ASCENT - Center for Technical Knowledge®
Autodesk® Inventor® 2018
iLogic
1st Edition

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Preface

The Autodesk® Inventor® 2018 iLogic student guide teaches students to use the iLogic functionality that exists in the Autodesk® Inventor® 2018 software. In this practice-intensive curriculum, students acquire the knowledge required to use iLogic to automate Autodesk Inventor designs.

In this student guide, students learn how iLogic functionality furthers the use of parameters in a model by adding an additional layer of intelligence. By setting criteria in the form of established rules they learn how to capture design intent, enabling them to automate the design workflow to meet various design scenarios in part, assembly, and drawing files.

Topics Covered

• iLogic functionality overview.
• iLogic workflow overview.
• Review of model and user-defined parameters, and equations and their importance in iLogic.
• Understanding the iLogic interface components (iLogic Panel, Edit Rule dialog box, and iLogic Browser).
• Rule creation workflow for Autodesk Inventor parts and assemblies.
• Using variations of conditional statements in an iLogic rule.
• Accessing and incorporating the various function types into an iLogic part, assembly, or drawing file rule.
• Event Triggers and iTriggers.
• Creating Forms to create a custom user interface for an iLogic rule.
Note on Software Setup

This student guide assumes a standard installation of the software using the default preferences during installation. Lectures and practices use the standard software templates and default options for the Content Libraries.

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With a dedication for engineering and education, Jennifer has spent over 20 years at ASCENT managing courseware development for various CAD products. Trained in Instructional Design, Jennifer uses her skills to develop instructor-led and web-based training products as well as knowledge profiling tools.

Jennifer has achieved the Autodesk Certified Professional certification for Inventor and is also recognized as an Autodesk Certified Instructor (ACI). She enjoys teaching the training courses that she authors and is also very skilled in providing technical support to end-users.

Jennifer holds a Bachelor of Engineering Degree as well as a Bachelor of Science in Mathematics from Dalhousie University, Nova Scotia, Canada.

Jennifer MacMillan has been the Lead Contributor for *Autodesk Inventor iLogic* since its initial release in 2013.
In this Guide

The following images highlight some of the features that can be found in this Student Guide.

**Practice Files**

The Practice Files page tells you how to download and install the practice files that are provided with this student guide.

**FTP link for practice files**

**Chapters**

Each chapter begins with a brief introduction and a list of the chapter’s Learning Objectives.
Side notes

Side notes are hints or additional information for the current topic.

Instructional Content

Each chapter is split into a series of sections of instructional content on specific topics. These lectures include the descriptions, step-by-step procedures, figures, hints, and information you need to achieve the chapter’s Learning Objectives.

Practice Objectives

Practices enable you to use the software to perform a hands-on review of a topic.

Some practices require you to use prepared practice files, which can be downloaded from the link found on the Practice Files page.

Chapter Review Questions

Chapter review questions, located at the end of each chapter, enable you to review the key concepts and learning objectives of the chapter.
Icons in this Student Guide

The following icons are used to help you quickly and easily find helpful information.

<table>
<thead>
<tr>
<th>Icon</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>New in 2018</td>
<td>Indicates items that are new in the Autodesk Inventor 2018 software.</td>
</tr>
<tr>
<td>Enhanced in 2018</td>
<td>Indicates items that have been enhanced in the Autodesk Inventor 2018 software.</td>
</tr>
</tbody>
</table>
Chapter 1

Introduction to iLogic

The Autodesk® Inventor® iLogic functionality provides an easy-to-use interface that enables you to automate designs by creating and manipulating rules that drive the model’s geometry. As opposed to making parameter changes in a single model, the use of iLogic enables you to set the controls for parameter change to build design intent into a model.

Learning Objectives in this Chapter

• Understand the various design automation tools in the Autodesk Inventor software.
• Understand how the iLogic functionality can be incorporated into a model to use parameters and equations to build design intent into a model.
• Describe the workflow for incorporating iLogic automation into an Autodesk Inventor file.
1.1 Design Automation Overview

Several different methodologies can be used for design automation. These range from out-of-the-box Autodesk Inventor solutions to highly customized and programmed solutions. The progression of these solutions is shown in Figure 1–1. As you progress through the options, the investment in training or customized programming increases. The key is to find a solution that balances a justifiable investment with the requirements for the solution.

Out-of-the-box

A number of tools in the Autodesk Inventor software can be used to incorporate intelligence into your designs. The progression of these solutions in terms of complexity is shown in Figure 1–2.

- Parameters and equations can be used to relate dimensions or features to one another. The relationships that are established build intelligence into your design so that the basic design intent is captured.
• The Autodesk Inventor iPart and iAssembly functionality enables you to create variations in your part designs quickly and easily. This provides an alternative to recreating the same model repeatedly while varying specific parameters in the model. This customization is accomplished by assigning variable model parameters to a table in which values are entered to create a specific variation.

• Parameters can be controlled directly in the software. However, a further level of customization enables you to control the parameter values using a spreadsheet. The spreadsheet can be embedded or linked to an Autodesk Inventor file to control the parameter values. This is ideal for users that might not have access to the Autodesk Inventor software or have limited knowledge of the software.

• Using iLogic functionality in the Autodesk Inventor software enables you to incorporate a rules-based design into your model that captures and automates an intelligent design process. Using the basic iLogic functions requires little or no programming knowledge, but programming experience can help with the advanced functions. This functionality is the focus of this student guide.

• Programming VBA Macros or Application Programming Interfaces (API), or incorporating custom add-ins for design automation can be done using the out-of-the-box Autodesk Inventor functionality. This level of customization requires programming knowledge.

Many users do not have API experience or a background with VB and VBA programming that is required for the top level of out-of-the-box customization. Therefore, iLogic, with its intuitive interface, is an excellent choice for design automation.

At the top end of the spectrum, standalone tools can be programmed that are highly customized and oriented toward solving a specific business or design need. For example, an engineer-to-order system or sales configurator tool might be required to limit the contact points in a sales cycle. This solution generally requires programming using the Autodesk® Inventor® Engineer-to-Order Series or other custom-built applications with considerable investment.
1.2 iLogic Overview

iLogic functionality takes the use of parameters and equations in a model one step further by adding an additional layer of intelligence in your design. iLogic enables you to set the criteria in the form of established rules that capture design intent, enabling you to reuse designs to suit various design scenarios. The rules are all incorporated as part of the digital prototype definition. The rules that define the design intent control the model and automate the design workflow to ensure that the model reacts correctly. iLogic rules are based on conditional statements and functions. Automation functions have been embedded in the rule creation dialog box enabling you to define the rules that can accomplish many different results in a design. No programming knowledge is required to use the basic iLogic functions, but programming experience can be an asset when using the advanced functions.

In the example shown in Figure 1–3, iLogic is used to control the thickness of the connecting rod, based on its length. The rule that controls the geometry indicates the range of values that are acceptable and what thickness value is to be used.

```vbnet
If length >= 60 And length < 80 Then
Parameter("thickness") = 1.5
Else If length >= 80 And length <= 100 Then
Parameter("thickness") = 3.0
Else If length < 60 Or length > 100 Then
MessageBox.Show("Please enter a value between 60-100 mm.", "Invalid Entry")
End If
```

![Length = 60 mm](image)

![Length = 90 mm](image)

Figure 1–3
iLogic Functionality

Some things that can be accomplished using iLogic rules include the following:

- Control model and user parameter values to ensure that the specifications and standards are met. Supports string (Text), boolean (True/False), and numeric parameters.

- Activate part and assembly features, assembly components, or assembly constraints based on conditional statements.

- Perform multiple operations based on a single user input.

- Perform checks (iProperties, dimensions, etc.) in the model for design situations.

- Update material and iProperty information in the model.

- Read document information (filename, path, extension, etc.).

- Measure entities in the model.

- Provide customized feedback based on specified conditions.

- Drive iFeature, iPart, and iAssembly configurations.

- Incorporate the execution of other rules in a parent rule.

- Interface directly with a predefined form to assign parameter values.

- Control drawing size, borders, and title block information based on user entry.

- Control view location and size or suppression in a drawing.
The assemblies shown in Figure 1–4 have been automated using iLogic. Custom forms have been created to easily enter the required data that drives all of the iLogic rules.
1.3 iLogic Workflow

As with most functionality in the Autodesk Inventor software, the use of iLogic also follows a workflow. This workflow can be broken into four steps.

Use the following general steps to create iLogic rules:

1. Prepare the Model/Drawing.
2. Rule Creation.
3. Set rule triggers.
4. Create and edit rules, as necessary.

Figure 1–5 highlights the steps graphically. Additional in-depth information is included for each of these steps as you progress through this student guide.

---

Step 1 - Prepare the Model/Drawing.

Before a rule can be created, all of the required dimensions, parameters, and equations must already have been added to the model. Rules are written based on this information and without it, the rules are not tied to the model geometry. Therefore, when you design a model, you should always consider the final design intent.

- Verify that feature dimensions capture the model's geometric intent, ensure that any required user parameters (Numeric, Text, and True/False) are included, and that they update if changes are required.

In the case of automating a drawing, a drawing must exist that contains all of the details required to communicate the design. iLogic rules in a drawing can control sheet sizes, title blocks and borders, view positioning, scaling, and suppression.
An iLogic rule is used to control the parameters, features, or components beyond what the user defined parameters and equations can do in the Autodesk Inventor software.

- iLogic rules are based on conditional statements and functions that capture the design intent, enabling you to reuse designs to meet various design scenarios.
- A substantial and varied list of functions are provided that can be included in a rule.

Rule triggers enable you to define when a rule is launched (triggered). iLogic provides a list of event triggers to which the established rules are assigned.

- The list of triggers varies slightly depending on whether a part, assembly, or drawing is active.
- Each trigger provides you with standard functions that are commonly used, such as before a document save, when a document is closed, or when part geometry is changed.
- An iTrigger can also be used to trigger rules by adding a user parameter to the document that in turn launches any rules that it contains.

Continue to add or edit rules, as required.

- A complete list of rules can be reviewed in the iLogic browser.
- The order in which rules are listed in the iLogic browser can affect the resulting geometry. Drag and drop rules in the iLogic browser to capture the model’s true design intent.
Practice 1a Working with a Logical Model

Practice Objectives

- Review parameters in a part and assembly model.
- Review iLogic rules that have been created in part and assembly models.
- Launch and edit an iLogic form to modify the key parameters used to configure an assembly and its components.

In this practice, you will open an Autodesk Inventor assembly in which iLogic rules and forms have been created in the top-level assembly model. A component in the assembly also has iLogic-rules. Using a custom iLogic form, you will make changes to the model by selecting and entering new parameter values. The intent of the practice is to show how multiple design configurations can easily be created after iLogic rules have been incorporated into a model.

Task 1 - Open a part model and review its parameters.

1. If the Autodesk Inventor software is not open, select Start> All Programs>Autodesk>Autodesk Inventor 2018> Autodesk Inventor 2018 or double-click on the Autodesk Inventor 2018 icon on the desktop.

2. In the Get Started tab>Launch panel, click (Projects) to open the Projects dialog box.

3. Click Browse, browse to C:\Autodesk Inventor 2018 iLogic Practice Files\ (or the directory of the installation files if you changed the default directory), and select iLogic.ipj. Click Open. The Projects dialog box updates and a checkmark displays next to the new project name, indicating that it is the active project. The project file tells the Autodesk Inventor software where your files are stored.

4. Click Done.
5. Open `Configured_Clip.ipt` in the `Overview` folder. The model displays as shown in Figure 1–6.

![Autodesk Inventor](image)

**Figure 1–6**

6. In the `Manage` tab>Parameters panel, click \( \text{Parameters} \) (Parameters) to open the Parameters dialog box. Expand the User Parameters node, if required. Five key user parameters have been created in the model. These have been added to the model for use in the iLogic rules and contain the required configuration options. For example, the Clip_Color parameter enables you to select from five material types.

7. Expand the Model Parameters node, if required. The four key model parameters (`Length`, Clip_Angle, Thickness, and Width) are also used in the iLogic rules.

8. Close the Parameters dialog box.

**Task 2 - Launch an iLogic Form to configure the model.**

1. In the Manage tab>iLogic panel, click \( \text{iLogic browser} \) (iLogic browser) to toggle on the display of the iLogic browser. The browser displays embedded in the Model browser.

2. In the Rules tab in the iLogic browser, note that seven iLogic rules have been created in the model. Their descriptive names help to identify their purposes. For example, the Clip_Color rule controls the clip color based on the material type that is selected.

3. Double-click on the Clip_Color rule to open the Edit Rule dialog box. This dialog box is used to program all of the iLogic rules. Click Close to close the dialog box without making any changes.
4. Select the *Forms* tab. It contains any forms that have been created in the model.

5. Click **Clip Configuration** to open the Clip Configuration form, as shown in Figure 1–7.

![](clip_configuration.png)

**Figure 1–7**

Each of the items in the form drives the following changes in the model:

- Assigns the client name for the project. This is a drop-down list of available client names.
- Defines the clip color. This is a drop-down list of available materials.
- Assigns the thickness and width of the clip. This is a drop-down list of the available sizes.
- Assigns the clip angle. This is a user-entry field. The permitted range of angular values is 7.5 to 20.
- Assigns the clip length. This is a user-entry field. The permitted range of values is 35 to 70mm.
- Assigns engraving to the surface of the clip. You can specify whether the engraving is to be included. If set to **True**, you can specify the engraving text. When engraving is included, a message displays details about the acceptable character length.

*If outside of the permitted range, an assigned value is set and you are provided with feedback.*
6. Change the model’s configuration by selecting and entering values for each of the fields.

7. The model updates with the changes as you make selections in the drop-down lists, or when you press <Enter> after entering a value.

8. Click **Done** to close the form.

9. Close the model without saving.

**Task 3 - Launch an iLogic form to configure the assembly.**

1. Open **Mechanical Pencil_Complete.iam** in the **Overview** folder. The model displays as shown in Figure 1–8.

![Figure 1–8](image)

2. The assembly should open in the iLogic LOD representation. If not, set the **iLogic** representation as the active Level of Detail. This representation is required for the Grip Style iLogic rule to run correctly. This is discussed in the Assembly functions content.

3. In the iLogic browser, select the **Rules** tab. The rules provided enable customization of the assembly.

4. Select the **Forms** tab. It contains any forms that have been created in the model.

To set the LOD representation, expand the **Representations** and **Level of Detail** nodes in the Model browser, and double-click on iLogic.
5. Click **Assembly Configuration** to launch the Assembly Configuration form, as shown in Figure 1–9.

![Assembly Configuration form](image)

**Figure 1–9**

Each of the items in the form drive changes in the assembly and clip models. Many of the options are the same as in the clip model and the values entered here are pushed to the clip model to make changes to it. The only item that is unique to the assembly is the Grip Style. This controls how the grip geometry is represented in the assembly.

6. Change the assembly’s configuration by selecting and entering values. The components display the changes as selections are made, or when you press <Enter> after entering a value.

7. Click **Done** to close the form.

8. Close the model without saving.
Chapter Review Questions

1. Which of the following design automation tools are available as part of the Autodesk Inventor software? (Select all that apply.)
   a. Parameters
   b. iPars
   c. iAssemblies
   d. iLogic
   e. API Programming
   f. Autodesk Inventor Engineer-to-Order Series

2. Which of the following best describes why it is important to ensure that the dimension scheme in a model captures the design intent before iLogic rules are added?
   a. Dimensions cannot be modified once iLogic rules have been incorporated into the model.
   b. Features cannot be modified once iLogic rules have been incorporated into the model.
   c. The dimension/model parameters are referenced in the rules to drive the model geometry. They must capture the model's intent for the rule referencing to work correctly.

3. Working with iLogic requires experience with API and VBA programming.
   a. True
   b. False

4. Which of the following can be accomplished using iLogic? (Select all that apply.)
   a. Perform multiple operations based on a single user input.
   b. Update material and iProperty information in the model.
   c. Provide user with customized feedback based on specified conditions.
   d. Drive iFeature, iPart, and iAssembly configurations.
   e. Interface directly with a predefined form to assign parameter values.