

## Exercise 02565-09

**Pro/ENGINEER - Sketching**

## Reading

Pro/ENGINEER Part Modelling User Guide.

## Purpose

The purpose of this exercise is to more systematically explore the facilities of the sketcher.

## Part 1

Make extruded, rotated, and swept solids based on the profiles shown in appendix A and B. Play with the extrusion vector, rotation angle and the path in the sweeping.

## Part 2

How would you be able to define the objects in appendix C using boolean operations. Make a geometric model of the object in ProEngineer.

## Part 3

Make a geometric model of the object defined in appendix D.

## Part 4

Make a geometric model by extruding a profile of your own choice. In defining the profile you should use as many “features” of the sketcher as possible. In particular you should work with the sketcher constraints, Trim/Divide and Relations. The spline curves are not included in the exercise.

Exercise 02565-09 **Appendix A**

Exercise 02565-09 **Appendix B**

Exercise 02565-09 **Appendix C**

Exercise 02565-09 **Appendix D**

Exercise 02565-09 Appendix E