Generative Design with solidThinking Inspire

Author: Khai Lam Le

Supervisor: Mathew Vihtonen

Examiner: Mirja Anderson

Degree Programme: Plastics Technology

DEGREE THESIS	
Arcada	
Degree Programme:	Plastics Technology
Identification number:	16565
Author:	Khai Lam Le
Title:	
Supervisor (Arcada):	Mathew Vihtonen
Commissioned by:	Mirja Andersson

Abstract :

This thesis presents a computational method generating 3D models based on topology optimization named solidThinking Inspire. The generative modeling procedure is launched with a set of objective conditions and an initial frame. Basing on the prescribed constraints, the program will produce a new shape for 3D model that has optimal performance, mass-reduction and incredible structure. The technique perfectly inspires all designers to disrupt all the baselines of ordinary modelling.

In the thesis, there are analytical comparisons about modelling methods and physical results between solidThinking Inspire and SolidWorks. The 3D models will be designed in SolidWorks. Then solidThinking Inspire will operate the optimization process. MakerBot Replicator+ and Formlabs Form 2 in Arcada lab are two 3D printers being used to print out the prototypes. From the optimizing results, it is clear that the solidThinking Inspire is able to transform a design with diminished weight and unchanging performance. However, It can be concluded that solidThinking Inspire 2016 currently should be utilized as an add-in for other 3D CAD software. Because there is a lack of technical delicacy during sufficing the optimizing shape. In some particular designs, this limit will induce flawed final prototypes.

Keywords:	solidThinking Inspire , Generative Design , Topology Optimization, Coffee Cup Holder.
Number of pages:	61
Language:	English
Date of acceptance:	

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FOREWORD

I wish to express my sincere gratitude to Mr Mathew Vihtonen for his valuable supervision and guidance throughout this thesis. I also feel thankful for the support from solidThinking for their license and the AN-CADSolutions Company for pricing info. In addition, I am also grateful to the technical staff Mr Stuart Buddle for the technical advices during my work in the 3D printing laboratory.

I would like to acknowledge Arcada University of Applied Sciences for providing an excellent education and other relevant facilities in order to complete this thesis. Lastly, I would like to thank my family and friends for their continuous encouragement in all time.

1 INTRODUCTION

1.1 Background

In order to developing a design idea, 3D CAD Traditional Modelling is currently first choice of every engineers. However, the traditional modelling method does not support the engineers in controlling the variation in design and optimizing the structure to adapt technical requirements.

Recently, Generative Design has been introduced as an innovative approach to build up 3D model. It has stepped into the new level in CAD technology allowing produce optimum model according to design objectives. At the moment, solidThinking Inspire is the most inventive software in producing generative design of parts.

solidThinking Inspire reflects a 3D computational designing tool basing on topology optimization. The optimization process is appointed the required specification such as loading condition and product's material. The initial structure will be transform into the ideal layout by analysing the applied preference. The final product is optimized with the conditional satisfactory, reduction in resource consumption and incredible visual.

solidThinking Inspire provides engineers a shorter way to approach the design in the more efficient mechanism. In industrial world, this software has ability to escalate the product's optimization such as material reduction in robot prototype or functional automotive parts' design with better quality. Apparently, the competence in computer-aided optimization is in high demand.

1.2 Objectives

- Demonstrate the utilities of solidThinking Inspire.
- Compare generative design to traditional solid modelling design.
- Optimize the designed parts by solidThinking Inspire.
- 3D Print the optimized designs by Formlabs Form 2 and MakerBot Replicator.

2 LITERATURE REVIEW

2.1 SolidWorks Modelling

2.1.1 Overview

SolidWorks is a 3D modelling program allowing users create 3D complex and functional designs exclusively on Windows operating system. This 3D software tools is issued and developed by Dassault Systèmes SOLIDWORKS Corporation. SolidWorks has been one of the prestigious mechanical design software beside Autodesk Inventor and Solid Edge in the current field.

In 1993, the industrial world was heading to the high demand in create 3D solid model with simple computer-aided techniques. Jon Hirschstick who is graduated from Massachusetts Institute of Technology established a 3D CAD technological company with his engineering team [1]. Two years later, SolidWorks was introduced to the world as the innovative 3D CAD software and amazed every engineer with its valuable functionalities. In 1997, Dassault Systèmes, a French enormous 3D design company acquired SolidWorks for $310 million in stock. [2]

Nowadays, SolidWorks has been used in at 23,400 locations in 80 countries with more than 3,073,600 users throughout the world. [2] It is presently one of the leading 3D modelling programs in designing 3D prototypes, constructing the drawing, analytical simulation tools, and variation in extensions with maximized productivity and simple user interfaces. SolidWorks' applications varied from massive industry such as aerospace, automotive and other kinds of large industry to small 3D printing prototypes. It is appropriate to conclude that SolidWorks has made the design steps more flexible and effortless in order to let engineers realize the design concept with required conditions into manufacturing.

SolidWorks 3D CAD contains three packages as Standard, Professional and Premium.

a. The Standard Package

With SolidWorks Standard, it is capable to design a complicated part and assemble parts into the functional product. SolidWorks Standard allows users to capture and modify design content by creating structural components, performing stress analysis and fulfil necessary characteristics [3].

b. The Professional Package

SolidWorks Professional is basically an upgraded version of SolidWorks Standard, which provides valuable proficiency to produce more advanced design. SolidWorks Professional includes:

- SolidWorks Toolbox illustrates a library of standard hardware that addresses the diverse needs of every model assembly [4] [5].
- SolidWorks Utilities provides users the abilities to overview and analyse for geometrical aspects in solid models [4] [5].
- Workgroup PDM help managing projects and remains process on the right track with project specifications [4] [5].
- Task Schedule is the application working independently from SolidWorks 3D CAD. Its function is organizing the tasks and ensuring processes performed on proper time [4] [5].
- SolidWorks Costing enforces the estimation of budget and resources adequately to ensure effectiveness during project execution [4] [5].
- PhotoView has ability to build up the sophisticated photorealistic renderings and animations inside SOLIDWORKS 3D CAD with optimum performance [4] [5].
- FeatureWorks is a method that recognizes and transfers non-SolidWorks-based files to be fully functional in SolidWorks. Additionally, it establishes an advantageous connection between SOLIDWORKS and other CAD programs to improve the design process [4] [5].
- SolidWorks Design Checker ensures and monitors elements such as dimensions, material, assemble and other kind of designing factors against its approved baseline [4] [5].

c. The Premium Package

SolidWorks Premium is the most comprehensive 3D CAD product in the SolidWorks series. It is the combination of SolidWorks Professional and Standard with more further expanding capabilities to manufacture more efficient and better quality products. The Premium Suit contains powerful tools such as:

- SolidWorks Simulation is a testing method enabling engineers evaluates the product designs in virtual environment. This extensive tool considerably diminish the design testing time and escalate the quality of product's assessment comparing to the traditional testing tool. SolidWorks Simulation can tackle with total design issues such as static, fatigue, fluid flow, dynamic and other types of failure mode in spite of the complex geometry of designs [6] [7].
- SolidWorks Motion provides the designers the animation of assembly from simple to complex in order to assure its proper behaviour against external elements.
- SolidWorks Routing is applied to design the routed system such as piping system, electrical cables distribution, engine routing harness and other types of routed design with the rapid processing time and desirable output [6] [7].
- ScanTo3D is the ultimate tool for building up 3D model from input data. By taking ScanTo3D, the users can mix and mesh the imported object in the most effective methods [6] [7].
- TolAnalyst is an acronym of tolerance analysis tool, which ensures the relation in dimensions and tolerances of the assemblies. Furthermore, it is capable to execute tolerance analysis promptly whenever it is required [6] [7].
- CircuitWork is a translating bridge between SolidWorks and Electrical Computer-Aided Design (ECAD) support engineers deliver Printed Circuit Board (PCB) designs between those two applications vice versa [6] [7].

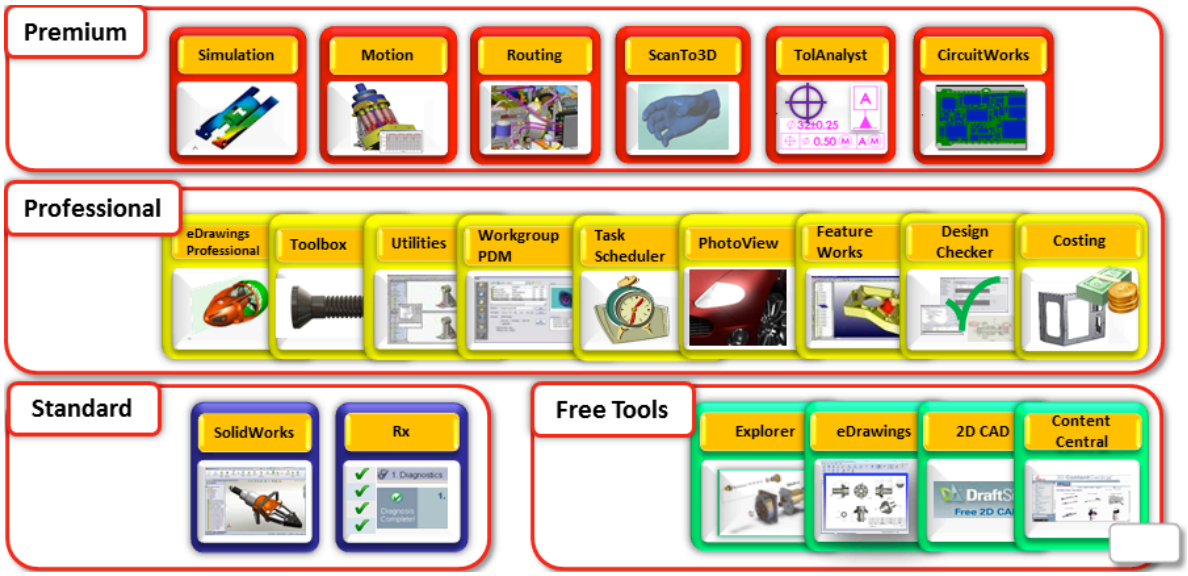


Figure 1 *Overview of SolidWorks packages [8]*

2.1.2 Modelling methods

Like another Computer-Aid Design programs, SolidWorks has been strengthened to sufficiently support engineers constructing 3D models with the best time-reduction and effectiveness. There are three basic steps in forming every 3D solid model:

Figure 2 *Basic steps of 3D modelling.*

In 3D modelling stage, it is crucial to select a plane in one of three basic planes as front, top and right plane. On that chosen plane, a new sketch will be created and custom by drawing geometrics such as lines, shapes and arcs. These sketches must be an enclosed boundary and attached to the origin. With SolidWorks modelling tools, the sketch can effortlessly be transformed into the solid part.

In SolidWorks, there are three sections as Part, Assembly and Drawing which link strongly with each other to provide a flawless result in different perspectives. When we start designing a component in assembly or a product in SolidWorks, it usually begins with various lines in specified plane geometries. These sketches will be generated into a solid part by various methods such as Extruded, Revolved, Loft and other modelling techniques.

For more complex parts, the designer initially creates surfaces in many different shapes depending on the final outcome. Then it is possible to operate the modelling methods on the current design until it meets all requirements.

Furthermore, the stress analysis is fundamental in SolidWorks, which must be performed to ensure the part is able to stand with external conditions. From the stress analysed results, the Factor of Safety is displayed for the designer to recognize fractured spots and determine the adjustment if necessary. The Factor of Safety is basically a division of the maximum stress over the allowable stress. To identify the failed or safe area of the design:

- If the Factor of Safety at an area is 1, the material at that area is on the point of failing.
- If the Factor of Safety at an area is less than 1, the material at that area has failed.
- If the Factor of Safety at an area is more than 1, the material at that area will be safe. [9]

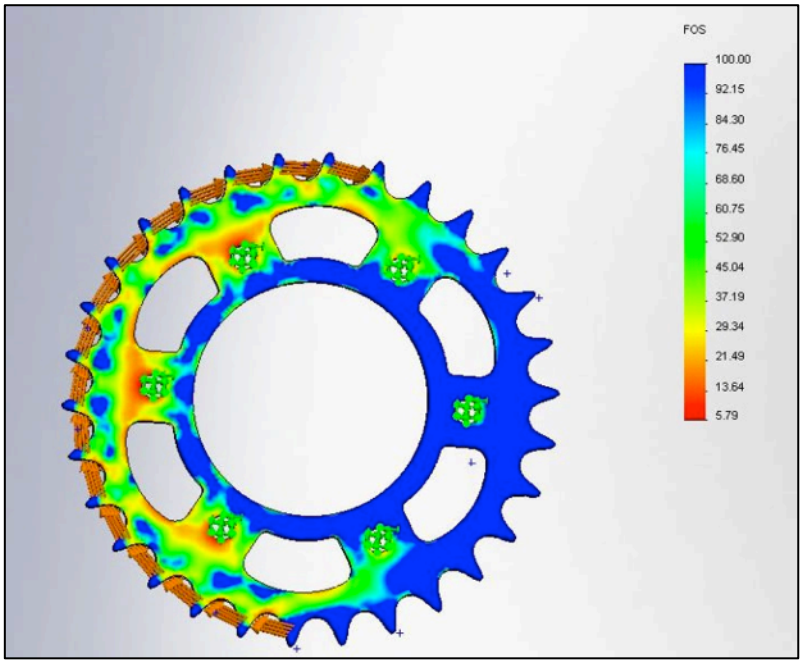


Figure 3 *Example of stress analysis in SolidWorks [10]*

Regarding to Assembly, it simply brings all of single part into a completed prototype. During assembling the parts, SolidWorks interference checking system will let the designer know whether parts performed correctly in the suitable fits. Afterward, the 2D CAD drawing is fundamental to step into the production stage, which can be executed by Drawing.

2.1.3 New enhancements in SolidWorks 2017

Every year at SolidWorks World, the newest version of SolidWorks is presented and released by their production team with abundance of enhancements for users. Likewise, SolidWorks 2017 was introduce with inventions in new featuring tools such as Multi-distance Chamfer, Advanced Hole Wizard…and other supportive functionalities such as Smarter Mate Placement, Transparent Section Views…There are all of tremendous features in SolidWorks 2017 such as:

a. Designing Features [11]

- Bi-directional circular pattern.
- New chamfer-fillet features.
- Transparent section view.
- Thread feature.
- Shaded sketch contours.
- Advanced hole wizard.
- New functionalities for sketch on surfaces.

b. Functionalities in assembly [11]

- Smarter mate system.
- Magnetic mates.

c. Advanced details in 2D Drawing [11]

- BOM Tables in Template
- Further details in 2D drawing outline.

d. Usage Supports [11]

- New enhancements in SolidWorks PDM, SolidWorks MBD and DimExpert.
- Format expansion in eDrawing.
- Establishing tree-house diagram.

Especially in this year, Dassault Systèmes SolidWorks has introduced a brand new cloud-based 3D modelling application, named SolidWorks Xdesign. This program runs on any browser on personal computer or mobile device meanwhile provides superior utilities in 3D modelling. SolidWorks currently launches a Beta Program allowing users go through all aspect of Xdesign. It is expected to be release an official version of Xdesign in 2017, which will turn the 3D CAD world into the new level [12].

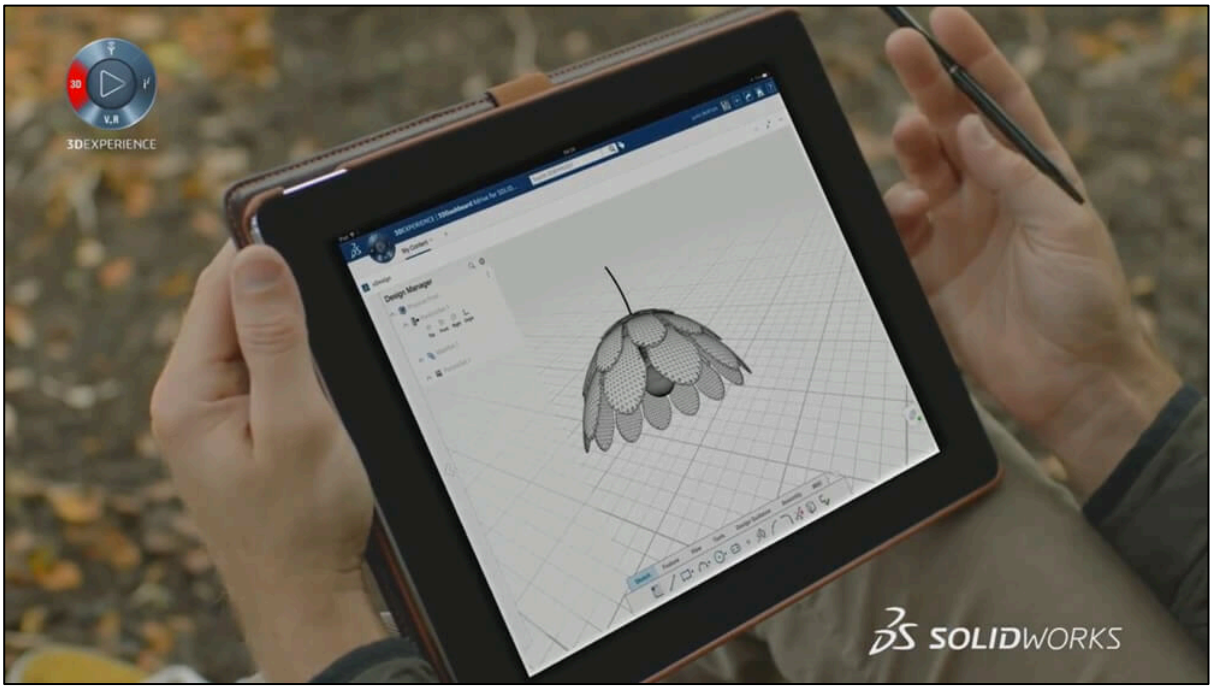


Figure 4 *SolidWorks XDesign runs on iPad [12].*

2.2 solidThinking Inspire Modelling

2.2.1 Overview

solidThinking, have developed by American engineering software company Altair HyperWorks, includes Evolve, Inspire, Click2Cast and Envision. Regarding to 3D optimization tool, solidThinking Inspire is being a leading solution in the field. This tool is a superior alternative for traditional design process with incredible technological power [13].

In 1991, two brothers Alex and Mario Mazzardo and Guido Quaroni develop 3D design project named solidThinking. At the beginning, solidThinking was designed to operated on NEXTSTEP system developing by NeXT. Two years later, it received an award "the best new CAD and 3D program " at NeXTWORLD EXPO held in San Francisco. In 1998, the Windows-based OlidThinking 3.0 was released with new features as the construction history and an extensive NURBS modelling toolset. After a year, solidThinking was able to run on the Mac OS X operating system [14].

In July 2008, US-based technology provider Altair Engineering acquired the company's assets meanwhile the release of solidThinking 7.6. In next 14 months, the generative tool solidThinking Inspire 8.0 and solidThinking 8.0 were introduced to the users. With the next version, the original solidThinking turned into solidThinking Evolve. Today all releases of solidThinking are simultaneously developed for Evolve and Inspire [14].

Nowadays, solidThinking contains four products as Evolve, Inspire, Click2Cast and Envision and constantly expands their product range in design industry.

a. solidThinking Inspire

solidThinking Inspire is an automatic generation tool that supports designer to create the product in the most optimized structure. Opposing to the tradition designing method, Inspire is able to generate the prototype according to pre-allocated external conditions. The product's layout is strong enough to stand under the pressures although there is a significant contraction in the material mass. solidThinking Inspire enables engineers to save processing time, manufacturing budget and material consumption and especially achieve marvellous frame [15].

In the solidThinking Inspire 2016, there are new enhancements that bring the software into higher level:

- PolyNURBS tools.
- Comparison the design performance.
- More advanced load types.
- Surfacing optimization [15].

b. solidThinking Evolve

While Inspire automatically explores the optimal form with desired design concepts, Evolve does not perform as a similar optimizing toolset. It is simply a set of modelling tools to visualize the initial sketch into the realistic model. solidThinking Evolve allows industrial designers to develop forms faster, using either Windows or Mac OS X. It enables you to capture an initial sketch, explore styling alternatives, and visualize products with realistic renderings generated in real time. Evolve provides organic surface modelling and parametric control, with NURBS-based surfaces and solids and a unique ConstructionTree history feature. It frees designers from the constraints of engineering-oriented CAD tools, while allowing the export of digital models required by others in the product development process [16].

c. Click2Cast

Click2Cast is simulation software enabling user strengthen the design part in casting processes. With Click2Cast Simulation, its user-friendly interface and simulation technology support users are able to detect all the typical casting issues effortlessly during the process [17].

d. Envision

Envision is a cloud-based business platform that can integrate perfectly with any third-party program. Furthermore, Envision can optimize productivity and processing performance in order to obtain the solutions for all customers' problem [18].

2.2.2 Applications

In Melbourne, there is an additive manufacturing studio naming HardMarque Future Factories. The Australia-based workshop utilized both solidThinking Inspire and Evolve to re-design a traditional piston to a better version. After the design and manufacturing process, the final result was 23.5 % lighter in weight than the tradition piston. Moreover, solidThinking Evolve and Inspire are obviously able to support and integrate for titanium additive manufacturing process.

Figure 5 *Design and manufacturing process of piston by HardMarque.*

Topology optimization also contributed its power in to the space industry. Altair ProductDesign was chosen to support the RUAG Space Switzerland design and optimized the antenna support arm 3D design for Sentinel-1 satellites. For this specific case, OptiStruct is one of Altair's toolset that was used to deliver an initial optimized layout according to imported conditions during launching or being in space.

From the basic layout, the surface modelling process was conducted with solidThinking Evolve. After 4 weeks, the new design of antenna support arm was generated. It was 3D printed by RUAG's 3D-printing technology provider. As expected, the final product was half of original weight, stronger structure and cost saving for launching into space.

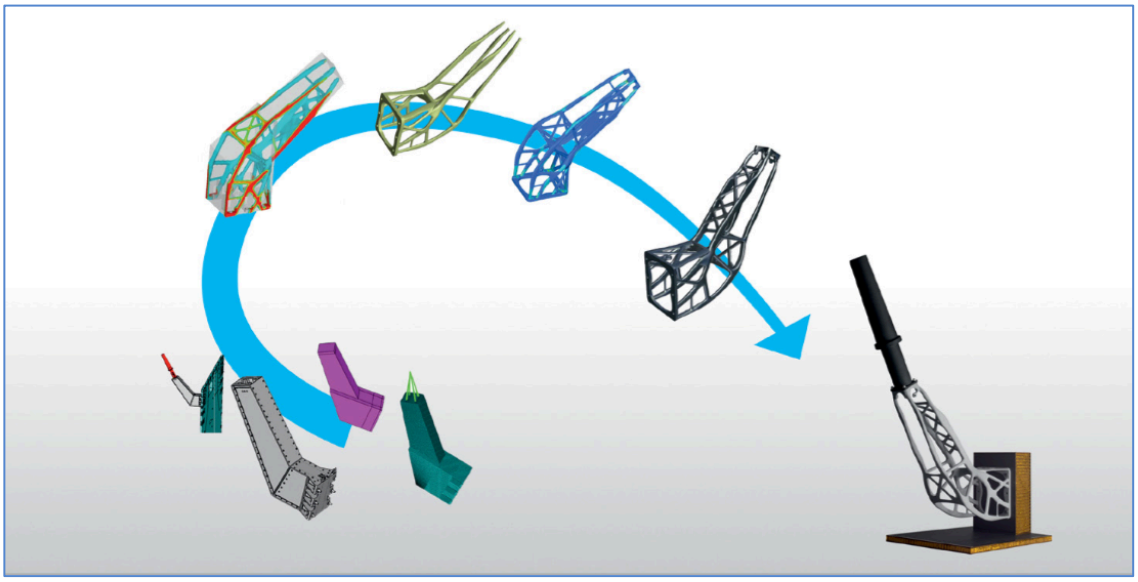


Figure 6 *Design process for antenna support arm for RUAG Space.*

2.2.3 License

In Finland, AN-Cadsolutions Company is the reseller of many 3D CAD software, 3D technological machine and distributes the 3D modelling training programs. The Tampere-based company offers solidThinking Inspire that includes both optimization functions and analysis functions with the Annual Unlimited Campus License, **1495€ / a year** .It contains Unlimited Network Licenses for a campus, node-locked licenses for named students and faculty, and updates, all for a one year period. Regarding to solidThinking Suite (Inspire & Evolve), the Annual Unlimited Campus License price is **1995€ / a year.** AN-Cadsolutions also distributes the commercial version of solidThinking as a below table.

Table 1 *Prices for commercial license of solidThinking.*

Products	**Annual Workstation License**	**Annual Concurrent License**
solidThinking Inspire - Shape (€)	7290	7290
solidThinking Inspire - Shape & Analysis (€)	9750	10970
solidThinking Suite - Inspire & Evolve (€)	10970	12190

Note: Price included VAT 0%

2.2.4 Topology optimization

Topology optimization is a computational method that grants the component's geometry with the most enhancement and minimization in material layout. By usage of topology optimization, engineers are able to eliminate capacity of material that response whole prescribed internal and external condition. As a result, it is essential to modify the optimized component depending on desired realization [19] [20].

a. Advantages

- Solve the time-consuming issue in design processes.
- Avoid the redundancy in designed parts.
- Construct the lightweight and material-optimized design, which results the remarkable cost-reduction.
- Decrease the pre-manufacturing tests such as compression test, fatigue test …etc.

b. Drawbacks

- Limit the development of the prototype due to deficiency in manufacturing capabilities.
- Unpredictable and interlaced design structure.

2.2.5 Features

a. Home Section

- Files: open and save the new files.
- Measure: measure the models' parameter such as lengths, widths, depths and angles.
- Move: drive and rotate the models.

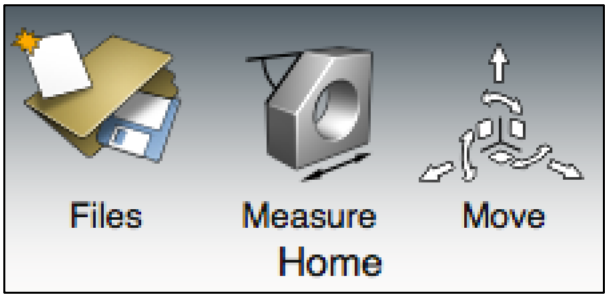


Figure 7 *Home section in Geometry tab*

b. Create Section

- Basic sketch tools: Points/Lines, Rectangles, Circles, Arcs, Trim / Break.
- Patch: fill in or remove the poor surfaces.
- PolyNURBS: construct the free-form solid structure from the post-optimized model.

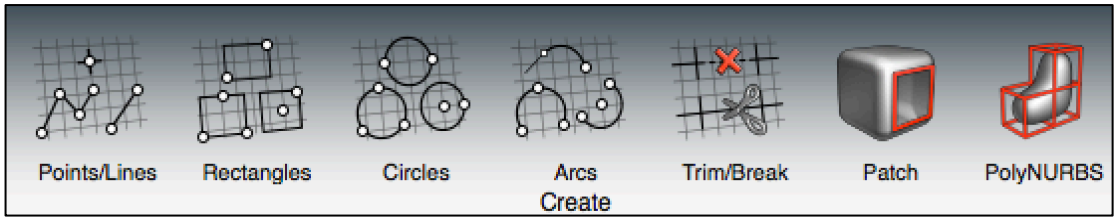
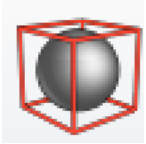
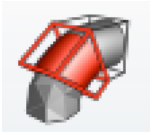
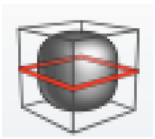
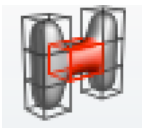


Figure 8 *Create section in Geometry tab*

Table 2 *Functionalities of PolyNURBS Toolset*

PolyNURBS Tools	Function
	Create a solid PolyNURBS object.
	Develop the PolyNURBS mass along the post-optimized structure.
	Add or remove the PolyNURBS cage.
	Create a loop over the existing PolyNURBS cage.
	Establish a bridge or opening a hole between two Poly-NURBS object.
	Sharpen the edges of PolyNURBS mass.

c. Modify section

- Push/Pull: extrude or revolve the surfaces.
- Mirror: create the parts across symmetry planes.
- Scale: modify the parts' size.
- Boolean: combine, subtract or intersect two different parts.
- Cut: cut the parts.
- Simplify: solve all the problematic geometry before analysing.
- Midsurface: generate the midsurfaces from solid parts.

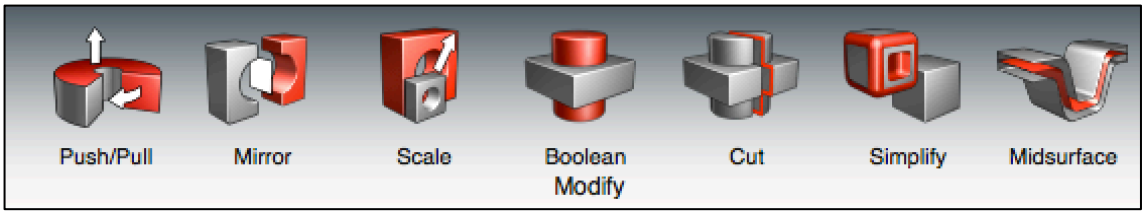


Figure 9 *Modify section in Geometry tab*

d. Connection Section

- Fasteners: to detect holes for bolting or screwing in the parts.
- Joints: to detect holes for allocating pins or slide pins in the parts.
- Contacts: to detect the touching area of two different parts.

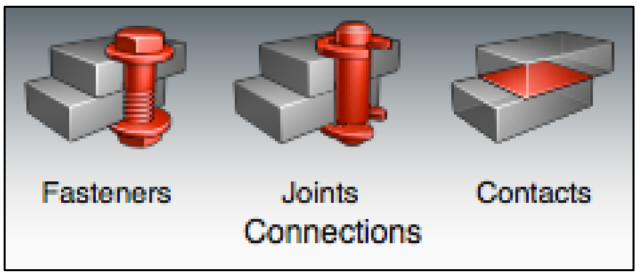


Figure 10 *Connection section in Structure tab*

e. Setup Section

- Loads: apply the specific loads onto the models.
- Disps: apply the displacement onto the models
- Accels: apply the gravity, angular velocity and acceleration.
- Temps: apply temperature loads onto the models.
- Materials: determine the models ‘material.
- Masses: setup the concentrated mass onto a random point or parts.
- Shape Controls: modify the design mass basing on specific purposes.

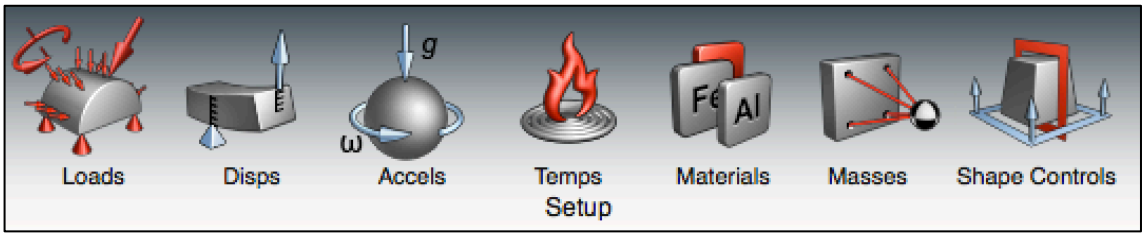


Figure 11 *Setup section in Structure tab*

f. Run Section

- Analyse: to run the analysis on the models.
- Optimize: to perform the optimization process.

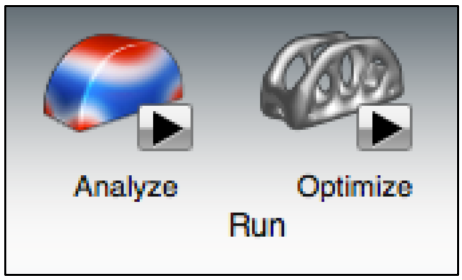


Figure 12 *Run section in Structure tab*

2.2.6 Modelling methods

Step 1: Create the part in solidThinking Evolve or Inspire or import the parts from another 3D CAD software.

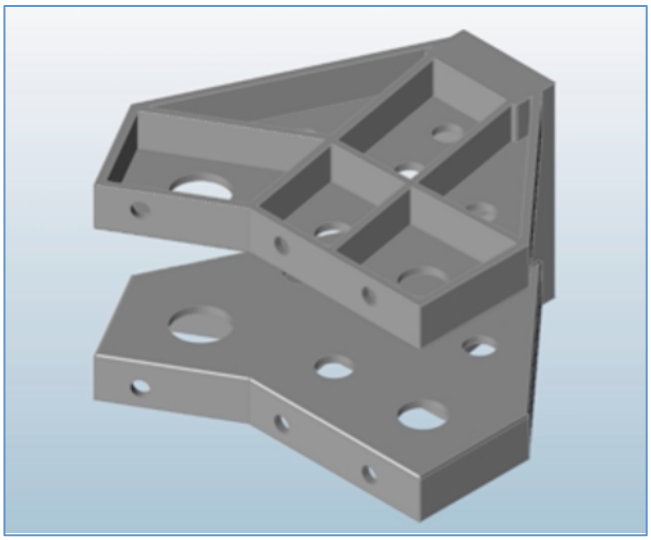


Figure 13 *Create or import models into solidThinking Inspire [21].*

Step 2: Simplify the part that allowing users to perform design configuration.

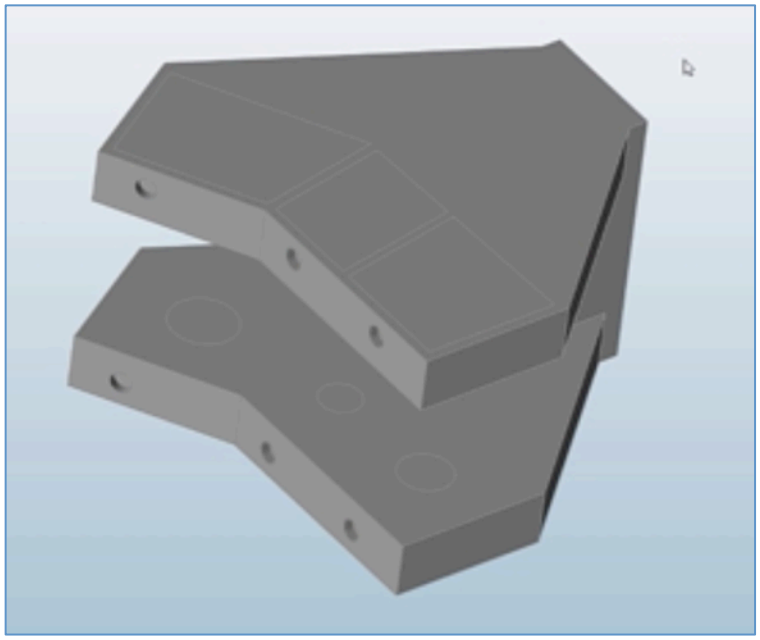


Figure 14 *Simplify the models [21].*

Step 3: Set all the features, material, external and internal conditions for the part.

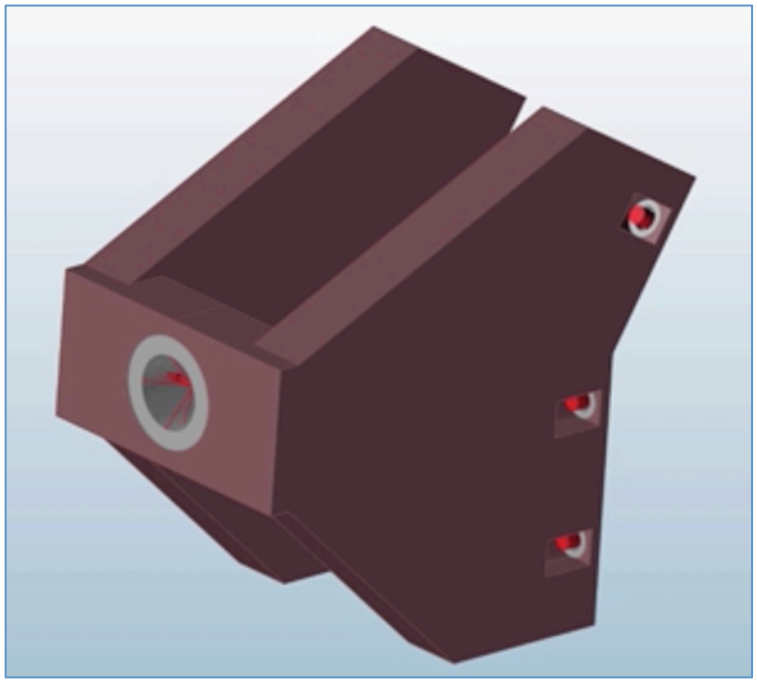


Figure 15 *Assign the external conditions for models [21].*

Step 4: Produce the optimized layout which base on prescribed conditions.

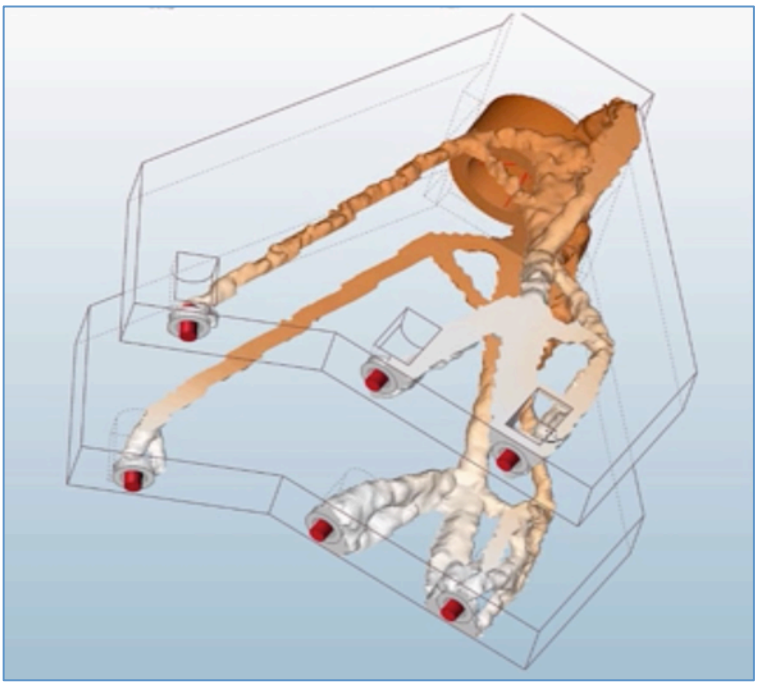


Figure 16 *Optimization process [21].*

Step 5: Build up material mass on the optimized layout with PolyNURBS toolset.

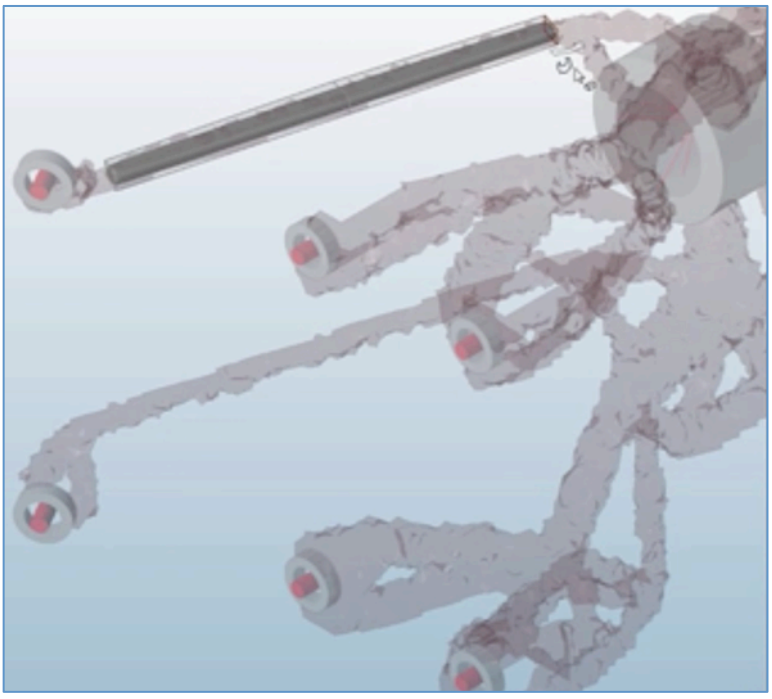


Figure 17 *Develop PolyNURBS cavity* [21].

Step 6: Execute demonstration to analyse the design performance.

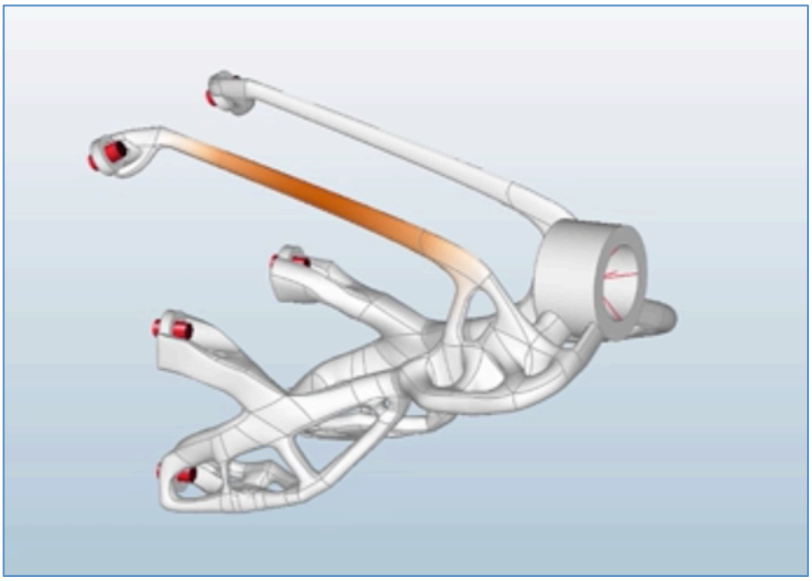


Figure 18 *Analysis in the model performance [21].*

2.3 MakerBot Replicator 5th Generation

MakerBot Replicator 5th generation is a desktop 3D printer developing by a company name MakerBot in New York, US. MakerBot Replicator produces the prototypes basing on Fused Deposition Modelling printing technology. It has 11 % larger build volume than MakerBot Replicator 2. Using this printer, users can directly print a part via a desktop or a mobile device with multiple CAD formats support. MakerBot Replicator provides The Smart Extruder that has a new sensor system. Whenever printing process get jammed, the extruded will pause and enable to recovery the print. In addition, The Smart Extruder+ can be another choice for more reliable performance and improved results. MakerBot Replicator 5th generation is a very worthy tool aiding design workflow with multiple purpose and affordable price [22].

Figure 19 *MakerBot Replicator 3D printer [22].*

2.4 Formlabs Form 2

Form 2 is an SLA 3D printer developed and manufactured by a US-based company Formlabs. In contrast to MakerBot Replicator+, its mechanism is using an ultraviolet laser to solidify a thin layer of resin in the resin tank. Thereafter, the print head is lifted up to have wiper clear detritus. This creates a solid layer and keeps repeating layer by layer until forming a prototype. Before printing, the software Preform would allow users to manipulate and assign necessary supports for 3D models. Compared to other SLA 3D printers, Formlabs Form 2 is able to deliver excellent results with sharp details and smooth surface in its price range [23].

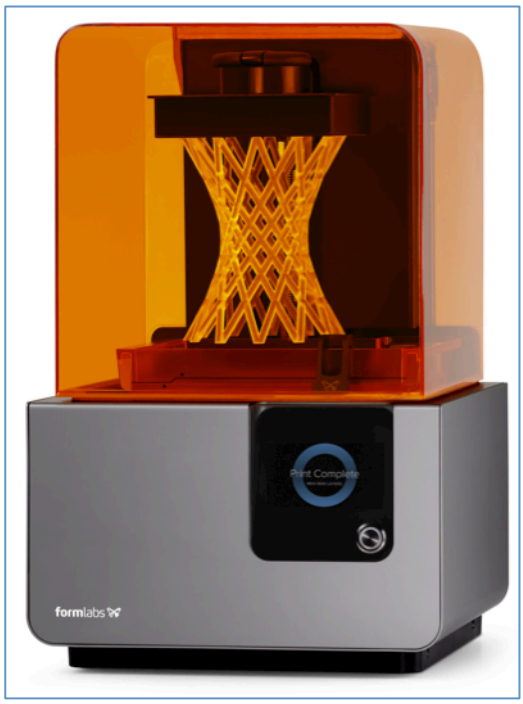


Figure 20 *Formlabs Form 2 3D Printer [23].*

3 METHOD

3.1 Phone holder

At the very beginning step, the initial design of a phone holder was established in SolidWorks, as **Figure 21** The sketch of Phone holder in SolidWorks. Afterward, the sketch was extruded with 10 cm depth and filleted with 1 cm radius, as **Figure 22**.

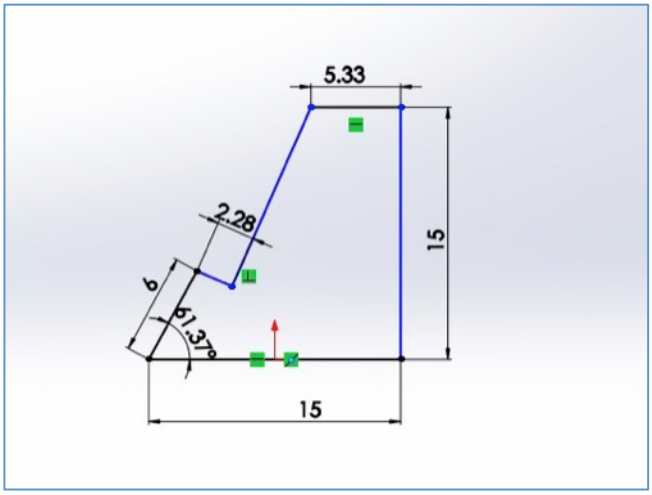


Figure 21 *The sketch of Phone holder in SolidWorks.*

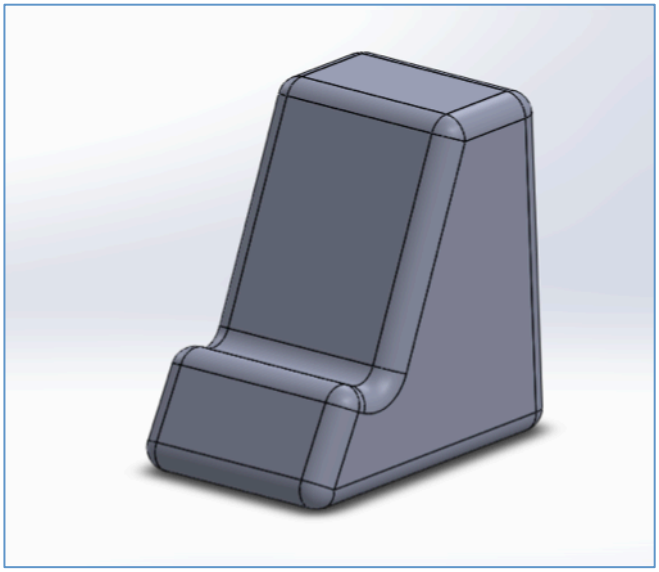


Figure 22 *Phone holder in SolidWorks.*

After saving the slprt file, the model was opened and transformed into solidThinking Inspire in order to perform the optimized process. It is crucial to set the support on the bottom and the loading condition of 50 N on the desired surfaces, as in ***Figure 23***.

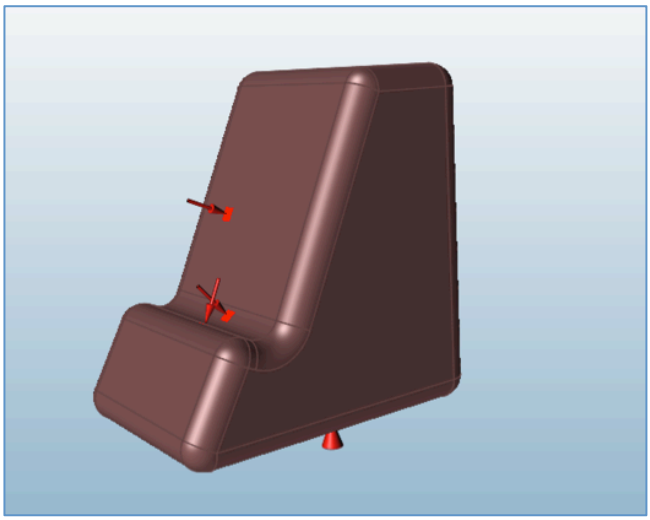


Figure 23 *Loading conditions of Phone holder in solidThinking Inspire.*

Based on the prescribed conditions, the software processed optimization with required parameter as in ***Figure 24*** and extracted an optimized frame as in ***Figure 25***.

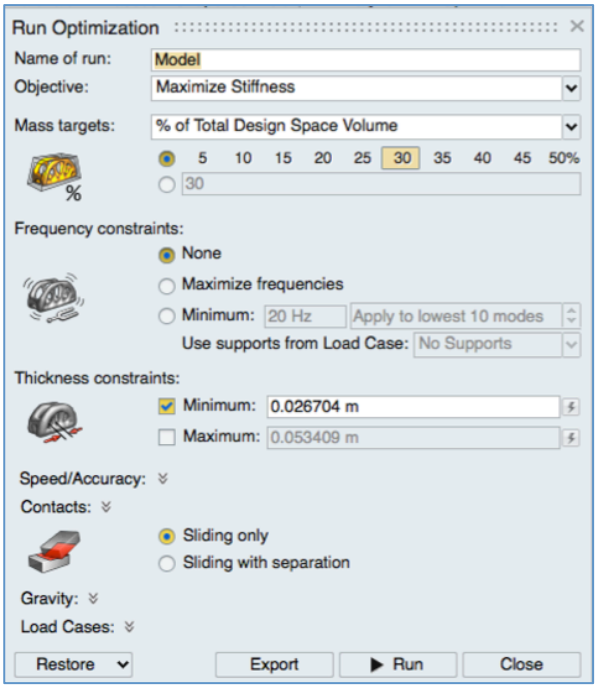


Figure 24 *Optimizing parameters of Phone holder.*

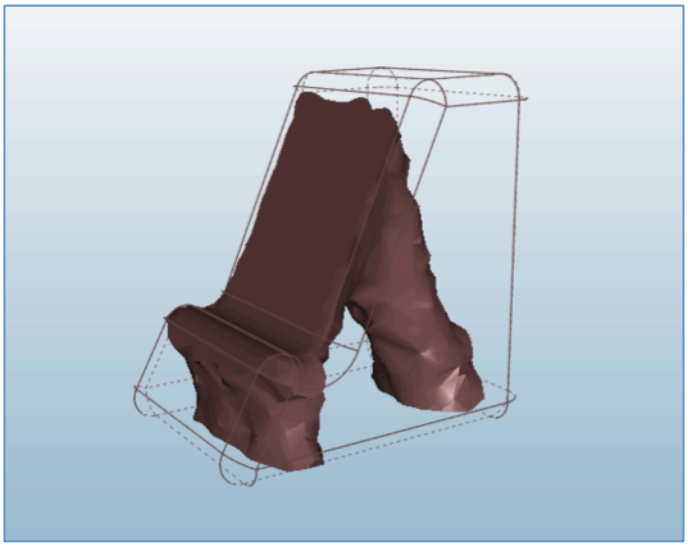


Figure 25 *Optimized result of phone holder design.*

Then the PolyNURBS tool was used to fill all the material mass to accomplish a model. Firstly, **Wrap** tool was used to create blocks within the optimizing frame as shown in ***Figure 26***.

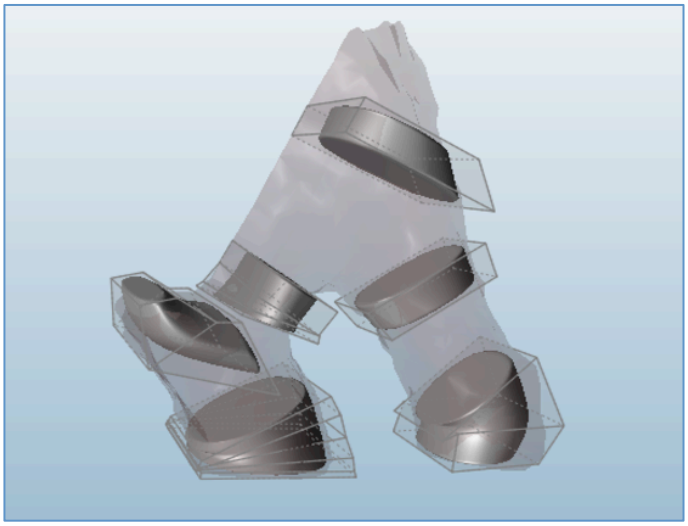


Figure 26 *Wrap PolyNUBRS mass for phone holder optimized frame*

Then we can connect those blocks by **Bridge** and adjust some details of design with +/- and **Sharpen**. Until the material mass amount to optimizing frame, final design is completed and displayed in ***Figure 27***. Thus, it is crucial to perform the simulation analysis for the model by **Analysis Explorer** tools to ensure the performance as ***Figure 28***.

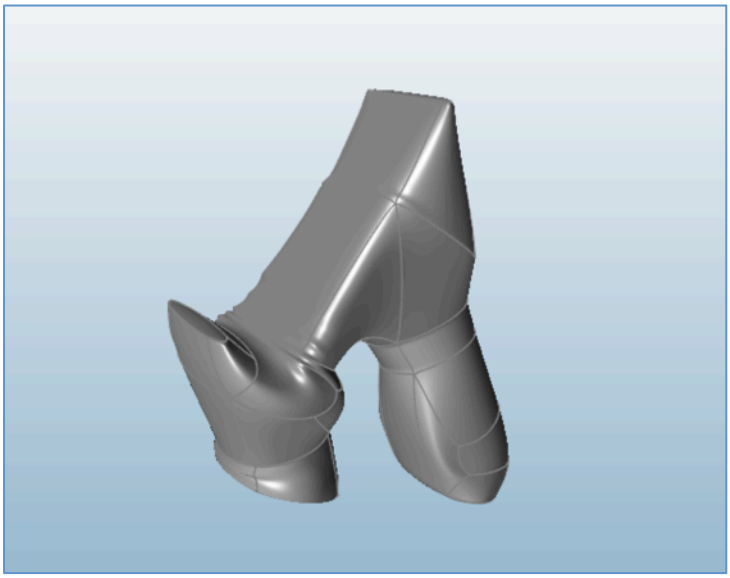


Figure 27 *Final optimized phone holder design.*

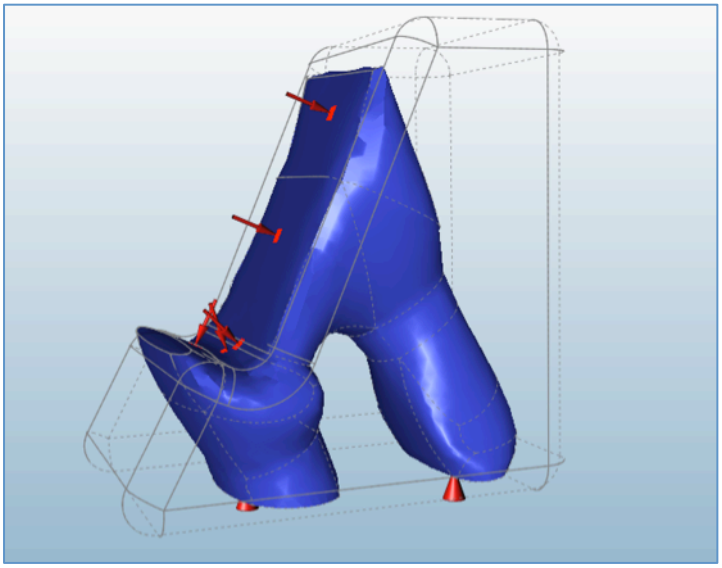


Figure 28 *Simulation analysis of phone holder.*

3.2 Push-up Equipment

In SolidWorks, a square with 35 mm side was laid out as **Figure 29** on the top plane, and then extruded with 6mm thickness.

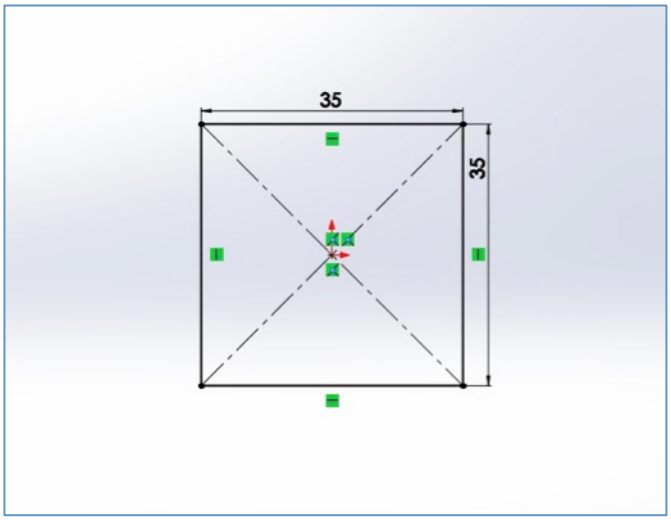


Figure 29 *First sketch of the Push-up equipment.*

On the right plane, the 6mm-thicked-handle as **Figure 30** was modelled by Extrude tool.

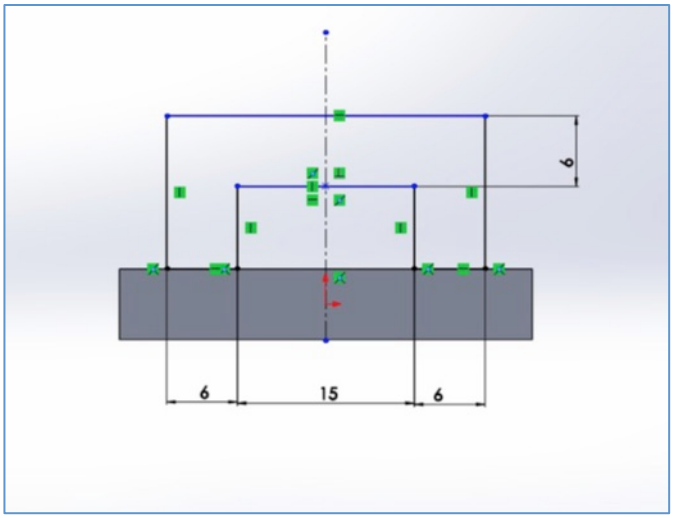


Figure 30 *Push-up handle sketch in SolidWorks.*

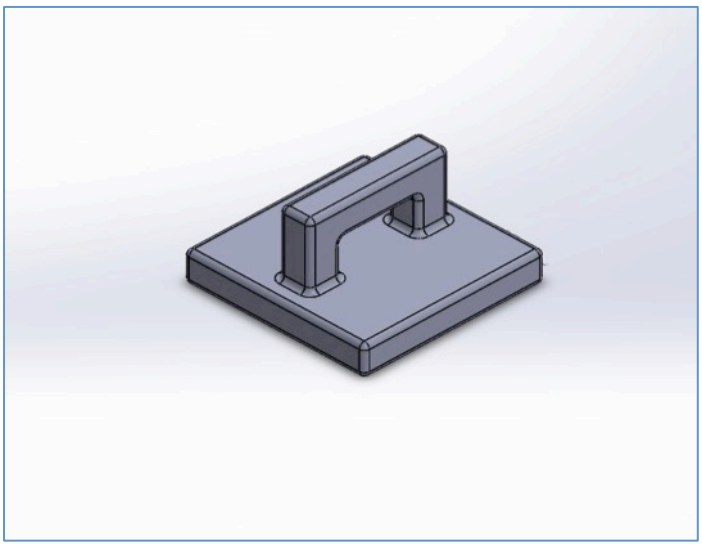


Figure 31 *Final design of Push-up equipment.*

Afterward, the file was transformed into solidThinking Inspire in order to perform the optimized process. The handle of the Push-up equipment was assigned to withstand the load of 500 Newton and be supported from the bottom surface as in ***Figure 32***.

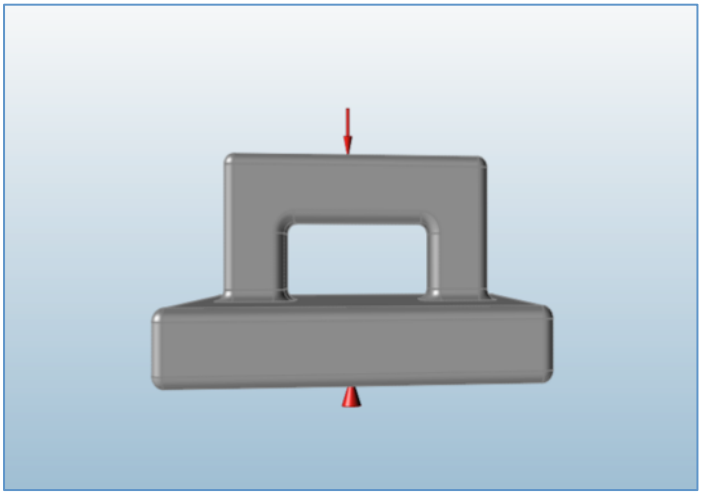


Figure 32 *Loading conditions for Push-up equipment.*

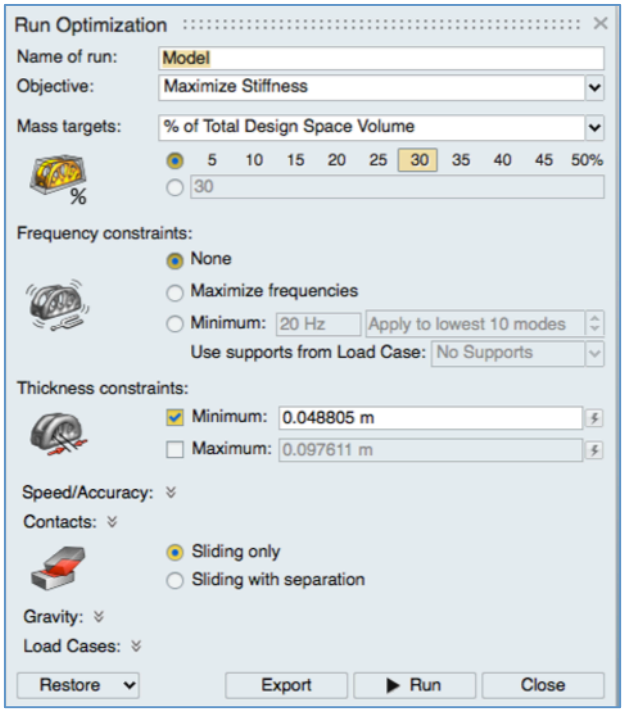


Figure 33 *Optimizing parameters of Push-up equipment.*

By prescribed conditions and parameters, the optimization was processed and generated the ideal shape.

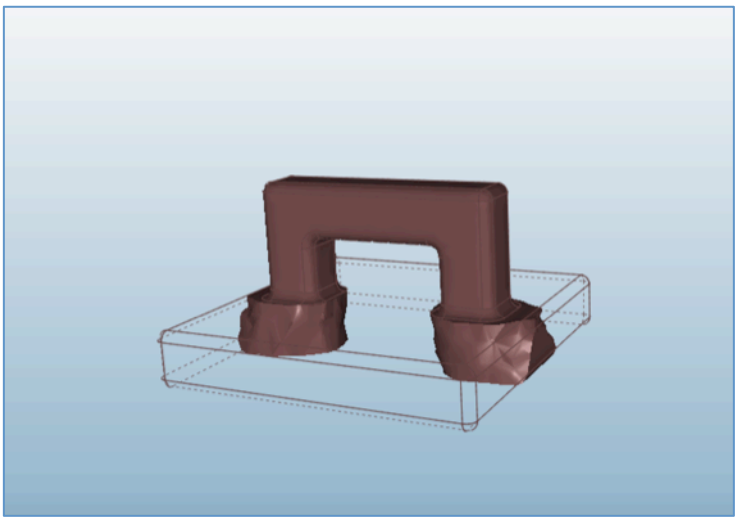


Figure 34 *Optimized result of Push-up equipment design.*

In order to fill in the ideal shape, certain blocks were formed by **Wrap** in **Figure 35** and connected those blocks by **Bridge**. As a result, the design is completed and shown as ***Figure 36***. At last, the simulation analysis was conducted to assure the performance of model with **Analysis Explorer** tools as in ***Figure 37***.

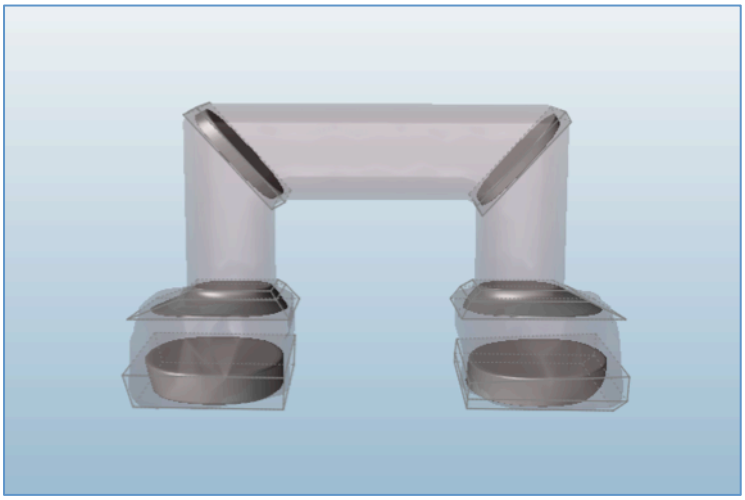


Figure 35 *Wrap PolyNUBRS mass for Push-up equipment optimized shape.*

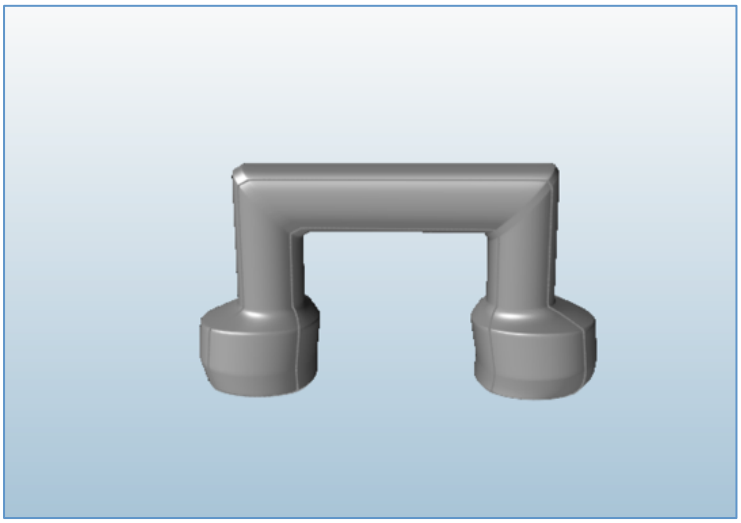


Figure 36 *The finished design of Push-up equipment.*

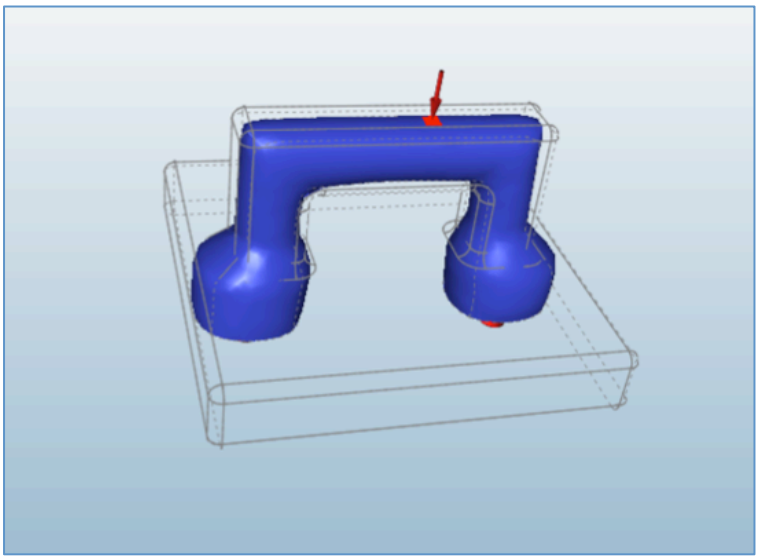


Figure 37 *Simulation analysis of Push-up equipment*

3.3 Coffee Cup Holder

The design objective is creating a holder that is able to bear a common paper coffee cup. SolidWorks is the chosen 3D modelling program for designing the coffee cup holder. The dimension of a paper coffee cup is shown in **Figure 38**.

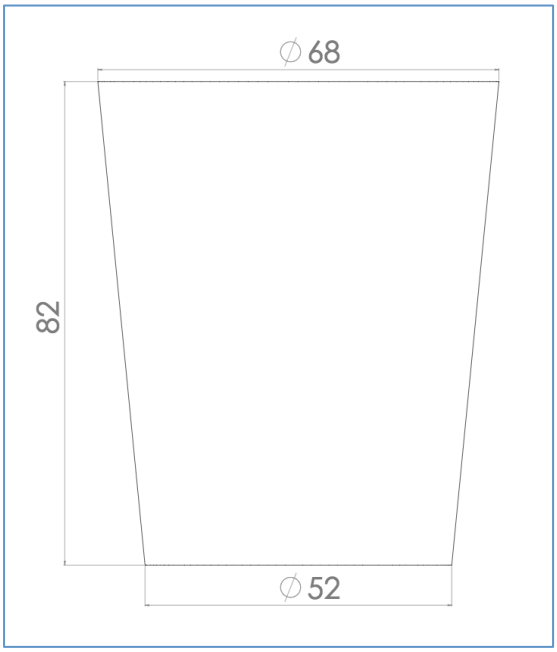


Figure 38 *A paper coffee cup's dimension*

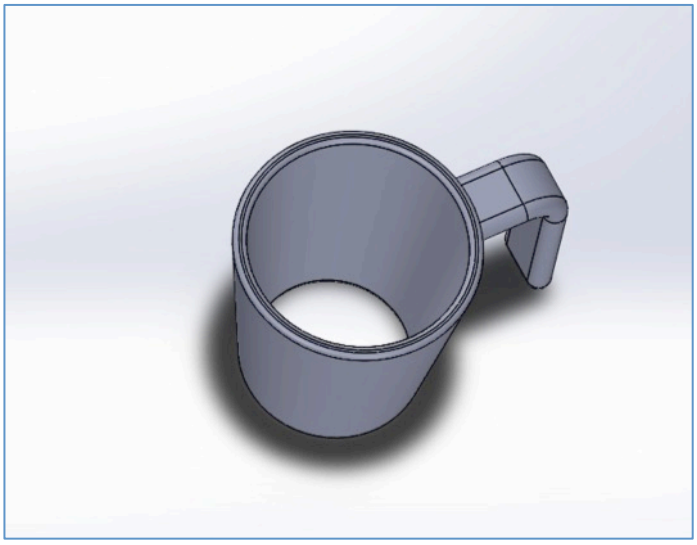


Figure 39 *Coffee cup holder design in SolidWorks.*

Then cup holder design was opened in solidThinking Inspire in order to perform the optimizing process. In the beginning stage, we assume that an inner surface will bear 50 N loading and be supported by a handle, as in **Figure 40** *and* **Figure 41**.

Figure 40 *A loading on the inner surface of Coffee cup holder.*

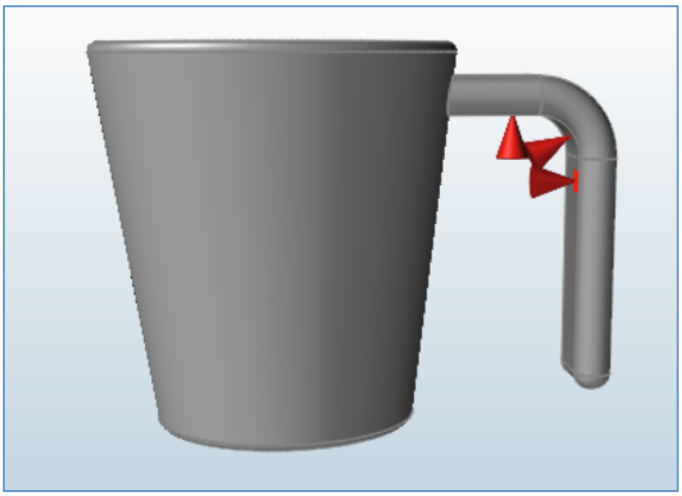


Figure 41 *Supports in the handle of Coffee cup holder.*

After assigning prescribed conditions, we run the optimization with the parameters in **Figure 42**.

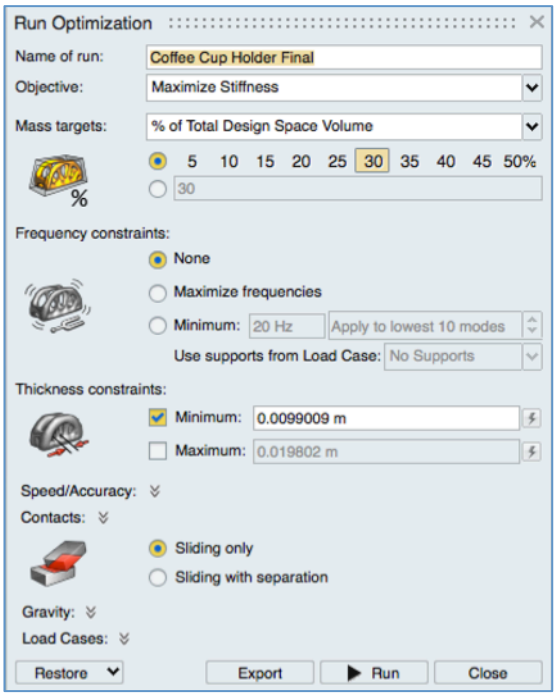


Figure 42 *Optimizing parameters of Coffee cup holder.*

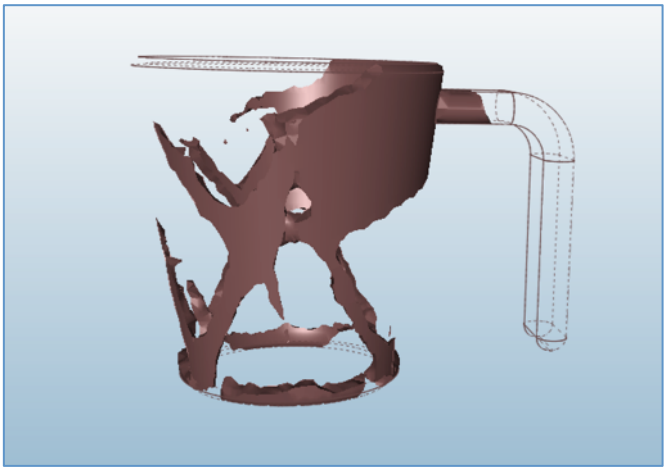


Figure 43 *Optimizing shape of Coffee cup holder.*

Similar to above models, the ideal shape was created and filled up with material mass by combination of **Bridge**. **Wrap**, **Create** in PolyNURBS tools.

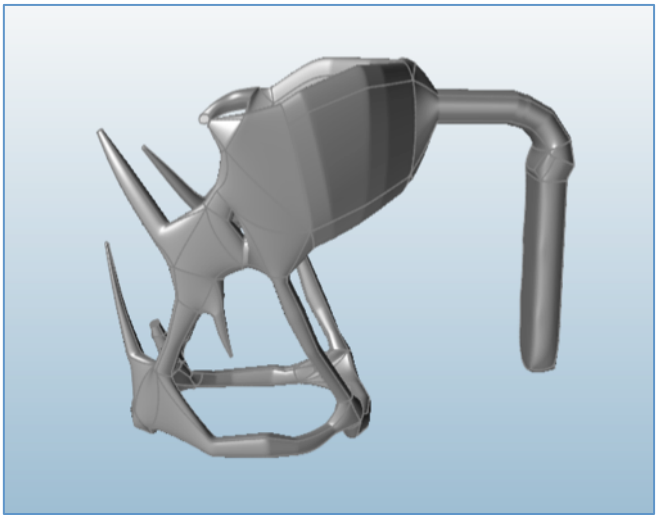


Figure 44 *Initial optimized Coffee cup holder*

After generating the PolyNURBS model, Coffee cup holder design was needed to be re-design because of its uneven inner surface. The solution was revolved-cut the inner surface in solidThinking Inspire. From the **Geometry** tab on the ribbon, the **Line** tool was used to create the face with dimension as **Figure 45**.

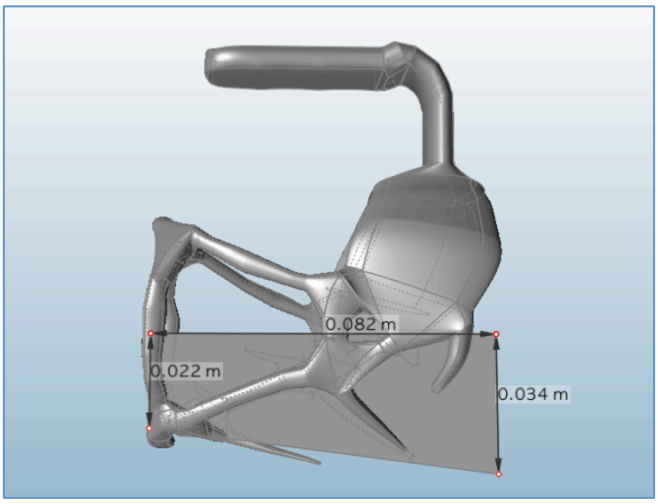


Figure 45 *Revolve-cut sketch of the optimized Coffee cup holder design.*

That face was revolved to a tapered cylindrical part by **Revolve** tool from the same tab. A newborn part was subsequently used as a tool to subtract the PolyNURBS coffee cup holder part as in **Figure 46**. As a result, we can have a nice smooth surface inside coffee cup holder. The final step is simulation the loading conditions of model by **Analysis Explorer** tools as ***Figure 47***.

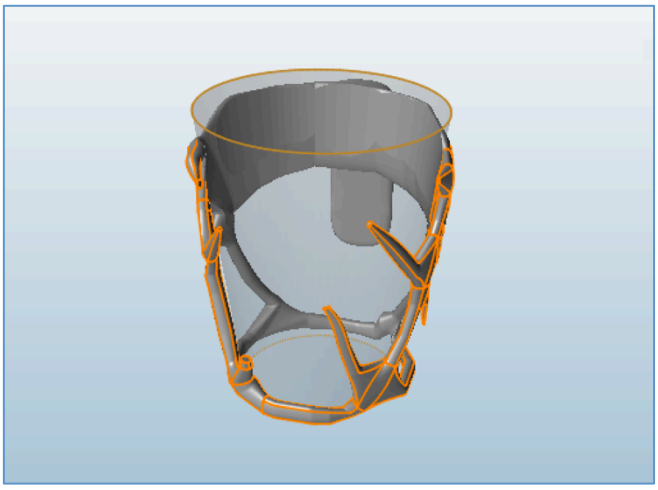


Figure 46 *Subtracting in the optimized Coffee cup holder design.*

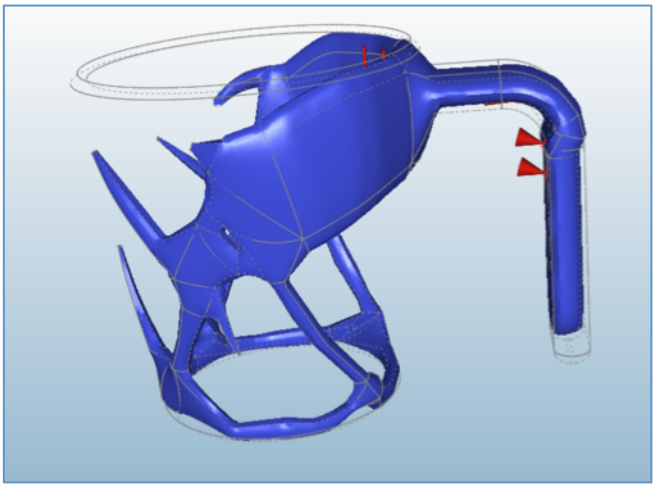


Figure 47 *Simulation analysis of the optimized Coffee cup holder.*

3.4 3D Printing Coffee Cup Holders

3.4.1 3D printing Coffee cup holders by MakerBot Replicator.

To operate the printing process, the initial step is access in MakerBot Print program on the computer and then imports a CAD file of original coffee cup holder design by Solid-Works. It is essential to rotate or rescale the design rescale in some cases until fitting the actual dimensions. Afterwards, we are able to edit any print settings as material, quality and other options in the Setting dialog as **Figure 48**.

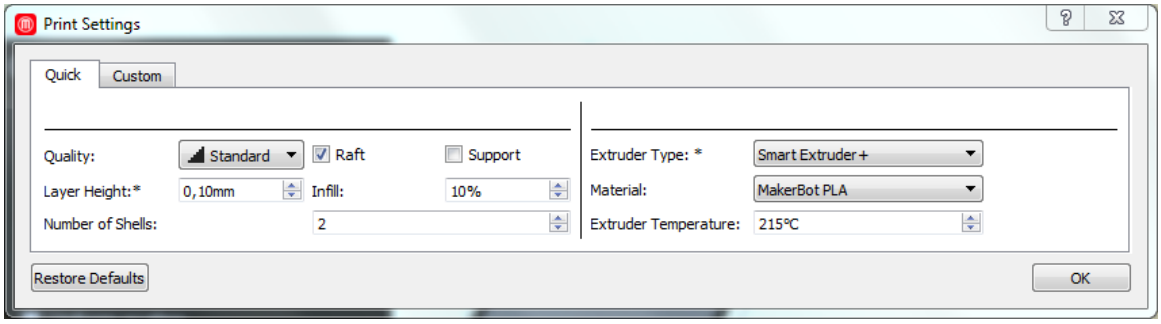


Figure 48 *Original Coffee cup holder print settings.*

Before moving to the printer, the Preview dialog allows users observing the model with generated support and printing time. In order to print in Replicator+, the printing file can be transfer via memory stick or Internet connectivity. In the control panel, it is important to navigate the printing file and then start the print.

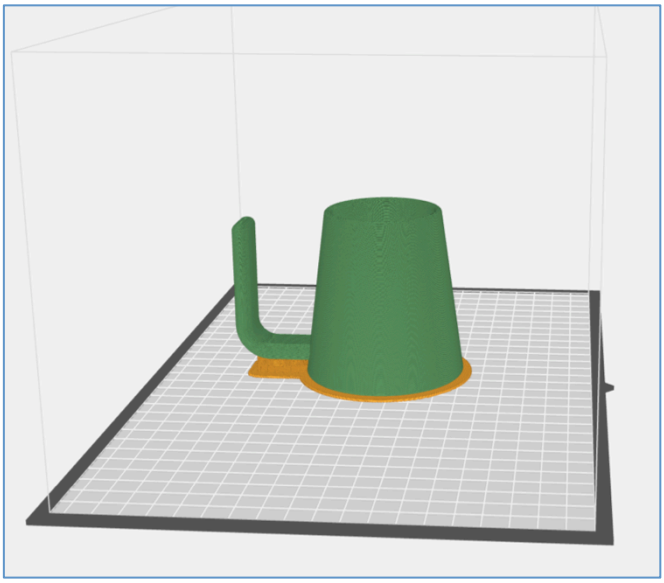


Figure 49 *Original Coffee cup holder print preview*

Regarding the optimized coffee cup holder design in solidThinking Inspire, we repeat all above steps to process the printing workflow. As a result, the optimized coffee cup holder print preview is shown in **Figure 50**.

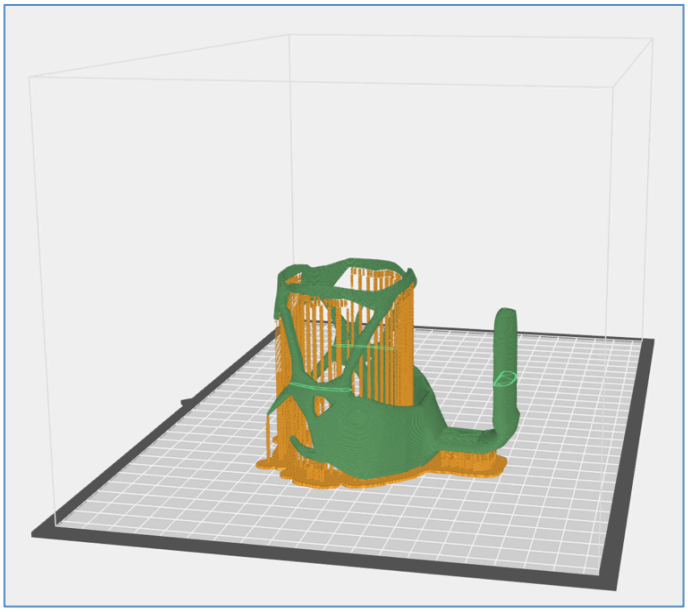


Figure 50 *Optimized Coffee cup holder print preview.*

3.4.2 3D Printing by Formlabs Form 2

At the first stage of printing optimized coffee cup, we start with PreForm software and select the proper settings in the Print Setup window. Then the CAD file was loaded to the system. The size of model can be rescaled and oriented by Preform toolset. For this model, it will generate best supports structure if we rotate the model 45 degrees clock-wise. The manual support can be used to assign supports in particular spots of the model. Finally, we send it to Form 2 printer.

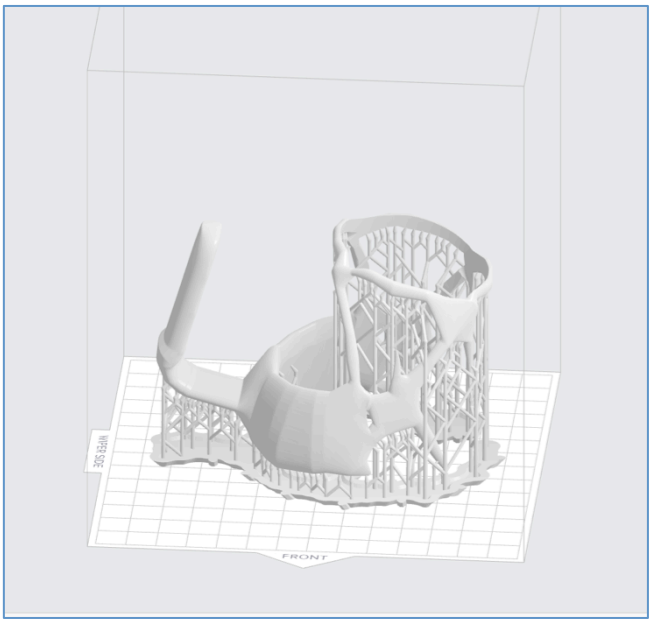


Figure 51 *Optimized Coffee cup holder print preview.*

Before starting printing, it is important to open the vent cap to ensure your resin tank fills appropriately. Then we confirm the print file on the machine. The resin will flow to the tank until its temperature goes to about 35 degree Celsius. The printing process will be started automatically. When the print is accomplished, the gloves should be worn due to the adhesion of model's surface. The platform is removed from the printer and attached to the jig. Thus, it is easier to lift the print off the platform. In order to wash out the mod-el, we rinse the model into two buckets with isopropyl alcohol (IPA) for 30 minutes and 10 minutes respectively. Afterward, the model should be clean gently with soap and dry out. The final step is trimming supports off the main body by the flush cutters.

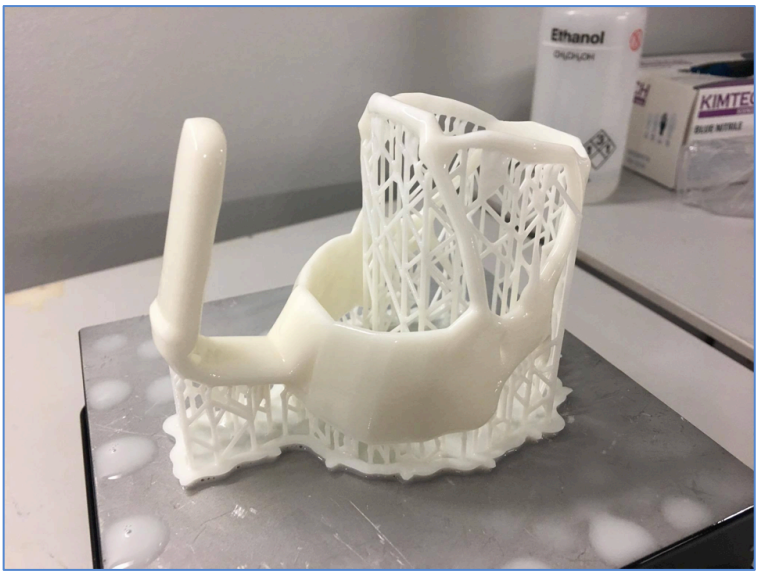


Figure 52 *Optimized Coffee cup holder prints by Formlab Form 2.*

Regarding the traditional coffee cup holder design in SolidWorks, we repeat all above steps to process the printing workflow. Subsequently, the result of tradition coffee cup holder is demonstrated as **Figure 53**.

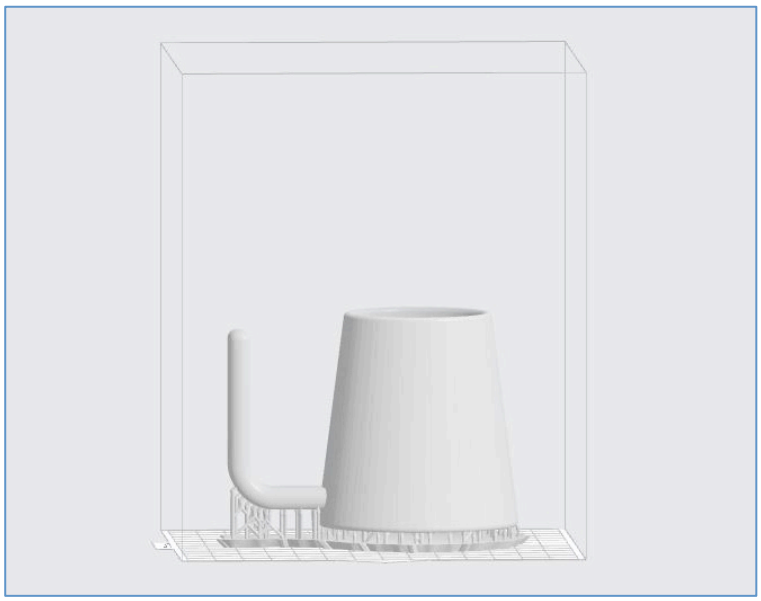


Figure 53 *Traditional Coffee cup holder print preview.*

4 RESULTS

After conducting several optimization processes, it is doubtless to realize the massive transformation of original parts. In next sections, the illustration of optimized results will be presented.

4.1 Phone Holder

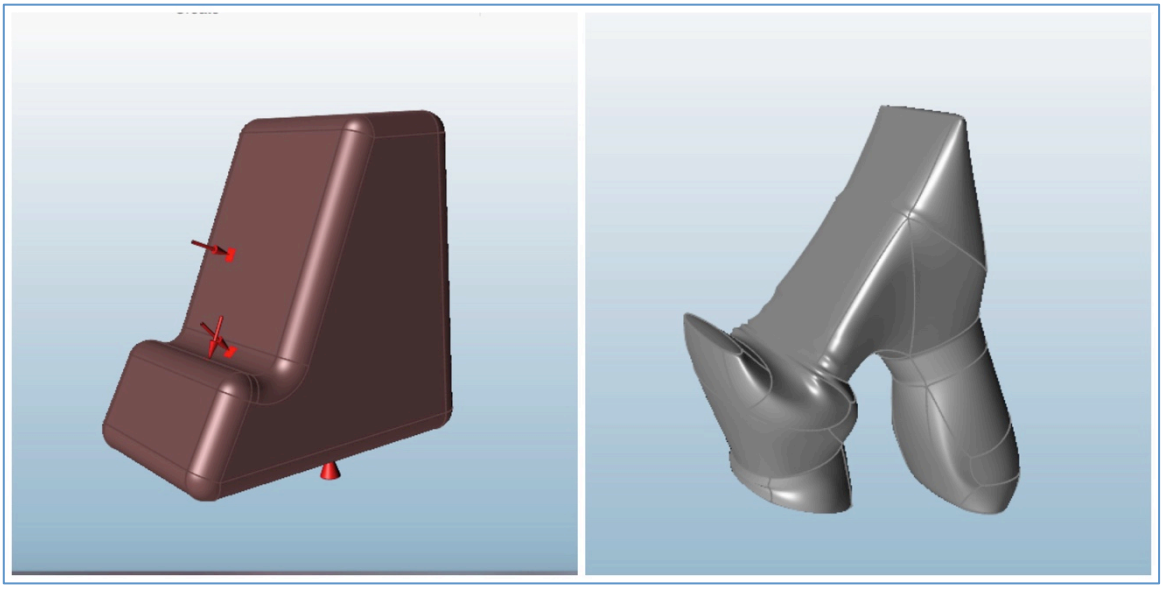


Figure 54 *Phone holder optimizing result.*

4.2 Pushup equipment

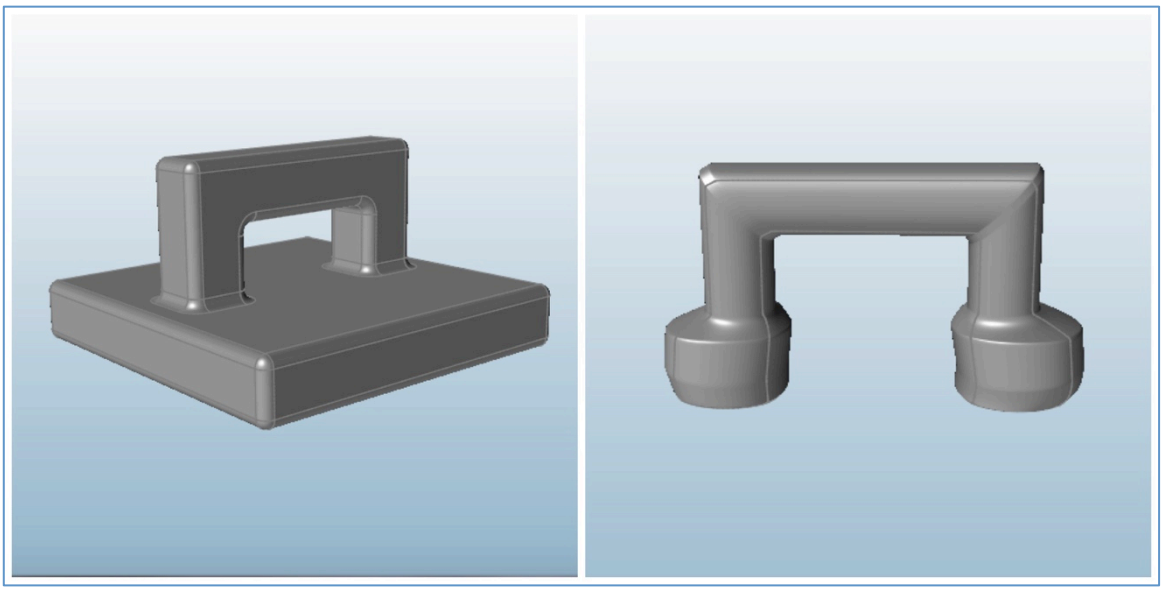


Figure 55 *Push-up equipment optimizing result.*

4.3 Coffee Cup Holder

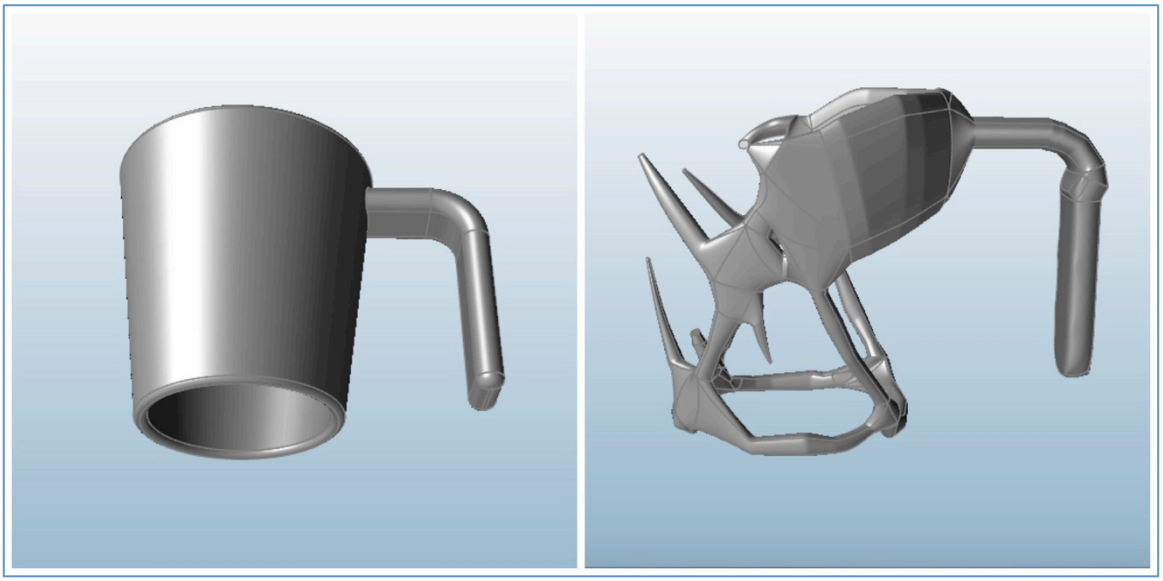


Figure 56 *Coffee cup holder optimizing result.*

4.3.1 3D printing Coffee cup holder by MakerBot Replicator+

Figure 57 *Coffee cup holder prototypes with MakerBot Replicator.*

4.3.2 3D printing Coffee cup holder by Formlabs Form 2

Figure 58 *Optimized Coffee cup holder prototype.*

4.4 Comparisons of results.

After achieving the final results, it is important to execute comparisons between original designs and optimized designs. In this specified comparisons, there are many crucial factors needed to study such as weight, printing time and material cost.

Table 3 *Comparison of weight reduction between models.*

	Phone holder	**Push-up equipment**	**Coffee cup holder**
Original weight (kg)	1,52	9,3	0,068
Optimized weight (kg)	0,47	2,2	0,032

Note: ABS is chosen material for the weight measurement.

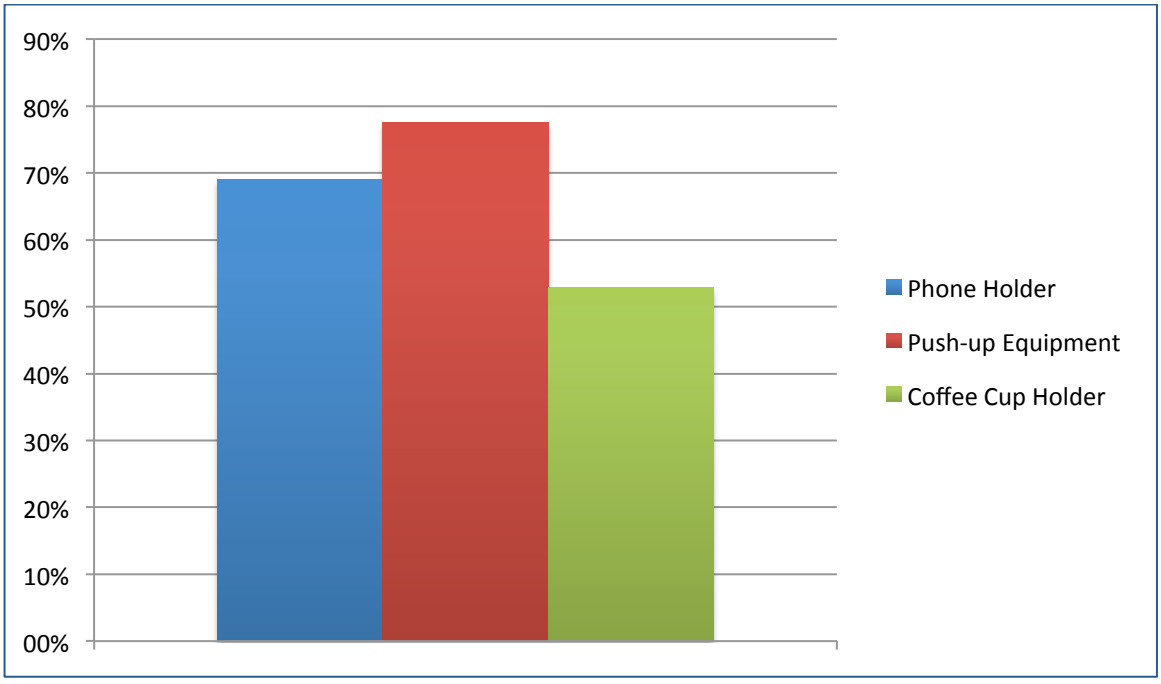


Figure 59 *Percentage of mass reduction in optimizing three models.*

Table 4 *Comparisons of result with MakerBot Replicator.*

	Original Coffee cup holder design	**Optimized Coffee cup holder design**
Weight including supports (g)	48,55 gram	27,45 gram
Weight (g)	43 gram	19 gram
Printing time	5 hours 53 minutes	5 hours 15 minutes
Material cost (€ / a piece)	3,3	1,86

Note: MakerBot PLA is a chosen material for this print (6.78€ / 100g).

Table 5 *Comparisons of result with Formlabs Form 2.*

	Original Coffee cup holder design	**Optimized Coffee cup holder design**
Volume including supports (g)	72.52 mL	45.56 mL
Volume (g)	59.25 mL	30.81 mL
Printing time	6 hours 4 minutes	5 hours 12 minutes
Material cost (€ / a piece)	11.75 € / a piece	7.4 € / a piece

Note: Formlabs White Resin is a chosen material for this print (162€ / 1 L).

5 DISCUSSION

solidThinking Inspire offers a variety of options in optimization such as selecting material, connection tools, thermal loading and shape controls. The technique perfectly inspires all designers to disrupt all the baselines of ordinary modelling. It is free for users to assign any loading conditions on the design and adjust the mass-reducing level. Basing on the prescribed constraints, it will produce a new shape for 3D model achieving required persistence and outstanding branching structure.

On the other hand, solidThinking Inspire sometimes cannot be operated effectively due to its difficulty in software interface. During the design process, it is slightly problematic in some basic manipulation such as rotating and zooming to the correct aspect. These disadvantages considerably delay the design process and induce the software interface becoming unfriendly to overall users.

Moreover, the PolyNURBS tool has a low resolution so it is hard to fill the mass into the curve shape to establish well-rounded parts. When the software extracts the optimized frame, there are some specified portions of a model, which cannot be selected and filled up with material mass by PolyNURBS tools. For designs having wiring structures, the PolyNUBRS is able to fill in the optimizing frame delicately. The more complex optimized frame is harder to fill the material mass in thoroughly. It will result in uneven shape and rough surfaces such as the initial optimized Coffee cup holder design.

6 CONCLUSION

solidThinking Inspire obviously takes a step into the next generation intelligent design software .The computer does not receive and execute commands from users. In another ways, users will propose an ideal to a computer which is an artist evolving that ideal into an aesthetic artwork.

Through the thesis, all of the designed parts were satisfactorily optimized with solidThinking Inspire. The material consumption in three parts was spectacularly decreased over 50 %. Especially, the prototypes of Coffee cup holder were both successfully printed out by Formlabs Form 2 and MakerBot Replicator. The results has indicated that the optimization of Coffee cup holder was lowered the weight from 43 grams to 19 grams and the material cost from 3,3 € to 1,86 € with MakerBot Replicator. With Formlabs Form 2, it occurred the cutback from 59,25 mL to 30,81 ml in volume and 11,75 € to 7,4 € in material cost. Furthermore, the PolyNURBS toolset is able to minimize a high load of modelling time while still achieving the design goals. It is definitely to accomplish the optimization process in less than an hour for general users.

After spending a certain time period with solidThinking Inspire, it can be said that it is almost impossible to produce the optimized structure with conventional 3D CAD programs. However, solidThinking Inspire has limitations to initiate a part traditionally while SolidWorks or other CAD software does it more efficiently. It is preferred to firstly model parts in SolidWorks or other conventional 3D CAD programs, then solidThinking Inspire will be in charge of processing the optimization. From my perspective, solidThinking Inspire should be utilized as an assistive tool for other 3D CAD software to optimize the designed parts.

Thanks to evolution of global technology, the demand in computing tools has been growing increasingly. solidThinking Inspire currently pursuits the software-enhancing marathon by adding new features year after year. It is strongly believed the next version of solidThinking Inspire will be upgraded to provide higher-level concept of engineering design and cover all its drawbacks in last version.

7 REFERENCES

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