

Converting Connected CAD Segments into a Polyline

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QUESTION

I have created multiple lines and arcs that are all connected to one another, but a single CAD object is not being generated. Instead, the individual lines and arcs can only be selected separately. What am I doing wrong?

ANSWER

Individual CAD lines and arcs can be snapped together end-to-end to form a polyline, but only when Endpoint Object Snaps are turned on and Connect CAD Segments is enabled.

To turn on Endpoint Object Snaps and enable Connect CAD Segments



1. Select **Edit> Snap Settings**  from the menu and make sure that **Object Snaps**  - and particularly, **Endpoint**  snaps - are turned on.
2. Navigate to **Edit> Edit Behavior**  and make sure that **Connect CAD Segments**  is enabled.

Connect CAD Segments can also be toggled on and off on the CAD panel of the Preferences dialog.

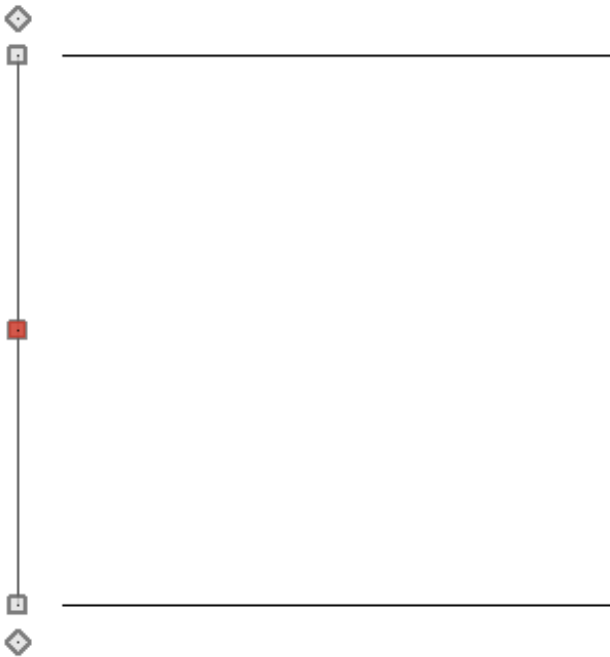
3. Once Endpoint Object Snaps and Connect CAD Segments are enabled, verify that all of the CAD objects that you'd like to be connected are properly snapped to one another to form a polyline.

After a polyline has been created, a single edge can be disconnected using the Disconnect Selected Edge edit tool.

To disconnect a segment from a polyline

1. Using the **Select Objects**  tool, click on the desired segment of the polyline.
2. Click on the **Disconnect Selected Edge**  edit tool.

The selected segment can now be deleted, moved or altered separate from the rest of the polyline.




Alternatively, select CAD> Lines> Disconnect Edges to toggle the Disconnect Edges option. With this option turned on, each segment selected will disconnect from the polyline.

Related Articles

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- [📄 Using the Fillet and Chamfer Edit Tools \(/support/article/KB-00389/using-the-fillet-and-chamfer-edit-tools.html\)](https://support/article/KB-00389/using-the-fillet-and-chamfer-edit-tools.html)



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