

Here is an idea but it's not the autocad way of doing it.

Draw a footprint but save it as a schematic child. You can insert this in your panel layout and select the parent from either the schematic or the first footprint file you inserted.

For example:

Insert power supply as a regular footprint file that contains Tag, MFG, DESC and Cat.

Insert cables as a child footprint file and link it to the power supply.

Insert chassis as a child footprint file and also link this to the power supply.

This way the cables and chassis will be "smart" footprints. If you change a TAG or delete the parent component the child footprint files will be updated.