

## 3D modeling of Complex Structure based on AutoCAD VBA

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**Abstract:** *In this paper, 3DSolid models are widely used in many fields. Although it is easy in modeling, the repeat work is large. Automatic building for 3DSolid models can be performed by developing programs based on VBA for AutoCAD. The 3DSolid modeling method by VBA is discussed in this paper, and notices in modeling by VBA, which is obtained in author's work, are given.*

**Keywords:** VBA; modeling; AutoCAD

### 1. INTRODUCTION

Three types of 3D models are supported by AutoCAD: wireframe model, curved surface model and solid model. Because of the most application of solid model in 3D space, this paper only discusses modeling methods using solid model. [1].

### 2. METHODS OF 3d MODELLING

#### 2.1 Creating fundamental 3D object directly

This way is to create a solid according to fundamental solids. VBA provides several modeling orders of solids: AddBox, AddCone, AddCylinder, AddEllipticalCone, AddEllipticalCylinder, AddSphere, AddTorus and AddWedge. Although this way is very easy, but compared with modeling with AutoCAD directly, it is a little more complex. Some points should be concentrated on: when using VBA to model, the coordinate used by all points is that in word coordinate system(WCS) and models are created under WCS. And the fundamental underside of models is parallel to XY plane in WCS. It means that models cannot be created in predesigned place by using user coordinate system(UCS) directly, which causes some troubles in modeling by using these orders. Reasons are as follows: the underside of models(like cylinder's two back planes and cone's underside) may be planes in space, which are not certainly parallel to WCS'XY plane, and modeling by using these orders cannot achieved directly. There are two methods of solving this problem.

Number1. Creating the model in WCS first, then put it at the correct place by moving, revolving, aligning and so on. Modeling in this way is similar to modeling in AutoCAD directly. It can be achieved but is comparatively complicated.

Number2. Creating models at certain place first in WCS then using UCS at this place, get the switch matrix of this UCS and put the model at predesigned place using this matrix. For example, to create the elliptical table showed in picture 1(located right at the picture, the model's underside is an ellipse that is not parallel to XY plane in WCS), the procedures to model are as follows: 1. setting the underside ellipse'core as origin, the ellipse's long axis as X axis and the ellipse's short axis as Y axis, create user coordinate system(name: NewUcsObj showed in the picture); 2. get relevant size parameters about the elliptical cone which

contains this elliptical table; 3. use order AddEllipticalCone to create an elliptical cone whose underside ellipse's core is WCS's origin; 4. use order SliceSolid to cut this elliptical cone as the needed elliptical table(its name is TYTObj, located left at the picture); 5. get UCS switch matrix by following orders and then put the elliptical table at designed place using this switch matrix.  
' set new-created UCS as current UCS  
ThisDrawing.ActiveUCS = NewUcsObj  
' get switch matrix of UCS  
Dim TransMatrix As Variant  
TransMatrix = NewUcsObj.GetUCSMatrix()  
' put the elliptical table from WCS to UCS(that is to put it to predesigned place)  
TYTObj.TransformBy (TransMatrix)  
Modeling the elliptical table can be achieved through these operations above

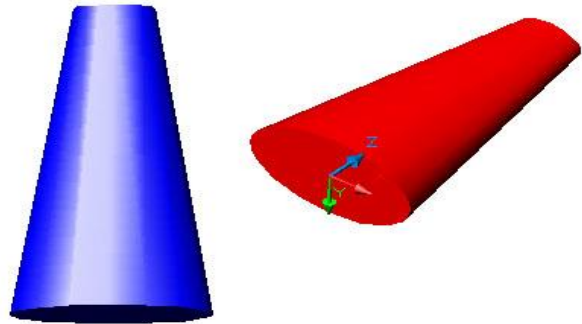


Fig 1. Elliptical table's modeling

#### 2.2 Creating solids by way of stretching, revolving and so on

This way is to create solids by stretching and revolving 2D or 3D line objects. VBA provides several modeling order : AddExtrudedSolid, AddExtrudedSolidAlongPath and AddRevolvedSolid.

This method is similar to modeling with AutoCAD directly, which needs to create a UCS and activate it. Then create planes which should be stretched or revolved to make them become solids in UCS.

Compared with the first method, this way is easier and more practical. For example, Box, cylinder, elliptical cylinder and so on can be created by being stretched; cone, cylinder, torus and so on can be made by revolving.

In addition, for some complicated solids, realizing their modeling should use operations like intersect, union, subtract and so on among solids based on that fundamental models have been created.

### 3 NOCICES IN 3d MODELLING BASED ON VBA

#### 3.1 user coordinate system's construction

It is easy to construct UCS in AutoCAD, taking picture 2a as an example. If you want to set point O as the coordinate'origin, line OA as X axis and the line which is perpendicular to line OA and up as Y axis, you can use the UCS order:founding coordinate system with three points. That is to choose point O as coordinate origin at first; set point A at positive X axis and choose point B directly among all points in positive Y axis. Then regardless of whether line OB is perpendicular to line OA or not,the UCS can be constructed successfully like just like picture 2b.

However, when founding UCS with VBA, line OB must be perpendicular to line OA. Otherwise, UCS can not be constructed. Therefore, to found UCS in plane AOB, two assistant perpendicular lines should be made . Then UCS's construction can be realized. The author applies following operations by practicing: first cut or stretch line OB to the strength of line OA and create a rhombus whose connecting sides are line OA and line OB; then use the characteristic that the rhombus's two diagonals are perpendicular and create two perpendicular lines to construct UCS; finally delete all assistant lines.

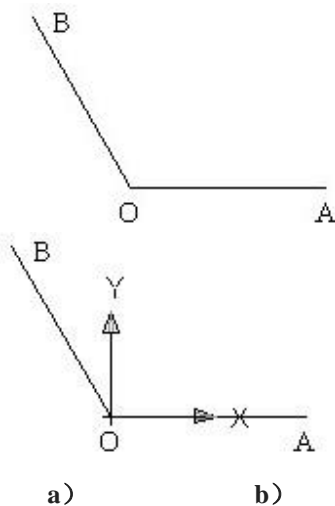


Fig 2. Construction of UCS

### 3.2 using method SendCommand correctly

When modeling with VBA, to visit AutoCAD and realize the alternation with AutoCAD, it is feasible to send commands to AutoCAD order line directly by using method SendCommand. This method sends single string which must contain parameters of commands to be operated to order line, and then arrays these parameters according to this command's reference order. The space bar and return key's ASCII in strings are equal to pressing ENTER key in a keyboard. For example, when drawing a circle whose radius is 10 and core is at point (0,0,0), the following two commands are equal:

```
ThisDrawing.SendCommand "_Circle 0,0,0 10 "
```

```
ThisDrawing.SendCommand "_Circle" & vbCr & "0,0,0" & vbCr & "10" & vbCr
```

Using method SendCommand can make up for VBA's defects in drawing commands'shortage. For instance, there is no command to align in VBA. When this command is required, sending it to AutoCAD order line directly using method SendCommand to complete this operation.

### 3.3 No dependence on events'order of occurrence and operation

When you are writing events operation programs, don't rely on the order that you think is the one in which this event occurs and is operated. For example, as for following programs:

```
ThisDrawing.SendCommand "_Circle 0,0,0 50 "
```

```
Dim Center(0 To 2) As Double
```

```
Dim Radius As Double
```

```
Center(0) = 0: Center(1) = 0: Center(2) = 0: Radius = 20
```

```
Dim circleObj As AcadCircle
```

```
Set circleObj = ThisDrawing.ModelSpace.AddCircle(Center, Radius)
```

```
ThisDrawing.SendCommand "_LINE 0,0,0 -20,80,100 "
```

The result(when the programs are alone) can be: drawing a circle whose radius is 50 first then drawing a circle whose radius is 20 and finally drawing a line. This order is consistent to programs' operation. However, the result(when programs are with the form and other sentences) can also be: drawing a circle whose radius is 20 first then drawing a circle whose radius is 50 and drawing a line finally. This order is not the same as that of programs' operation. That is, when VBA and SendCommand order are used together, they are not operated in the order which they are sent in certainly.

### 3.4 plane domain's formation and application

In AutoCAD, when a plane domain is formed, each part(line, curve) that contributes to this plane domain will not be alone and then become an element of this plane domain. What's more, when the plane domain is stretched or revolved, it will disappear. It will be a part of the constructed solid. However, in VBA, when a plane domain is formed, each part of the plane domain exists alone as well. Similarly, when the plane domain is stretched or revolved, it will be reserved. Therefore, they should be deleted.

### 3.5 Using circumference ratio $\pi$ correctly

When forming revolved solid with method AddRevolvedSolid, its unit for revolving angle is radian. Circumference ratio is used here. If the revolving angle is less than  $360^\circ$ , choices of  $\pi$  will not have any influence on the picture to be formed. But when the revolving angle is equal to  $360^\circ$ , the last number of  $\pi$ 'value after the point must be plus 1. Otherwise, the formed revolved solid will not close. For example, the value of  $\pi$  can be 3.14160 but not 3.14159. Picture 3 is the figure when  $\pi$  is 3.14159. From it we can see that after romancing, there is a crack in this figure. But when  $\pi$  is 3.14160, the figure closes and this phenomenon does not appear.

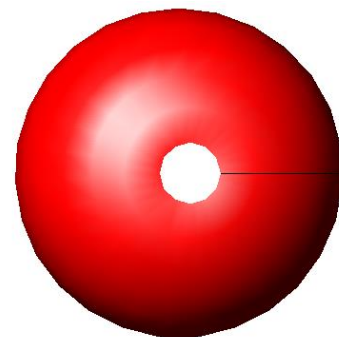


Fig 3. unclosed revolved solid

### 3.6 Using stretching and inclining angle correctly

Just like modeling in AutoCAD directly, when using AddExtrudedSolid to stretch a plane domain in VBA, if the stretching inclining angle is positive, it means that the plane domain is getting thinner gradually from standard object by being stretched; otherwise, if the value is negative, the plane domain is stretched to be thicker gradually from standard object. The default angle 0 means that the plane domain is stretched towards the direction which is perpendicular to the 2D object's plane.

### 3.7 Addition

When modeling by using VBA, some commands' input parameters are different from those in modeling directly with AutoCAD. It is necessary to figure out significance of these input parameters carefully. For example, when constructing a cylinder (or cone), its underside's core should be input; however, in VBA, we should input its own core (that is, core of its height). The input array number is one more than actual value when using command ArrayPolar to realize annular array in VBA. Parameters of rotation axis are its starting point and the 3D vector pointing its direction when using method AddRevolvedSolid to form solids by revolving. But we should input rotation axis's starting point and end point in AutoCAD.

## 4 CONCLUSIONS and PERORATIONS

As a secondary development tool of AutoCAD, VBA is easy to use and its functions are very strong. Developing parameterized modeling programs by using it, designers can be set free from complicated and repeated work. Work efficiency can be improved obviously.

## 5 References

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