CAD 2

LEC 4

Drawing shoulder joint in solidworks

Steps to Drawing shoulder joint in solidworks

- **1. Create the Ball (Humeral Head)**

- The ball of the shoulder joint is a spherical component that fits into the socket.

- **Step 1:** Open a new **Part** document in SolidWorks.

- **Step 2:** On the **Top Plane**, create a **Sketch**. Select the **Circle** tool and draw a circle to represent the ball (the humeral head). Define the diameter (e.g., 50 mm).

- **Step 3:** Exit the sketch. Go to the **Features** tab and select **Revolve Boss/Base**.

- **Step 4:** Select the vertical axis of the circle (usually the centerline) and set the revolve to 360 degrees to create a full sphere.

- **Step 5:** Add any necessary fillets or features to the ball if required for specific geometry (e.g., smooth contours).

- **Step 6:** Save the ball as a part file, for example, **Ball.sldprt**.

- **2. Create the Socket (Glenoid Cavity)**

- The socket of the shoulder joint is a concave surface that the ball will fit into.

- **Step 1:** Open a new **Part** document in SolidWorks.

- **Step 2:** Create a **Sketch** on the **Top Plane**. Draw a circle larger than the ball's diameter to represent the outer edge of the glenoid cavity (socket).

- **Step 3:** Create a concentric smaller circle that will form the inner cavity (glenoid) for the ball to fit into. Ensure the diameter of the inner circle is slightly larger than the ball diameter to allow a good fit.

- **Step 4:** Select the **Extruded Boss/Base** feature and extrude the socket to the desired depth (typically 10–15 mm, depending on the required joint size).

- **Step 5:** To create the spherical surface for the socket, use **Fillet** or **Shell** features to smooth the inside surface.

- **Step 6:** Save the socket part as a separate file, for example, **Socket.sldprt**.

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- **3. Assemble the Ball and Socket**

- Now that you have the ball and the socket, you can assemble them into a working shoulder joint.

- **Step 1:** Open a new **Assembly** document in SolidWorks.

- **Step 2:** Insert both the **Ball** and **Socket** parts into the assembly.

- **Step 3:** Apply **Mate** constraints to position the ball inside the socket:
    - Use a **Concentric Mate** to center the ball with the socket's inner edge.
    - Use a **Tangent Mate** to allow the ball to roll along the socket surface.

- **Step 4:** You can use a **Limit Mate** or **Distance Mate** to control the range of motion of the ball inside the socket.
    - For example, use a **Distance Mate** to control how far the ball can move along the axis.
    - Apply **Limit Mates** to restrict the angular movement of the ball to replicate the shoulder's range of motion.

- **4. Simulate the Joint Movement**

- **Step 1:** To visualize and test the joint's movement, you can use the **Motion Study** feature in SolidWorks to simulate the ball's movement within the socket.

- **Step 2:** Apply the **Motor** or **Mate Controllers** to create realistic motion for the shoulder joint.

- **Step 3:** Check for interference or collisions during movement. Adjust the socket's size or ball radius if necessary.