

INTRODUCTION

PCBs, which stand for Printed Circuit Boards, have emerged as critical components in modern electronic devices, enabling the interconnection and functioning of complex electronic circuits. They have played a pivotal role in shaping the electronic industry, and their continuous development has facilitated the miniaturization, efficiency, and reliability of electronic devices. I will assist my Ph.D. mentor and analyze different PCBs to be constructed on a miniature camera

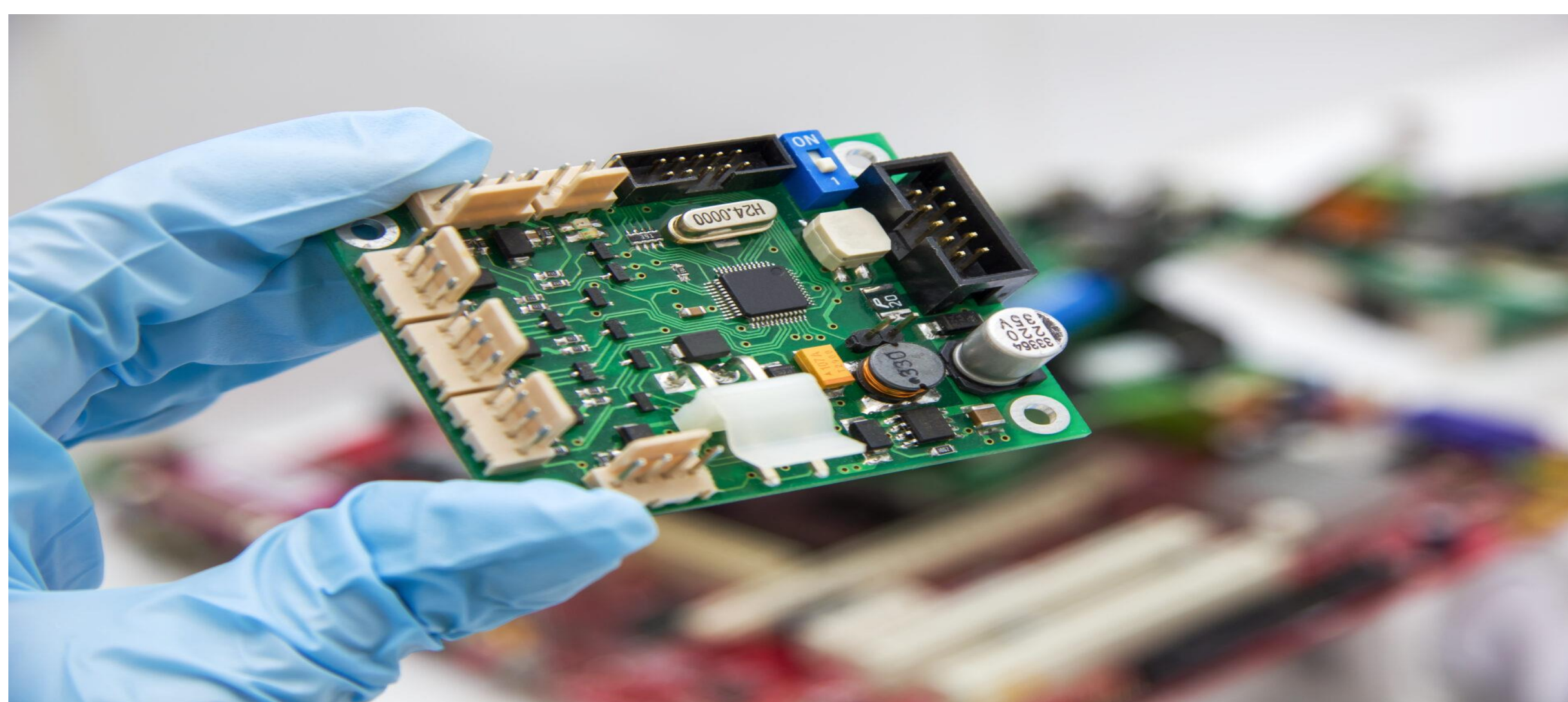


Fig 1. How a Printed Circuit Board looks like

METHODS

We use Altium Designer for our PCB design due to its accessibility, ease of use, and availability of online learning resources for beginners. We designed a simple PCB design containing two components to get familiar with the Altium Designer. One component is an Omnetics connector which is connected to a series of pads on the other side. Using the component datasheet, we implement the exact footprint of Omnetics connector. This requires designing the pads with the correct width and spacing. The board we designed has two layers for simplifying the routing. The vias placement at the appropriate location ensures straightforward metal routing without making it cumbersome. The electric pads provide components to be electrically mounted on soldering. Vias are placed on the edge of the pads to connect tracks to the bottom, and top layers. We manually route the connections as opposed to using the auto-route feature of the tool due to the relatively smaller number of routes, making the implementation easier while illustrating the process of designing the PCB. Lastly ensure the design meets the requirements of the manufacturer's constraints, which can be done using the design rule check (DRC). We run the DRC to identify all the electrical shorts and other design constraints. All the violations related to the track and via clearances were resolved before finishing the design.

RESULTS

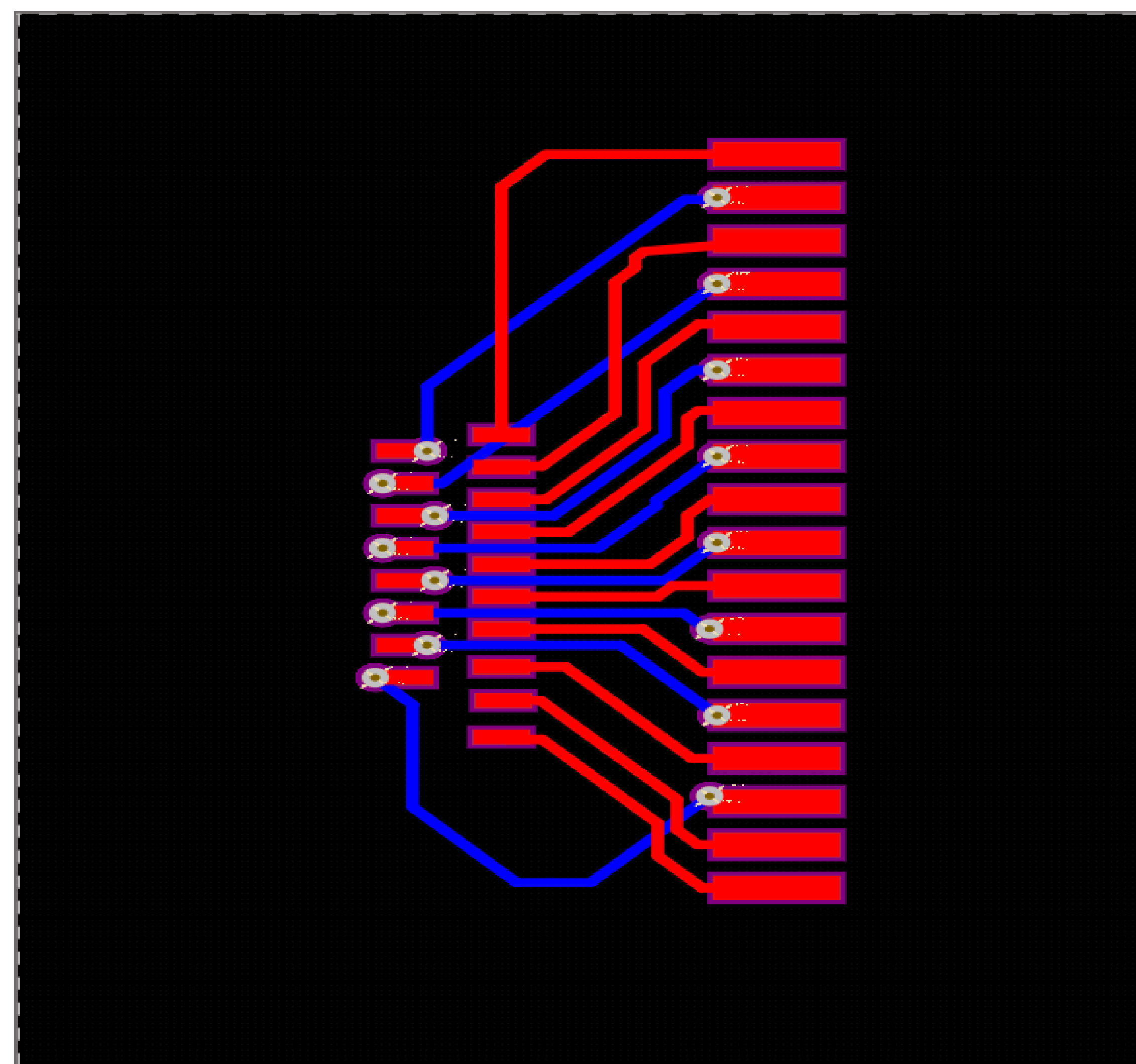


Fig 2. FINAL LAYOUT FOR THE PCB DESIGN ON ALTIUM

CONCLUSION

The research demonstrates the feasibility of integrating a custom-designed PCB within a camera system. The study shows that PCB's can effectively facilitate the complex electronic connections required for camera functionality.

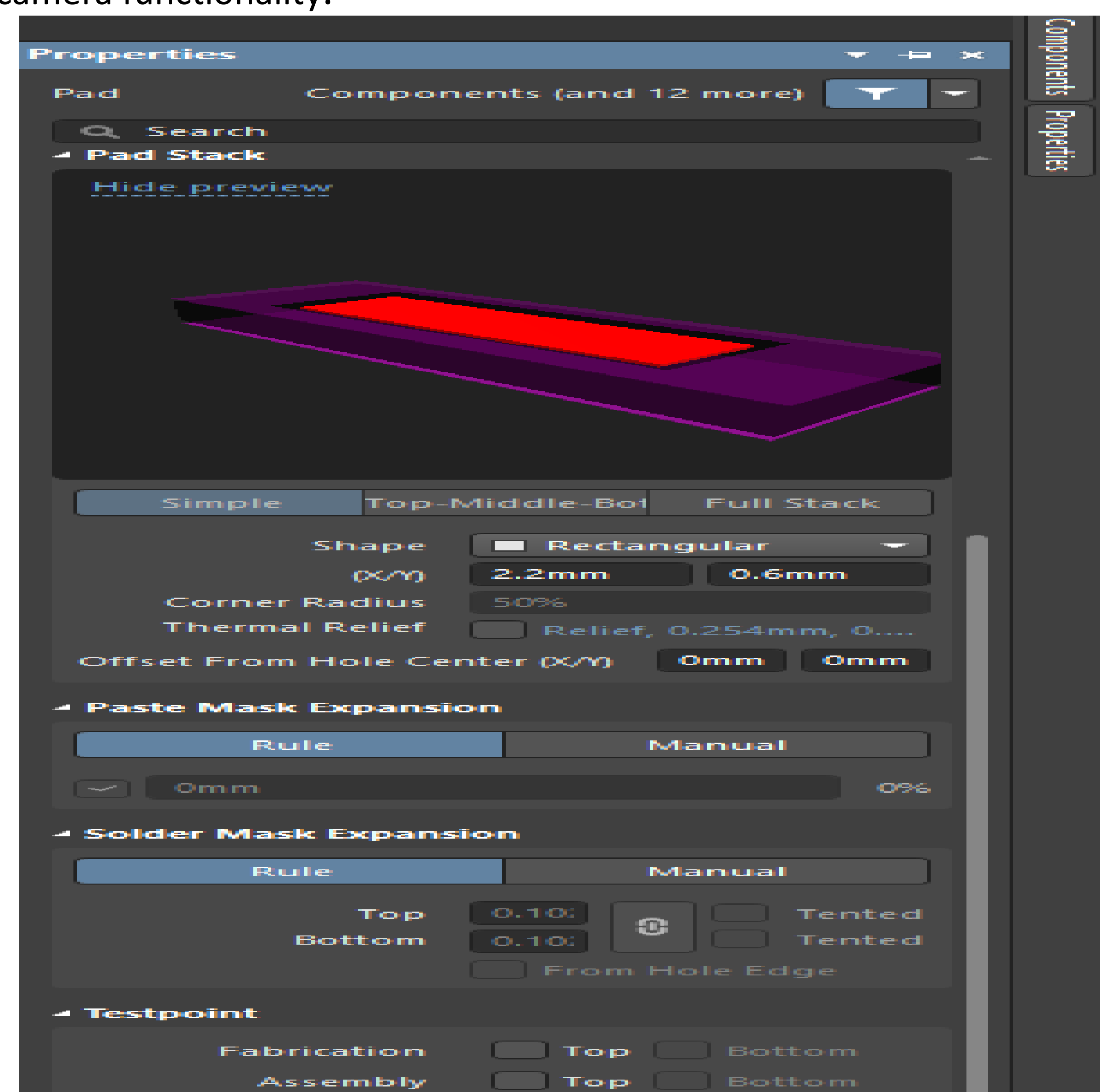


Fig 3. Properties of a Pad in Altium