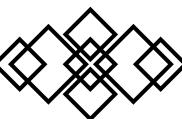


PROJECT REPORT

PTC CREO

JANUARY 2024

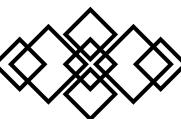
**NAME: MUHAMMAD
HASHIR ARIF
DEGREE: 44-ME
SYNDICATE: C
COURSE: ME 211
COMPUTER AIDED
DRAWING
TO: SIR
HASSAN**



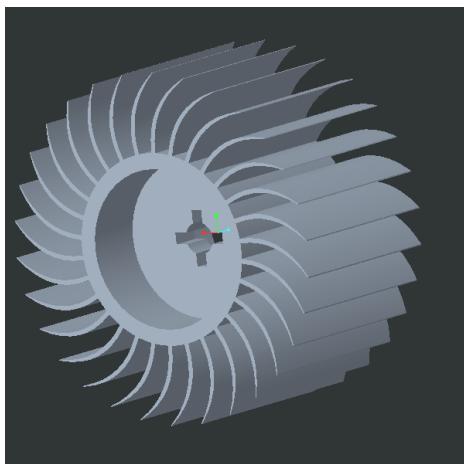
Project Title: vacuum mower

Description:

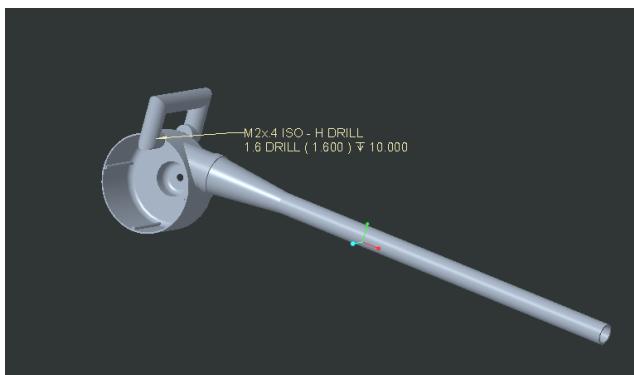
The vacuum mower is a new product that combines the functionality of a lawn mower with a vacuum blower pump that sucks up grass, leaves, and other debris. The vacuum blower pump is attached to the lawn mower and is designed to collect all of the debris in a bag, which can be easily emptied and disposed off. The vacuum mower is designed to be environmentally friendly, as it reduces the amount of waste produced by traditional lawn mowers. It is also optimized for ease of use and maintenance, with a simple control panel that allows the user to adjust the speed and suction power of the vacuum blower pump. The vacuum mower is a versatile and compact machine that can be used in a variety of environments, making it an ideal choice for homeowners and landscapers alike.



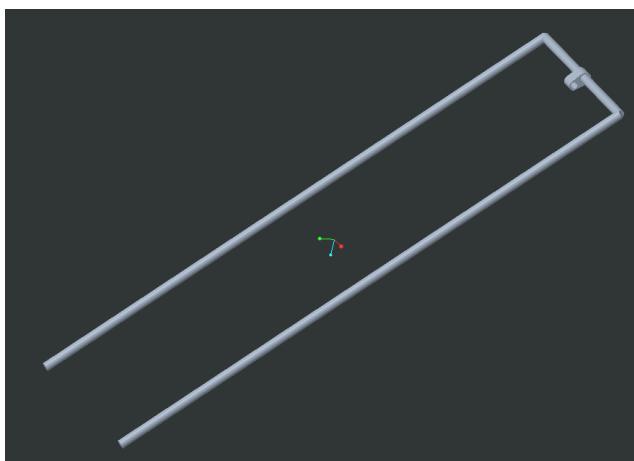
PARTS



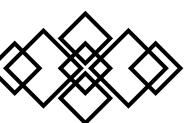
FAN

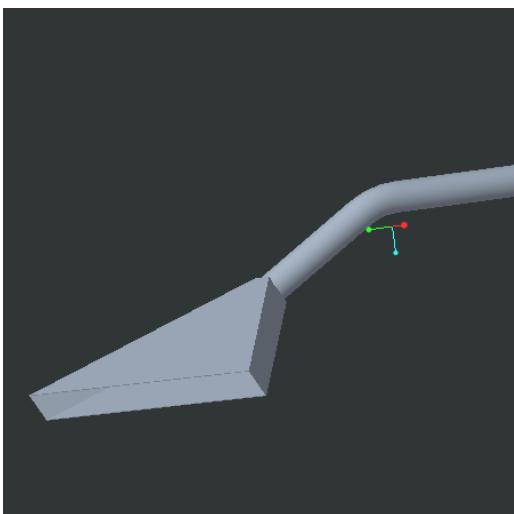


FAN CASING

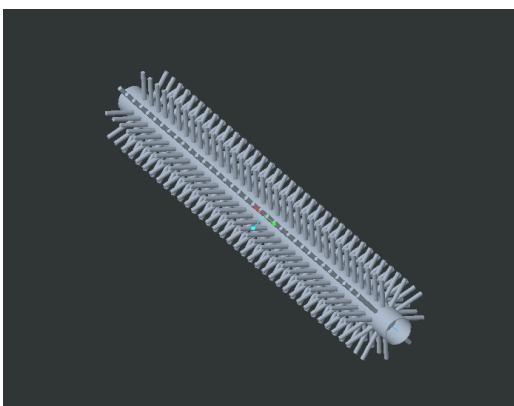


HANDLE BAR

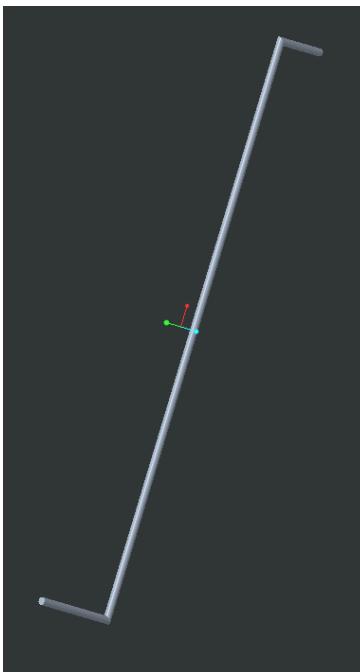




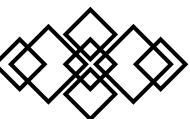
VACCUM SUCCION

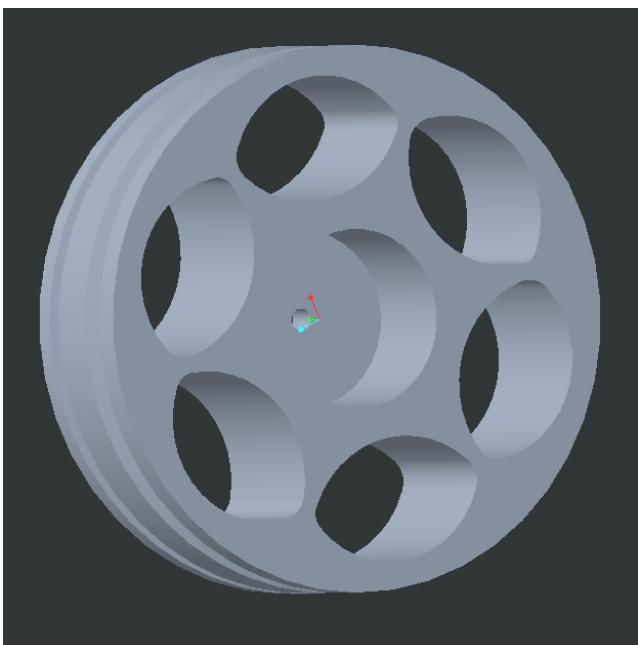


CUTTER

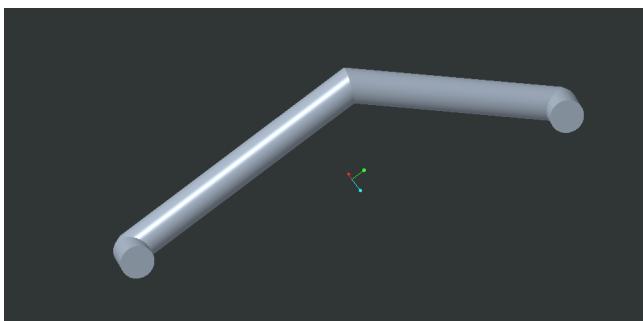


WHEEL MOUNT

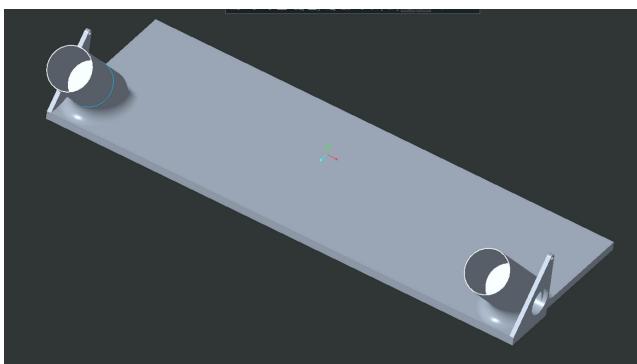




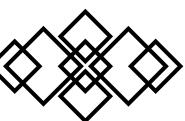
WHEEL



CUTTER MOUNT



BASE



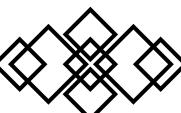
TOOLS USED

SWEEP

The Sweep command in Creo is used to create a solid or surface feature by sweeping a cross-sectional sketch along one or more selected trajectories. Material can be added or removed as the sketch sweeps along the trajectory

BLEND

The Blend command in Creo is used to create a continuous feature from multiple parallel planar sections. It can be used to blend edges, surfaces, or faces

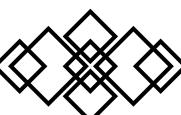


PATTERN

The Pattern command in Creo is used to create a pattern of features or parts. It can be used to create patterns of Creo Elements/Direct Machining features, Creo Elements/Direct Mold Design features, or face set features or parts

REVOLVE

The Revolve command in Creo is used to create a solid or surface feature by revolving a sketched section around an axis of revolution. The feature can be added or removed, and the orientation, rotation, and geometry of the object can be controlled

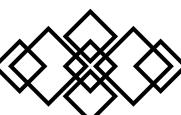


EXTRUDE

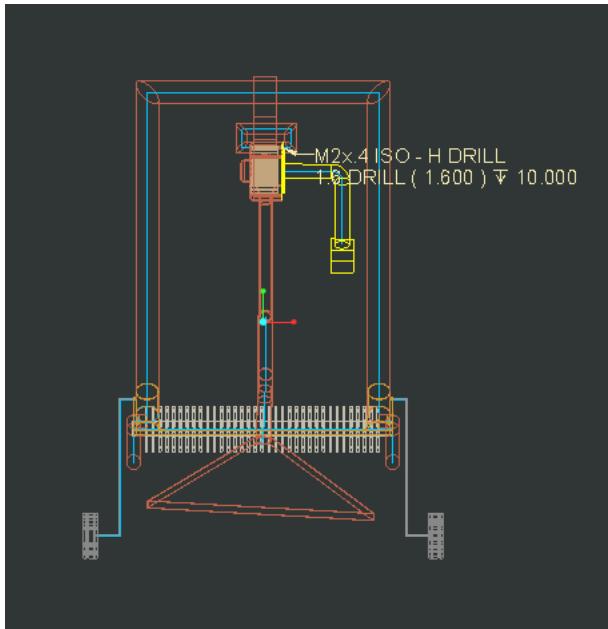
The Extrude command in Creo is used to create a solid or surface feature by translating a two-dimensional sketch normal to the sketch plane, for a pre-defined distance or up to a specified reference. Material can be added or removed

ROUND

The Round command in Creo is used to add a radius or chord to one or more edges, an edge chain, or the space between surfaces. The feature can be created using one of the following modes: Full, Variable, Chordal, or Surface-to-Surface



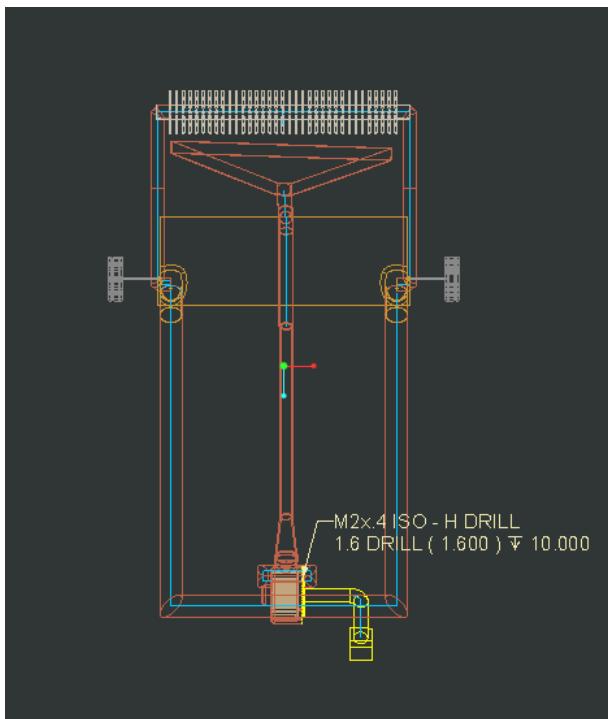
2D VIEWS



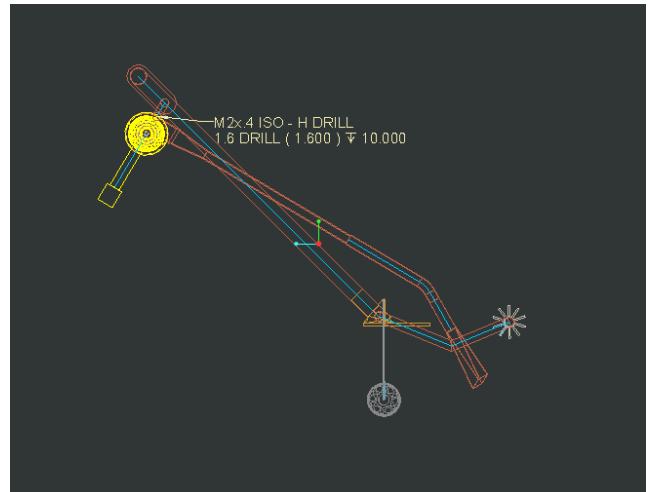
TOP VIEW



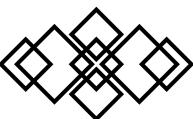
STANDARD VIEW



FRONT VIEW



SIDE VIEW



EXPLODED VIEW

