Creating Wings in SolidWorks

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1 Introduction

The goal of this tutorial is to introduce a method of creating aircraft wings in the 3D computer aided design program `SolidWorks`. This tutorial is appropriate for users who have limited to no experience with `SolidWorks`. Supplementary guides and tutorials can be found on the MEAM.Design Wiki page at `http://alliance.seas.upenn.edu/ medesign/wiki`.

2 Defining the Airfoil

There are two straightforward methods of defining the airfoil profile.

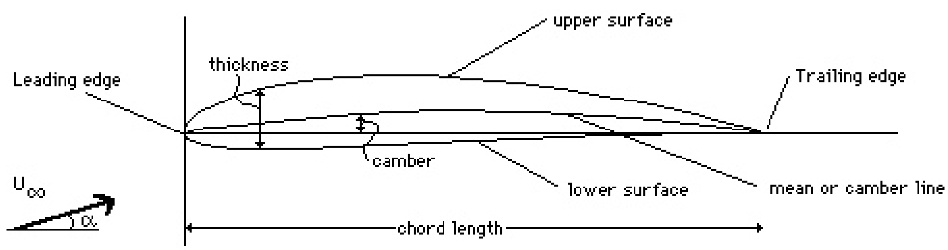

Figure 1: Sketch of Airfoil Geometry, from Kroo "Airfoil Geometry" [1]

Online Profile Generator The first method involves using a web-based NACA 4-digit series profile generator [2]. The airfoil profile is determined by three parameters, which correspond to the four digits of the NACA airfoil. The first digit corresponds to the degree of camber, the second to the position of the camber, and the third and forth to the thickness (expressed as a percentage of chord length).

1. This tool can be found at `http://www.ppart.de/aerodynamics/profiles/NACA4.html`.[1] Toggle the sliders until you achieve the desired airfoil.

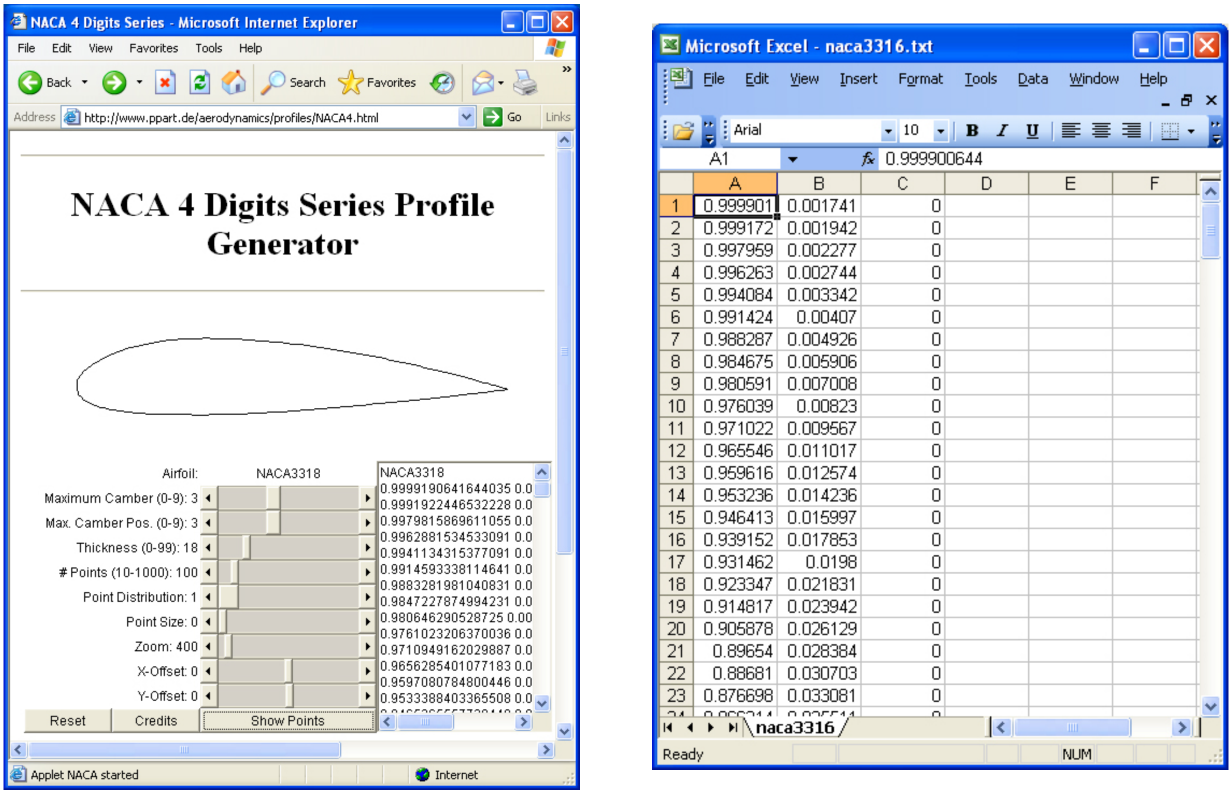

Figure 2: Web-based profile generator and `Excel` file containing airfoil profile data points

2. Copy the coordinate points and paste into a new `Microsoft Excel` document.
3. Depending on the version of `Excel` being used, sometimes both the x and y data points show up in the same column when copied from the Profile Generator table. To fix this problem so that column A corresponds to x-axis points only, and column B corresponds to y-axis points only, click on *Data* (in the toolbar) $\rightarrow$ *Text to Columns*. Within the *Convert Text to Columns Wizard*, select *delimited* for original data type and then *tab* and *space* as the delimiters. Now the x and y data should be in two different columns.

Using the Equation For a 4-Digit NACA Airfoil The second method involves using NACA formulas to define the shape of the airfoil.

[1] Profile generators for NACA 5 digits series and ellipses can also be found at `http://www.ppart.de/aerodynamics/profiles/NACA5.html` and `.../profiles/Ellipse.html`, respectively.

1. To define the shape of a symmetrical 4-digit NACA airfoil (i.e. NACA 00xx):

$$y = c\frac{t}{0.20}\left[0.2969\left(\frac{x}{c}\right)^{\frac{1}{2}} - 0.1260\left(\frac{x}{c}\right) - 0.3517\left(\frac{x}{c}\right)^{2} + 0.2843\left(\frac{x}{c}\right)^{3} - 0.1015\left(\frac{x}{c}\right)^{4}\right]$$

2. For a cambered 4-digit airfoil, we must first define the mean camber line using the following piecewise equation:

$$y = \begin{cases} m\frac{x}{p^2}\left(2p - \frac{x}{c}\right) & 0 \leq x < pc \\ m\frac{c-x}{(1-p)^2}\left(1 + \frac{x}{c} - 2p\right) & pc \leq x \leq c \end{cases}$$

   Once the mean camber line is defined, the coordinates of the upper and lower airfoil surface (x_U,y_U) and (x_L,y_L) are given by:

$$x_U = x - y\sin\theta, \qquad y_U = y_c + y\cos\theta$$

$$x_L = x + y\sin\theta, \qquad y_L = y_c - y\cos\theta$$

3 Drawing the Airfoil Curve

1. Create a new `SolidWorks` part document
2. From the toolbar select *Tools* → *Options* → *Document Properties* (tab) *to units.* Select the desired unit system. Since we have data points for a unit-length airfoil, the chord of the airfoil will be one of whatever unit the document is in. The chord length can then be specified by scaling the profile after it is inserted.
3. From the toolbar select *Insert* → *Curve* → *Curve Through XYZ Points...*
4. A window titled *Curve File* will appear. Within this window, select *Browse...*. Change *Files of type:* from *Curves (*.sldcrv)* to *Text Files (*.txt)* and then select the file containing the airfoil profile points.
5. Once the profile points are in the *Curve File* window, select *OK*. A curve is now inserted into the `SolidWorks` model view.
6. The curve we have just inserted shows up as *Curve1* in the FeatureManager design tree on the left hand side of the screen. However, in order to create solid geometry, we must first convert this curve into a sketch.

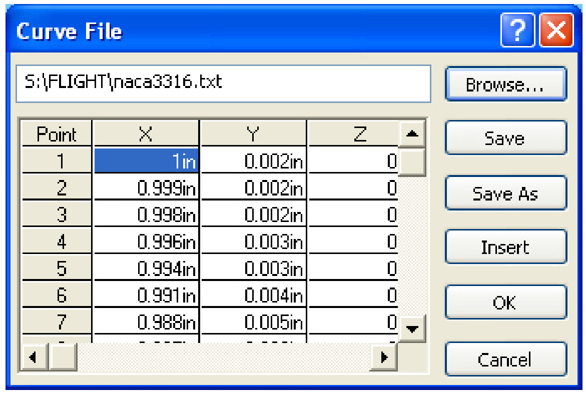

Figure 3: Curve file window

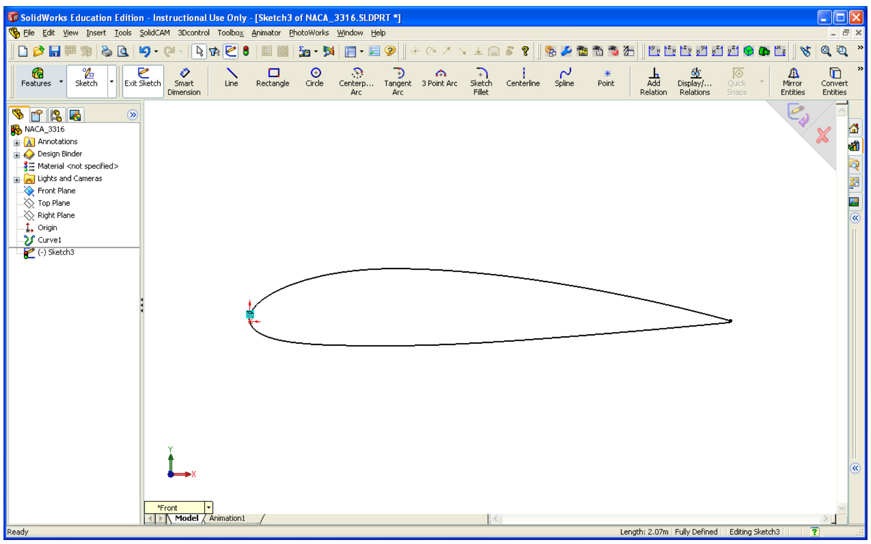

Figure 4: After creating a sketch and converting curve entities, the airfoil appears with black lines in the `SolidWorks` model view.

7. To create a sketch that is based on the curve, first insert a new sketch by selecting the *Sketch* tool from the CommandManager toolbar. `SolidWorks` should by default insert the airfoil curve on the front plane, so select the front plane as the sketch plane.

8. Now we can create a sketch defined by the airfoil curve by first selecting the curve and then selecting *Convert Entities*, which can be found in the CommandManager toolbar. The curve should appear black in color, which indicates a fully defined sketch. We are now ready to use this sketch to create 3D geometry.

4 Creating 3D Geometry

This section details methods of creating 3D wings from the 2D airfoil profile.

Extruded Feature Extruding the sketch profile will create a simple wing. This feature will not provide any curvature to the wing, nor will it allow the wing to be tapered gradually towards the tip.

1. From the CommandManager toolbar, select *Extruded Boss/Base*.
2. If your sketch is not already selected, you will be asked to select an existing sketch to use for the feature. Select the sketch of the airfoil profile.
3. From the FeatureManager, change the depth of the extrusion to achieve the desired span.
4. Click the green check mark to execute the feature, and save the `.SDLPRT` file.

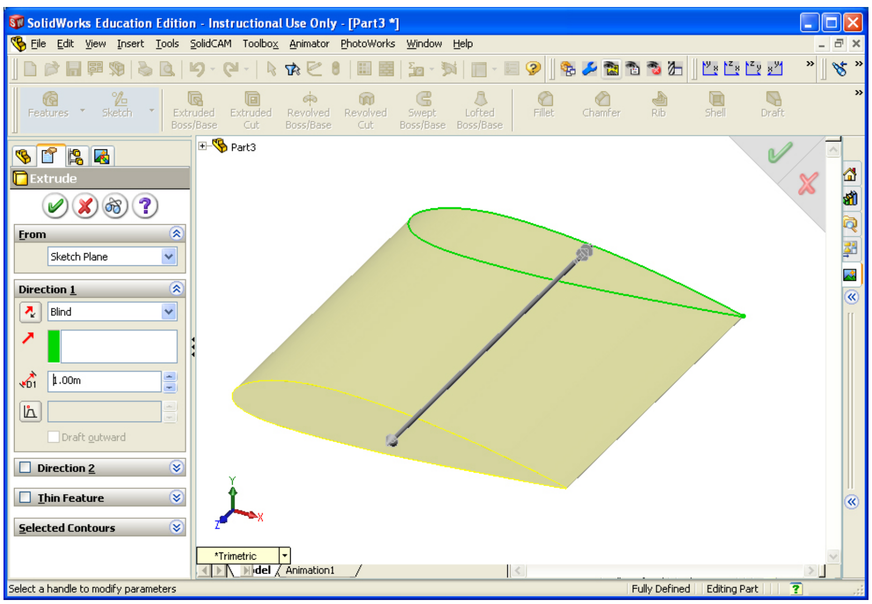

Figure 5: An extruded feature

Swept Feature Sweeping the sketch profile through a path provides a higher degree of flexibility in comparison to the extrusion feature.

1. To begin, we must first insert a path line through which the profile will be swept. Once again, a curve can be inserted from a `.txt` file containing data points in the same manner in which the airfoil profile was inserted, but it is also possible to simply insert a new sketch and draw a line or a spline.
2. Make sure you have exited the sketch containing the profile, and then select *Sketch* and then *Sketch* again from the CommandManager toolbar.

3. The panel on the left hand side of the screen will ask you to select a plane on which to create a sketch. With the airfoil profile already oriented on the front plane, select the top plane (if the planes are not displayed, open up the part tree).

4. Orient the view such that you are normal to the top plane. This can be accomplished be either opening the *Standard Views* drop-down menu in the *View* toolbar, or by pressing the spacebar on the keyboard and selecting *top*.

5. Select either the *line* or *spline* tool from the CommandManager toolbar, and create the desired sweep path next to the airfoil profile, avoiding any automatic relations `SolidWorks` may try to insert.

6. Select *Add Relation* from the CommandManager toolbar and select the endpoint of the path line and the profile sketch.

7. Within the *Add Relations* side panel, define the relation as *Pierce*. Select the green check mark to insert the relation. This relation is necessary to create a swept feature.

8. Exit the sketch and select *Swept Boss/Base* from the feature option in the CommandManager toolbar.

9. The *Sweep* sidebar will ask you to select a sweep profile and a sweep path. Select the airfoil profile sketch as the sweep profile, and the line/spline sketch as the sweep path.

10. Notice that the span and shape of the wing depends on the sweep path sketch that was previously inserted. If you would like to go back and change the length or path of the sweep path, first select the green check mark to execute the sweep feature. An item entitled *Sweep1* appears in the FeatureManager design tree. Expand the sweep feature by selecting the plus symbol next to it, right-click on the sketch that contains the sweep path, and select *Edit Sketch*. Any changes made to the sweep path will automatically rebuild in the sweep feature after you exit the sketch.

11. The sweep feature can also be edited by right-clicking on it in the design tree. Some options that may be of interest can be found in the *Orientation/twist type* drop-down menu. For example, if you are creating prop blades, you may want to have the airfoil profile twist along the path.

Lofted Feature Lastly, we can employ the loft feature to create wings that have longitudinal twist, bend, and a non-constant chord length. [2]

[2] Additionally, we could even use lofts to create wings with non-constant airfoil geometry.

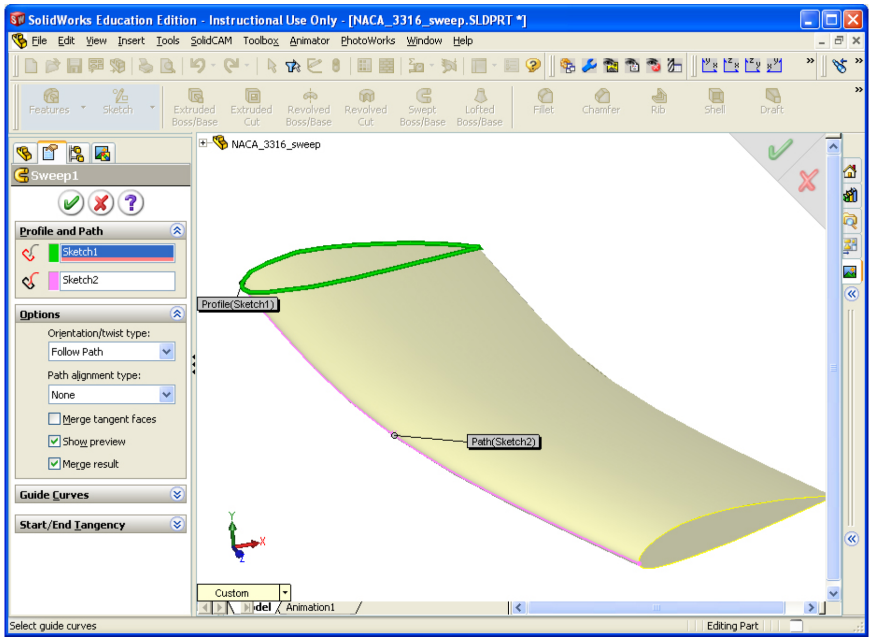

Figure 6: A swept feature with a curved profile path

1. To use the loft feature, we must insert a new plane as reference geometry. Before dong this, first right-click the front plane in the FeatureManager design tree, and select *show*.
2. Now go to *Insert* → *Reference Geometry* → *Plane*. Select the front plane from the model view, and set the distance between this plane and the new plane to be the desired span of the wing. Select the green check mark to insert the plane. The reference geometry plane should appear in the model view and the design tree.
3. Insert a new sketch on the reference plane (probably named *Plane1*).
4. Orient the view normal to the reference plane, select the airfoil profile that is sketched on the front plane, and convert entities. This tool copies the airfoil curve into the sketch on the reference plane.
5. In the case of wanting to create a wing with a non-constant chord length (*i.e.* a 'taper') we can scale the airfoil sketch on the reference plane. To do this, select *Tools* → *Sketch Tools* → *Scale*.
6. If the sketch isn't already selected, select the airfoil curve on the reference plane. *Spline2* will appear in the *Entities to Scale* box in the side panel.
7. Under *Parameters*, you must choose a point to *scale about*. You can click anywhere in the sketch to define a point to scale about. Finally, specify a *scale factor*, and select the green check mark to execute.

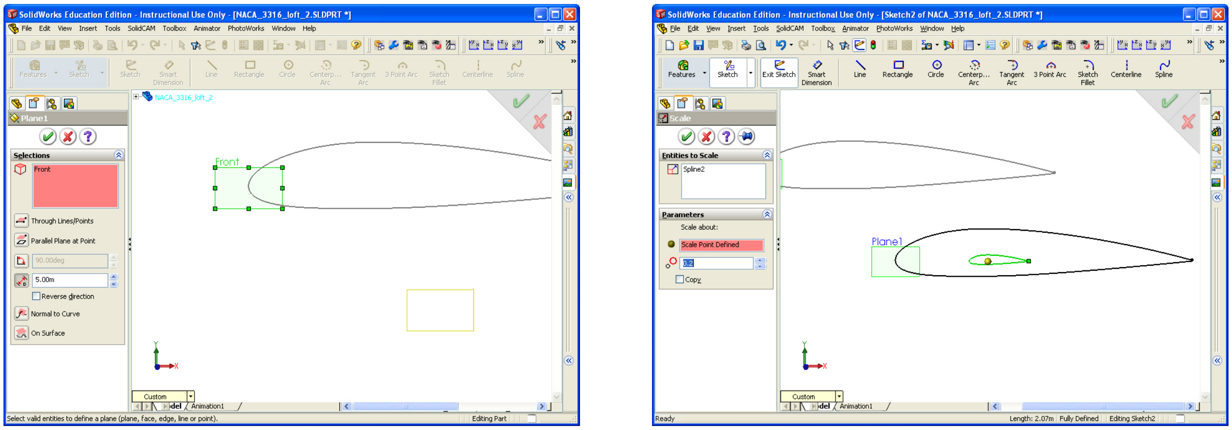

Figure 7: Inserting a reference plane and scaling a sketch

8. Insert a loft by selecting *Lofted Boss/Base* from the CommandManager toolbar.
9. In order to create a loft, we must select at least two profiles. Select the airfoil profile sketch on the front plane and the scaled-down profile sketch on the reference plane.
10. The loft generated by `SolidWorks` is most likely not the loft that is desired. There are many different ways in which we can modify the loft feature. First, you will notice a teal-blue circle that appears on each of the selected profiles. These circles mark the points that are synchronized between the profiles. These markers can be clicked-and-dragged around to modify the synchronization between the profiles.

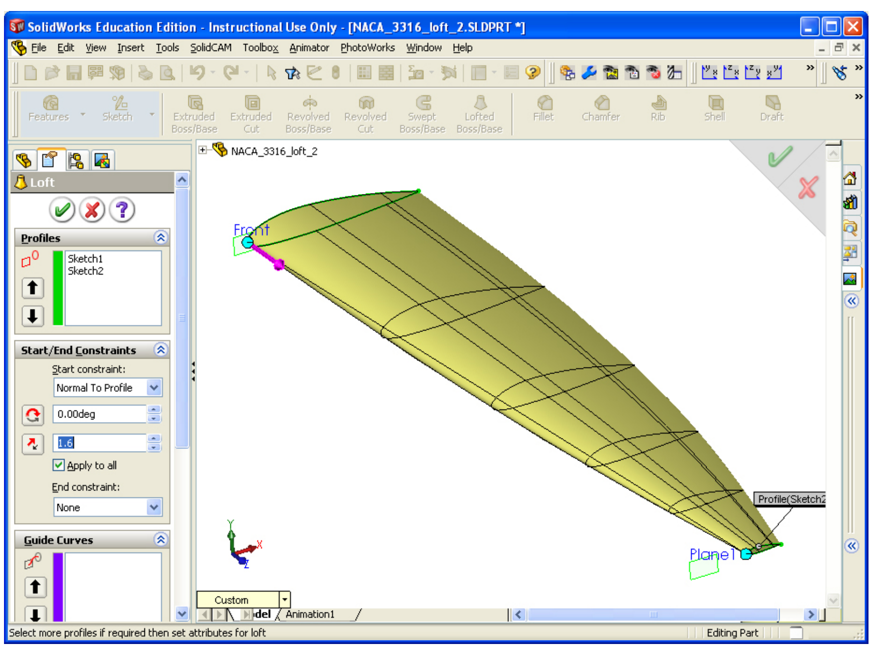

Figure 8: A lofted feature with a scaled-down end profile

11. The start/end constraints can also be modified to create the differently shaped wings. Within the *Start/End Constraints* box in the Loft editor, change *Start Constraint* from *None* to *Normal to Profile*. This option causes the loft faces to start normal to the sketch profile. If the start constraint for the airfoil profile sketch is *Normal to Profile* at the root, and there is no constraint on the scaled-down profile at the tip of the wing, the resulting loft will be parabolic in shape.

12. Guide curves can also be used for lofted features in a way that is similar to how they are used for swept features. In lofts, guide curves specify the geometry between the profiles.

13. Although there are other options that can be used in loft features, the last option I will mention is using multiple scaled and/or rotated airfoil profiles in a single loft. In this way, the wing geometry can be highly specified.

5 Measurements, Mass Properties, Rendering

Once the wing is created, `SolidWorks` is capable of telling you some useful information about the wing, such as the dimensions, surface area, and moments about various axes (once a material is assigned). Finally, realistic material renderings can be produced using the PhotoWorks tool.

Measurements `SolidWorks` is capable of determining the dimensions and surface area of the wing.

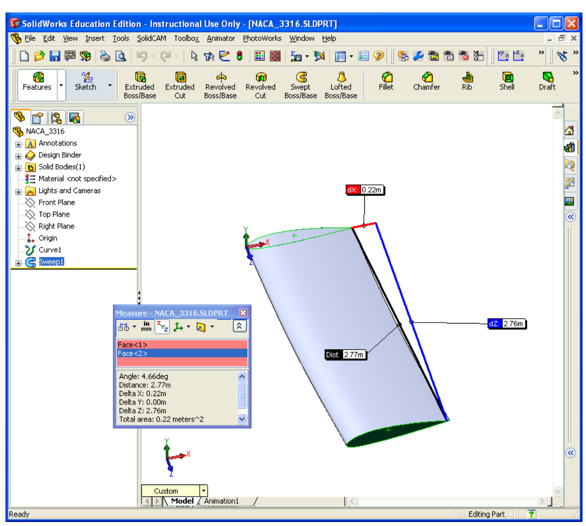

Figure 9: Making measurements

1. Go to *Tools* → *Measure*.
2. To determine the span of the wing select the flat face at both the root and tip end of the wing. `SolidWorks` will display the distance between the two faces, as well as a breakdown of the distance in *(x,y,z)* components.
3. The surface area of the wing can be determined by selecting the curved surface of the geometry.

Mass Properties Additionally, one can assign a material to the geometry to obtain information on the mass properties of the wing, such as total mass, center of mass, and the moments of inertia about various axes.

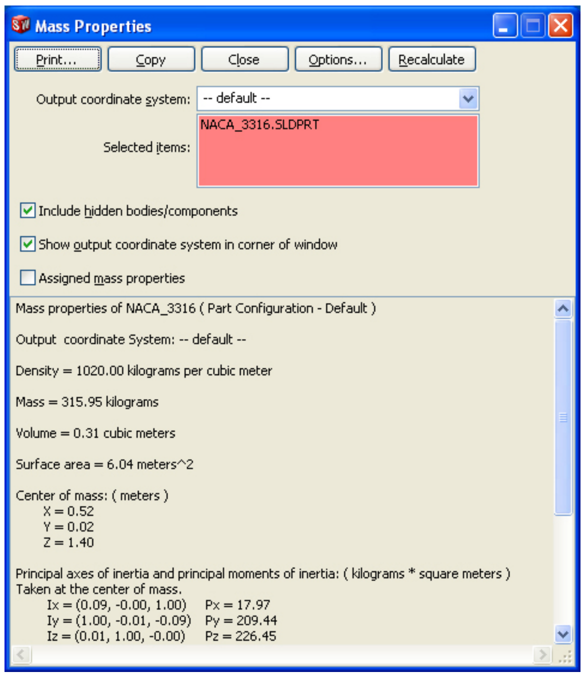

Figure 10: Mass properties window

1. A material must be assigned to the wing before `SolidWorks` can provide information about its mass properties. To do this, right-click in the FeatureManager design tree where it says *Material not specified*, and select *Edit Material*.
2. If the wing is to be printed using the `Dimension Elite` 3-D printer, select *ABS* (under plastics) as the material. If the wing is going to be fabricated from a series of laser-cut acrylic segments, select *Acrylic (Medium-high impact)*.
3. Notice that at the bottom of the Materials Editor side panel there is a box titled *Physical Properties*. Properties such as elastic modulus, Poisson's ratio, shear

modulus, density, etc. are provided by `SolidWorks`, assuming that a material was assigned.

4. Select the green check mark to save the material to the wing.
5. Now, select *Tools* → *Mass Properties*. Various mass properties, such as center of mass (relative to the part origin), and the moments of inertia about various axes are computed.

Rendering Lastly, one can prepare attractive `SolidWorks` renderings of the wing geometry using the PhotoWorks feature, a rendering tool that is built in to the `SolidWorks` program.

1. On the right-hand side of the screen, select the PhotoWorks Items button to display the PhotoWorks task pane.
2. Select a renderable material from the task pane, and drag the material name onto the `SolidWorks` part.
3. To add a background scene and lighting, do the same for any of the various Studios. Lights can also be inserted manually by dragging a light into the window containing the part.

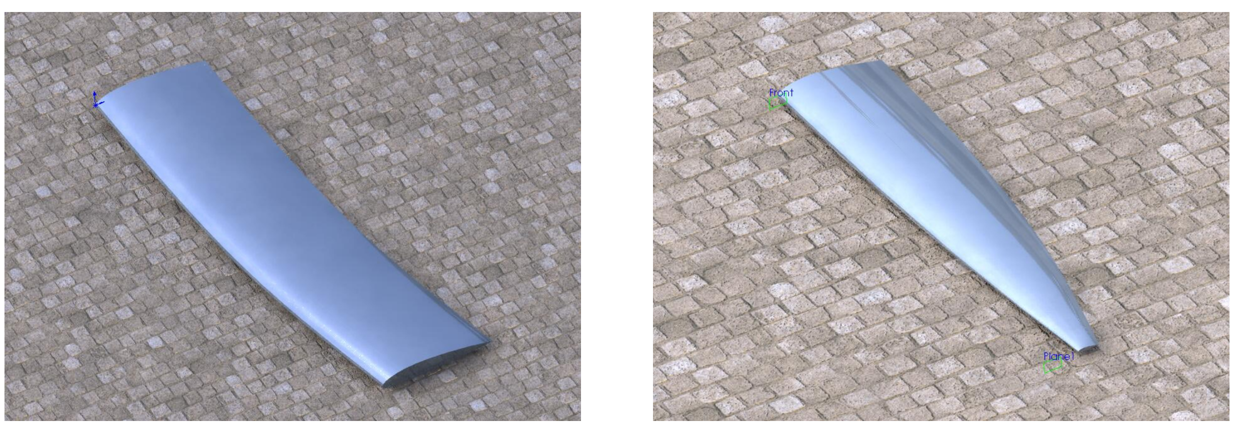

Figure 11: Renderings of the swept feature (left) and lofted feature (right). The material shown is polished aluminum and the studio is *Daytime.*

References

[1] Kroo, "Airfoil Geometry", http://adg.stanford.edu/aa241/airfoils/airfoilgeometry.html (2006)

[2] Trapp, Zores, “NACA 4 Digits Series Profile Generator”, http://www.ppart.de/aerodynamics/profiles/NACA4.html (2009)

[3] Ladson, Brooks, Hill, ”Computer Program To Obtain Ordinates for NACA Airfoils”, NASA Technical Memorandum 4741, National Aeronautics and Space Administration, (1996)