Design Workflow for Additive Manufacturing

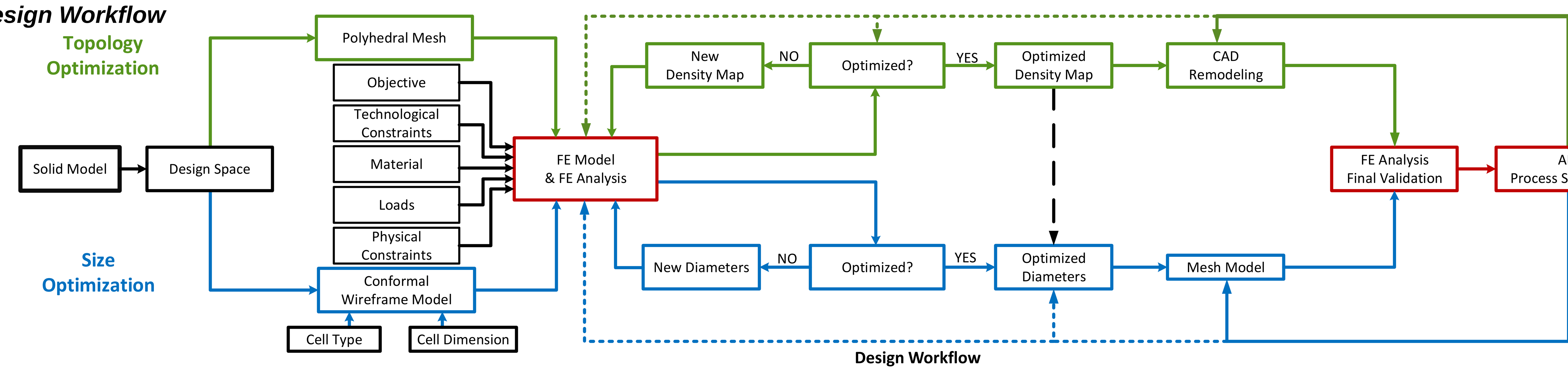
Federico Uriati[1], Stefano Rosso[1], Filippo Da Rin Betta[1], Gianpaolo Savio[2], Roberto Meneghello[1], Gianmaria Concheri[2]
[1] University of Padova - Department of Management and Engineering - Laboratory of Design Tools and Methods in Industrial Engineering
[2] University of Padova - Department of Civil, Environmental and Architectural Engineering - Laboratory of Design Tools and Methods in Industrial Engineering

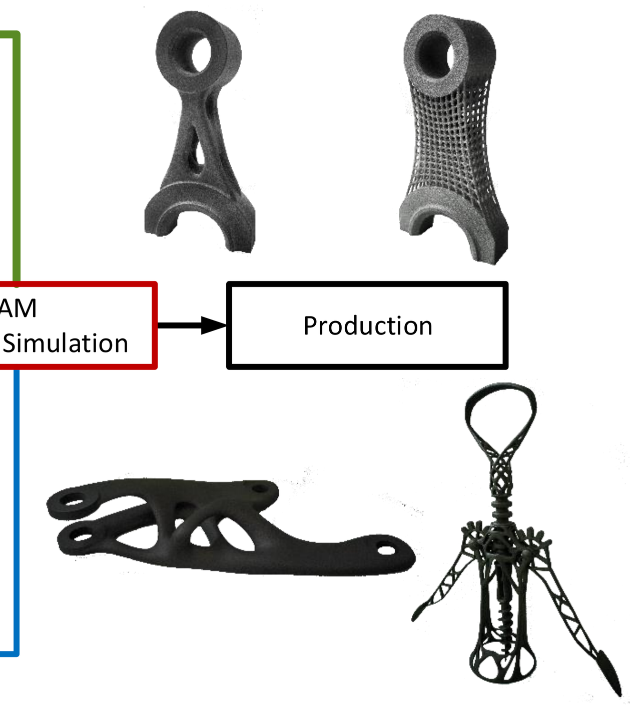
Introduction Additive Manufacturing (AM) technologies introduce a completely new approach for the design of customized and optimized parts; AM allows producing complex parts with a limited increase in production cost and time. Optimization approaches such as size, shape, and topology optimization are being re-discovered, since the models resulting from these methods are challenging to be produced with traditional manufacturing techniques.
These methods enable designers to reinvent solutions of different structural problems with the aim of obtaining improved mechanical characteristics and higher overall performances, especially the reduction of weight that comes with these new geometries.
High computational resources and advanced Computer-Aided Engineering software make possible to perform the optimization in the conceptual design stage, where the design has not yet been refined to a specific topology.

Functional requirements → Conceptual design → Product Optimization → Detailing + Validation + Process Simulation

The aim of this work is to illustrate the possibilities of new design workflow in relation with AM potentialities and limits.
Different geometric modeling opportunities and structural optimization techniques are presented: commercial software is used to perform topology optimization and redesign of the optimized results. As alternative, a novel method, developed by the research group, allows to design conformal lattice structures with size optimization performed on the beams, which allows to automatically obtain a smooth mesh model. The proposed workflow is validated on a set of test cases, adopting different design methods based on Lattice structures, PolyNurbs, and parametric CAD to reach innovative solutions.

Design Workflow





Design Workflow

Topology optimization → material is arranged in order to find the best distribution under a set of boundary conditions and respecting the structural and dimensional performances requirements.
The goal of the optimization can be to maximize the stiffness, imposing a desired reduction of mass; these targets are reached varying the density of each element of a polyhedral mesh, which is related to the mechanical properties.
As a result, a density map is obtained, which is contoured to a specific level of density (threshold), obtaining a mesh surface. The optimized mesh of the design space is taken as an "inspiration" to the further modeling of the part in a CAD environment, often operated manually. Finally, Finite Element Analysis (FEA) is performed to verify that the stress conditions are respected. As an alternative, the density map can be used to assign the dimension to the elements of a lattice structure.
Tools: SOLIDWORKS, Inspire Studio.

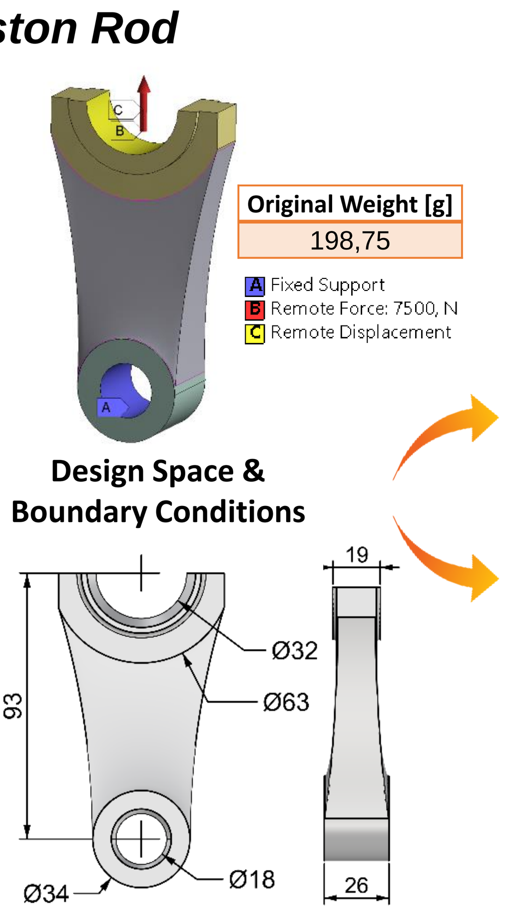
Size Optimization→ a conformal wireframe is obtained specifying the unit cell type and the minimum cell size; then, a size optimization is performed taking in consideration boundary conditions and manufacturing constraints. As a result, the optimal diameter of each beam is obtained.
A lattice structure is then modeled from the wireframe and the optimized diameters with a mesh modeling method. Each beam is modeled with a 8-faces, 12-nodes coarse mesh, assuming a double truncated pyramidal shape.
The mesh approach then exploits the Catmull-Clark subdivision surface algorithm to automatically obtain smooth surfaces; indeed, each quad face is subdivided in 4 smaller quad faces at every iteration. This allows to reduce stress concentration, especially at nodal points, enhancing the mechanical properties and the fatigue life of the lattice.
Tools: Rhinoceros 6, Grasshopper, Python.

Computer Aided Engineering → simulation software are used to investigate the properties of the parts from the mechanical point of view and simulate the AM process.
Finite Element Analysis can be used in different phases of the design process. During optimization, FEA are iteratively performed to recognize weak points of the structures and consequently optimized dimensions of the elements are obtained. At the end of the modeling phase, FEA is performed to mechanically validate the resistance of the final part.
The validated model undergoes an AM process simulation to evaluate the possible issues that could occur during the manufacturing phase, i.e. thermal deformation and residual stress. If the deformation is higher than requirements, is necessary to remodel the component to compensate the error; if the residual stresses are too high, is necessary to re-perform the FE analysis.
Tools: ANSYS Mechanical 2019 R1, ANSYS Additive Suite 2019 R1, Karamba3D.

Case Study – Piston Rod

Structural Analysis Parameters	
Material	AlSi11Cu2Fe
ρ [g/cm3]	2,7
E [Gpa]	71
YS [Mpa]	140
UTS [Mpa]	240
Safety Factor	1,5
σ_{adm} [Mpa]	93,3

Process Simulation Parameters	
AM Technology	SLM
Baseplate temperture	170°C
Hatch spacing	80 µm
Slicing stripe width	5 mm
Laser power	300W
Scan speed	2667 mm/s



Original Weight [g]
198,75

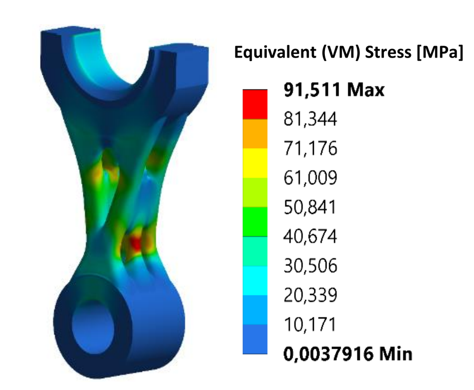
Design Space & Boundary Conditions

Topology optimization

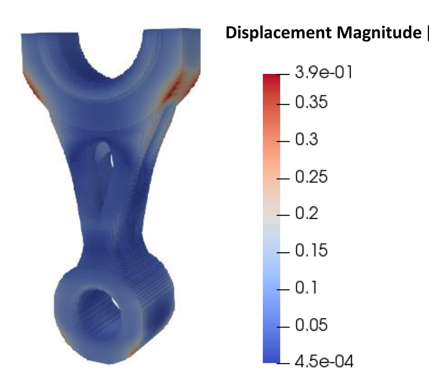
Weight [g]	Mass Reduction
115,53	41,87%

Size Optimization

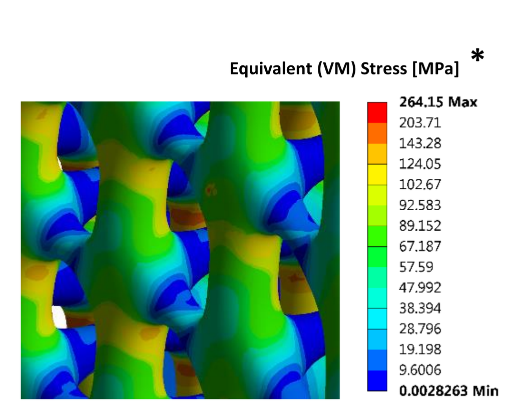
Weight [g]	Mass Reduction
124,55	37,33%

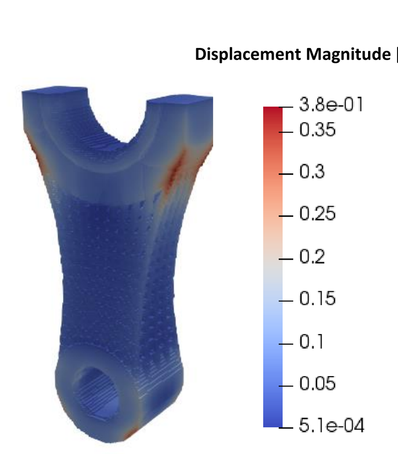


Finite Element Analysis



AM Process Simulation

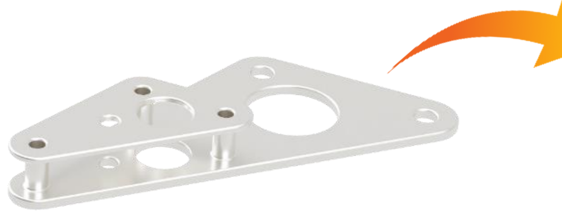




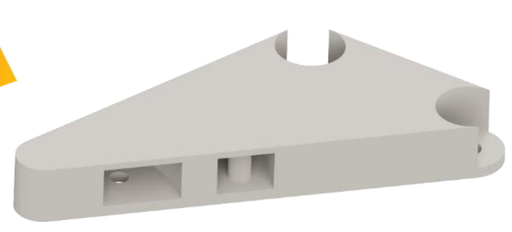
The results reported in the table show the similar percentage of mass reduction obtained with these two methods applied to a piston rod.
Nevertheless, both the approaches present pros and cons. The topology optimization workflow can be performed using commercial software but requires a manual remodeling of the optimized part because the mesh resulting from the optimization is really coarse. The mesh approach allows a size optimization of the lattice structures and the final mesh presents a smooth curvature (C2) thanks to the Catmull-Clark subdivision surface algorithm.
Final verification through FEA demonstrates that the load cases are correctly sustained and with the AM process simulation is possible to observe the displacements, deformation, and residual stress that will occur during the AM process.
* The higher value of stress observed is related to the absence of fillet between the lattice structure and the body.

Sailing Boat Barber- Single part Topology Optimization

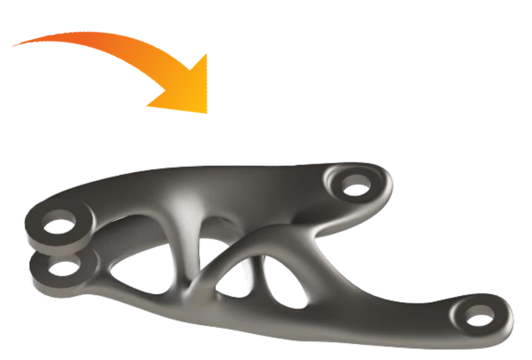
Material	ρ [g/cm³]	E [Gpa]	YS [Mpa]	UTS [Mpa]	Safety Factor	σ_{adm} [Mpa]
Ti6AlV	4,429	110	786	862	1,5	524



Original Geometry



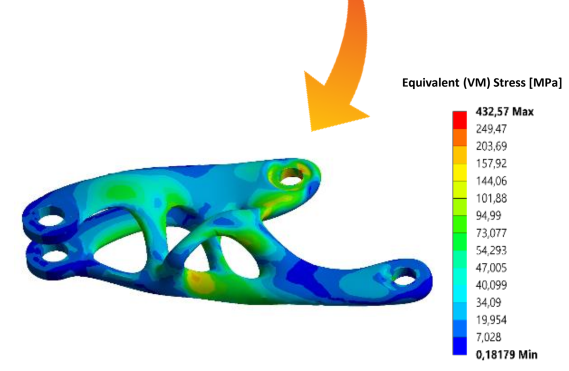
Design Space



Topology Optimization and Remodeling

A sailing boat barber is redesigned adopting topology optimization. The starting geometry is made up by 5 parts connected with fasteners.
Topology optimization guarantees mass reduction and parts consolidation, resulting in one lighter part. Part consolidation is an advantage offered by Topology Optimization because with the reduction of number of components, the weakness related with connection point decreases.
The optimized geometry is then manually remodeled and mechanically verified through FEA.

Volume[cm³]	Original Weight [g]	Final Weight [g]	Mass reduction
46,347	257,9	205,27	20,4%



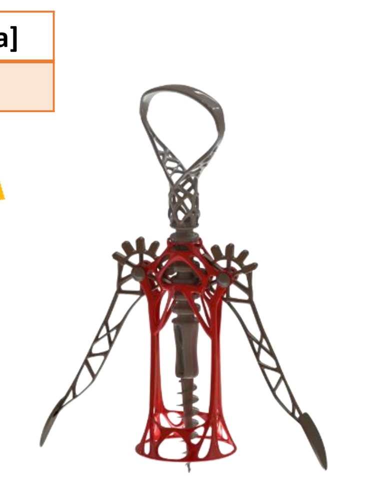
Finite Element Analysis

Corkscrew - Assembly Topology Optimization

Material	ρ [g/cm³]	E [Gpa]	YS [Mpa]	UTS [Mpa]	Safety Factor	σ_{adm} [Mpa]
SS316L	7,99	190	494	624	1,5	329,3

Original Geometry

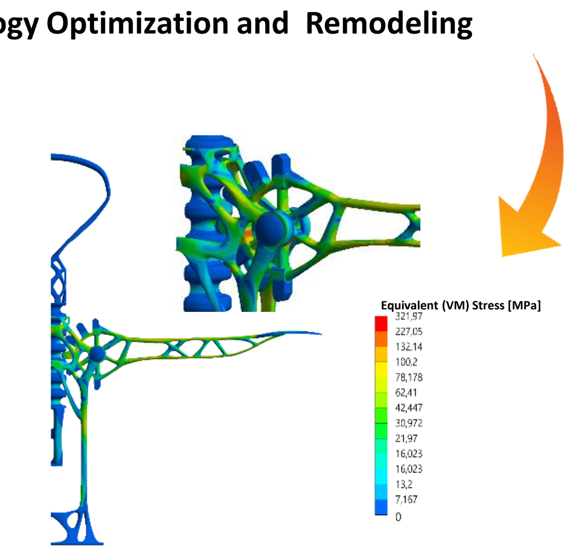
Design Space



Topology Optimization and Remodeling

Topology optimization can be applied to assembly. In the corkscrew case, the topology optimization is performed taking into account both external loads and internal ones generated from contact points.
A considerable global reduction of mass is obtained and the FEA validate the new geometry.

Part	Original Weight [g]	Final Weight [g]	Mass Reduction
Arm(x1)	25,28	10,53	58,3%
Body	60,95	29,72	51,2%
Screw	78,70	47,94	39,0%
TOTAL	190,21	99,02	47,9%



Finite Element Analysis

Acknow-ledgements
The authors gratefully acknowledge generous in-kind support from A. Sandi, 3Dfast Srl, for fabricating test parts .
This work was partially funded by Fondazione CARIPARO, Cassa di Risparmio di Padova e Rovigo and the grant “FSE 2105-37-11-2018: Hybrid manufacturing strategies through selective laser sintering for the customization of series components” by Regione Veneto.

References

Bendsøe, M. P. (1989) ‘Optimal shape design as a material distribution problem’, Structural optimization. Springer, 1(4), pp. 193–202.
Catmull, E. and Clark, J. (1978) ‘Recursively generated B-spline surfaces on arbitrary topological meshes’, Computer-Aided Design. Elsevier, 10(6), pp. 350–355. doi: 10.1016/0010-4485(78)90110-0.
Savio, G., Meneghello, R. and Concheri, G. (2018) ‘Geometric modelling of lattice structures for additive manufacturing’, Rapid Prototyping Journal, 24(2), pp. 351–360. doi: 10.1108/RPJ-07-2016-0122.