**Task 2: Custom Footprint Creation for AMS1117 (SOT-223)**

**Aim:**  
To design a custom PCB footprint for the AMS1117 voltage regulator using datasheet dimensions and implement it accurately in KiCad.

**Submission Includes:**  
Footprint screenshot, and design justification based on mechanical drawing.

**Write Up:**

The footprint was designed as per the AMS1117 (SOT-223) package dimensions. Pad-to-pad horizontal spacing is maintained at **2.29 mm**, with the large tab (Pad 4) having a width of **2.95–3.15 mm** and height of **6.71–7.29 mm**. The smaller pins have widths of **0.84–1.04 mm** and heights of **3.30–3.71 mm**, exactly matching the mechanical drawing.

