Effective heat exchange is key for many energy applications including heat exchangers, heat extraction from heat source, and heat rejection to ambient thermal sink. This study focuses on the investigation for a specific heat exchange configuration, namely heat removal within a concentric annular passage using helical turbulators and jet impingement. Numerical testing was used to see how the different geometric parameters affect the heat transfer and pressure drop within the annulus by using helicoil turbulators. A vast range of designs were studied by changing the turbulator shape, pitch, and blockage ratio while maintaining a constant Reynolds number of 25,000. CFD was performed in STARCCM+ using the realizable $\kappa$-$\varepsilon$ turbulence model. Results show that turbulence and heat transfer increase with a higher blockage ratio and smaller pitch but the pressure drop is subsequently increased as well. The square turbulator promoted higher heat transfer compared to the circle turbulator but the pressure drop was significantly increased when the helix angle was greater than 20° and blockage ratio greater than 0.48. Experimental and numerical efforts were used to find the heat transfer due to impingement jets on the target surface. Multiple flows as a function of jet Reynolds number ranging from 16,000-55,000 were tested for two geometries. Temperature Sensitive Paint (TSP) was utilized to observe local heat transfer. It was observed that jet degradation occurs after the 6th row of stream-wise impingement jets for both cases experimentally and it was difficult to numerically capture the effect of the cross flow from previous jets but managed to follow the same trend. The numerical results showed that they can be used with good agreement to predict the surface averaged Nusselt number to be within the 12% uncertainty found from experimental efforts. Geometry 7 was determined to perform better in terms of heat transfer as opposed to Geometry 6 with the same pressure loss.