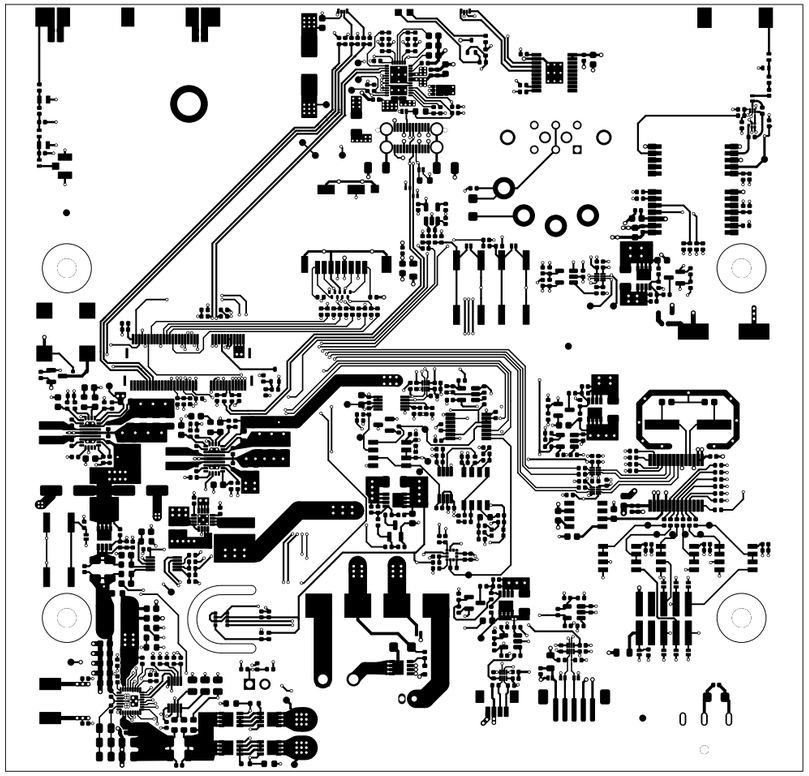


Layer Name	Type	Material	Thickness (mm)	Color	Epsilon R	Loss Tangent
F.Silkscreen	Top Silk Screen	Not specified	0 mm	White	1	0
F.Paste	Top Solder Paste		0 mm		1	0
F.Mask	Top Solder Mask	Not specified	0.02 mm	Black	3.3	0
Top	copper		0.035 mm		1	0
Dielectric	prepreg	FR4	0.2 mm	Not specified	4.6	0.02
Gnd	copper		0.035 mm		1	0
Dielectric	core	FR4	1 mm	Not specified	4.6	0.02
Power	copper		0.035 mm		1	0
Dielectric	prepreg	FR4	0.2 mm	Not specified	4.6	0.02
Bottom	copper		0.035 mm		1	0
B.Mask	Bottom Solder Mask	Not specified	0.02 mm	Black	3.3	0
B.Paste	Bottom Solder Paste		0 mm		1	0
B.Silkscreen	Bottom Silk Screen	Not specified	0 mm	Not specified	1	0

BOARD CHARACTERISTICS

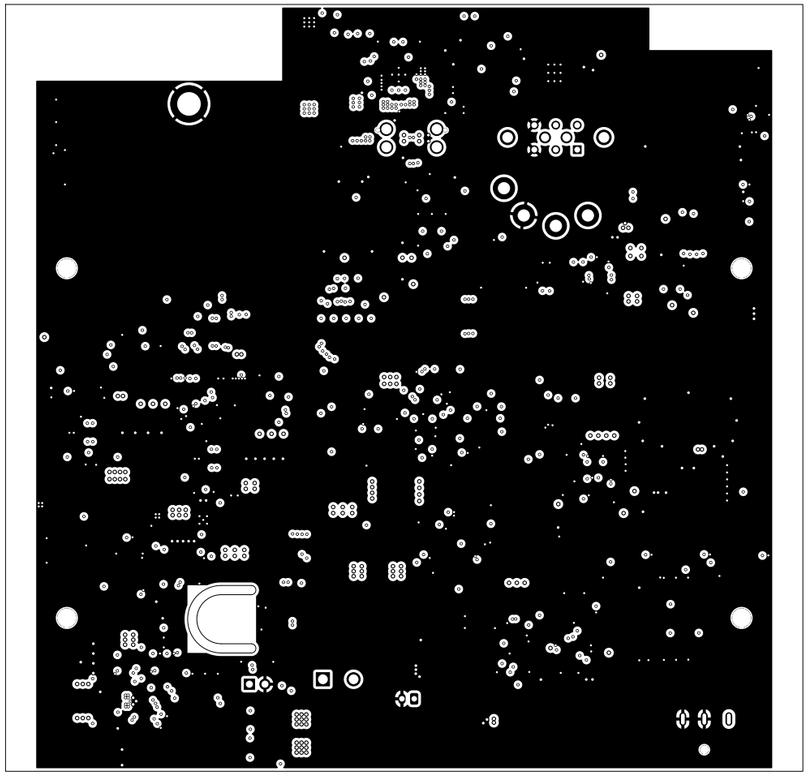
Copper Layer Count:	4	Board Thickness:	1.5800 mm
Board overall dimensions:	130.0000 mm x 125.0000 mm		
Min track/spacing:	0.1000 mm / 0.2000 mm	Min hole diameter:	0.2000 mm
Copper Finish:	HAL lead-free	Impedance Control:	Yes
Castellated pads:	No	Plated Board Edge:	No
Edge card connectors:	No		

IMPEDANCE CONTROL TABLE

LAYER	TRACE (MM)	SPACING (MM)	IMPEDANCE SINGLE-ENDED	IMPEDANCE DIFFERENTIAL	TOLLERANCE
TOP	0.35	–	50ohm	–	+/-10%
TOP/BOTTOM	0.27	0.2	–	90ohm	+/-10%

FAE TECHNOLOGY
via C.Battisti 136, 24025, Gazzaniga (Bg), Italy
Mail: info@fae.technology Tel: +39 035738130

Project **Flatburn_V4**		Board name **Flatburn_V4_layout**
Designer **Luca Brighenti**		
Internal code **-**		Customer **MIT**
Date 07/03/25	Rev. **1**	Rev. changes See Root Page
		Kicad Version **KiCad E.D.A. 9.0.0**
		Sheet **1/12**



Layer Name	Type	Material	Thickness (mm)	Color	Epsilon R	Loss Tangent
F.Silkscreen	Top Silk Screen	Not specified	0 mm	White	1	0
F.Paste	Top Solder Paste		0 mm		1	0
F.Mask	Top Solder Mask	Not specified	0.02 mm	Black	3.3	0
Top	copper		0.035 mm		1	0
Dielectric	prepreg	FR4	0.2 mm	Not specified	4.6	0.02
Gnd	copper		0.035 mm		1	0
Dielectric	core	FR4	1 mm	Not specified	4.6	0.02
Power	copper		0.035 mm		1	0
Dielectric	prepreg	FR4	0.2 mm	Not specified	4.6	0.02
Bottom	copper		0.035 mm		1	0
B.Mask	Bottom Solder Mask	Not specified	0.02 mm	Black	3.3	0
B.Paste	Bottom Solder Paste		0 mm		1	0
B.Silkscreen	Bottom Silk Screen	Not specified	0 mm	Not specified	1	0

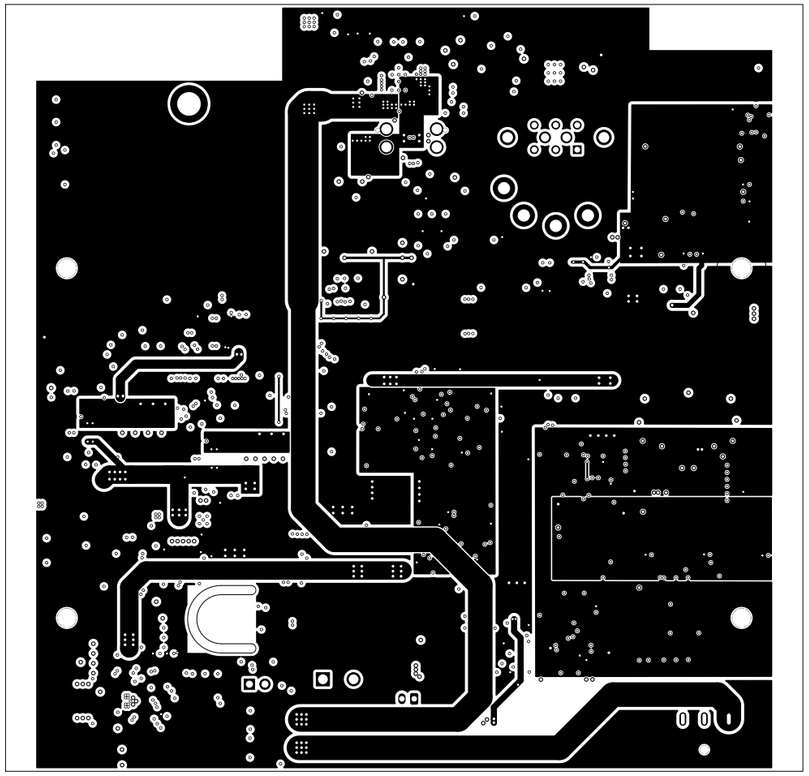
BOARD CHARACTERISTICS

Copper Layer Count:	4	Board Thickness:	1.5800 mm
Board overall dimensions:	130.0000 mm x 125.0000 mm		
Min track/spacing:	0.1000 mm / 0.2000 mm	Min hole diameter:	0.2000 mm
Copper Finish:	HAL lead-free	Impedance Control:	Yes
Castellated pads:	No	Plated Board Edge:	No
Edge card connectors:	No		

IMPEDANCE CONTROL TABLE					
LAYER	TRACE (MM)	SPACING (MM)	IMPEDANCE SINGLE-ENDED	IMPEDANCE DIFFERENTIAL	TOLLERANCE
TOP	0.35	-	50ohm	-	+/-10%
TOP/BOTTOM	0.27	0.2	-	90ohm	+/-10%

FAE TECHNOLOGY
via C.Battisti 136, 24025, Gazzaniga (Bg), Italy
Mail: info@fae.technology Tel: +39 035738130

Project	**Flatburn_V4**	Board name	**Flatburn_V4_layout**
Designer	**Luca Brighenti**		
Internal code	**-**	Customer	**MIT**
Date 07/03/25	Rev. **1**	Rev. changes See Root Page	Kicad Version **KiCad E.D.A. 9.0.0** / Sheet **2/12**



Layer Name	Type	Material	Thickness (mm)	Color	Epsilon R	Loss Tangent
F.Silkscreen	Top Silk Screen	Not specified	0 mm	White	1	0
F.Paste	Top Solder Paste		0 mm		1	0
F.Mask	Top Solder Mask	Not specified	0.02 mm	Black	3.3	0
Top	copper		0.035 mm		1	0
Dielectric	prepreg	FR4	0.2 mm	Not specified	4.6	0.02
Gnd	copper		0.035 mm		1	0
Dielectric	core	FR4	1 mm	Not specified	4.6	0.02
Power	copper		0.035 mm		1	0
Dielectric	prepreg	FR4	0.2 mm	Not specified	4.6	0.02
Bottom	copper		0.035 mm		1	0
B.Mask	Bottom Solder Mask	Not specified	0.02 mm	Black	3.3	0
B.Paste	Bottom Solder Paste		0 mm		1	0
B.Silkscreen	Bottom Silk Screen	Not specified	0 mm	Not specified	1	0

BOARD CHARACTERISTICS

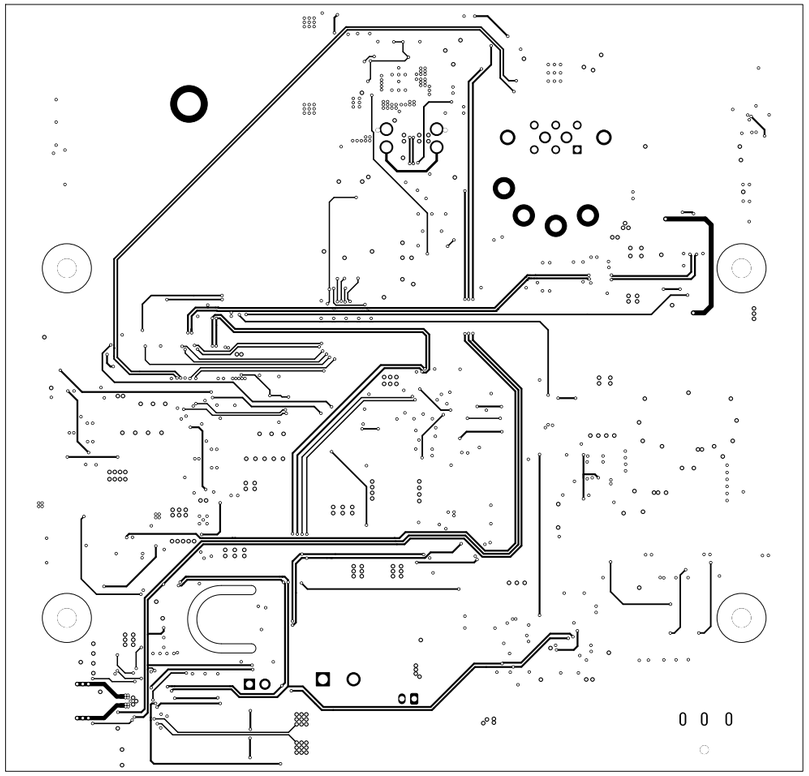
Copper Layer Count:	4	Board Thickness:	1.5800 mm
Board overall dimensions:	130.0000 mm x 125.0000 mm		
Min track/spacing:	0.1000 mm / 0.2000 mm	Min hole diameter:	0.2000 mm
Copper Finish:	HAL lead-free	Impedance Control:	Yes
Castellated pads:	No	Plated Board Edge:	No
Edge card connectors:	No		

IMPEDANCE CONTROL TABLE

LAYER	TRACE (MM)	SPACING (MM)	IMPEDANCE SINGLE-ENDED	IMPEDANCE DIFFERENTIAL	TOLLERANCE
TOP	0.35	-	50ohm	-	+/-10%
TOP/BOTTOM	0.27	0.2	-	90ohm	+/-10%

FAE TECHNOLOGY
via C.Battisti 136, 24025, Gazzaniga (Bg), Italy
Mail: info@fae.technology Tel: +39 035738130

Project	**Flatburn_V4**	Board name	**Flatburn_V4_layout**
Designer	**Luca Brighenti**		
Internal code	**-**	Customer	**MIT**
Date 07/03/25	Rev. **1**	Rev. changes See Root Page	Kicad Version **KiCad E.D.A. 9.0.0** — Sheet **3/12**



Layer Name	Type	Material	Thickness (mm)	Color	Epsilon R	Loss Tangent
F.Silkscreen	Top Silk Screen	Not specified	0 mm	White	1	0
F.Paste	Top Solder Paste		0 mm		1	0
F.Mask	Top Solder Mask	Not specified	0.02 mm	Black	3.3	0
Top	copper		0.035 mm		1	0
Dielectric	prepreg	FR4	0.2 mm	Not specified	4.6	0.02
Gnd	copper		0.035 mm		1	0
Dielectric	core	FR4	1 mm	Not specified	4.6	0.02
Power	copper		0.035 mm		1	0
Dielectric	prepreg	FR4	0.2 mm	Not specified	4.6	0.02
Bottom	copper		0.035 mm		1	0
B.Mask	Bottom Solder Mask	Not specified	0.02 mm	Black	3.3	0
B.Paste	Bottom Solder Paste		0 mm		1	0
B.Silkscreen	Bottom Silk Screen	Not specified	0 mm	Not specified	1	0

BOARD CHARACTERISTICS

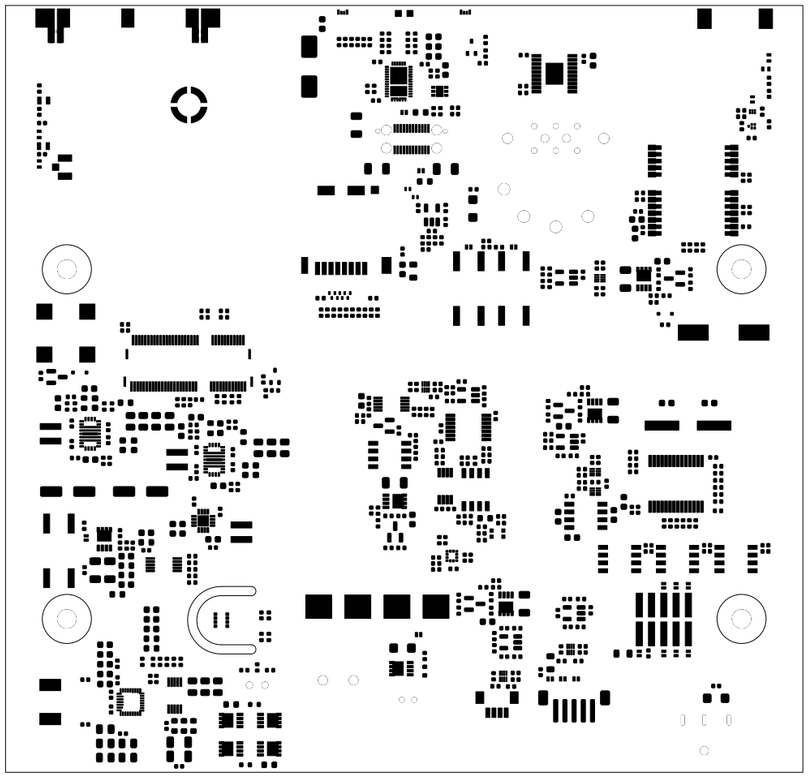
Copper Layer Count:	4	Board Thickness:	1.5800 mm
Board overall dimensions:	130.0000 mm x 125.0000 mm		
Min track/spacing:	0.1000 mm / 0.2000 mm	Min hole diameter:	0.2000 mm
Copper Finish:	HAL lead-free	Impedance Control:	Yes
Castellated pads:	No	Plated Board Edge:	No
Edge card connectors:	No		

IMPEDANCE CONTROL TABLE

LAYER	TRACE (MM)	SPACING (MM)	IMPEDANCE SINGLE-ENDED	IMPEDANCE DIFFERENTIAL	TOLLERANCE
TOP	0.35	-	50ohm	-	+/-10%
TOP/BOTTOM	0.27	0.2	-	90ohm	+/-10%

FAE TECHNOLOGY
via C.Battisti 136, 24025, Gazzaniga (Bg), Italy
Mail: info@fae.technology Tel: +39 035738130

Project	**Flatburn_V4**	Board name	**Flatburn_V4_layout**
Designer	**Luca Brighenti**		
Internal code	**-**	Customer	**MIT**
Date 07/03/25	Rev. **1**	Rev. changes See Root Page	Kicad Version **KiCad E.D.A. 9.0.0** — Sheet **4/12**



Layer Name	Type	Material	Thickness (mm)	Color	Epsilon R	Loss Tangent
F.Silkscreen	Top Silk Screen	Not specified	0 mm	White	1	0
F.Paste	Top Solder Paste		0 mm		1	0
F.Mask	Top Solder Mask	Not specified	0.02 mm	Black	3.3	0
Top	copper		0.035 mm		1	0
Dielectric	prepreg	FR4	0.2 mm	Not specified	4.6	0.02
Gnd	copper		0.035 mm		1	0
Dielectric	core	FR4	1 mm	Not specified	4.6	0.02
Power	copper		0.035 mm		1	0
Dielectric	prepreg	FR4	0.2 mm	Not specified	4.6	0.02
Bottom	copper		0.035 mm		1	0
B.Mask	Bottom Solder Mask	Not specified	0.02 mm	Black	3.3	0
B.Paste	Bottom Solder Paste		0 mm		1	0
B.Silkscreen	Bottom Silk Screen	Not specified	0 mm	Not specified	1	0

BOARD CHARACTERISTICS

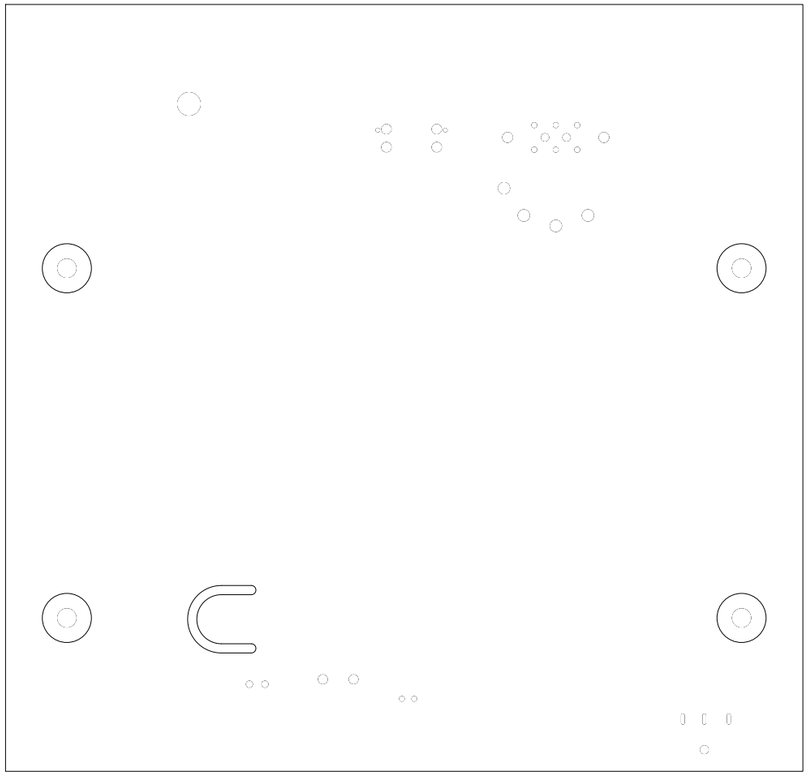
Copper Layer Count:	4	Board Thickness:	1.5800 mm
Board overall dimensions:	130.0000 mm x 125.0000 mm		
Min track/spacing:	0.1000 mm / 0.2000 mm	Min hole diameter:	0.2000 mm
Copper Finish:	HAL lead-free	Impedance Control:	Yes
Castellated pads:	No	Plated Board Edge:	No
Edge card connectors:	No		

IMPEDANCE CONTROL TABLE

LAYER	TRACE (MM)	SPACING (MM)	IMPEDANCE SINGLE-ENDED	IMPEDANCE DIFFERENTIAL	TOLLERANCE
TOP	0.35	–	50ohm	–	+/-10%
TOP/BOTTOM	0.27	0.2	–	90ohm	+/-10%

FAE TECHNOLOGY
via C.Battisti 136, 24025, Gazzaniga (Bg), Italy
Mail: info@fae.technology Tel: +39 035738130

Project **Flatburn_V4**		Board name	**Flatburn_V4_layout**
Designer **Luca Brighenti**			
Internal code **-**		Customer **MIT**	
Date 07/03/25	Rev. **1**	Rev. changes See Root Page	Kicad Version **KiCad E.D.A. 9.0.0** Sheet **5/12**



Layer Name	Type	Material	Thickness (mm)	Color	Epsilon R	Loss Tangent
F.Silkscreen	Top Silk Screen	Not specified	0 mm	White	1	0
F.Paste	Top Solder Paste		0 mm		1	0
F.Mask	Top Solder Mask	Not specified	0.02 mm	Black	3.3	0
Top	copper		0.035 mm		1	0
Dielectric	prepreg	FR4	0.2 mm	Not specified	4.6	0.02
Gnd	copper		0.035 mm		1	0
Dielectric	core	FR4	1 mm	Not specified	4.6	0.02
Power	copper		0.035 mm		1	0
Dielectric	prepreg	FR4	0.2 mm	Not specified	4.6	0.02
Bottom	copper		0.035 mm		1	0
B.Mask	Bottom Solder Mask	Not specified	0.02 mm	Black	3.3	0
B.Paste	Bottom Solder Paste		0 mm		1	0
B.Silkscreen	Bottom Silk Screen	Not specified	0 mm	Not specified	1	0

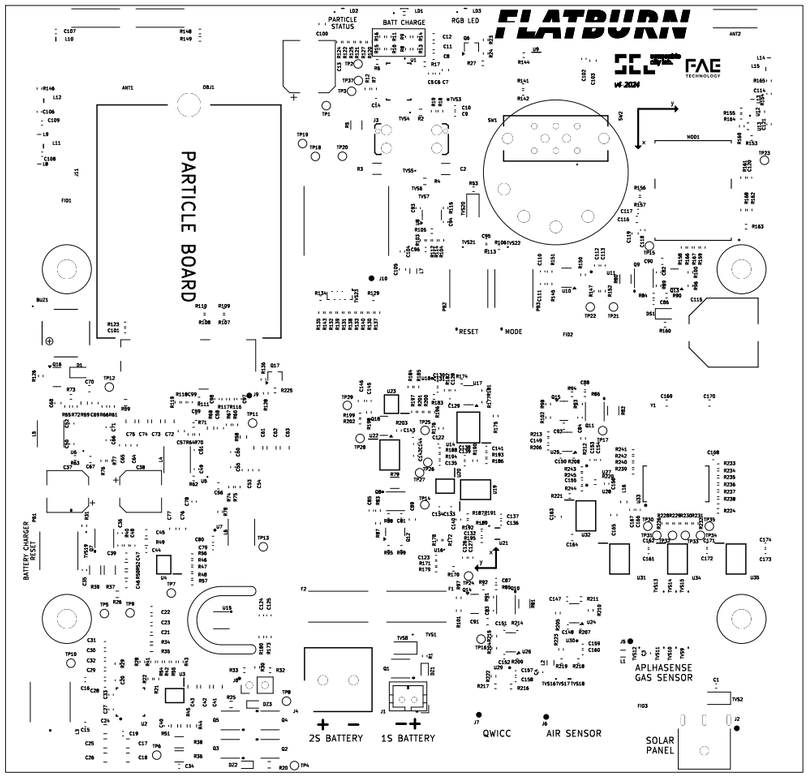
BOARD CHARACTERISTICS

Copper Layer Count:	4	Board Thickness:	1.5800 mm
Board overall dimensions:	130.0000 mm x 125.0000 mm		
Min track/spacing:	0.1000 mm / 0.2000 mm	Min hole diameter:	0.2000 mm
Copper Finish:	HAL lead-free	Impedance Control:	Yes
Castellated pads:	No	Plated Board Edge:	No
Edge card connectors:	No		

IMPEDANCE CONTROL TABLE					
LAYER	TRACE (MM)	SPACING (MM)	IMPEDANCE SINGLE-ENDED	IMPEDANCE DIFFERENTIAL	TOLLERANCE
TOP	0.35	-	50ohm	-	+/-10%
TOP/BOTTOM	0.27	0.2	-	90ohm	+/-10%

FAE TECHNOLOGY
via C.Battisti 136, 24025, Gazzaniga (Bg), Italy
Mail: info@fae.technology Tel: +39 035738130

Project **Flatburn_V4**		Board name **Flatburn_V4_layout**	
Designer **Luca Brighenti**			
Internal code **-**		Customer **MIT**	
Date 07/03/25	Rev. **1**	Rev. changes See Root Page	Kicad Version **KiCad E.D.A. 9.0.0** / Sheet **6/12**



Layer Name	Type	Material	Thickness (mm)	Color	Epsilon R	Loss Tangent
F.Silkscreen	Top Silk Screen	Not specified	0 mm	White	1	0
F.Paste	Top Solder Paste		0 mm		1	0
F.Mask	Top Solder Mask	Not specified	0.02 mm	Black	3.3	0
Top	copper		0.035 mm		1	0
Dielectric	prepreg	FR4	0.2 mm	Not specified	4.6	0.02
Gnd	copper		0.035 mm		1	0
Dielectric	core	FR4	1 mm	Not specified	4.6	0.02
Power	copper		0.035 mm		1	0
Dielectric	prepreg	FR4	0.2 mm	Not specified	4.6	0.02
Bottom	copper		0.035 mm		1	0
B.Mask	Bottom Solder Mask	Not specified	0.02 mm	Black	3.3	0
B.Paste	Bottom Solder Paste		0 mm		1	0
B.Silkscreen	Bottom Silk Screen	Not specified	0 mm	Not specified	1	0

BOARD CHARACTERISTICS

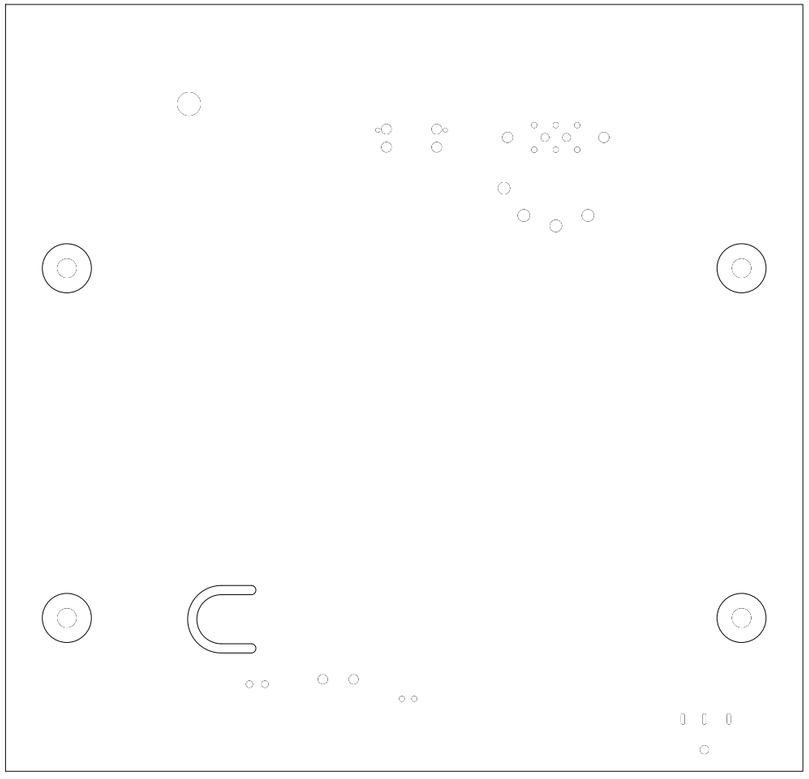
Copper Layer Count:	4	Board Thickness:	1.5800 mm
Board overall dimensions:	130.0000 mm x 125.0000 mm		
Min track/spacing:	0.1000 mm / 0.2000 mm	Min hole diameter:	0.2000 mm
Copper Finish:	HAL lead-free	Impedance Control:	Yes
Castellated pads:	No	Plated Board Edge:	No
Edge card connectors:	No		

IMPEDANCE CONTROL TABLE

LAYER	TRACE (MM)	SPACING (MM)	IMPEDANCE SINGLE-ENDED	IMPEDANCE DIFFERENTIAL	TOLLERANCE
TOP	0.35	–	50ohm	–	+/-10%
TOP/BOTTOM	0.27	0.2	–	90ohm	+/-10%

FAE TECHNOLOGY
via C.Battisti 136, 24025, Gazzaniga (Bg), Italy
Mail: info@fae.technology Tel: +39 035738130

Project **Flatburn_V4**		Board name **Flatburn_V4_layout**	
Designer **Luca Brighenti**			
Internal code **-**		Customer **MIT**	
Date 07/03/25	Rev. **1**	Rev. changes See Root Page	Kicad Version **KiCad E.D.A. 9.0.0** / Sheet **7/12**



Layer Name	Type	Material	Thickness (mm)	Color	Epsilon R	Loss Tangent
F.Silkscreen	Top Silk Screen	Not specified	0 mm	White	1	0
F.Paste	Top Solder Paste		0 mm		1	0
F.Mask	Top Solder Mask	Not specified	0.02 mm	Black	3.3	0
Top	copper		0.035 mm		1	0
Dielectric	prepreg	FR4	0.2 mm	Not specified	4.6	0.02
Gnd	copper		0.035 mm		1	0
Dielectric	core	FR4	1 mm	Not specified	4.6	0.02
Power	copper		0.035 mm		1	0
Dielectric	prepreg	FR4	0.2 mm	Not specified	4.6	0.02
Bottom	copper		0.035 mm		1	0
B.Mask	Bottom Solder Mask	Not specified	0.02 mm	Black	3.3	0
B.Paste	Bottom Solder Paste		0 mm		1	0
B.Silkscreen	Bottom Silk Screen	Not specified	0 mm	Not specified	1	0

BOARD CHARACTERISTICS

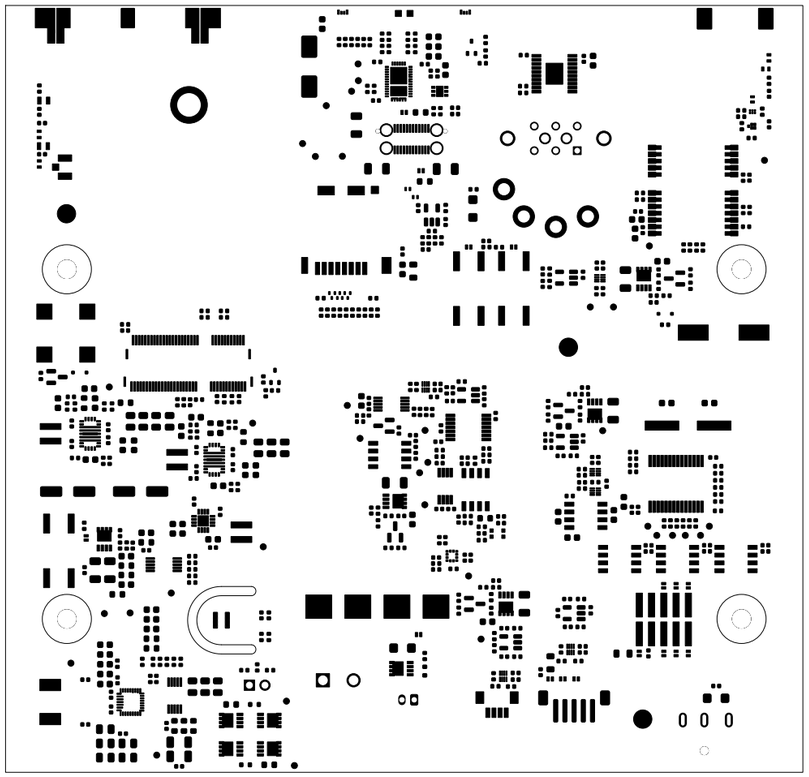
Copper Layer Count:	4	Board Thickness:	1.5800 mm
Board overall dimensions:	130.0000 mm x 125.0000 mm		
Min track/spacing:	0.1000 mm / 0.2000 mm	Min hole diameter:	0.2000 mm
Copper Finish:	HAL lead-free	Impedance Control:	Yes
Castellated pads:	No	Plated Board Edge:	No
Edge card connectors:	No		

IMPEDANCE CONTROL TABLE

LAYER	TRACE (MM)	SPACING (MM)	IMPEDANCE SINGLE-ENDED	IMPEDANCE DIFFERENTIAL	TOLLERANCE
TOP	0.35	–	50ohm	–	+/-10%
TOP/BOTTOM	0.27	0.2	–	90ohm	+/-10%

FAE TECHNOLOGY
via C.Battisti 136, 24025, Gazzaniga (Bg), Italy
Mail: info@fae.technology Tel: +39 035738130

Project **Flatburn_V4**		Board name **Flatburn_V4_layout**
Designer **Luca Brighenti**		
Internal code **-**		Customer **MIT**
Date 07/03/25	Rev. **1**	Rev. changes See Root Page
		Kicad Version **KiCad E.D.A. 9.0.0**
		Sheet **8/12**



Layer Name	Type	Material	Thickness (mm)	Color	Epsilon R	Loss Tangent
F.Silkscreen	Top Silk Screen	Not specified	0 mm	White	1	0
F.Paste	Top Solder Paste		0 mm		1	0
F.Mask	Top Solder Mask	Not specified	0.02 mm	Black	3.3	0
Top	copper		0.035 mm		1	0
Dielectric	prepreg	FR4	0.2 mm	Not specified	4.6	0.02
Gnd	copper		0.035 mm		1	0
Dielectric	core	FR4	1 mm	Not specified	4.6	0.02
Power	copper		0.035 mm		1	0
Dielectric	prepreg	FR4	0.2 mm	Not specified	4.6	0.02
Bottom	copper		0.035 mm		1	0
B.Mask	Bottom Solder Mask	Not specified	0.02 mm	Black	3.3	0
B.Paste	Bottom Solder Paste		0 mm		1	0
B.Silkscreen	Bottom Silk Screen	Not specified	0 mm	Not specified	1	0

BOARD CHARACTERISTICS

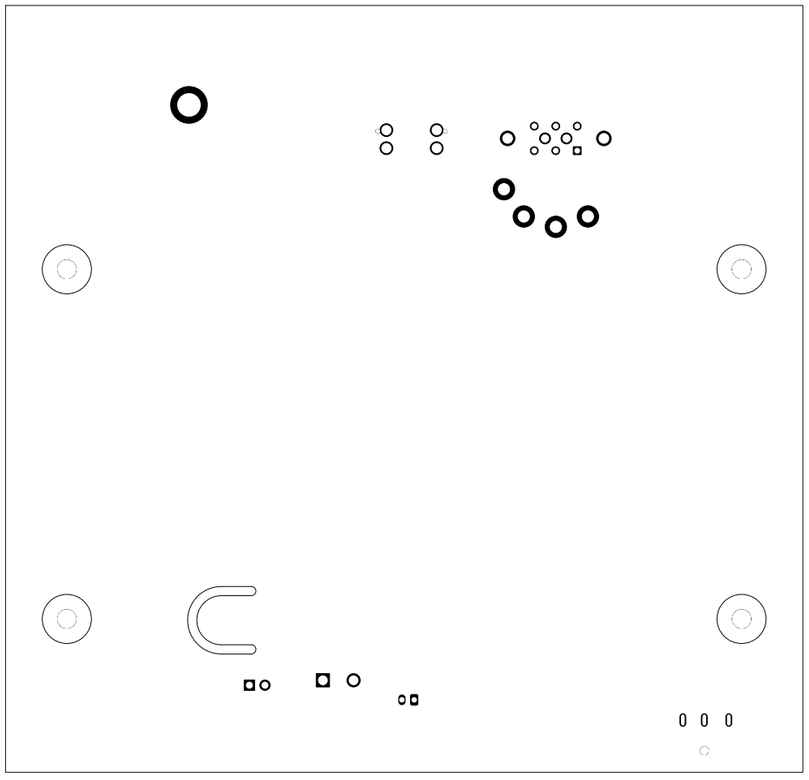
Copper Layer Count:	4	Board Thickness:	1.5800 mm
Board overall dimensions:	130.0000 mm x 125.0000 mm		
Min track/spacing:	0.1000 mm / 0.2000 mm	Min hole diameter:	0.2000 mm
Copper Finish:	HAL lead-free	Impedance Control:	Yes
Castellated pads:	No	Plated Board Edge:	No
Edge card connectors:	No		

IMPEDANCE CONTROL TABLE

LAYER	TRACE (MM)	SPACING (MM)	IMPEDANCE SINGLE-ENDED	IMPEDANCE DIFFERENTIAL	TOLLERANCE
TOP	0.35	–	50ohm	–	+/-10%
TOP/BOTTOM	0.27	0.2	–	90ohm	+/-10%

FAE TECHNOLOGY
via C.Battisti 136, 24025, Gazzaniga (Bg), Italy
Mail: info@fae.technology Tel: +39 035738130

Project	**Flatburn_V4**	Board name	**Flatburn_V4_layout**
Designer	**Luca Brighenti**		
Internal code	**-**	Customer	**MIT**
Date 07/03/25	Rev. **1**	Rev. changes See Root Page	Kicad Version **KiCad E.D.A. 9.0.0** — Sheet **9/12**



Layer Name	Type	Material	Thickness (mm)	Color	Epsilon R	Loss Tangent
F.Silkscreen	Top Silk Screen	Not specified	0 mm	White	1	0
F.Paste	Top Solder Paste		0 mm		1	0
F.Mask	Top Solder Mask	Not specified	0.02 mm	Black	3.3	0
Top	copper		0.035 mm		1	0
Dielectric	prepreg	FR4	0.2 mm	Not specified	4.6	0.02
Gnd	copper		0.035 mm		1	0
Dielectric	core	FR4	1 mm	Not specified	4.6	0.02
Power	copper		0.035 mm		1	0
Dielectric	prepreg	FR4	0.2 mm	Not specified	4.6	0.02
Bottom	copper		0.035 mm		1	0
B.Mask	Bottom Solder Mask	Not specified	0.02 mm	Black	3.3	0
B.Paste	Bottom Solder Paste		0 mm		1	0
B.Silkscreen	Bottom Silk Screen	Not specified	0 mm	Not specified	1	0

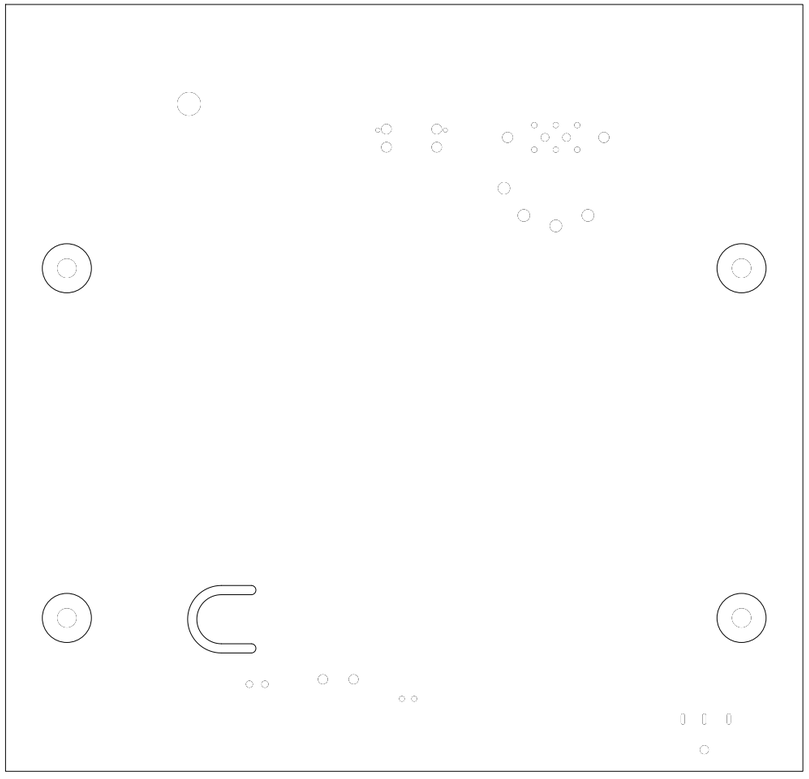
BOARD CHARACTERISTICS

Copper Layer Count:	4	Board Thickness:	1.5800 mm
Board overall dimensions:	130.0000 mm x 125.0000 mm		
Min track/spacing:	0.1000 mm / 0.2000 mm	Min hole diameter:	0.2000 mm
Copper Finish:	HAL lead-free	Impedance Control:	Yes
Castellated pads:	No	Plated Board Edge:	No
Edge card connectors:	No		

IMPEDANCE CONTROL TABLE					
LAYER	TRACE (MM)	SPACING (MM)	IMPEDANCE SINGLE-ENDED	IMPEDANCE DIFFERENTIAL	TOLLERANCE
TOP	0.35	-	50ohm	-	+/-10%
TOP/BOTTOM	0.27	0.2	-	90ohm	+/-10%

FAE TECHNOLOGY
via C.Battisti 136, 24025, Gazzaniga (Bg), Italy
Mail: info@fae.technology Tel: +39 035738130

Project **Flatburn_V4**		Board name **Flatburn_V4_layout**
Designer **Luca Brighenti**		
Internal code **-**		Customer **MIT**
Date 07/03/25	Rev. **1**	Rev. changes See Root Page
		Kicad Version **KiCad E.D.A. 9.0.0**
		Sheet **10/12**



Layer Name	Type	Material	Thickness (mm)	Color	Epsilon R	Loss Tangent
F.Silkscreen	Top Silk Screen	Not specified	0 mm	White	1	0
F.Paste	Top Solder Paste		0 mm		1	0
F.Mask	Top Solder Mask	Not specified	0.02 mm	Black	3.3	0
Top	copper		0.035 mm		1	0
Dielectric	prepreg	FR4	0.2 mm	Not specified	4.6	0.02
Gnd	copper		0.035 mm		1	0
Dielectric	core	FR4	1 mm	Not specified	4.6	0.02
Power	copper		0.035 mm		1	0
Dielectric	prepreg	FR4	0.2 mm	Not specified	4.6	0.02
Bottom	copper		0.035 mm		1	0
B.Mask	Bottom Solder Mask	Not specified	0.02 mm	Black	3.3	0
B.Paste	Bottom Solder Paste		0 mm		1	0
B.Silkscreen	Bottom Silk Screen	Not specified	0 mm	Not specified	1	0

BOARD CHARACTERISTICS

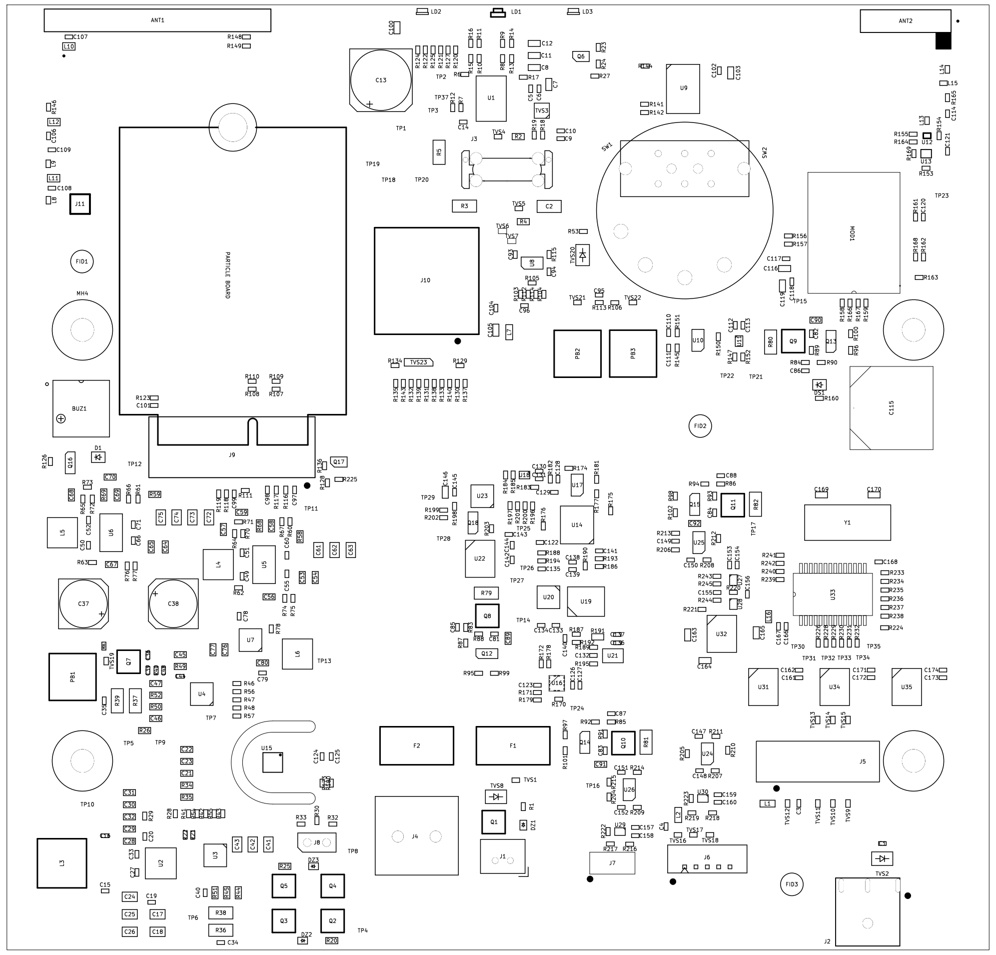
Copper Layer Count:	4	Board Thickness:	1.5800 mm
Board overall dimensions:	130.0000 mm x 125.0000 mm		
Min track/spacing:	0.1000 mm / 0.2000 mm	Min hole diameter:	0.2000 mm
Copper Finish:	HAL lead-free	Impedance Control:	Yes
Castellated pads:	No	Plated Board Edge:	No
Edge card connectors:	No		

IMPEDANCE CONTROL TABLE

LAYER	TRACE (MM)	SPACING (MM)	IMPEDANCE SINGLE-ENDED	IMPEDANCE DIFFERENTIAL	TOLLERANCE
TOP	0.35	–	50ohm	–	+/-10%
TOP/BOTTOM	0.27	0.2	–	90ohm	+/-10%

FAE TECHNOLOGY
via C.Battisti 136, 24025, Gazzaniga (Bg), Italy
Mail: info@fae.technology Tel: +39 035738130

Project **Flatburn_V4**		Board name **Flatburn_V4_layout**
Designer **Luca Brighenti**		
Internal code **-**		Customer **MIT**
Date 07/03/25	Rev. **1**	Rev. changes See Root Page
		Kicad Version **KiCad E.D.A. 9.0.0**
		Sheet **11/12**



Layer Name	Type	Material	Thickness (mm)	Color	Epsilon R	Loss Tangent
F.Silkscreen	Top Silk Screen	Not specified	0 mm	White	1	0
F.Paste	Top Solder Paste		0 mm		1	0
F.Mask	Top Solder Mask	Not specified	0.02 mm	Black	3.3	0
Top	copper		0.035 mm		1	0
Dielectric	prepreg	FR4	0.2 mm	Not specified	4.6	0.02
Gnd	copper		0.035 mm		1	0
Dielectric	core	FR4	1 mm	Not specified	4.6	0.02
Power	copper		0.035 mm		1	0
Dielectric	prepreg	FR4	0.2 mm	Not specified	4.6	0.02
Bottom	copper		0.035 mm		1	0
B.Mask	Bottom Solder Mask	Not specified	0.02 mm	Black	3.3	0
B.Paste	Bottom Solder Paste		0 mm		1	0
B.Silkscreen	Bottom Silk Screen	Not specified	0 mm	Not specified	1	0

BOARD CHARACTERISTICS

Copper Layer Count:	4	Board Thickness:	1.5800 mm
Board overall dimensions:	130.0000 mm x 125.0000 mm		
Min track/spacing:	0.1000 mm / 0.2000 mm	Min hole diameter:	0.2000 mm
Copper Finish:	HAL lead-free	Impedance Control:	Yes
Castellated pads:	No	Plated Board Edge:	No
Edge card connectors:	No		

IMPEDANCE CONTROL TABLE

LAYER	TRACE (MM)	SPACING (MM)	IMPEDANCE SINGLE-ENDED	IMPEDANCE DIFFERENTIAL	TOLLERANCE
TOP	0.35	-	50ohm	-	+/-10%
TOP/BOTTOM	0.27	0.2	-	90ohm	+/-10%

FAE TECHNOLOGY
via C.Battisti 136, 24025, Gazzaniga (Bg), Italy
Mail: info@fae.technology Tel: +39 035738130

Project	Flatburn_V4	Board name	Flatburn_V4_layout
Designer	Luca Brighenti		
Internal code	-	Customer	MIT
Date 07/03/25	Rev. 1	Rev. changes See Root Page	Kicad Version KiCad E.D.A. 9.0.0
		Sheet	12/12