Open Cascade 中的布尔操作

Modeling Algorithms Boolean Operations

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布尔操作（*Boolean Operations*）是通过两个形状（*S1*，*S2*）的组合来生成新的形状。布尔操作有如下几种类型：
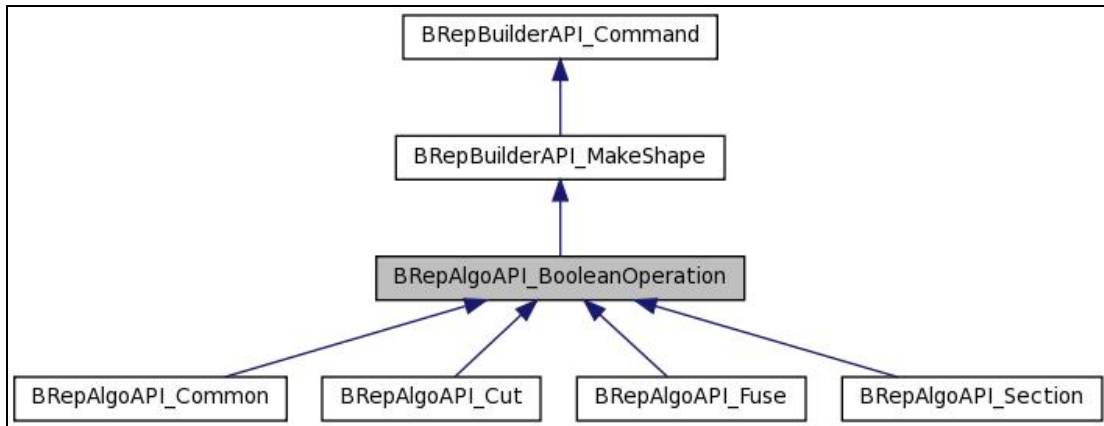
- 并集操作 *Fusion*：*Gets all the points in S1 or S2*;
- 交集操作 *Common*：*Gets all the points in S1 and S2*;
- 差集操作 *Cut S1 by S2*：*Gets all the points in S1 and not in S2*;

下图所示为三种布尔操作：
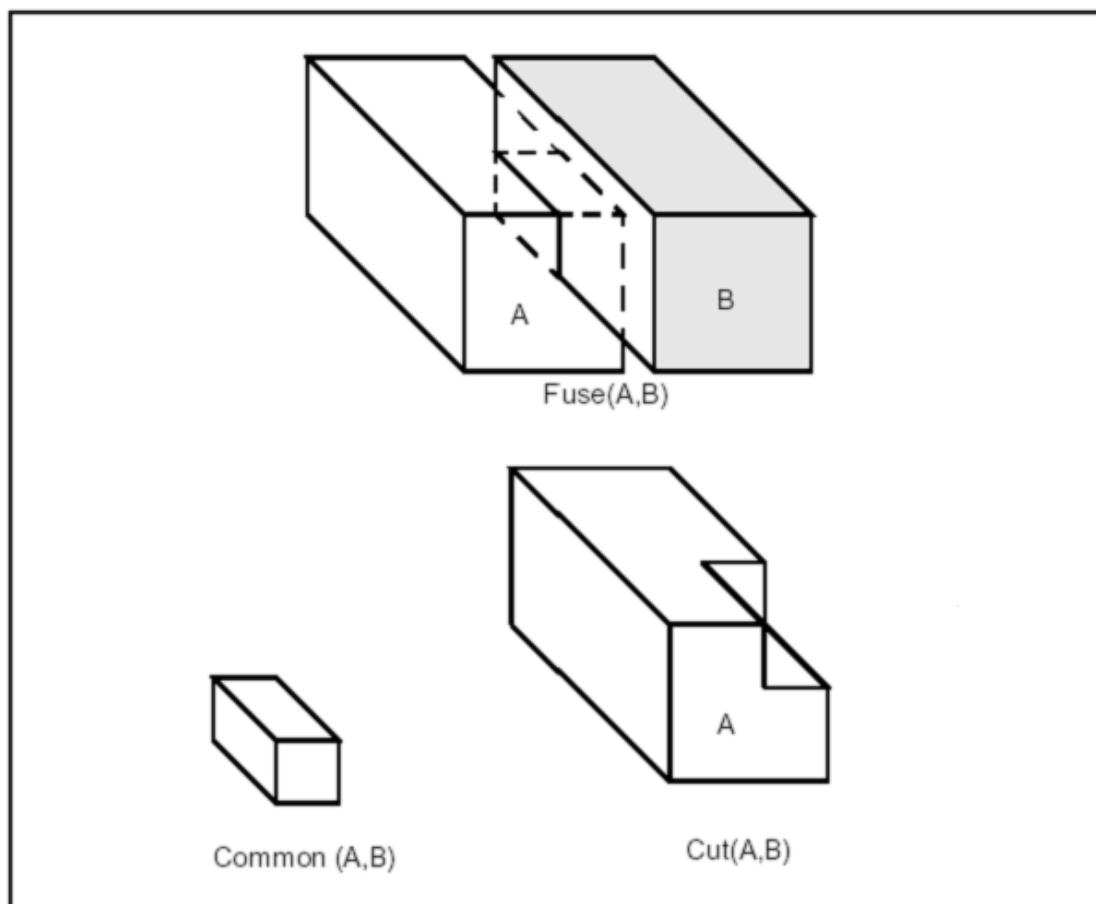
1. ***BRepAlgoAPI_BooleanOperation***

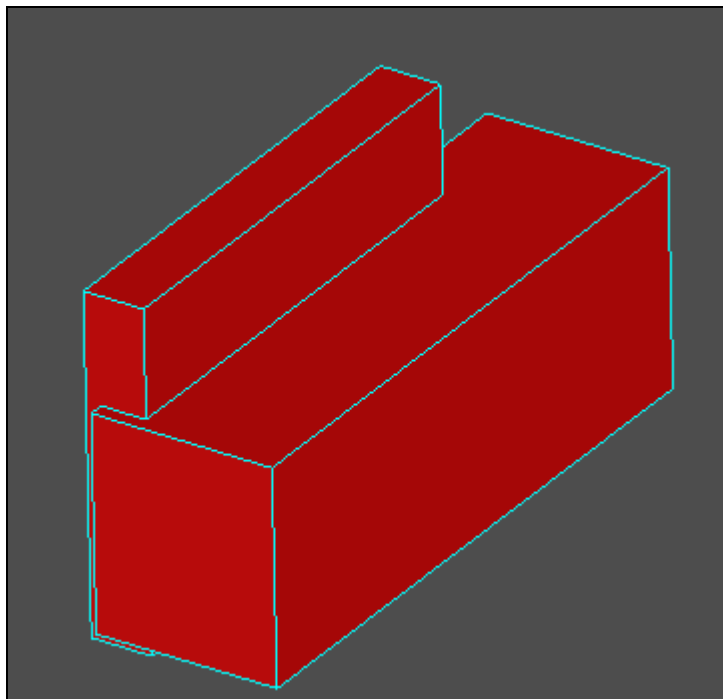   类 ***BRepAlgoAPI_BooleanOperation*** 是布尔操作的基类。

2. ***BRepAlgoAPI_Fuse***

类 ***BRepAlgoAPI_Fuse*** 执行布尔并集操作。如下所示：

```
TopoDS_Shape theBox1 = BRepPrimAPI_MakeBox(50, 200, 70);
TopoDS_Shape theBox2 = BRepPrimAPI_MakeBox(-30, 150, 70);
TopoDS_Shape FusedShape = BRepAlgoAPI_Fuse(theBox1, theBox2);
```
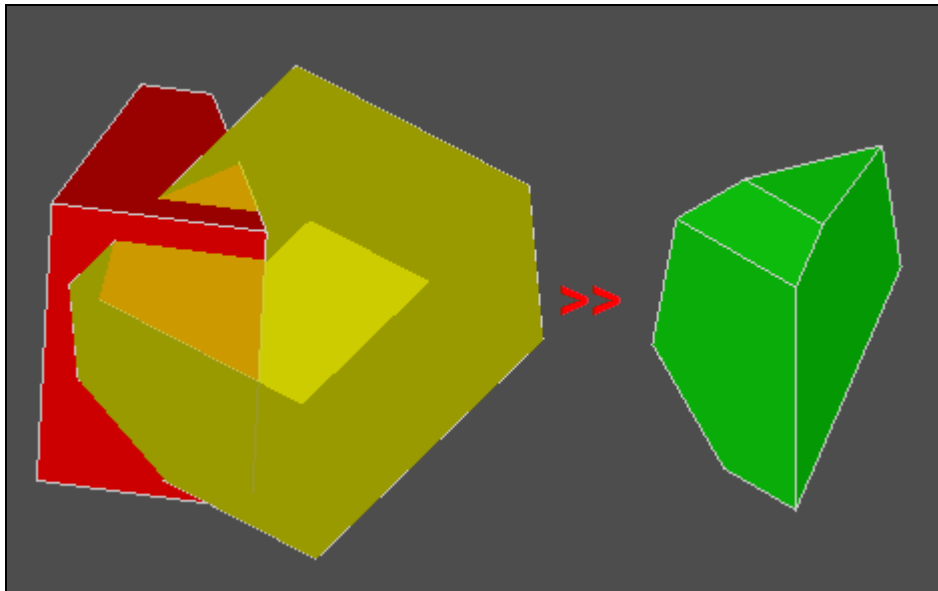

Fuse two boxes

3. *BRepAlgoAPI_Common*

类 *BRepAlgoAPI_Common* 执行布尔交集操作，如下所示：

```
gp_Ax2 axe(gp_Pnt(10,10,10),gp_Dir(1,2,1));
TopoDS_Shape theBox = BRepPrimAPI_MakeBox(axe,60,80,100);
TopoDS_Shape theWedge = BRepPrimAPI_MakeWedge(60.,100.,80.,20.);
TopoDS_Shape theCommonSurface = BRepAlgoAPI_Common(theBox,theWedge);
```
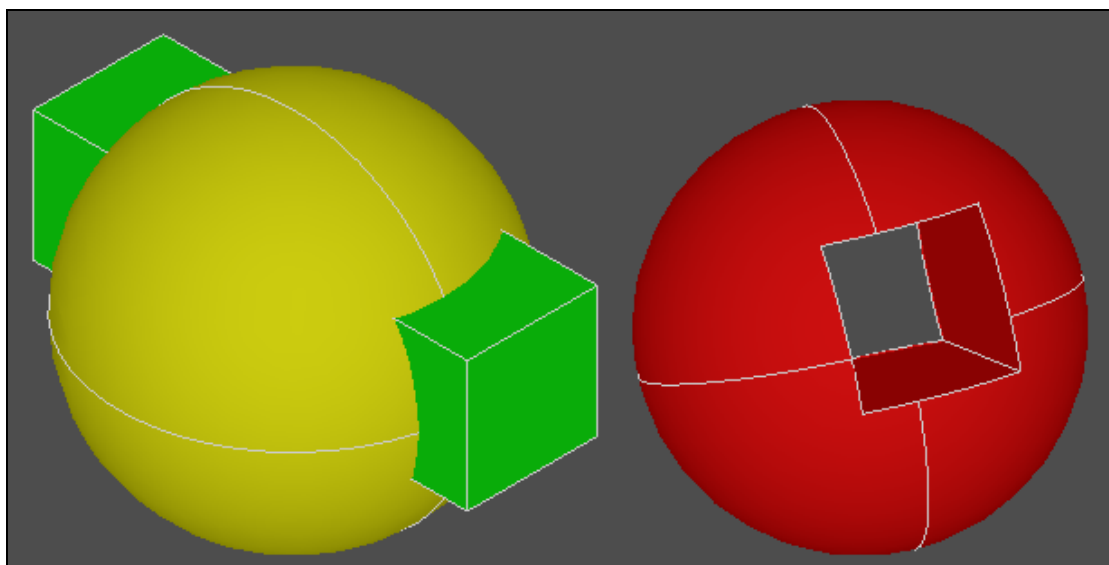

Compute the common surface

4. *BRepAlgoAPI_Cut*

类 *BRepAlgoAPI_Cut* 执行布尔差集操作，如下所示：

```
TopoDS_Shape theBox = BRepPrimAPI_MakeBox(200,40,40);
TopoDS_Shape theSphere = BRepPrimAPI_MakeSphere(gp_Pnt(100,20,20),80);
TopoDS_Shape ShapeCut = BRepAlgoAPI_Cut(theSphere,theBox);
```


5. *BRepAlgoAPI_Section*