EDUCATION OF SOLID MODELING TO FRESHMAN OF INDUSTRIAL DESIGN COURSE

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Abstract — This paper discusses the Solid Modeling to Freshman of Industrial Design Course (SMFID) that has evolved in the Department of Design at Shizuoka University of Art and Culture (SUAC). The SMFID course, which consists of lectures and exercises, mainly aims to educate students in the theory and techniques needed to create solid models on a computer. In the lectures, students learn the thought process and the theory necessary to model solids, while in the exercises, they learn to model solids using three-dimensional computer-aided design (3D-CAD) software. SMFID enables students to learn to model solids efficiently in a short time with "learning assistance" which is composed of effective teaching materials developed by SUAC. The contents of SMFID and learning assistance are detailed in this paper.

Index Terms — CAD Education, Feature Base Solid Modeling, Geometric Modeling.

INTRODUCTION

With the increasing use of 3D-CAD, the importance of modeling solids is increasing in the design field. However, in the field of education, there is still much trial and error in teaching the modeling of solids as shown in Fig. 1. We report on SMFID developed at SUAC. SMFID consists of lectures on the thought processes and theory with regard to modeling and exercises using 3D-CAD. The exercises cover six topics; (1) sketching of figures, (2) modeling I: a solid defined only by the front view, (3) modeling II: a solid defined by two views, (4) modeling III: a solid defined by three views, (5) modeling IV: a solid which requires auxiliary views and (6) modeling V: a solid of revolution. To facilitate the efficient learning of these topics in the exercises, SUAC has developed SMFID learning assistance and has introduced it in the exercises. This paper concerns the contents of SMFID and learning assistance. Figure 2 shows a schedule of SMFID.

SKETCH

Students learn drawing and editing of two-dimensional (2D) figures, and dimensioning. In the drawing of 2D figures, elements such as a line, a circle, an arc, a polygon and a free-form curve are taught. In terms of the editing of 2D figures, move, copy and delete of figures are taught. Figure 2 shows examples of a sketch. Figure 2 shows the drawn figure and its dimensions. The sketch in Fig. 2 was produced by the Technical Drawing method.

Figure 3 shows the shape of a lock. Since the thickness of this shape is constant, this figure can be drawn using only the front view. On the sketch plane, the front view is drawn, and this front view is protruded by 38 mm to form the model shown in Fig. 4. There are three holes in this lock, which are defined by protruding circles with the front view outline of

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the lock. Solids, such as this lock, which are defined by only a front view can be modeled by protruding an outline drawn on the sketch plane. Accordingly, solids can be defined by the “sketch features”.

MODELING II

Figure 5 shows the shape of a link arm. To define this shape by projection drawing front and top views are needed. In the modeling, an outline of the front view is protruded to define the solid model shown in Fig.6, which is the base model. Then, holes are defined as shown in Fig.7. The cut part drawn in top view is defined and then the solid model shown in Fig.8 is formed. The modeling process proceeds in the order of Figs. 6, 7 and 8. Accordingly, the sketch feature in which the outline of the front view is protruded is defined as shown in Fig.6; the features of hole are defined as shown in Fig. 7, and the cut feature is defined as shown in Fig.8.

In modeling II, first a base model is formed by protruding a line drawing of a front view. Then, the line drawing of the top view (or side view) is manipulated and a final model is defined.

MODELING III

Figure 9 shows the shape of a cutting frame. The projection of this shape requires three views: front, top and side. In the modeling, we first focus on the front view. According to the rules of the projection method, a figure that most completely presents the original shape is drawn as the front view. Therefore, the solid model formed by protruding the front view figure defines the characteristics of the shape, as shown in Fig.10, and becomes the base model. Next, we focus on the side view. The side view shows an arch with 30 mm radius. Defining the arch in the base model produces the solid model shown in Fig. 11. Finally, we focus on the top view which shows a quadrangular pocket; one hole with a diameter of 38 mm and four holes with a diameter of 12 mm. Defining the pocket and the holes produces the solid model shown in Fig. 12. The modeling process proceeds in the order of Figs. 10, 11 and 12 (the order of Figs. 10, 12 and 11 is also acceptable). Accordingly, the sketch feature in which an outline of the front view is protruded is defined as shown in Fig. 10, the cut feature for arches is defined as shown in Fig 11, the cut feature for a pocket and the hole features are defined as shown in Fig. 12.

In modeling III, first a base model is formed by protruding a front view line drawing. Then, a line drawing of the top view (or side view) is produced. Finally, a line drawing of the side view (or top view) is produced and the model is defined.

MODELING IV

Figure 13 shows the shape of a clamp hook. The characteristic feature of this shape is a hook inclined by 30 degrees toward a plate with three holes. Therefore, this figure cannot be represented by three views (front, top and side), even though its shape is simple. An auxiliary view is
required, which is a key in this modeling method.
In modeling IV, first, a solid base model is defined as shown in Fig. 14. Next, the datum plane is drawn to define the position of the hook. A sketch drawn on the plane is protruded to define the solid model. By rounding the edge, the solid model in Fig. 15 is formed. The modeling process proceeds in this order. Accordingly, the sketch feature in which an outline of the front view is protruded is defined as shown in Fig. 14, the datum plane is defined, the sketch feature in which the hook is drawn is defined, and the round feature is defined as shown in Fig. 15. In modeling IV, first a base model is formed by protruding a front view line drawing. A line drawing of a top view (or side view) is manipulated. Then, a line drawing of a side view (or top view) is manipulated; modeling continues as in modeling III until this process is completed. Subsequently, a datum plane is defined in order to form the inclined part. Finally, the inclined part is formed on the basis of the datum plane and the model is defined.

**MODELING V**

Figure 16 shows the shape of a slinger. The characteristics of this shape are that it has axial symmetry and eight blades divided between two disks, four on each disk placed at equal intervals. In the modeling the characteristic of axial symmetry is defined as shown in Fig. 17. Modeled blades are shown in Figs. 18. By copying these blades and placing them at equal intervals around the axis of symmetry, the solid model in Fig. 19 is formed. The modeling process proceeds in this order. Accordingly, the sketch feature of a rotation is defined in Fig. 17, and sketch features in which outlines of blades are protruded are defined. Students study the process of modeling based on the axial symmetric characteristic in modeling V.

We have developed learning assistance for SMFID that enables students to master the modeling of solids efficiently in a short time. The learning assistance in word file format as shown in Fig. 20 is disclosed to promote free use by students. In a modeling exercise using learning assistance, all participating students were able to finish modeling within the time allocated for the exercise. We obtained feedback on the efficacy of teaching materials from questionnaires given to participating students. Figure 21 shows students performing the exercise. Figures 22 to 26 show shapes of the exercise and modeling test.
Using the solid model defined by 3D-CAD, the Physical model is produced by RP System.

**FIGURE. 20**
LEARNING ASSISTANCE

**FIGURE. 21**
CLASSROOM OF SMFID

**FIGURE. 22**
MODELING PROCESS

**FIGURE. 23**
MODELING PROCESS

**FIGURE. 24**
MODELING PROCESS

**FIGURE. 25**
MODELING PROCESS

**FIGURE. 26**
MODELING PROCESS
Using the solid model defined by 3D-CAD, the physical model is produced by rapid prototyping system. Figures 27 and 28 show the rapid prototyping system. Figure 29 shows samples of physical model. Figure 30, 31 and 32 show design practices of robotics toy.

**CONCLUSION**

This paper discusses the course on SMFID that has evolved in the Department of Design at SUAC. SMFID consists of lectures on the thought processes and theory with regard to modeling and exercises using 3D-CAD. The exercises cover six topics; (1) sketching of figures, (2) modeling I: a solid defined only by the front view, (3) modeling II: a solid defined by two views, (4) modeling III: a solid defined by three views, (5) modeling IV a solid which requires auxiliary views and (6) modeling V: a solid of revolution. GMOC enables students to learn to model solids efficiently in a short time with "learning assistance" which is composed of effective teaching materials developed by SUAC.