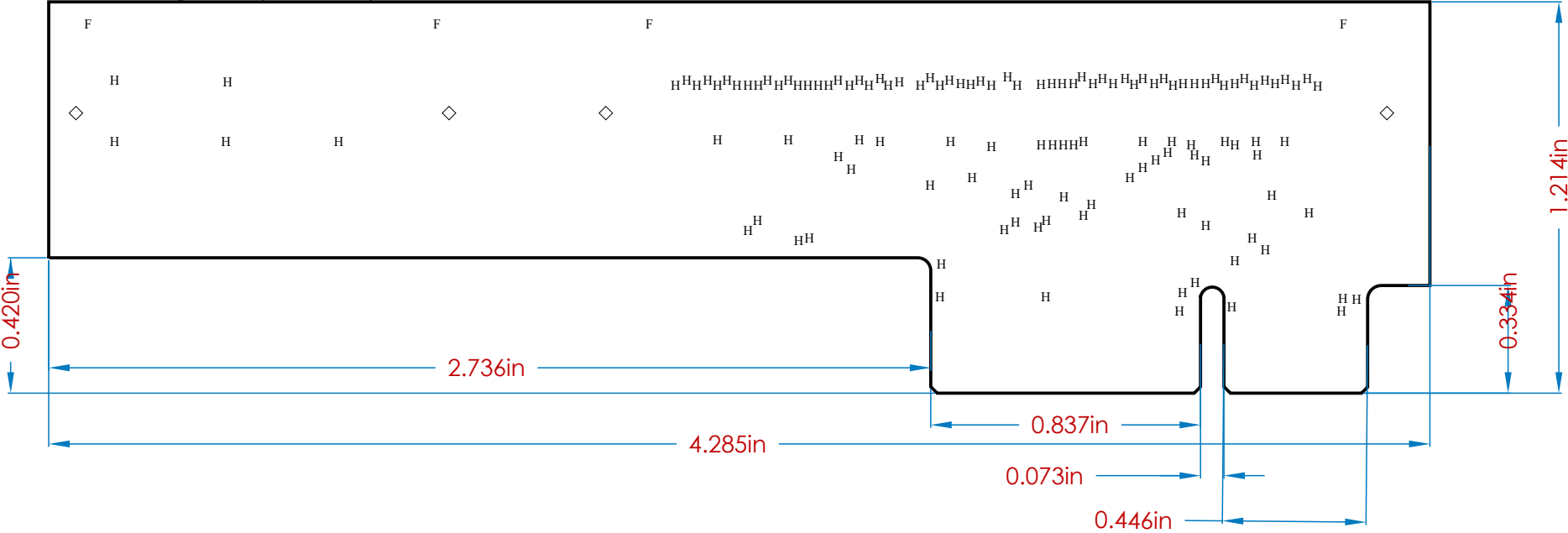


Drill Drawing View (Scale 2:1)



Layer Stack Legend

Layer	Thickness	Gerber
Top Overlay		GTO
Top Solder	0.000in	GTS
TOP	0.001in	GTL
	0.010in	
INNER1	0.001in	G1
	0.012in	
INNER2	0.001in	G2
	0.010in	
Layer 1	0.001in	G3
	0.012in	
Layer 2	0.001in	G4
	0.010in	
Bottom Layer	0.001in	GBL
Board Layer Stack Bottom Solder	0.000in	GBS
Board Layer Stack Bottom Overlay		GBO
Total thickness: 0.063in		

Notes:

- Fabricate per IPC 6012 Class II
- Material: FR4 equivalent.  
Tg>=170°
- Cu weight all layers: 4 oz. outer, 2 oz. inner
- Total board thickness after plating: 0.063 +/- 10%
- Final finish: ENIG
- Soldermask both sides using photo-imagable process. Color: green
- Silkscreen both sides using non-conductive ink. Color : white
- All holes to be +/- 0.003 unless otherwise specified. Hole sizes are given after plating. Plated thru-hole shall have a min of 0.002 Cu internal plating.
- Remove all burrs and sharp edges 0.015 min.

Drill Table

Symbol	Count	Hole Size	Plated	Hole Type	Hole Tolerance	Hole Length
H	125	0.012in	Plated	Round		
F	4	0.032in	Plated	Round		
◇	4	0.055in	Non-Plated	Round		
	133 Total					



[No Variations]

Title: NXP To TI Adapter Board

Drawing No: =DrawingNumber

Date: Apr. 2024

Size: B

Revision: 1.0

Sheet: 1 of 1