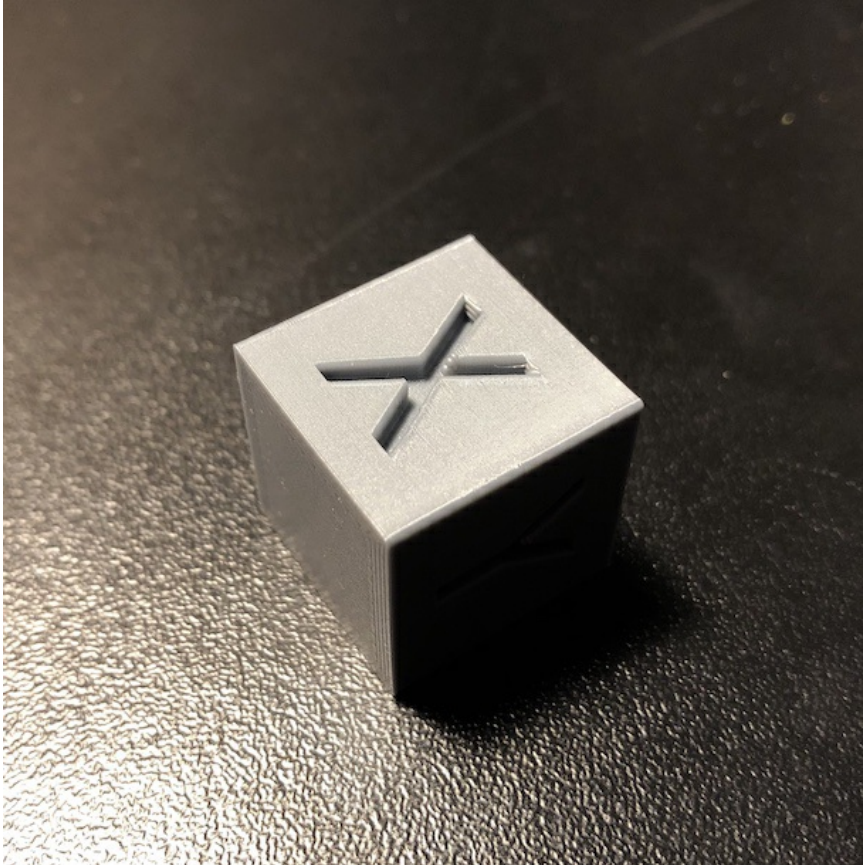# Overview

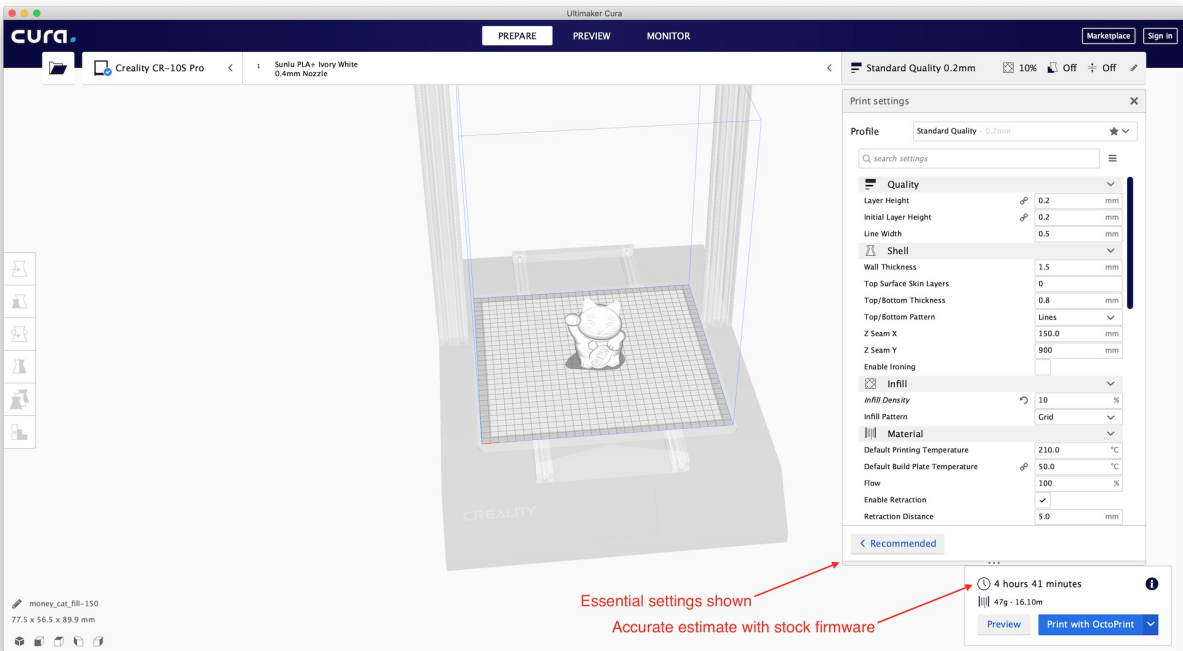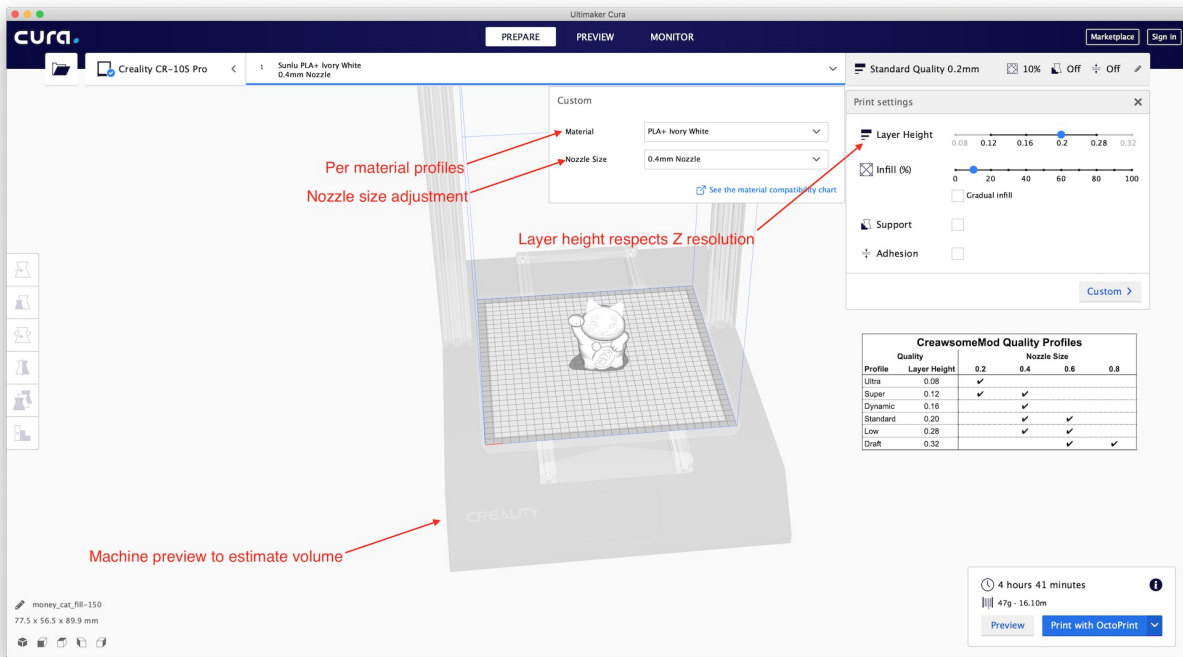## How this mod will improve my print quality ?

It depends how have you been able to tune your profiles and printer so far ! You can see few sample below. You should be able to decide by yourself if this mod worths to be tested. **Keep in mind you still need to adjust settings according to your material and model.**

xyzCalibration_cube.stl printed at 0.12mm layer height, 50mm/s Source File

money_cat_fill.stl printed at 0.20mm layer height, 60mm/s Source File

Top screenshot annotations:

- Per material profiles
- Nozzle size adjustment
- Layer height respects Z resolution
- Machine preview to estimate volume

Cura interface (top):

Ultimaker Cura

PREPARE PREVIEW MONITOR

Marketplace Sign in

Creality CR-10S Pro

Sunlu PLA+ Ivory White
0.4mm Nozzle

Standard Quality 0.2mm   10%   Off   Off

Custom

| Material | PLA+ Ivory White |
| Nozzle Size | 0.4mm Nozzle |

See the material compatibility chart

Print settings

Layer Height   0.08 0.12 0.16 0.2 0.28 0.32

Infill (%)   0 20 40 60 80 100
☐ Gradual infill

Support ☐

Adhesion ☐

Custom >

CreawsomeMod Quality Profiles

Profile	Quality Layer Height	Nozzle Size 0.2	0.4	0.6	0.8
Ultra	0.08	✔			
Super	0.12	✔	✔		
Dynamic	0.16		✔		
Standard	0.20		✔	✔	
Low	0.28		✔	✔	
Draft	0.32			✔	✔

4 hours 41 minutes
47g - 16.10m
Preview   Print with OctoPrint

money_cat_fill–150
77.5 x 56.5 x 89.9 mm

Bottom screenshot annotations:

- Essential settings shown
- Accurate estimate with stock firmware

Cura interface (bottom):

Ultimaker Cura

PREPARE PREVIEW MONITOR

Marketplace Sign in

Creality CR-10S Pro

Sunlu PLA+ Ivory White
0.4mm Nozzle

Standard Quality 0.2mm   10%   Off   Off

Print settings

Profile   Standard Quality - 0.2mm

search settings

Quality		
Layer Height	0.2	mm
Initial Layer Height	0.2	mm
Line Width	0.5	mm
Shell		
Wall Thickness	1.5	mm
Top Surface Skin Layers	0	
Top/Bottom Thickness	0.8	mm
Top/Bottom Pattern	Lines	
Z Seam X	150.0	mm
Z Seam Y	900	mm
Enable Ironing		
Infill		
Infill Density	10	%
Infill Pattern	Grid	
Material		
Default Printing Temperature	210.0	°C
Default Build Plate Temperature	50.0	°C
Flow	100	%
Enable Retraction	✔	
Retraction Distance	5.0	mm

< Recommended

4 hours 41 minutes
47g - 16.10m
Preview   Print with OctoPrint

money_cat_fill–150
77.5 x 56.5 x 89.9 mm

Installation

Requirements

- Cura 4.0

Install Cura 4

If you haven't Cura 4 installed yet, please proceed with it, and run it a first time. Running it at least once is required to go through your OS application signature check.

Reset Cura

Please refer to the Cura wiki to find the location for your *user settings* and *application cache* and clean them up. **You should already have backup your settings using Cura backup feature before the modification !**
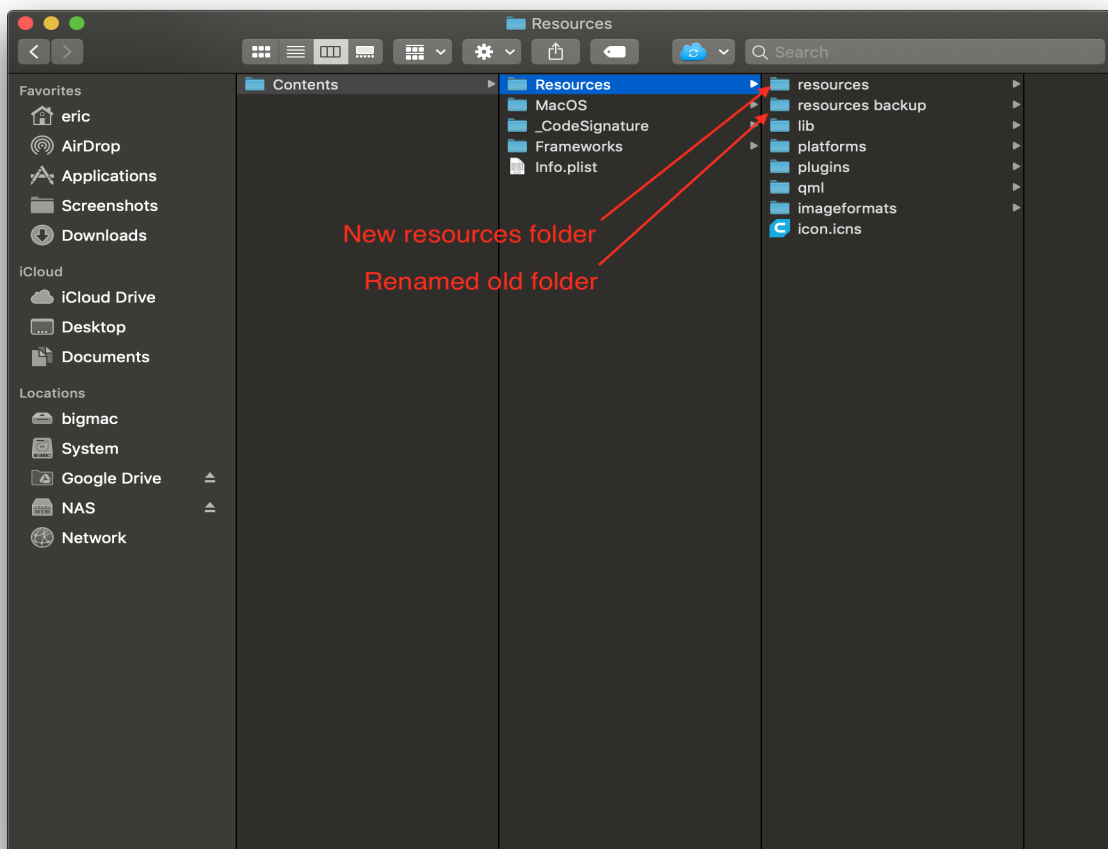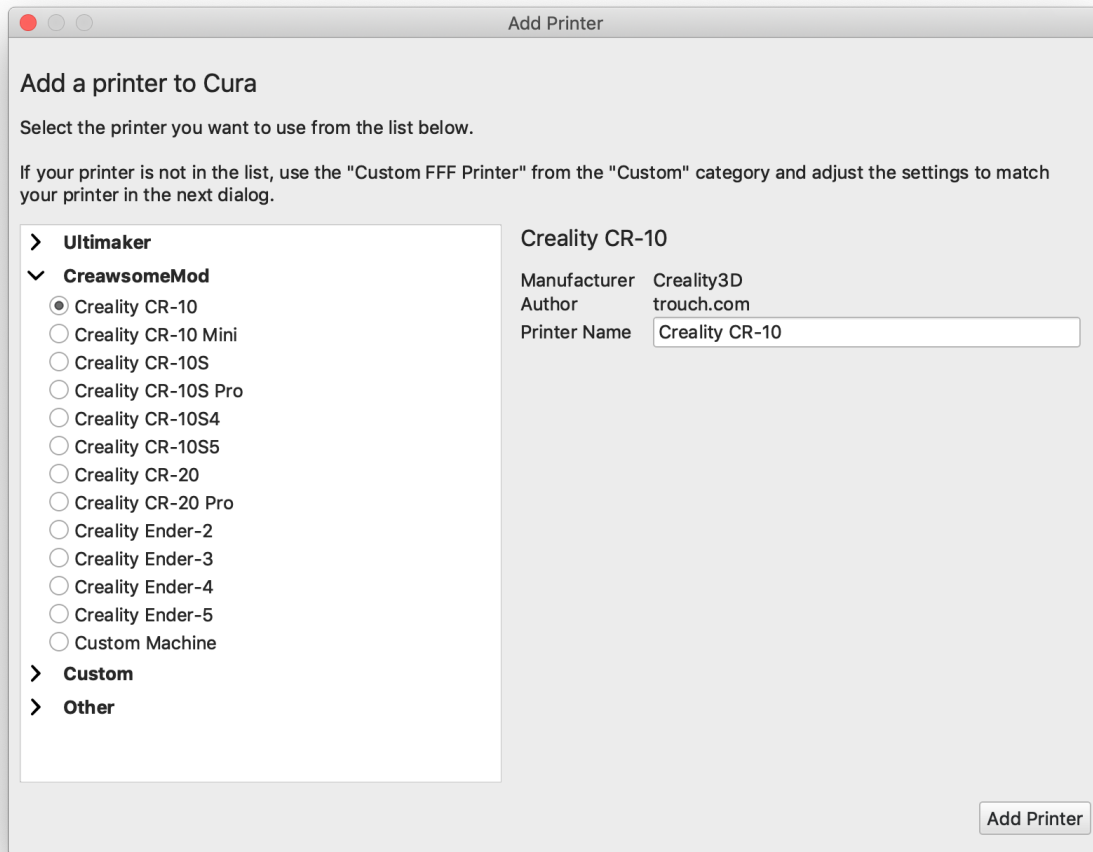
Install CreawsomeMod

Windows

Extract the zip file somewhere on your computer. Go to your `Program Files\Ultimaker Cura` folder, locate the `resources` folder and rename it for backup purpose. Copy and past the `resources` folder from the zip file to modify your Cura installation. To revert, simply delete the new folder and rename back the old one.
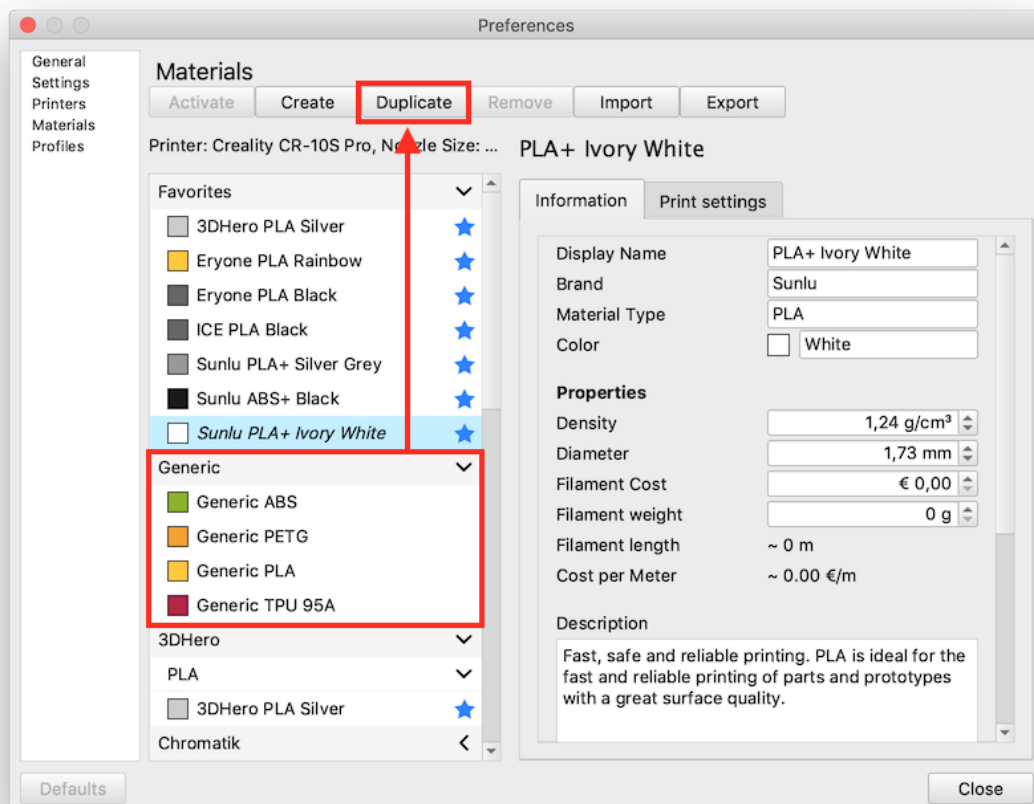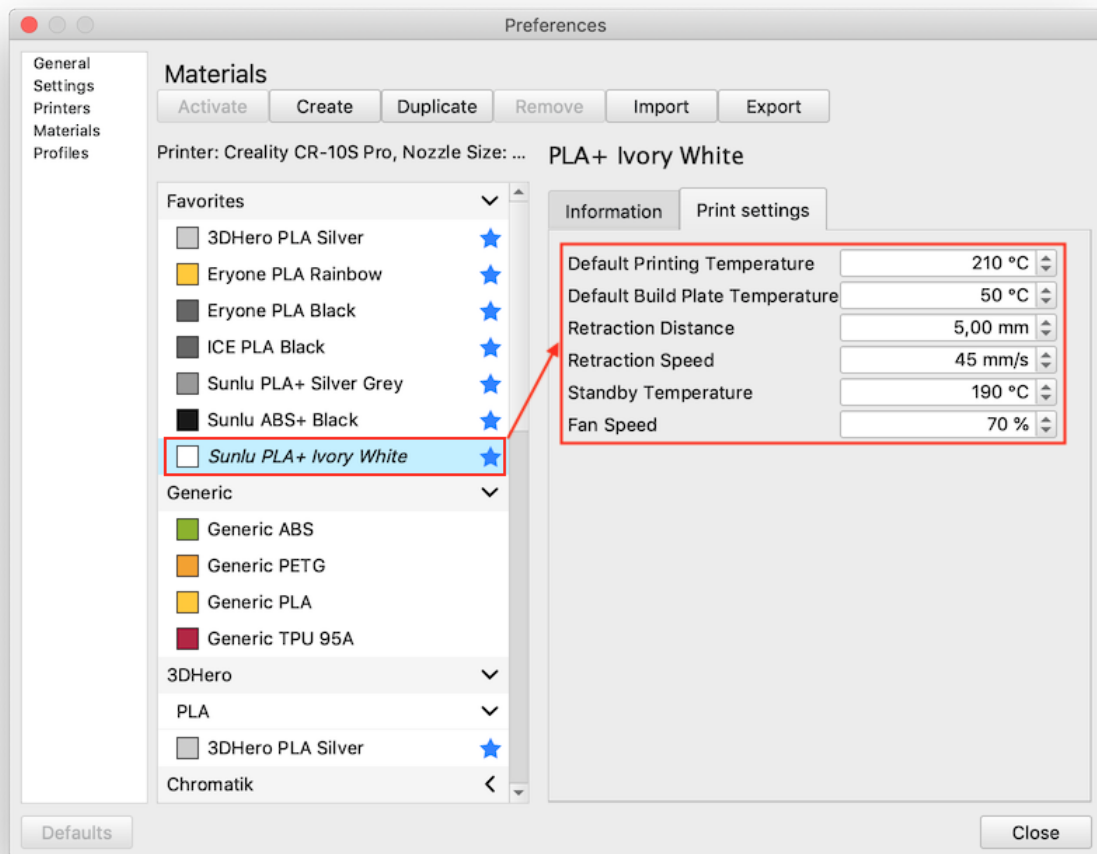
MacOS

Extract the zip file somewhere on your computer. Go to your `Applications` folder, select `Ultimaker Cura.app`, right-click or ctrl-click on it and select *Show Package Content*. Navigate through the folders to find the `resources` one and rename it for backup purpose. Copy and past the `resources` folder from the zip file to modify your Cura installation. To revert, simply delete the new folder and rename back the old one.

 Feel free to check the start g-code and adjust it to your need.

Material Settings

It is really important that you use the material profiles capabilities to duplicate Generic ones and adjust temperatures as well as retractation settings there for each of your spool !

Tuning Guide

No matter of the printer, slicer, mod or not, I highly recommend this Guide to tune your printer. Most importants are extruder and flow calibration as well as properly tuning temperature for each of your spools !

Known Issues

Clogging

If you encounter any nozzle clogging during retract, you probably have a small leak between the PTFE and nozzle ! Creality users, try this Original Creality hot end ptfe fix by Luke Hatfield, aka "OneBadMarine". This ensure to keep presure between the PTFE tube and nozzle to avoid any leak.

Missing print details

CreawsomeMod sets the line width to 125% per default for convenience in calculation and overral better looking. However some models aren't compatible with such setting. In case you lose details, or have some wall replaced with thin infill, try to reduce the line width to 0.45mm or 0.40mm.

Stringing

For most of Creality printer, the extruder feedrate is limited to 25mm/s ! CreawsomeMod includes a start g-code to set it to 50mm/s using an *M203* command and warns users when setting retraction above that value. Make sure to adjust start g-code if you need a retract speed above 50mm/s.

macOS user privileges

In case you can't modify your Cura install, please check your user privileges and Cura application access rights.

Incompatible user profiles

Since CreawsomeMod create new machine definitions with advanced features such as per-material preset and nozzle choice, any previous profile is not compatible with CreawsomeMod. Even it if was ever possible, it could bring unexpected behavior.

Z-Seam

Probably the most challenging issue with FDM. Per default, CreasomeMod sets the Z-seam at the back of the model. This gives amazing overall looking and accuracy, but it may brings a non esthetic line on curved models. That's CreawsomeMod's Achilles' heel.

1. **Don't set the seam to be random** as it will give overall worst result.
2. Change Z-seam location for your model as shown in the picture above
3. Fine tune your retract setting
4. Enable and calibrate Linear Advance (incompatible with "silent" Creality boards using TMC2208)
5. Use Cura coasting (incompatible with Linear Advance)