



Orca Upgrades

Handguard, Stock, and Barrel Mount

1 INTRODUCTION

This upgrade pack is for the Orca V5.3 3D Printed AR-15.

It contains a mid-length handguard that features M-LOK slots and picatinny top rail for mounting lights, lasers, and tape switches. An extended butt that adds another inch to the length of pull. And an updated barrel mount that prints at a forty five degree angle, resulting in a cleaner print with less chances of warping.

The Docs from the original Orca V5.3 release will be referenced as the primary source of information. Only details pertinent to these upgrade parts will be covered here. The only major change is that rather than built in support, support enforcers are used. This was done to save time on the release, and as an experiment. Details on how to use these are provided.

Both the handguard and extended butt are optional based on your preference. However, the barrel mount is an improvement that is highly recommended, if you have not already printed the older V5.3 mount. If you already have a V5.3 mount, then the updated one won't do you much good, as the improvements are primarily related to printability.

2 TABLE OF CONTENTS

1	Introduction	1
3	Printing	1
3.1	Print Orientation	1
3.2	Using Support Enforcers	2
3.3	Support Settings	2
3.3.1	Barrel Mount	2
3.3.2	Handguard	2

3 PRINTING

Be sure to read the Docs from the original Orca release. Print settings for the updated parts are the same as the originals. Only change is the way the support is done.

3.1 PRINT ORIENTATION

All parts come pre-orientated. You can double check print orientation by looking at the obvious print surfaces. The barrel mounts do not contain an obvious surface, and are supported by the support material. If you look closely, there is a small pad of material in the mesh files for the mounts. This is to ensure that the slicer does not drop the mounts onto the bed fully.

3.2 USING SUPPORT ENFORCERS

The support enforcers are added to the slicer in a similar way to the infill modifiers. They act in the same way as paint on supports. The areas of the part that the enforcers overlap with are supported, other areas are not. This ensures that the minimal amount of support is added.

In Prusa Slicer (and Bambu Studio*) enforcers are added by right clicking on the part > Add Support Enforcer > Load. Then you will select the file for the enforcer, included in the same folder as the part. After this is done select “Support for Enforcers Only” in Prusa or one of the “Manuel” support modes in Bambu.

NOTE: There is a bug in the version of Bambu Studio that I tested with. The slicer does not lock the enforcers or infill modifiers to the part. You will need to manually center them. This a pain, and I hope they fix it soon.

In Cura you will import the enforces same as the part, and then under the “Per Model Settings” select “Print as Support”. After this is done, you can select the modifiers and part and merge them together by right clicking and selecting “Merge Models”.

3.3 SUPPORT SETTINGS

The stock butt does not require any support, but the handguard and barrel mounts do. Most stock support settings will work, and if you have a favorite set of support settings they will probably work great.

- 0.1 MM XY separation between support and part.
- 0.2 MM Z separation between support and part.
- Two interface layers.
- 0.2 MM interface pattern spacing.

3.3.1 Barrel Mount

Standard or tree / organic support will work just as well as the other. Snug supports in Prusa slicer work very well for this.

3.3.2 Handguard

Tree / organic supports are best. If you must use standard supports, disable the bottom interface layers to make it easier to dislodge.

That's really about all!