Case Study

Design for Metal Additive Manufacturing – Part 2: Topology Optimization and Build Preparation

Developed and curated by the Ansys Academic Development Team

János Plocher and Navid Manai

education@ansys.com

Summary

Ansys Discovery is a simulation-driven design tool that combines instant physics simulation, high fidelity simulation, and interactive geometry modeling in a single easy-to-use experience. This enables designers and engineers to illustrate a wide variety of concepts in the fields of design, structures, fluids, and heat transfer with the help of simulation.

Ansys SpaceClaim is the perfect modeling solution for engineers who want access to 3D answers but do not have the time or inclination to learn complex traditional CAD systems. It provides you with tools to accelerate geometry preparation and get to simulation sooner while eliminating delays between design teams. It includes and Add-In called Ansys AdditvePrep which is a tool that allows you to prepare parts that will be additively manufactured, *i.e.* find an optimal orientation, create support structures, adjust the build strategy and parameters and generate a build file.

This case study demonstrates how an aeronautical bracket component is topology optimized to reduce component weight using Ansys Discovery while considering additive manufacturing. It highlights the effect of design and manufacturing constraints on the build file and explores the build orientation and the associated trade-off with respect to the distortion tendency, build time and support structure requirements, using Ansys SpaceClaim's AdditvePrep.

Table of Contents

1. Introduction

1.1 Background

Different types of brackets can be found in various sections of an airplane. Weight plays a critical role in the design of such airplane components. Thus, today's airplanes are commonly built from lightweight composite materials and high-performance alloys. Utilizing materials with a high stiffness and strength to weight ratio is one key aspect in aircraft design, however, the geometry of the components also contributes significantly to the specific performance. In this case study we will consider that the bracket will be fabricated using additive manufacturing (AM), due to its potential for reducing the component's weight.

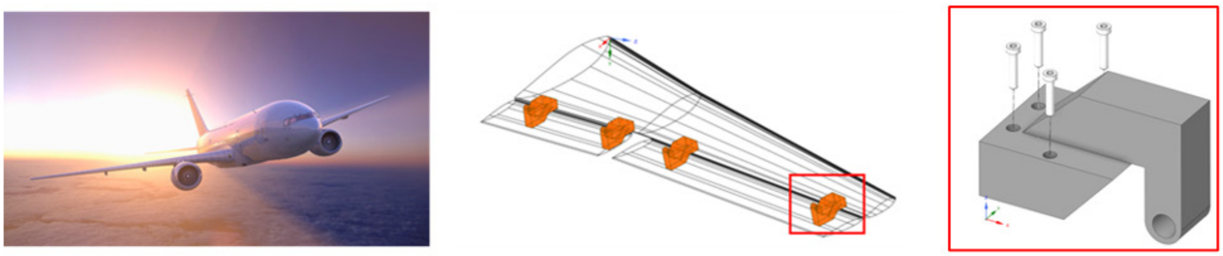


Figure 1: Airplane flap bearing bracket example (not to scale).

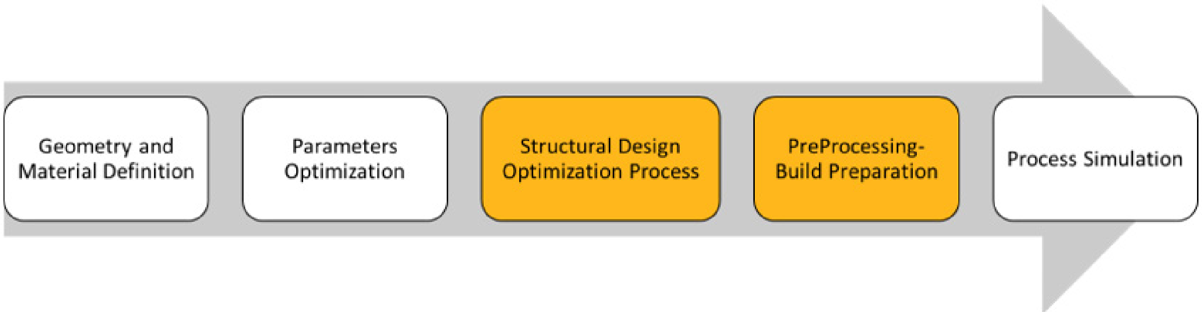
The case study is split into three parts, as shown in Table 1, following the design for AM (DfAM) workflow (see Figure 2). With A360 (AlSi10Mg), a suitable material candidate for the bracket was determined in the first part of this study. The objective of the second part is to (i) find an alternative design to the existing (over-engineered) flap bearing bracket (see Figure 1) that is saving weight while not compromising safety and ensuring the points of contact (fixtures, bores, *etc.* remain intact so that other parts of the assembly do not need to be re-designed), to (ii) determine the best possible part orientation on the baseplate and to (iii) create the necessary support structures and build files for additive manufacturing.

Table 1: Division of the metal additive manufacturing case study in parts.

Part	Content	Software	Hyperlink
1	• Strategic Materials Selection • LPBF process parameters optimization	• Ansys Granta EduPack • Ansys Additive Suite	Find case study here
2	• Topology optimization • Build preparation (orientation, support structures, *etc.*)	• Ansys Discovery • Ansys SpaceClaim	Current document
3	• Stress and distortion analysis in LPBF	• Ansys Additive Suite	Find case study here

As shown in Figure 2, first the initial geometry is selected, the material candidate is defined and the process parameters are optimized. Subsequently, the initial design is optimized to reduce weight, for which the loads, boundary conditions and connections must be applied. Once the structural simulation and optimization has been solved, outputs like stress, strain and displacement can be evaluated (*i.e.* comparison to the functional requirements). Then, the build set-up is defined. This pre-processing step includes the build orientation, the support structure generation, the toolpath generation and optionally nesting of multiple parts on the build plate. Prior to the physical printing of the part, in the third and last step, the LPBF process is simulated to analyze the stress and distortion that can occur within the AM process and check if there may be possible issues with the build process as a result.

Figure 2: Design for additive manufacturing (DfAM) workflow.



There are different lightweighting design approaches, ranging from heuristic or expertise-driven methods of introducing recesses (*e.g.* holes in ribs of airplane wings) or lattices in areas of low stress (presupposes a preliminary structural analysis) to numerical (mathematically-driven) approaches like topology optimization (TO), which can also be used to inform variable-density lattice design. Additive manufacturing (AM) enables the production of such complex designs with ease, thus making numerical methods an attractive approach for structural optimization to exploit the lightweight potential AM has on offer.

1.2 Problem Statement

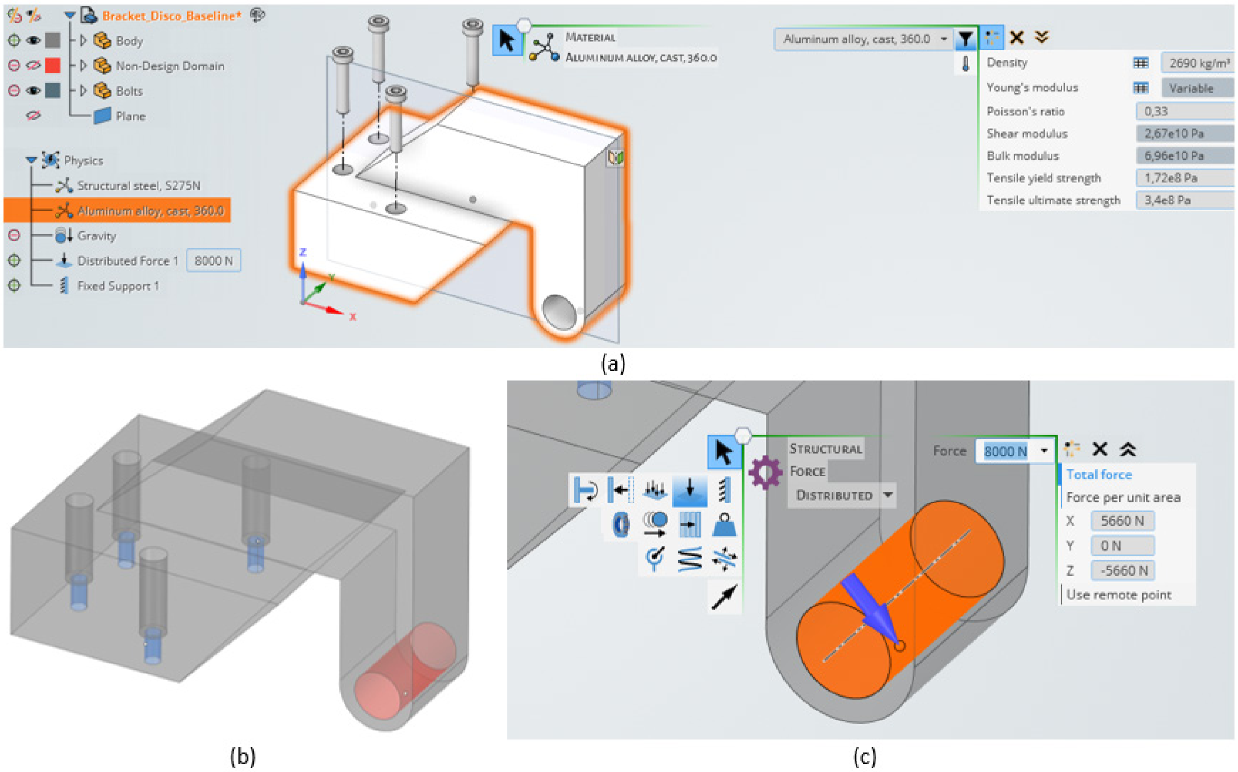
The aim of this work is to structurally optimize the component through topology optimization while ensuring the safety factor and stiffness is no less than 75% and 90% of the baseline (original) components, respectively. Furthermore, the component shall be prepared for AM, meaning either an optimal build orientation should be determined, or a relevant design constraint shall be considered to reduce the support volume requirements while considering build time and distortion tendencies. Finally, a build file, including the support structure geometry and machine files shall be created.

2. Pre-processing

In the pre-processing with Ansys Discovery, the focus lies on applying the loads and boundary representations and assigning the relevant material properties. If you are new to Ansys Discovery, the free Static Structural Simulation in Ansys Discovery course is recommended to learn the basics of the software interface.

As per part one of the case study, the aluminum alloy A360 (AlSi10Mg) has been chosen for the bracket and is thus applied to the component (see Fig.6(a)), while the bolts are assigned the default structural steel. The bottom section of the bolt drilling is assigned a fixed boundary condition (see Fig. 6(b)), whereas a load of 8000N is applied to the flap link (see Fig 6(c)), which is composed of force components in x- and y-direction of equal magnitude (Force in x: 5660N and force in y: -5660N). The simulation is run in the Explore Mode with a fidelity/mesh size of 1.5 mm (the mesh size can be verified with the Size Preview function in the Simulation ribbon). It is of note that the default fidelity in the Explore Mode is dependent on your computer's GPU performance and that any difference in the approach and mesh size may lead to different results (see AIC on obtaining numerically accurate results: Numerically Accurate Results). A topology optimization is a mesh-dependent problem, yielding different topologies and thus the part performance is based on the number of finite elements used. Lastly, it is of note that gravity was not considered for this simulation.

Figure 4: Pre-processing setup with (a) the material assignment, (b) the definition of the design domains for



the fixed boundary conditions and loads and (c) the applied load components.

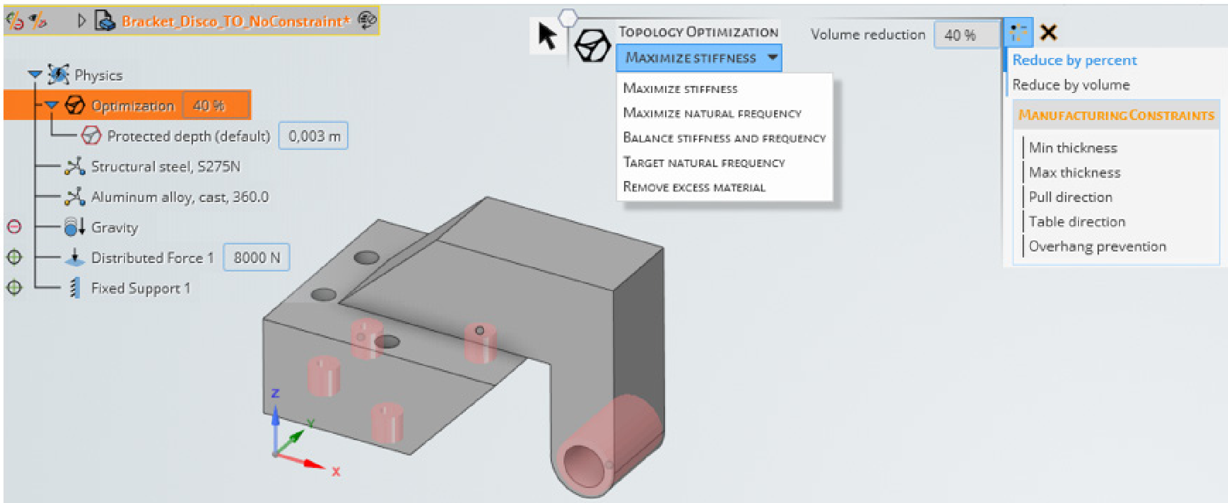
The default parameters that will be provided as outputs are the factor of safety (yield strength of material divided by maximum equivalent stress in the part), max. deformation and max. Von Mises stress. When additionally selecting topology optimization in the simulation ribbon, the volume and mass will be added to the default monitors, enabling the tracking of the convergence towards the target constraint.

3. Solution Method

3.1 Topology Optimization

The topology optimization is an iterative process, aimed at finding the optimal lay-out of a structure within a given/specified design domain under consideration of applied loads, boundary conditions as well as a volume constraint. Additional design restriction in form of manufacturing constraints may also be included. For the TO of the bracket the multi-physics tool Ansys Discovery is employed which uses a Level-Set TO, which can be set up from the simulation ribbon. If you are new to topology optimization in Ansys Discovery, the free Topology Optimization Ansys Innovation Course is recommended to learn the basics of how to set up the simulation. By default, the objective function of the TO in Discovery is set to maximize the stiffness (minimize compliance) and remove material by a given percentage or volume. Moreover, a protected depth is automatically applied to the surfaces at which boundary conditions and loads are applied (see Figure 5). This protected depth can be extended to other geometrical features that shall remain intact (unaffected by optimization) and effectively reduces the design domain (volume in which material can be removed).

Figure 5: Heads-Up Display (HUD) for the Topology Optimization feature, showing the different optimization



objectives together with the volume and manufacturing constraints. A protected depth around the boundary conditions and loading areas (highlighted in red) is automatically applied.

For this case study, two scenarios are considered, as detailed in Table 2: Maximize stiffness for a 40% lighter bracket with (i) no manufacturing constraints and (ii) an overhang prevention constraint set to 45° (most common inclination angle, below which most 3D printers need to apply support structures to guarantee part can be built). It is of note that another manufacturing constraint relevant to AM related to the minimum thickness, *i.e.* minimum feature size (needs to be aligned with the printer resolution) could also be explored in Ansys Discovery but was not considered in this study.

Table 2: Bracket designs parameters for topology optimization.

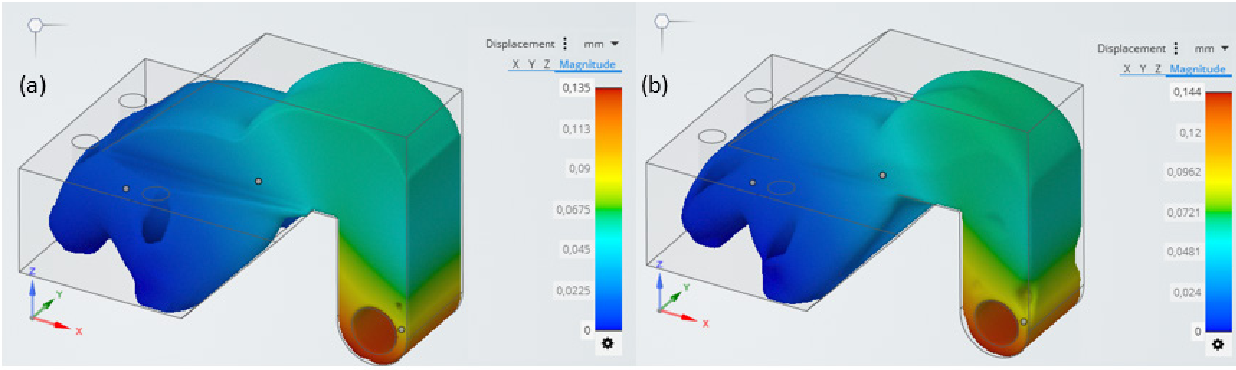
Component	Objective	Volume reduction [%]	Protected depth [mm]	Manufacturing constraint
Type-A	Max. Stiffness	40	2	n.a.
Type-B				45° Overhang prevention

4. Post-Processing

4.1 Solution

In Figure 6, the two optimized topologies considering no manufacturing constraints as well as a 45° degree overhang angle are displayed and Table 3 lists the corresponding performance metrics.

Figure 6: Topology optimized brackets with the color coding for the displacement.



(a) Without and (b) with overhang constraint.

From Table 3 it becomes clear that the factor of safety and stiffness of the component is reduced after the removal of 40% of the volume. However, the requirements defined in Section 1.2 are met.

Table 3: Results of the key state variables and parameters of the baseline geometry and the topology optimized counterparts.

Component	Factor of Safety	Max. Displacement [m]	Max. Von Mises Stress [MPa]	Mass [kg]	Volume [m^3]
Baseline	2.11	1.31E-4	81	0.947	3.5E-4
Type-A	1.62	1.34E-4	106	0.565	2.1E-4
Type-B	1.66	1.44E-4	104	0.566	2.1E-4

4.2 Reverse Engineering

The first step after running the topology optimization in Ansys Discovery is to create a faceted (triangulated) geometry (click on the button in the results arc that says: 'Add optimized body to model') that is extracted from the finite element mesh. This necessary to obtain an accurate representation of the structure's boundaries and contours. The newly created faceted geometry will be shown in the model tree and can be – among other post-processing options - converted into a solid by right-clicking on the faceted geometry and choosing one of the three options. In this case study we have re-created a solid body using surfaces (see Figure 7). It is of note that it is not necessary to create a solid model for the subsequent steps within Ansys SpaceClaim, but a STL file would suffice.

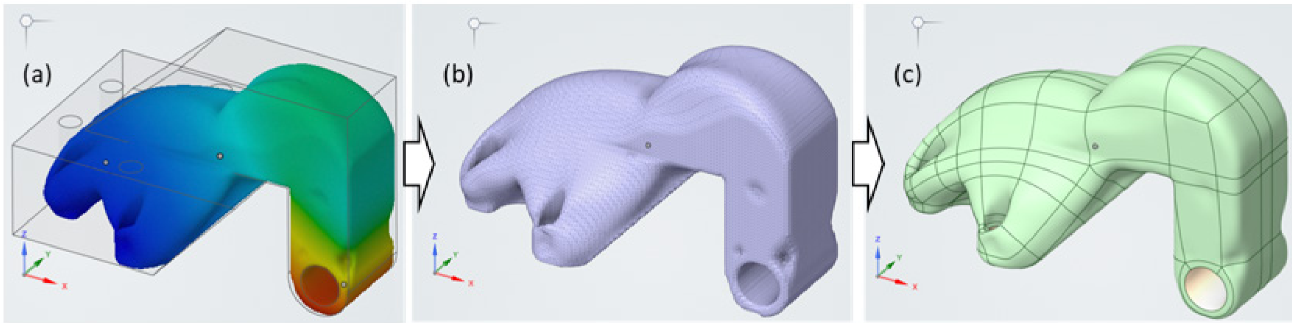


Figure 7: (a) Simulation result of topology optimized bracket, (b) faceted geometry and (c) solid body that was reverse engineered using surfaces.

5. Build Preparation

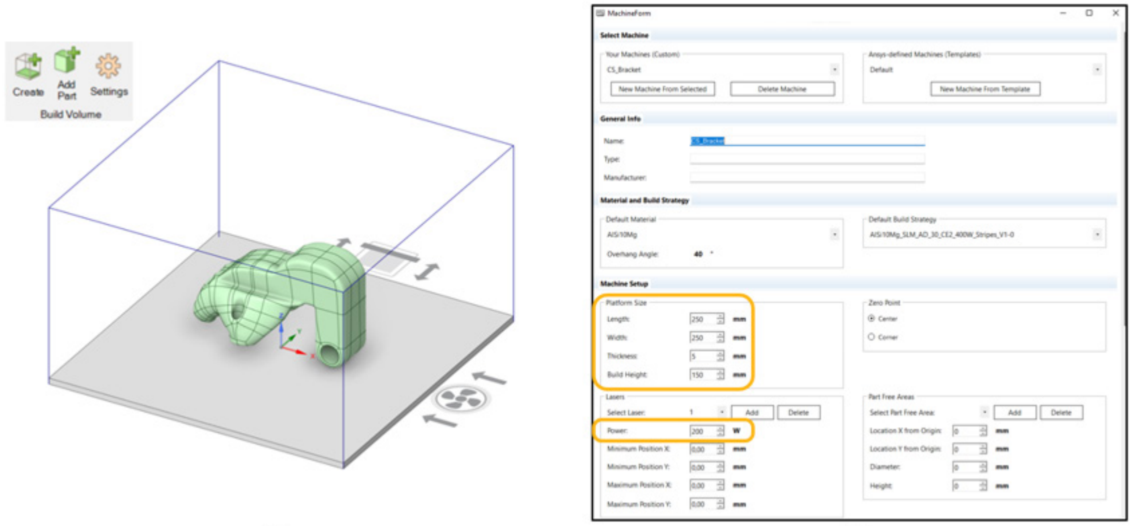
Once the two optimized designs of the bracket have been created the parts will be prepared for metal additive manufacturing (MAM) in Ansys' direct modeling CAD tool Ansys SpaceClaim. If you would like to learn more on SpaceClaim, how to use it and its functionalities, we recommend to have a look at two Innovation Courses: Ansys SpaceClaim - Tools Tutorial and Solid Modeling With Ansys SpaceClaim.

AdditivePrep, is an Add-In tool in SpaceClaim, helping you prepare you part for MAM, either by sending it directly to your build chamber for print or by continuing your AM-workflow with simulation and consequently optimization of the process using Ansys Additive to explore the AM-induced stresses and distortions.

You can use AdditivePrep to orient your part(s) based on factors that are most relevant to you. These include build time, volume of supports, distortion tendency, stair-step error, and shadow area. Subsequently you can automatically cerate support structures for the component and adjust your build strategy and parameters. Next, AdditivePrep will enable you to view and animate the scan vectors within a slice or the slices within a build in the Slice Viewer. Finally, you can generate a build file for print and simulation. It is of note that this Add-In must be enabled first. In SpaceClaim, click File > SpaceClaim Options > License and then check AdditivePrep (also ensure the Add-In is active in the Settings window). Once activated a new Ribbon called 'Additive' will appear.

5.1 Build Volume

First, the build volume will be created and then the part will be added (see Figure 8). This automatically defines the recoater and gas flow directions with respect to the part. In the settings you can choose from different machine types, edit existent ones and create new machine configurations. For this case study, we are choosing a build volume of 250mm x 250mm x 150mm (L x W x H) with a baseplate thickness of 5mm. Furthermore, 200W is selected for the laser in printing with a LPBF (Laser-Power-Bed-Fusion) machine. It is of note, that the overhang angle is set to 40° as default but will be changed to 45° at a later stage. Following these steps will create new components in the structures tree, including *e.g.* baseplate, the build chamber, the part, as well as the support regions and support, which are still empty because they have not yet been defined.



(a) (b)

Figure 8: Ansys AdditivePrep interfaces for the (a) Build Volume and (b) its editable machine settings.

5.2 Orientation Map

In the second step, the parts will be orientated within the created build volume by dragging the cross-hair to the darkest green location (optimal) of the combined orientation map. Areas in red are indicating the worse possible orientation. In this study the support volume, build time, and distortion tendency are considered with an equal weighting (34% prioritization for support volume and 33% for build time and distortion tendency, respectively) between the three (see Figure 9). Moreover, the default Z-Offset was kept at 5 mm (default distance between the part and the baseplate), to ensure easy cut-off from the baseplate. Note, the change in the combined map will yield a respective change of location of the cross-hairs in the individual maps.

Part of the aim of this study is to elucidate the effect of design decisions (*e.g.* 45° overhang constraint in specimen Type-B) on the optimality of the part orientation. Thus, three different scenarios will be compared (see Table 4):

- Specimen Type-A: Optimally oriented part (through using the orientation map)
- Specimen Type-B: Unchanged (z-direction) build orientation in accordance with the initial design decision (of including a 45° overhang constraint in setting up the TO)
- Specimen Type-A*: The part orientation of the unconstrained design (Type-A) will not be optimized but rather kept identical to the part orientation of Type-B.

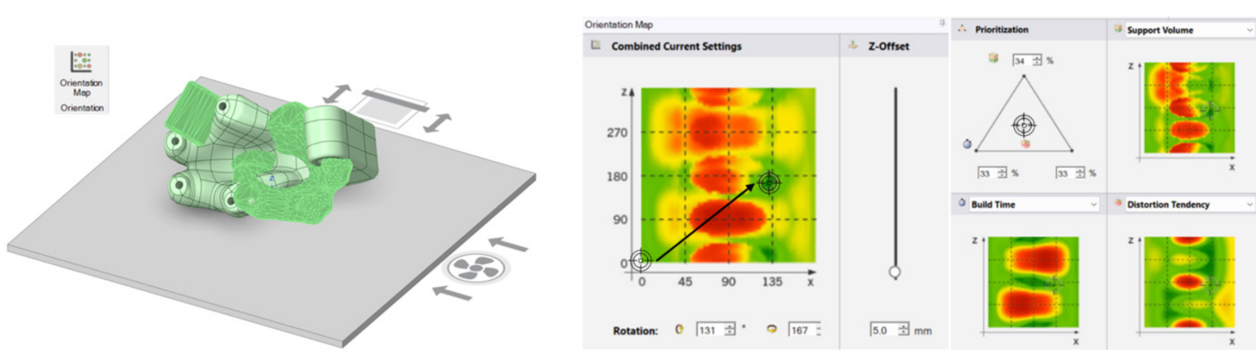
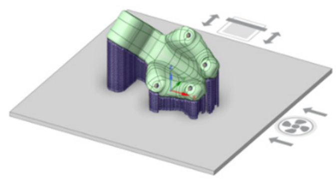
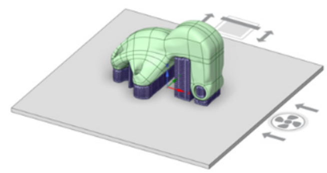
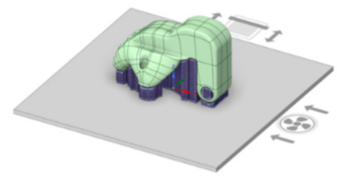


Figure 9: Ansys AdditivePrep interfaces for the (a) Orientation Map and (b) its editable orientation.

The qualitative result of the print orientation derived from the orientation maps for the 3 configurations, as shown in Table 4, reveal that even though Type-B was optimized for a minimum support volume it still does not outperform Type-A, which was oriented optimally. Moreover, Type-A achieves a better build time and a lower distortion tendency as Type-B; thus, yielding a higher score. In fact, the distortion tendency of Type-B is critical and worse compared to Type-A*, which has the same part orientation. The quantitative difference in support volume requirement will be discussed in Section 5.4. Given these findings, it must be carefully assessed whether a manufacturing/design constraint is necessary (*e.g.* when induced material anisotropy due to the layered manufacturing must be tightly controlled to meet performance requirements), as in this case, an unconstrained design yields higher structural performance (recall Table 3) and naturally offers scope for choosing a better orientation.

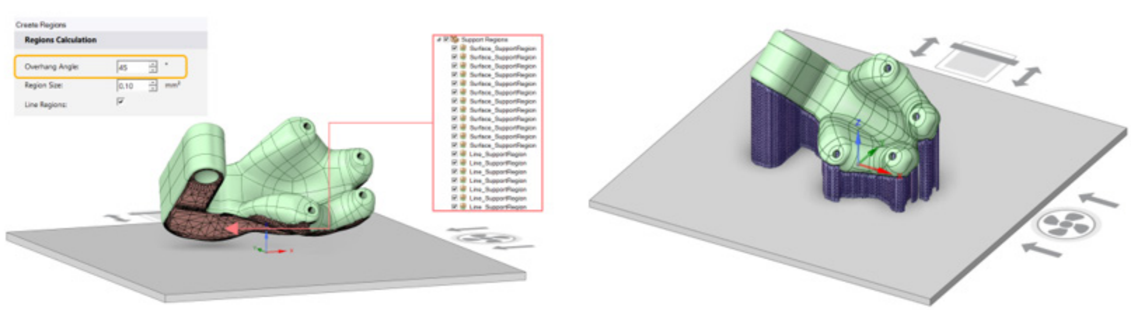
Table 4: Summary of the qualitative part orientation analysis for the two designs under consideration with optimal (Type-A), as-designed (Type-B) and suboptimal (Type-A*) orientation.

	Type-A		Type-B	Type-A*
				
Overall				
Support Vol.				
Build Time				
Dist. Tend.				

Note: The colors denote an ***Excellent****,* ***Good****,* ***Critical*** *or* ***Bad*** *part orientation.*

5.3 Support Generation

Once the orientation is determined, the support regions are automatically determined and subsequently created. It is of note, that the support angle is changed from the default 40° to 45° to align with the design constraint (see Figure 10(a)). The default region size of 0,1 mm2 was kept and line regions were considered. Block and line supports were primarily generated, accompanied by other support types or manual support adjustments (*e.g.* deletion of small fragmented regions) if necessary. The default settings for the general, support and contour parameters were applied.



(a) (b)

Figure 10: Build preparation of specimen Type-A with (a) support area determination for an overhang angle of 45° and (b) support structure generation.

5.4 Build File

Next, the Build Processor tab in the Build File category is selected to define parameters for the slicing and defining the scan pattern. For this purpose, we will start off with a pre-defined build strategy template with the designation: *AlSi10Mg_SLM_AD_30_CE2_400W_Stripes_V2-0*. This signifies the usage of the aluminum alloy for this case study (recall Section 2) to be processed using selective later melting (SLM) with a slice thickness of 30 microns, a maximum laser power of 400 W and the striped hatch pattern. For this case study, we will deviate from this pre-defined strategy by changing the slice thickness of the part and the supports to 80 microns (see Figure 11), which will automatically offer the opportunity to save as a new build strategy. Furthermore, the hatch distance of the stripe-based volume infill strategy was changed from 0,17 mm to 0,15 mm (as this was determined in the first part of the case study when optimizing the processing parameters).

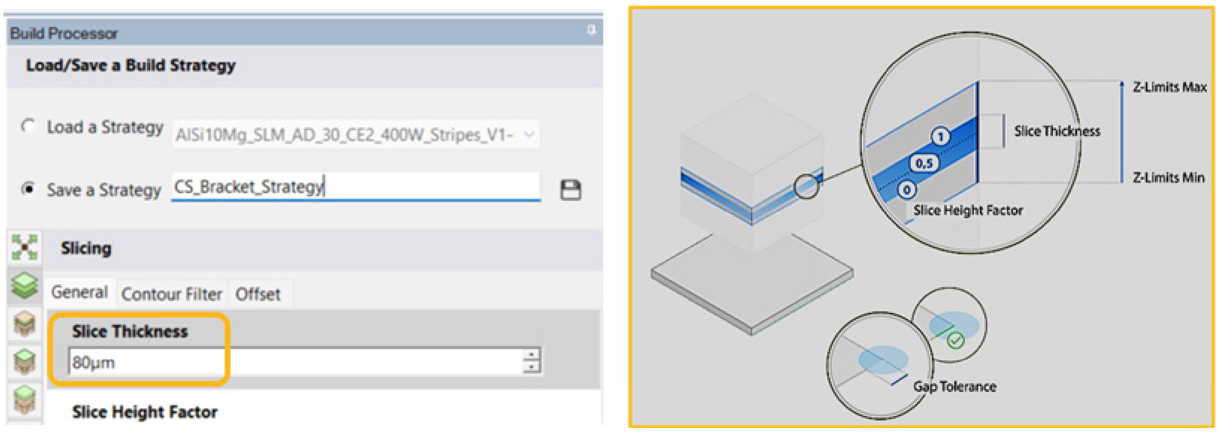


Figure 11: Build processer interface highlighting the adjustment of the slice thickness, yielding a new strategy.

Next, the Slice Viewer tool offers the opportunity to examine the individual 2D slices of the bracket, the sequence of slices within the build and the progression of the laser scan vectors in the direction they are created. No specific actions were taken in this stage, this merely serves as sanity check for the build process and parameters applied.

The last step within the *Build File* category constitutes the cost estimator tool, determining the total cost to build the job based on the cost, material, and build time parameters defined for any selected SLM machine (see Table 5). As such a machine profile was not explicitly chosen (recall Section 5.1), the machine cost will be zero in this case (otherwise it would be the largest contributing factor, which is directly related to the build time). Specimen Type-A results in the highest cost due to a greater material usage which is associated with the part orientation that yields the largest part height and thus a greater build chamber powder fill requirement. On the contrary, specimen Type-B yields the lowest cost. The build time, which is comprised of the exposure times (path and jump times) and the recoated times is greatest for build configuration Type-A* while Type-B can be printed 2% faster. Table 5, also confirms the effect of using the overhang constraint, leading to a 100% lower support volume requirement of specimen Type-B compared to the other two specimens. The material loss, *i.e.* unrecoverable powder loss due to gas flow during the build and post-processing, is around 10% for all specimens but highest for specimen Type-A, which should be important to consider in a potential life-cycle-assessment.

Table 5: Part Cost Estimator summary chart for the build configurations under consideration.

	Type-A	Type-B	Type-A*
Total Cost [/]	363	321.2	325.6
Build Time	0d 7h 45min	0d 7h 37min	0d 7h 47min
Total Material	10.74 kg	8.89 kg	8.99 kg
Support Material	0.02 kg	0.01 kg	0.02 kg
Material Loss	1.07 kg	0.89 kg	0.9 kg

Lastly, you can export the build file as command line interface (CLI) file, as required by *e.g.* EOS machines or export the full set of files associated with the build. The latter can then be used for further process simulation (see Table 1 and Figure 2).

6. Further Steps and Conclusion

It is of note that the structural analysis and optimization was performed considering various assumptions and simplifications. For instance, the material properties were considered as bulk, whereas layered manufacturing techniques often introduce material orthotropy. Furthermore, the study only considers a single loading case whereas multiple loading cases (*e.g.* different flap angles, wind loads, *etc.*) would be realistic. Moreover, potential cyclic (fatigue) loading (heating and cooling regime during and after flight and/or fatigue behavior) or damage analysis (potentially relevant for safety regulations), which could be conducted using Ansys' flagship FEA software Ansys Mechanical, were not considered.
Regarding the topology optimization of the component, a stiffness-optimal design was conducted, however, Ansys Mechanical offers the opportunity to include a customer criterion or consider an optimal topology for a given volume fraction considering a minimum stress, which could be relevant in such aeronautical components. In light of this, after a minimum compliance optimization, a common additional step (which was not conducted in this study), would be a shape optimization to reduce potential stress concentrations. Besides the AM manufacturing constraint of considering the overhang angle, another constraint related to the minimum feature size could have been added to ensure agreement of the final layout with the printer resolution.

The build preparation has elucidated the effect of (i) including/excluding a design constraint (45° overhang angle during setup of the TO) as well as (ii) improving or leaving the part orientation unchanged on the build time, support structure requirements and distortion tendency. It is of note that the selection of the right design and manufacturing constraint, *i.e.* part design, alongside the part orientation or general process plan (all the process information making up the build file) is non-trivial as it is usually tightly connected to the business plan and the general trade-off considerations between cost, time and quality. The latter can be controlled to some extent using the right process parameters and simulation tools, which will be showcased in the pursuing part of this case study, in which AM-induced stresses and distortions are determined to help decide on post-processing requirements or even design and build orientation changes.

Use and Reproduction

The content used in this resource may only be used or reproduced for teaching purposes; and any commercial use is strictly prohibited.

Document Information

This case study is part of a set of teaching resources to help introduce students to multiphysics topics.

Ansys Education Resources

To access more undergraduate education resources, including lecture presentations with notes, exercises with worked solutions, MicroProjects, real life examples and more, visit www.ansys.com/education-resources.

Feedback

If you notice any errors in this resource or need to get in contact with the authors, please email us at education@ansys.com.

ANSYS, Inc.
Southepointe
2600 Ansys Drive
Canonsburg, PA 15317
U.S.A.
724.746.3304
ansysinfo@ansys.com