TEACHING SOLID MODELING WITH AUTOCAD

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Abstract
Solid modeling is the creation of an envisioned or existing part or assembly in digital solid form in 3D space. A digital solid is a 3D model consisting of vertices, edges, faces, and partially filled or entirely filled interior. It is a complete and unambiguous representation of the object in a precisely enclosed and filled volume in digital space. Solid models are used in many industries: engineering, manufacturing, marketing, entertainment, healthcare, etc. Modern manufacturing methods such as rapid prototyping, rapid manufacturing, and 3D printing use solid model geometries for production.

AutoCAD is a computer modeling software from Autodesk that can be used to create 2D and 3D geometric models of products. It is the most popular CADD (Computer-Aided Drafting and Design) software in the market worldwide. In fact it is the software commonly used in introducing CAD (Computer-Aided Drafting) to students at High Schools, Junior Colleges and in many Universities. AutoCAD is unquestionably the most popular software for 2D design drafting but is not so commonly used in solid modeling. However, with good mastering, AutoCAD solid modeling is highly versatile and can be competitive in small and mid-sized engineering design and consulting firms who lack financial resources for investment in rapidly changing parametric solid modelers.

This paper explores the teaching of solid modeling using AutoCAD. It presents a two phase procedural technique (planning and construction) that helps students to master solid modeling fundamentals. It is based on the author’s experience from years of teaching design drafting with AutoCAD as one of the main software. The planning phase can become purely a mental exercise when properly mastered so this phase is independent of software. The construction phase is software dependent. The technique combines visual and operational skills that students need in design graphic modeling. The application of the procedure is not limited to AutoCAD software because the author has successfully used the same approach to teach Solid Edge, Inventor, and Solidworks. The solid modeling technique is outlined; illustrations and samples of classroom presentations are provided.

Introduction
Solid modeling skills have become the vogue in the technical industry. Engineering and allied companies seek design drafters with these skills because solid modeling produces more accurate design graphic models faster and 2D drawings can be quickly generated from them. Traditionally, 2D drawings used to be created from instructions, isometric sketches, orthographic sketches, and isometric drawings. This practice is steadily vanishing due to the prevalence of 3D solid modeling software and design drafters who can use them.

CAD is an acronym that may be interpreted as Computer-Aided Drafting or Computer-Aided Design. The foundation for CAD goes back to the early 1940s when the mathematical description of curves was developed [1]. In 1963, Ivan Sutherland, formulated the theoretical basis for CAD graphics in his Ph.D. thesis titled “Sketch Pad”[1, 2]. He demonstrated that
graphic entities could be interactively picked on the computer screen using a light pen. This was the beginning of interactive computer graphics in engineering as it gave birth to the development of techniques for representing images in digital form. During the 1970s, commercial applications of CAD in 2D Drafting started. 2D CAD drawings consist of lines and arcs and are thus wireframe graphic models.

Initially, the CAD systems could only be run on large computer systems (mainframes and minicomputers) because of the memory size requirements for the computations, connections, and storage of the graphic entities. Advances in the development of computer hardware made larger memory and greater computational capacities possible, allowing CAD systems to migrate from mainframe computers to minicomputers, then workstations, desktops, and laptops. 3D wireframe models soon followed the 2D capabilities in the late 1960s. Though this was a noticeable improvement on 2D CAD systems, it lacked many practical attributes such as surfaces of physical objects. Hence 3D surface modeling technologies emerged in the early 1970s. Surface models are essentially wireframe models modified by covering their faces, but with the inside empty. Real objects are solids, though some may have cavities. Being able to create solid models thus attracted and still attracts a lot of attention. By the middle of 1970s, CAD systems with solid modeling capabilities have emerged [1, 2]. The development of 3D solid model capabilities brought engineering analysis of graphic models to the computer screen just like engineering drawings. Higher processing speeds, larger memories, and smaller sizes of computers at affordable prices have made solid modeling the mainstream in CAD applications today. Parametric and feature-based graphic modeling appears to be the advancing rapidly. Automatic drawing generation and dimension placement are also advancing at a fast pace.

Solid models are the most accurate mathematical description, the most realistic representations of objects, and the best visualization tools available. They can be associated with materials, texture or surface properties and volume. Solid models are sometimes easier to create than surface models. They are the most accurate 3D representation and most complete dimensionally. When rendered, solid models may appear realistic enough to be considered virtual models, minimizing the need for physical models. They are less costly to develop than physical models. Solid models accurately communicate design intent and can be used for engineering design analysis in finite element method (FEM). Fits and tolerances can be visualized in assemblies and clashes can be checked and eliminated. Accuracy and completeness of design is easy to achieve in a short time with solid models and they offer minimum error-checking efforts. This is very important in large and complex products. Manufacturing codes for computer numerical control (CNC) can be generated from solid models in computer-aided manufacture (CAM). Physical models can be generated directly from solid models in rapid prototyping, rapid manufacturing, and 3D printing. Solid models can be used directly by other departments such as marketing, purchasing, etc. 2D dimensioned drawings can be generated from solid models easily and with minimum efforts. For detail engineering analysis, information is required on material properties such as density, Young’s modulus, Poisson’s ratio, yield strength, ultimate strength, thermal conductivity, linear expansion coefficient, specific heat, etc. Geometric properties such as centroid, cross-sectional area, surface area, volume, section modulus, and second moment of area (area inertia) can be evaluated. Also physical properties like mass, weight, center of gravity, and second moment of mass (mass inertia) can be computed.
AutoCAD was formed by John Walker and his team of programmers in April 1982. AutoCAD, as a 2D drafting software, was first released by the company in December of the same year running on microcomputers or personal computers (PCs). It was derived from a program that was initiated in 1977 and released in 1979 as Interact CAD, called MicroCAD in Autodesk documents. By March of 1986, AutoCAD had become the most widely used software for design drafting worldwide, a position it holds today [3]. Parametric modeling in AutoCAD was introduced in 2010 version with geometric and dimensional constraints [4]. Since 2010, AutoCAD 360 has been available as mobile web-and-cloud based application. In March, 2014, AutoCAD 2015 was released, making it the 29th major releases by Autodesk for Windows. AutoCAD 2014 was the fourth consecutive year of releases for Mac computers.

Digital Solid Models in AutoCAD
A digital solid is a 3D model consisting of vertices, edges, faces, and partially filled or entirely filled interior. AutoCAD started as a 2D drafting software, but its 3D modeling capabilities have been enhanced greatly over the years. Today, its 3D work environment is probably more flexible than contemporary solid modelers but perhaps not as user-friendly or intuitive. AutoCAD uses constructive solid geometry (CSG) method to create complex solids from a combination of simple and compound solid forms. Simple solid forms are commonly called “primitives” and examples include spheres, boxes, cones, torus, wedges, pyramids, etc. CSG is a relatively simple method of constructing solid models. Simple and compound solid forms can be constructed from 2D profiles by extrusion or revolution. In an extrusion process a profile is swept in a path that may be a straight line or a curve. Revolution can be viewed as extrusion in a circular path.

Solid forms are combined using three Boolean operations of “Union”, “Subtract” and “Intersect”. In a union operation, two solid primitives are fused together into a single solid. In a subtract operation the overlapping portion of one solid form with another is removed from the second solid form. In an intersect operation, the common overlapping portion of two solid forms is retained while the remaining portions of the solid forms are discarded.

Generally 3D wireframe models can be displayed in hidden view mode but not in shaded mode. Only surface and solid models can be displayed in shaded modes. An isometric model is a 2D object because it is constructed on a 2D coordinate plane though it appears as a 3D image.

Part Modeling Technique
The solid modeling technique for a part developed and presented in this section has two phases of planning and construction. The planning involves mental exercises and freehand sketching and is independent of software. The construction phase requires mastering the use of software, so it is more mechanical.

Planning Phase
A flow chart for the planning phase is shown in Figure 1. With practice, the planning phase can be done mentally. However, for instructions of new students taking solid modeling course the first time, this phase should be religiously implemented. It is in this phase that 3D visualization skills are developed. Once mastered, solid modeling becomes easier and faster to do. This phase also help in developing critical thinking and brain storming skills that are vital in problem solving. A brief description of the steps in the planning phase follows.
1. Create freehand part isometric: The first task in the planning phase is to create a freehand isometric sketch of the part. This is necessary if part drawing or sketch is in orthographic views (orthoviews). If an isometric view (isoview) of part is available, then skip this step.

2. Decompose part into segments/features: The next task is to decompose the part into segments and features. So take a close look at part isometric and decide on the number of minimum segments for its creation. A segment must have a unique form. Unique forms of segments are based on shapes and sizes. Forms of different shapes are unique. Forms of the same shape but different sizes are unique. Forms of the same shape and size but different thicknesses or heights are unique. Further explanation of segments and features follows.

An engineered product can be decomposed into units, devices, and parts. Parts are also called components and are the basic building blocks of simple and complex products. Parts must be accurately modeled and documented in detail drawings. A part has a form which is a 3D geometric representation describing the visual appearance of an object. A part is considered in this modeling technique to consist of segments as shown in Fig. 2.

Fig. 1: Planning phase

Fig. 2: Part breakdown
A segment is a solid portion of a part that can be created from a single or compound profile. Segments are forms of specific shapes and sizes that combine to give a component its form. A segment may be a simple form or a compound form which is combination of simple forms. A segment can contain features that have special attributes and functions which usually modify segments. Form features may be considered as special segments. They include holes, counterbores, countersinks, spot faces, keyways, bosses, lugs, webs, lugs, threads, etc. Features can be constructed from simple forms though some CAD programs have built-in utilities for “inserting” some of them in models.

Each part or component must be carefully examined to identify its segments and features. To identify unique segments, focus on geometry and size. Portions of a component with the same shape or form but different sizes are different segments. For example, a stepped cylinder with a large base has two segments because the top portion diameter is different from that of the bottom. Each segment may be associated with a form primitive such as a box, cylinder, cone, etc. In summary, identify unique segments (geometry, size and features e.g. screws), and identify shapes associated with the segments and features of a component.

3. Create freehand isometric of segment/feature: The third task is to create a freehand isometric sketch of each segment and feature. This sketch helps in identifying form primitives such as boxes, cylinders, cones, etc. that are needed for the modeling. Also special features like screws, bosses, webs, etc. can be identified easily on the segment.

4. Choose solid transformation technique: The next task is to choose a solid transformation technique for the segments and features. The option is either extrusion or revolution. Any segment can be created by extrusion but it may not be the best option. Revolution is good for segments with rounded outer contour (cylinder, cone, frustum, sphere, etc.); especially those with relatively more complex internal features. An axis of revolution is required for a revolution operation.

5. Identify shapes and construction planes: After choosing a transformation technique, the shapes and the construction planes (Front, Top, Right) that the segments or features are on should be identified. This step is very important as a lot of time can be wasted rotating objects in 3D frame if the correct plane for the shape is not used to create its profile. To minimize modeling efforts, choose a construction plane with minimum shapes. Shapes with arcs and circles are priority for construction planes. Then look for planes that have the best shape description of the segments and features.

6. Create outline views: The last task in the planning phase is to create a freehand sketch of the outline view of each segment on its construction plane. Outline views represent the external contour of segments and discard hidden features. They result in a collection of shapes that are needed for the creation of profiles from which solid segments are made.

It must be emphasized again that the planning phase is independent of modeling software. So the approach above can be used to teach any solid modeling software. The implementation of the
profile sketches from this phase will depend on the specifics of the software being used for modeling.

**Construction Phase**

Fig. 3 represents the construction phase in creating solid segments and features of a part using AutoCAD software. The construction phase of Fig. 3 is somewhat restricted, because it focuses mainly on AutoCAD software. However, it can be adapted, perhaps easily, for other solid modeling software. The tasks in this phase are computer based. A brief description follows.

1. Create a segment shape sketch: The first task is to create a computer sketch of the shape from the last step of the planning phase based on the outline views developed. This involves choosing the right construction plane (identified in step 5 of planning phase) and creating the shape sketch on it. For each segment, construct the sketch on the appropriate construction plane.

2. Convert sketch into shape: The second task is to convert the sketch into a shape which is a closed transverse on a 2D surface. This is done by ensuring that the line and arc segments in a sketch are connected and not overlapping. This results in a shape that can be converted to a profile. Check to ensure that the shape is properly closed; line and arc segments are joined and not overlapping.
3. Convert shape into profile: The third task is to convert a shape to a profile which is a properly closed and dimensioned single-entity shape. A profile may be a cross-sectional view or longitudinal view of a segment. Some tools in AutoCAD such as rectangle, circle, and ellipses generate single-entity shapes automatically. By default, these shapes are profiles once sizes are added. When a shape has straight sides, the pline tool can be used to create it. An advantage with the pline tool is that a closed loop will be created only if all the vertices lie on the same plane. This ensures that the resulting profile produces a proper solid. For complicated shapes consisting of line and arc segments, the shapes should be converted to profiles using the region tool. Check to ensure that a profile is a closed shape. Then add dimensions or size constraints to the shape. Also add geometric constraints to the shape. It is better to handle profiles of segments one at a time. If revolution is chosen for profile transformation, then ensure that there is an axis of revolution for the transformation. Note that when AutoCAD Classic is used, constraints are implicitly added to shapes during creation. This is because of the requirement to explicitly provide size and direction information for line and arc segments during shape creation. If AutoCAD 3D Modeling module is used, then size and geometric constraints can be added explicitly to sketches as it is done in other popular solid modelers.

My experience indicates that many students have some challenges understanding the differences between a sketch and a profile. This usually leads to initial frustration in solid modeling skills development. So consider Fig. 4 below which gives three representations of a triangle. The first figure is an open sketch because the vertices of the triangle are not connected. It consists of three dis-joined line segments. This sketch will give a disconnected surface model if extruded in AutoCAD. It is not good for a solid model.

![Fig. 4: Triangle](image)

The second sketch is a closed figure giving a true triangular shape. The line segments in this shape may be joined or not. If the line segments are not joined and the shape is extruded in AutoCAD, a surface model will result. If the model is shaded, it will appear as a solid but the inside is completely empty! This does not represent the design intent of having a solid, so it is unacceptable. If the line segments are joined, the sketch becomes a shape with one entity like the third sketch but without the dimensions. The third sketch is closed and the line segments are joined, it is a one-entity shape. Also it has dimensions of length and angles which make the shape to be of a fixed size. It is a profile which
unambiguously defines the design intent of a desired shape and size. If a profile is extruded or revolved, a solid object is obtained. Summarizing Fig. 4, a sketch may be open or closed. When a sketch is closed it becomes a shape. When a shape is joined and constrained, it becomes a profile. A profile cannot be formed from a shape or closed sketch with overlapping graphic elements. An open sketch will not convert to a profile. A profile must be a closed, non-overlapping single-entity graphic shape that is partially or fully constrained geometrically and dimensionally. A new design is created by changing any of the constraints of a profile.

4. Transform profile into solid segment: Using the solid transformation technique identified in step 4 of the planning phase, transform the profile into a solid segment. Profiles are transformed into solids by extrusion or revolution. In an extrusion process a profile is translated or moved in a specified direction. AutoCAD can perform four types of extrusion: straight, path, sweep, and loft extrusions. In a straight extrusion process, a profile is swept in a direction normal to it. In path extrusion, the profile is swept in a path that may be a straight line, 2D or 3D curve but perpendicular to the profile plane. A sweep extrusion is an extrusion along a straight or curved part that is not perpendicular to the profile plane. It is more flexible than a straight or path extrusion. In straight, path, and sweep extrusions, the profile has a constant cross-section (shape and size of profile is fixed). A loft extrusion is like sweeping, except that the profile shape and size can vary along the extrusion path.

Revolution is another technique for constructing solid segments. This can be viewed as extrusion in a circular path. Revolution is good only for segments with circular, spherical or conical contours. Revolution profiles are longitudinal views of segments and a half section view is the profile required for a revolution operation. When parts have complicated internal features or contours and they have rounded exteriors, then revolution is the best technique of construction. It is a faster technique than extrusion when rounded objects have several hidden features. An axis of revolution must be identified and created for revolution operation.

Some assessment must be done for each segment or feature to determine whether it should be constructed by extrusion or revolution. However, extrusion is more versatile and should be used when revolution is not suitable. Virtually any solid form can be created by a series of extrusion processes. Cross-sectional view profiles are used mainly in extrusion operation while longitudinal section profiles are used in revolution operations. Use extrusion for simple profiles of polygons, non-symmetrical, and non-rounded segments. Use revolution for externally rounded segments that have complicated internal contours.

5. Assemble solid segments and features into part: After all solid segments and features have been created, they must be assembled together to form the part. This is done by first stacking the segments and features together in their proper locations. The stacking creates a single part of two or more separate segments and features. Since the part is a single entity, the separate components are joined by performing Boolean operations of union, intersect, and subtract. Union and intersect operations can be done in any order, but the
subtract operation should be done after union and or intersect are performed. This will avoid creating blind holes instead of through ones in a part.

Illustrations
Two examples are considered in this section to demonstrate the application of the technique enunciated above. The first example is Fig. 5 which shows the three orthoviews of a bracket. Fig. 6, the second example, shows the isometric view of another bracket design.

Example 1
Fig. 7a is the result of step 1 in planning phase because Fig. 5 shows orthoviews of the part. This sketch helps in visualizing the part to be modeled. Fig. 7b and Fig. 7c result from applying steps 2 and 3 in the planning phase. The component of Fig. 7a is broken into two segments as shown in these figures. Note that a second option is a rectangular base segment and top angled segment which are not shown. Encourage the students to generate as many options as possible to help them understand that some options minimize computer storage spaces. For Step 4, it is obvious from the forms of the segments in Fig. 7b and Fig. 7c that extrusion (specifically, straight extrusion) will do the transformation of the segment profiles to solids. For Step 5 of the planning
phase, the profile plane of Fig. 7b is the front plane and the profile plane of Fig. 7c is chosen to be the top, though the front or right plane could be used. The top plane is the default in AutoCAD 3D space, so no switching of view direction is necessary. Fig. 7d shows the sketch of the profile for Fig. 7b on the front plane while Fig. 7e shows the sketch of the profile for Fig. 7c on the top plane. This completes the planning phase for Example 1.

![Fig. 8: Example 1 segments](image)

For Step 1 in the construction phase for Example 1, the 2D shape of Fig. 8a was created with a “line” tool from Fig. 7d. Constraints were applied by using the “ortho” mode in AutoCAD, so a shape was created instead of a sketch. Hence Steps 1 and 2 were combined in the execution. The shape of Fig. 8a consists of dis-jointed but touching line segments. It is made up of five line segments without gaps. It is included to demonstrate a common error beginners in AutoCAD solid modeling make.
For Step 3 in the construction phase, the profile for Fig. 7b is shown in Fig. 8b as the 2D figure. It is a single-entity closed transverse or shape made up of continuous line segments. The profile was created from the shape of Fig. 8a by using the “pline” tool to re-trace the shape of the figure. If the shape was copied and “regioned”, a profile would be created also.

Step 4 of the construction phase was performed by extruding the shape of Fig. 8a and the profile of Fig. 8b. The shaded figure of Fig. 8a resulted from extruding the shape in front of it. It is a surface, not a solid model. A shape was extruded, not a profile. The shaded figure of Fig. 8b resulted from extruding the profile in front of it. It is a solid model because a profile was extruded.

The above processes were repeated for the second segment of Fig. 7c. Fig. 8c shows a 2D shape using the “line” tool while Fig. 8d shows a 2D profile using the “rectangle” tool to create the shape sketch of Fig. 7e. The corresponding extruded figures are shown behind the 2D figures. Again, a surface model is obtained in Fig. 8c while a solid model is obtained in Fig. 8d.

Step 5 of the construction phase has two separate tasks. The first task is “staking” of segments, and the second task is applying appropriate Boolean operations. Fig. 9a shows the “staking” together of the shaded figures of Fig. 8a and Fig. 8c. Similarly, Fig. 9b shows the “staking” together of the shaded figures of Fig. 8b and Fig. 8d. These figures consist of two separate segments. Fig. 9c shows the result of applying the Boolean operation of “Union” to Fig. 9a. No change is observed because the two surface models still remain separate. Fig. 9d shows the result of applying the Boolean operation of “Union” to Fig. 9b. A change is observed because the two solid segments have combined into a single solid model.

**Example 2**
The isoview of the part is shown in Fig. 6, so Step 1 of the planning phase is not necessary. Fig 10a to Fig. 10e shows the results of planning Steps 2 and 3. Five segments are identified and sketched in isoviews. Extrusion was chosen for transformation of segment profiles into solids for all five segments. Application of planning Steps 5 and 6 yielded the sketches of the segment profiles shown in Fig 11a to Fig. 11e. Figs. 11a, 11b, 11c, and 11e are on the front plane while Fig. 11d is on the right plane. This concludes the planning phase.

![Fig. 10: Part segments](image-url)
Fig. 11: Shapes and profile planes

Fig. 12: Example 2 segments

Fig. 13: Example 2 assembly
Steps 1 to 3 of the construction phase were performed separately for each of the five segments of Example 2. Fig. 12a, Fig. 12c, Fig. 12e, Fig. 12g, and Fig. 13c show these profiles. Note that Fig. 12c shows two overlapping profiles. This is allowed in AutoCAD so long as the profiles are independently. In this case the rectangle was created with the “rectangle” tool and the circle was created with the “circle” tool. The approach avoided the alternative which requires a U-shape (rectangle without the top horizontal line segment) to be created with the “line” tool, creating a semi-circular arc on the U-shape or creating a circle on the U-shape, and trimming the circle portion inside the U-shape. Then the “region” tool is used to combine the semi-circle with the U-shape sketch. This would mean creating a compound profile from the shape depicted in Fig. 11b. A compound profile is created from a combination of two or more shapes. The use of compound profiles may speed up the modeling process in some complicated parts.

In Step 4, extrusion was used for the profiles created to obtain Fig. 12b, Fig. 12d, Fig. 12f, Fig. 12h, and Fig. 13d. Initially, two separate solids were obtained in Fig. 12d but the Boolean operation of “Union” was applied so that a single solid resulted.

In Step 5 of the construction phase, the first four segments were stacked and Fig. 13a shows the outcome. Then the Boolean “Union” operation was applied to Fig. 13a to obtain Fig. 13b. The four solid segments were combined into one. Next, the segment of Fig. 13d was positioned on Fig. 13b as shown in Fig. 13e. Fig. 13f is the desired part which was obtained by applying the Boolean operation “Subtract” to Fig. 13e. That is, Fig. 13d was subtracted from Fig. 13b.

**Classroom Exercises**

Fig. 14a and Fig. 14b are examples of classroom exercises. The sketches were drawn on the whiteboard while taking the students through the planning phases for the parts. The part of Fig. 14a was decomposed into two segments. The top segment is a frustum so a revolution transformation was used to create the segment solid. The profile of the base needed to be “regioned” because of the straight tangent edges. The part of Fig. 14b was decomposed into three segments that were extruded. Fig. 15a shows the solid model of Fig. 14a while fig. 15b shows the solid model of Fig. 14b.

![Fig. 14a: Class exercise 1](image1.png)  ![Fig. 14b: Class exercise 2](image2.png)
Students’ Feedback
The technique described above has been used several times in DRFT 134: Mechanical Drafting. This course emphasizes documentation of mechanical parts with views generated from solid models. It was taught last in spring 2014 with 10 students initially enrolled, but two students withdrew. During the first week of spring 2015, the author administered a questionnaire to the 8 students who took this course about their experience in solid modeling. Students were requested to rate ten statements with 1: strongly disagree; 2: disagree; 3: not sure, 4: agree; and 5: strongly agree. Six of the 8 students responded to the questionnaire. Table 1 shows the questionnaire and the average rating of responses to the statements in the questionnaire is given in the last column. No conventional statistical analysis apart from the arithmetic mean was attempted because of the sample size limitation. Since the minimum average response is 4 (Statement #8), one may conclude that the technique presented helped the students generally. Statement #10 is of note because it deals with transferability of skills to other solid modeling software. The average of 4.67 to this statement clearly demonstrates that the technique helps in perhaps reducing the learning curve for these other software and or in understanding them better. (We offer Solid Edge and NX in other courses in the Program). Based on the overall responses in Table 1, it seems safe to say that using AutoCAD as a platform for introducing students to solid modeling helps them to better master solid modeling skills.

Since taking DRFT 134, some of the students have learnt to use Solid Edge and some are learning NX this semester (Spring 2015). In a follow-up discussion in week 7 of this current semester, most of the students expressed the sentiment that learning AutoCAD solid modeling has helped them to reduce the learning curve in Solid Edge and NX. Also, most of the students who took DRFT 134 earlier continue to make AutoCAD their preferred solid modeling software in other 3D projects. Whether such preference holds in the workplace is a different matter altogether.
<table>
<thead>
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<th>Question #</th>
<th>Statement</th>
<th>Rating</th>
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<tr>
<td>1</td>
<td>Breaking down a component into segments of basic forms and sizes helped to improve my understanding of solid modeling principles.</td>
<td>4.17</td>
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<td>2</td>
<td>The teaching approach of using segment diagramming helped me to better understand solid modeling.</td>
<td>4.50</td>
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<td>3</td>
<td>The use of segment iso-sketch helped me to easily identify the profile construction plane.</td>
<td>4.33</td>
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<tr>
<td>4</td>
<td>The use of outline orthoviews was helpful in determining the minimum number of segment profiles.</td>
<td>4.50</td>
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<tr>
<td>5</td>
<td>The concept of a profile was explained satisfactorily.</td>
<td>4.33</td>
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<tr>
<td>6</td>
<td>Understanding the concept of a profile reduces mistakes and time in modeling.</td>
<td>4.17</td>
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<tr>
<td>7</td>
<td>Identifying the profile plane is a very important step in solid modeling.</td>
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<td>8</td>
<td>It is important to relate profiles to the correct construction planes.</td>
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<td>9</td>
<td>Graphic entities that default to profiles speed up solid modeling process.</td>
<td>4.00</td>
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<tr>
<td>10</td>
<td>The fundamental principles of solid modeling learnt in DRFT 134 have helped me in the use of other solid modeling software.</td>
<td>4.67</td>
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**Conclusion**

AutoCAD is perhaps not the first choice for most solid modelers today because of the availability of other packages with more advanced parametric modeling features. Also, most of these other packages use macros to automate the “Union, Subtract, and Intersect” operations in AutoCAD; thereby making solid modeling relatively faster and or easier for users. But AutoCAD solid modeling is still performed in small and large firms who do business with companies who have AutoCAD as their only software. This happens to be more prevalent with small individual design drafting companies that cannot afford multiplicity of solid modeling packages. Hence learning AutoCAD solid modeling is a valuable skillset for these companies. Haven had the opportunity of teaching solid modeling with Inventor (previous employment), Solid Edge, and Solid Works; I think a better understanding of solid modeling operations is gained in AutoCAD solid modeling.

A two phase structured approach for the solid modeling of a part with AutoCAD is presented. A flow chart for the planning phase is shown in Figure 1. The planning phase is when a part is
decomposed into segments and features. This phase helps students develop 3D visualization, critical thinking, and problem-solving skills. When mastered, the planning phase can be done mentally and it is independent of software.

The second phase is the construction phase in which the sketched shape(s) of the first phase are converted into profiles that are transformed into solid segments using AutoCAD software. Shapes and profiles must be created on the appropriate planes: Top, Front, Right planes. Fig. 3 shows a flow chart for this phase. The construction phase is software dependent so the procedure would need some modification when the software is not AutoCAD. It should be noted that newer versions of AutoCAD tend to have more enhanced solid modeling capabilities, including parametric modeling. Thus for some other solid modelers, the modification would be minimal.

The technique outlined combines visual and operational skills that students need in design graphic modeling. The use of part isoview, segment freehand isoview, and outline views make the technique graphic intensive, aiding the development of strong visualization skills. The procedural approach encourages systematic development of operational skills. When no other software is available for solid modeling (consider colleges in many third world countries), AutoCAD solid modeling is a good way to prepare students in solid modeling skillset.

References

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