Panelize with CLI:

Open kicad command prompt

CMD:kikit panelize –preset {.json file path} {INPUT file} {OUTPUT file}

Example:

* cd C:\MyProjects\temp\Orchid\_E\_S\_LED\Panel
* kikit panelize --preset kikit.json ..\Orchid\_E\_S\_LED.kicad\_pcb Panel.kicad\_pcb

kikit.json -> the preset file.

..\Orchid\_E\_S\_LED.kicad\_pcb -> Input file found one directory above.

Panel.kicad\_pcb -> output file.