Lesson 16 Helical Sweeps and Annotations
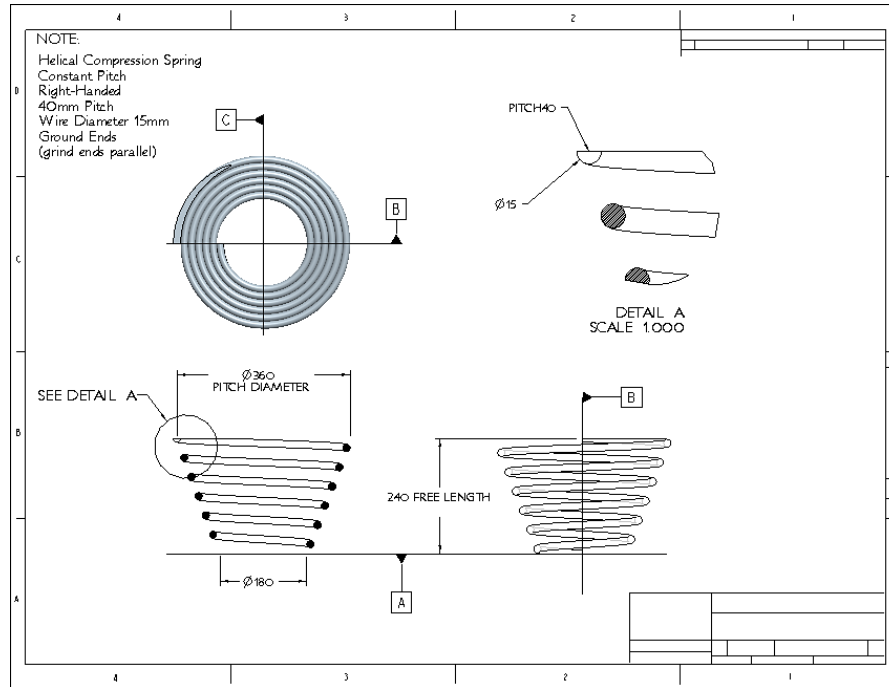

Figure 16.1 Helical Compression Spring Drawing

OBJECTIVES

- Create a **helical compression spring** with a **Helical Sweep**
- Use sweeps to create **hooks** on **extension springs**
- Create **plain ground** or **hook ends** on a spring
- Create **3D Notes** and **Annotation Features**

REFERENCES AND RESOURCES

For **Resources** go to **www.cad-resources.com >** click on the PTC Creo Parametric 3.0 Book cover

- Lesson Lecture
- Book Projects PDF
- Project Lectures
- Quick Reference Card
- Configuration Options

Helical Sweeps and Annotations

A **helical sweep** (Fig. 16.1) is created by sweeping a section along a helical *trajectory*. The trajectory is defined by both the *profile* of the *surface of revolution* (which defines the distance from the section origin of the helical feature to its *axis of revolution*) and the *pitch* (the distance between coils). The trajectory and the surface of revolution are construction tools and do not appear in the resulting geometry.

   Annotation features are data features that you can use to manage the model annotation including surface finish, geometric tolerances, notes, and so on. **Model notes** are pieces of text, which can contain links (URL's) to World Wide Web pages, which you can attach to objects in Creo. Model notes, increase the amount of information that you can attach to any entity in your model.

Helical Sweeps

The Helical Sweep command is available (Fig. 16.2) for both solid and surface features. You can define the helical sweep feature using the following options:

- **Keep constant section** The pitch is constant
- **Vary section** The pitch is variable and defined by a graph
- **Thru axis of revolution** The section lies in a plane that passes through the axis of revolution
- **Normal To trajectory** The section is oriented normal to the trajectory
- **Normal to projection** The section is oriented normal to the projection
- **Use right handed rule** The trajectory is defined by the right-hand rule
- **Use left handed rule** The trajectory is defined by the left-hand rule
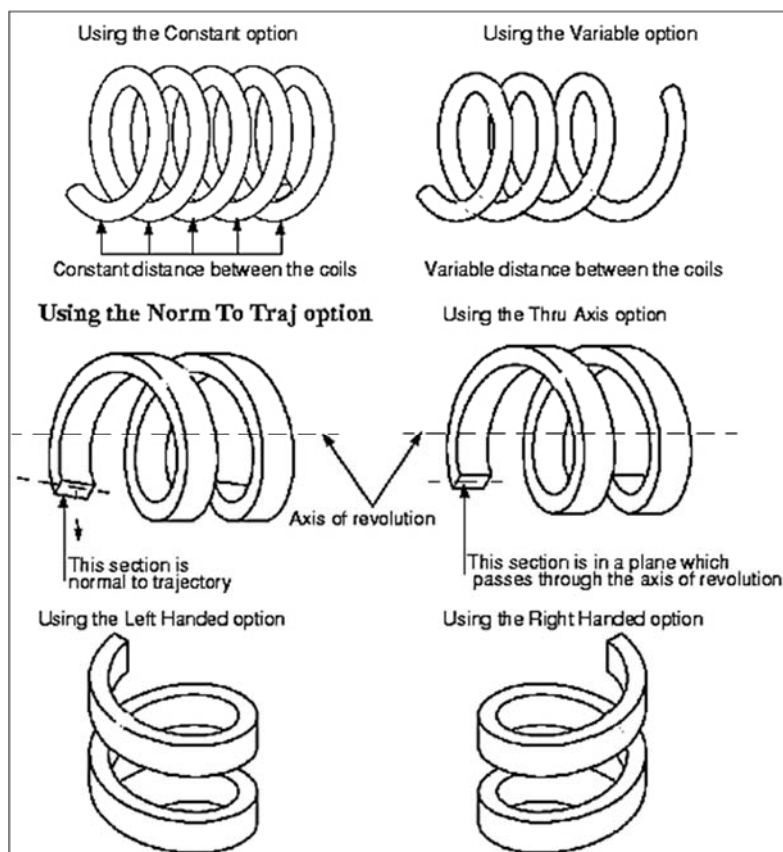
Figure 16.2 Helical Sweeps

Annotations

Model notes are text strings, which can be placed flat to the screen (view plane) in model space (Fig. 16.3). Note(s) can be attached to any entity in your model. When you attach a note to an entity, that entity is considered the parent of the note. If you delete the parent entity, all child note(s) are deleted with it. You can also allocate a URL to each model note. You can use model notes to communicate with members of your workgroup as to how to review or use a model, explain how you approached or solved a design problem when modeling, and explain changes that you have made to the features of a model over time.

Annotation features can also be notes, but also include: symbols, surface finish, geometric tolerance, set datum tags, ordinate baseline dimensions, driven dimension, and so on.
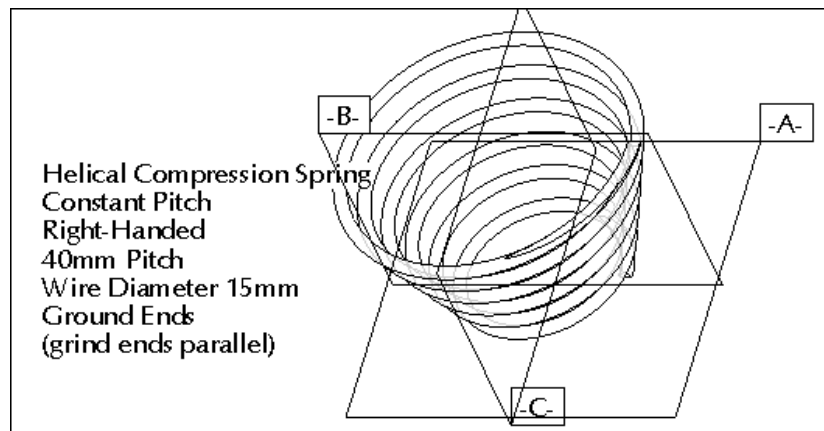

Helical Compression Spring
Constant Pitch
Right-Handed
40mm Pitch
Wire Diameter 15mm
Ground Ends
(grind ends parallel)

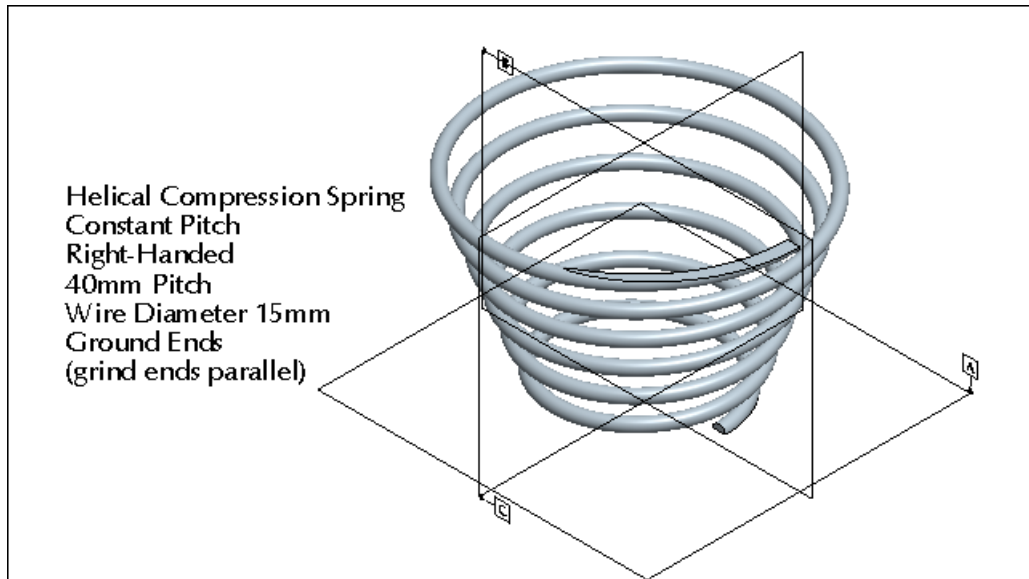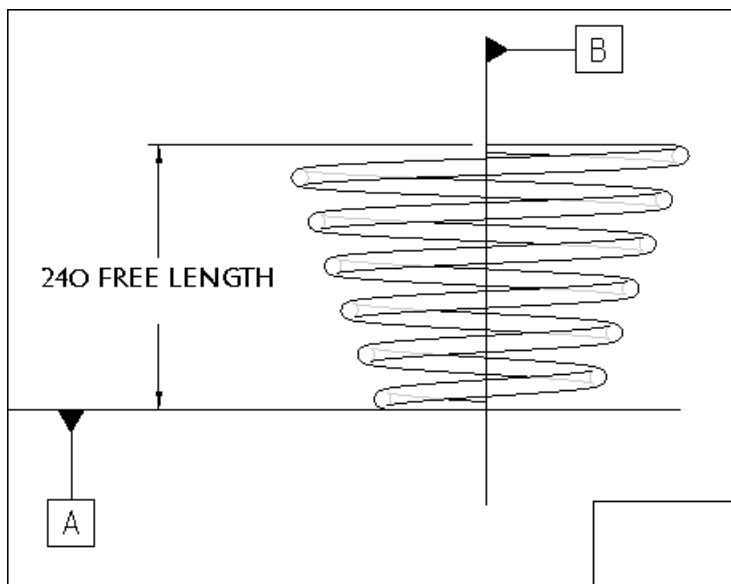**Figure 16.3** Model Notes

Lesson 16 STEPS

Figure 16.4(a) Helical Compression Spring with Datum Planes and Model Note

Helical Compression Spring

Springs [Fig. 16.4(a)] and other helical features are created with the Helical Sweep command. A helical sweep is created by sweeping a *section* along a *trajectory* that lies in the *surface of revolution:* The trajectory is defined by both the *profile* of the surface of revolution and the distance between coils. The model for this lesson is a *constant-pitch right-handed helical compression spring with ground ends, a pitch of **40 mm**, and a wire diameter of **15 mm** [Figs. 16.4(b-e)].
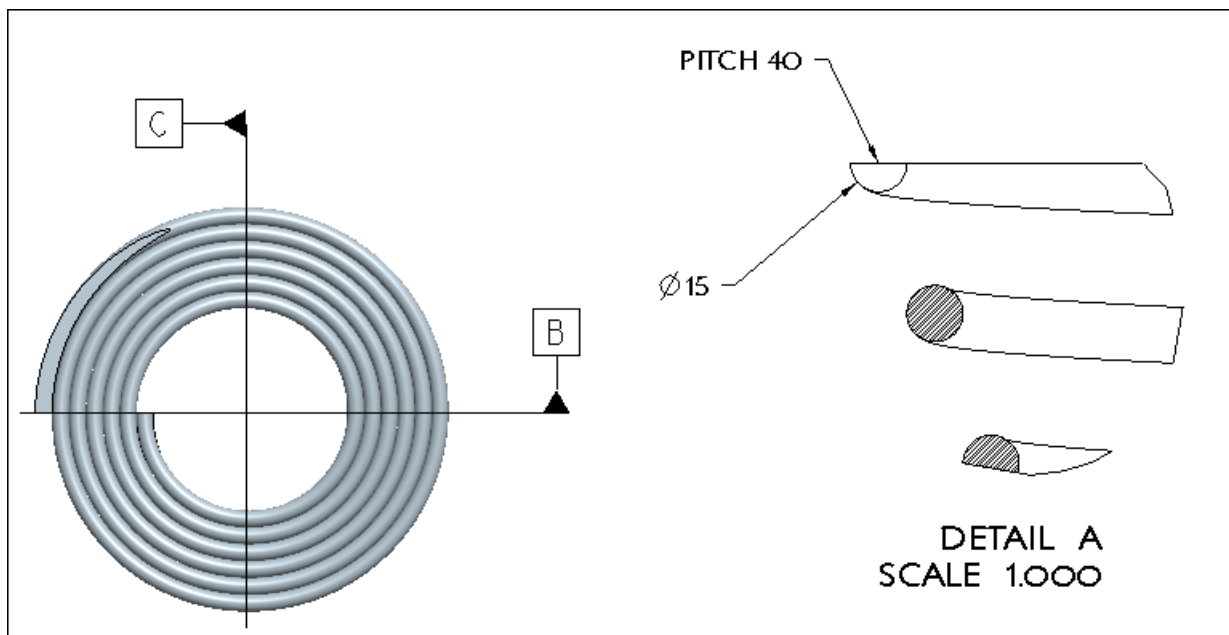
Figure 16.4(b) Helical Compression Spring Drawing: DETAIL A

   687
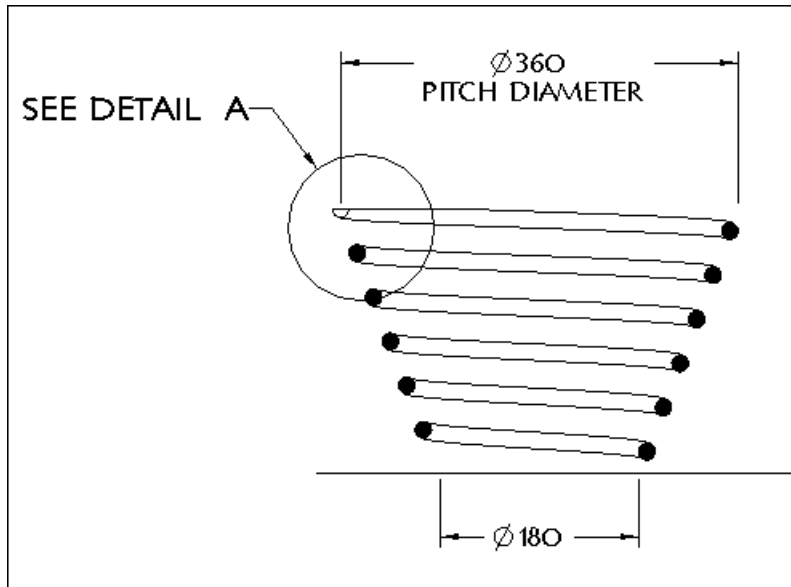
Figure 16.4(c) Helical Compression Spring Drawing, Section

Figure 16.4(d) FREE LENGTH **240**

NOTE:
Helical Compression Spring
Constant Pitch
Right-Handed
40mm Pitch
Wire Diameter 15mm
Ground Ends
(grind ends parallel)

Figure 16.4(e) 3D Model Note

Start a new part. Click: ☐ **Create a new model >** [◉ ☐ Part] **>** Name **helical_compression_spring >** [✓ Use default template] **> OK > File > Prepare > Model Properties** (set the material and units):

- **Material** = ss.mtl
- **Units** = millimeter Newton Second

Set Datum [◄▌◄] and **Rename** the default datum planes and coordinate system:

- Datum TOP = **A**
- Datum FRONT = **B**
- Datum RIGHT = **C**
- Coordinate System = **CSO**

Click: [Sweep] > [▼] > [≋ Helical Sweep] **Helical Sweep > References** tab **> Define** [Fig. 16.5(a)] **>** select datum **B > Sketch >** [icon] **> RMB > Axis of Revolution >** add a vertical centerline along datum **C > MMB > RMB > Line Chain >** starting on the edge of datum create the angled line **> MMB > MMB** [Fig. 16.5(b)]
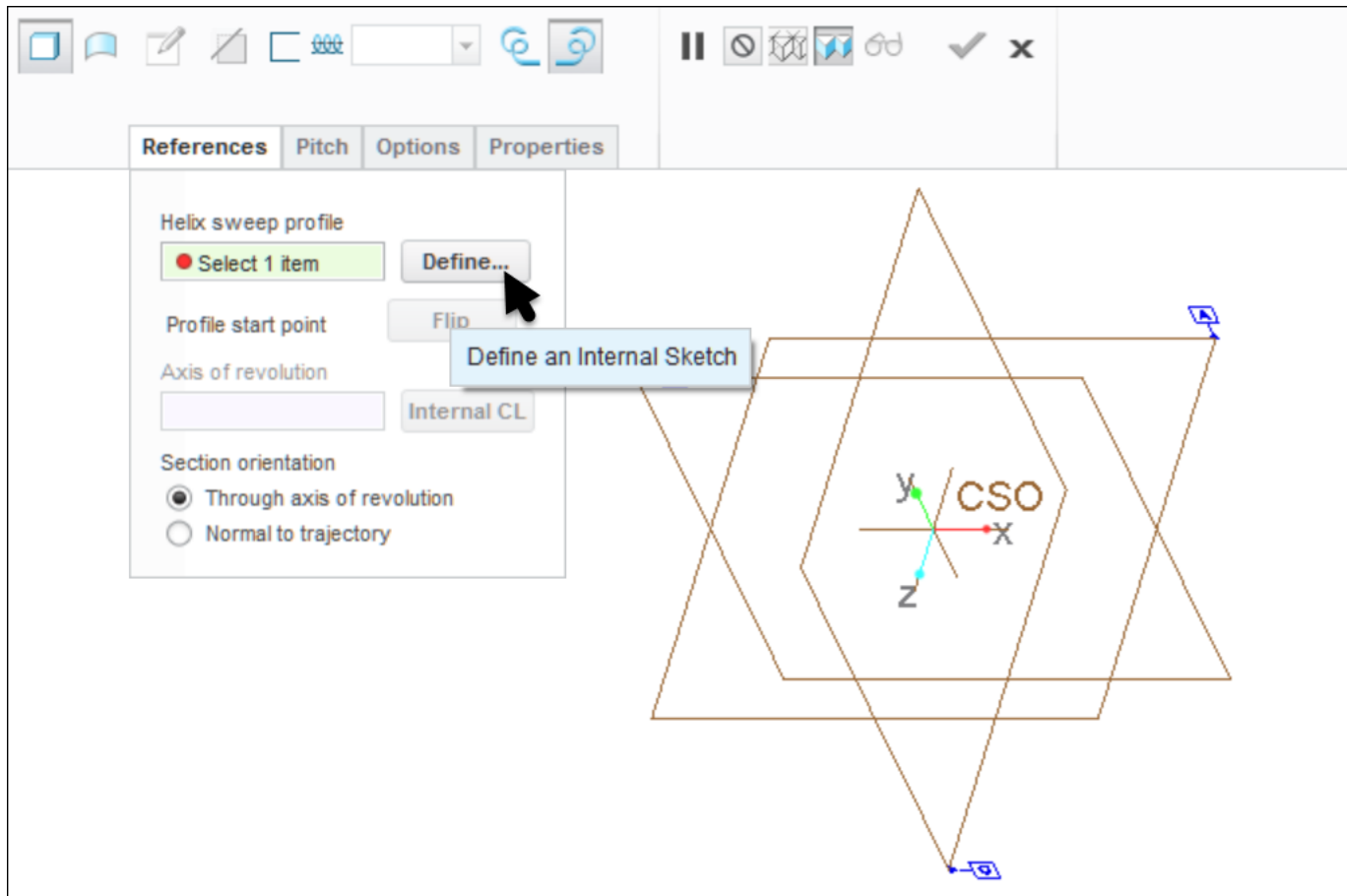
Figures 16.5(a) Helical Sweep Tool
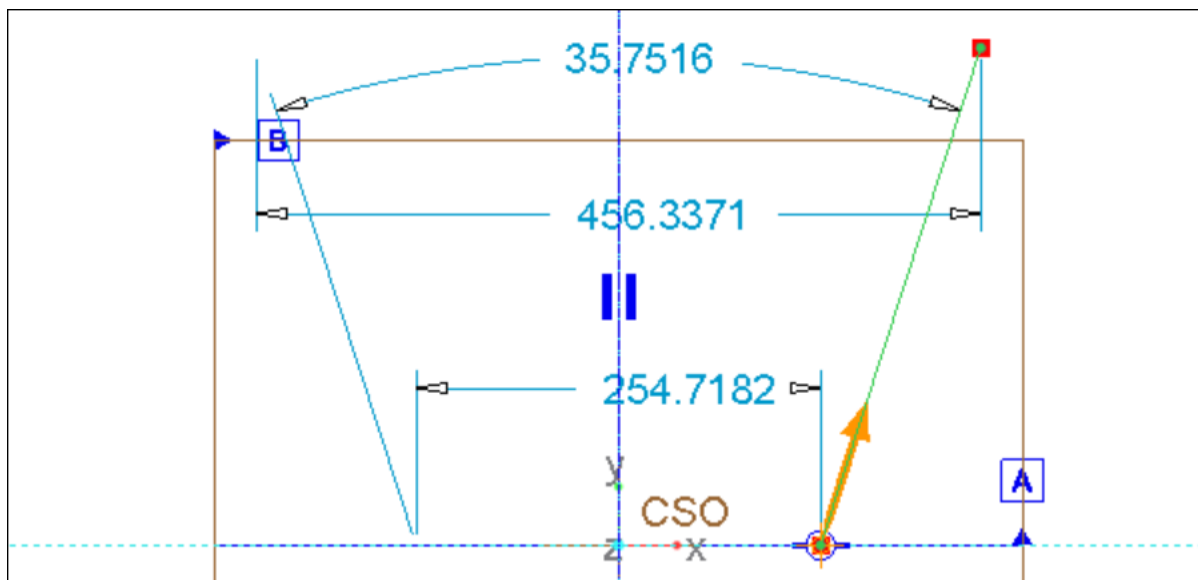
Figure 16.5(b) Helix Sweep Profile Sketch. Note the Start Arrow Direction.

Press: **RMB > Dimension >** add the height (free length) dimension **> MMB > MMB** [Fig. 16.5(c)] **>** window-in the dimensions **>** press **RMB > Modify** change the values to the design sizes [Fig. 16.5(d)] **> ✔**
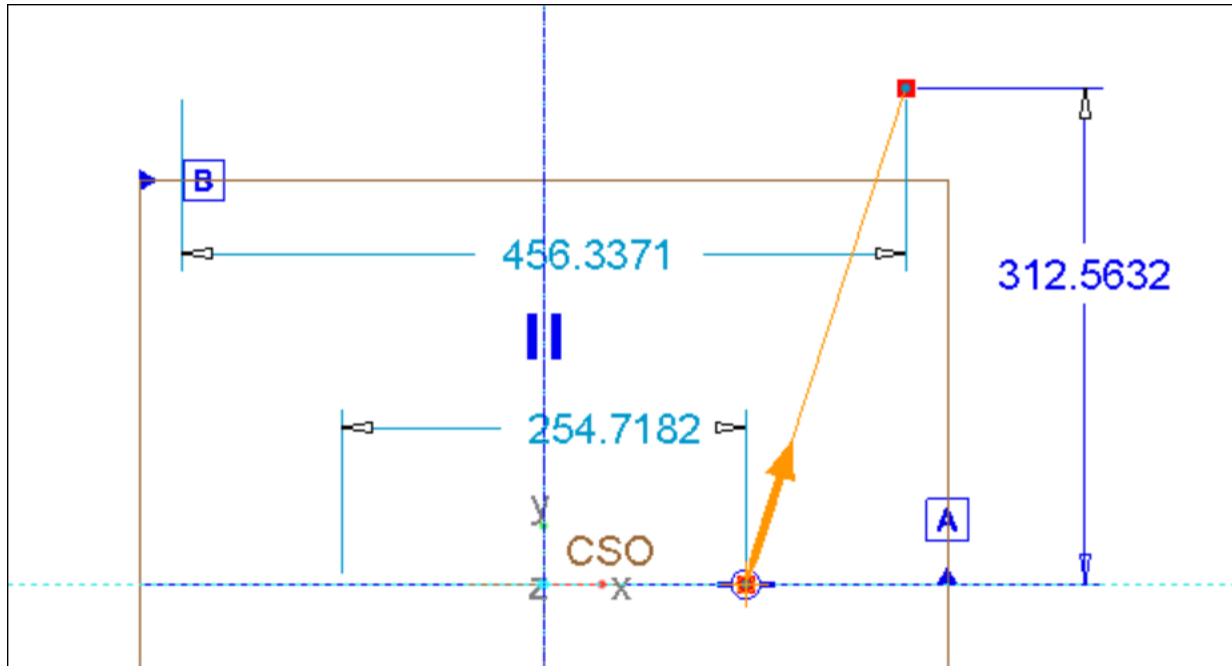
Figure 16.5(c) Dimensioned Sketch
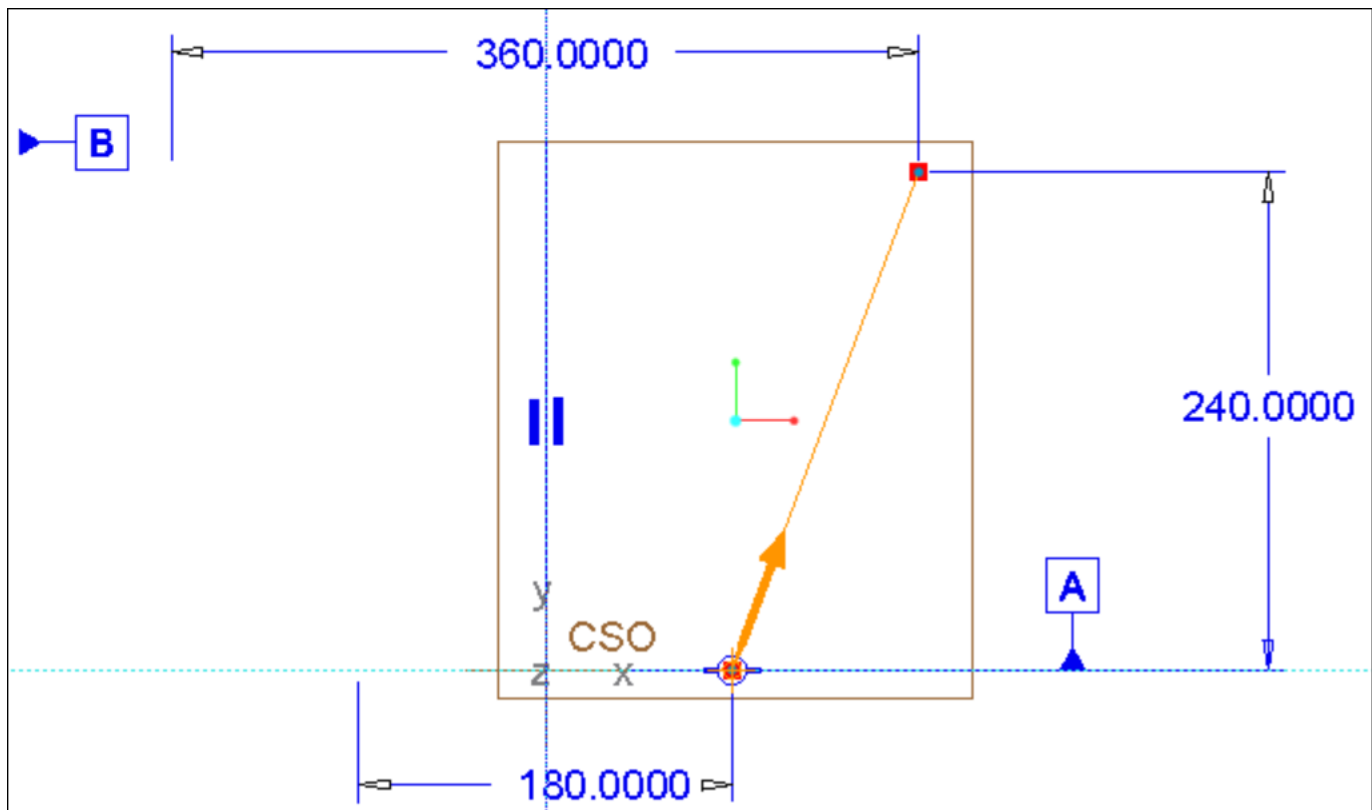
Figure 16.5(d) Modified Dimensions

Enter the pitch value **40** [icon] 40.0000 [v] **> Enter** [Fig. 16.5(e)] > [icon] **Create or edit sweep section** from the Dashboard > [icon] **Sketch View** > [icon] **Center and Point** sketch the section geometry of the spring at the intersection of the crosshairs [Fig. 16.5(f)] **> MMB > LMB >** select the dimension **> press RMB > Modify >** *type* **15 > Enter > OK > LMB** [Fig. 16.5(g)]
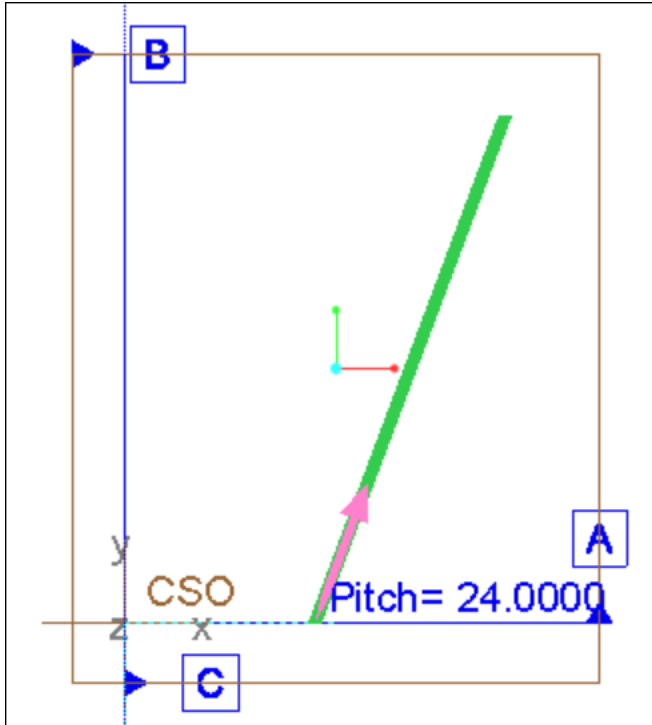
Figure 16.5(e) Pitch **24**, change to **40**
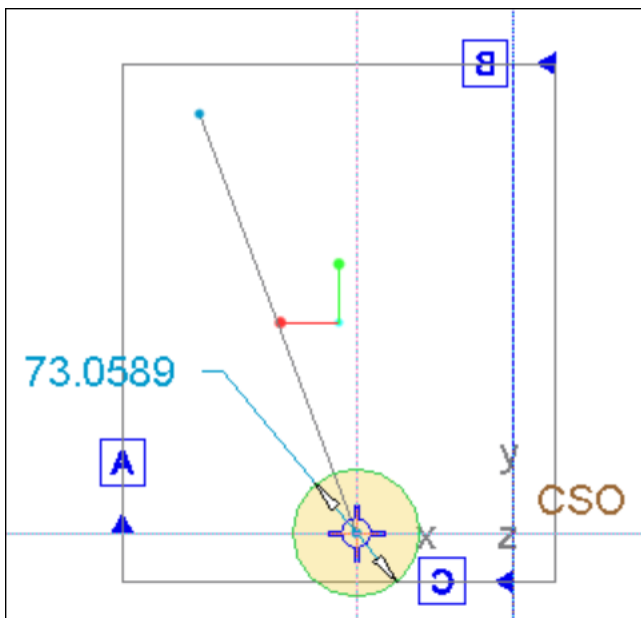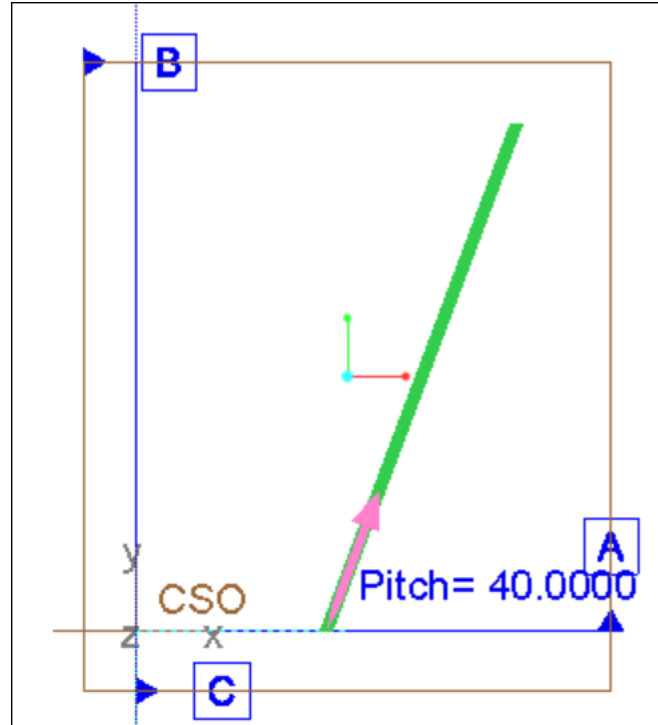
Figure 16.5(f) Sketch a Circle
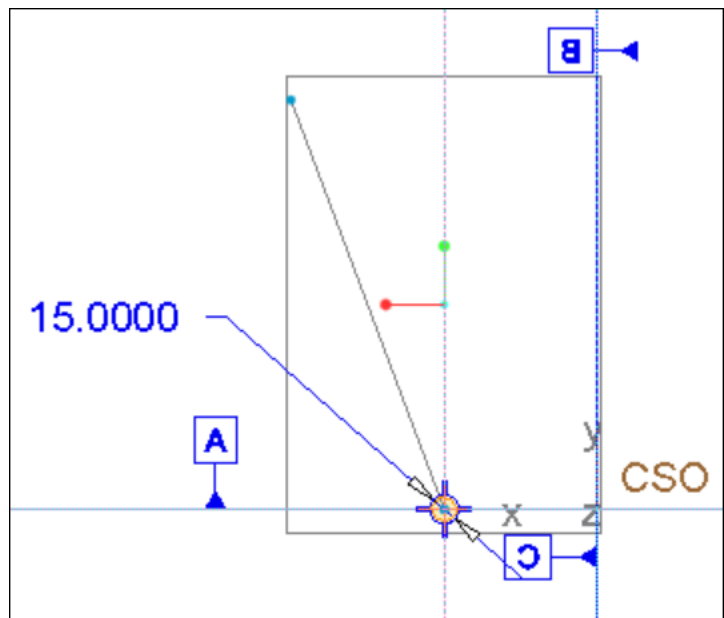
Figure 16.5(g) Wire Diameter **15**

Click: **RMB > OK** [Fig. 16.5(h)] > **RMB > Show Section Dimensions > Ctrl+D >** ✓ > **View** tab > **Appearance Gallery >** change the color of the part > **Ctrl+S > OK** [Fig. 16.5(i)] > change your model color
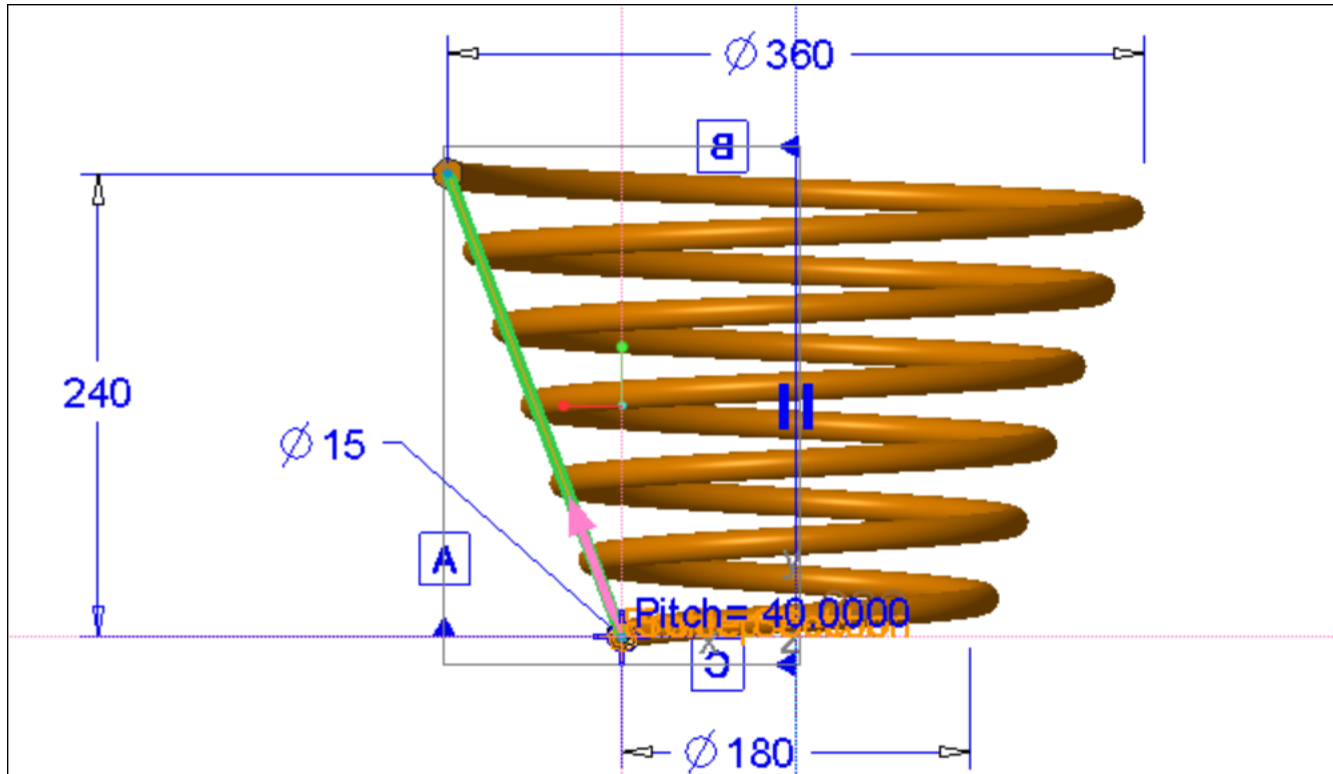
Figure 16.5(h) Helix Preview
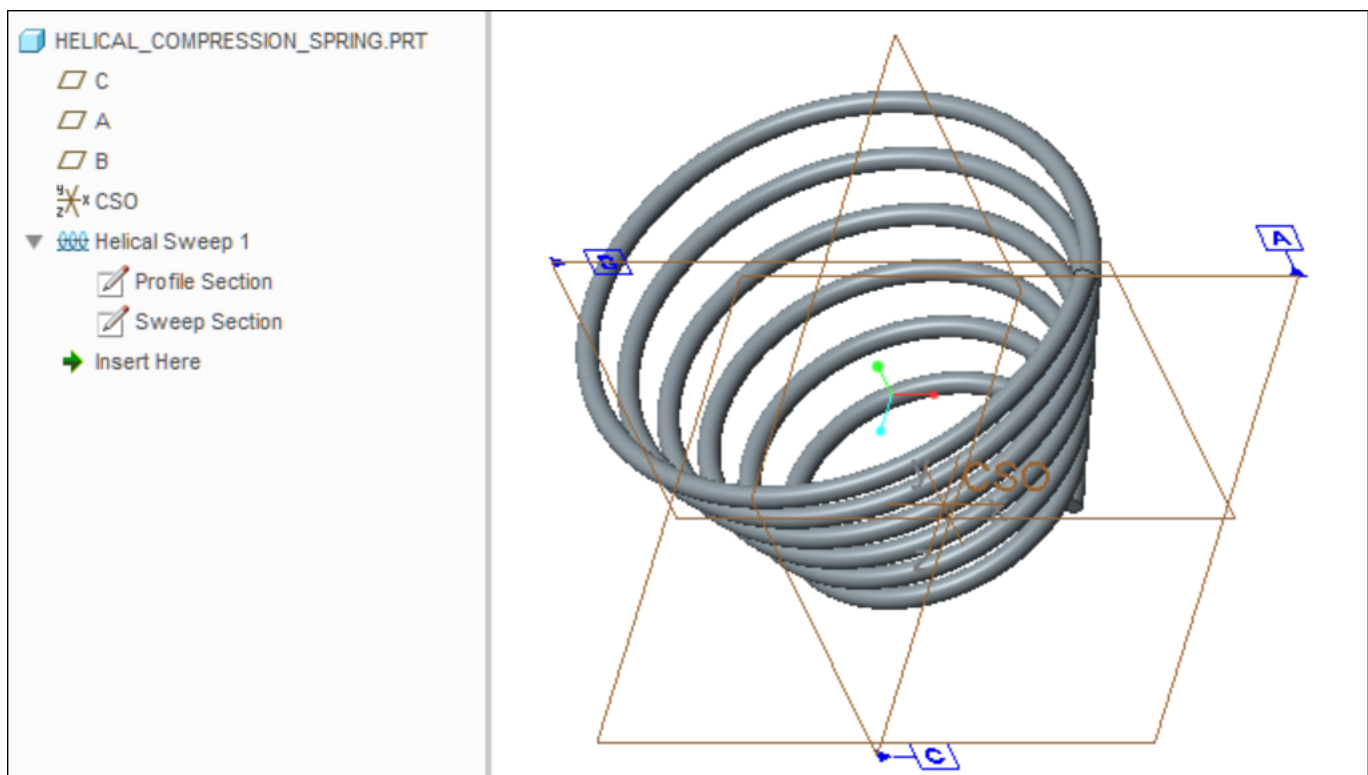
Figure 16.5(i) Completed Helical Sweep

Create the *ground ends*, click: **Model** tab > [⬚ Extrude] > [⬚] ▼ expand depth options by opening slide-up panel > [⬚] **Extrude on both sides** > [⬚] **Remove Material** > in the Graphics Window, press **RMB** > **Define Internal Sketch** > Sketch Plane- pick datum **C** > Reference- pick datum **A** > Orientation- **Bottom** [Fig. 16.6(a)] > **Sketch** > [⬚] > **RMB** > **Line Chain** > draw a horizontal line > **MMB** > **MMB** > **LMB** > modify the dimension [Fig. 16.6(b)] > spin the model as needed > [✔]
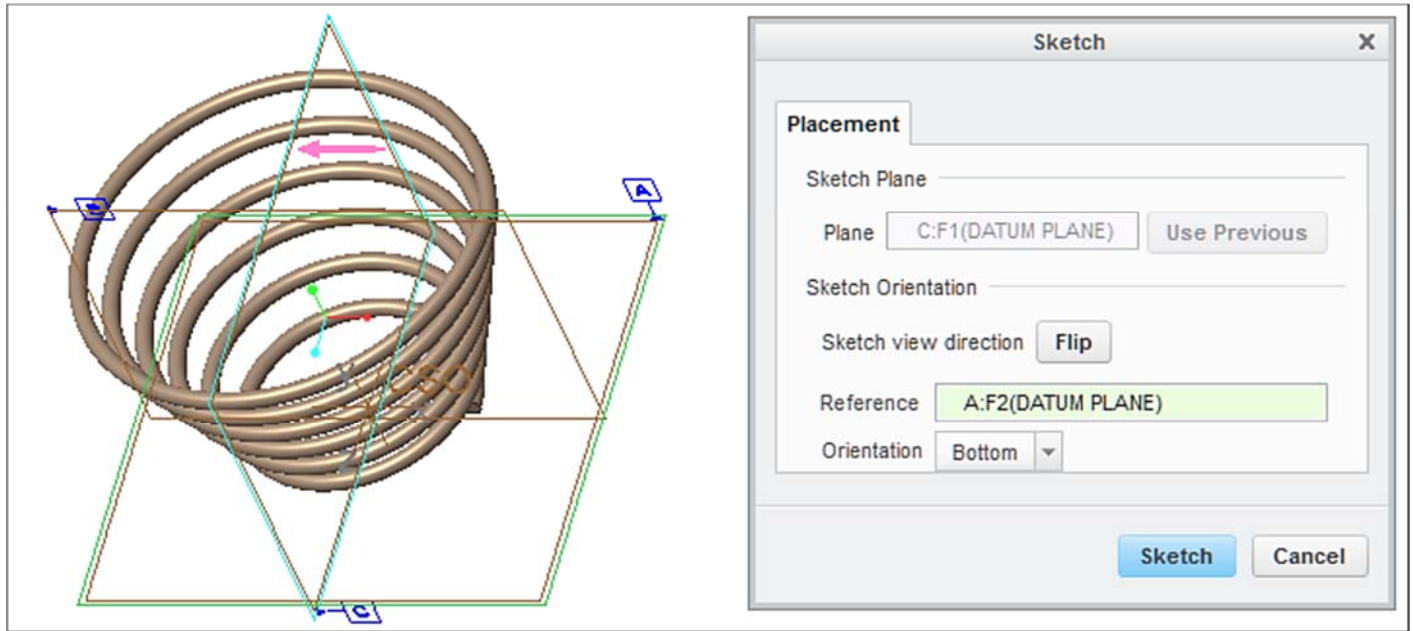
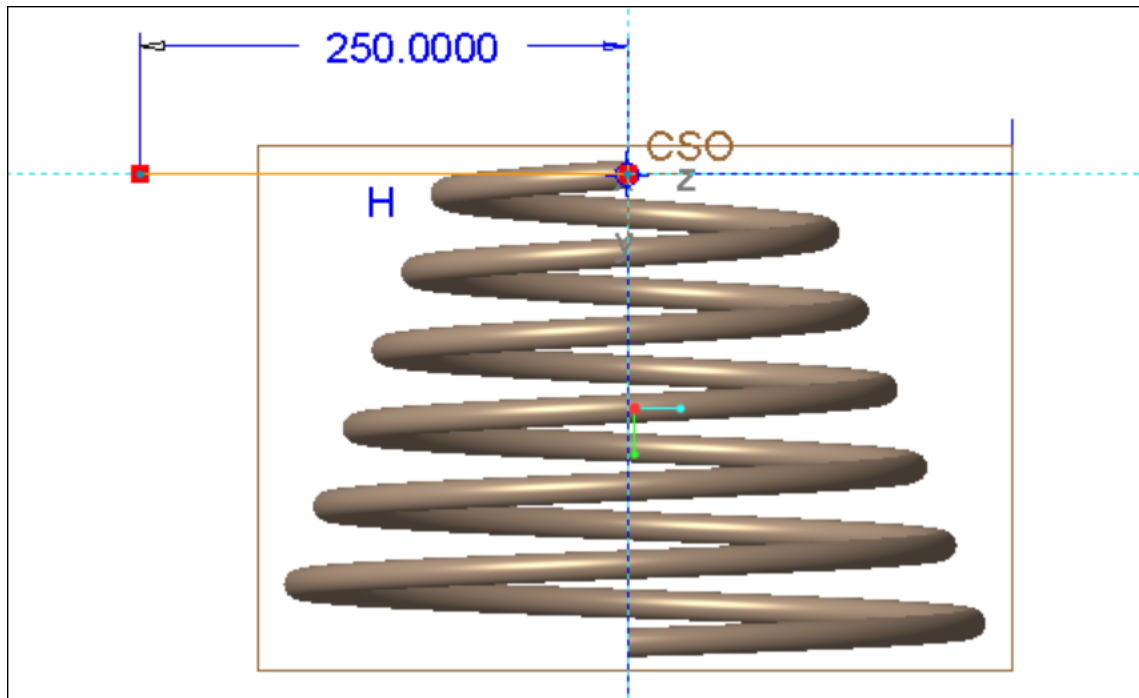**Figure 16.6(a)** Cut Sketch Orientation

Figure 16.6(b) Creating Ground Ends (any length will work as long as it goes beyond the spring)

Extend a depth handle to **300** so as to include the full spring **>** in the Graphics Window, press **RMB > Flip Material Side** [Fig. 16.6(c)] > ✅ [Fig. 16.6(d)] > **LMB**
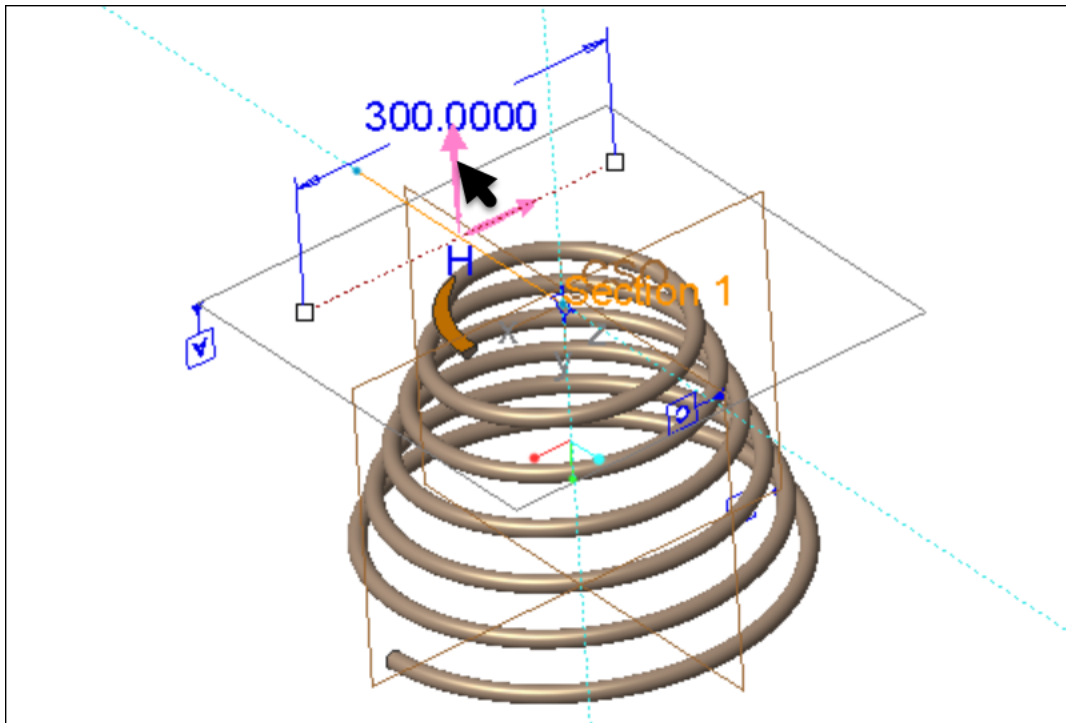

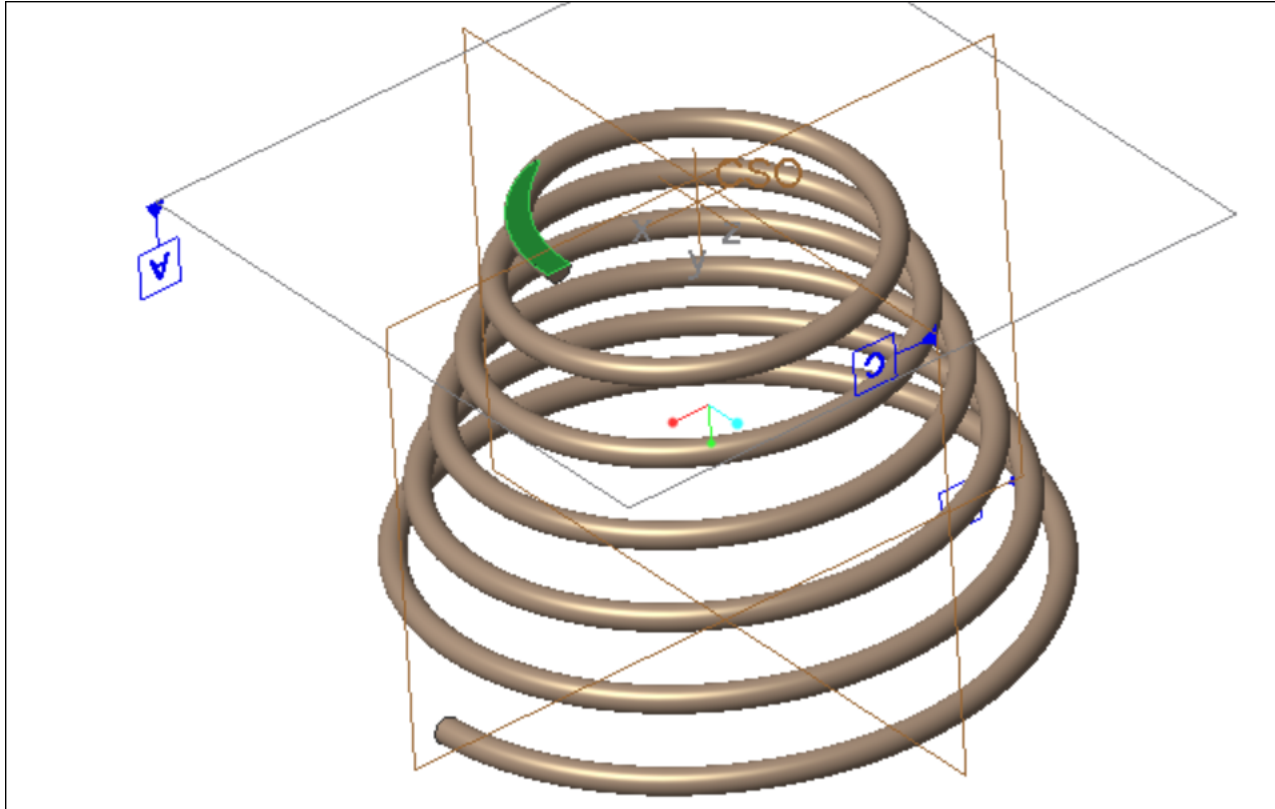**Figure 16.6(c)** Depth Handles (Squares) and Material Side Arrow (currently pointing upward)

Figure 16.6(d) Completed Cut for One Ground End

The second ground end is created using similar commands [Fig. 16.7(a)]. **>** Complete the spring [Fig. 16.7(b)].
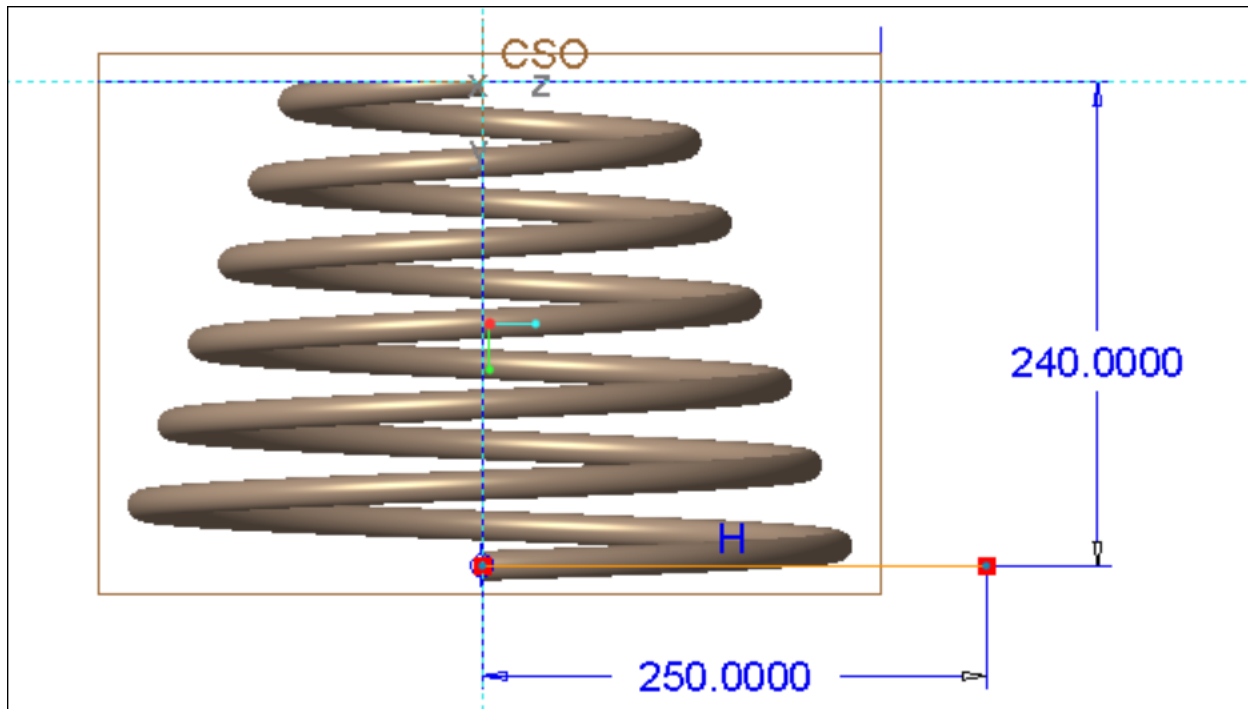**> Ctrl+D > Ctrl+S**

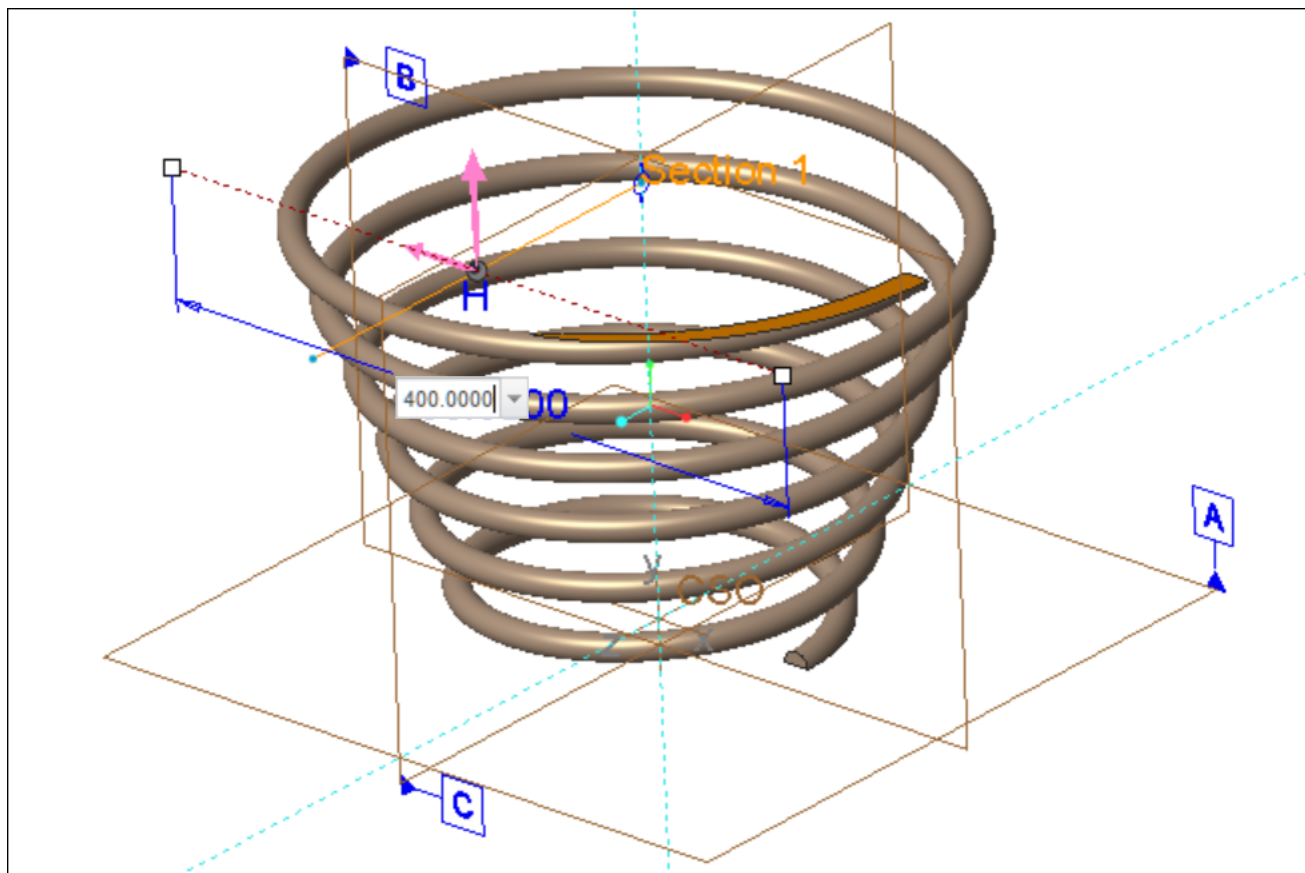**Figure 16.7(a)** Creating the Second Ground End

Figure 16.7(b) Preview of the Completed Cut for the Second Ground End

695

Figure 16.8 provides an **ECO** (Engineering Change Order) for a new spring. Copy the file you are working on by clicking: **File > Save As > Save a Copy > HELICAL_EXTENSION_SPRING > OK > File > Close > File > Open > helical_extension_spring.prt > Open >** delete the existing ground ends > modify the pitch to **10 mm >** change the wire diameter to **7.5 mm >** complete the extension spring [Figs. 16.9(a) through 16.10(d)]. The free length is to be **120 mm**. The large diameter will now be **180 mm**, and the small diameter will be **120 mm**. **> Ctrl+S > OK > File > Close**

Figure 16.8 ECO to Create a Helical Extension Spring **[You are not creating this ECO drawing; you are making a new part from an existing part (copied) using different dimensions and features]**
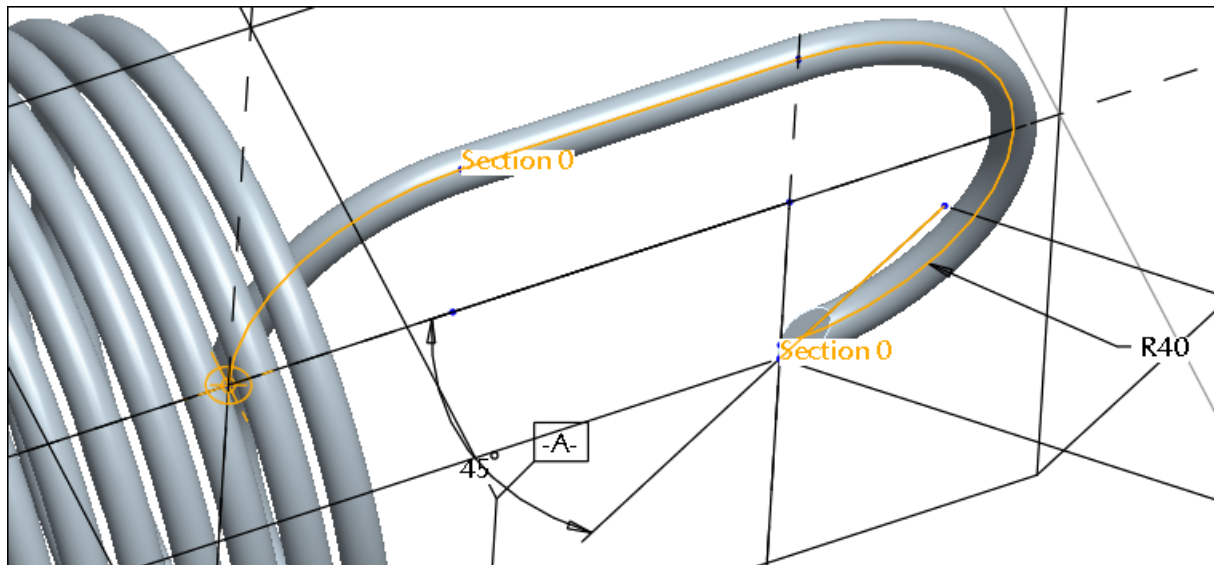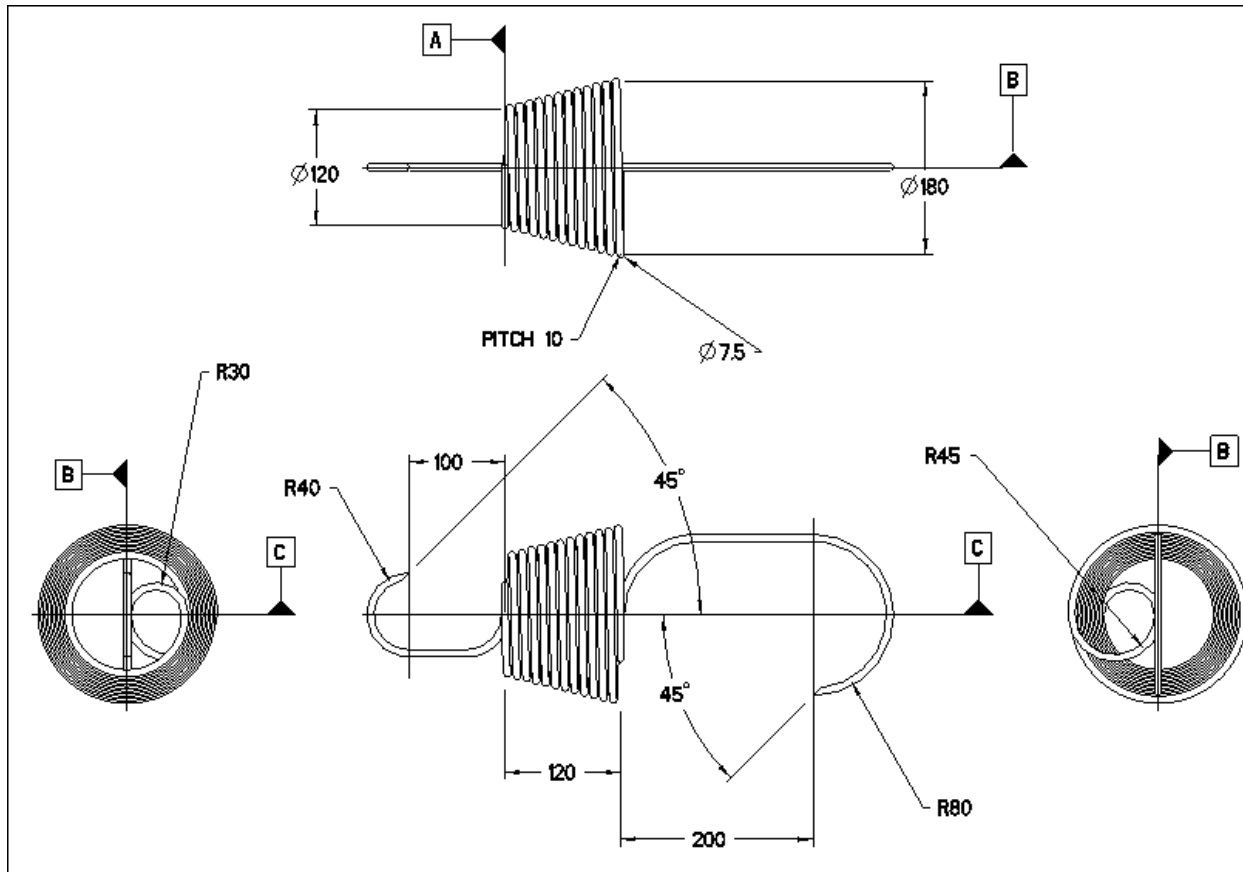

Figure 16.9(a) Ground End

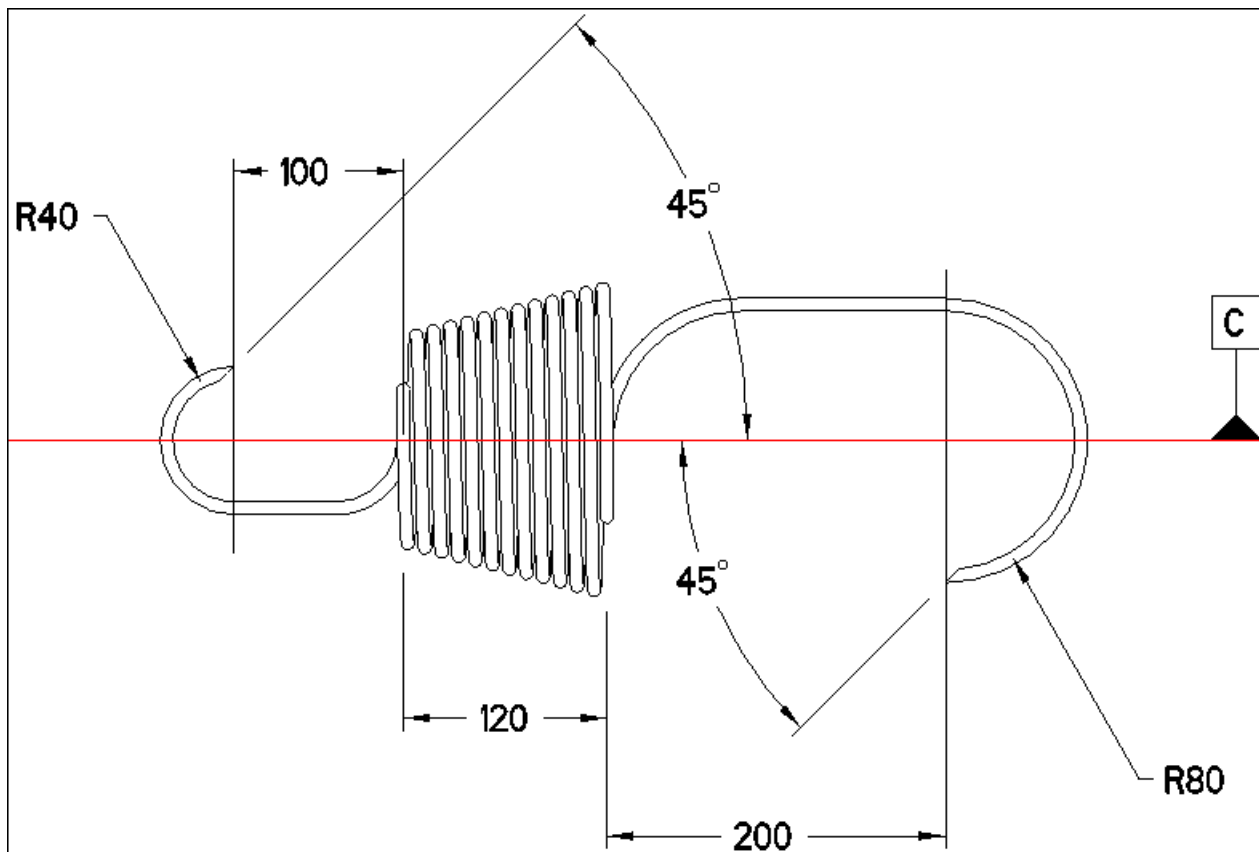**Figure 16.9(b)** Detail Drawing of Helical Extension Spring with Machine Hook Ends

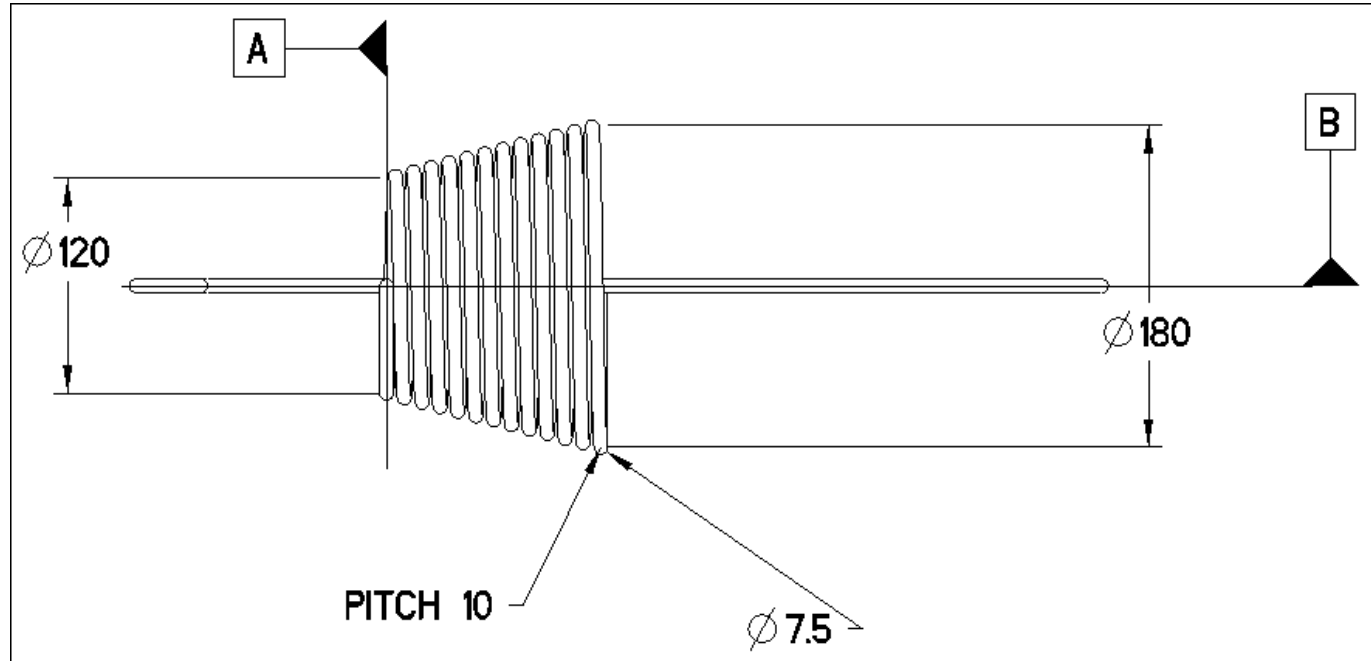**Figure 16.9(c)** Front View

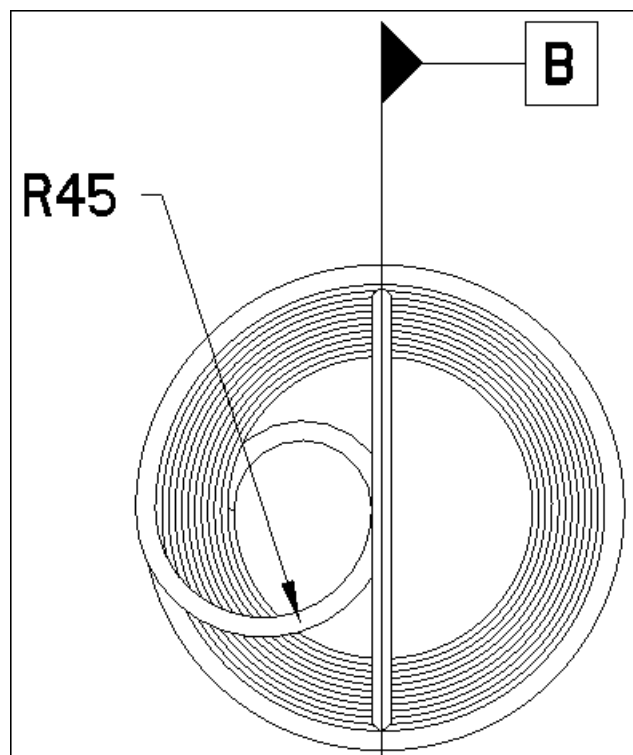**Figure 16.9(d)** Top View

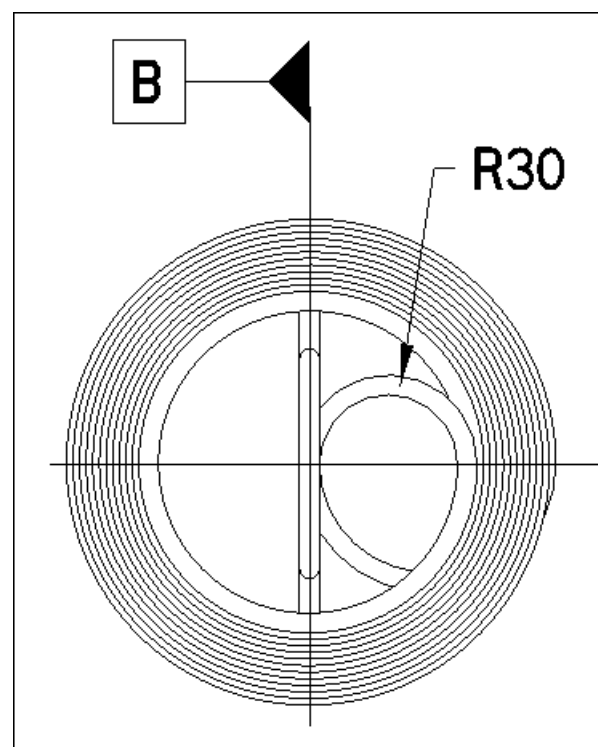**Figure 16.9(e)** Right Side View

Figure 16.9(f) Left Side View

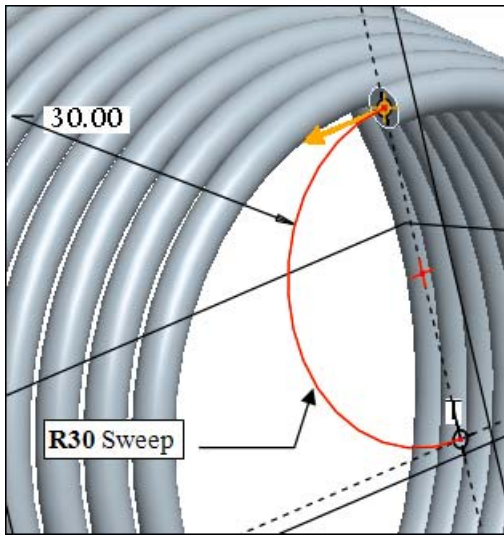Create the machine hooks using simple sweeps and cuts, as shown in Figures 16.10(a) through 16.10(d).
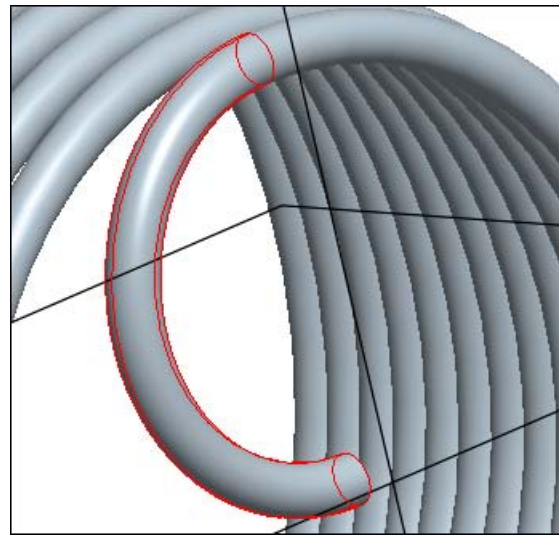

Figure 16.10(a) Sweep **R30**
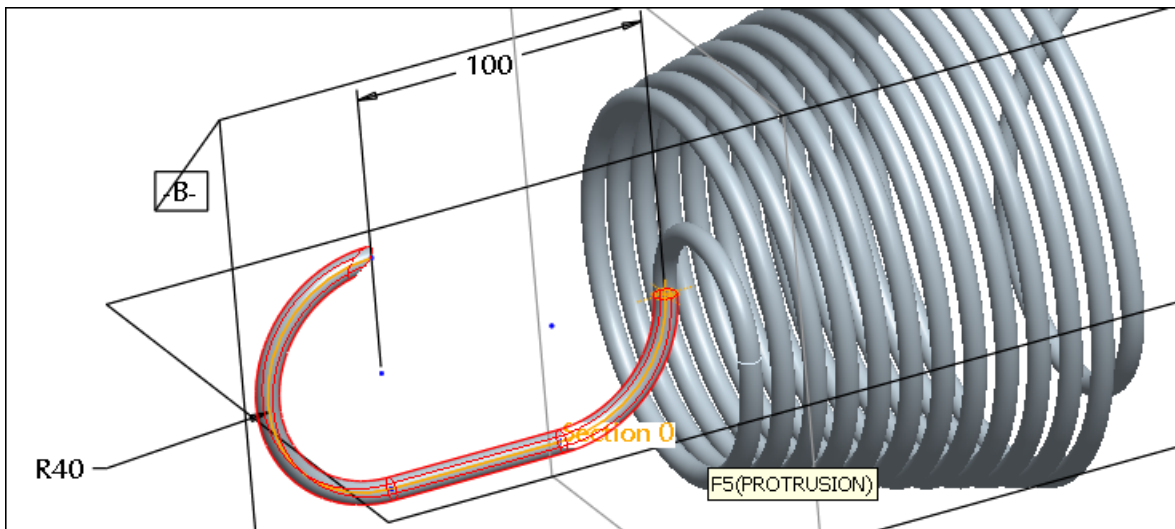

Figure 16.10(b) Completed Sweep

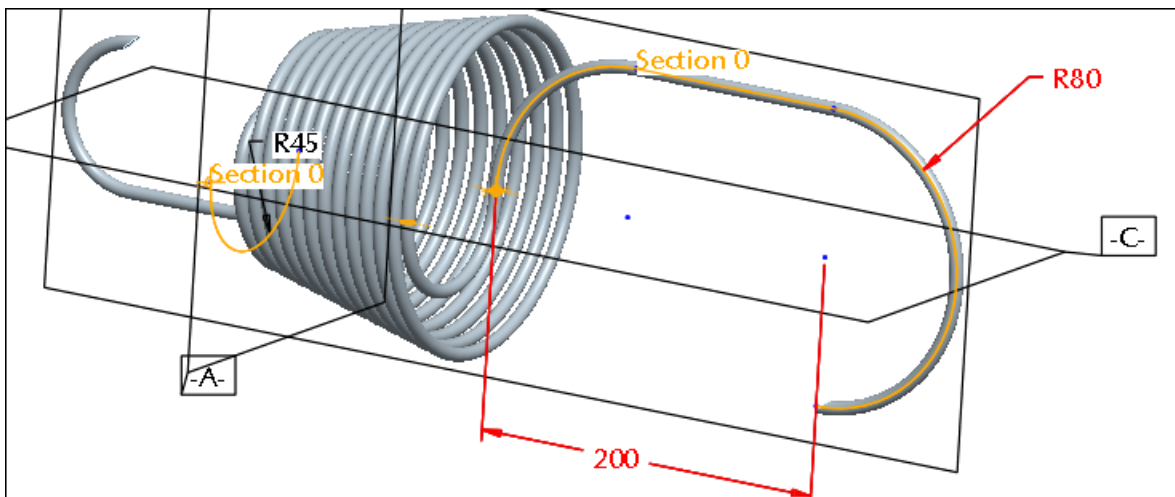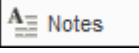**Figure 16.10(c)** Small Hook End Sweep

Figure 16.10(d) Large Hook End Sweep

Annotations

When you attach a note to an entity, that entity is considered the "parent" of the note. Deleting the parent deletes all of the notes of the parent. You can attach model notes anywhere in the model; they do not have to be attached to a parent. Here we will add a note to the part and describe the spring.

Click: **File > Open > helical_compression_spring.prt > Open > Annotate** tab **> FLAT TO SCREEN >**

RMB on the command button **> Set >** [A≣ Notes] **Unattached Note** [Fig. 16.11(a)] **>** select a place on the screen to place the note **>** type the following note [Fig. 16.11(b)]:

> **Helical Compression Spring**
> **Constant Pitch**
> **Right-Handed**
> **40 mm Pitch**
> **Wire Diameter 15mm**
> **Ground Ends**
> **(grind ends parallel)**

Helical Compression Spring
Constant Pitch
Right-Handed
40 mm Pitch
Wire Diameter 15mm
Ground Ends
(grind ends parallel)

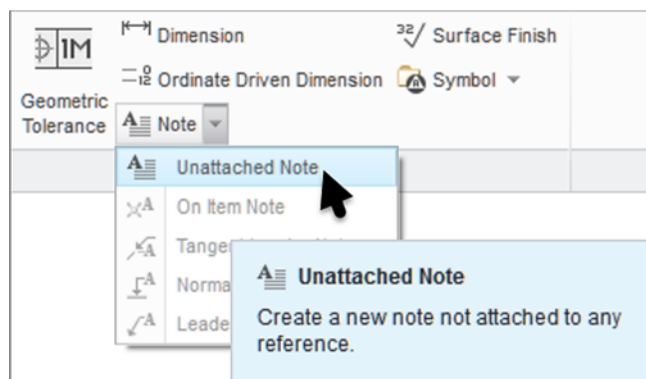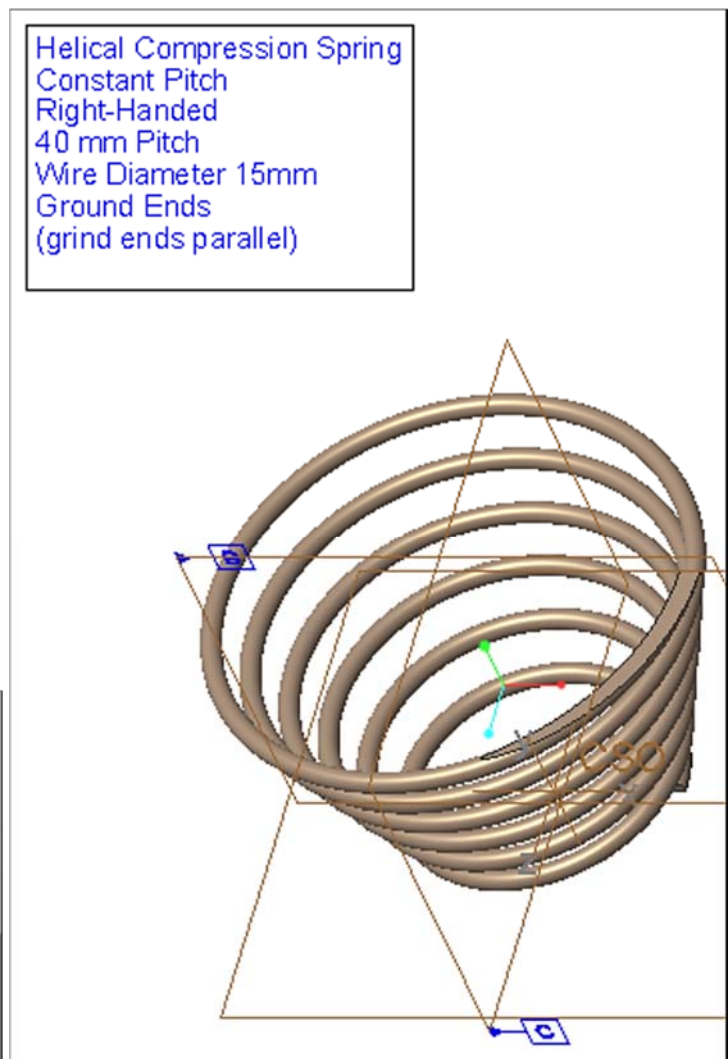**Figure 16.11(a)** 3D Notes   **Figure 16.11(b)** Note Dialog Box

Click: **MMB** [Fig. 16.11(c)] > move the note as needed > with the note still selected (highlighted); **RMB >**
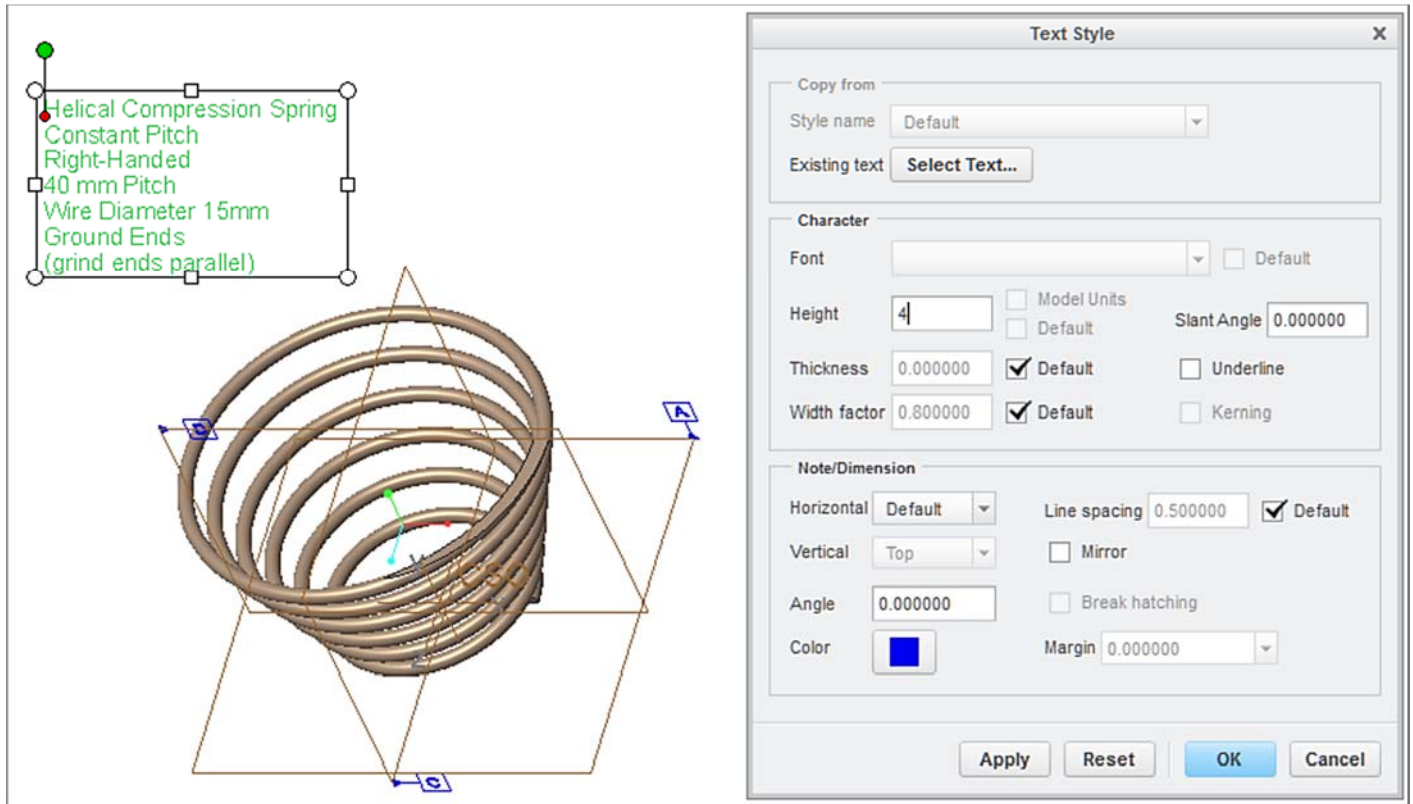**Text Style >** ☐ Default > Height **4** > **Enter** [Fig. 16.11(d)] > **OK** > **LMB** > **Ctrl+S**
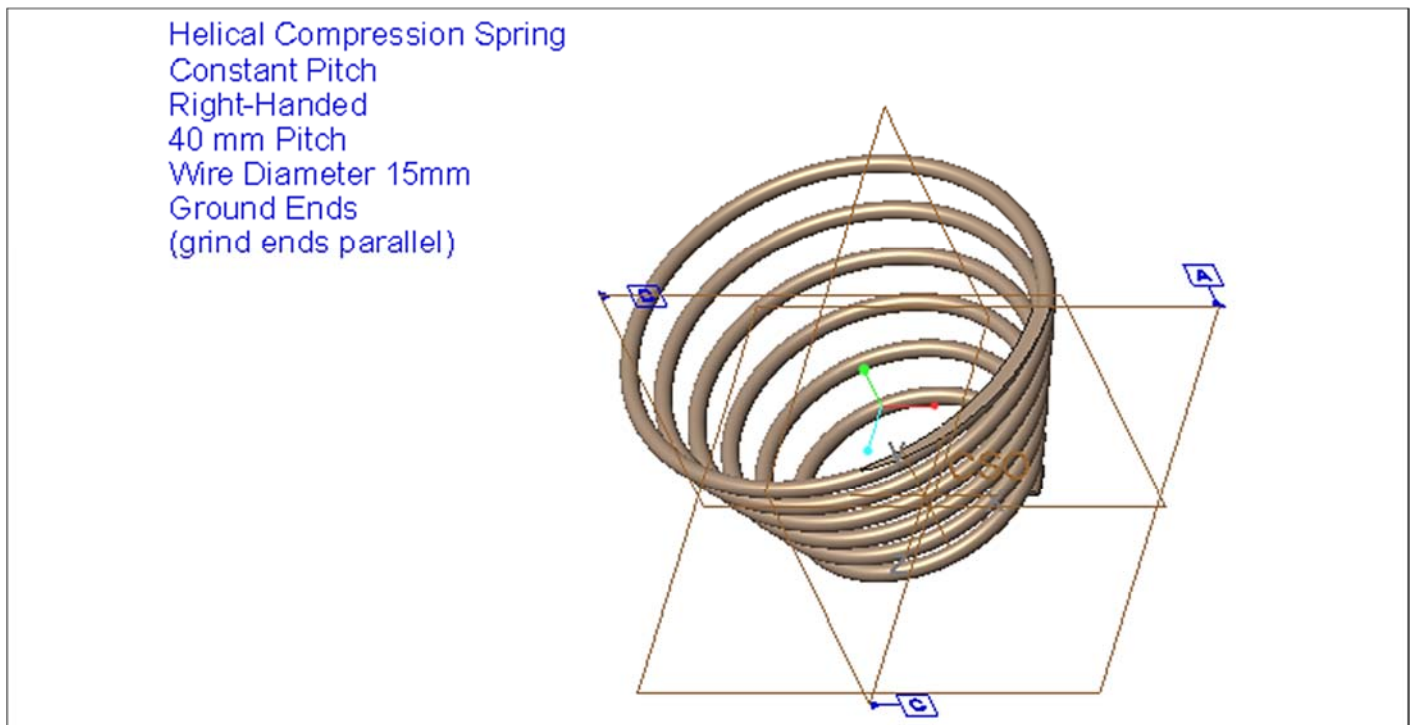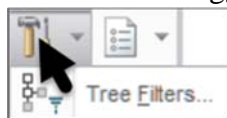

Figure 16.11(c) Placing the Note

Figure 16.11(d) Completed Note

701

You can toggle model annotations on and off using [icon] **Turn on or off 3D annotations and annotation elements >** toggle the annotations *off* and *on* **>** display the note in the Model Tree by clicking:

[icon] Tree Filters... **>** [icon] Tree Filters... **>** toggle all *on* [Fig. 16.12(a)] **> Apply > OK** [Fig. 16.12(b)]
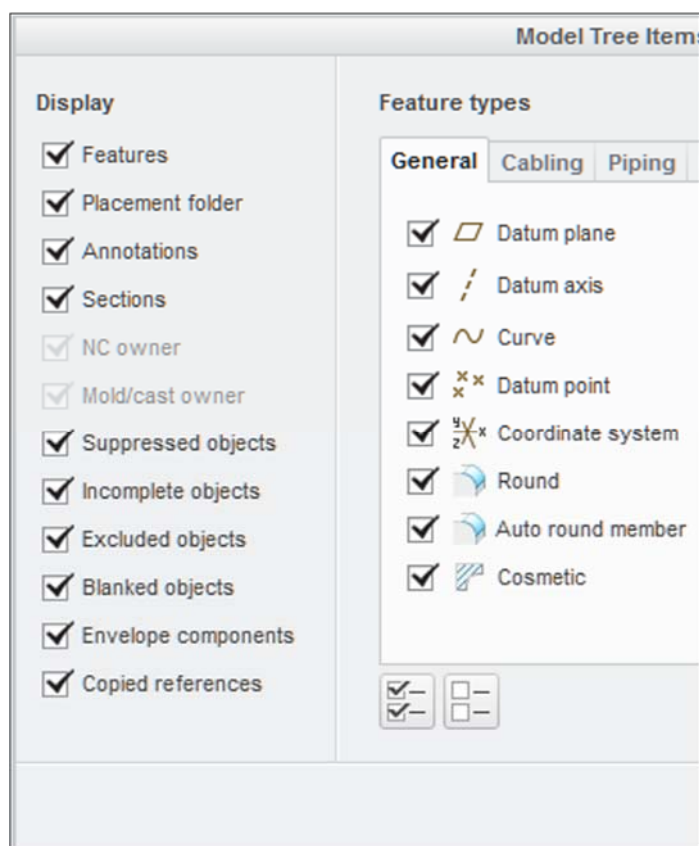

Figure 16.12(a) Displaying 3D Notes (Annotations) in Model Tree

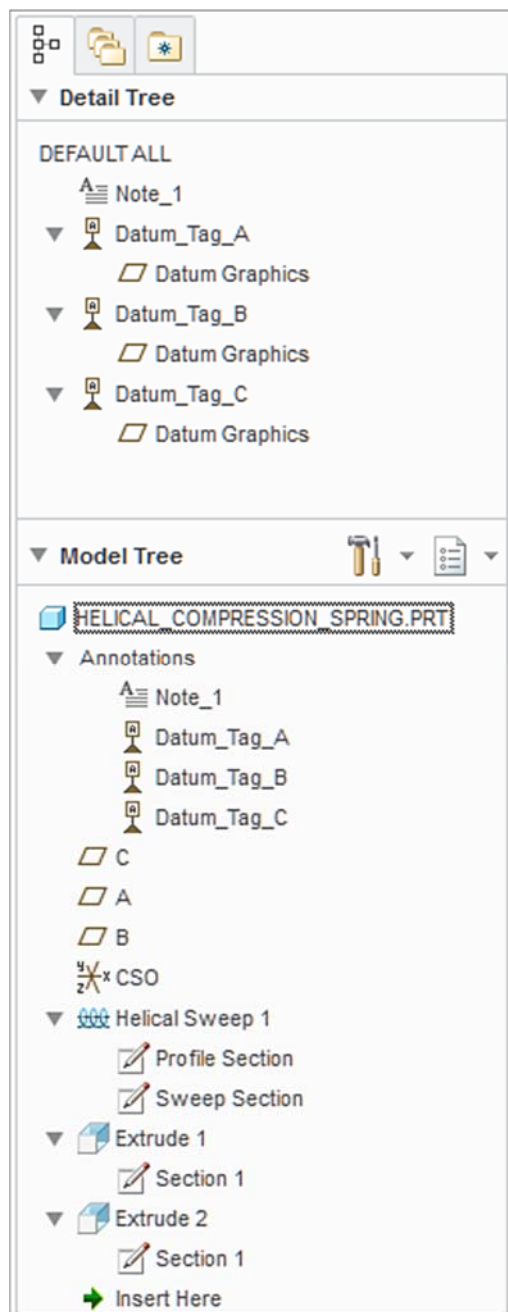**Figure 16.12(b)** Detail Tree and Model Tree

Click on ⊠ Note_1 in the Model Tree **> RMB > Rename >** type: **Compression_Spring > Enter > RMB > Add Link [**Fig. 16.13(a)] **>** type: **http://www.americanprecspring.com/** [Fig. 16.13(b)] **> OK > OK > LMB >** place your pointer over the note in the graphics area to see the Screen Tip [Fig. 16.13(c)]
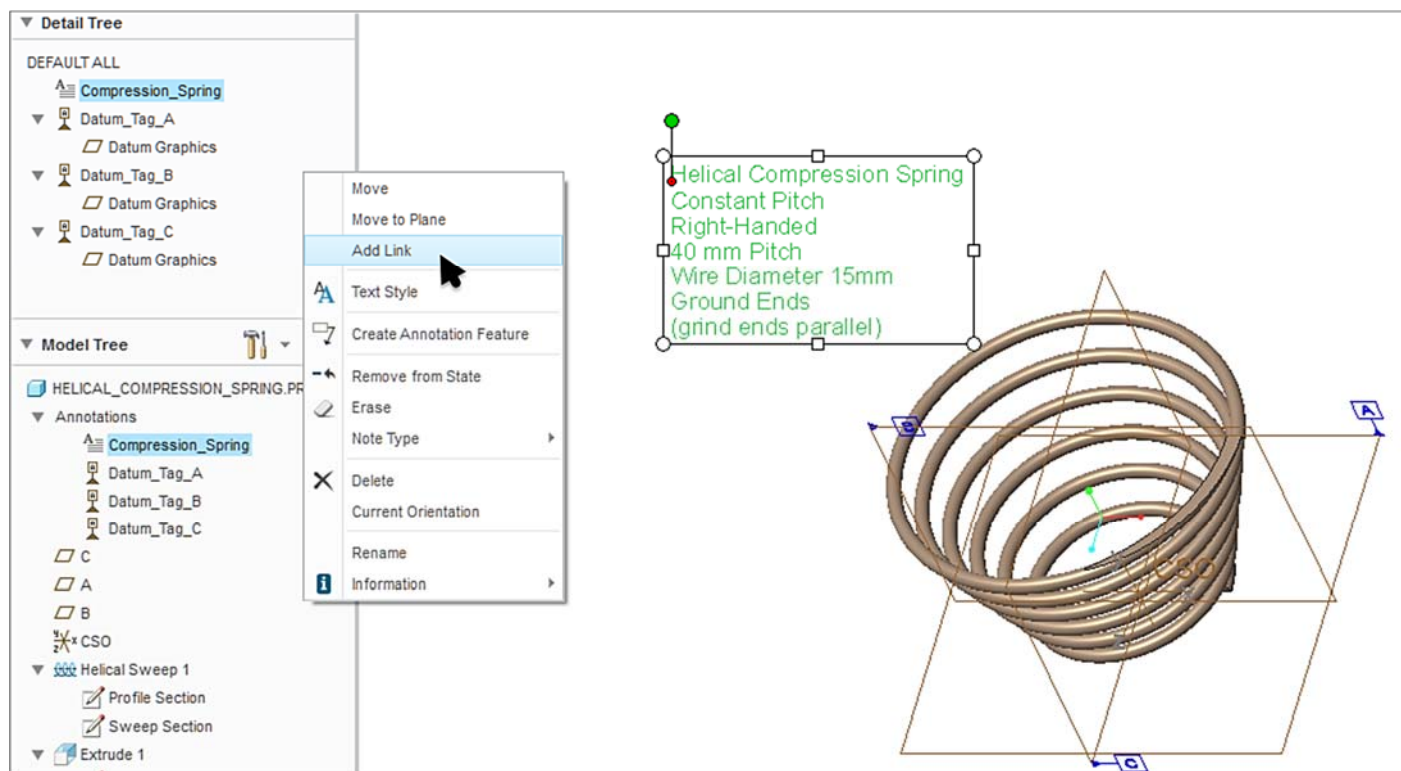

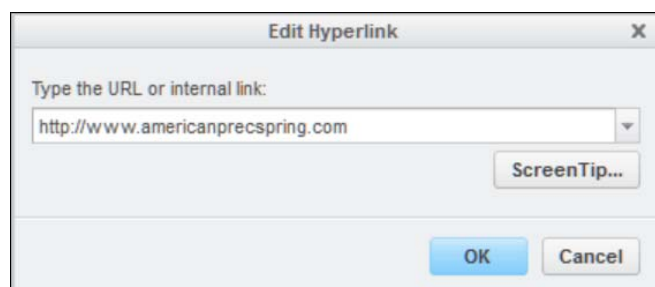**Figure 16.13(a)** Press RMB **>** Properties (your options list may appear differently)

Figure 16.13(b) Hyperlink
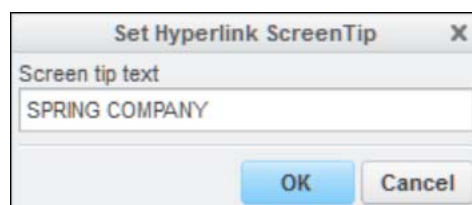

Figure 16.13(c) Screen Tip

Create a screen tip that will display as the pointer passes over the note, click: ⊠ ScreenTip... **>** *type* **SPRING COMPANY** [Fig. 16.13(d)] **> OK > OK > OK >** place the pointer over the note [Fig. 16.13(e)] **>** move the pointer off of the note **> LMB** to deselect

Figure 16.13(d) Note
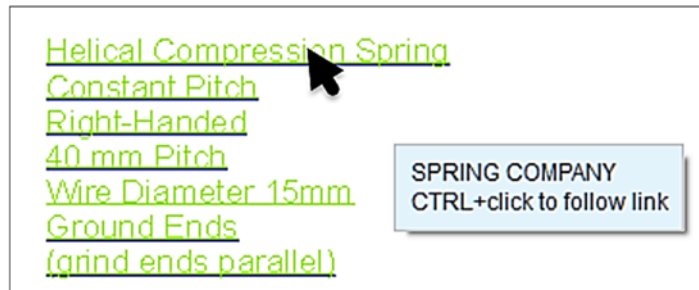
Figure 16.13(e) Screen Tip Displayed

703

Open the URL, click: **A≡ Compression_Spring** from the Model Tree **> RMB > Open URL** [Fig. 16.13(f)] *(URL opens in the browser window)* [Fig. 16.13(g)] > close the **Browser > LMB** to deselect **> Ctrl+D > Ctrl+S > File > Manage File > Delete Old Versions > Enter**

Figure 16.13(f) Open URL
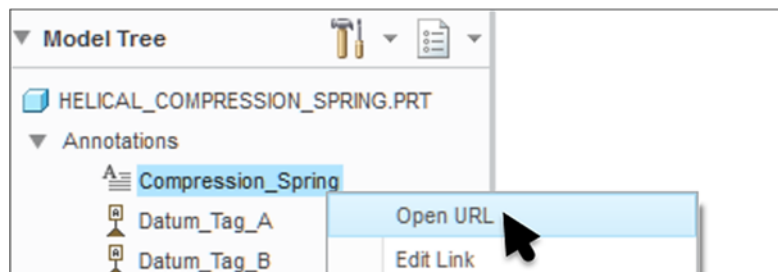

Figure 16.13(g) American Precision Spring Website (this web page may have since been updated)

704

Annotation Features

3D Notes can also be added to an entity using **Annotation Features**. Annotation features are data features that you can use to manage the model annotation and propagate model information to other models, or to manufacturing processes. The Annotation Feature Tool options correspond to the **ASME Y14.41 Digital Product Definition Data Practices.**

An Annotation feature consists of one or more Annotation Elements. Each Annotation Element (AE) can contain one annotation item, along with associated references and parameters. You can include the following types of annotations in an Annotation Element:

- Note
- Symbol
- Surface Finish
- Geometric Tolerance
- Set Datum Tag
- Ordinate Baseline
- Driven Dimension
- Ordinate Driven Dimension
- Reference Dimension
- Ordinate Reference Dimension
- Existing Annotation

Digital Product Definition Data Practices

"ASME Y14.41 establishes requirements for preparing, organizing and interpreting 3-dimensional digital product images (Fig. 16.14). Digital Product Definition Data Practices, which represents an extension of the popular Y14.5 standard for 2-dimensional drawings, reflects the growing need for a uniform method of documenting the data created in today's computer-aided design (CAD) environments. The standard provides a guide for CAD software developers working on improved modeling and annotation practices for the engineering community. ASME Y14.41 sets forth the requirements for tolerances, dimensional data, and other annotations. ASME Y14.41 advances the capabilities of Y14.5, Dimensioning and Tolerancing, the standard pertaining to 2-D engineering drawings".

In the following steps you will create a single-view 3D definition of the model for manufacturing, instead of a traditional multi-view drawing.
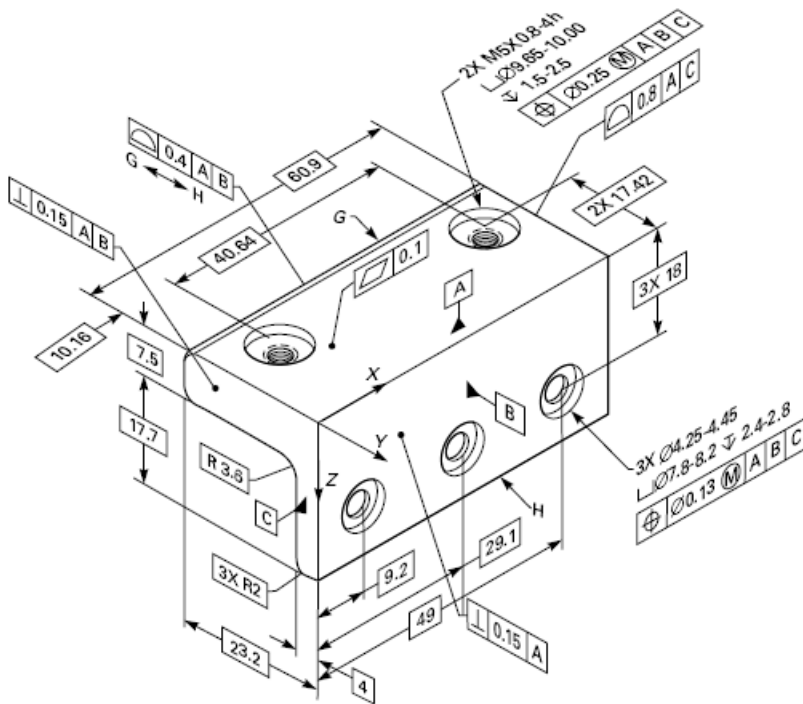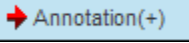

Figure 16.14 Digital Product Definition, ASME Y14.41

Click: 🔲 **Open the View Manager > Orient** tab **> New >** *type* **Annotation > Enter >** rotate the view **>** click

on ➔ Annotation(+) **> RMB > Save** (Fig. 16.15) **> OK** *(the + sign will disappear)* **> Close > Ctrl+S**
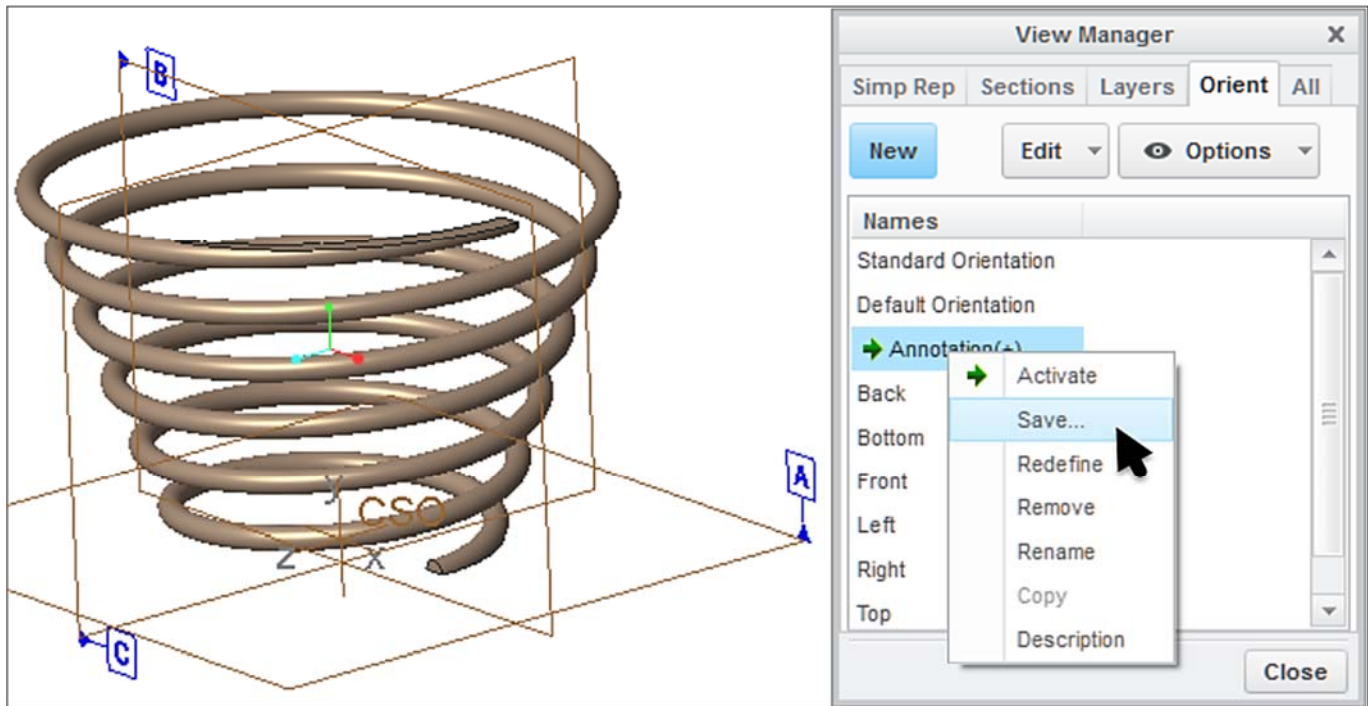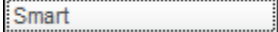
Figure 16.15 Reorient the Model

Click: Smart **>** Annotation **>** select the 3D note [Fig. 16.16(a)] **>** select a new note position [Fig. 16.16(b)] **> LMB** to deselect **> Ctrl+S**
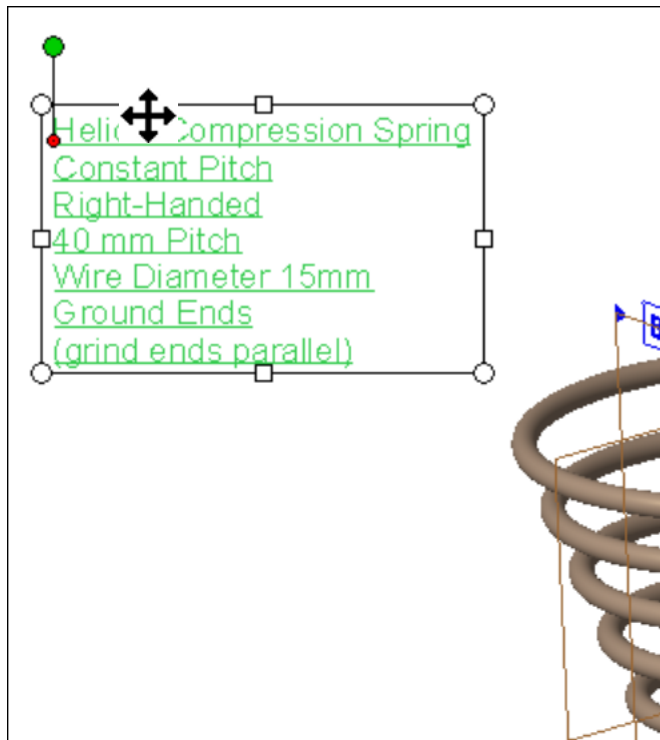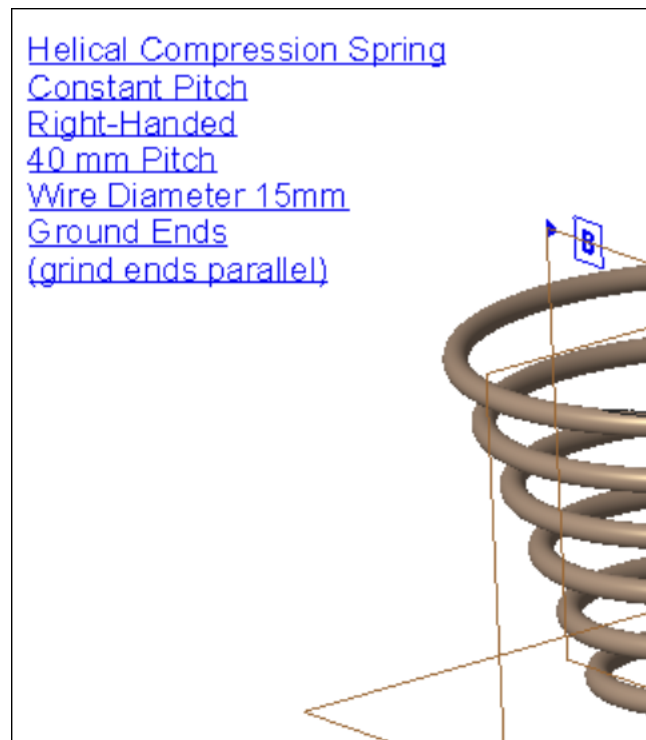

Figure 16.16(a) Move

Figure 16.16(b) Place the Note

Click: [Annotation] > [Smart] > with the **Annotate** tab active, select the Helical Sweep feature > press **RMB** > **Show Annotations** [Fig. 16.17(a)] > [⊢⊣] tab > [☑] > **OK** from the Show Annotations dialog box [Fig. 16.17(b)] > **LMB** to deselect
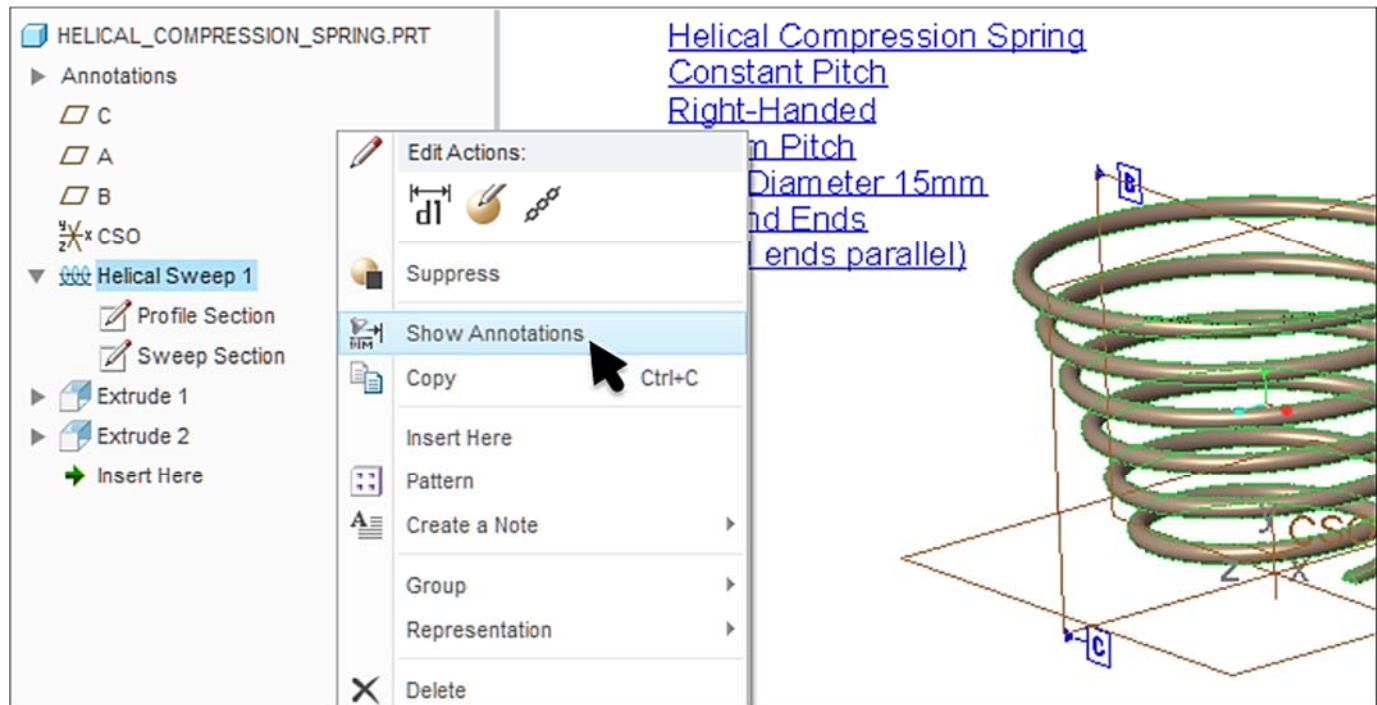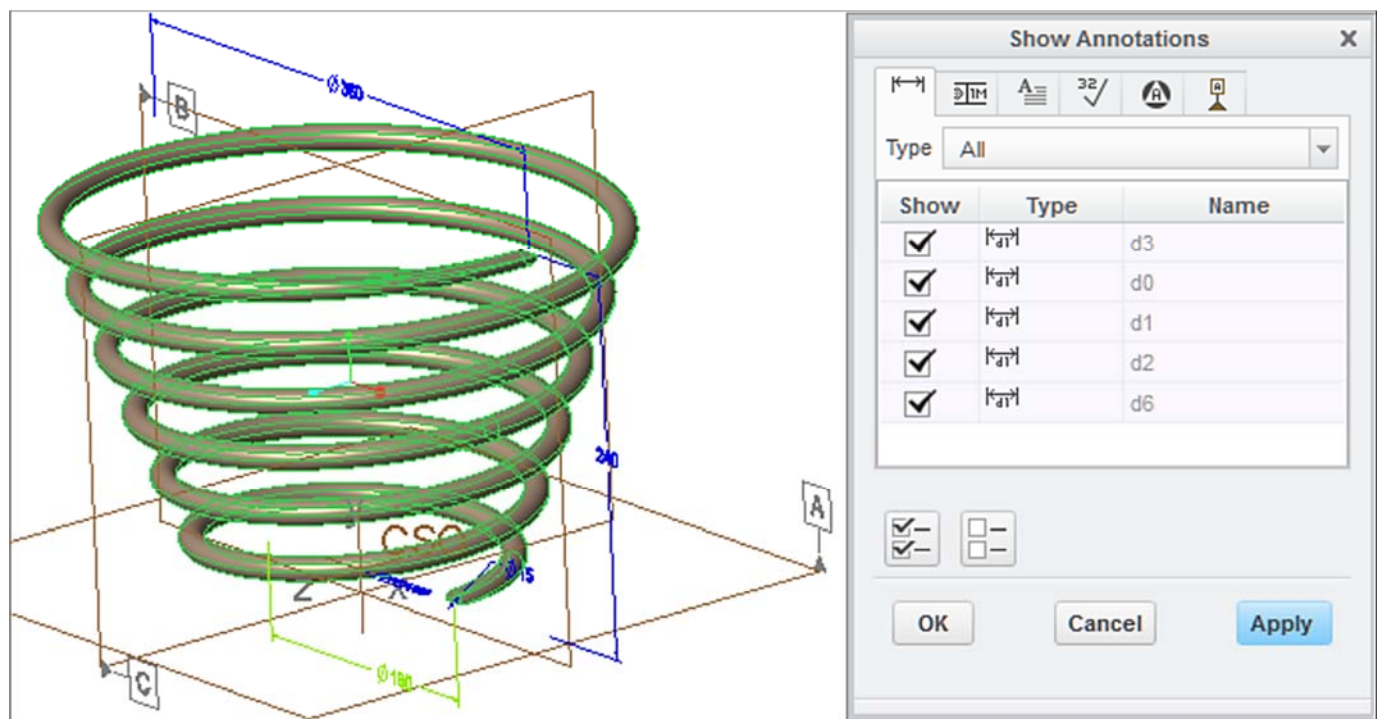

Figure 16.17(a) Create Driving Dimension AE

Figure 16.17(b) Displayed Driving Dimensions

Select **Datum_Tag_B >** press **RMB > Flip** [Fig. 16.18(a)] **>** select an annotation **>** press and hold down the **LMB >** move the pointer to a new location **>** release the **LMB >** move each annotation to a better location **>** select the **PITCH 40** annotation **>** press **RMB > Current Orientation** [Figs. 16.18(b-c)]
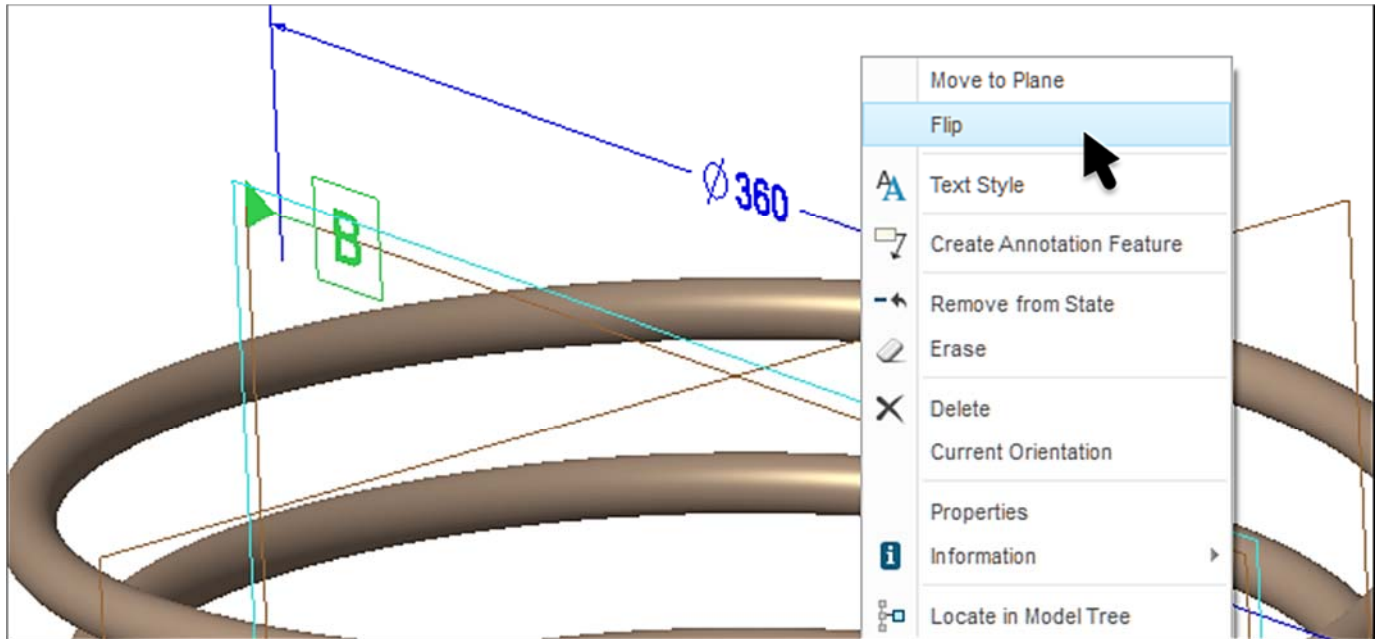

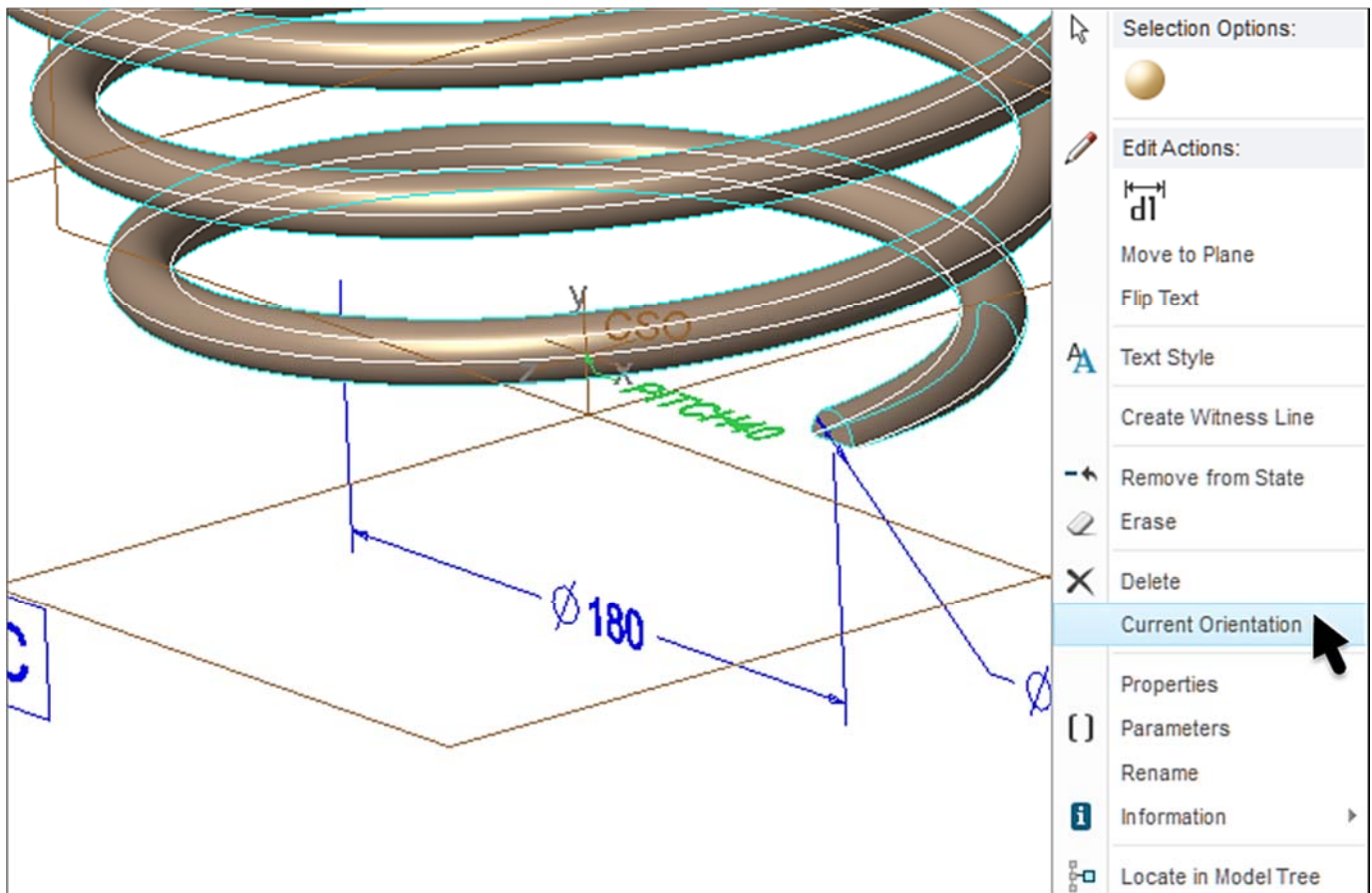**Figure 16.18(a)** Flip **Datum_Tag_B**

Figure 16.18(b) Current Orientation (your options list may appear differently)

Click: ⦿ Named Model Orientation > ▼ > **TOP** [Fig. 16.18(d)] > **Flip > OK >** move the **PITCH 40** annotation as needed **>** rotate the model [Fig. 16.18(e)]
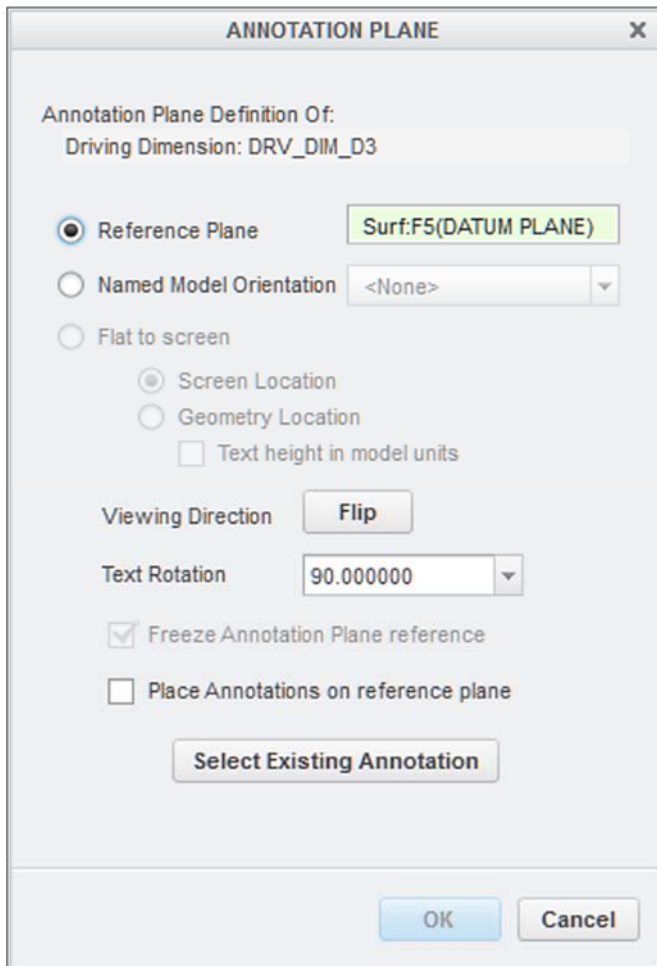
ANNOTATION PLANE   ✕

Annotation Plane Definition Of:
    Driving Dimension: DRV_DIM_D3

⦿ Reference Plane   Surf:F5(DATUM PLANE)

○ Named Model Orientation   <None>   ▼

○ Flat to screen
    ⦿ Screen Location
    ○ Geometry Location
        ☐ Text height in model units

Viewing Direction   **Flip**

Text Rotation   90.000000   ▼

☑ Freeze Annotation Plane reference

☐ Place Annotations on reference plane

        Select Existing Annotation

                    OK   Cancel

Figure 16.18(c) Annotation Plane Dialog Box

ANNOTATION PLANE   ✕

Annotation Plane Definition Of:
    Driving Dimension: DRV_DIM_D3

○ Reference Plane   Select items

⦿ Named Model Orientation   TOP   ▼

○ Flat to screen
    ⦿ Screen Location
    ○ Geometry Location
        ☐ Text height in model units

Viewing Direction   **Flip**

Text Rotation   90.000000   ▼

☑ Freeze Annotation Plane reference

☐ Place Annotations on reference plane

        Select Existing Annotation
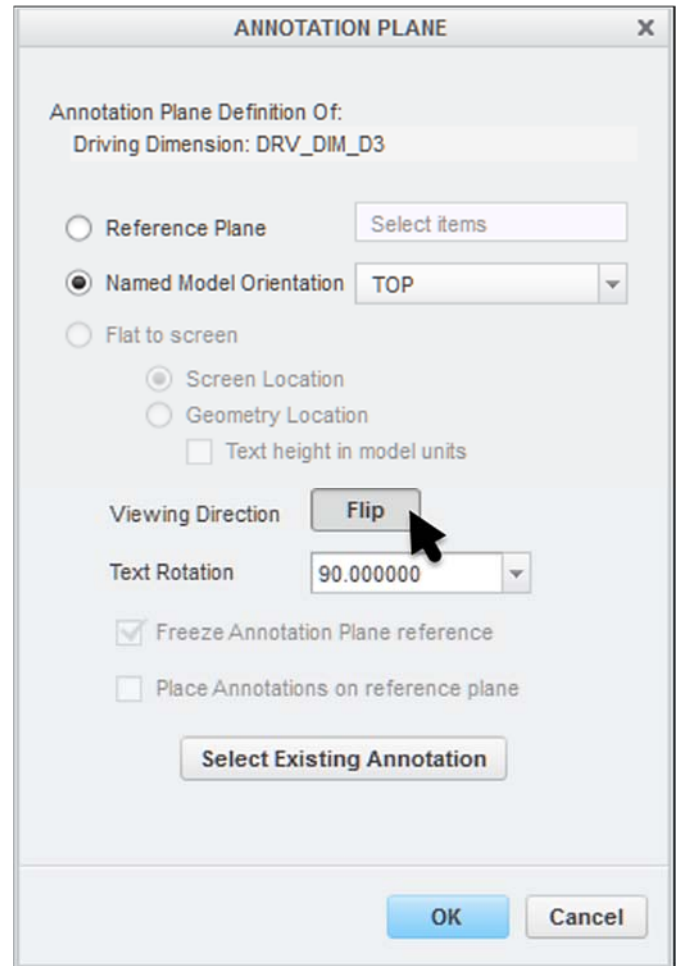
                    OK   Cancel
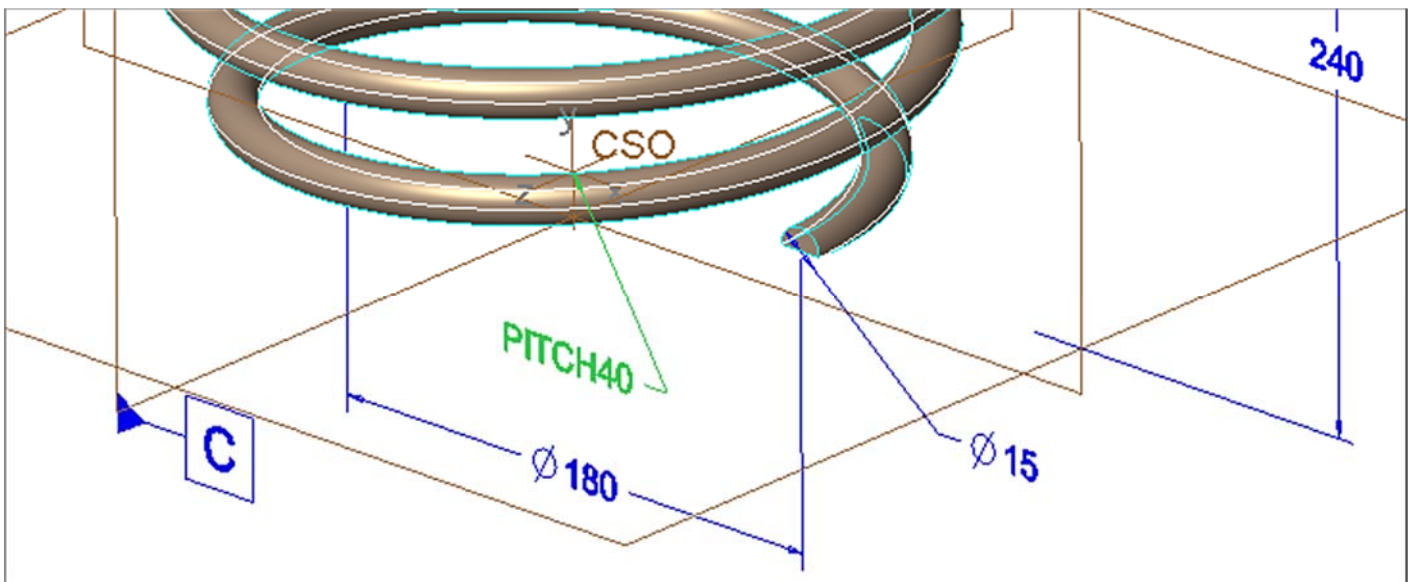
Figure 16.18(d) Named Model Orientation: TOP

Figure 16.18(e) Moved Dimensions

Click: [icon] **Open the View Manager > Orient** tab **>** click on **Annotation(+) > RMB > Save > OK > Close >**
Annotation tab **>** select the **360** dimension [Fig. 16.19(a)] **>** press **RMB > Properties > Display** tab [Fig.
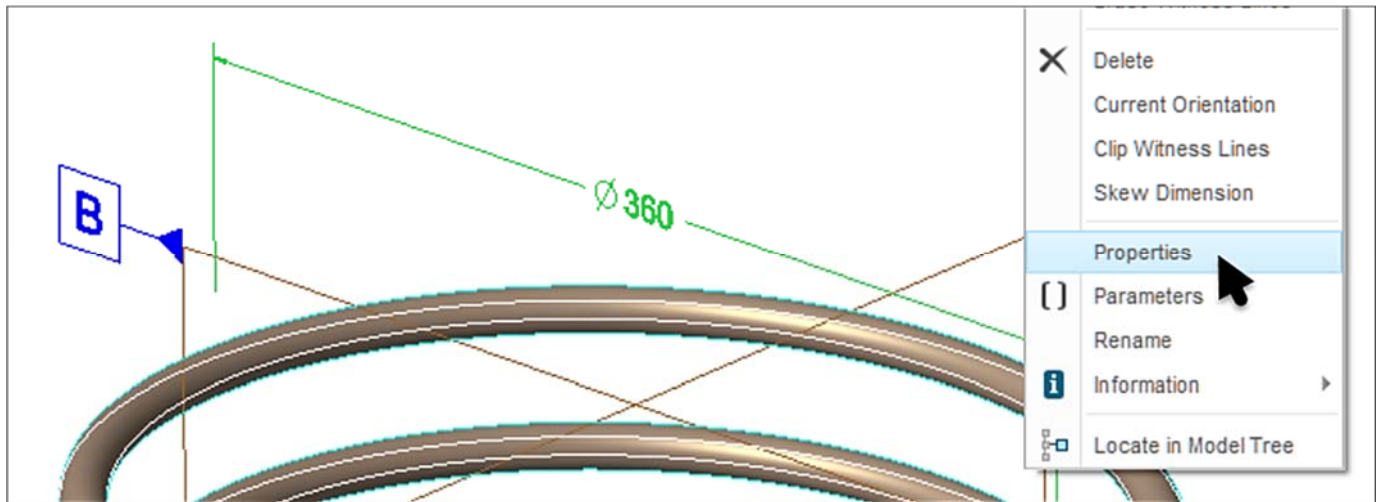16.19(b)] **>** *type* **PITCH DIAMETER > OK > LMB** to deselect **> Ctrl+S** [Fig. 16.19(c)]

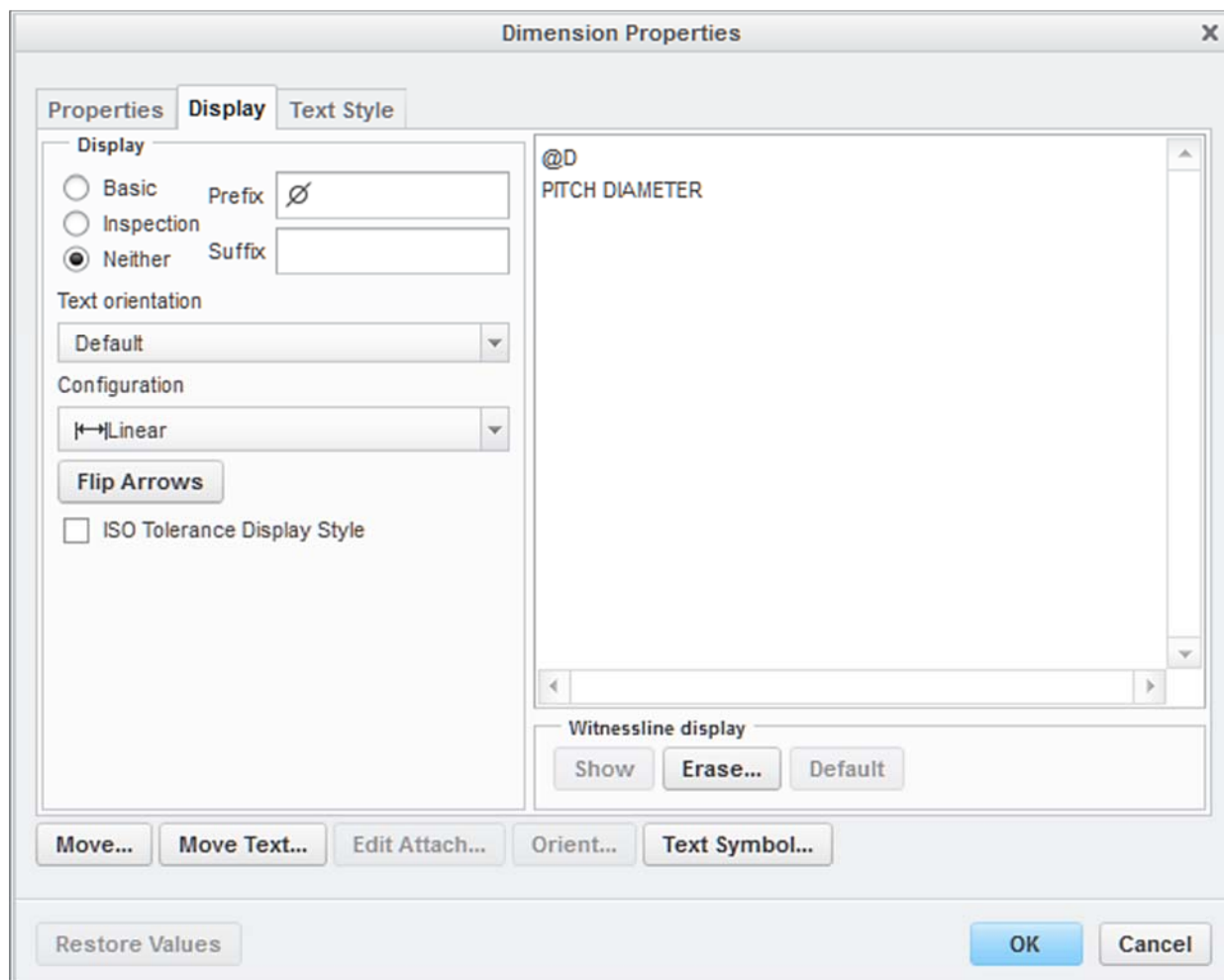**Figure 16.19(a)** Dimension Properties (your options list may appear differently)

Figure 16.19(b) Dimension Properties Dialog Box, Display Tab *(type in added text)*
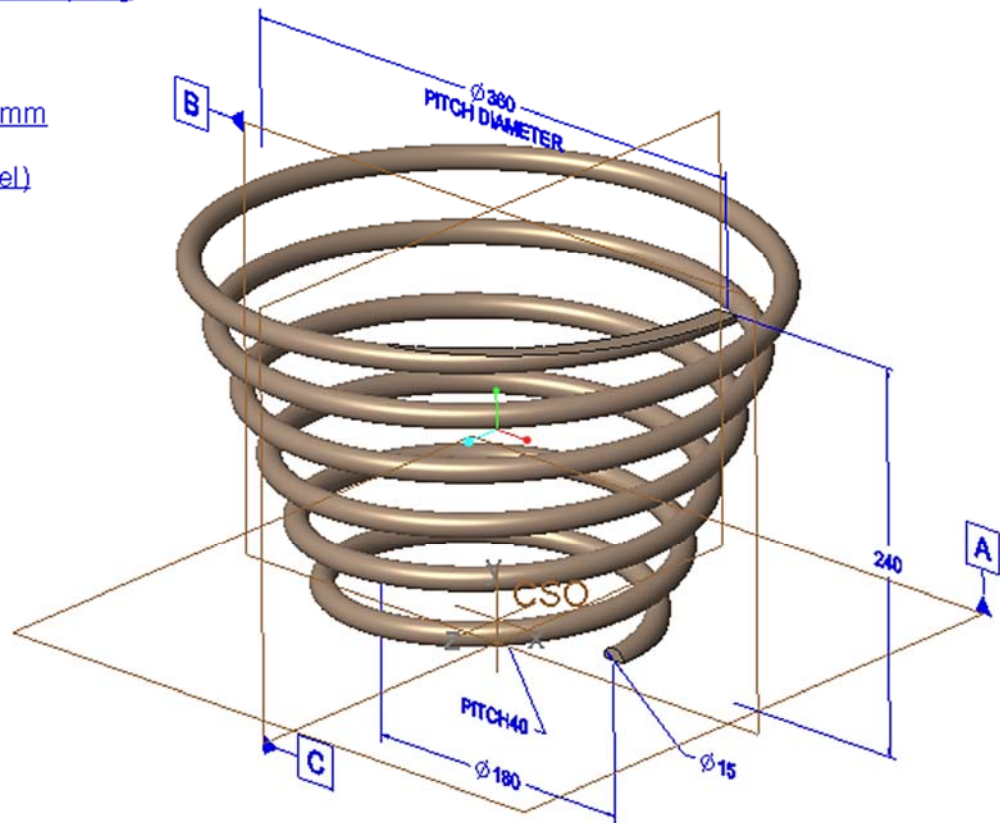
Figure 16.19(c) Annotated Part

Click: [icon] **Annotation Feature >** [icon] tab [Fig. 16.20(a)]
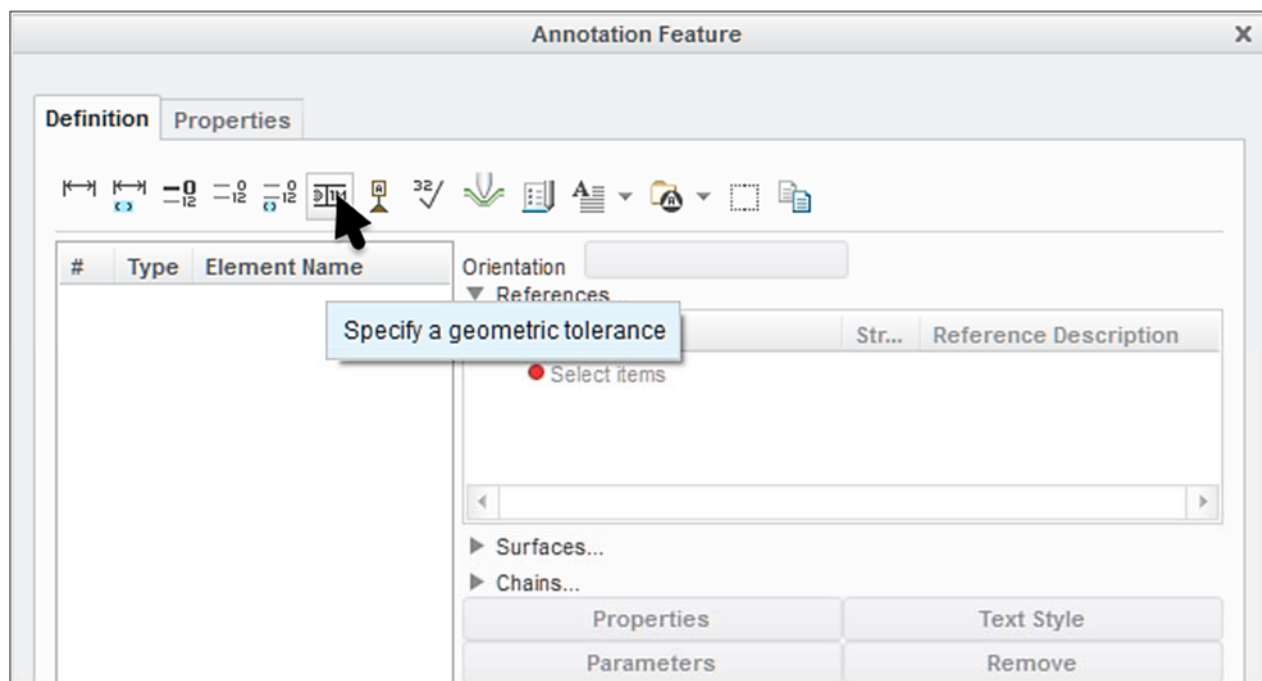
Figure 16.20(a) Annotation Feature Dialog Box, Geometric Tolerance

Click: [//] **Parallelism** [Fig. 16.20(b)] **>** Reference: Type: [▼] **> Surface >** select the surface [Fig. 16.20(c)]
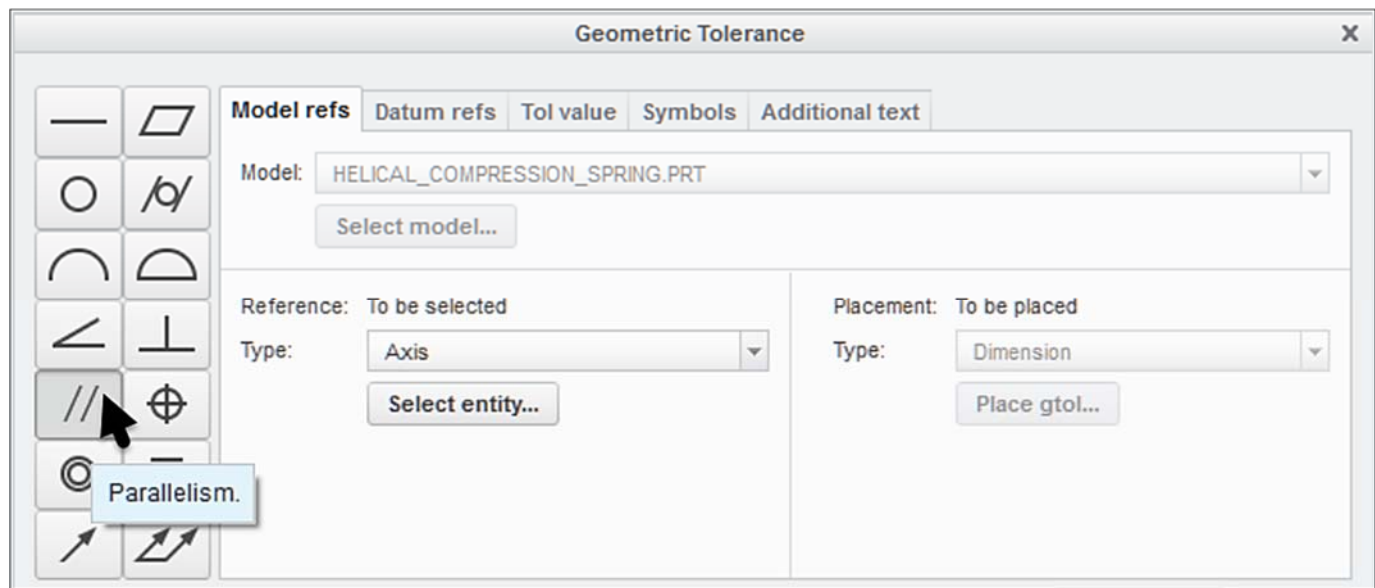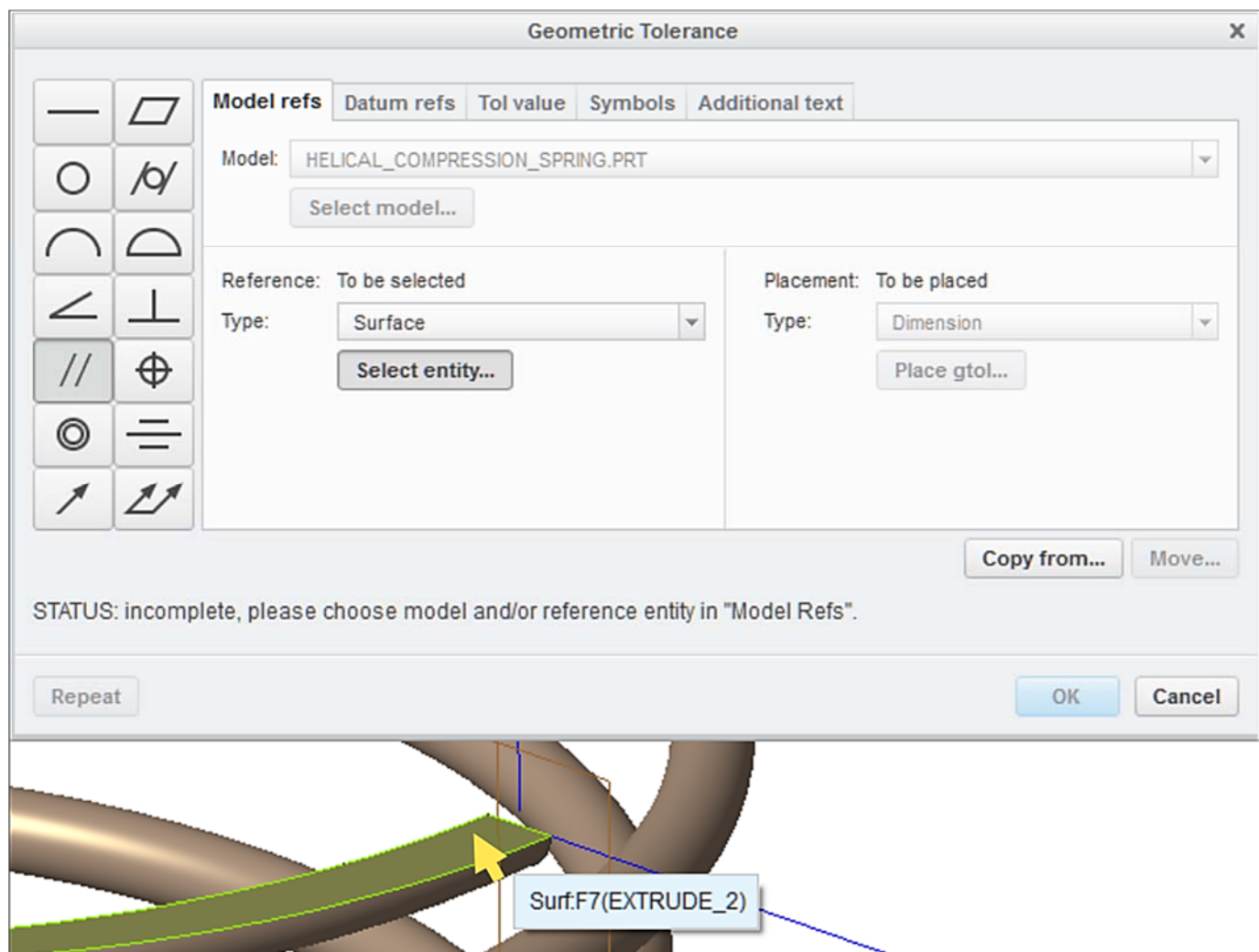

Figure 16.20(b) Geometric Tolerance Dialog Box

Figure 16.20(c) Select the Ground (Cut) Surface

Click: Placement: To Be Placed- Type [▼] > **Dimension** > **Place Gtol** [Fig. 16.20(d)] > select the **240** dimension [Fig. 16.20(e)] > **Datum Refs** tab > **Primary** tab > Basic [▼] > **A** [Fig. 16.20(f)] > **OK**
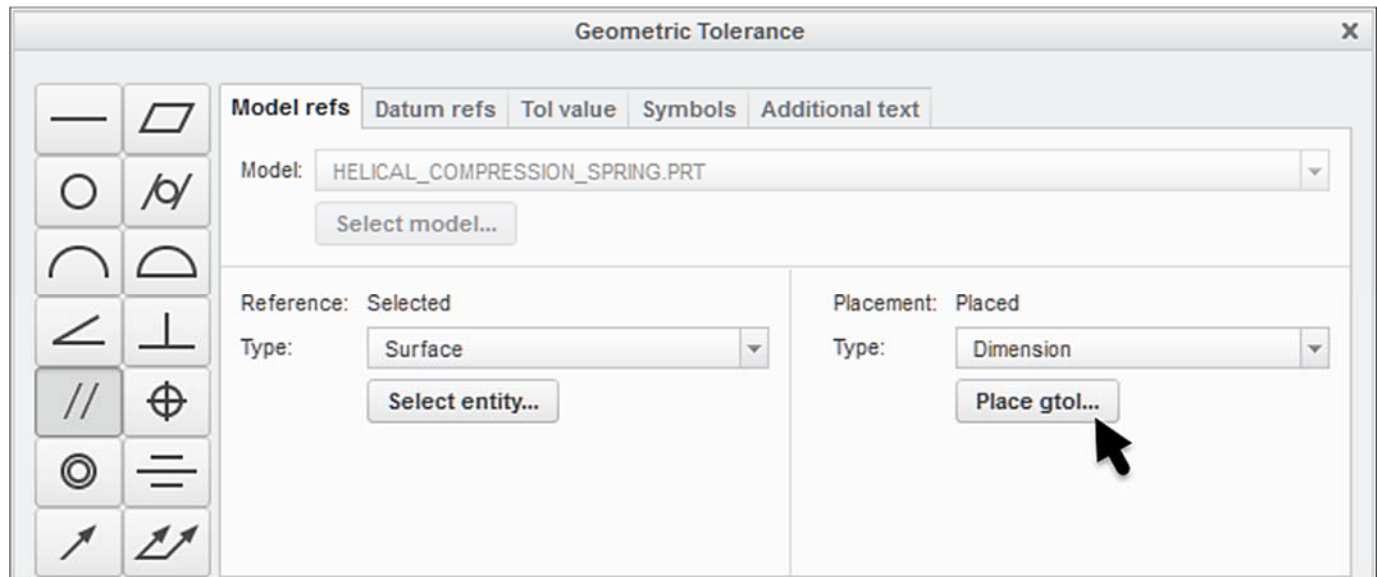
Figure 16.20(d) Type **Dimension**
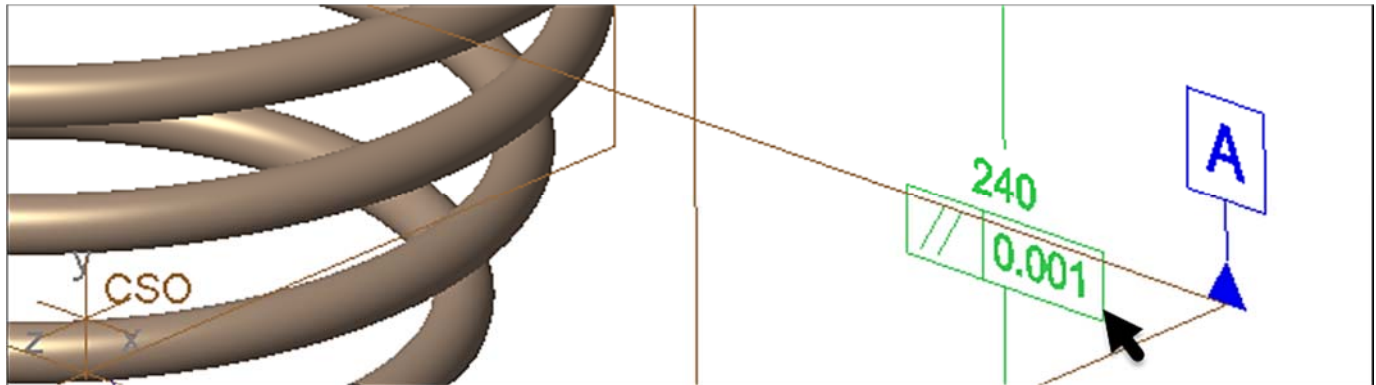
Figure 16.20(e) Select the **240** Dimension
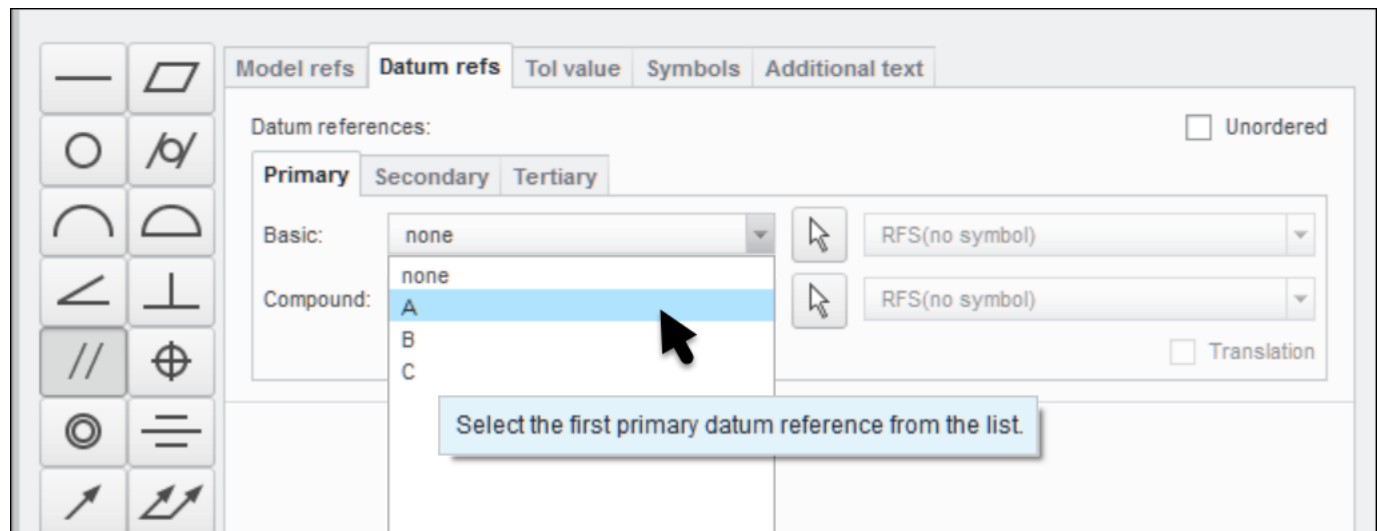
Figure 16.20(f) Select Primary **A**

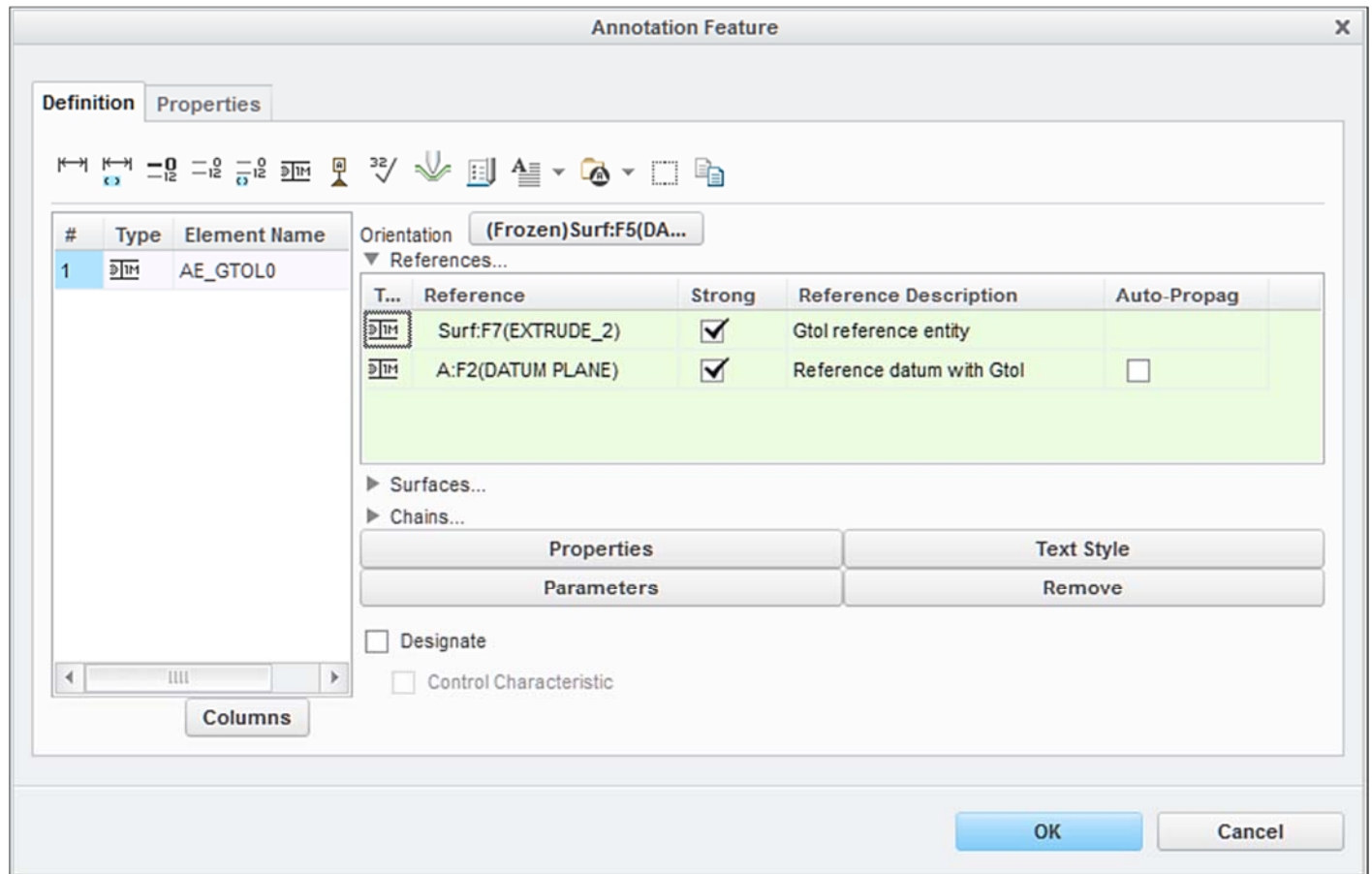Click: **OK** [Fig. 16.20(g)] > **Ctrl+S** [Fig. 16.20(h)] > **LMB** to deselect
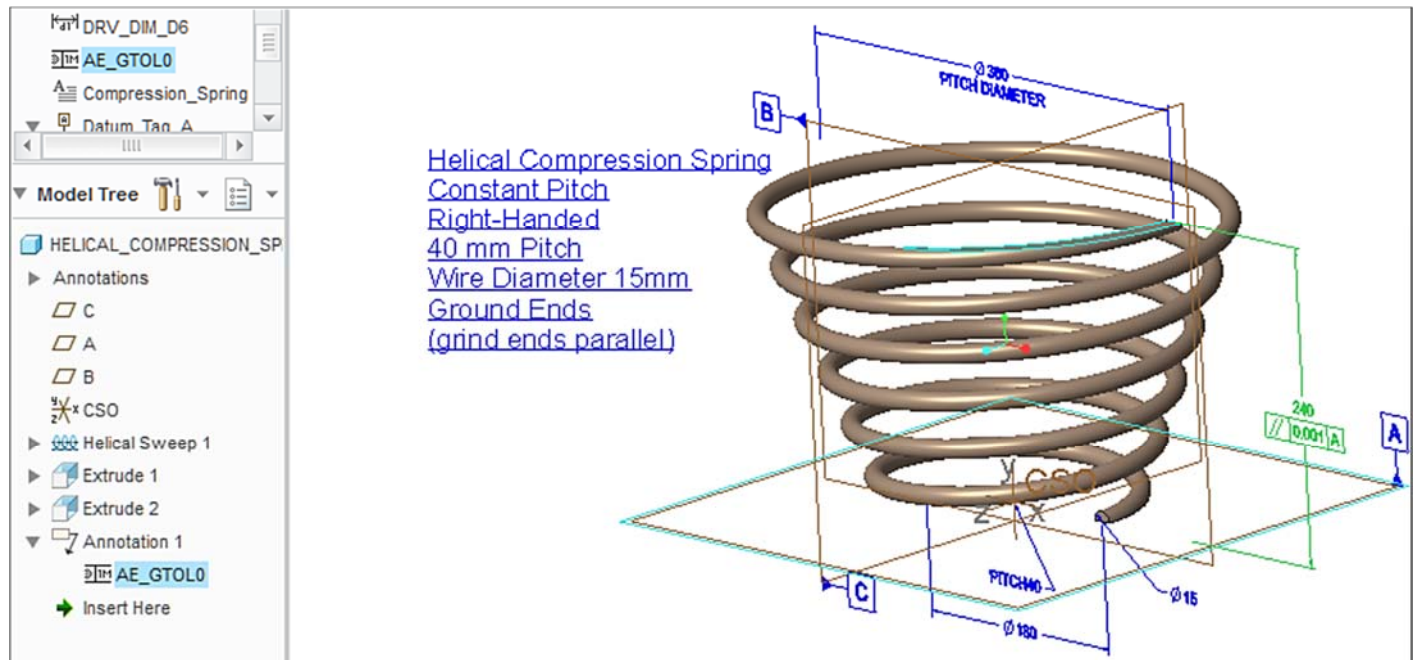

Figure 16.20(g) Annotation Feature Dialog Box

Figure 16.20(h) Annotation Feature Completed

Click: **FRONT > RMB > Set >** ▭⁊ **Annotation Feature >** ³²√ **Create a surface finish** [Fig. 16.21(a)] **> double-click** on the **machined** folder > ▭ standard1.sym **> Preview** [Fig. 16.21(b)]
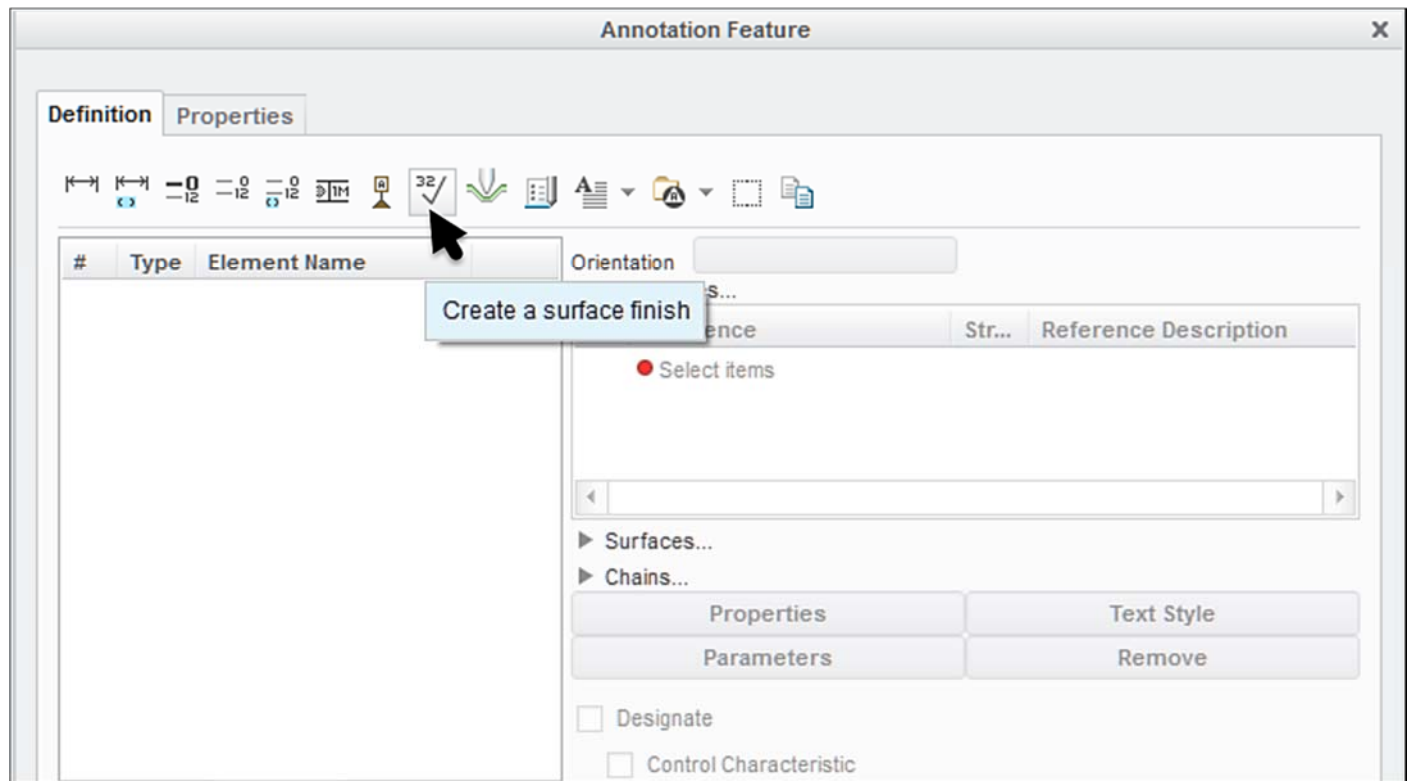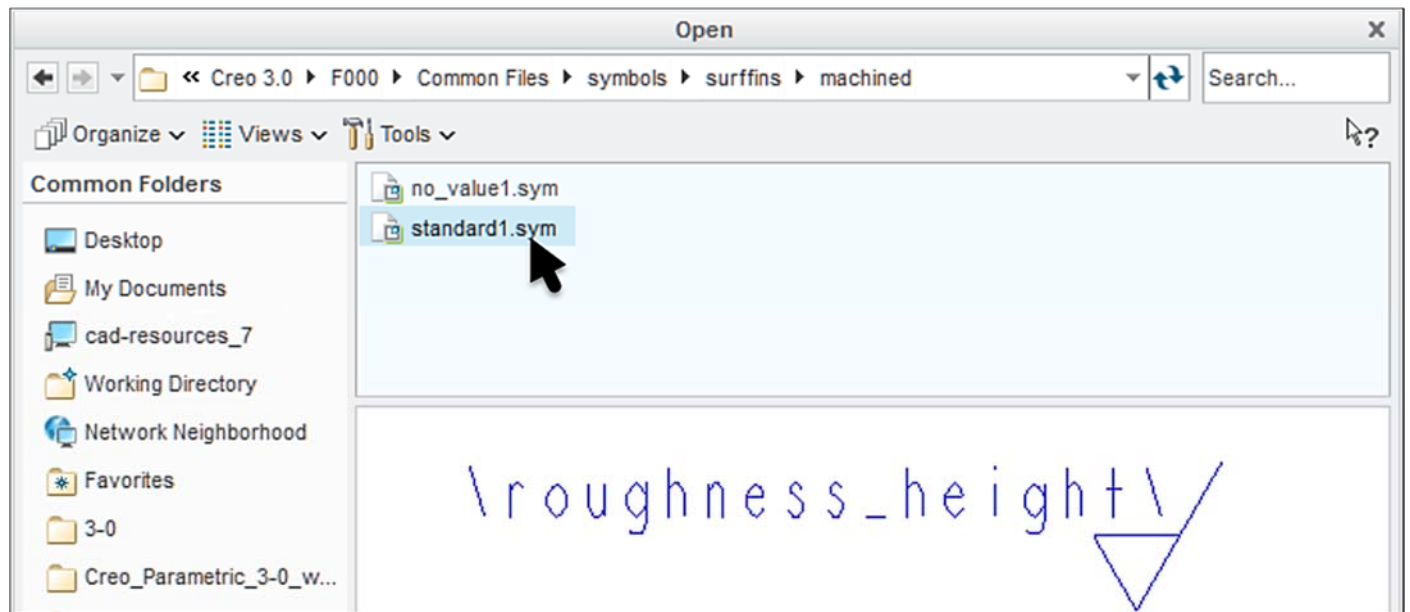

Figure 16.21(a) Create a Surface Finish

Figure 16.21(b) Preview of Surface Symbol **standard1.sym**

Click: **Open** [Fig. 16.21(c)] and the Surface Finish dialog box opens with its References collector active > select the ground surface [Fig. 16.21(d)]
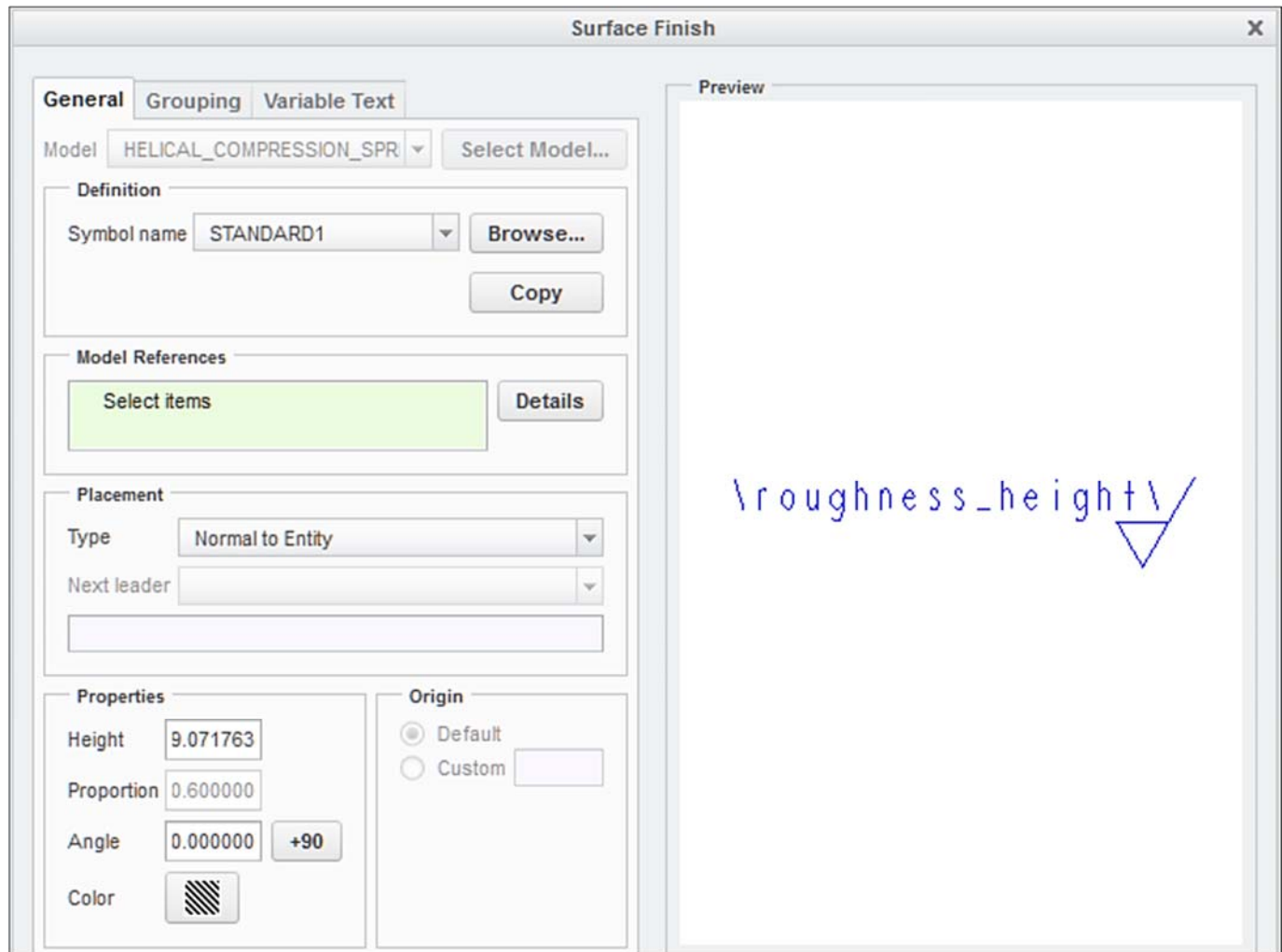

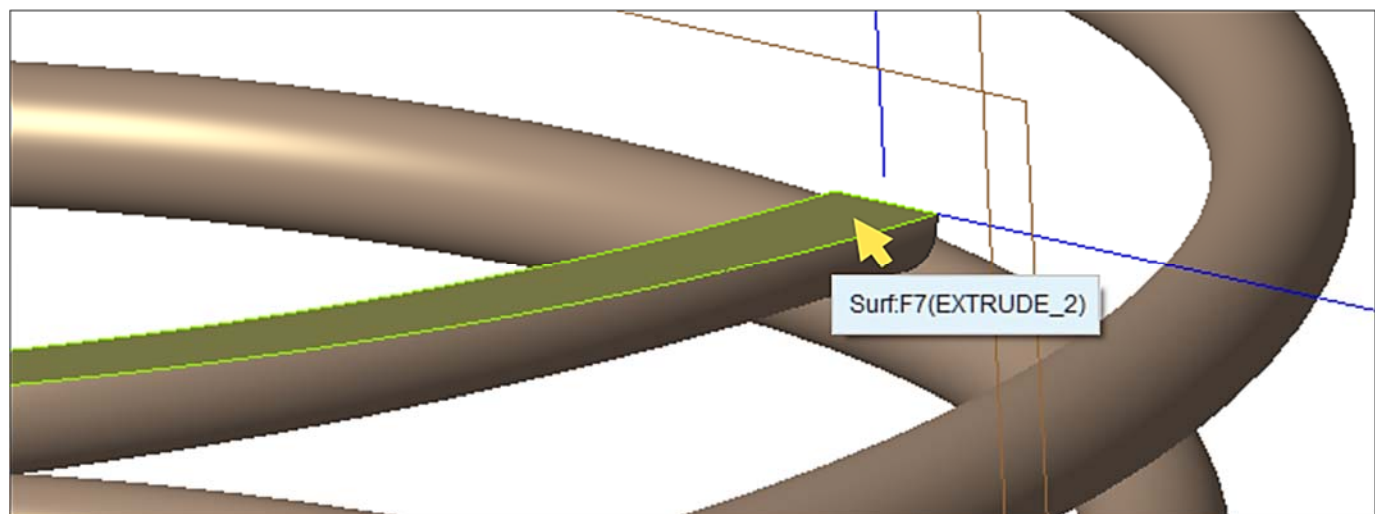
Figure 16.21(c) Surface Finish Dialog Box

Figure 16.21(d) Reference Surface

Click inside the Placement collector for Attachment references [Fig. 16.21(e)] > select the symbol position on the cut surface [Figs. 16.21(f-g)] > **MMB** [Fig. 16.21(h)]
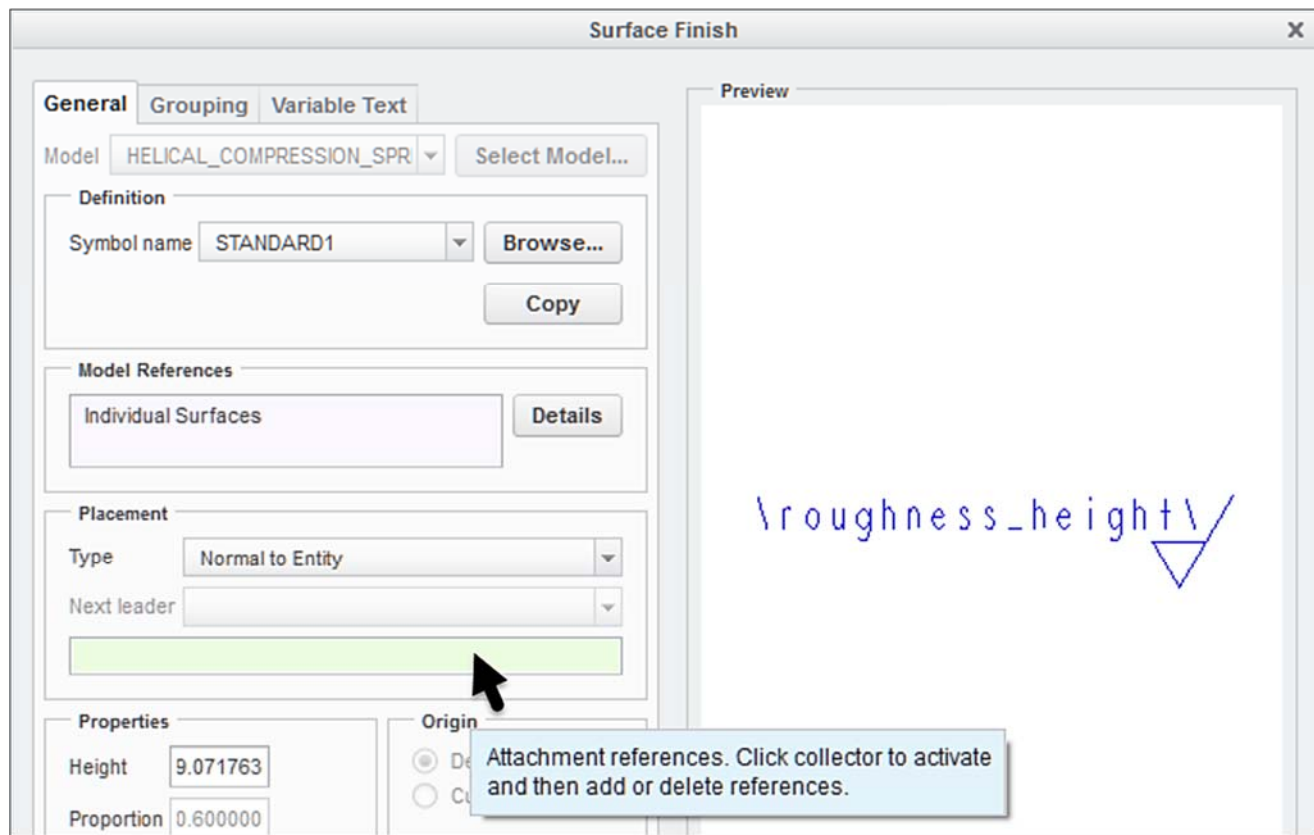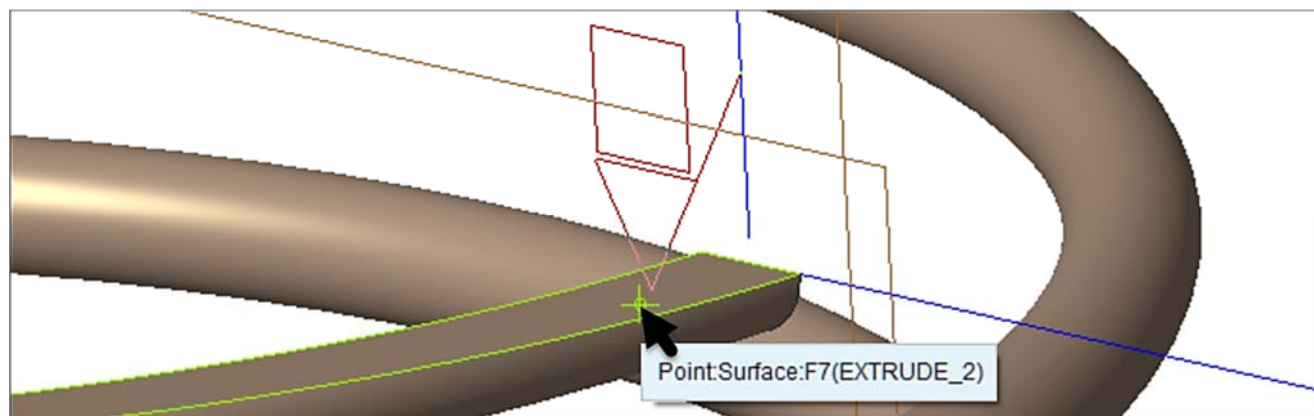

Figure 16.21(e) Placement Collector

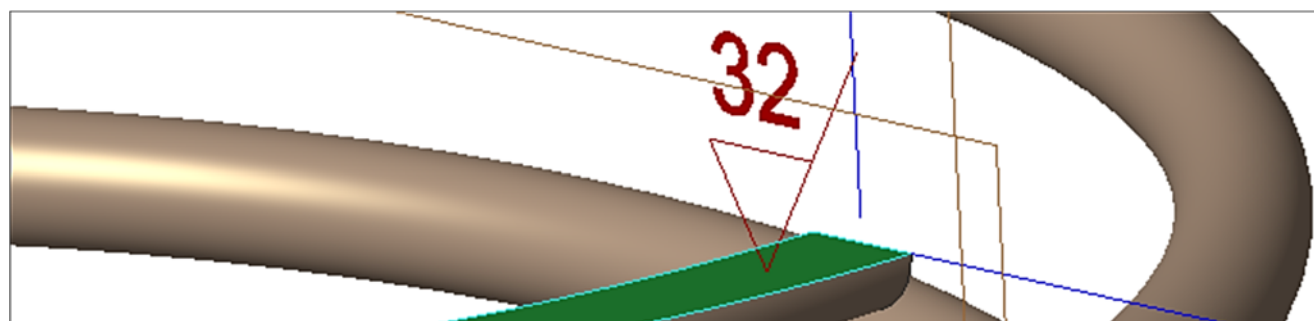**Figure 16.21(f)** Select the Surface Finish Symbol Position

Figure 16.21(g) Completed Symbol Placement

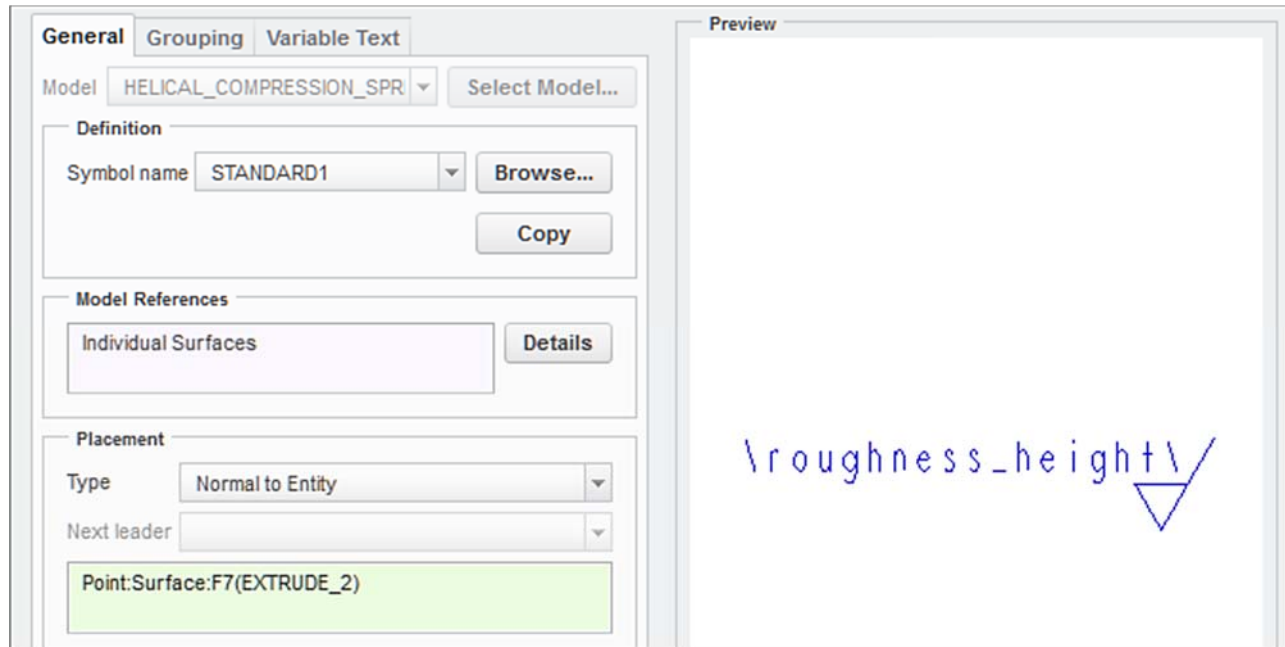Click: **Variable Text** tab **> MMB > OK** [Fig. 16.21(i)]

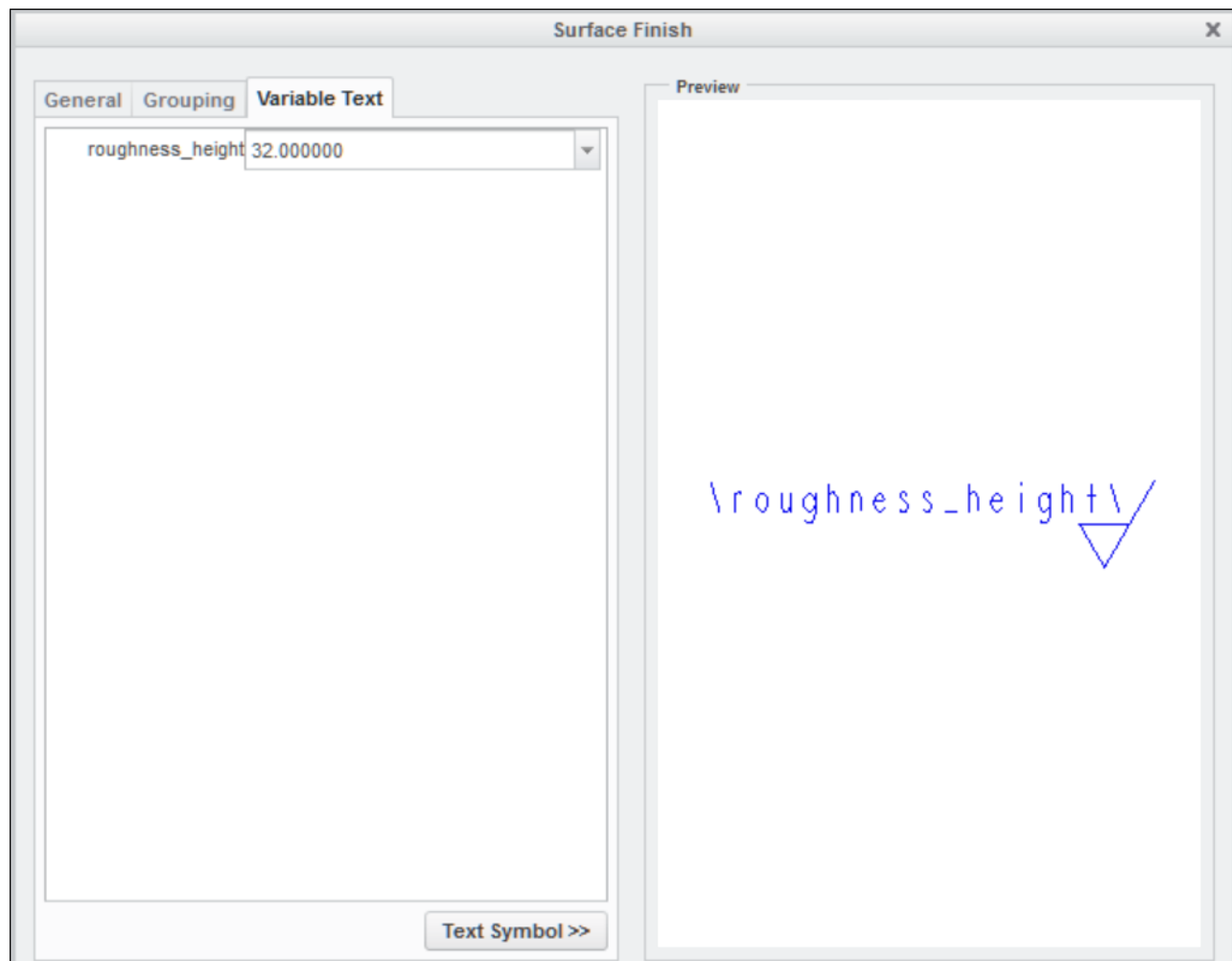**Figure 16.21(h)** General Tab Selections Completed

Figure 16.21(i) Variable Text Tab

Click: ![32 checkmark icon] tab [Fig. 16.21(j)] > repeat the process to create an annotation feature finish symbol on the opposite end of the spring [Fig. 16.21(k)] > **OK** from the Annotation Feature dialog box > **LMB** to deselect
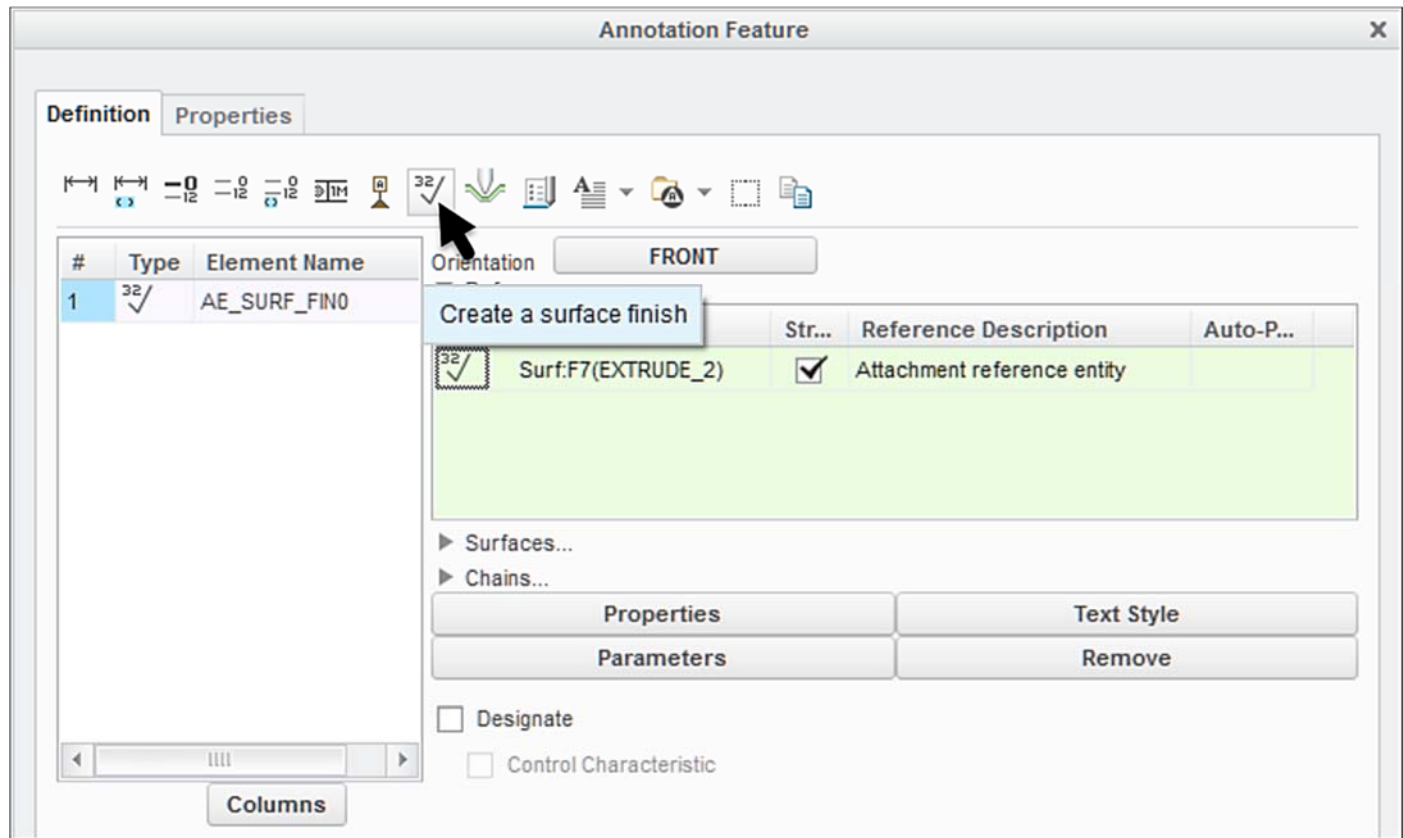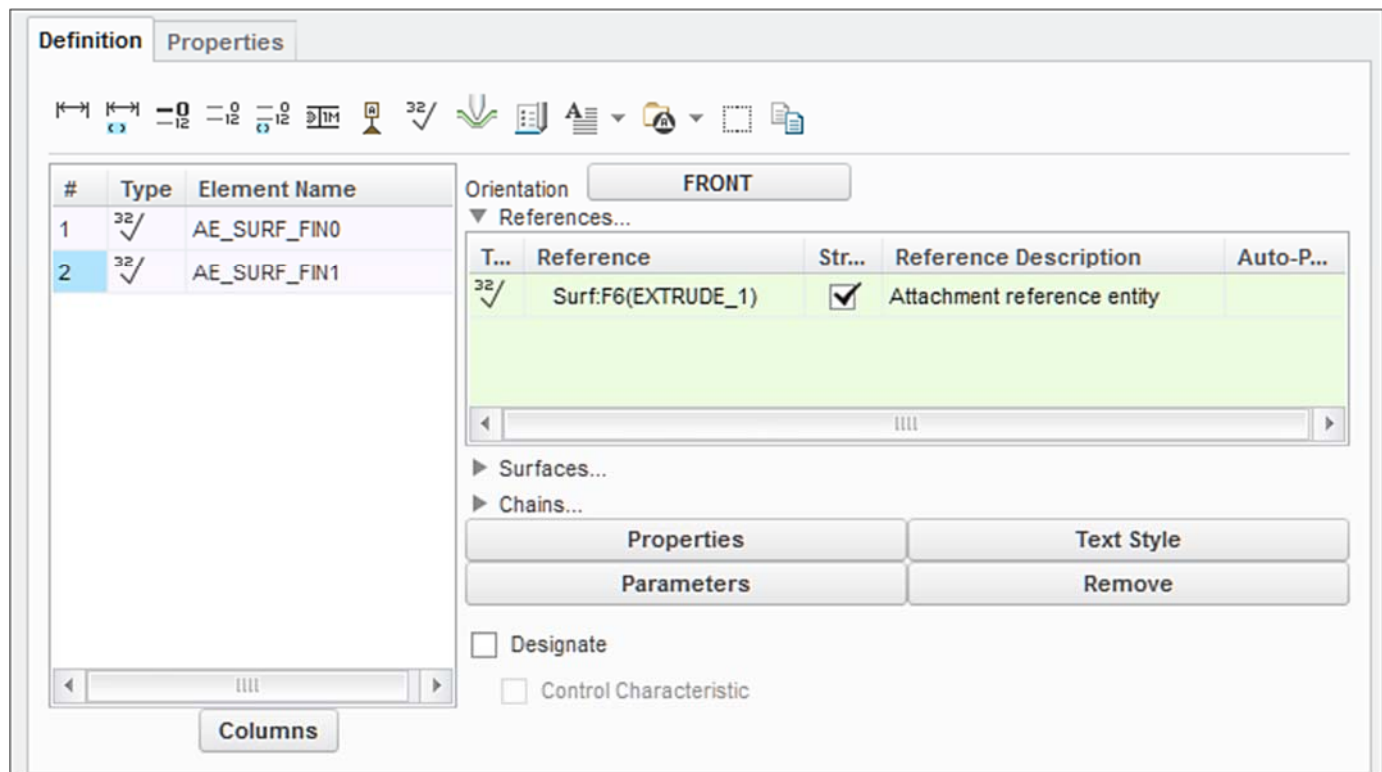

Figure 16.21(j) Annotation Feature Dialog Box

Figure 16.21(k) Second Surface Finish Annotation

719

Click: > **View tab** > **View Manager** > **Orient** > click on **Annotations(+)** > **Save** > **OK** > **Close** > **FLAT TO**

SCREEN > [cube icon] **FRONT** (Fig. 16.22) > [save icon] > **File** > **Manage File** > **Delete Old Versions** > **Enter** > **File**
> **Save As** > Type [icon] > **Zip File (*.zip)** > **OK** > upload > **File** > **Close** > **File** > **Exit** > **Yes**
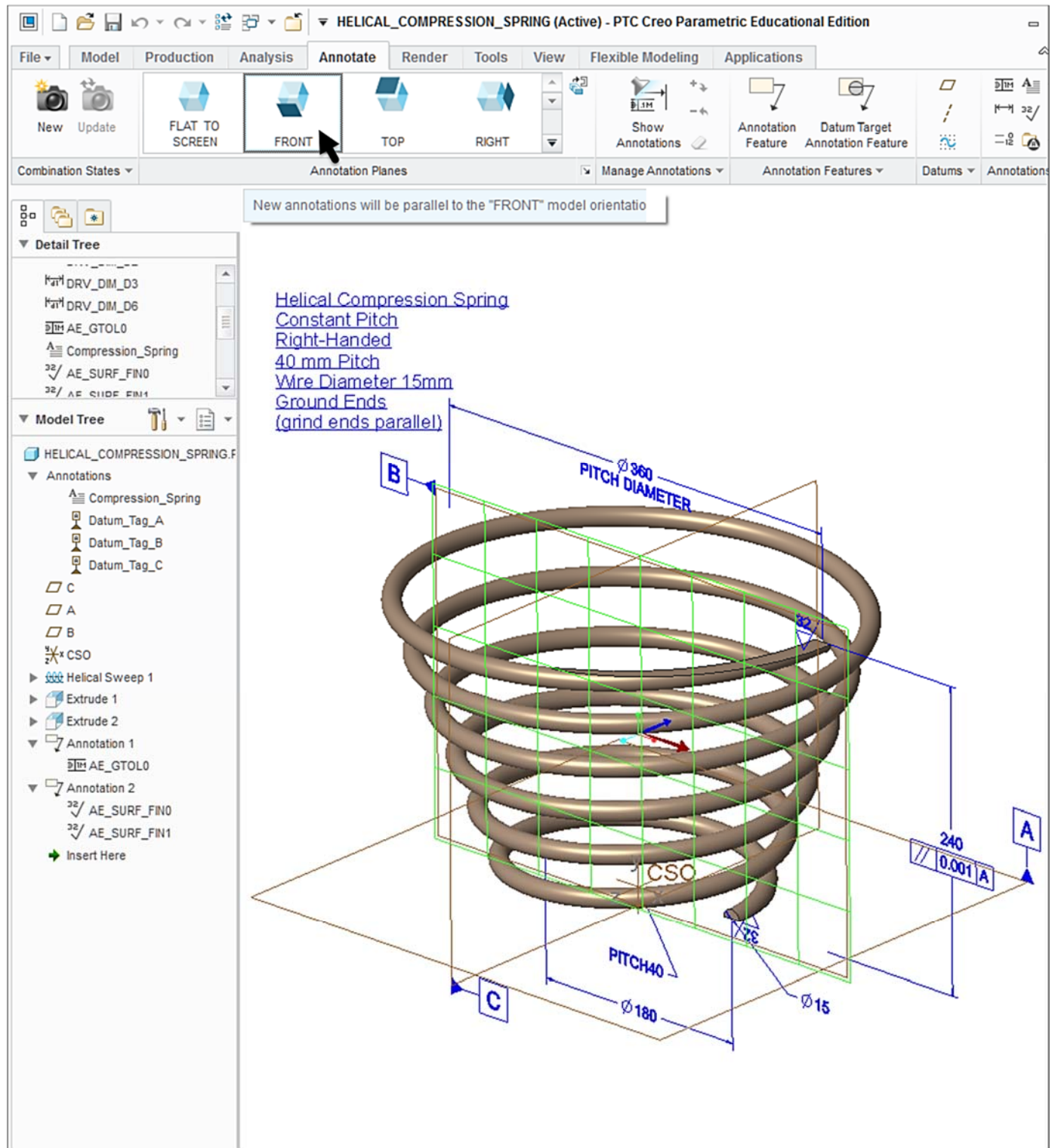

Figure 16.22 Active Annotation Orientation Plane (Grid Shown in Green)

Download a different spring project from ***www.cad-resources.com***.