

Undergraduate Research Paper

5/4/17

Jerry L. Collins II

Researching for Dr. Marcus Hohlmann
High Energy Physics

For the Spring 2017 semester, I had the pleasure of engaging in undergraduate research under the guidance of Dr. Marcus Hohlmann, whom I have had previous research experience with, being a member of his high energy physics research group three years. While the typical duties centered around detector assembly and their respective Quality Control testing for the CMS experiment at the Large Hadron Collider at CERN, I was tasked with heading a different project for the purposes of this specific undergraduate research course. That task was to produce a printed circuit board (PCB) version of a Thick GEM design within the program Altium Designer, with which an outside company could manufacture the boards on our behalf. The following steps towards the completion of this task was done with the assistance of fellow undergraduate and research group member Francisco Jimenez.

As shown in Figure 1 below, a version of this board had already been produced in the program Solidworks, to the required specifications. However it was determined that the solidworks files were not compatible with the requirements of the company, nor could the software natively produce the required files. Therefore the design needed to be reproduced in Altium Designer.

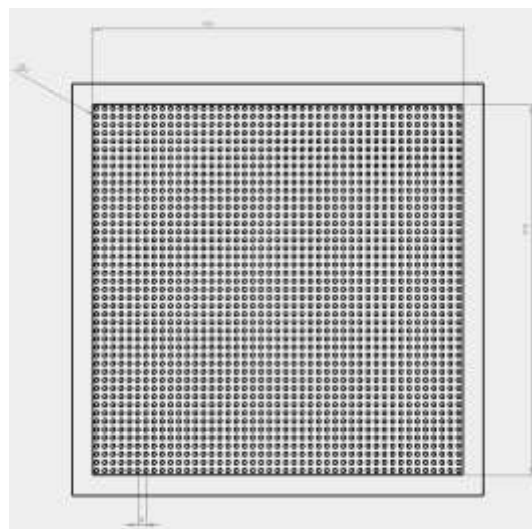


Figure 1: Thick GEM Solidworks design

The biggest issue in the creation of this design in Altium Designer was, as one would expect, the production of all of the required holes. The holes being very small and plentiful, it was at first seemingly a somewhat daunting task to complete their creation. However, after contacting and discussing with Dr. Aiwu Zhang of Brookhaven National Labs, a plan was brought forth on how to tackle the situation.

Before this however, the preliminary groundwork for the board was set up. Instead of beginning with the entire 10x10 centimeter board dimensions, it was decided that the 9x9 cm active region would be constructed first, minimizing the number of variables, as having the extra space there to avoid could be distracting while creating the rest of the design. In addition to this, having a strict border also makes it easier to stay within the required active region confines. Thus the board dimensions were restricted to 9x9 cm and it became time to populate the active region with the holes.

This was the primary step for which we utilized the advice given by Dr. Zhang. The creation of the holes (which consists of creating a pad of a specified diameter, 1mm, and shape then declaring a hole within said pad of an equivalent diameter and, in our case, not plated with metal) was known and relatively simple. Even populating the entire field with the holes was possible, though not simple as it would require either knowledge of code-based manipulation of components within the software, or one-by-one placement of each hole. Dr. Zhang suggested that instead of making all holes individually, to make one row of holes, then duplicate those, then duplicate the two, then the four, and so on until the field was populated. Though there was a limit to the number of rows which were able to be duplicated simultaneously, this did prove to be an overall effective method for filling the region with the required number of holes.

There were some setbacks however. Upon first attempt to utilize this method of hole population, there seemed to be an alignment issue. This issue seemed to cause each row to be shifted ever so slightly to the right, causing an overall skew in the hole alignment, which would lead to a trapezoidal arrangement of the holes, instead of a 9x9 square. Fortunately, after repeated trials, the issue was resolved and the holes were now arranged in the proper square pattern within the necessary 9x9 region. It should be noted that while the pattern was what we desired, there remained an insignificant alternating pattern of the rows being shifted 0.01mm back and forth. Since every other row returned to the proper x-axis alignment, this was not an overall skew. That combined with the shifting being only 0.01mm, which is negligible, this factor was considered an insignificant issue.

Now that the largest hurdle of the project was complete, it was time to complete the final requirements. A fill was added to the entire region (besides the holes) which called for the placement of metal over that whole area (Altium defaults to copper but the manufacturer can be told externally to use silver in this case). This was done on both the top and bottom layers of the board to ensure there would be silver on both sides. After that, it came time to form the board into the 10x10 cm dimensions required. In the keep-out layer, the border lines were moved on all sides to form a square of 10x10 cm. The board was then redefined to be of that new size. Now with the 10x10 size created, it came time for the final step.

This final step was to place holes in the corners of the board so that it can be affixed within our setup. This required measurement of the proper diameter (2.8mm) needed for each corner hole, as well as its proper placement location such that it will align properly within the poles on our apparatus.

All of the required features of the Thick GEM were now complete within the Altium design, and both the gerber and NCDrill files were produced for the manufacturer and for Dr. Hohlmann. Figure 2 below shows an image (at an angle) of what Altium projects the finished board to look like (using copper to surround the holes instead of silver).

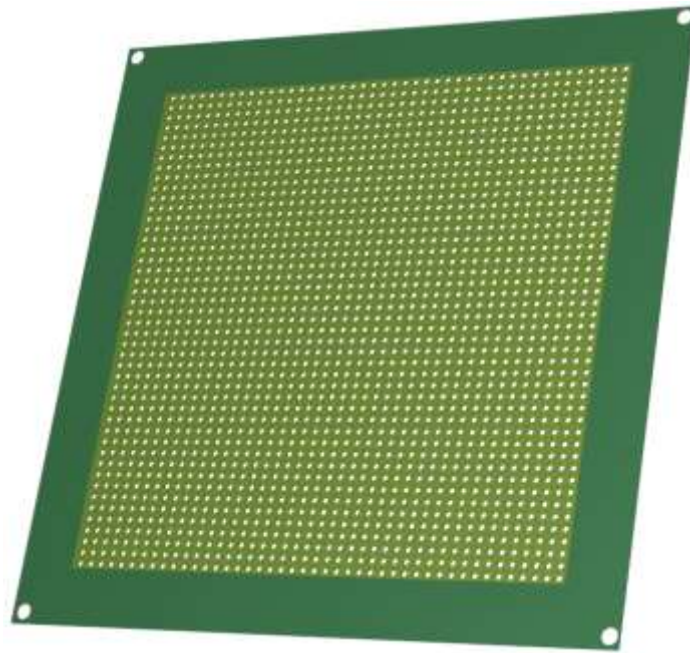


Figure 2: 3D view of the complete board design generated by Altium Designer
Will all requirements met, the design stands ready for production, barring any additional changes. If that were to be determined necessary, those changes could be made promptly and the files resubmitted. As things stand currently however, the project is complete.