Experiment No. - 05

Aim :- To create a 3-D design of Gear in Creo Parametric software.

Equipments required :- A computer system with creo parametric software installed in it.

Theory :-

Designing a gear in Creo Parametric involves creating a base circular sketch and using features like Extrude to form the gear body. Advanced features, such as Pattern tools, allow for the replication of gear teeth around the base. Parameters like module, pressure angle, and pitch diameter are set to ensure the correct profile and fit.

Proper gear design is critical in mechanical systems to ensure efficient power transmission and longevity. An accurately designed gear reduces noise, minimizes wear, and enhances mechanical efficiency. In this experiment, designing a gear in Creo provides insights into the precision and adaptability required for industrial applications.

Key parameters for designing a gear include:

- Module (m): Defines the size of the teeth; it's the ratio of the pitch diameter to the number of teeth.
- Pitch Diameter (D): The diameter at which two gears effectively make contact.

- Number of Teeth (Z): Determines the gear ratio and affects torque and speed.
- Pressure Angle (α): The angle at which force is transmitted between meshing gears, impacting load capacity and wear.
- Addendum and Dedendum: The height of the gear tooth above and below the pitch circle, respectively.

Procedure :

To create a gear design we will follow following steps :-

1.) Start a New Sketch for the Gear Profile

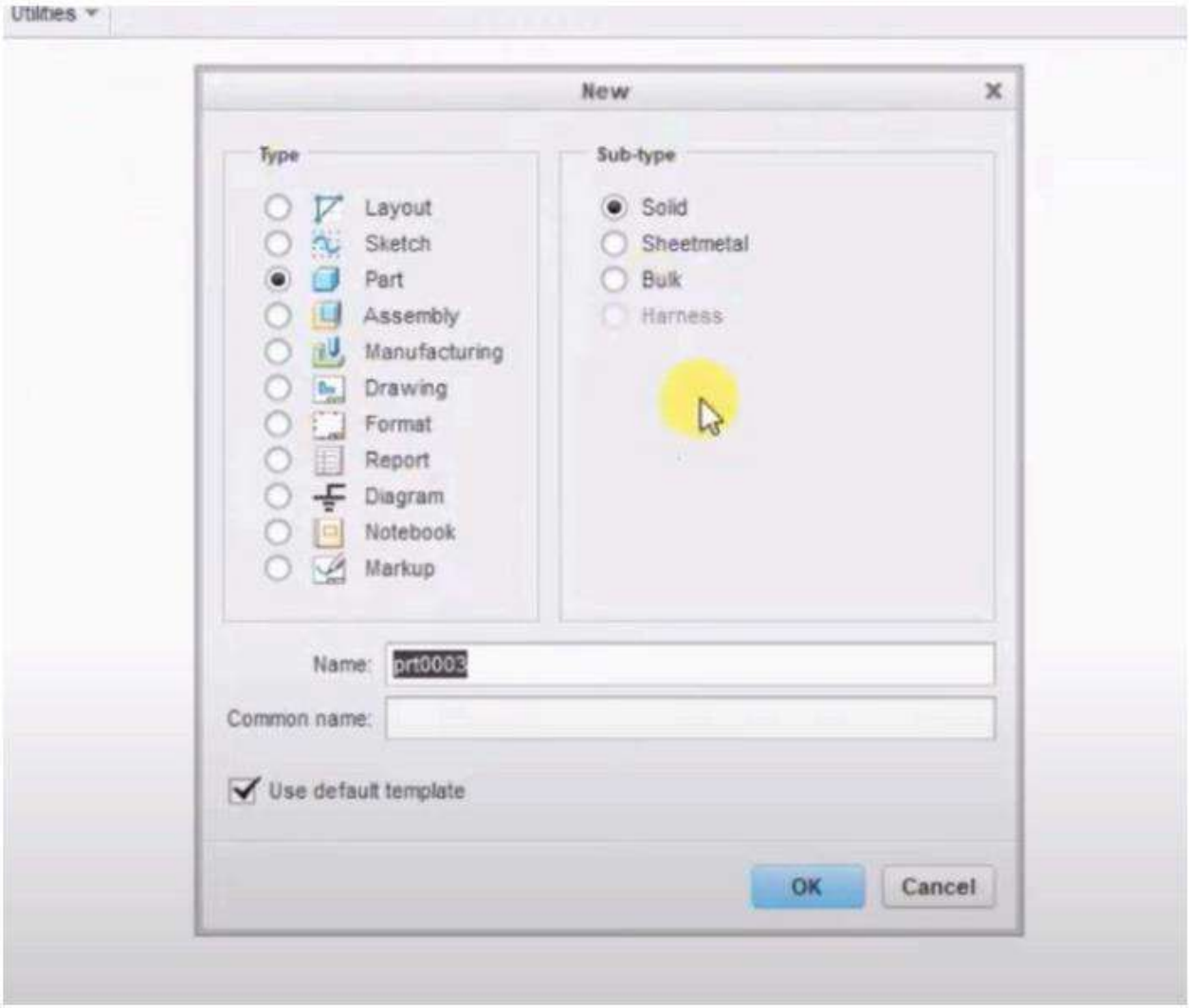
Open a new part file and create a 2D sketch on an appropriate plane. Draw a circle representing the pitch circle diameter, which serves as the basis for gear tooth placement.

2.) Draw the Gear Tooth Profile

Sketch a single gear tooth profile on the pitch circle. Use constraints and dimensions to control the tooth's height, width, and angle, ensuring it meets the design specifications.

3.) Use the Pattern Tool

Once the gear tooth profile is complete, use the **Pattern** feature



to replicate it around the pitch circle. Select the appropriate parameters (like number of teeth and spacing) to evenly distribute the teeth along the circle.

4.) Extrude the Gear Profile

Use the **Extrude** feature to add thickness to the sketched profile, creating a 3D model of the gear. Set the extrusion depth based on the required width of the gear.

5.) Add Fillets and Chamfers

Add fillets or chamfers to the gear edges to reduce stress concentrations and enhance durability. This step helps in improving the strength and longevity of the gear in mechanical Applications.

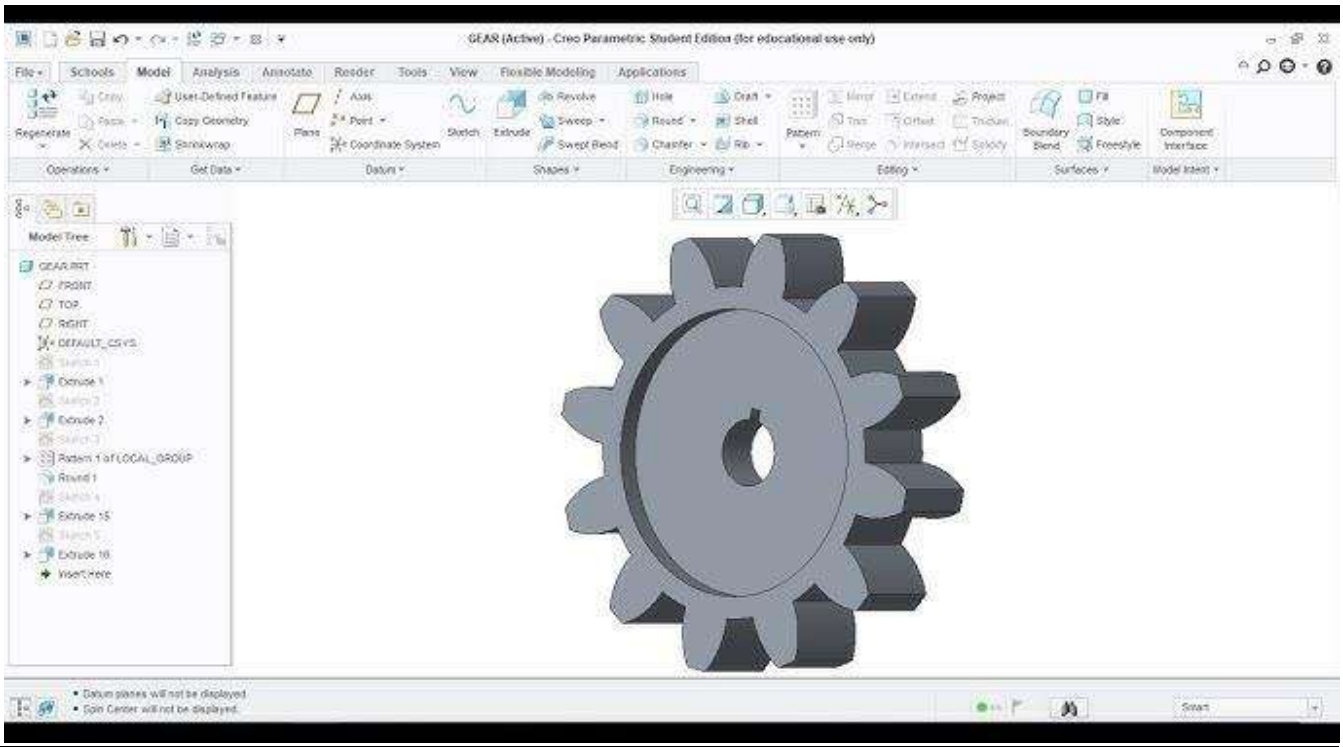
6.) Apply Final Adjustments

Review the model and make any necessary adjustments to parameters like module or tooth profile. The parametric nature of Creo allows easy modifications that will automatically update the entire model.

7.) Finalize the Model

After finalizing all design aspects, save the model. Optionally, use

Creo's rendering tools to visualize the gear or export it for further analysis, like stress testing or meshing with another gear.



Conclusion :

In conclusion, the gear design in Creo Parametric demonstrated effective use of parametric and feature-based tools to create a precise and adaptable model. By following structured steps—from defining parameters to patterning teeth and adding final adjustments—the experiment highlighted the software's capability to support efficient, accurate gear modeling. This exercise reinforced essential CAD skills applicable to real-world engineering tasks, emphasizing the importance of precise design in mechanical applications.