










## Board Setup

- Board Stackup
  - Board Editor Layers
  - Physical Stackup
  - Board Finish
  - Solder Mask/Paste
- Text & Graphics
  - Defaults
  - Formatting
  - Text Variables
- Design Rules
  - Constraints
  - Pre-defined Sizes
  - Teardrops
  - Length-tuning Patterns
  - Net Classes
  - Custom Rules
  - Violation Severity



### Copper

	Minimum clearance:	<input type="text" value="0.127"/>	mm
	Minimum track width:	<input type="text" value="0.2"/>	mm
	Minimum connection width:	<input type="text" value="0.25"/>	mm
	Minimum annular width:	<input type="text" value="0.05"/>	mm
	Minimum via diameter:	<input type="text" value="0.4"/>	mm
	Copper to hole clearance:	<input type="text" value="0.254"/>	mm
	Copper to edge clearance:	<input type="text" value="1"/>	mm

### Holes

	Minimum through hole:	<input type="text" value="0.3"/>	mm
	Hole to hole clearance:	<input type="text" value="0.25"/>	mm

### uVias

	Minimum uVia diameter:	<input type="text" value="0.2"/>	mm
	Minimum uVia hole:	<input type="text" value="0.1"/>	mm

### Silkscreen


Minimum item clearance:	<input type="text" value="0.153"/>	mm
Minimum text height:	<input type="text" value="1"/>	mm
Minimum text thickness:	<input type="text" value="0.153"/>	mm


### Arc/circle approximated by segments

Maximum allowed deviation:  mm

Note: zone filling can be slow when < 0.005 mm.

### Zone fill strategy

 ☐ Allow fillets outside zone outline

 Minimum thermal relief spoke count:

### Length tuning

☐ Include stackup height in track length calculations

Import Settings from Another Board...

OK

Cancel

Board Setup

Board Stackup

Board Editor Layers

Physical Stackup

Board Finish

Solder Mask/Paste

Text & Graphics

Defaults

Formatting

Text Variables

Design Rules

Constraints

Pre-defined Sizes

Teardrops

Length-tuning Patterns

Net Classes

Custom Rules

Violation Severity

Copper layers: 2

☐ Impedance controlled

Add Dielectric Layer...

Remove Dielectric Layer...

Layer	Id	Type	Material	Thickness		Color	Epsilon R	Loss Tan
	F.Silkscreen	Top Silk Screen	Not specified			White		
	F.Paste	Top Solder Paste						
	F.Mask	Top Solder Mask	Not specified	0.01 mm		Not specified	3.3	0
	F.Cu	Copper		0.035 mm				
	Dielectric 1	Core	FR4	1.51 mm	<input checked="" type="checkbox"/>	Not specified	4.5	0.02
	B.Cu	Copper		0.035 mm				
	B.Mask	Bottom Solder Mask	Not specified	0.01 mm		Not specified	3.3	0
	B.Paste	Bottom Solder Paste						
	B.Silkscreen	Bottom Silk Screen	Not specified			White		

Board thickness from stackup: 1.6 mm

Adjust Dielectric Thickness

Export to Clipboard

Import Settings from Another Board...

OK

Cancel

## Board Setup

- Board Stackup
  - Board Editor Layers
  - Physical Stackup
  - Board Finish
  - Solder Mask/Paste
- Text & Graphics
  - Defaults
  - Formatting
  - Text Variables
- Design Rules
  - Constraints
  - Pre-defined Sizes
  - Teardrops
  - Length-tuning Patterns
  - Net Classes
  - Custom Rules
  - Violation Severity

☐ Has castellated pads

☐ Plated board edge

Copper finish: HAL SnPb

Edge card connectors: None

Import Settings from Another Board...

OK

Cancel

## Board Setup

- Board Stackup
  - Board Editor Layers
  - Physical Stackup
  - Board Finish
  - Solder Mask/Paste
- Text & Graphics
  - Defaults
  - Formatting
  - Text Variables
- Design Rules
  - Constraints
  - Pre-defined Sizes
  - Teardrops
  - Length-tuning Patterns
  - Net Classes
  - Custom Rules
  - Violation Severity



Use your board manufacturer's recommendations for solder mask expansion and minimum web width.  
If none are provided, setting the values to zero is suggested.

Solder mask expansion:  mm

Solder mask minimum web width:  mm

Solder mask to copper clearance:  mm

☐ Allow bridged solder mask apertures between pads within footprints

☒ Tent vias

Solder paste absolute clearance:  mm

Solder paste relative clearance:  %

*Note: Solder paste clearances (absolute and relative) are added to determine the final clearance.*

Import Settings from Another Board...

OK

Cancel

## Board Setup

- Board Stackup
  - Board Editor Layers
  - Physical Stackup
  - Board Finish
  - Solder Mask/Paste
- Text & Graphics
  - Defaults
  - Formatting
  - Text Variables
- Design Rules
  - Constraints
  - Pre-defined Sizes
  - Teardrops
  - Length-tuning Patterns
  - Net Classes
  - Custom Rules
  - Violation Severity

## Default properties for new graphics and text:

	Line Thickness	Text Width	Text Height	Text Thickness	Italic	Keep Upright
Silk Layers	0.12 mm	1 mm	1 mm	0.15 mm	<input type="checkbox"/>	<input type="checkbox"/>
Copper Layers	0.2 mm	1.5 mm	1.5 mm	0.3 mm	<input type="checkbox"/>	<input type="checkbox"/>
Edge Cuts	0.05 mm					
Courtyards	0.05 mm					
Fab Layers	0.1 mm	1 mm	1 mm	0.15 mm	<input type="checkbox"/>	<input type="checkbox"/>
Other Layers	0.1 mm	1 mm	1 mm	0.15 mm	<input type="checkbox"/>	<input type="checkbox"/>

## Default properties for new dimension objects:

Units: Automatic ▾

Units format: 1234 mm ▾

Precision: 0.0000 ▾

☐ Suppress trailing zeroes

Text position: Outside ▾

☒ Keep text aligned

Arrow length: 1.27 mm

Extension line offset: 0.5 mm

Import Settings from Another Board...

OK

Cancel

