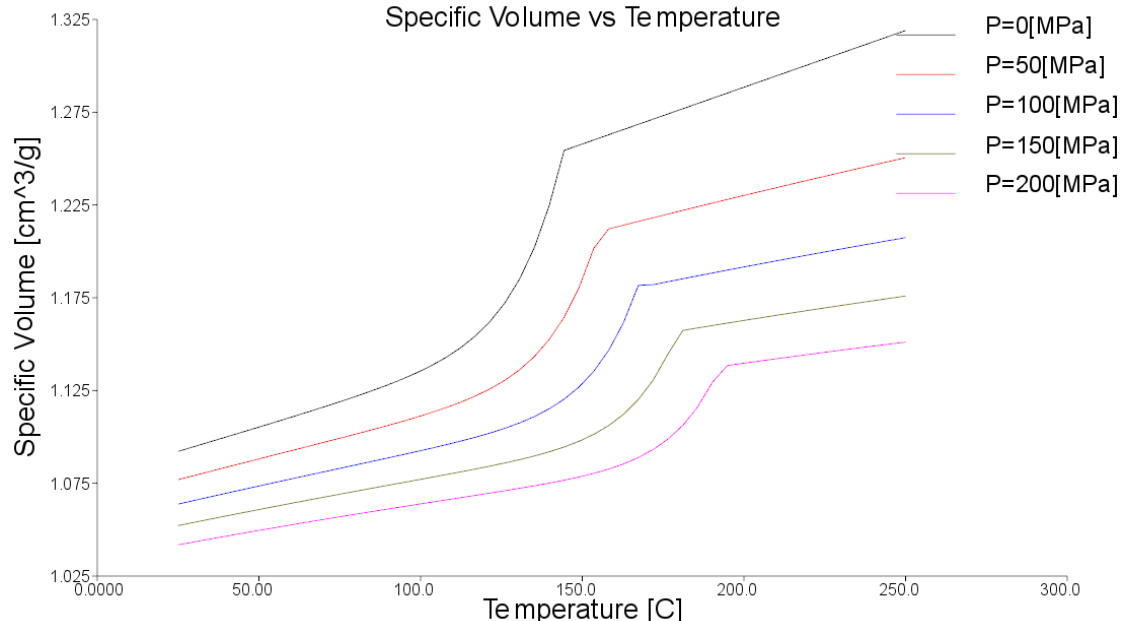
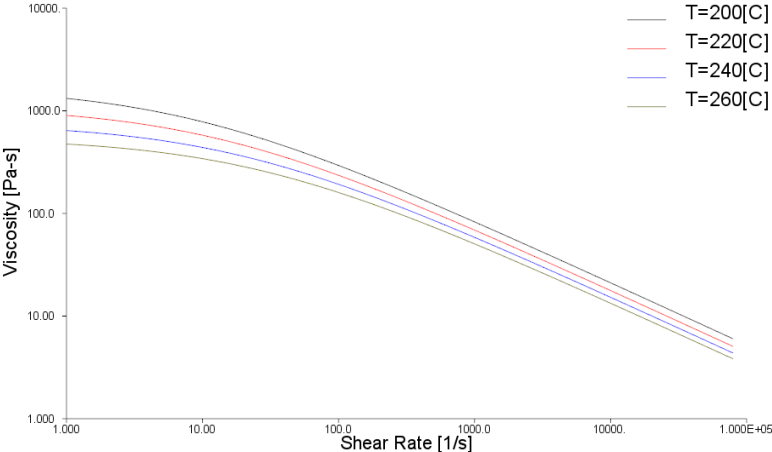
Juncture loss method coefficients	c1	Not provided
	c2	Not provided
Transition temperature	131°C	
Moldflow Viscosity index	VI(240)0063	
Melt mass-flow rate (MFR)	Temperature	230°C
	Load	2.16 kg
	Measured MFR	10 g/10min
Specific heat data	Not provided	
Thermal conductivity data	Not provided	
Melt density	0.76165 g/cm³	
Solid density	0.91562 g/cm³	
Specific volume vs Temperature		
<div><div><div>Specific Volume vs Temperature</div><div></div></div></div>		
Elastic modulus, 1 st principal direction (E1)	1340 MPa	
Elastic modulus, 2 nd principal direction (E2)	1340 MPa	
Poissons ratio (v12)	0.392	
Poissons ratio (v23)	0.392	
Shear modulus (G12)	481.3 MPa	
Observed nominal shrinkage	Parallel	1.3%
	Perpendicular	1.532%

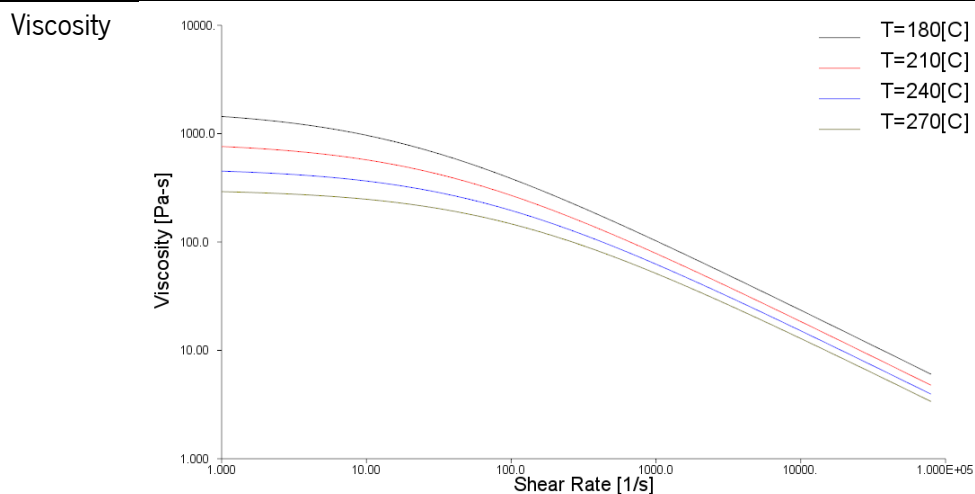
Table 23 – ExxonMobil Chemical - Exxtral HMU202 material properties.

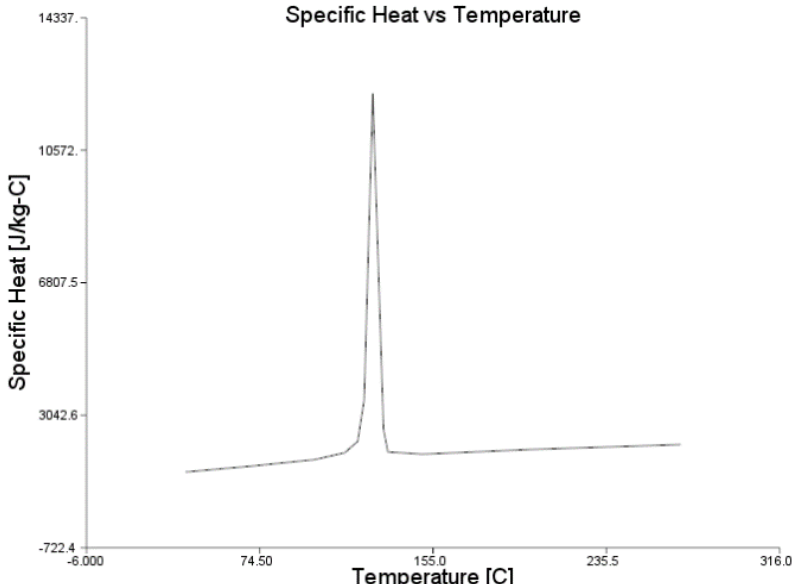
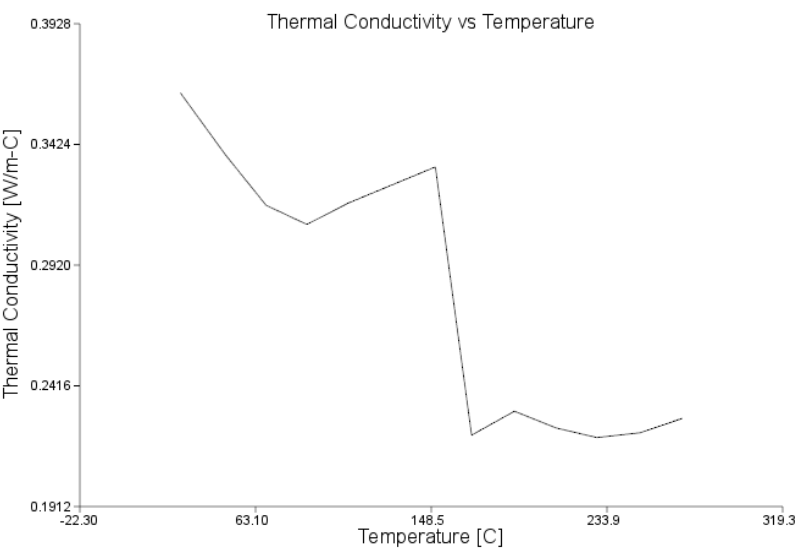
Family name	POLYPROPYLENES (PP)	
Trade name	Exxtral HMU202	
Manufacturer	ExxonMobil Chemical	
Family abbreviation	PP	
Material structure	Crystalline	
Data source	Moldflow Corporation: pvT-Measured: mech-Supplemental	
Date last modified	25-JUL-06	
Date tested	01-NOV-99	
Grade code	HMU202	
Supplier code	EXXONMOB	
Fibers/fillers	20% Talc Filled	
Mold surface temperature	55°C	
Melt temperature	230°C	
Mold temperature range (recommended)	Minimum	20°C
	Maximum	90°C
Melt temperature range (recommended)	Minimum	200°C
	Maximum	260°C
Absolute maximum melt temperature	300°C	
Ejection temperature	112°C	
Maximum shear stress	0.25 MPa	
Maximum shear rate	100000 1/s	
Viscosity	<div><div><div>T=200[C]</div><div>T=220[C]</div><div>T=240[C]</div><div>T=260[C]</div></div></div>	
Juncture loss method coefficients	c1	Not provided

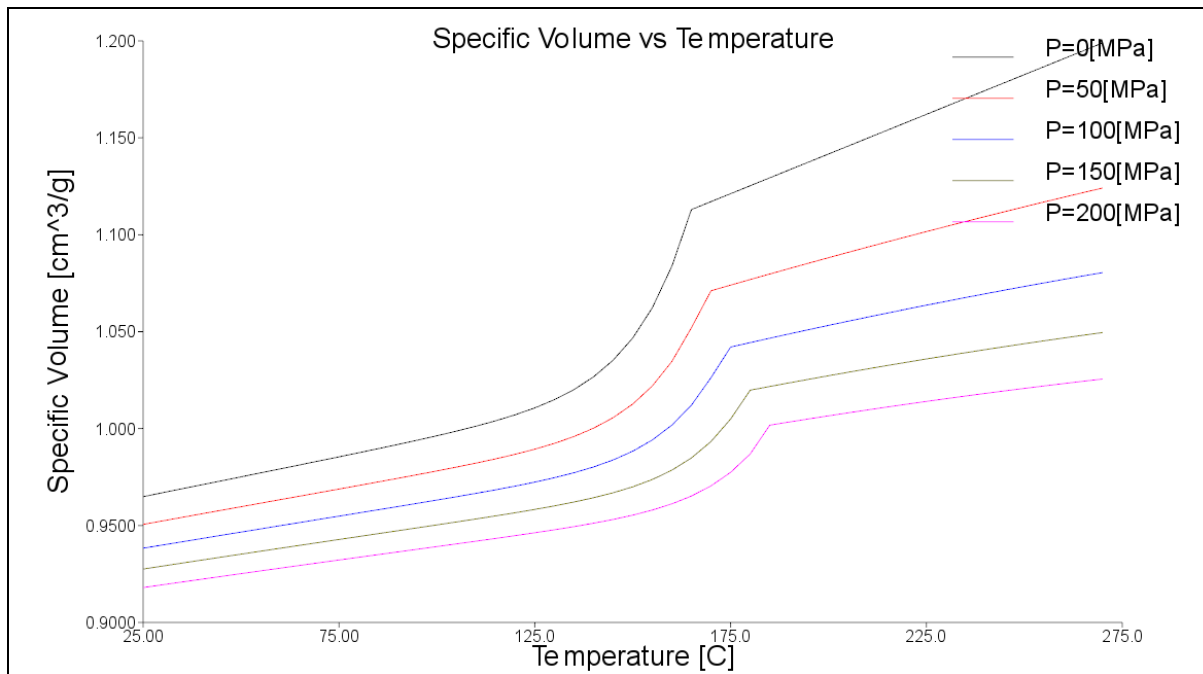
	c2	Not provided
Transition temperature	143°C	
Moldflow Viscosity index	VI(240)0058	
Melt mass-flow rate (MFR)	Temperature	Not provided
	Load	Not provided
	Measured MFR	Not provided
Specific heat data	Not provided	
Thermal conductivity data	Not provided	
Melt density	0.91086 g/cm³	
Solid density	1.0727 g/cm³	
Specific volume vs Temperature		
<div><p>Specific Volume vs Temperature</p><p>Y-axis: Specific Volume [cm³/g]</p><p>X-axis: Temperature [°C]</p><p>Legend:</p><ul style="list-style-type: none">P=0[MPa]P=50[MPa]P=100[MPa]P=150[MPa]P=200[MPa]</div>		
Elastic modulus, 1 st principal direction (E1)	1561.62 MPa	
Elastic modulus, 2 nd principal direction (E2)	1561.62 MPa	
Poissons ratio (v12)	0.384	
Poissons ratio (v23)	0.384	
Shear modulus (G12)	564.178 MPa	
Observed nominal shrinkage	Parallel	0.9228%
	Perpendicular	1.092%

Table 24 – SunAllomer Ltd - TRC104N material properties.

Family name	POLYPROPYLENES (PP)	
Trade name	TRC104N	
Manufacturer	SunAllomer Ltd	
Family abbreviation	PP	
Material structure	Crystalline	
Data source	Moldflow Plastics Labs : pvT-Measured : mech-Measured	
Date last modified	28-NOV-12	
Date tested	06-FEB-08	
Grade code	MAT4081	
Supplier code	SUNALL	
Fibers/fillers	20% Mineral Filled	
Mold surface temperature	40°C	
Melt temperature	225°C	
Mold temperature range (recommended)	Minimum	10°C
	Maximum	70°C
Melt temperature range (recommended)	Minimum	180°C
	Maximum	270°C
Absolute maximum melt temperature	300°C	
Ejection temperature	123°C	
Maximum shear stress	0.25 MPa	
Maximum shear rate	100000 1/s	



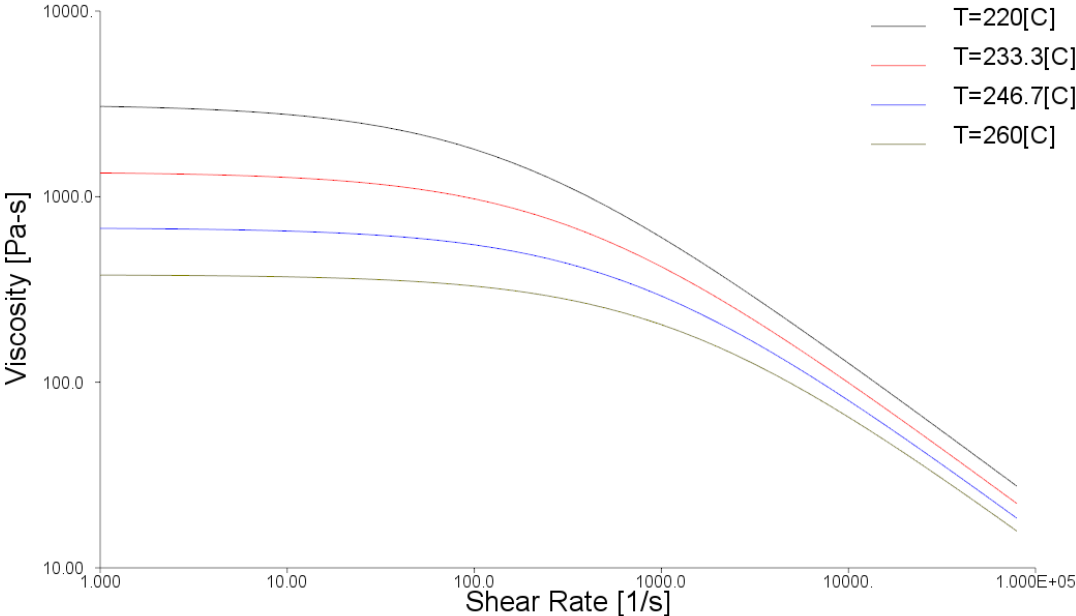
Juncture loss method coefficients		c1	3.192e-05
		c2	2.098
Transition temperature		131°C	
Moldflow Viscosity index		VI(240)0062	
Melt mass-flow rate (MFR)		Temperature	230°C
		Load	2.16 kg
		Measured MFR	23 g/10min
Specific heat data	<div><div>Specific Heat vs Temperature</div></div>		
Thermal conductivity data	<div><div>Thermal Conductivity vs Temperature</div></div>		
Melt density		0.86049 g/cm³	
Solid density		1.0364 g/cm³	
Specific volume vs Temperature			



Elastic modulus, 1 st principal direction (E1)	2274.6 MPa	
Elastic modulus, 2 nd principal direction (E2)	1927.17 MPa	
Poissons ratio (v12)	0.372	
Poissons ratio (v23)	0.688	
Shear modulus (G12)	966.25 MPa	
Observed nominal shrinkage	Parallel	0.5698%
	Perpendicular	0.7661%

Table 25 – Mitsubishi Chemical Corporation - Durabio D7340R material properties.

Family name	POLYCARBONATES (PC)
Trade name	Durabio D7340R
Manufacturer	Mitsubishi Chemical Corporation
Family abbreviation	PC
Material structure	Amorphous
Data source	Manufacturer (Mitsubishi Chemical Corporation) : pvT-Measured : mech-Measured
Date last modified	13-OCT-17
Date tested	13-OCT-17
Grade code	MAT6343
Supplier code	MITSUBIC
Fibers/fillers	Unfilled

Mold surface temperature	80°C	
Melt temperature	240°C	
Mold temperature range (recommended)	Minimum	70°C
	Maximum	90°C
Melt temperature range (recommended)	Minimum	220°C
	Maximum	260°C
Absolute maximum melt temperature	270°C	
Ejection temperature	95°C	
Maximum shear stress	1.5 MPa	
Maximum shear rate	15000 1/s	
Viscosity		
		
Juncture loss method coefficients	c1	Not provided
	c2	Not provided
Transition temperature	115°C	
Moldflow Viscosity index	VI(240)0350	
Melt mass-flow rate (MFR)	Temperature	230
	Load	2.16
	Measured MFR	11

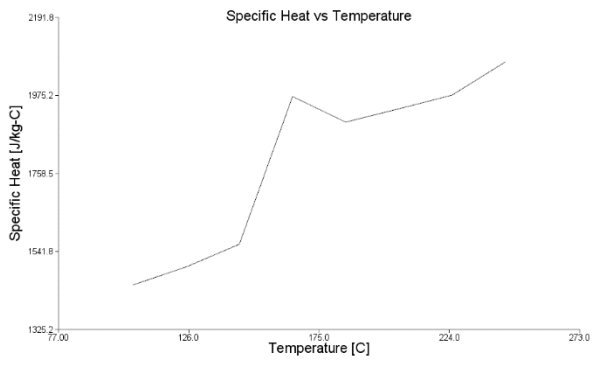
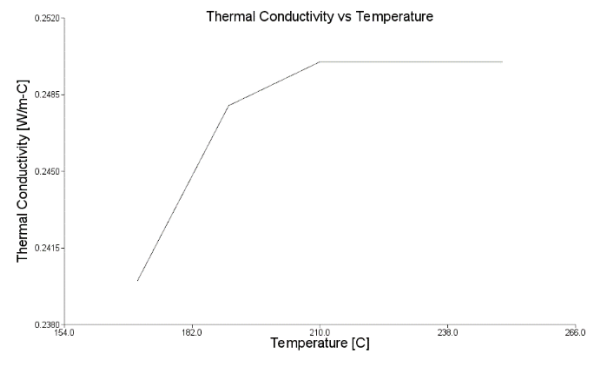
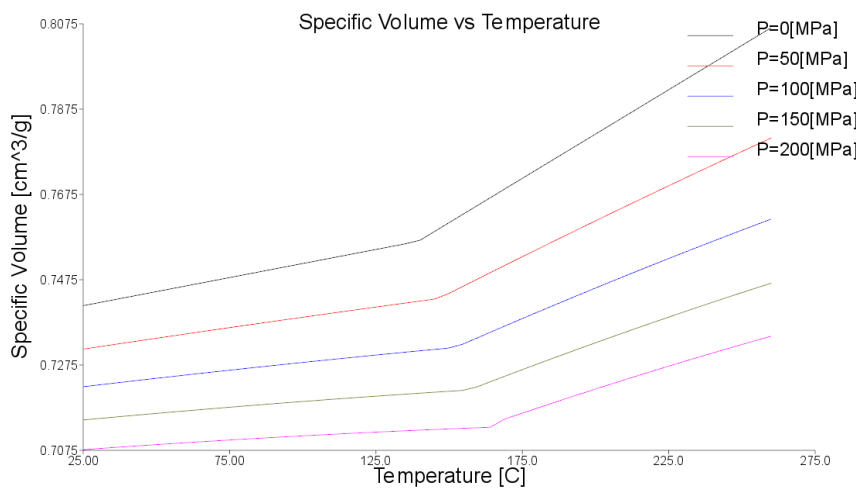
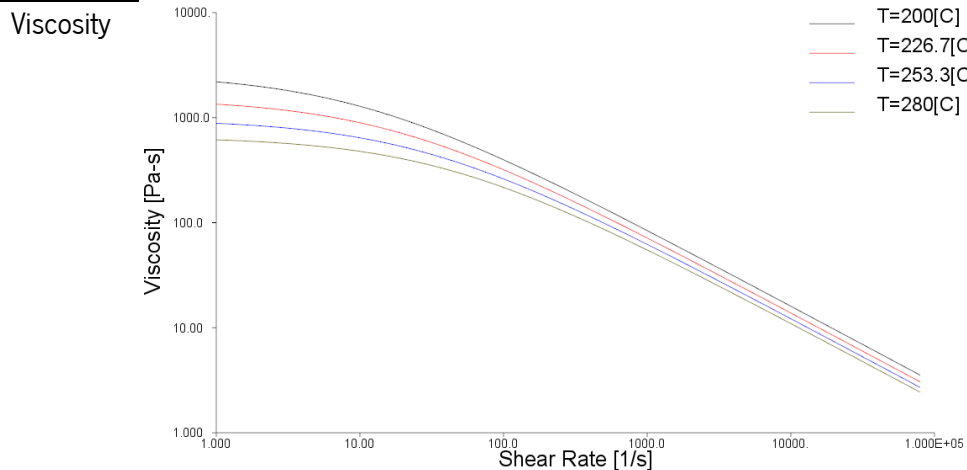
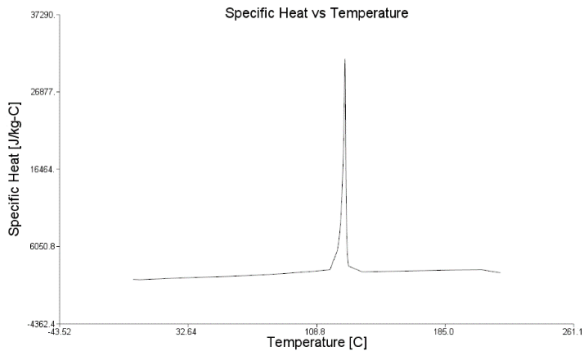
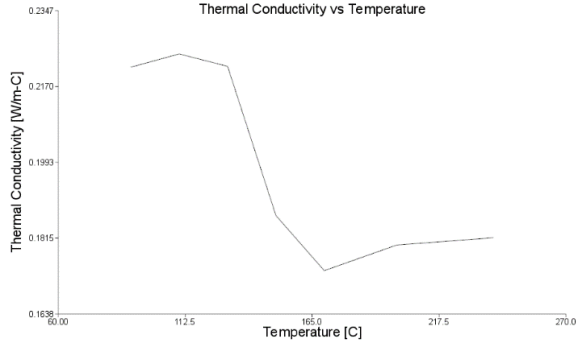
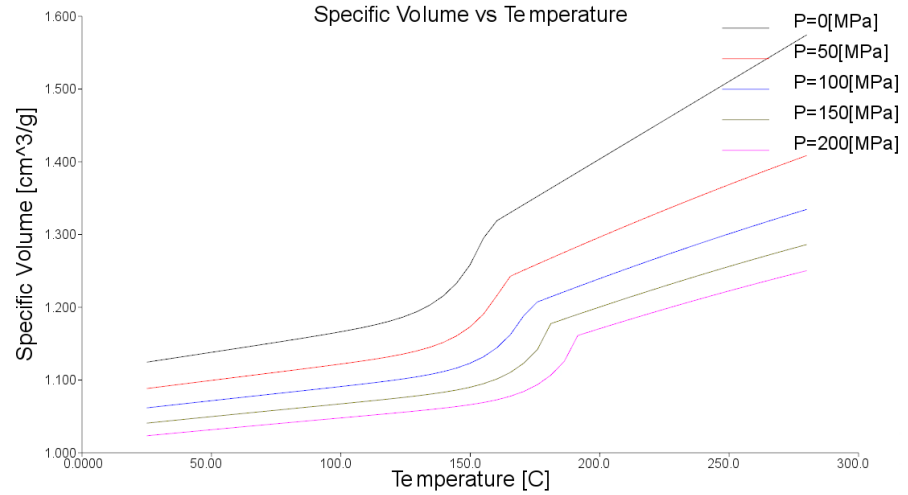
Specific heat data		
Thermal conductivity data		
Melt density	1.2527 g/cm ³	
Solid density	1.3488 g/cm ³	
Specific volume vs Temperature		
		
Elastic modulus, 1 st principal direction (E1)	2100 MPa	
Elastic modulus, 2 nd principal direction (E2)	2100 MPa	
Poissons ratio (v12)	0.35	
Poissons ratio (v23)	0.35	
Shear modulus (G12)	777.778 MPa	
Observed nominal shrinkage	Parallel	Not provided
	Perpendicular	Not provided

Table 26 – Borealis Europe – BD950MO material properties.

Family name	POLYPROPYLENES (PP)	
Trade name	BD950MO	
Manufacturer	Borealis Europe	
Family abbreviation	PP	
Material structure	Crystalline	
Data source	Manufacturer (Borealis Europe) : pvT-Measured : mech-Measured	
Date last modified	08-FEV-13	
Date tested	Not provided	
Grade code	CM15055	
Supplier code	BOREALEUR	
Fibers/fillers	Unfilled	
Mold surface temperature	40°C	
Melt temperature	240°C	
Mold temperature range (recommended)	Minimum	20°C
	Maximum	60°C
Melt temperature range (recommended)	Minimum	200°C
	Maximum	280°C
Absolute maximum melt temperature	300°C	
Ejection temperature	117°C	
Maximum shear stress	0.25 MPa	
Maximum shear rate	100000 1/s	



Juncture loss method coefficients	c1	2.74e-05
	c2	2.098
Transition temperature	129°C	
Moldflow Viscosity index	VI(240)0066	
Melt mass-flow rate (MFR)	Temperature	230°C
	Load	2.16 kg
	Measured MFR	7 g/10min
Specific heat data		
Thermal conductivity data		
Melt density	0.67154 g/cm³	
Solid density	0.88927 g/cm³	
Specific volume vs Temperature		
		

Elastic modulus, 1 st principal direction (E1)	1500 MPa	
Elastic modulus, 2 nd principal direction (E2)	1500 MPa	
Poissons ratio (v12)	0.41	
Poissons ratio (v23)	0.41	
Shear modulus (G12)	534 MPa	
Observed nominal shrinkage	Parallel	Not provided
	Perpendicular	Not provided

Appendix C

Tool_1

Table 27 – Tool_1 with FGS simulation statistics in quasi-static regime.

Simulation regime:	Quasi-static	Model Size:	1016849 nodes
Time to solve:	4h:18min:23s		747593 elements
RAM aviable/used:	128Gb/69.48Gb	Hardware:	Intel Xeon E5-1630 V4 3.7GHz

Table 28 – Tool_1 with FGS simulation statistics in transient regime.

Simulation regime:	Transient	Model Size:	1016849 nodes
Time to solve:	1d:5h:17min:28s		747593 elements
RAM aviable/used:	128Gb/76.78Gb	Hardware:	Intel Xeon E5-1630 V4 3.7GHz

Table 29 – Tool_1 with LGS simulation statistics in quasi-static regime.

Simulation regime:	Quasi-static	Model Size:	1072547 nodes
Time to solve:	3h:55min:22s		784373 elements
RAM aviable/used:	128Gb/65.53Gb	Hardware:	Intel Xeon E5-1630 V4 3.7GHz

Table 30 – Tool_1 with LGS simulation statistics in transient regime.

Simulation regime:	Transient	Model Size:	1072547 nodes
Time to solve:	1d:4h:33min:9s		784373 elements
RAM aviable/used:	128Gb/78.29Gb	Hardware:	Intel Xeon E5-1630 V4 3.7GHz

Tool_2

Table 31 – Tool_2 with FGS simulation statistics in quasi-static regime.

Simulation regime:	Quasi-static	Model Size:	6238316 nodes
Time to solve:	6h:30min:40s		4621388 elements
RAM aviable/used:	128Gb/49.82Gb	Hardware:	Intel Xeon E5-1630 V4 3.7GHz

Table 32 – Tool_2 with FGS simulation statistics in transient regime.

Simulation regime:	Transient	Model Size:	6238316 nodes
Time to solve:	2d:7h:53min:4s		4621388 elements
RAM aviable/used:	128Gb/57.6Gb	Hardware:	Intel Xeon E5-1630 V4 3.7GHz

Table 33 – Tool_2 with LGS simulation statistics in quasi-static regime.

Simulation regime:	Quasi-static	Model Size:	6924415 nodes
Time to solve:	9h:4min:41s		5087005 elements
RAM aviable/used:	128Gb/49.19Gb	Hardware:	Intel Xeon E5-1630 V4 3.7GHz

Table 34 – Tool_2 with LGS simulation statistics in transient regime.

Simulation regime:	Transient	Model Size:	6924415 nodes
Time to solve:	2d:4h:36min:48s		5087005 elements
RAM aviable/used:	128Gb/57.64Gb	Hardware:	Intel Xeon E5-1630 V4 3.7GHz

Fat_1

Table 35 – FAT_1 simulation statistics.

Simulation regime:	Quasi-static	Model Size:	14388814 nodes
Time to solve:	3d:17h:22min:56s		12342144 elements
RAM aviable/used:	128Gb/141.9Gb	Hardware:	Intel Xeon E5-1630 V4 3.7GHz

Fat_2

Table 36 – FAT_2 simulation statistics.

Simulation regime:	Quasi-static	Model Size:	16837392 nodes
Time to solve:	1d:19h:6min:24s		13471777 elements
RAM aviable/used:	128Gb/104.8Gb	Hardware:	Intel Xeon E5-1630 V4 3.7GHz

Fat_3

Table 37 – FAT_3 simulation statistics.

Simulation regime:	Quasi-static	Model Size:	5051377 nodes
Time to solve:	6h:57min:38s		3847551 elements
RAM aviable/used:	128Gb/38.89Gb	Hardware:	Intel Xeon E5-1630 V4 3.7GHz

Fat_4

Table 38 – FAT_4 simulation statistics.

Simulation regime:	Quasi-static	Model Size:	7979105 nodes
Time to solve:	11h:45min:46s		5944782 elements
RAM aviable/used:	128Gb/51.71Gb	Hardware:	Intel Xeon E5-1630 V4 3.7GHz

Fat_5

Table 39 – FAT_5 simulation statistics.

Simulation regime:	Quasi-static	Model Size:	13947829 nodes
Time to solve:	1d:12h:46min:18s		10825340 elements
RAM aviable/used:	128Gb/96.7Gb	Hardware:	Intel Xeon Gold 6244 - 3.6GHZ