

To Work with GC_BOOLOP — GstarCAD 2027

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Here, we use two Boolean operations as shown below to complete the design of mechanical parts.

1.Start command, the dialog box is displayed.

2.Select the option "Union" and uncheck the option "Delete the second selected entity", as shown below.

3.First select the exterior circle and then the small circles around it, it will keep all small circles around enabled for the next operation and you will get the first polyline as shown below.

4.Start again the Boolean operations, select option "Subtraction" and check "Delete the second selected entity", as shown.

5. Then select the interior circle and then the small circles around it, then the small circles around it will be automatically deleted and you will get the final result as shown below.

Tip: It will be easier to complete this operation if you hide some of the other objects.

Other versions with matching text

[2023](#) · [2024](#) · [2025](#) · [2026](#)

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