

# SPLINE command

2364 GstarCAD MY /KW August 23, 2021 [CAD Commands](#) 0 1811

The **SPLINE** command is used to create smooth curves through or near a group of fit points.

## Command Access:

**Ribbon** : Home > Draw > Spline

**Menu** : Draw > Spline

**Command** : SPLINE

## Command Prompts:

Specify first point or [Object]:

Specify next point:

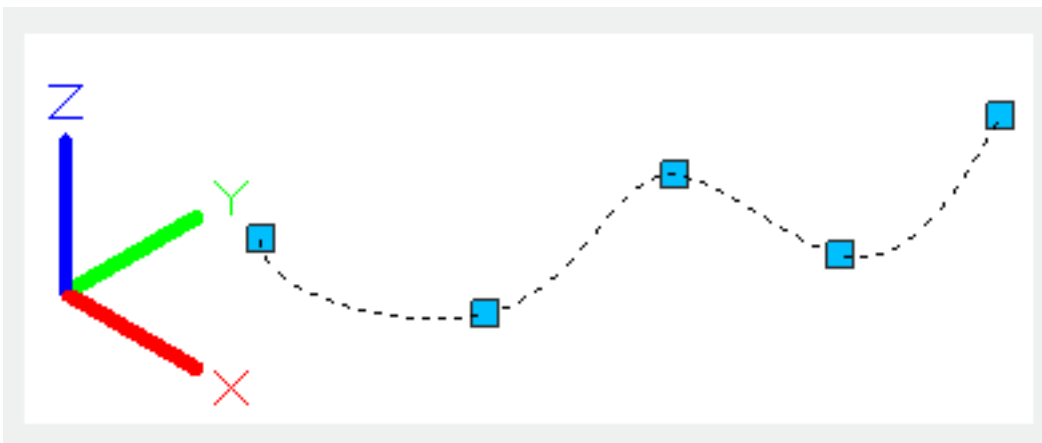
Specify next point or [Close/Fit tolerance] :

Specify start tangent:

Specify end tangent:

## Function Description:

The **SPLINE** is called non-uniform rational B-spline (NURBS), for simplicity, called splines. It is defined by fit points. By default, the fit points are coinciding with spline.



## Relative Glossary:

### First point:

Specify the first point of a spline.

### Object:

Convert 2D or 3D polylines that fitted by quadric splines or cubic splines to equivalent splines. Users could save or quit original polylines by setting the DELOBJ system variable.

### Next point:

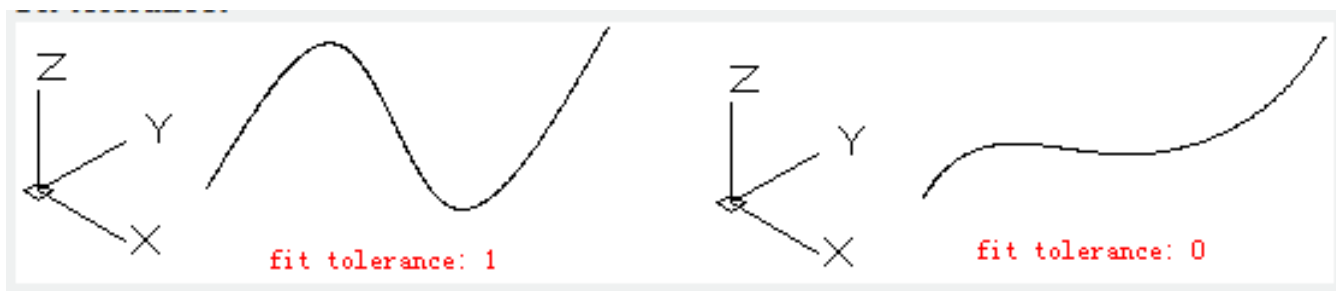
Prompt to create other splines, press ENTER to end the command prompts.

### Close:

Close the spline by defining the last point coincides with the first point. By default, the closed spline is periodicity and keeps curvature continuously along the entire ring.



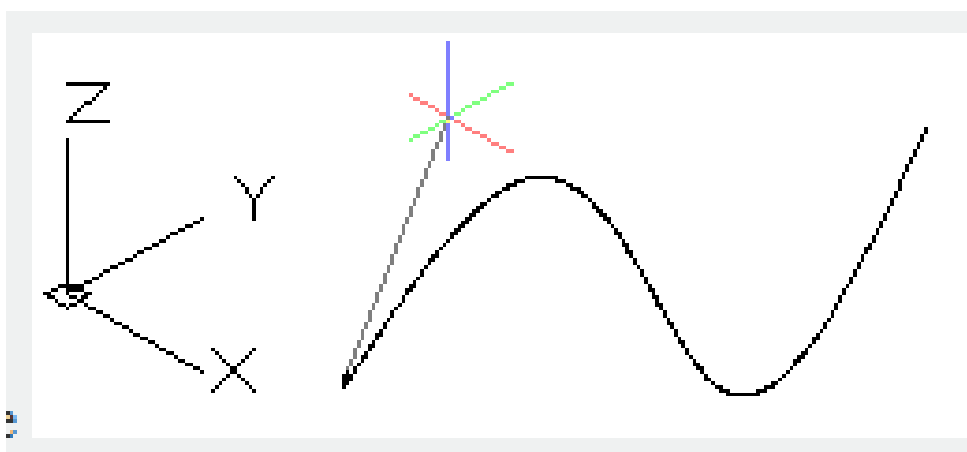
**Fit tolerance:**



Specify the offset of fit points of a spline. If the fit tolerance is 0, the created spline goes through all fit points. The fit tolerance is applied on all fit points (except the start point and the end point).

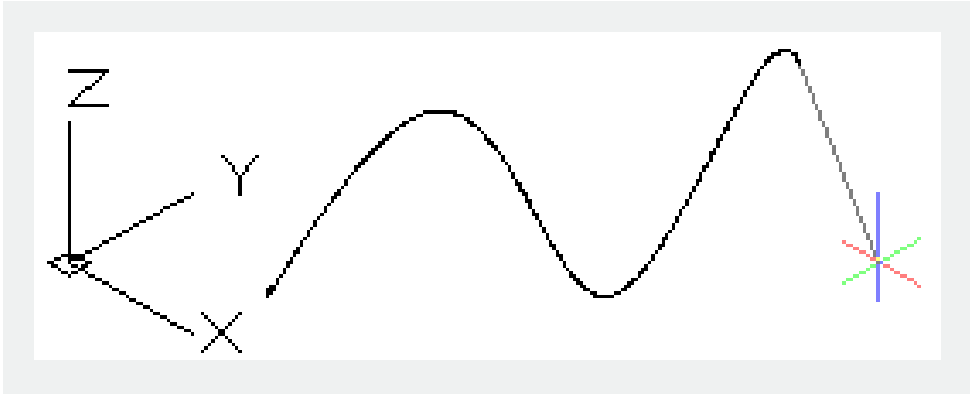
**Start tangent:**

Specify the tangent condition at the start point of a spline



**End tangent:**

Specify the tangent condition at the end point of a spline.



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