Chapter 2
The Direct Stiffness Method

Learning Objectives

When you have completed this lesson, you will be able to:
- Understand system equations for truss elements.
- Understand the setup of a Stiffness Matrix.
- Apply the Direct Stiffness Method.
- Create an Extruded solid model using 1-DEAS.
- Use the Display Viewing commands.
- Use the Sketch In Place command.
- Create Cutout features.
2.1 Introduction

The direct stiffness method is used mostly for Linear Static analysis. The development of the direct stiffness method originated in the 1940s and is generally considered the fundamental of finite element analysis. Linear Static analysis is appropriate if deflections are small and vary only slowly. Linear Static analysis omits time as a variable. It also excludes plastic action and deflections that change the way loads are applied. The direct stiffness method for Linear Static analysis follows the laws of Statics and the laws of Strength of Materials.

![Stress-Strain diagram of typical ductile material](image)

This chapter introduces the fundamentals of finite element analysis by illustrating an analysis of a one-dimensional truss system using the direct stiffness method. The main objective of this chapter is to present the classical procedure common to the implementation of structural analysis. The direct stiffness method utilizes matrices and matrix algebra to organize and solve the governing system equations. Matrices, which are ordered arrays of numbers that are subjected to specific rules, can be used to assist the solution process in a compact and elegant manner. Of course, only a limited discussion of the direct stiffness method is given here, but we hope that the focused practical treatment will provide a strong basis for understanding the procedure to perform finite element analysis with I-DEAS.

The later sections of this chapter demonstrate the procedure to create a solid model using I-DEAS Master Modeler. The step-by-step tutorial introduces the I-DEAS user interface and serves as a preview to some of the basic modeling techniques demonstrated in the later chapters.
2.2 One-dimensional Truss Element

The simplest type of engineering structure is the truss structure. A truss member is a slender (the length is much larger than the cross section dimensions) two-force member. Members are joined by pins and only have the capability to support tensile or compressive loads axially along the length. Consider a uniform slender prismatic bar (shown below) of length L, cross-sectional area A, and elastic modulus E. The ends of the bar are identified as nodes. The nodes are the points of attachment to other elements. The nodes are also the points for which displacements are calculated. The truss element is a two-force member element, forces are applied to the nodes only, and the displacements of all nodes are confined to the axes of elements.

In this initial discussion of the truss element, we will consider the motion of the element to be restricted to the horizontal axis (one-dimensional). Forces are applied along the X-axis and displacements of all nodes will be along the X-axis.

For the analysis, we will establish the following sign conventions:

1. Forces and displacements are defined as positive when they are acting in the positive X direction as shown in the above figure.

2. The position of a node in the undeformed condition is the finite element position for that node.

If equal and opposite forces of magnitude F are applied to the end nodes, from the elementary strength of materials, the member will undergo a change in length according to the equation:

$$\delta = \frac{FL}{EA}$$

This equation can also be written as $$\delta = \frac{F}{K}$$, which is similar to Hooke’s Law used in a linear spring. In a linear spring, the symbol K is called the spring constant or stiffness of the spring. For a truss element, we can see that an equivalent spring element can be used to simplify the representation of the model, where the spring constant is calculated as $$K = \frac{EA}{L}$$.  

![Diagram of a truss element with forces and displacements](image-url)
2-4 Introduction to Finite Element Analysis

Force-Displacement Curve of a Linear Spring

We will use the general equations of a single one-dimensional truss element to illustrate the formulation of the stiffness matrix method:

By using the Relative Motion Analysis method, we can derive the general expressions of the applied forces ($F_1$ and $F_2$) in terms of the displacements of the nodes ($X_1$ and $X_2$) and the stiffness constant ($K$).

1. Let $X_1 = 0$,

Based on Hooke’s law and equilibrium equation:

$$\begin{cases} F_2 = K X_2 \\ F_1 = -F_2 = -K X_2 \end{cases}$$
2. Let $X_2 = 0$,

Based on Hooke’s law and equilibrium:

\[
\begin{align*}
F_1 &= K X_1 \\
F_2 &= -F_1 = -K X_1
\end{align*}
\]

Using the *Method of Superposition*, the two sets of equations can be combined:

\[
\begin{align*}
F_1 &= K X_1 - K X_2 \\
F_2 &= -K X_1 + K X_2
\end{align*}
\]

The two equations can be put into matrix form as follows:

\[
\begin{bmatrix}
F_1 \\
F_2
\end{bmatrix} = \begin{bmatrix} +K & -K \\ -K & +K \end{bmatrix} \begin{bmatrix} X_1 \\ X_2 \end{bmatrix}
\]

This is the general force-displacement relation for a two-force member element, and the equations can be applied to all members in an assemblage of elements. The following example illustrates a system with three elements.

**Example 2.1:**
Consider an assemblage of three of these two-force member elements. (Motion is restricted to one-dimension, along the X-axis.)
The assemblage consists of three elements and four nodes. The Free Body Diagram of the system with node numbers and element numbers labeled:

Consider now the application of the general force-displacement relation equations to the assemblage of the elements.

Element 1:

\[
\begin{bmatrix}
F_1 \\
F_{21}
\end{bmatrix} = \begin{bmatrix}
+K_1 & -K_1 \\
-K_1 & +K_1
\end{bmatrix} \begin{bmatrix}
X_1 \\
X_2
\end{bmatrix}
\]

Element 2:

\[
\begin{bmatrix}
F_{22} \\
F_3
\end{bmatrix} = \begin{bmatrix}
+K_2 & -K_2 \\
-K_2 & +K_2
\end{bmatrix} \begin{bmatrix}
X_2 \\
X_3
\end{bmatrix}
\]

Element 3:

\[
\begin{bmatrix}
F_{23} \\
F_4
\end{bmatrix} = \begin{bmatrix}
+K_3 & -K_3 \\
-K_3 & +K_3
\end{bmatrix} \begin{bmatrix}
X_2 \\
X_4
\end{bmatrix}
\]

Expanding the general force-displacement relation equations into an Overall Global Matrix (containing all nodal displacements):

Element 1:

\[
\begin{bmatrix}
F_1 \\
F_{21} \\
0 \\
0
\end{bmatrix} = \begin{bmatrix}
+K_1 & -K_1 & 0 & 0 \\
-K_1 & +K_1 & 0 & 0 \\
0 & 0 & 0 & 0 \\
0 & 0 & 0 & 0
\end{bmatrix} \begin{bmatrix}
X_1 \\
X_2 \\
X_3 \\
X_4
\end{bmatrix}
\]
Element 2:
\[
\begin{bmatrix}
0 \\
F_{22} \\
F_3 \\
0
\end{bmatrix} = \begin{bmatrix}
0 & 0 & 0 & 0 \\
0 & +K_2 & -K_2 & 0 \\
0 & -K_2 & +K_2 & 0 \\
0 & 0 & 0 & 0
\end{bmatrix} \begin{bmatrix}
X_1 \\
X_2 \\
X_3 \\
X_4
\end{bmatrix}
\]

Element 3:
\[
\begin{bmatrix}
0 \\
F_{23} \\
0 \\
F_4
\end{bmatrix} = \begin{bmatrix}
0 & 0 & 0 & 0 \\
0 & +K_3 & 0 & -K_3 \\
0 & 0 & 0 & 0 \\
0 & -K_3 & 0 & +K_3
\end{bmatrix} \begin{bmatrix}
X_1 \\
X_2 \\
X_3 \\
X_4
\end{bmatrix}
\]

Summing the three sets of general equation: (Note $F_2 = F_{21} + F_{22} + F_{32}$)
\[
\begin{bmatrix}
F_1 \\
F_2 \\
F_3 \\
F_4
\end{bmatrix} = \begin{bmatrix}
K_1 & -K_1 & 0 & 0 \\
-K_1 & (K_1+K_2+K_3) & -K_2 & -K_3 \\
0 & -K_2 & K_2 & 0 \\
0 & -K_3 & 0 & +K_3
\end{bmatrix} \begin{bmatrix}
X_1 \\
X_2 \\
X_3 \\
X_4
\end{bmatrix}
\]

Once the Overall Global Stiffness Matrix is developed for the structure, the next step is to substitute boundary conditions and solve for the unknown displacements. At every node in the structure, either the externally applied load or the nodal displacement is needed as a boundary condition. We will demonstrate this procedure with the following example.

**Example 2.2:**

Given:

- $K_1 = 50$ lb/in
- $K_2 = 30$ lb/in
- $K_3 = 70$ lb/in
- $F = 40$ lbs.

Node 1 $\rightarrow$ Node 2 $\rightarrow$ Node 3 $\rightarrow$ Node 4

$F = 40$ lbs.
Find: Nodal displacements and reaction forces.

Solution:

From example 2.1, the overall global force-displacement equation set:

$$\begin{bmatrix} F_1 \\ F_2 \\ F_3 \\ F_4 \end{bmatrix} = \begin{bmatrix} 50 & -50 & 0 & 0 \\ -50 & (50+30+70) & -30 & -70 \\ 0 & -30 & 30 & 0 \\ 0 & -70 & 0 & 70 \end{bmatrix} \begin{bmatrix} X_1 \\ X_2 \\ X_3 \\ X_4 \end{bmatrix}$$

Next, apply the known boundary conditions to the system: the right-end of element 2 and element 3 are attached to the vertical wall; therefore, these two nodal displacements ($X_3$ and $X_4$) are zero.

$$\begin{bmatrix} F_1 \\ F_2 \\ F_3 \\ F_4 \end{bmatrix} = \begin{bmatrix} 50 & -50 & 0 & 0 \\ -50 & (50+30+70) & -30 & -70 \\ 0 & -30 & 30 & 0 \\ 0 & -70 & 0 & 70 \end{bmatrix} \begin{bmatrix} X_1 \\ X_2 \\ 0 \\ 0 \end{bmatrix}$$

The two displacements we need to solve the system are $X_1$ and $X_2$. Remove any unnecessary columns in the matrix:

$$\begin{bmatrix} F_1 \\ F_2 \\ F_3 \\ F_4 \end{bmatrix} = \begin{bmatrix} 50 & -50 \\ -50 & 150 \\ 0 & -30 \\ 0 & -70 \end{bmatrix} \begin{bmatrix} X_1 \\ X_2 \end{bmatrix}$$

Next, include the applied loads into the equations. The external load at Node 1 is 40 lbs. and there is no external load at Node 2.

$$\begin{bmatrix} 40 \\ 0 \\ F_3 \\ F_4 \end{bmatrix} = \begin{bmatrix} 50 & -50 \\ -50 & 150 \\ 0 & -30 \\ 0 & -70 \end{bmatrix} \begin{bmatrix} X_1 \\ X_2 \end{bmatrix}$$

The Matrix represents the following four simultaneous system equations:

$$
\begin{align*}
40 &= 50 X_1 - 50 X_2 \\
0 &= -50 X_1 + 150 X_2 \\
F_3 &= 0 X_1 - 30 X_2 \\
F_4 &= 0 X_1 - 70 X_2
\end{align*}
$$
From the first two equations, we can solve for $X_1$ and $X_2$:

$$X_1 = 1.2 \text{ in.}$$
$$X_2 = 0.4 \text{ in.}$$

Substituting these known values into the last two equations, we can now solve for $F_3$ and $F_4$:

$$F_3 = 0X_1 - 30X_2 = -30 \times 0.4 = 12 \text{ lbs.}$$
$$F_4 = 0X_1 - 70X_2 = -70 \times 0.4 = 28 \text{ lbs.}$$

From the above analysis, we can now reconstruct the Free Body Diagram (FBD) of the system:

The above sections illustrated the fundamental operation of the direct stiffness method, the classical finite element analysis procedure. As can be seen, the formulation of the global force-displacement relation equations is based on the general force-displacement equations of a single one-dimensional truss element. The two-force-member element (truss element) is the simplest type of element used in FEA. The procedure to formulate and solve the global force-displacement equations is straightforward, but somewhat tedious. In real-life application, the use of a truss element in one-dimensional space is rare and very limited. In the next chapter, we will expand the procedure to solving two-dimensional truss frameworks.

The following sections illustrate the procedure to create a solid model using I-DEAS Master Modeler. The step-by-step tutorial introduces the basic I-DEAS user-interface and the tutorial serves as a preview to some of the basic modeling techniques demonstrated in the later chapters.
2.3 Basic Solid Modeling using I-DEAS Master Modeler

One of the methods to create solid models in I-DEAS Master Modeler is to create a two-dimensional shape and then extrude the two-dimensional shape to define a volume in the third dimension. This is an effective way to construct three-dimensional solid models since many designs are in fact the same shape in one direction. Computer input and output devices used today are largely two-dimensional in nature, which makes this modeling technique quite practical. This method also conforms to the design process that helps the designer with conceptual design along with the capability to capture the design intent. I-DEAS Master Modeler provides many powerful modeling tools and there are many different approaches available to accomplish modeling tasks. We will start by introducing the basic two-dimensional sketching and parametric modeling tools.

The L-Block design

Starting I-DEAS

1. Select the I-DEAS icon or type “ideas” at your system prompt to start I-DEAS. The I-DEAS Start window will appear on the screen.
2. Fill in and select the items as shown below:

<table>
<thead>
<tr>
<th>Project Name: (Your account Name)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Model File Name: L_Block</td>
</tr>
<tr>
<td>Application: Design</td>
</tr>
<tr>
<td>Task: Master Modeler</td>
</tr>
</tbody>
</table>

3. After you click OK, two warning windows will appear to tell you that a new model file will be created. Click OK on both windows as they come up.

4. Next, I-DEAS will display four windows, the graphics window, the prompt window, the list window and the icon panel. A line of quick help text appears at the bottom of the graphics window as you move the mouse cursor over the icons.

### Units Setup

When starting a new model, the first thing we should do is determine the set of units we would like to use. I-DEAS displays the default set of units in the list window.

1. Use the left-mouse-button and select the Options menu in the icon panel as illustrated.

2. Select Units.

3. Inside the graphics window, pick Inch (pound f) from the pop-up menu. The set of units is stored with the model file when you save.
Applying the BORN Technique

The basic concept of the “Base Orphan Reference Node” (BORN) technique is to create a Cartesian coordinate system as the first feature prior to creating any solid features. With the Cartesian coordinate system established, we then have three mutually perpendicular datum planes (the XY, YZ, and ZX planes) available for use as sketching planes.

1. Choose Coordinate Systems in the icon panel. (The icon is located in the first row of the task icon panel.)

2. In the prompt window, the message “Pick entity for coordinate system to reference” is displayed. Select the workplane by left-clicking once on any of the edges.

3. In the prompt window, the message “Select coordinate system component to define (Done)” is displayed. Press the ENTER key or the middle-mouse-button to accept the default settings.

By default, the user coordinate system is aligned to the world coordinate system.

4. Choose Name Parts… in the icon panel. (The icon is located in the last row of the application specific icon panel.)

5. The message “Pick part to name” is displayed in the prompt window. Pick any edge of the coordinate system we just created. The Name window appears.

6. In the Name window, we can assign or change the name, number, and bin of a part. Each field on the window is limited to 80 characters. In the Name box, type in “L_Block.”

7. Click on the OK icon to proceed.

8. The message “Pick part to name (Done)” is displayed in the prompt window. Press the ENTER key or the middle-mouse-button to end the Name Parts command.
Display Icon Panel

The Display icon panel contains various icons to handle different viewing operations. These icons control the screen display, such as the view scale, the view angle, redisplay, and shaded and hidden line displays.

View icons:

*Front, Side, Top, Bottom, Isometric, and Perspective:* These six icons are the standard view icons. Selecting any of these icons will change the viewing angle. Try each one as you read its description below.

- **Front View (X-Y Workplane)**
- **Right Side View**
- **Top View**
- **Bottom View**
- **Isometric View**
- **Perspective View**
Creating the Base Feature

1. Choose **Isometric View** in the display viewing icon panel.

2. Choose **Zoom-All** in the display viewing icon panel.

3. Choose **Sketch in Place** in the icon panel.

4. In the prompt window, the message “*Pick plane to sketch on*” is displayed. Pick one of the edges of the **XY plane** of the coordinate system. Note that even though the workplane is at the same orientation of the XY-plane of the coordinate system, selecting the XY-plane assures the sketch is attached to the established part, **Guide_Block**.

2-D Sketching

In this tutorial we will begin with building a 2-D sketch, as shown below.
I-DEAS provides many powerful tools for sketching 2-D shapes. In the previous generation CAD programs, exact dimensional values were needed during construction, and adjustments to dimensional values were quite difficult once the model was built. In I-DEAS, we can now treat the sketch as if it is being done on a piece of napkin, and it is the general shape of the design that we are more interested in defining. The I-DEAS part model contains more than just the final geometry. It also contains the design intent that governs what will happen when geometry changes. The design philosophy of “shape before size” is implemented through the use of I-DEAS Variational Geometry. This allows the designer to construct solid models in a higher level and leave all the geometric details to I-DEAS. We will first create a rough sketch, by using some of the visual aids available, and then update the design through the associated control parameters.

1. Pick Polylines in the icon panel. (The icon is located in the second row of the task specific icon panel. If the icon is not on top of the stack, press and hold down the left-mouse-button on the displayed icon to display all the choices. Select the desired icon by clicking with the left-mouse-button when the icon is highlighted.)

Graphics Cursors

Notice the cursor has changed from an arrow to a crosshair when graphical input is expected. Look in the prompt window for a description of what you are to choose. The cursor will change to a double crosshair when there is a possibly ambiguous choice. You can press the middle-mouse-button to accept the highlighted pick or choose a different item.

2. The message “Locate start” is displayed in the prompt window. Left-click a starting point of the shape, roughly at the center of the graphics window; it could be inside or outside of the displayed grids. In I-DEAS, the sketch plane actually extends into infinity. As you move the graphics cursor, you will see a digital readout in the upper left corner of the graphics window. The readout gives you the cursor location, the line length, and the angle of the line measured from horizontal. Move the cursor around and you will also notice different symbols appear along the line as it occupies different positions.
Dynamic Navigator

I-DEAS provides us with visual clues as the cursor is moved across the screen; this is the I-DEAS Dynamic Navigator. The Dynamic Navigator displays different symbols to show you alignments, perpendicularities, tangencies, etc. The Dynamic Navigator is also used to capture the design intent by creating constraints where they are recognized. The Dynamic Navigator displays the governing geometric rules as models are built.

- **Vertical** indicates a line is vertical
- **Horizontal** indicates a line is horizontal
- **Dashed line** indicates the alignment to the center point or endpoint of an entity
- **Parallel** indicates a line is parallel to other entities
- **Perpendicular** indicates a line is perpendicular to other entities
- **Endpoint** indicates the cursor is at the endpoint of an entity
- **Intersection** indicates the cursor is at the intersection point of two entities
- **Center** indicates the cursor is at the centers or midpoints of entities
- **Tangent** indicates the cursor is at tangency points to curves
3. Move the graphics cursor directly below point 1. Pick the second point when the vertical constraint is displayed and the length of the line is about 2 inches.

4. Move the graphics cursor horizontally to the right of point 2. The perpendicular symbol indicates when the line from point 2 to point 3 is perpendicular to the vertical line. Left-click to select the third point. Notice that dimensions are automatically created as you sketch the shape. These dimensions are also constraints, which are used to control the geometry. Different dimensions are added depending upon how the shape is sketched. Do not worry about the values not being exactly what we want. We will modify the dimensions later.

5. Move the graphics cursor directly above point 3. Do not place this point in alignment with the midpoint of the other vertical line. An additional constraint will be added if they are aligned. Left-click the fourth point directly above point 3.

6. Move the graphics cursor to the left of point 4. Again, watch the displayed symbol to apply the proper geometric rule that will match the design intent. A good rule of thumb is to exaggerate the features during the initial stage of sketching. For example, if you want to construct a line that is five degrees from horizontal, it would be easier to sketch a line that is 20 to 30 degrees from horizontal. We will be able to adjust the actual angle later. Left-click the fifth point horizontally from point 4.

7. Move the graphics cursor directly above the last point. Watch the different symbols displayed and place the point in alignment with point 1. Left-click the sixth point directly above point 5.

8. Move the graphics cursor near the starting point of the sketch. Notice the Dynamic Navigator will jump to the endpoints of entities. Left-click point 1 again to end the sketch.
9. In the prompt window, you will see the message “Locate start.” By default, I-DEAS remains in the Polylines command and expects you to start a new sequence of lines.

10. Press the **ENTER** key or the middle-mouse-button to end the Polylines command.

**Dynamic Viewing Functions**

I-DEAS provides a special user interface called *Dynamic Viewing* that enables convenient viewing of the entities in the graphics window. The *Dynamic Viewing* functions are controlled with the function keys on the keyboard and the mouse.

**Panning – F1 and the mouse**

Hold the **F1** function key down, and move the mouse to pan the display. This allows you to reposition the display while maintaining the same scale factor of the display. This function acts as if you are using a video camera. You control the display by moving the mouse.

**Zooming – F2 and the mouse**

Hold the **F2** function key down, and move the mouse vertically on the screen to adjust the scale of the display. Moving upward will reduce the scale of the display, making the entities display smaller on the screen. Moving downward will magnify the scale of the display.
Basic Editing – Using the Eraser

One of the advantages of using a CAD system is the ability to remove entities without leaving any marks. We will delete one of the lines using the Delete command.

1. Pick **Delete** in the icon panel. (The icon is located in the last row of the application icon panel. The icon is a picture of an eraser at the end of a pencil.)

2. In the prompt window, the message “Pick entity to delete” appears. Pick the line as shown in the below figure.

3. The prompt window now reads “Pick entity to delete (done).” Press the **ENTER** key or the **middle-mouse-button** to indicate you are done picking entities to be deleted.

4. In the prompt window, the message “OK to delete 1 curve, 1 constraint and 1 dimension? (Yes)” will appear. The “1 constraint” is the parallel constraint created by the Dynamic Navigator.

5. Press **ENTER**, or pick Yes in the pop-up menu to delete the selected line. The constraints and dimensions are used as geometric control variables. When the geometry is deleted, the associated control features are also removed.

6. In the prompt window, you will see the message “Pick entity to delete.” By default, I-DEAS remains in the Delete command and expects you to select additional entities to be erased.

7. Press the **ENTER** key or the **middle-mouse-button** to end the Delete command.
Creating a Single Line

Now we will add another line at the same location by using the *Lines* command.

1. Pick **Lines** in the icon panel. (The icon is located in the same stack as the *Polylines* icon.) Press and hold down the left-mouse-button on the *Polylines* icon to display the available choices. Select the *Lines* command with the left-mouse-button when the option is highlighted.

2. The message “*Locate start*” is displayed in the prompt window. Move the graphics cursor near *point 1* and, as the *endpoint* symbol is displayed, pick with the left-mouse-button.

3. Move the graphics cursor near *point 2* and click the **left-mouse-button** when the *endpoint* symbol is displayed.

   - Notice the *Dynamic Navigator* creates the parallel constraint and the dimension as the geometry is constructed.

4. The message “*Locate start*” is displayed in the prompt window. Press the **ENTER** key or use the middle-mouse-button to end the *Lines* command.

Consideration of Design Intent

While creating the sketch, it is very important to keep in mind the design intent. Always consider the functionality of the part and key features of the design. Using I-DEAS, we can accomplish and maintain the design intent at all levels of the design process.
The dimensions automatically created by I-DEAS might not always match with the designer’s intent. For example, in our current design, we may want to use the vertical distance between the top two horizontal lines as a key dimension. Even though it is a very simple calculation to figure out the corresponding length of the vertical dimension at the far right, for more complex designs it might not be as simple, and to do additional calculations is definitely not desirable. The next section describes re-dimensioning the sketch.

Step 2: Apply/Delete/Modify constraints and dimensions

As the sketch is made, I-DEAS automatically applies some of the geometric constraints (such as horizontal, parallel and perpendicular) to the sketched geometry. We can continue to modify the geometry, apply additional constraints, and/or define the size of the existing geometry. In this example, we will illustrate deleting existing dimensions and adding new dimensions to describe the sketched entities. To maintain our design intent, we will first remove the unwanted dimension and then create the desirable dimension.

1. Pick **Delete** in the icon panel. (The icon is located in the last row of the application icon panel.)

2. Pick the dimension as shown.
3. Press the **ENTER** key or the **middle-mouse-button** to accept the selection.

4. Pick **Yes** in the *popup menu* or press the **ENTER** key or the **middle-mouse-button** to delete the selected dimension and end the *Delete* command.

### Creating Desired Dimensions

1. Choose **Dimension** in the icon panel. The message *“Pick the first entity to dimension”* is displayed in the prompt window.

2. Pick the **top-horizontal-line** as shown in the below figure.

3. Pick the **second horizontal line** as shown.

4. Place the text to the right of the model.

5. Press the **ENTER** key or the **middle-mouse-button** to end the **Dimension** command.

- In I-DEAS, the **Dimension** command will create a linear dimension (distance in between the two lines) if two parallel lines are selected. Selecting two lines that are not parallel will create an angular dimension (angle in between the two lines.)
Modifying Dimensional Values

Next we will adjust the dimensional values to the desired values. One of the main advantages of using a feature-based parametric solid modeler, such as I-DEAS, is the ability to easily modify existing entities. The operation of modifying dimensional values will demonstrate implementation of the design philosophy of “shape before size.” In I-DEAS, several options are available to modify the dimensional values. In this chapter, we will demonstrate two of the options using the Modify command. The Modify command icon is located in the second row of the application icon panel; the icon is a picture of an arrowhead with a long tail.

1. Choose Modify in the icon panel. (The icon is located in the second row of the application icon panel. If the icon is not on top of the stack, press and hold down the left-mouse-button on the displayed icon, then select the Modify icon.) The message “Pick entity to modify” is displayed in the prompt window.

2. Pick the dimension as shown (the number might be different than displayed). The selected dimension will be highlighted. The Modify Dimension window appears.

   In the Modify Dimension window, the value of the selected dimension is displayed and also identified by a Name in the format of “Dxx,” where the “D” indicates it is a dimension and the “xx” is a number incremented automatically as dimensions are added. You can change both the name and the value of the dimension by clicking and typing in the appropriate boxes.

3. Modify the dimensional value to 3.25 as shown.

4. Click on the OK button to accept the value you have entered.
I-DEAS will adjust the size of the object based on the new value entered.

5. On your own, click on the top horizontal dimension and adjust the dimensional value to 0.625.

6. Press the **ENTER** key or the **middle-mouse-button** to end the *Modify Dimension* command.

The size of our design is automatically adjusted by I-DEAS based on the dimensions we have entered. I-DEAS uses the dimensional values as control variables and the geometric entities are modified accordingly. This approach of rough sketching the shape of the design first then finalizing the size of the design is known as the “shape before size” approach.

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**Pre-selection of Entities**

I-DEAS provides a flexible graphical user interface that allows users to select graphical entities BEFORE the command is selected (*Pre-selection*), or AFTER the command is selected (*Post-selection*). The procedure we have used so far is the *Post-selection* option. To pre-select one or more items to process, hold down the **SHIFT** key while you pick. Selected items will stay highlighted. You can **deselect** an item by selecting the item again. The item will be toggled on and off by each click. Another convenient feature of pre-selection is that the selected items remain selected after the command is executed.

1. Pre-select all of the dimensions by holding down the **SHIFT** key and clicking the **left-mouse-button** on each dimension value.

   **PRE-SELECT** **SHIFT** + **LEFT-mouse-button**

2. Select the **Modify** icon. The *Dimensions* window appears.
3. Move the *Dimensions* window around by “clicking and dragging” the window’s title area with the left-mouse-button. You can also use the *Dynamic Viewing* functions (activate the graphics window first) to adjust the scale and location of the entities displayed in the graphics window (F1 and the mouse, F2 and the mouse).

![Diagram](image1)

![Diagram](image2)

4. Click on one of the dimensions in the pop-up window. The selected dimension will be highlighted in the graphics window. Type in the desired value for the selected dimension. **DO NOT** hit the **ENTER** key. Select another dimension from the list to continue modifying. Modify all of the dimensional values to the values as shown.

![Diagram](image3)

5. Click the **OK** button to accept the values you have entered and close the *Dimensions* window.

- I-DEAS will now adjust the size of the shape to the desired dimensions. The design philosophy of “shape before size” is implemented quite easily. The geometric details are taken care of by I-DEAS.
Changing the Appearance of Dimensions

The top-horizontal-dimension we modified is displayed as 0.62, instead of the entered value (0.625.) We can adjust the appearance of dimensions by using the Appearance command.

1. Choose Appearance in the icon panel. (The icon is located in the second row of the application icon panel. If the icon is not on top of the stack, press and hold down the left-mouse-button on the displayed icon, then select the Appearance icon.) The message “Pick entity to modify” is displayed in the prompt window.

2. Pick the top horizontal dimension as shown in the figure.

3. The message “Pick entity to modify (Done)” is displayed in the prompt window. Press the ENTER key or use the middle-mouse-button to accept the selected object.

4. In the Product & Manufacturing Information window, switch on the Autoscale option.

5. Click on the Units/Decimal Places...button. The Units & Decimal Places window appears.
6. Set the decimal places to 3 to display three digits after the decimal point.

7. Click on the OK button to exit the Units & Decimal Places window.

8. Click on the OK button to exit the Product & Manufacturing Information window.

9. Press the ENTER key or the middle-mouse-button to end the Appearance command.

Repositioning Dimensions

1. Choose Move in the icon panel. (The icon is located in the first row of the application icon panel.) The message “Pick entity to move” is displayed in the prompt window.

2. Select any of the dimensions displayed on the screen.

3. Move the cursor to position the dimension in a new location. Left-click once to accept the new location.

4. Press the ENTER key or the middle-mouse-button to end the Move command.
Step 3: Completing the Base Solid Feature

Now that the 2D sketch is completed, we will proceed to the next step: create a 3D part from the 2D profile. Extruding a 2D profile is one of the common methods that can be used to create 3D parts. We can extrude planar faces along a path. We can also specify a height value and a tapered angle. In I-DEAS, each face has a positive side and a negative side; the current face we're working on is set as the default positive side.

1. Choose **Extrude** in the icon panel. The Extrude icon is located in the fifth row of the task specific icon panel. Press and hold down the left-mouse-button on the icon to display all the choices and slide the mouse up and down to switch between different options. In the prompt window, the message “Pick curve or section” is displayed.

2. Pick any edge of the 2-D shape. By default, the **Extrude** command will automatically select all segments of the shape that form a closed region. Notice the different color signifying the selected segments.

3. Notice the I-DEAS prompt “Pick curve to add or remove. (Done).” We can select more geometry entities or deselect any entity that has been selected. Picking the same geometry entity will again toggle the selection of the entity “on” or “off” with each left-mouse-button click. Press the **ENTER** key to accept the selected entities.

4. The **Extrude Section** window will appear on the screen. Enter **2.5** as the **extrusion distance** and confirm that the **New part** option is switched on as shown.

5. Click on the **OK** button to accept the settings and extrude the 2-D section into a 3-D solid.

- Notice all of the dimensions disappeared from the screen. All of the dimensional values and geometric constraints are stored in the database by I-DEAS and they can be brought up at any time.
Display Viewing Commands

3-D Dynamic Rotation – F3 and the mouse

The I-DEAS Dynamic Viewing feature allows users to do “real-time” rotation of the display. Hold the F3 function key down and move the mouse to rotate the display. This allows you to rotate the displayed model about the screen X (horizontal), Y (vertical), and Z (perpendicular to the screen) axes. Start with the cursor near the center of the screen and hold down F3; moving the cursor up or down will rotate about the screen X-axis while moving the cursor left or right will control the rotation about the screen Y-axis. Start with the cursor in the corner of the screen and hold down F3, which will control the rotation about the screen Z-axis.

Dynamic Rotation

F3 + MOUSE

Step 4: Adding additional features

➢ Sketch In Place

One option to manipulate the workplane is with the Sketch in Place command. The Sketch in Place command allows the user to sketch on an existing part face. The workplane is reoriented and is attached to the face of the part.

1. Choose Isometric View in the display viewing icon panel.

2. Choose Zoom-All in the display viewing icon panel.
3. Choose **Sketch in Place** in the icon panel. In the prompt window, the message “Pick plane to sketch on” is displayed.

4. Pick the top face of the horizontal portion of the 3-D object by left-clicking the surface, when it is highlighted as shown in the below figure.

Notice that, at this point, I-DEAS automatically orients the *workplane* to the selected surface. The surface selected is highlighted with a different color to indicate the attachment of the *workplane*. 
The Rectangle command

Next, we will use the Rectangle command to cut out a corner of the 3-D object. The Rectangle icon is located at the same location as the Polyline icon in the application icon panel. I-DEAS provides three options to create rectangles: Rectangle by 2 Corners, Rectangle by 3 Corners, and Rectangle by Center. The different icons denote the different methods to create rectangles.

1. Choose **Rectangle by 2 Corners** in the icon panel. This command requires the selection of two locations to identify the two opposite corners of a rectangle. The message “Locate first corner” is displayed in the prompt window.

2. Move the graphics cursor near the front corner of the 3-D object. Notice the Dynamic Navigator automatically snaps to the corner and also displays the endpoint symbol. Pick the front corner as shown.

3. The message “Locate second corner” is displayed in the prompt window. At this point, move the mouse around to adjust the size of the rectangle. Notice the movement of the mouse is mapped to the workplane. Create a rectangle by clicking at a location toward the inside of the sketching plane; do not align it to the center of the face.

4. Press the **ENTER** key or use the **middle-mouse-button** to end the Rectangle by 2 Corners command.
Modifying the Size of the Rectangle

1. *Pre-select* the two dimensions of the rectangle by holding down the **SHIFT** key while left-clicking the values. The selected dimensions are highlighted.

   **PRE-SELECT**  **SHIFT**  +  **LEFT-mouse-button**

2. Choose **Modify** in the icon panel. The *Dimensions* window appears.

3. In the *Dimensions* window, change the values to the values as shown below.

4. Click on the **OK** button to accept the values you have entered and I-DEAS will adjust the size of the object based on the newly entered values.

5. Press the **ENTER** key or the **middle-mouse-button** to end the **Modify** command.
Extrusion – Cutout

1. Choose Extrude in the icon panel. In the prompt window, the message “Pick curve or section” is displayed.

2. Pick the rectangle at a location other than a side that coincides with an edge of the L-Shape part.

   Attempting to select a line where two entities lie on top of one another (i.e. coincide) causes confusion as indicated by the double line cursor symbol and the prompt window message “Pick curve to add or remove (Accept)**.” This message indicates I-DEAS needs you to confirm the selected item. If the correct entity is selected, you can continue to select additional entities. To reject an erroneously selected entity, press the [F8] key to select a neighboring entity or press the right-mouse-button and highlight Deselect All from the popup menu.

3. At the I-DEAS prompt “Pick curve to add or remove (Done),” press the ENTER key or the middle-mouse-button to accept the selection.

4. The Extrude Section window appears. Set the extrude option to Cutout. Note the extrusion direction displayed in the graphics window.

5. Click and hold down the left-mouse-button on the Depth menu and select the Thru All option. I-DEAS will calculate the distance necessary to cut through the part.

6. Click on the OK button to accept the settings. The rectangle is extruded and the front corner of the 3D object is removed.
7. On your own, generate a shaded image of the 3D object.

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**Save the Part and Exit I-DEAS**

1. From the icon panel, select the **File** pull-down menu. Pick the **Save** option. Notice that you can also use the **Ctrl-S** combination (pressing down the Ctrl key and hitting the “S” key once) to save the part. A small watch appears to indicate passage of time as the part is saved.

2. Now you can leave I-DEAS. Use the left-mouse-button to click on **File** in the toolbar menu and select **Exit** from the pull-down menu. A pop-up window will appear with the message “Save changes before exiting?” Click on the **NO** button since we have saved the model already.
Questions:

1. The truss element used in Finite Element Analysis is a two-force member element. List and describe the assumptions of a two-force member.

2. What is the size of the stiffness matrix for a single element? What is the size of the overall global stiffness matrix in example 2.2?

3. What is the first thing we should consider when starting a new CAD model in I-DEAS?

4. How does the I-DEAS Dynamic Navigator assist us in sketching?

5. How do we remove the dimensions created by the Dynamic Navigator?

6. How do you modify more than one dimension at a time?

7. What is the difference between **Protrude** and **New Part** when extruding?

8. Identify and describe the following commands:

   (a)
   
   \[\text{SHIFT} + \text{LEFT mouse button}\]

   (b)

   (c)

   (d)
   
   \[\text{F3} + \text{Mouse}\]
Exercises:

1. Determine the nodal displacements and reaction forces using the direct stiffness method.

2. (Diagram not provided in text)