Design Graphics Solid Modelling Exercise

(MECH289)DGSMEx5C.tex

December 30, 2005

1 Introduction

An important aspect of geometric modelling is the conversion of elements, say, details, from the design sketch into proper solid models that are used for all sorts of things like trial assembly to check that everything fits together properly and, ultimately, to produce instructions for automated manufacture, possibly on an NC lathe.

2 Selecting a Detail

Refer to Fig. 6 in the document **Design Graphics Project '06**. The "input shaft bearing sleeve" will be selected as an example. It is important to choose a short, descriptive name for all details that helps to identify them. Ambiguous names like "First Thing-a-ma-jig" are anathematic. The name selected should immediately draw attention to the hatched item containing two bearings that support the shaft of least diameter at upper left in the lower, section view in Fig. 6.

2.1 Creating a "Sweep" Section

The 2D linework of the element selected was isolated from the assembly shown in Fig. 6. The fastening cap screws and the cylindrical gear reducer upper housing are gone. Then key radial and axial dimensions are scaled off the drawing. These appear as the larger figures, everything is in mm. Then the coordinates (x, y) of 12 corner points of the section are similarly measured off the design sketch. The origin has been taken as the intersection of the bore centre line and the left end face. Note that the four slightly larger that 10^{ϕ} diameter mounting holes, that must provide clearance for $M10$ cap screws, have been omitted as have any necessary snap-ring grooves, undercuts, chamfers, fillets and rounds that are needed but not shown on the design sketch. All this is summarized in Fig. 1. The entry "163.75 OA" at the bottom refers to the overall axial length dimension, $41.25 + 97.5 + 25 = 163.75$. *Caution*, you must *never* dimension a complete circular feature by radius. That must always be called out as a diameter.

2.2 Creating a Solid Model

The information in the first sketch is plotted in 3D. In my case $(0, 50)$ -plotted as $(0, 50, 0)$- was the first point and then the others followed in clockwise sequence around the path of orthogonal and parallel lines of the sleeve wall section outline. With a single, almost anticlimactic, command, the wall section was swept full circle, through 32 positions, about the bore axis defined on the origin and any point $(x, 0, 0)$, $x > 0$. The result, with hidden lines removed is shown in Fig. 2.

3 Conclusion

The exercise described above assumes a wire-frame and "B-rep" (boundary representation) modelling environment. Many modern CAD systems make extensive, though not exclusive, use of so-called solid primitives, *i.e.*, rectangular blocks, cylinders of revolution, spheres, *etc.*, that may be controllably stretched and squashed. These may then be pierced with other primitives and assembled, using Boolean combinations, into the final, desired shape. Take a look at sections **4 Symmetry and Solid Modeling** on p.35 and **4.3 Creation and Manipulation** on pp.38-44 in the document **Fundamentals of Geometric Construction** on my website. As a thought exercise make neat freehand sketches of how you might model the piece shown in Fig. 2 using four cylinders that have each been axially pierced by a cylindrical hole and then appropriately assembling the four resulting "doughnuts" into final form.

(MECH289)DGSMEx5C.tex

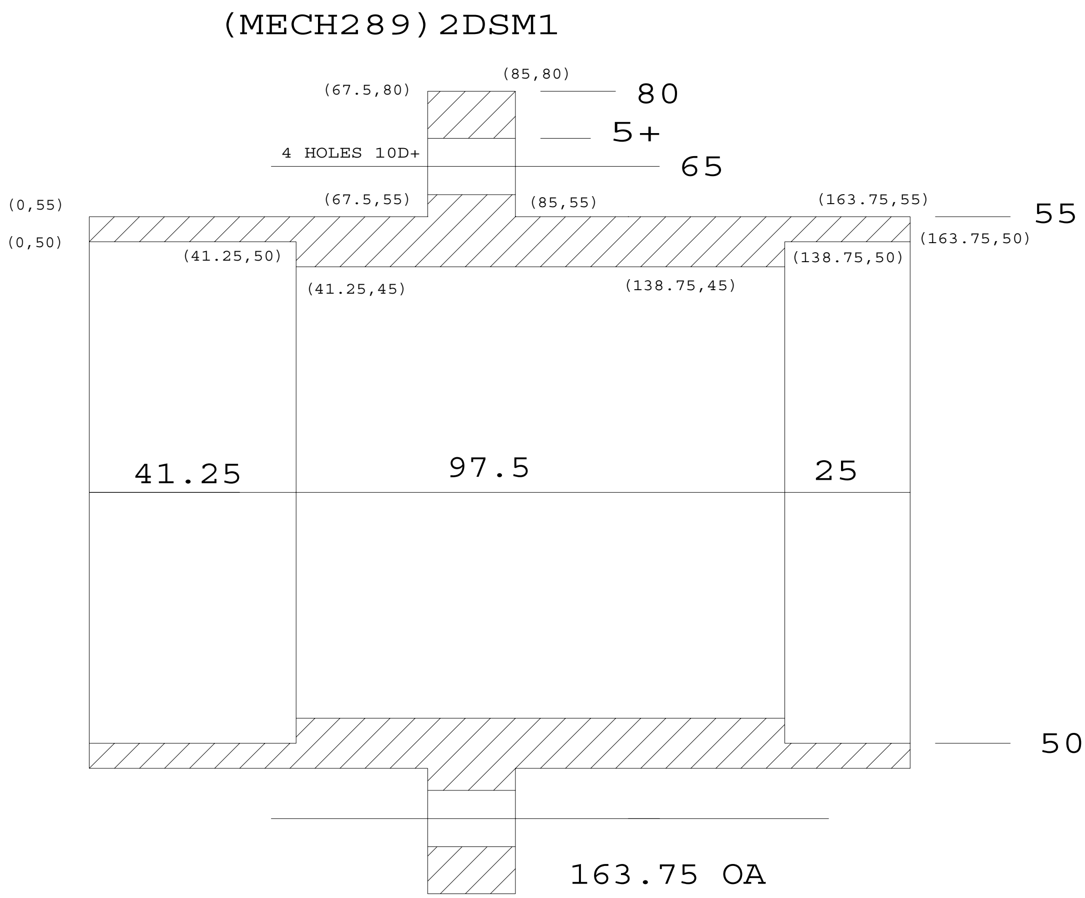

Figure 1: Sweep Section

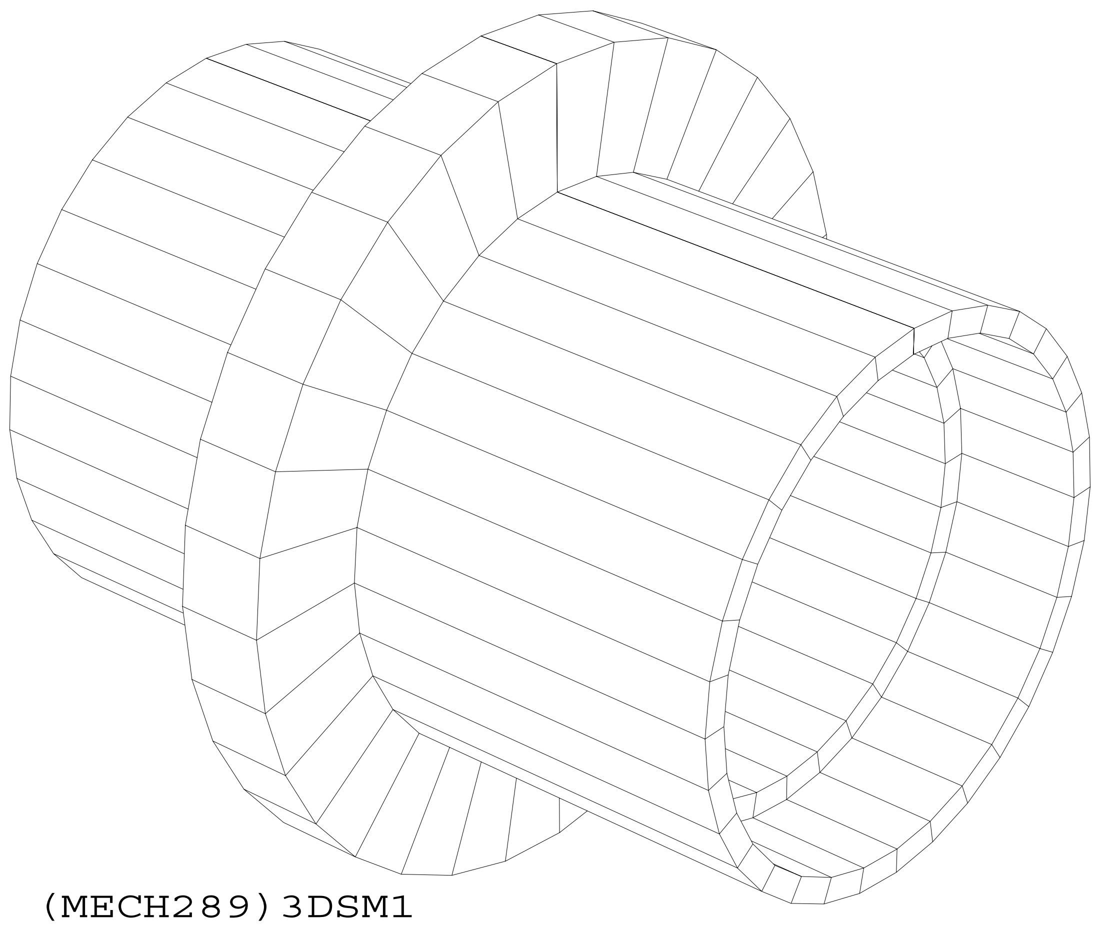

Figure 2: Primitive Solid Model