CAD 2

LEC 4

Drawing Drawing ball and soket joint in solidworks

Steps to Drawing Drawing ball and soket joint

- **1. Create the Ball (Spherical Part)**
- **Step 1:** Open a new part document in SolidWorks.
- **Step 2:** In the top plane, start a new **Sketch**. Select the **Circle** tool and draw a circle with the desired radius (for the ball).
- **Step 3:** Exit the sketch, and on the **Features** tab, select **Revolve Boss/Base**.
- **Step 4:** Choose the vertical axis of the sketch to revolve and specify a full 360-degree revolution to create a sphere.
- **Step 5:** Save the ball part as a separate file (e.g., **Ball.sldprt**).
- **2. Create the Socket (Hollow Part)**
- **Step 1:** Create a new part document in SolidWorks.
- **Step 2:** In the top plane, start a **Sketch**. Draw a circle that is slightly larger than the radius of the ball, which will form the outside of the socket.
- **Step 3:** Draw a smaller circle concentric to the first one. This smaller circle will define the interior of the socket.
- **Step 4:** Extrude the sketch to the desired length to form a hollow cylinder.
- **Step 5:** Now, you need to create a spherical cut to make the socket part. Use the **Insert > Cut > Revolve** feature.
- **Step 6:** In the sketch for the revolve, draw a profile for the cut that is the shape of the ball, but only for one-quarter of the sphere, and revolve it to remove material.
- **Step 7:** Save the socket part as a separate file (e.g., **Socket.sldprt**).

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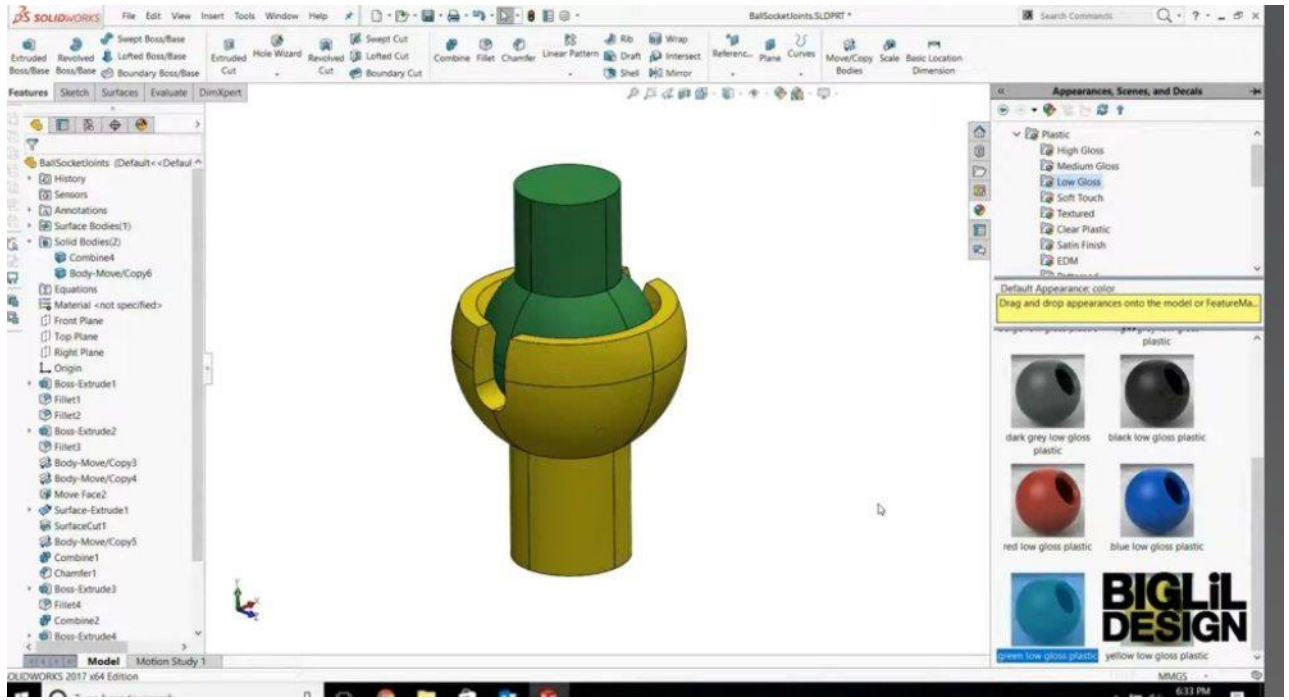
- **3. Assemble the Ball and Socket**
- **Step 1:** Open a new assembly document.
- **Step 2:** Insert both the ball and the socket parts into the assembly.
- **Step 3:** Apply a **Mate** to align the ball with the socket. Use the **Mate** tool to insert a **Concentric Mate** between the ball's center and the socket's interior and a **Distance Mate** to control how far the ball sits in the socket.
- **Step 4:** You can also apply a **Tangent Mate** if necessary to prevent movement along certain directions.
- **4. Finalize the Assembly**
- **Step 1:** You can add additional mates to lock the ball in position (e.g., **Limit Mate**) to restrict how much the ball can move inside the socket.
- **Step 2:** Test the assembly to ensure the ball and socket function correctly, and adjust the mates as needed.


SOLIDWORKS
File Edit View Insert Tools Window Help
BallSocketJoints.SLDPRT *
Search Commands
Extruded Boss/Base
Revolved Boss/Base
Swept Boss/Base
Lofted Boss/Base
Boundary Boss/Base
Extruded Cut
Hole Wizard
Revolved Cut
Swept Cut
Lofted Cut
Boundary Cut
Combine
Fillet
Chamfer
Linear Pattern
Rib
Draft
Shell
Wrap
Intersect
Mirror
Reference...
Plane
Curves
Move/Copy Bodies
Scale
Basic Location Dimension
Features
Sketch
Surfaces
Evaluate
DimXpert
BallSocketJoints (Default<<Defaul
History
Sensors
Annotations
Surface Bodies(1)
Solid Bodies(2)
Combine4
Body-Move/Copy6
Equations
Material <not specified>
Front Plane
Top Plane
Right Plane
Origin
Boss-Extrude1
Fillet1
Fillet2
Boss-Extrude2
Fillet3
Body-Move/Copy3
Body-Move/Copy4
Move Face2
Surface-Extrude1
SurfaceCut1
Body-Move/Copy5
Combine1
Chamfer1
Boss-Extrude3
Fillet4
Combine2
Boss-Extrude4
Model
Motion Study 1
Appearances, Scenes, and Decals
Plastic
High Gloss
Medium Gloss
Low Gloss
Soft Touch
Textured
Clear Plastic
Satin Finish
EDM
Default Appearance: color
Drag and drop appearances onto the model or FeatureMa...
dark grey low gloss plastic
black low gloss plastic
red low gloss plastic
blue low gloss plastic
green low gloss plastic
yellow low gloss plastic
BIGLiL DESIGN
SOLIDWORKS 2017 x64 Edition
MMGS
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