

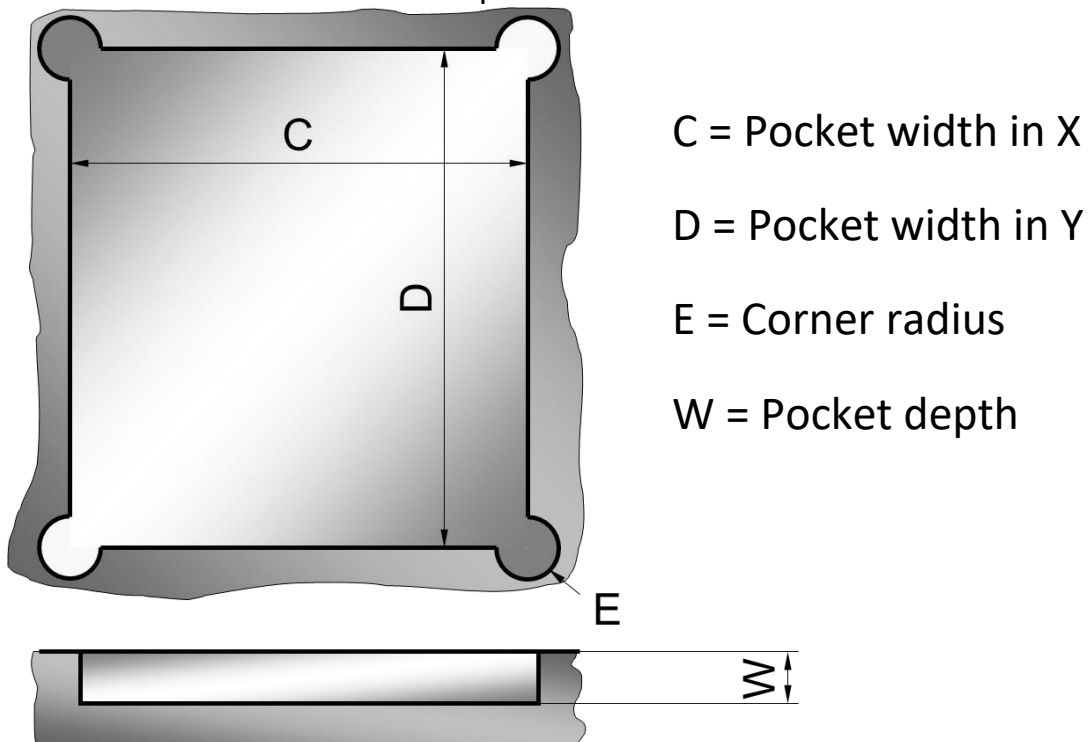
Ersa Dip&Print Station

Form dimensional properties for stencil order

Dimensional properties for Print-Stencils with component pocket:

For correct dimensioning of a Print-Stencil with pocket (depression for fixing a component against shifting in XY direction) following points have to be checked:

- How large are the outside measurements in length and width?**
 The dimensions of the component have to fit to the component pocket (length x width), without a too high amount of looseness. Please note the measure tolerances of the component manufacturer.



- How high soldering paste has to be printed?**
 The height of the soldering paste print defines the deposition quantity largely. Commonly the print height (and the aperture sizes) should be similar to the print stencil used during production. Goal is to achieve the same volume which means the same solder amount.



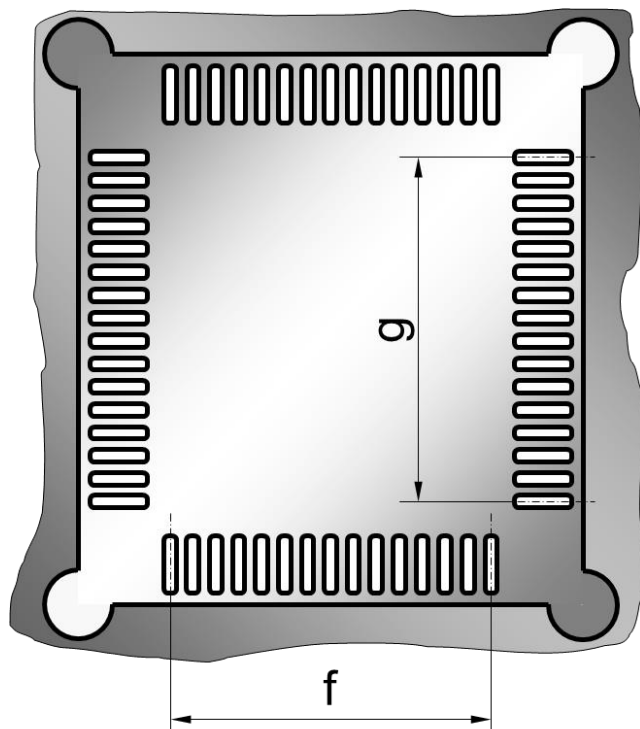
T = Stencil thickness (= Height of solder paste print)

Ersa Dip&Print Station

Form dimensional properties for stencil order

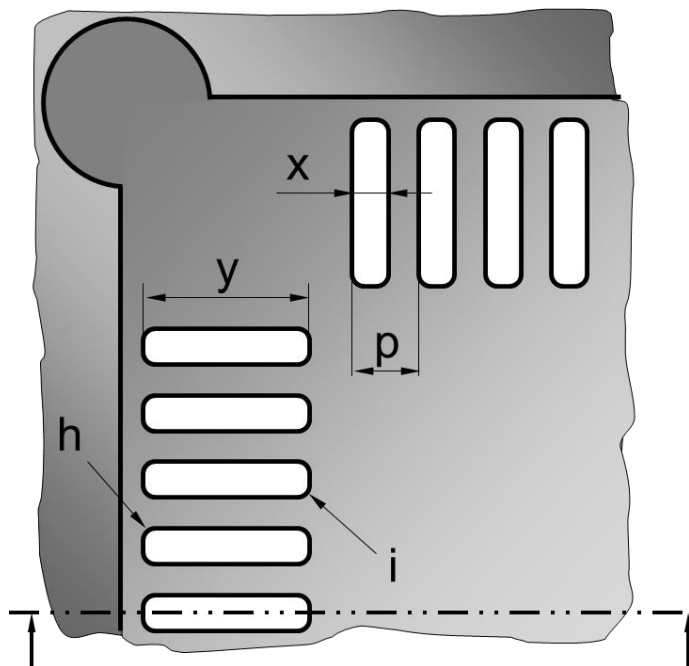
- **How large has to be the aperture?**

The size of the aperture (together with the print height) defines the deposition volume of the solder paste. The aperture has to seal the print area (pad) as much as possible. Otherwise the soldering paste might smear by squeezing out at the leakage indefinitely. This could result in solder bridges and solder balls.



f = Center-to-center distance outer apertures in X

g = Center-to-center distance outer apertures in Y



x = Aperture width

y = Aperture length

p = Aperture pitch

h = Outer corner radius

i = Inner corner radius

Ersa Dip&Print Station

Form dimensional properties for stencil order

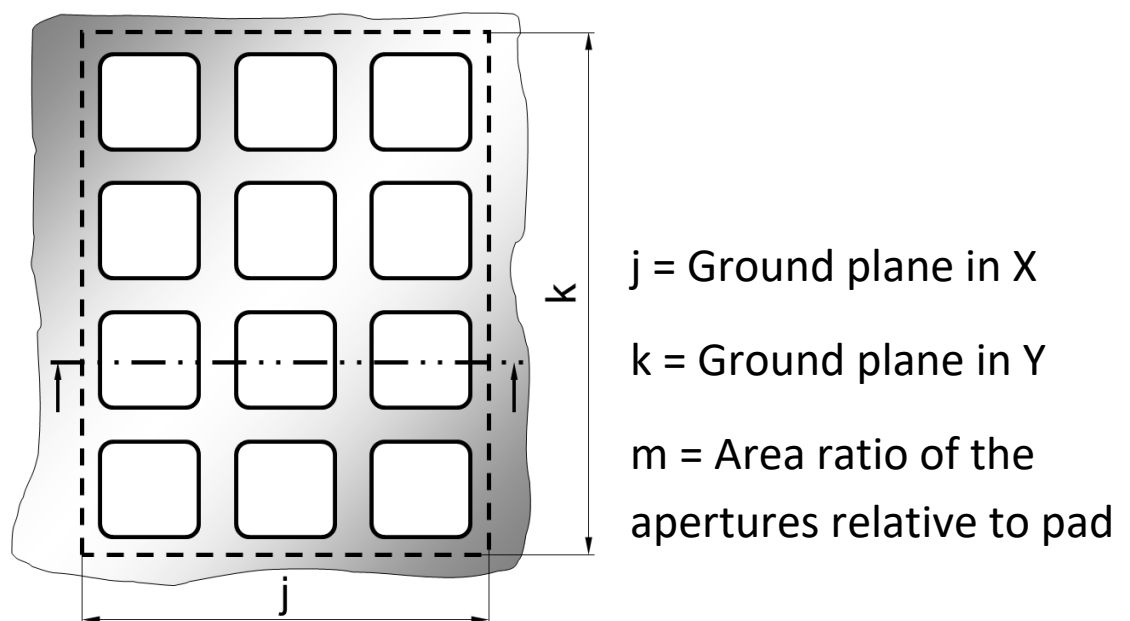
- **Printing to QFN, MLF, LPCC and BTC:**

Components with bottom termination can be printed perfectly because there are termination areas at the component body only (pads). Often there is an additional ground plane at the center.

➔ While the surrounding pads are no problem usually, the ground plane in the center requires special attention:

- In case too much solder paste is applied, the component might float at the center solder. This may result in open joints at the surrounding pads.
- In case insufficient solder paste at the ground plane, the minimum junction might be decreased. This may result in thermal problems at the component (at later operating).
- Often the solder paste is applied to the ground plane in a segmented print to avoid the entrapment of flux. This may result in voids that may create problems in the thermal junction again.
- Two possibilities exist for definition of the measures for the aperture of the ground plane:

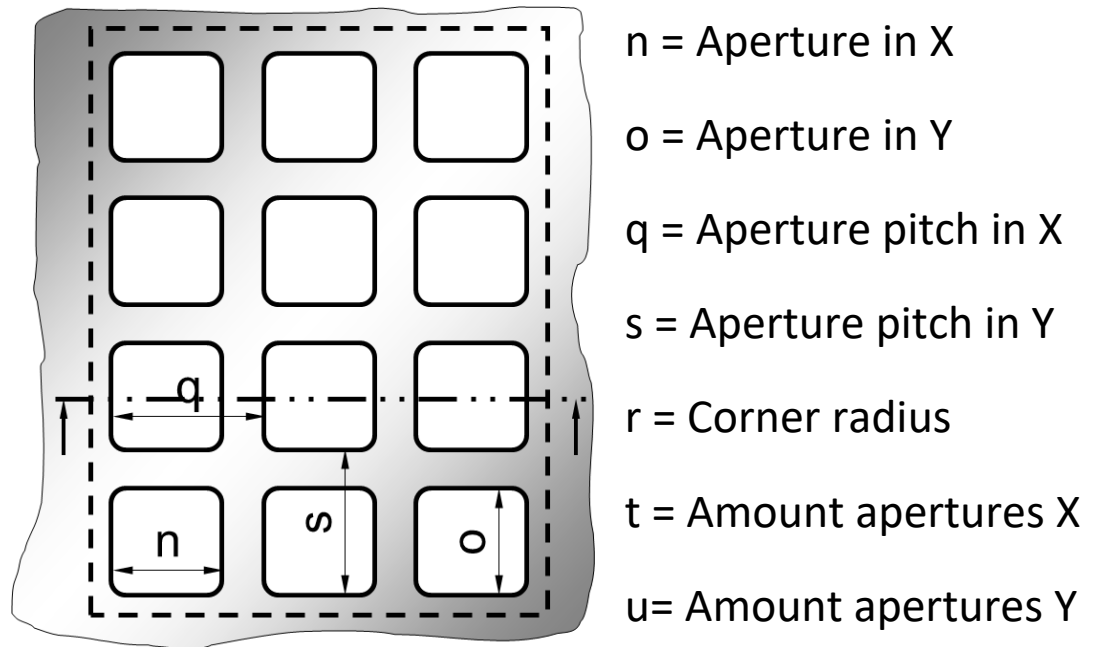
1. Area ratio of the apertures relative to the area of the ground plane:



Ersa Dip&Print Station

Form dimensional properties for stencil order

2. Detailed definition of measures for the apertures



Other geometry requires exact separate specification.

- **Printing to QFP and similar:**

Printing to the feet of standard QFP is complicated because of the manufacturing tolerances. It should be considered only if no other solution is available. Standard QFP often may be dipped to a special dip-solder-paste, as long as there is enough clearance between feet and component bottom (see Dip-Stencils). Paste printing to component is just for QFP with ground plane the mostly only left method because here the clearance between feet and component bottom is extremely small or even zero.

➔ The aperture has to be smaller than the footprint of the feet. Otherwise the solder paste squeezes out of the gap.

➔ For the ground plane the same rules are applicable like for printing to QFN / BTC.

Ersa Dip&Print Station

Form dimensional properties for stencil order

Please fill the desired measures to the table:

C		mm	Pocket width in X	
D		mm	Pocket width in Y	
E		mm	Corner radius	If not defined: Standard radius 1 – 3 mm (depending on pocket dimensions)
W		µm	Pocket depth	Resulting from stencil thickness – plate thickness. (Standard plate thickness: 500 µm)
T		µm	Stencil thickness	If not defined: Standard stencil thickness 120 µm
f		mm	Center-to-center distance outer apertures in X	
g		mm	Center-to-center distance outer apertures in Y	
x		mm	Aperture width	
y		mm	Aperture length	
p		mm	Aperture pitch	
h		mm	Outer corner radius	If not defined: Standard radius 0.07 mm
i		mm	Inner corner radius	If not defined: Standard radius 0.07 mm
j		mm	Ground plane in X	
k		mm	Ground plane in Y	
m		%	Area ratio of the apertures relative to the pad	
n		mm	Aperture in X	
o		mm	Aperture in Y	
q		mm	Aperture pitch in X	
s		mm	Aperture pitch in Y	
r		mm	Corner radius	If not defined: Standard radius 0.07 mm
t		Stk.	Amount of apertures in X	
u		Stk.	Amount of apertures in Y	

Tolerance of all measures, except T: - 0.00 mm + 0.25 mm

Tolerance T: - 10 µm + 10 µm

Ersa GmbH