SECTION 7: SLICING FOR FDM PRINTING
SECTION 7.1: SLICING FOR ZORTRAX PRINTERS USING Z-SUITE

Select A Printer.

- M200
  Maximum Build Volume: 200 x 200 x 180 mm

- M300
  Maximum Build Volume: 300 x 300 x 300 mm

Download and Install Z-Suite. Use key Z4C694F96

Add your .stl or .obj 3D models by dragging in or locating through the ‘Add Files’ Browser.
SECTION 7.1: SLICING FOR ZORTRAX PRINTERS USING Z-SUITE

When orienting your model,
- Try to always print on a flat base
- Orient to limit overhangs
- Place print centrally to limit warpage
- Avoid scaling down a model unless you have already accounted for tolerances and minimum layer thicknesses
- If you are printing multiple .stls in one print, make sure they do not intersect each other.

Select the Model, and place your model in the ideal orientation for printing.

Use the MOVE, RESIZE & ROTATE functions. These can be accessed through either the top or side menu bar.

Once you are happy with the orientation of your model, move on to ANALYSIS.
SECTION 7.1 : SLICING FOR ZORTrax PRINTERS USING Z-SUITE

**ANALYSIS**

Analysis is a short stage of the Z-Suite process in which the model is scanned for unprintable features.

If your model is highlighted in RED, return to your modeling software and ensure:

- There are wall thicknesses on all parts of your model
- Your model is a close object containing no non-manifold edges
- No wall thickness should be less than 0.4mm

If your model is highlighted in GREEN, proceed to SUPPORT.
SECTION 7.1: SLICING FOR ZORTRAX PRINTERS USING Z-SUITE

SUPPORT

Support material is external, detachable helpers that print in the sections where your model needs it.

Unless you are confident that your model requires no support, ALWAYS generate support.

Support will generate and be shown in Grey. The support angle dictates the highest degree an overhang will be before support is generated. 20–45° is recommended.

The lower the angle, the greater the support.

Once support has been generated proceed to PRINT SETTINGS.
SECTION 7.1: SLICING FOR ZORTRAX PRINTERS USING Z-SUITE

PRINT SETTINGS - Normal options

1. Material Group: Zortrax Materials
   For M300’s: Z-HIPS, Z-GLASS
3. Layer Thickness: Layer thickness directly relates to the quality of your model. The thinner the layer, the higher the resolution or finish of your model. The thinner your layers the longer your print time.
4. Print Quality: Unless printing a final model, leave this setting at Normal. Draft can be used for fast 3D prints
5. Pattern: Pattern informs the shape of your infill, for most prints it is recommended to keep on PATT.0 as this is the quickest of all Z-Suites Patterns.
6. Infill Density: Infill is the internal support structure of a 3D print. Reducing wasted material by using layered 3 dimensional patterns infill saves on both print time and material. It is recommended to use 20–30% infill, only stepping up to 60% if your part will need severe structural integrity. No prints should be set to solid infill

Often flat models will require more infill to achieve a successful print.

Once you are happy with your print settings click PREVIEW to generate your sliced model.
SECTION 7.1 : SLICING FOR ZORTRAX PRINTERS USING Z-SUITE

PRINT SETTINGS
Advanced options

1. Material Group: see previous page
2. Material: see previous page
3. Support: LITE: recommended will generate support that is easier to remove
            SMART BRIDGES: recommended will generate less unnecessary support
4. Layer Thickness: see previous page
5. Print Quality: see previous page
6. Type: see previous page
7. Pattern: see previous page
8. Infill Density: see previous page
9. Surface Layers: Number of layers with FULL surface coverage. The more surface layers, the smoother the surface and the longer the print time.
                   Top: recommended 4–7
                   Bottom: recommended 3–5

Seam: A seam is the starting point of each 3D printed layer. A NORMAL seam will result in a distinct line on the edge of your model. This will lead to a faster print time. A RANDOM seam will result in no line, this will lead to a longer print time.

For further information regarding Surface Layers, Offsets, Seams and Layer Gaps visit https://support.zortrax.com/z-suite-2-manual/

Once you are happy with your print settings click PREVIEW to generate your sliced model.
SECTION 7.1: SLICING FOR ZORTRAX PRINTERS USING Z-SUITE

**PREVIEW**
Here you will see a preview of exactly how your model is expected to be printed.

- **Timeline:**
  This can be used to inspect different parts of your print. Ensure all your model is identified as ‘model’ and support structures are present.

- **Report:**
  This will give you an overview of your print settings AND print time.

  As you slide through the timeline, the report will update with the estimated print time. This can be useful to estimate how long you will need to stay for the first 10 layers of your print to complete.

**Move timeline slider to 100%**
If your slider is not at 100% your FULL PRINT WILL NOT EXPORT

**Click ‘Export File’**
SECTION 7.1 : SLICING FOR ZORTRAX PRINTERS USING Z-SUITE

- Deselect ‘Export the report to a .txt file’
- Click ‘EXPORT FILE’
- Save file to an SD card. Ensure extension is .zcode
Download and Install the latest version of Ultimaker Cura

Select your printer

If your printer is not already available, you will need to add a new printer.

Settings > Printer > Add Printer ...

Select the Ultimaker Extended +

Click 'Add Printer'
SECTION 7.2: SLICING FOR ULTIMAKER 2 EXTENDED + PRINTERS USING CURA

Re-orienting Model

Adding Model

Select the Model, and place your model in the ideal orientation for printing.

Use the MOVE, SCALE & ROTATE functions.
These can be accessed through the side toolbar

When orienting your model,
- Try to always print on a flat base
- Orient to limit overhangs
- Place print centrally to limit warpage
- Avoid scaling down a model unless you have already accounted for tolerances and minimum layer thicknesses
- If you are printing multiple .stls in one print, make sure they do not intersect each other.

Add your .stl or .obj 3D model/s by dragging in or locating through the ‘Add Files’ Browser.
Once your model is in the ideal orientation to print you can configure your print settings.

**MATERIAL**
PLA, the Ultimaker 2 Extended+ will only ever be loaded with PLA

**NOZZLE**
0.04mm

**LAYER HEIGHT | PRINT SPEED**
Your layer height dictates the quality of your print. The thinner the layer height, the higher the resolution of your print but the print time will be increased. We advise 0.15 mm for most 3D prints as these prints are relatively fast and of decent resolution.

**INFILL**
Infill is the internal support structure of a 3D print. Reducing wasted material by using layered 3-dimensional patterns. Infill saves on both print time and material. It is recommended to use 20–40% infill, only stepping up to 60% if your part will need severe structural integrity.

**SUPPORT**
Support material is external, detachable helpers that print along with your 3D model, allowing for gravity-defying overhangs. Unless you are confident your model requires no support, always Generate Support.

**BUILD PLATE ADHESION**
Build plate adhesion is additional material used to adhere your print to the build plate, ensuring your print does not detach from the build plate mid-print. Unless you require a glossy bottom surface to your print, always apply Build Plate Adhesion.

Once you have set up your print settings in RECOMMENDED, switch to the CUSTOM tab to further refine settings.
SECTION 7.2 : SLICING FOR ULTIMAKER 2 EXTENDED + PRINTERS USING CURA

1. **INFILL**
   - Infill Density
     - It is recommended to use 20-40% infill, only stepping up to 60% if your part will need severe structural integrity.
   - Infill Pattern can be selected using the drop down:
     - Grid: Strong 2D infill
     - Triangles: Strong 2D infill
     - Cubic: Strong 3D infill
     - Octet: Strong 3D infill
     - Concentric: Flexible 3D infill
     - Cross: Flexible 3D infill
     - Zig-zag: A grid-shaped infill, printing continuously in one diagonal direction

2. **SUPPORT**
   - Generate Support = Y
   - Support Placement = Everywhere
   - Support Overhang = 45
   - Support Pattern = Zig Zag or Triangle are recommend as they are the easiest to remove

3. **BUILD PLATE ADHESION**
   - Build Plate Adhesion type can be selected using the drop down:
     - Raft (recommended): A raft adds a thick grid with a roof between the model and the build plate
     - Brim: A brim adds a single layer flat area around the base of the model to prevent warping.
     - Skirt: A skirt is a line printed around the object on the first layer, but not connected to the object.

Once you are happy with all settings click ‘Prepare’ to slice your file. This will take a few seconds - minutes depending on the complexity of your file.
Ensure your model, support and build plate adhesion have all fully loaded.

The timeline sliders can be used to inspect the model.

Take note of your print time and make a booking using the TeamUp Calendar.

Click the arrow beside ‘Save to Removable Drive’ and select ‘Save to File’.

Once file is sliced change your view to Layer to see how your model will print.
Click ‘Save to File’ and navigate to your SD Card. Do not delete the ‘UM2E’ prefix. Ensure the file is saved as a .gcode (not gcode.gz)
Download and install the latest version of Ultimaker Cura.

Select your printer.

If your printer is not already available, you will need to add a new printer.

Settings > Printer > Add Printer...

Select the Ultimaker Extended +.

Click ‘Add Printer’.
Select the Model, and place your model in the ideal orientation for printing.

Use the MOVE, SCALE & ROTATE functions. These can be accessed through the side toolbar.

When orienting your model,
- Try to always print on a flat base
- Orient to limit overhangs
- Place print centrally to limit warpage
- Avoid scaling down a model unless you have already accounted for tolerances and minimum layer thicknesses
- If you are printing multiple .stls in one print, make sure they do not intersect each other.
Once your model is in the ideal orientation to print you can configure your print settings. Firstly,

1. **Select Extruder 2**

   **MATERIAL**
   PVA, the Ultimaker Extended 3’s Extruder 2 will always be set to PVA, however, you will not print using this extruder.

2. **PRINT CORE**
   BB 0.4

3. **PRINT CORE**
   AA 0.4

4. **Select Extruder 1**

   **MATERIAL**
   PLA, the Ultimaker Extended 3’s Extruder 1 will always print with PLA

5. **PRINT CORE**
   AA 0.4

6. **LAYERS HEIGHT | PRINT SPEED**
   Your layer height dictates the quality of your print. The thinner the layer height, the higher the resolution of your print but the print time will be increased. We advise 0.15 mm for most 3D prints as these prints are relatively fast and of decent resolution.

7. **INFLILL**
   Infill is the internal support structure of a 3D print. Reducing wasted material by using layered 3 dimensional patterns. Infill saves on both print time and material. It is recommended to use 20-40% infill, only stepping up to 60% if your part will need severe structural integrity.

8. **SUPPORT. Always Set Support Extruder to Extruder 1**
   Support material is external, detachable helpers that print along with your 3D model, allowing for gravity defying overhangs. Unless you are confident your model requires no support, always Generate Support.

9. **BUILD PLATE ADHESION**
   Build plate adhesion is additional material used to adhere your print to the build plate, ensuring your print does not detach from the build plate mid print. Once you have set up your print settings in RECOMMENDED, switch to the CUSTOM tab to further refine settings.
SECTION 7.3: SLICING FOR ULTIMAKER 3 EXTENDED PRINTERS USING CURA

**INFILL**

- **Infill Density**
  
  It is recommended to use 20-40% infill, only stepping up to 60% if your part will need severe structural integrity.

- **Infill Pattern** can be selected using the drop-down:
  
  - Grid: Strong 2D infill
  - Triangles: Strong 2D infill
  - Cubic: Strong 3D infill
  - Octet: Strong 3D infill
  - Concentric: Flexible 3D infill
  - Cross: Flexible 3D infill
  - Zig-zag: A grid shaped infill, printing continuously in one diagonal direction

**SUPPORT**

- **Generate Support** = Y
- **Support Extruder** = Extruder 1
- **Support Placement** = Everywhere
- **Support Overhang** = 45-60
- **Support Pattern** = Zig-Zag or Triangle are recommended as they are the easiest to remove

**BUILD PLATE ADHESION**

- **Build Plate Adhesion Type** can be selected using the drop-down:
  
  - Raft (recommended): A raft adds a thick grid with a roof between the model and the build plate
  - Brim: A brim adds a single layer flat area around the base of the model to prevent warping.
  - Skirt: A skirt is a line printed around the object on the first layer, but not connected to the object.

- **Build Plate Adhesion Extruder**: Extruder 1

Once you are happy with all settings click ‘Prepare’ to slice your file. This will take a few seconds - minutes depending on the complexity of your file.
SECTION 7.3 : SLICING FOR ULTIMAKER 3 EXTENDED PRINTERS USING CURA

Once file is sliced change your view to Layer to see how your model will print.

Ensure your model, support and build plate adhesion have all fully loaded.

The timeline sliders can be used to inspect the model.

Take note of your print time and make a booking using the TeamUp Calendar.
Click ‘Save to File’ and navigate to your SD Card.

Do not delete the ‘UM2E’ prefix.

Ensure the file is saved as a .gcode (not .gcode.gz)
SECTION 7.4: SLICING CHECKLIST

This handy checklist is included to help you identify issues you may encounter while slicing your 3D print. The items highlighted in red are examples of errors that can be easily identified and solved prior to exporting your file for printing.

It is strongly recommended that you browse the table and double check your sliced file for any of the potential errors mentioned - as this can save much time and prevent unnecessary stress!

So please ensure that...

<table>
<thead>
<tr>
<th>SLICING CHECKLIST</th>
<th>Yes</th>
<th>No</th>
</tr>
</thead>
<tbody>
<tr>
<td>You have consulted the Slicing Guide?</td>
<td></td>
<td></td>
</tr>
<tr>
<td>You chose the correct printer model in the slicing software?</td>
<td></td>
<td></td>
</tr>
<tr>
<td>You chose the correct material type?</td>
<td></td>
<td></td>
</tr>
<tr>
<td>You chose the correct infill amount?</td>
<td></td>
<td></td>
</tr>
<tr>
<td>You have added a raft to your model?</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Your model is in the ideal orientation for its geometry?</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Your model has enough surface contact with the build platform to adhere well?</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Your model is within the recommended build volume?</td>
<td></td>
<td></td>
</tr>
<tr>
<td>You do not have intersecting models?</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

1. **RECTIFY**

2. **PROCEED**