Product development process aided by numerical simulation applied to vehicle components

Rafael K. P. Santos[1], Marcos V. H. Silveira[2], Modesto H. Ferrer[3]
[1, 2, 3](Department of Mobility Engineering / Federal University of Santa Catarina, Brazil)
Received 03 January 2023; Accepted 17 January 2023

Abstract: - Considering technology development and the raise in computational power, numerical methods became important device in developing new products and in improving existing ones, being a differential for market competition among companies, consequently from the reduction of time and cost in projects. This work has the main goal of presenting the development steps of a structural component of the suspension system, the steering knuckle, for Formula SAE competition vehicles, aided by computational aided engineering. Starting from the geometrical and functional configuration of the vehicle, a multibody model was generated by the Altair Motion View software to simulate high performance maneuvers, and then estimate two cases of loads. The first case estimates the critical loads which it would be submitted. The second evaluates high frequency loads, to assess a cyclic load case. A preliminary geometrical model of the suspension knuckle was created, serving as base to simulate the topological optimization through the finite element method. At this stage, the project has suffered iterations until the conception that meets the structural criteria, evaluated from the load cases, and reaching its final geometry. Finally, the final product was evaluated by simulation regarding the component durability on the cyclic load case.

Keywords: -Computer aided numerical simulation, finite element method, product development process, steering knuckle, suspension system

I. INTRODUCTION

To meet customer demands for performance, weight reduction in vehicle has been the main requirement in structural parts development. In the case of suspension parts, this new demand in terms of optimization has lead in many cases to higher complexity parts [1]. Product development methods and strategies have been converged to most of the biggest automobile companies, meeting downsizing requirement, in addition to cost, quality and time, by the use of computational aided engineering (CAE) software extensively in their processes.

Virtual models and simulation have provided important data and knowledge for development, like mass estimation, systems functionality, load cases and preliminary shapes, even in the initial phases of projects. Furthermore, it represents a cost and time reduction for physical prototype tests evaluation, and validation, becoming an important competitiveness factor for the company.

Formula SAE is a student competition that challenges students to conceive, design, fabricate, and compete with small formula style racing cars. In this kind of series, the use of CAE tools is even more important, due to limited resources in vehicle development, and to compensate lack of experience or team historical data of past competitions and projects.

Motivated by that scenario, this present work main objective is to show and propose a simulation based design method of suspension structural part, the steering knuckle, for Formula SAE completion vehicle, using different Computer Aided Engineering tools and steps in this process.

II. STEERING KNUCKLE

The steering knuckle is the structural part that connects different components responsible for vehicle control, including suspension control arms, steering, and tie rod connections, wheel hub and brake caliper.

To meet high-volume series production vehicles, these components must satisfy a number of requirements along their life [2]: Service life and function; geometry and connection to adjacent components; material properties (corrosion, strength, stiffness, heat resistance, weight, etc.); design, ergonomics, assembly, disassembly; environmental factors, recyclability, disposal; legal requirements.

In another way to continually develop lightweight steering knuckles, new materials and manufacturing processes have been used, which can provide improved strength and fatigue properties. These include Austempered Cast Iron (ADI) and spheroidal graphite cast iron alloys with increased amounts of silicon and boron [2]. Components manufactured with those materials have increased more than 25% in toughness and dynamic strength. Furthermore, high silicon cast iron has better distribution of mechanical properties in different sections of geometry, increasing toughness and machinability.

The subject of this work has been studied for many authors. Klava [3] developed the steering knuckle using traditional product development process, passing through concept, preliminary evaluation and detailed project, using structural simulation. The author proposes geometry optimization, resulting in a stronger and heavier part.

Ilzhöfer [4] presented an automated structural optimization, using prediction of durability methods, coupled with structural analysis by finite element method. This work considers ride, turns and braking. It was possible to consider load frequency history.

Flesch [5] mentioned in his work details of steering knuckle development for Formula SAE series. It was evaluated thermal and mechanical loads, using analytical and computational methods. Author considered stress distribution in the part to decide regions of mass optimization, but has not considered compliance of final part.

III. METHODOLOGY

Being the established process model in automotive engineering, the V model allows creative and analytical processes, by the extensive use of simulation since the first steps of the vehicle development project. As shown in Fig 1, the 'V model' starts with the specification of the desired complete vehicle characteristics, moving downward for decomposition and specification of subsystems, and then to the design and evaluation of parts. Moving upwards, the systems created out of designed components are tested and validated, reaching the complete vehicle sign-off [6].

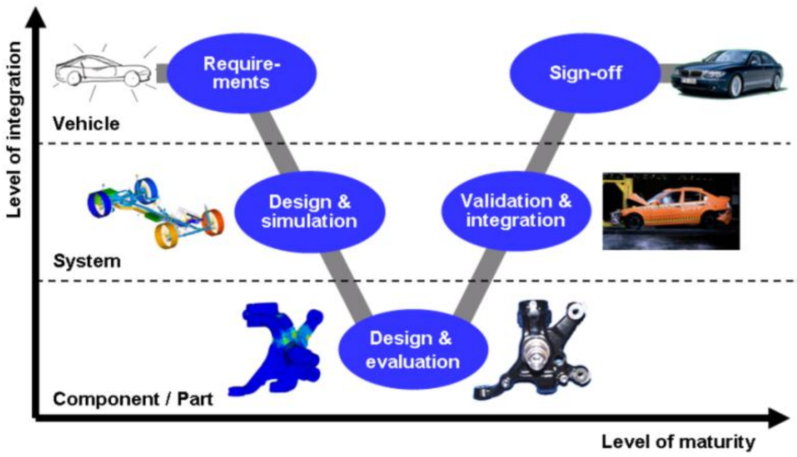


Figure 1. V Model PDP

The V model can be applied in different levels of vehicle development. In suspension products development process, two kinds of simulation are commonly used, the Multi-body dynamics simulation (MBD or MBS) and the finite element method (FEM) for structural analysis and optimization.

MBD has been used in complex vehicle subsystems analysis, composed by a lot of bodies connected, in order to determine relative motions, forces and moments in which those bodies are submitted. This kind of simulation can be used in the first two phases of V model, the Full Vehicle and Subsystem simulation and evaluation. Their outputs will provide data for the component/part design level.

For component design level in the V model, FEM is used to evaluate modal characteristics and structural behaviors, in order to ensure that part will not fail during its life cycle. t

Based on those processes, it was developed a workflow that had driven the steering knuckle development in this work. The structural product development process is presented as follows, in Fig 2.

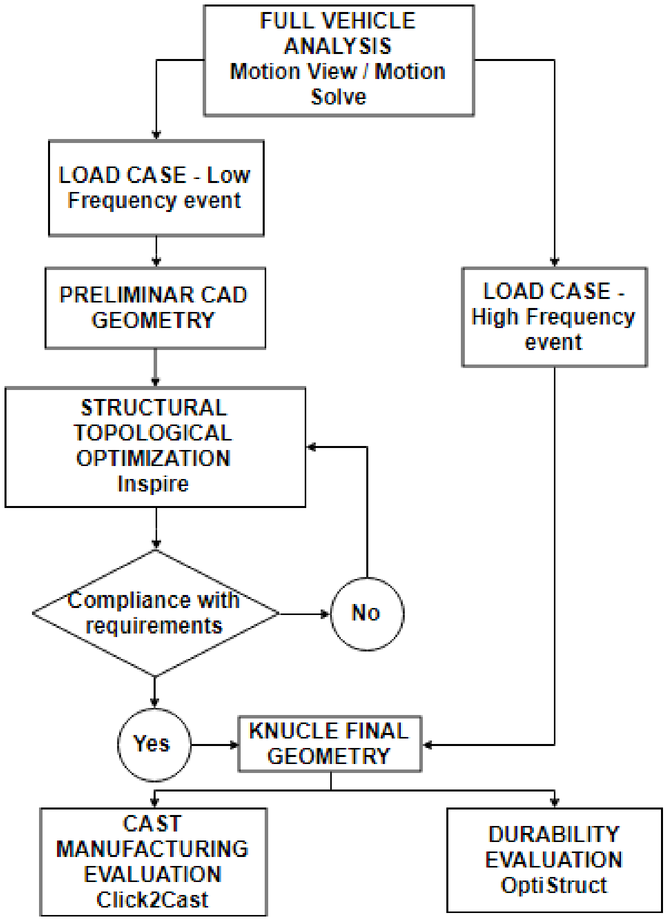


Figure 2. Simulation Based Design Workflow.

Starting from vehicle initial specification, provided by previous studies about suspension geometry, a generic multi-body model was generated in Altair Motion View, adjusting suspension hardpoints coordinates, mass and inertia moment of sprung and unsprung masses from FSAE available vehicle template.

The generated model was simulated in two different situations by Altair Motion Solve. The first one was an extreme case, representing the maximum values of lateral accelerations resulted by a misuse in competition real conditions. The second situation was a standard maneuver, performed in a velocity that represents a high frequency event, found in FSAE competition. Both simulation results were used as inputs for FEM next steps, in the concept development and posterior durability analysis, respectively.

A preliminary CAD model respecting components connection and functionalities was created, to be used as a reference for structural optimization in Inspire software and serve as concept guideline. The optimized geometry is refined, using cast manufacturing design guidelines using the feature of shape smoothing Polynurbs.

After that, the preliminary concept design passes through iterations of static structural analysis, and reaches its final concept geometry that meets structural criteria. Two additional steps for analysis are recommended for this simulation based method, to check part manufacturing feasibility and its mechanical durability.

Casting manufacturing simulation using Click2Cast was performed to evaluate and adjust geometry to its manufacturing process, resulting in pre-molded shape of steering knuckle. Then, fatigue structural simulation was performed to evaluate component number of cycles, and relate to competition endurance performance.

IV. MULTI-BODY DYNAMICS MODEL AND FULL VEHICLE SIMULATION

Multi-body dynamics simulation (MBD or MBS) consists of the study of interconnected bodies by joints, influenced by forces and moments, and restricted by constraints. It can evaluate dynamic behavior of rigid and flexible components [7].

Many previous studies, using MBD analysis, have already been developed for Formula SAE series, and most of commercial software have FSAE template models to encourage the use of those tools.

Altair Motion View was used to create a generic formula SAE model, using full vehicle FSAE library available. Previous studies developed by [8] and [9] were used in this work to adjust suspension hardpoints, mass and inertia of virtual vehicle model. Fig 3 shows MBD model for next analysis.

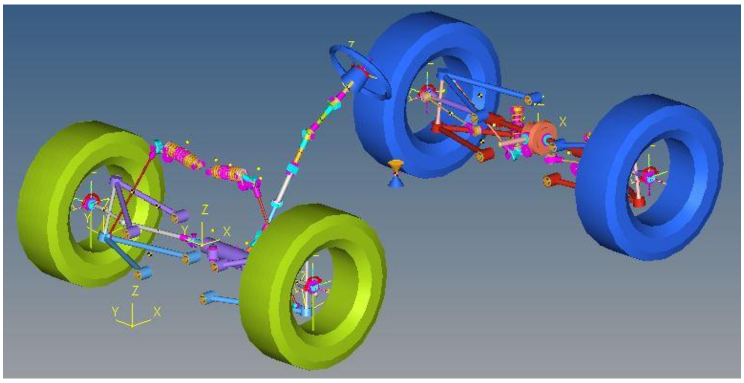


Figure 3. MBD Motion Solve Model.

Based on [10] work, two extreme load cases for static structural analysis and topological optimization were considered. The J-Turn, evaluating cornering and maximum brake force.

The J-Turn maneuver starts in 100 km/h, turning the steering wheel 120 degrees right in 0,2 seconds. Then, after 0,8 seconds, 240 degrees left steering is performed in 0,4 seconds, finishing the maneuver after 3 seconds. The resulting lateral force curve is shown in Fig 4, which maximum peak of force was found in the outer tire, starting the second cornering.

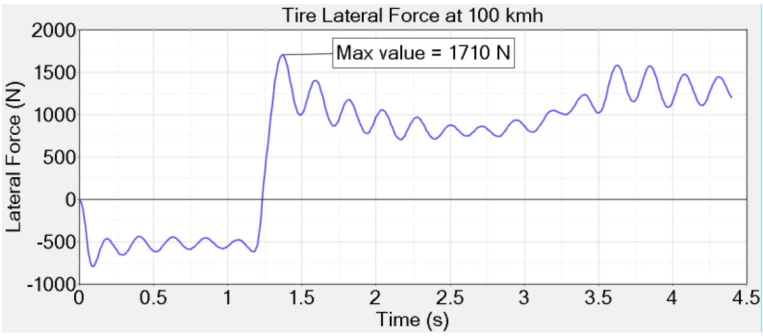


Figure 4. Lateral Force at 100km/h.

Braking maximum force was estimated by analytical model, also based in [10] work. The load cases extracted for static structural analysis for compliance is compiled in Table 1, as follows.

Table 1. Load Cases for static analysis.

Cornering load case			
References	x(longitudinal)	y (lateral)	z(vertical)
Tire contact parch	0	1710	1480
Braking load case			
References	x(longitudinal)	y (lateral)	z(vertical)
Tire contact parch	-1256,775	0	1048
Brake caliper support	0	0	-3700

This case was used to input forces for structural topological optimization, which had driven steering knuckle geometry to a final concept and its validation for static analysis. Respecting Formula SAE competition characteristics of track, it was assumed that the vehicle will have mean velocity of 60 km/h during endurance event. So the J-Turn maneuver was performed, using the same parameters, for this velocity. Fig 5 has shown J-Turn results.

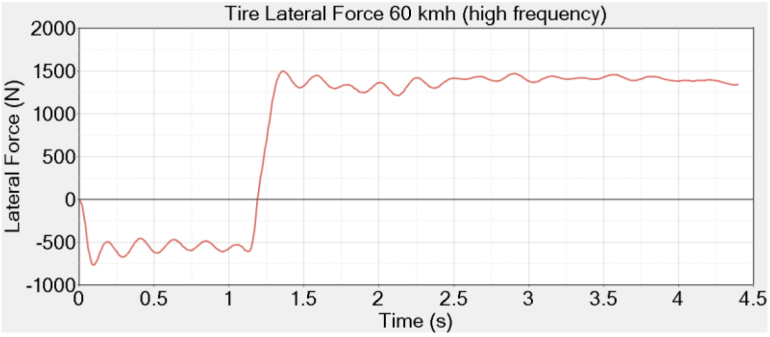


Figure 5. Lateral Force at 60 km/h.

This case had feed durability in fatigue analysis, after knuckle reach its final concept level, to evaluate how many cycles the component will endure.

V. PRELIMINARY CAD GEOMETRY

As mentioned early, preliminary CAD has served as reference for steering knuckle concept development, assuming its assembly and functionality constraints. Fig 6 shows the baseline assembly model for finite element model.

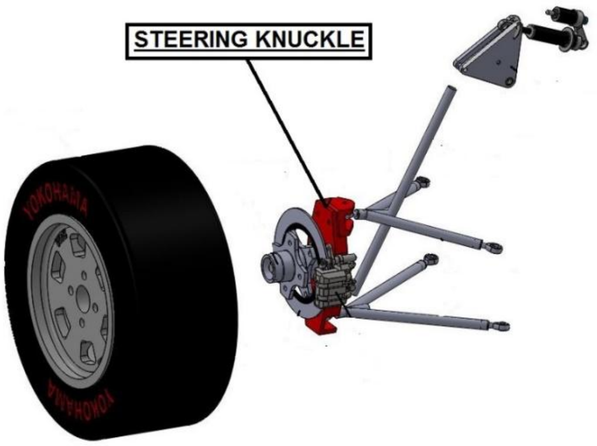


Figure 6. Baseline assembly for steering knuckle mount.

The steering knuckle, shown in red, is composed by SAE 1045 steel spindle axle and steering knuckle main body, considered in next analysis that has made by GJS - 600 Cast Iron, the mechanical properties can be seen at Table 2.

Table 2. Mechanical Properties of Nodular Cast Iron GJS-600

Material Properties – GJS600		
Property	Value	Unit
Young Modulus	174	GPa
Density	7.2	kg/dm^3
Poisson's Coefficient (v)	0.275	
Yield Tensile Strengh	600	MPa

The Fig 7 shows Steering Knuckle assembly in detail. This preliminary CAD model is the base for topological optimization in Inspire and new concept generation.

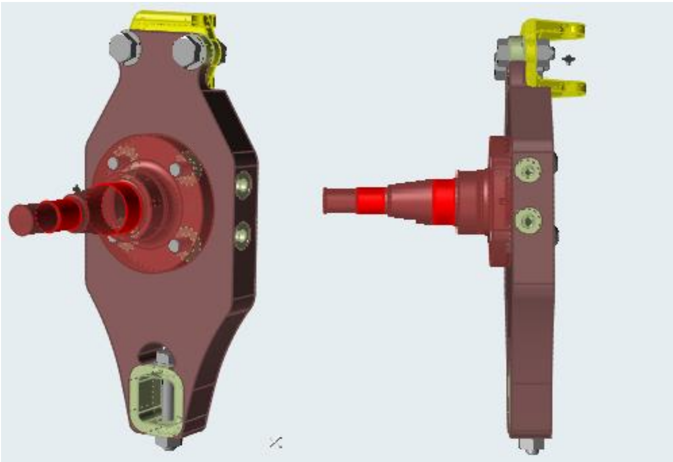


Figure 7. Steering Knuckle

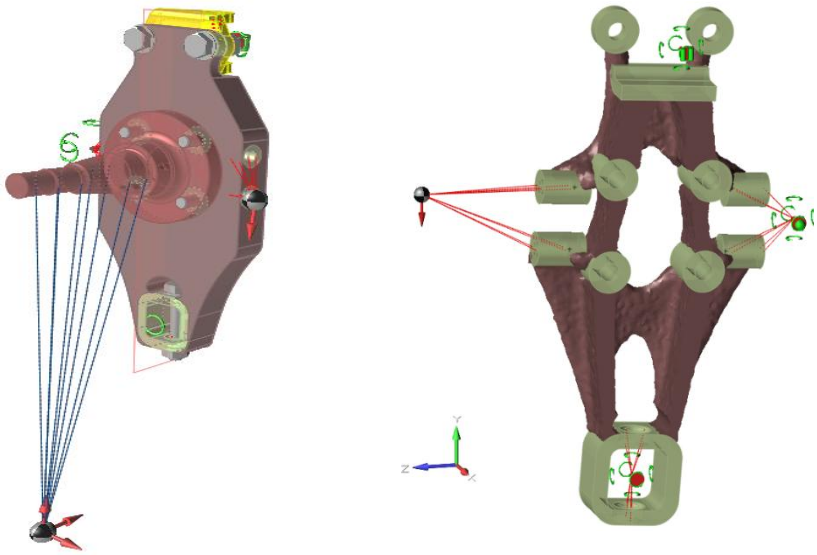
VII. STRUCTURAL TOPOLOGICAL OPTIMIZATION

Based on CAD assembly constraints, the finite element model was created in Inspire 2017. Suspension components, in exception of steering knuckle, were changed to rigid connections, respecting joints, dimensions, and force point locations, in order to reduce complexity of model and reach the nearest possible physical representation.

Load cases presented in Table 1 were applied in respective force point locations, the tire contact patch representation and caliper support. The FE model in Inspire is shown as follows in Fig 8(a).

The software package of Altair Engineering has one tool dedicate to concept design, it is the SolidThinking INSPIRE. This software offers the structure optimization, so in this project the topology optimization would be used.

The Inspire solver calculates stiffness geometry for each load case, and excludes elements that are not necessary in the structure [11]. This exclusion respects manufacturing pattern settings, which was casting in this work. The resulting model in optimization is shown in Fig 8(b).



(a) (b)

Figure 8. (a) Inspire FEM Model, (b) Optimized Shape in Inspire Simulation.

That geometry can be smoothed by Polynurbs tool, present in Inspire software, to give a real product shape. Then, new shape can be evaluated in FEM static analysis, using mechanical structural criteria. The load cases used in structural simulation were the same, and criteria were Von Mises Stress, due to material ductile characteristics. Fig 9 and 10 represent cornering and braking contour plot results.

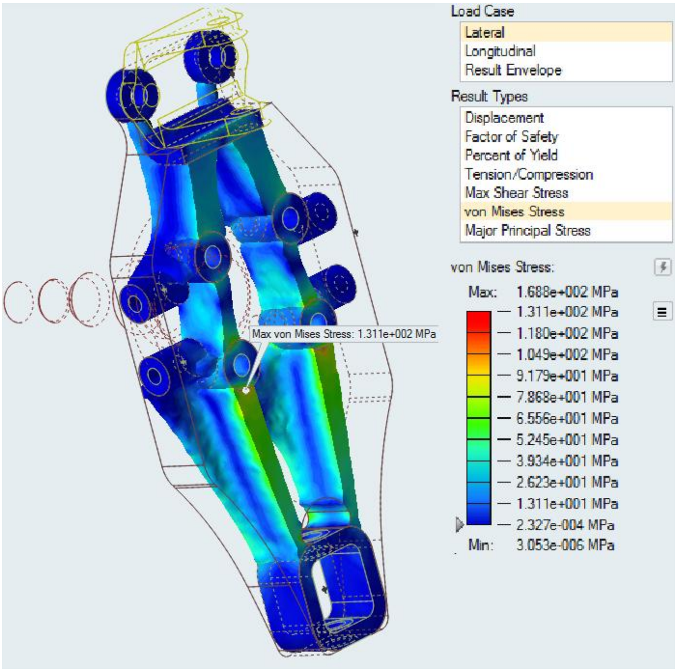


Figure 9. Cornering results.

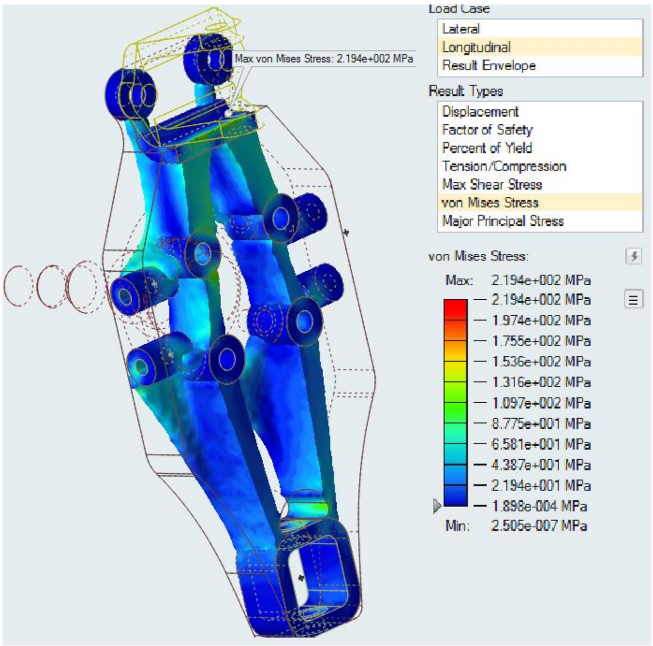


Figure 10. Braking results.

Taking both Von Misses Tensile results, the maximum stress supported by component was 219,4 MPa, resulting in Safety Factor of 2,73 for ductile static failure criteria of Von Mises – Hencky. Part is still over dimensioned for static case but there was a great weight reduction of 70%, from 4,3 kg to 1,38 kg. So, the new part achieves its final geometry.

VIII. DURABILITY ANALYSIS

This full model of the static simulation could be export to other Altair HyperWorks platform, the HyperMesh, this finite element preprocessor defines the condition for durability design to solve on OptiStruct [12]. The load history comes from multibody dynamic results. The critical cycle for Formula SAE track is the maximum deceleration longitudinal in a straight braking and then enter a maneuver of Fish-Hook, it would be maximized the lateral and normal forces. The load history can be seen at Fig 11.

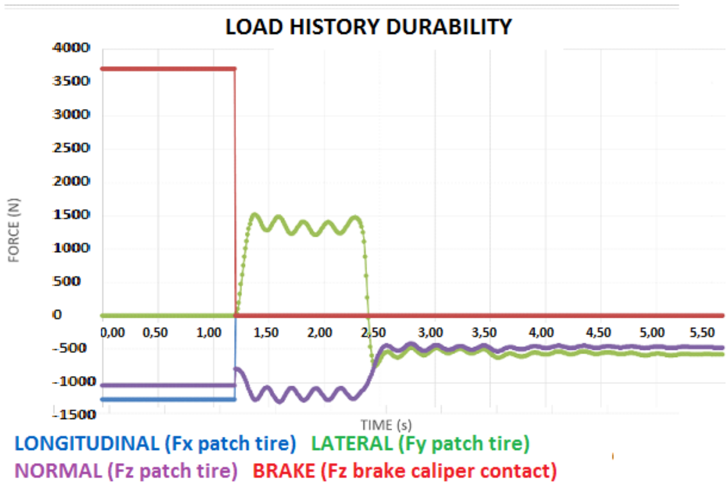
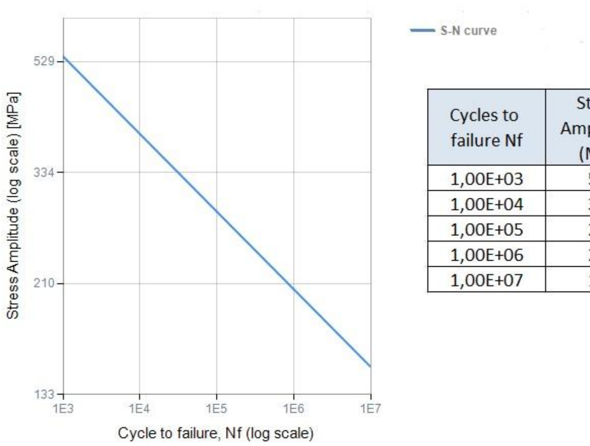


Figure 11. Graph load history

The cycle loads indicate a multiaxial situation, so the critical plane methodology was chosen to measure the multiaxial accumulate damage by Goodman, more conservative method. The Cast Nodular Iron GJS-600 has the SN curve at Fig 12.



Cycles to failure Nf	Stress Amplitude (Mpa)
1,00E+03	540
1,00E+04	391
1,00E+05	284
1,00E+06	206
1,00E+07	149

Figure 12. SN curve of GJS-600.

The cast part rough surface was considered. The durability calculation ran and estimates the maximum damage around 0.0005, where the spindle axis was fixed on steering knuckle, as shown in Fig 13. The life would finish around 1950 cycles, according to the result shown in Fig 14 and 15.

The average characteristic of the endurance FSAE track has 9 situations like a Fish Hook maneuver per lap, then 9 cycles per lap. Based on this prediction, the steering knuckle didn't accumulate any plastic damage before 216 laps, around 10 endurances event.

Figure 13. Fatigue Damage Results

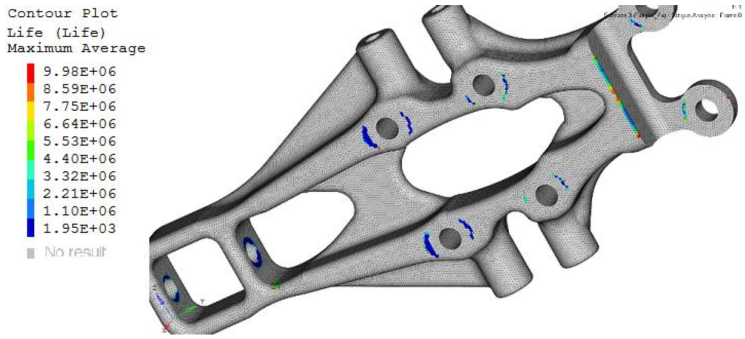


Figure 14. Fatigue Life Results (the color blue is the warning zone with less life cycle)

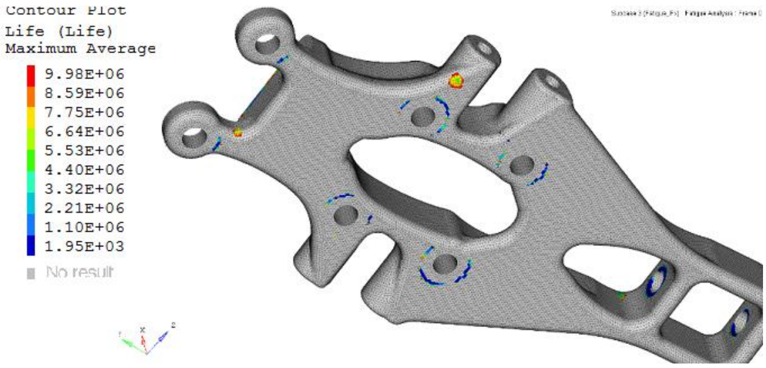


Figure 15. Fatigue Life Results (the color blue is the warning zone with less life cycle)

IX. CAST MANUFACTURING ANALYSUS

The gravity cast manufacturing was available the viability through the Click2Cast software. Click2Cast is casting process simulation software, based on finite elements. The software can analysis some methods of cast process, as gravity die, gravity sand, investment, high pressure, low pressure die casting and tilt pouring. It allows to enhance and optimize their manufactured components avoiding typical casting defects such as air entrapment, porosity, mold degradation, cold shots, mold filling and solidification simulation. The focus is to find a way to minimize the porosity inside the part, minimize the refuse of with less feeder volume and runner. This concept manufacturing design study too the best position of the inlet and part position as gravity direction. It doesn't consider the contraction percent.t.

In first way, the cast simulation focus in solidification process, then it could be the pre-design of feeders, analyzing the final zone of solidification. This process could be done in INSPIRE manufacturing tools and Click2Cast. The first analysis could be seen in Fig 16.

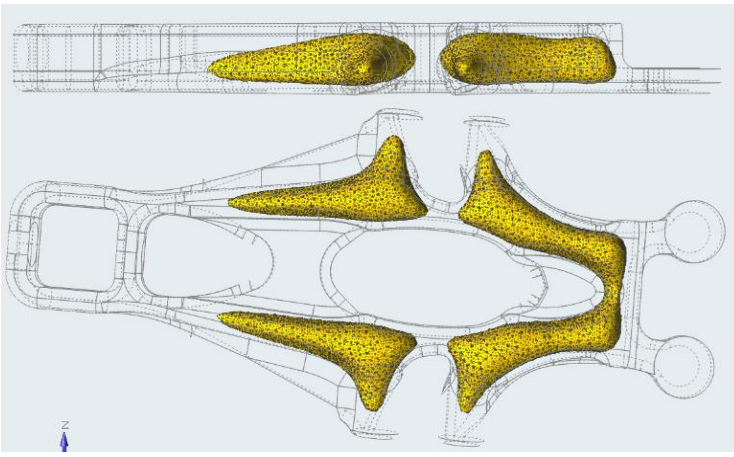


Fig 16. Cast simulation solidification process, final zone.

The filling process in Click2Cast was done and verify the best place to define the inlet. The feeders were design in specific regions, that increase the mass 68%, but it eliminated the high porosity level inside the part. The porosity volume results get the maximum value about 300 mm³, then it can be seen in Fig 17.

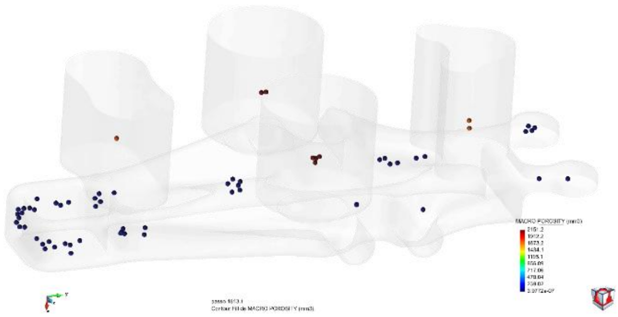


Fig 17. Cast Simulation filling process – porosity volume

X. CONCLUSION

The main objective of this paper was to present a simulation based design methodology for structural parts, aiming Formula and Mini Baja SAE teams, showing the steering knuckle development for a generic FSAE car as an example. The conclusions reached through this paper are listed, as follows:

- For Formula SAE and Baja SAE Teams, and fresh racing teams, which would exist a gap in experimental data and limited budget for product development, the use of CAE software can be the main strategy to be competitive, decreasing time in physical tests;
- The proposed methodology was based in the best practices and state of art in suspension parts development, which shows the importance of students using CAE tools, preparing futures engineers for current market;
- Time expended in creative phase of concept for structural parts can be reduced, by using optimization simple CAE tolls, meeting structural requirements and doing lighter components, resulting in performance;
- Design for manufacturing can be done fast;
- Suggested future work: physical testing and simulation results validation, improving current methodology.

REFERENCES

[1] A. S. de Farias, T. C. de Oliveira, *Suspension support developed in high silicone ductile iron – A practical application* (SAE Brasil. 2010).

[2] B. Heissing, M. Ersoy, *Chassis Handbook – Fundamentals, Driving Dynamics, Components, Mechatronics, Perspectives* (Wiesbaden, ViewegTaubner Verlag, 2011).

[3] B. Klava, *Desenvolvimento das etapas de análise e otimização estrutural de uma manga de eixo*, TCC, Universidade Federal de Santa Catarina, Florianópolis, 2003.

[4] B. Ilzhöfer, Shape Optimization Based on Parameters from Lifetime Prediction, *NAFEMS Seminar: Fatigue Analysis*, Institut für Maschinenkonstruktionslehre und Kraftfahrzeugbau, Universität Karlsruhe, FE – Design GmbH, Germany, 2000.

[5] E. L. Flesch, *Projeto de um par de mangas de eixo automotivas para a fórmula SAE*, TCC, Universidade Federal do Rio Grande do Sul, Porto Alegre, 2010.

[6] J. Weber, *Automotive Development Process - Processes for Successful Customer Oriented Vehicle Development* (Dortrecht, Springer, 2009).

[7] *Modeling and Simulation of Full Vehicle for Analyzing Kinematics and Compliance Characteristics of Independent* (Macpherson strut and Semi Independent suspension system. Pag. 17).

[8] *Second International Conference on Emerging Trends in Engineering - SICETE* (Page Dr.J.J.Magdum College of Engineering, Jaysingpur).

[9] L.B. de Menezes, *Estudo no desenvolvimento de suspensão duplo A e aplicação em veículo de classe formula SAE*, TCC, Universidade Federal de Santa Catarina, Joinville, 2016.

[10] R. K. P. Santos, *Desenvolvimento de produto para sistema de suspensão de veículo de competição*, TCC, Universidade Federal de Santa Catarina, Joinville, 2016.

1. Altair, *University Academic Program. Practical Aspects of Finite Element Simulation: Study Guide* (3 ed. Altair Engineering Inc., 2015).
2. Altair, *University Academic Program. Practical Aspects of Structural Optimization: Study Guide* (2. ed. Altair Engineering Inc., 2015).