

Reference designators

Reference designators are considered one of the most crucial concepts in schematic design as it tells us the type of the components and their numbers for example say you got 100 resistor in your schematic with different values how can you differentiate between them? By numbering them like R1 R2 R3 R...., by this way you can ensure that every component in your schematic has a unique name. Although the reference designators are very useful in schematic design, It's not that important in PCB layout. For sure when every component is placed near to its designator it makes the PCBA more easy but what if there was no place enough for the texts and you should consider that traces, routing and components placement are more important than texts so we have to use another approach named "Assembly notes". Assembly notes is a layer contains the designator of each component under its footprint and this helps a lot when we are in PCBA process.