



## How to Create a Gear Library

During the design process, we can define a 3D model as the customized part library. By reusing the library, designers can reduce the modelling time, increase the standard parts reusability, and hence improve work efficiency.

Design ideas: Firstly, designers should define a group of design parameters that control the shape and the topological relation, as well as the driving parameters of the part family. Then, define the part library by configuration table which can be reused in any file.

### 1. Create the Sample Part

Let's make a standard involutes gear for the example.

#### 1.1 The base parameters of the gear:

Name	Expression	Value	Unit	Description
Spur Gear				
$\pi$ B	10	10	mm	Tooth Width
$\pi$ M	1	1		Module
$\pi$ z	16	16		Number of Teeth
$\pi$ a	20	20	deg	Pressure Angle
$\pi$ d	$M * z$	16	mm	Reference Dia.
$\pi$ da	$M * (z + 2)$	18	mm	Tip Dia.
$\pi$ dr	$M * (z - 2.5)$	13.5	mm	Root Dia.
$\pi$ db	$d * \cos(a)$	15.03598	mm	Base Dia.

Figure 1. the Parameters of the Gear

#### 1.2 Create the Reference Circle, Base Circle, Tip Circle, Root Circle by using the Wireframe function.

The diameter of each Circle is driven by variables.

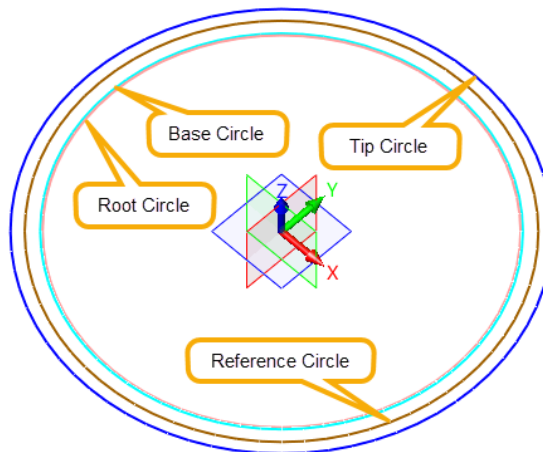


Figure 2. Create 4 Important Circles



1.3 Create the involute by Equation Curve command.

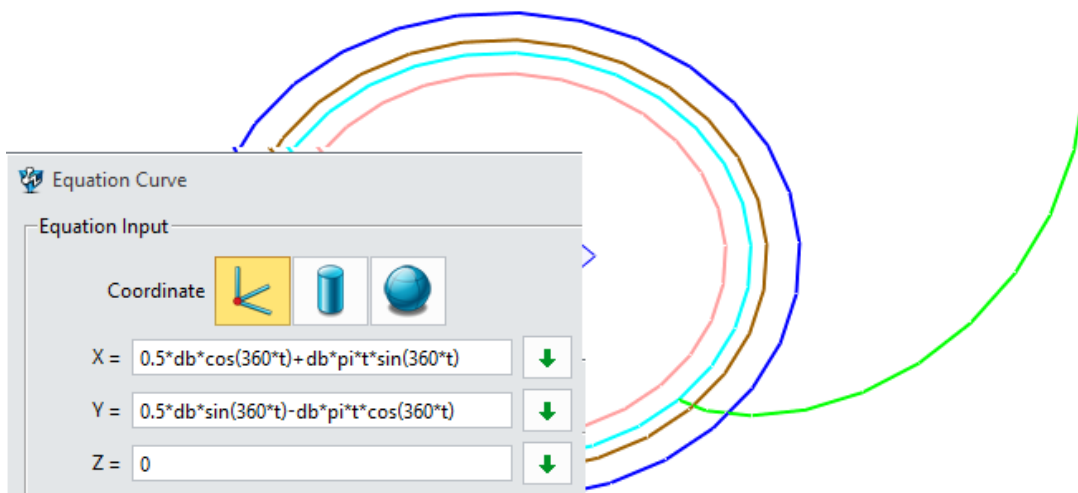


Figure 3. Create the Involute Line

1.4 Create a line and then concatenate it with the involute.

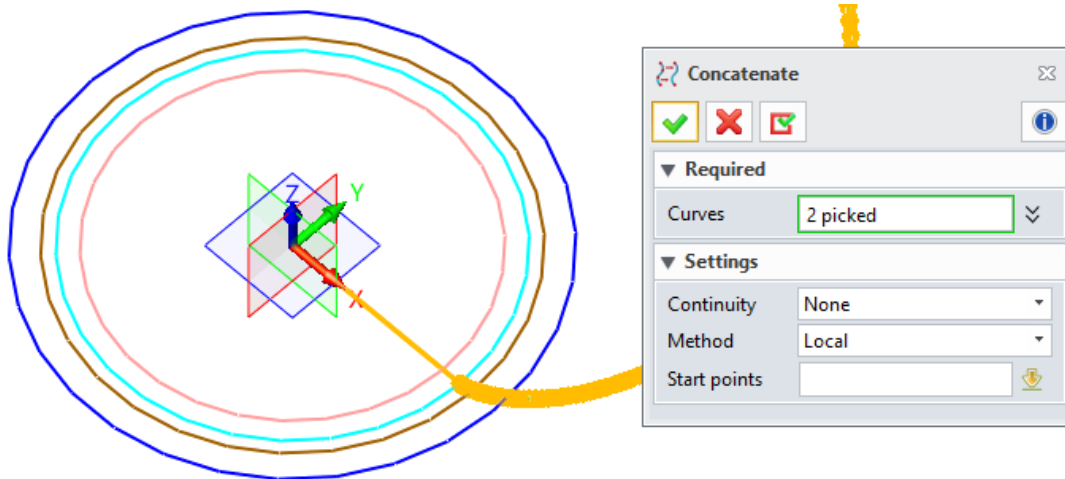


Figure 4. Concatenate a Line with the Involute

1.5 Rotate this connected curve - by 90/z degree (z is the number of teeth). Then mirror the curve along the XZ Plane.

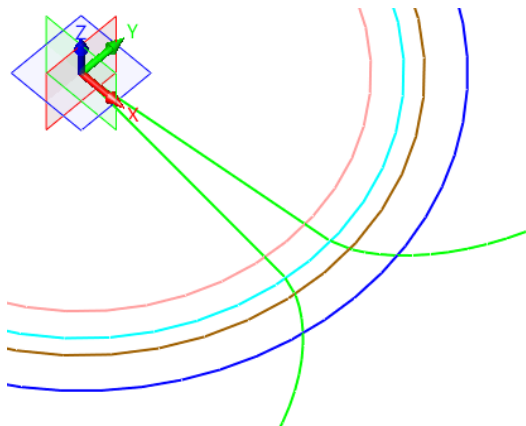


Figure 5. Create another Contour Line



1.6 Create the Extrude\_Cut profile by Sketch command.

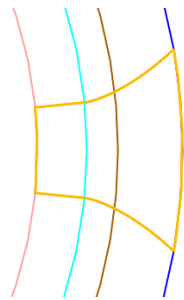


Figure 6. Create the Extrude\_Cut Profile

1.7 Extrude the Tip Circle to get the gear body. The height is the thickness of the gear which is driven by variable B. And then use the Sketch1 to create a tooth space.

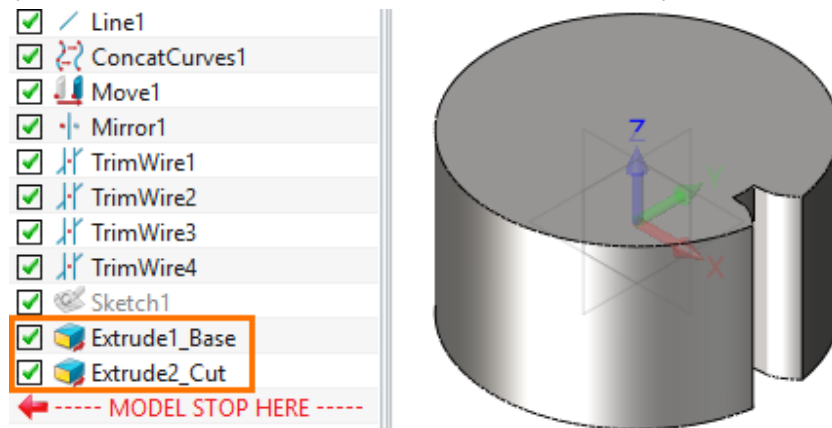


Figure 7. Extrude the Base and Create a Tooth Space

1.8 Pattern the Extrude\_Cut feature to create the other tooth space.

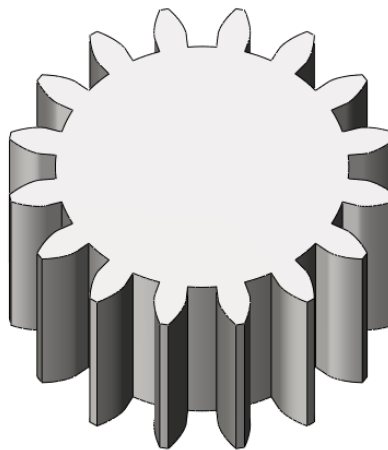


Figure 8. Gear Model

## 2. Create a Gear Library

Because the Number for Teeth, Module, Pressure Angle and Tip Coefficient have different standard values in different standards. So, it's easy to get a new gear shape by changing key parameters, including Module(M), Number of Teeth(z) and Tooth Width(B).



2.1 Open the Config Table, select the key parameters B, M and z. Then these 3 parameters are listed in the right window. Then many more configurations can be quickly created based on default one.

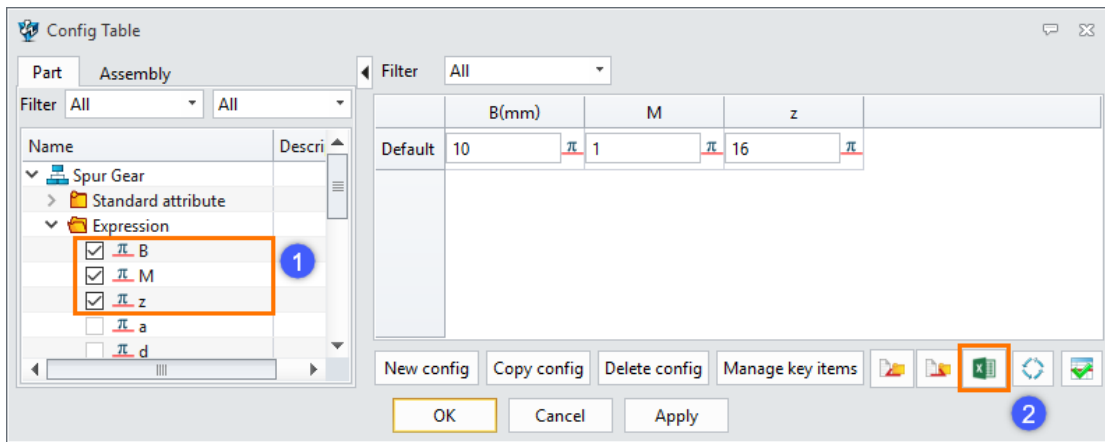


Figure 9. Config Table

2.2 Set the Gear Library parameters in Excel. Save and exit the Excel. Then go back to ZW3D, all parameters data have been automatically imported into the Config table.

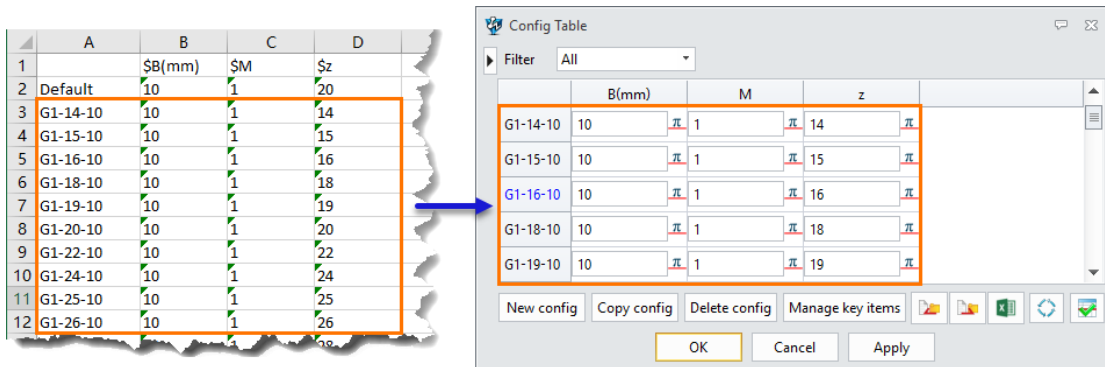


Figure 10. Set the Gear Library Parameters

2.3 As well, all Part Config items are listed in History Manager. Let's check it. Right click the G1-30-10 Config and active it, then the model is updated. Now the current gear has 30 teeth.

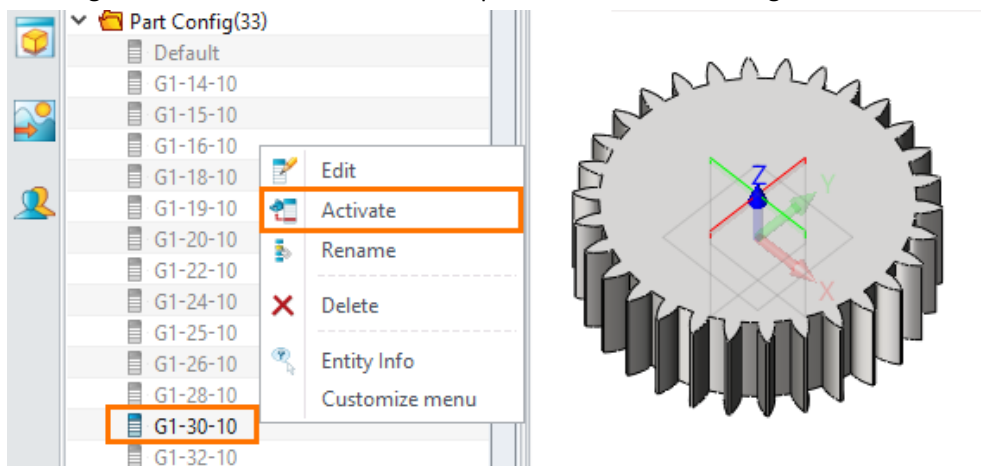


Figure 11. Active a Part Config



2.4 Save the file in a specific location, for example:

D:\ZWsoft\ZW3D 2018 Eng (x64)\Reuse Library\Reuse Examples\Gear

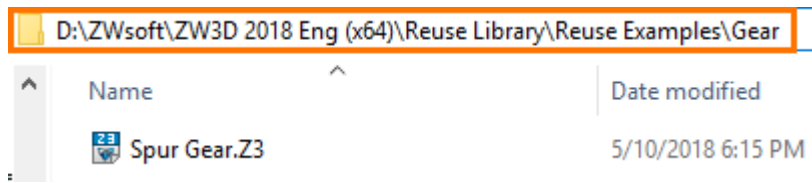


Figure 12. Save the Gear Model

### 3. Reuse the Gear

3.1 Open a new file and find Reuse Library command from the Assembly Ribbon Tab. Find the model file in Reuse Library Manager, select the parameters of the gear and pick OK.

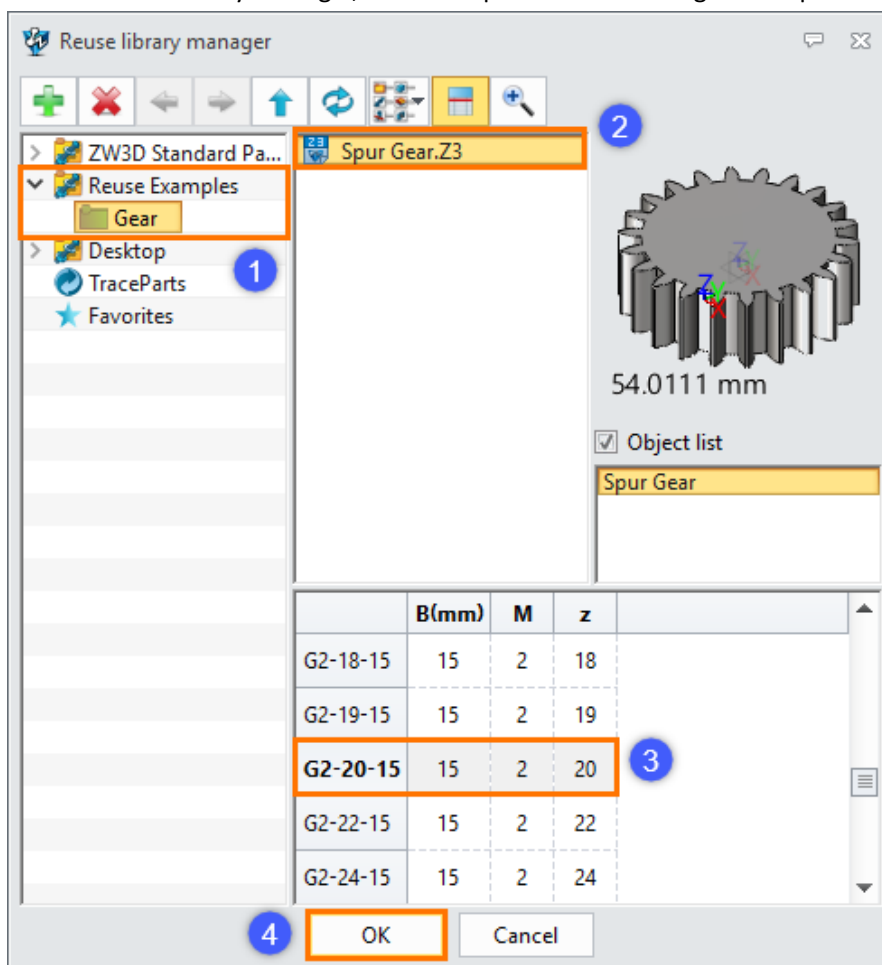


Figure 13. Select the Parameters of the Gear



3.2 Next, we can reconfirm the parameters again or set other parameters. Click ok to insert the gear part into the current file.

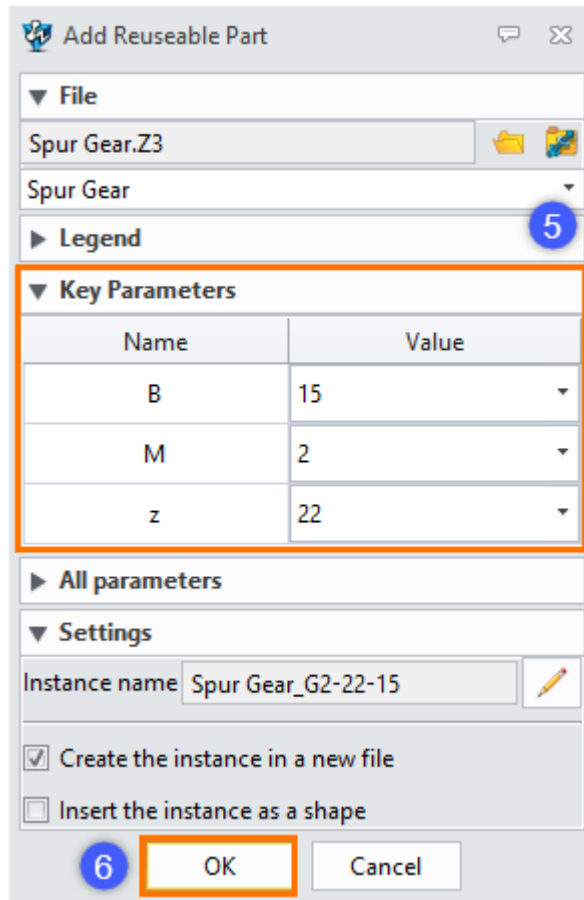


Figure 14. Reconfirm the Parameters

3.3 So far, the reuse of a gear library is verified.

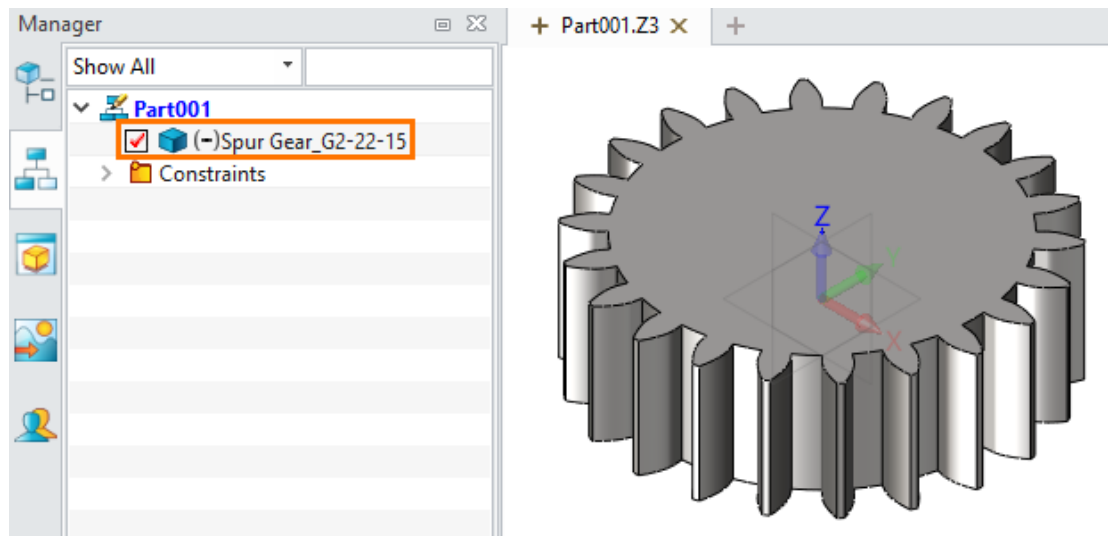


Figure 15. Reuse the Gear from Library