INVENTOR PARAMETRIC DESIGN: LEVEL 1

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Inventor Parametric Design: Level 1

A Guide to Parametric Modeling and Technical Drawing

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Institution: Western Technical College
Edition: Initial Release
Date: June 2, 2025

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This text was compiled on 05/15/2026

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About the Author

Alex Bahl is an experienced mechanical designer with a strong foundation in design and manufacturing. With over a decade of expertise in CAD modeling and engineering technology, Alex has specialized in refining workflows, enhancing product quality, and optimizing production efficiency. His deep knowledge of Inventor, SolidWorks, and AutoCAD, combined with his strategic approach to problem-solving, has enabled him to lead cross-functional teams in developing innovative solutions that reduce costs and improve customer satisfaction.

Alex has a Bachelor of Science in Mechanical Engineering Technology from the University of Wisconsin Oshkosh and an Associate of Applied Science in Mechanical Design Technology from Western Technical College in La Crosse, Wisconsin. His professional experience spans roles as a Mechanical Designer at Nelson Global Products, a CAD Designer at Wabash National, and currently, as an Instructor at Western Technical College, where he imparts practical engineering knowledge to the next generation of designers.

With a passion for precision, efficiency, and innovation, Alex's approach to parametric design leverages industry-leading techniques to create adaptable and intelligent models. His textbook provides a structured methodology for mastering Autodesk Inventor, equipping learners with the skills to build scalable, parameter-driven designs suited for modern manufacturing environments.

Acknowledgments

Thank you to my instructors and professors that passionately shared their expertise in classrooms and virtual meetings. Thank you too, industry professionals that kindly shared their knowledge and experience with me, a curious young CAD designer.

Resource Overview

This textbook, *Inventor Parametric Design: Level 1*, provides a project-structured methodology for learning Autodesk Inventor. The three projects included will equip learners to build parameter-driven designs, giving them fundamental skills that are scalable for modern manufacturing environments. The author, Alex Bahl, is an experienced mechanical designer with a strong foundation in design and manufacturing. He is currently an instructor of Mechanical Design at Western Technical College in La Crosse, WI. Alex's approach to parametric design leverages industry-leading techniques to create adaptable and intelligent models.

Note: This resource is also available as a formatted PDF and editable document on WISELearn.

The three chapters cover:

- **Technical Drawings** with emphasis on continuous improvement through peer reviews.
- **Bottom-Up Design** with emphasis on data management of vendor supplied parts.
- **Top-Down Design** with emphasis on applying functional tolerances to parametric designs.

Competencies addressed in the text include:

1. Apply techniques for mathematical and scientific problem-solving to support design and modeling processes.
2. Explore user interface, command entry, sequence, file preferences, and options in various software to develop proficiency in digital tools.
3. Revise and update working drawings to refine accuracy and incorporate design improvements.
4. Complete working drawings and views of parametric models using acceptable industry standards to establish effective technical documentation.
5. Sketch 2D geometry using 3D parametric software to build foundational design skills.
6. Constrain 2D geometry using dimensions and geometric constraints to ensure precision in sketches.
7. Create a basic parametric model to transition from 2D sketches to functional 3D representations.
8. Create features on a model to enhance complexity and functionality in parametric design.
9. Assemble parts using parametric assembly concepts to construct multi-component systems.
10. Conceptualize and model components of a given assembly to refine and adapt designs for specific applications.
11. Differentiate between dimensioning and tolerancing methods and understand their effects to ensure precision and manufacturability.

CHAPTER OVERVIEW

1: Technical Drawings

SECTION OVERVIEW

1.1: Understanding Technical Drawings

Topic hierarchy

1.1.1: Overview of Technical Drawings in Engineering Design

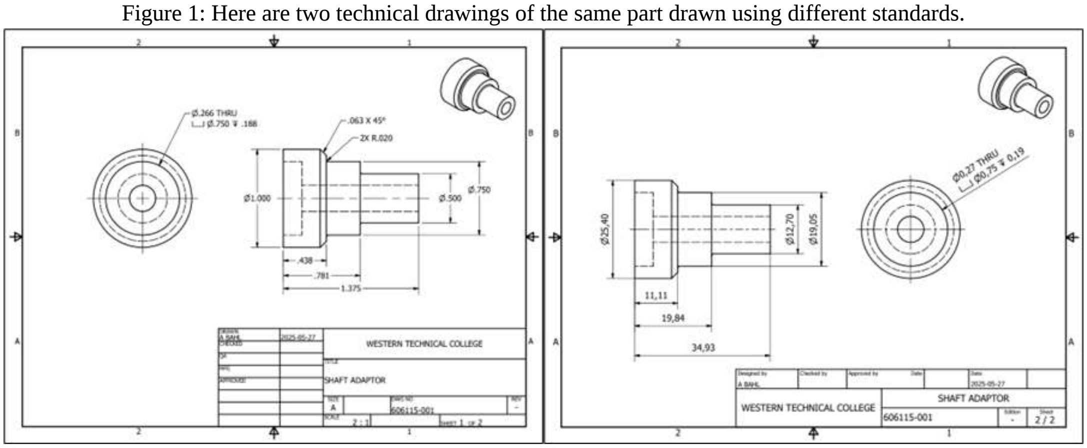
Technical drawings are used to convey all the specifications and tolerances required to manufacture a product. They follow strict industry standards to ensure that designs can be accurately interpreted by engineers, machinists, and manufacturers worldwide.

1.1.2: Standardized Drafting Conventions

Industry standards ensure uniformity across technical drawings, reducing errors and misinterpretation. The most widely used standards include:

- **ASME -** American Society of Mechanical Engineers
- **ISO -** International Organization for Standardization
- **DIN -** Deutsches Institut für Normung

Figure 1: Here are two technical drawings of the same part drawn using different standards.



ASME Standard ISO Standard

1.1.3: Overview of Relevant ASME Standards

ASME is the primary standardization organization in the American market and will be the focus of this book. ASME publishes standards, and the Y14 standards describe the proper format, best practices, technical requirements, definitions of terms and rules of how to interpret a technical drawing, and CAD models. We will cover topics from these specific standards:

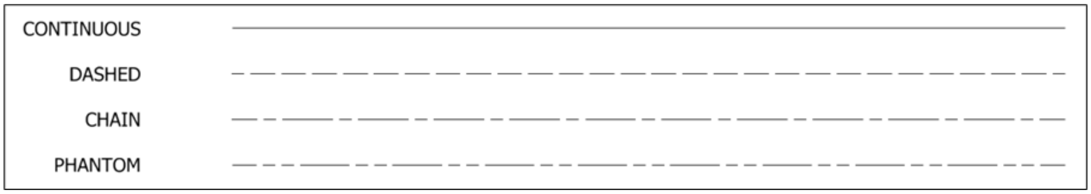
- ASME Y14.2 Line Conventions and Lettering
- ASME Y14.3 Orthographic and Pictorial Views
- ASME Y14.5 Dimensioning and Tolerancing
- ASME Y14.35 Revision of Engineering Drawings and Associated Documents

1.1.4: Lines and Lettering

The way a line is drawn on a technical drawing communicates to the print reader. Here are line types and their uses. (American Society of Mechanical Engineers [ASME], 2015)

Line Type	Uses
Continuous	Visible Edges, dimensions, and more
Dashed	Hidden Lines, Threads
Chain	Centerlines, Center marks
Phantom	Cutting Plane Lines (Section Views), Alternate Positions

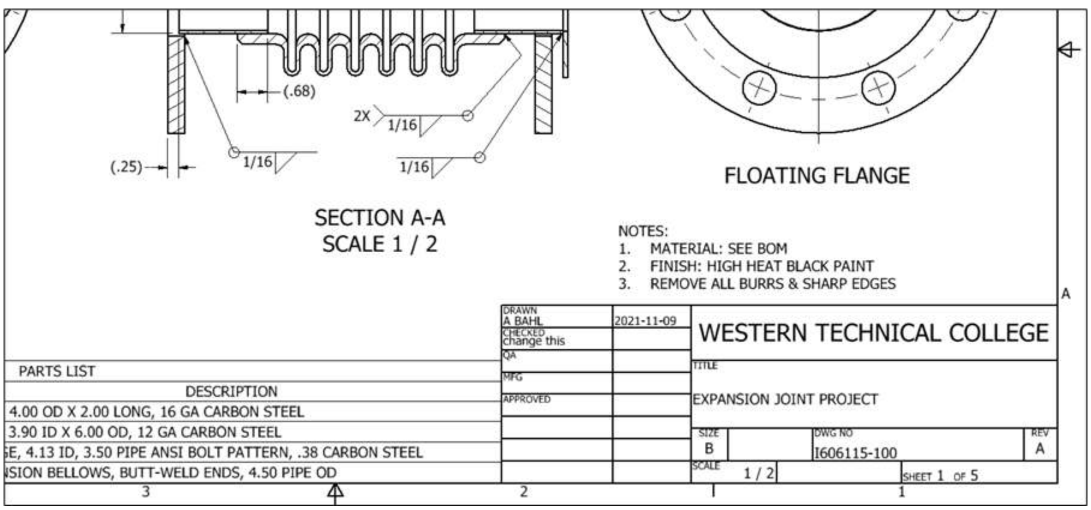
Figure 2: Line Types



Lettering on a technical drawing must be easy to read. The Y14.2 Standard specifies the minimum letter height of text on a drawing.

Height	Uses
.12” (3mm)	Notes, Dimensions, and Title Block Text
24” (6mm)	Titles, View Labels, Section View Identifiers
.10” (2.5mm)	Title Block Labels

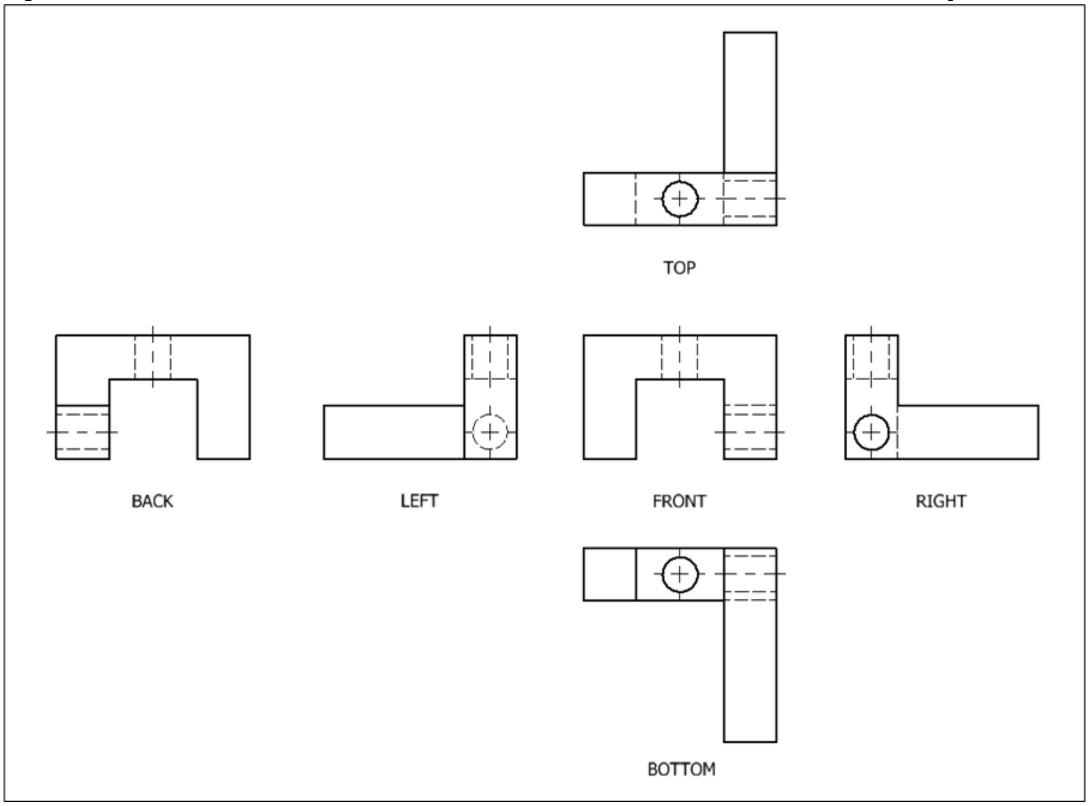
Figure 3: Lettering Example



1.1.5: Orthographic Drawings Views

Orthographic drawing is a way of drawing a three-dimensional object on paper. “Ortho” is a Greek root word meaning “Straight,” and is used to describe how they drawing views are aligned. Starting with the Front View, we will place the Right View to the right, and Top View above. There are Six Principal Views: Front, Right, Left, Top, Bottom, and Back. Imagine placing the object you would like to draw inside of a glass box and then you look through the front window of the box and draw what you see. This would be the Front View. If you look through the right side of the box, that would be the Right View and should be placed to the right of the Front View and aligned so the top, bottom, and all features line up horizontally. (ASME, 2013)

Figure 4: The Six Principal Views

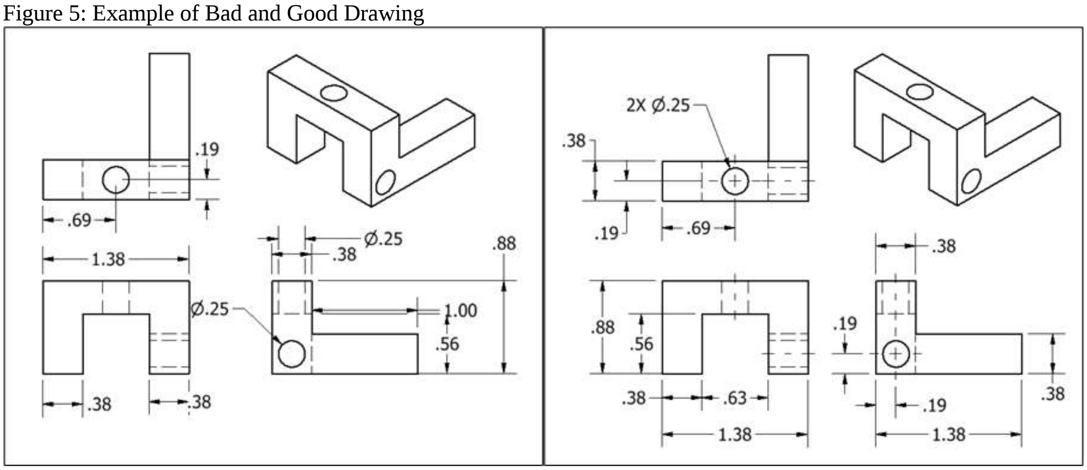


1.1.6: Dimensioning and Tolerancing

1.1.6: Dimensioning and Tolerancing

The important thing to remember is that dimensioning and tolerancing must clearly define design intent. The ASME Y14.5 standard governs geometric dimensioning and tolerancing practices and covers a whole class's worth of material, as you may know. We will review the more nuanced topics in the standard that aid in drafting a technical drawing that is clear and easy to read. Let us start by comparing a poorly dimensioned drawing, and a properly dimensioned drawing. (ASME, 2019)

Figure 5: Example of Bad and Good Drawing



Bad Good

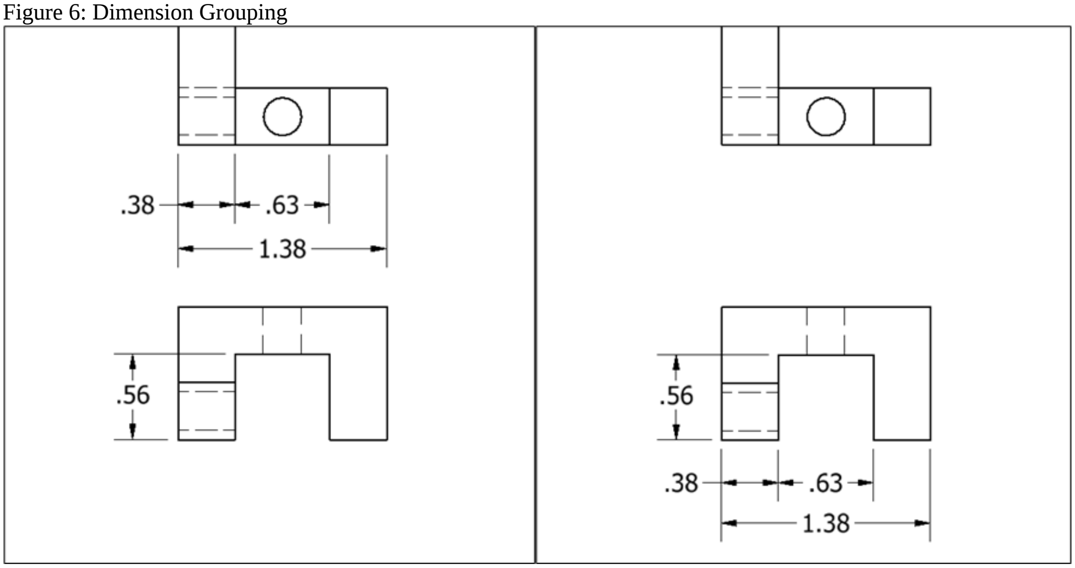
Here are common mistakes to look out for on your drawings and the drawings you peer review.

Dimension Rules

Careful placement of dimensions on a drawing will greatly enhance the readability. By grouping dimensions of a common feature, staggering dimensions that are too close together, and including a visible gap between the object and dimension's extension line, you can improve drawing clarity.

Always dimension a feature where you can see the outline the best. Additionally, by grouping dimensions of a feature together in one drawing view it reduces the time a print reader must search for the dimension they are looking for.

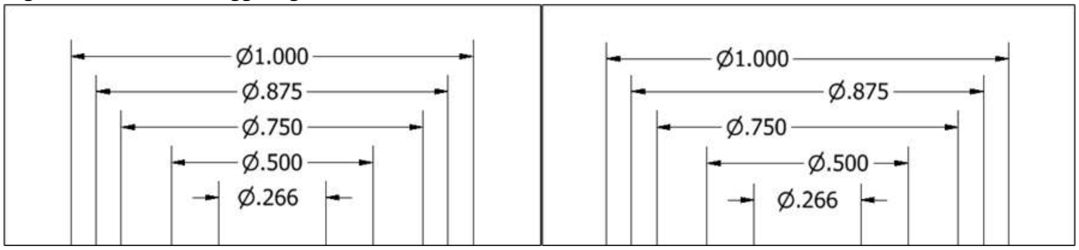
Figure 6: Dimension Grouping



Bad Grouping Good Grouping

Staggering the dimension text will enhance the readability of a technical drawing.

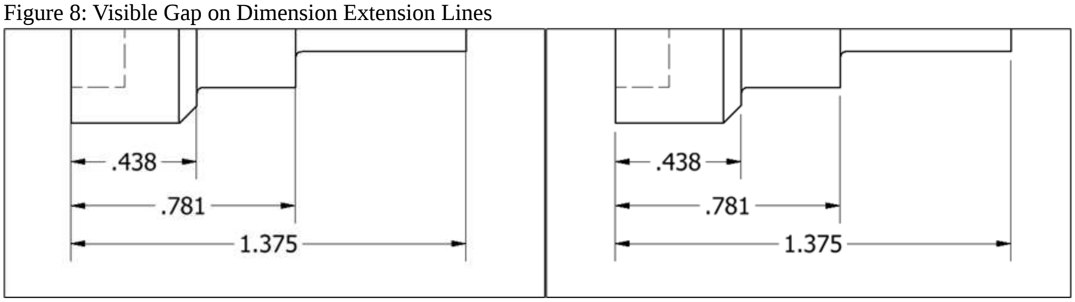
Figure 7: Dimension Staggering



Bad Staggering Good Staggering

A visible gap between object lines and extension lines enhances readability. To ensure a visible gap in your Inventor drawings, do the following: select the dimension by single clicking it, click and drag the end grip of the extension line without a visible gap, drag and release it on the outermost object vertex or line. Repeat on the other side, as necessary. You may have to do this after arranging the dimensions for better dimension grouping.

Figure 8: Visible Gap on Dimension Extension Lines



Bad Visible Gap Good Visible Gap

TIP: For simple dimensions without tolerance, it can sometimes be quicker to delete the dimension and replace it with careful selection of object lines or vertices to achieve the desired look. However, if there are tolerances or feature control frames attached to the dimension, it may be more efficient to drag the end grip of the extension lines to reduce the time spent recreating the dimension tolerance.

Leader Rules

There are rules you could follow when placing leaders to help make your drawing more readable. The first rule is that leaders "float up." That is, they are easier to find if they are toward the top of the drawing, so inclining the leader above the horizontal is ideal. Sometimes other dimensions prevent leaders from being placed above the horizontal, and they must be placed below the horizontal, and that is okay if it improves the clarity of the drawing. The preference is that they are placed above the horizontal, but the ultimate goal is to produce a readable drawing.

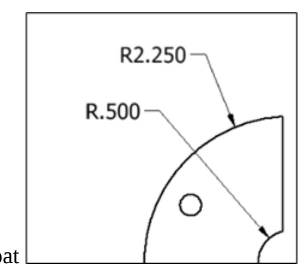


Figure 9: Leaders "Float

Another rule with leaders is to avoid placing leaders too horizontally or too vertically. The best angle is 45 degrees, followed by 30 and 60 degrees, but certainly between 15 and 75 degrees. This avoids leader lines being mistaken for object lines and improves readability. The figure on the right display leader notes in the first quadrant, however the rule applies to leader notes placed in any quadrant.

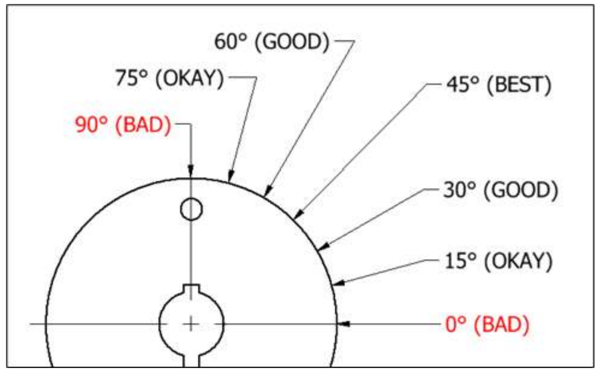


Figure 10: Preferred Leader Position

Diameter and Radius Dimension Rules

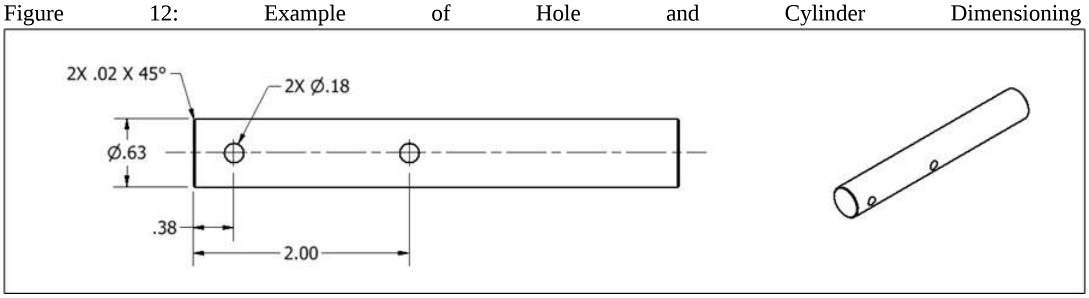
The Leader Rules above also apply to diameter and radius dimensions. Additionally, diameter and radius dimensions should always point to the center point of the arc to which they are attached. If it is not, that dimension is likely an overtyped leader note. The downside there is that the leader note is not linked to the model parameter and will not update automatically.

Full circles such as holes and cylinders should be dimensioned as diameters, and partial circles should be dimensioned as a radius. This is the default behavior of Inventor, but occasionally you will want to change the dimension from one to the other. One example is below. In this example, the shape would be best interpreted as a full circle and therefore would be best dimensioned as a diameter. To change it: after the dimension is placed, press <esc> key to back out of the dimension command, if needed, then right click the dimension you would like to change, hover over Dimension Type, and choose the desired option.

Figure 11: Changing Dimension Type

In the same way we dimension a feature where its outline is shown best, holes and cylinders should be dimensioned following the rules in the ASME standard. Cylinders should be dimensioned from the side, and holes should be dimensioned where they appear as a true circle. Whether it is a cylinder or a hole, they should always be located in the circle view with extension lines extending from the center point.

Figure 12: Example of Hole and Cylinder Dimensioning Rules



Threaded Holes

The example below shows a **1/4-20 UNC – 2B ↧ .500** hole callout. Let us examine how to read a thread note like this.

Thread Note Measurement	Explanation
1/4	.250-inch major diameter. A 1/4” diameter bolt would thread into this hole.
20 UNC	20 threads-per-inch, in the Unified National Course thread form.
2B	Thread tolerance. This is optional and if omitted 2B is assumed.
↧ .500	Thread depth. This is the depth of the fully formed threads.

Threaded Hole Manufacturing Process

It is important to understand how holes are manufactured. Below we see a threaded hole with a ¼-20 UNC thread profile cut to a depth of .500 inches.

A. Blind Hole: A blind hole is a hole that does not go through the other side of the material. When a threaded hole must be made, it starts with a plain hole 1X diameter deeper than the threaded hole finished depth.
B. Threaded Hole: A cutting tool called a “tap” is used to cut the threads for a threaded hole. The end of the tap is tapered to make it easier to start the cutting tool, resulting in the first four threads to not be fully formed threads. This is why the blind hole was cut deeper.
C. Chamfer/Countersink: If desired, a chamfer or countersink is cut to facilitate easy assembly. The diameter shown for the countersink is the diameter measured at the surface of the part.

Dashed lines are used to represent threads in both section view and hidden line views.

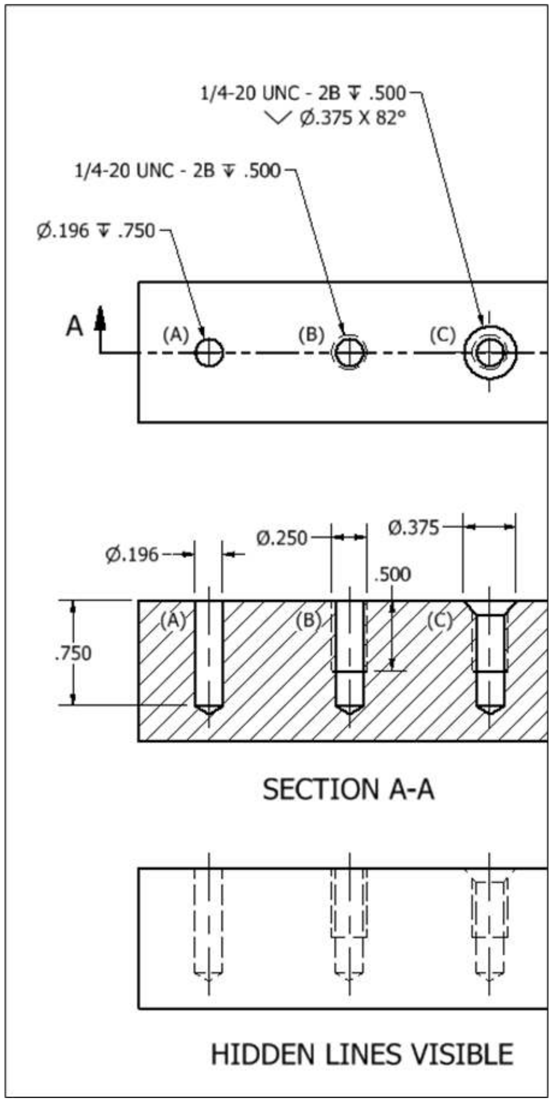


Figure 13: Threaded Hole Process

Clearance Holes

Selecting dimensions for clearance holes has never been easier with the convenience of moder CAD systems like Autodesk Inventor. Clearance hole dimensions have long been standardized but they have not always been built into Inventor's Hole tool. Before then, designers were required to dig out their copy of Machinery's Handbook and look up clearance hole sizes out of the book. (Oberg, 2016)

D. Countersink: This is a clearance hole for a machine screw, which has a conical head profile to match the angle of the countersink. Note that the countersink diameter is the diameter measured at the surface of the part.
E. Counterbore: This is a clearance hole for a cap head screw. Holes (E), (F), and (G) are all counterbored clearance holes for the same 1/4-inch cap head screw, but different fit. This is a **normal** fit.
F. Same as (E), but a **close** fit.
G. Same as (E), but a **loose** fit.

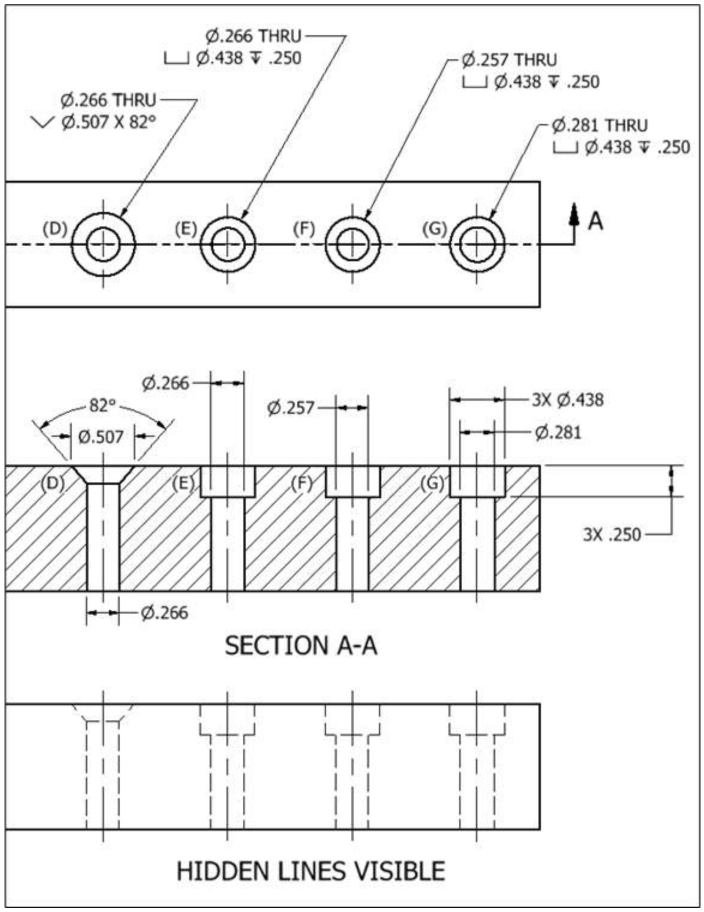


Figure 14: Clearance Hole Types

Assigning Fit in the CAD Model

In the modeling environment you can choose the type of fit of your clearance hole by changing the Fit in the Hole tool.

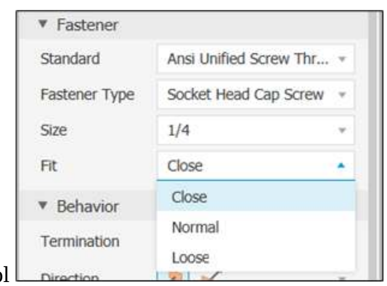


Figure 15: Changing Fit in the Hole Tool

SECTION OVERVIEW

1.2: Peer Review Process and Markup Drawings

Topic hierarchy

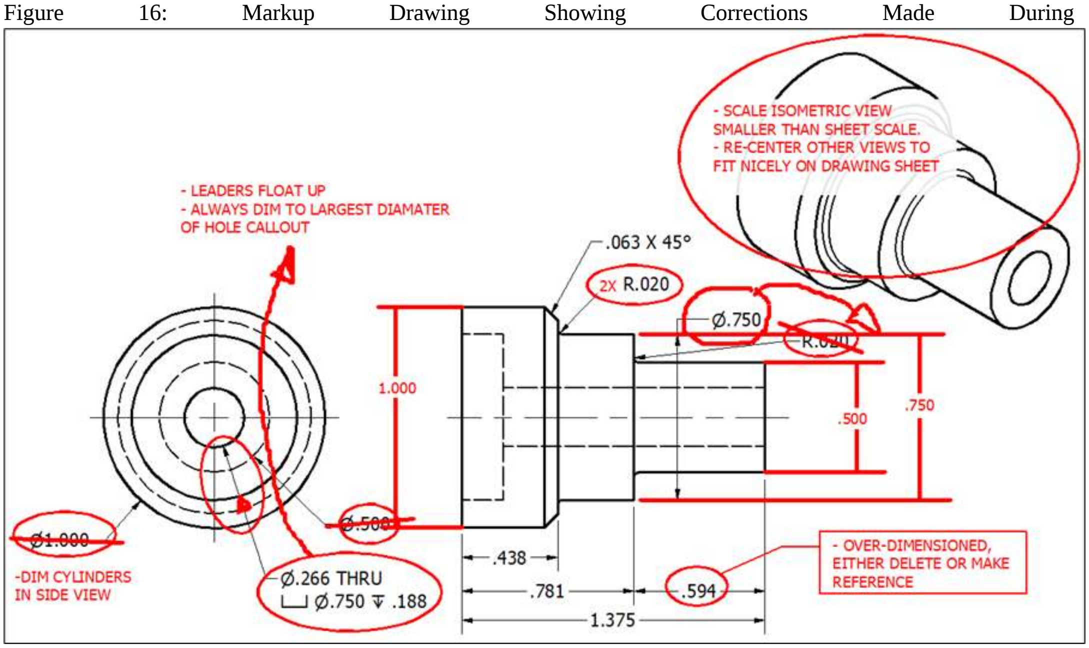
1.2.1: The Role of Peer Review in Engineering Design

Peer review ensures accuracy and manufacturability in technical drawings. The person that reviews the drawing is called the **Checker**, and their name is placed in the **Title Block** for each drawing they check. The designer and checker collaborate to refine dimensions and tolerances, checking for missing details, and manufacturability.

Additionally, peer review encourages continuous improvement by helping designers and engineers learn better ways to represent details on a technical drawing. By catching errors early in the design process, companies can reduce expensive rework and delays, not to mention reputational damage if a subpar product reaches the customer. In local and global companies alike, peer review is also critical for ensuring compliance with government or industry-adopted safety and performance standards.

Bottom line, having drawings checked will improve the design, minimize risk, and ensure efficient manufacturing.

Figure 16: Markup Drawing Showing Corrections Made During Review



1.2.2: Best Practices for Markup and Revision Cycles

When checking a drawing, remember The Three C's. **The Three C's of technical drawing are: Clear, Complete, and Concise.** Whether you are the Drafter or Checker, asking yourself these three questions will ensure a quality technical drawing. Is it clear? Is it Complete? Is it Concise? Here is a detailed list of questions to ask relating to each of The Three C's:

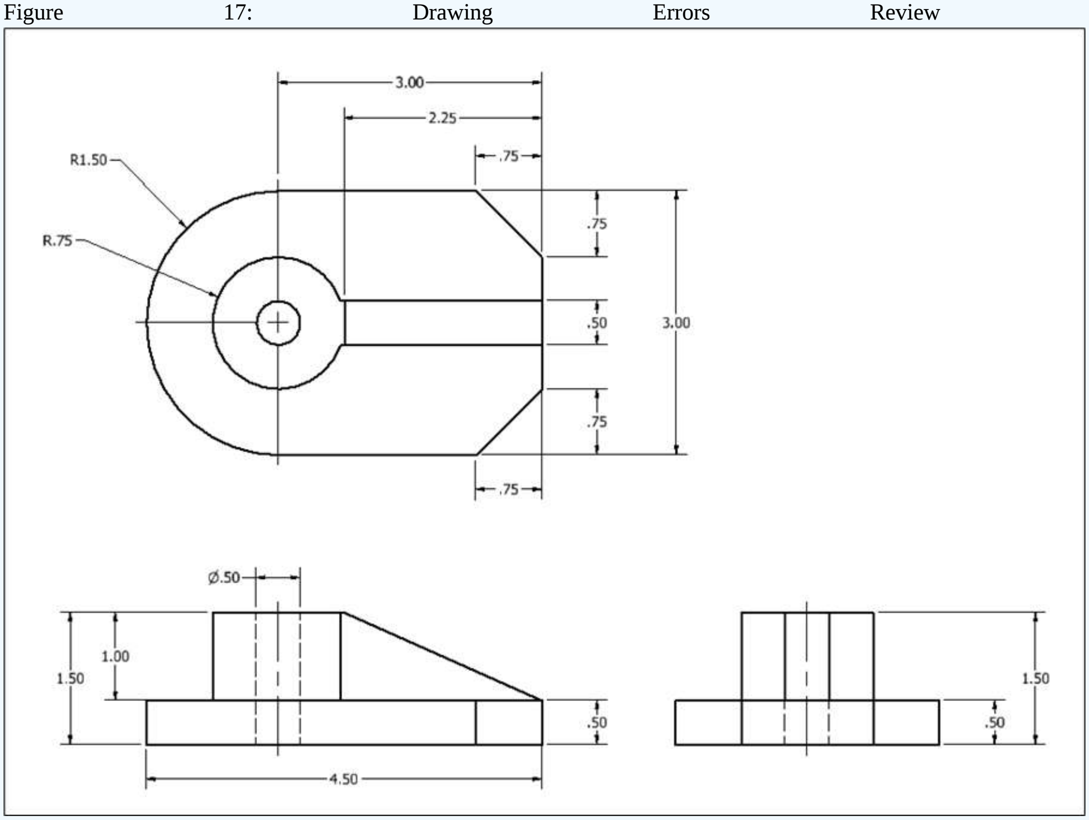
- **Is it Clear?** Are the dimensions readable? Is any text covered by lines? Are any object lines or dimensions outside the border? Does anything look cluttered? Is everything spelled correctly? Are Notes and Annotations properly formatted? Does the text fit inside the Title Block cells? Is there an isometric view?
- **Is it Complete?** Do you have all the dimensions needed? Is the material and finish specified? Are more drawing views needed to describe the object? Is the Title Block complete with Designer, Checker, Date, Company, Title, Description, Part Number, Scale, Revision. Is there a Block Tolerance? Are dimension tolerances appropriate?
- **Is it Concise?** Is any dimension duplicated? Is the part over-dimensioned anywhere? Should any dimensions be reference dimensions, or should a reference dimension be added? Is there any conflicting information? Are there any drawing views that can be eliminated?

Depending on the complexity of the product, the drawing review process may take a few iterations. The first review could improve the readability of the drawing, and once the drawing has been revised a second review could analyze the tolerances and manufacturability. It is important that these intermediary drawings are not mistaken for production drawings, so they should have a note explaining the drawing is "PRELIMINARY," or "NOT FOR PRODUCTION." It is not until a drawing is released to production that it has its status changed to "Initial Release." Once a drawing has been released to production, any changes to the drawing will trigger a **revision bump**, and the history of the changes recorded in the **Revision Table.** (ASME, 2014)

? Exercise Runaway argument for \eatSpaces?

💡 **Exercise:** Review the flawed technical drawing below and suggest necessary corrections.

Figure 17: Drawing Errors Review Exercise



1.2.3: Autodesk Design Review

Autodesk Design Review is a free download from Autodesk. You can publish an Inventor drawing (.IDW) as a Design Web Format (.DWF) file and send it to reviewers and other collaborators. Reviewers can mark up the 2D drawing using Autodesk Design Review, then return the file with comments. Markups are saved in the file, allowing for review and status updates without Autodesk Inventor. (Autodesk, 2018)

Markup Workflow

1. **Publish and Share:** The designer creates a DWF file in Autodesk Inventor and sends it for review.
2. **Review and Markup:** Reviewers check the drawing for The Three C's and then redline and comment in Autodesk Design Review.
3. **Make Corrections:** The designer opens the marked-up DWF, views markups, makes changes, and republishes for further review if needed, or releases the drawing to production.

Figure 18: Autodesk Design Review Ribbon and Markup Tools

SECTION OVERVIEW

1.3: Taking a Tour of Inventor's User Interface

Topic hierarchy

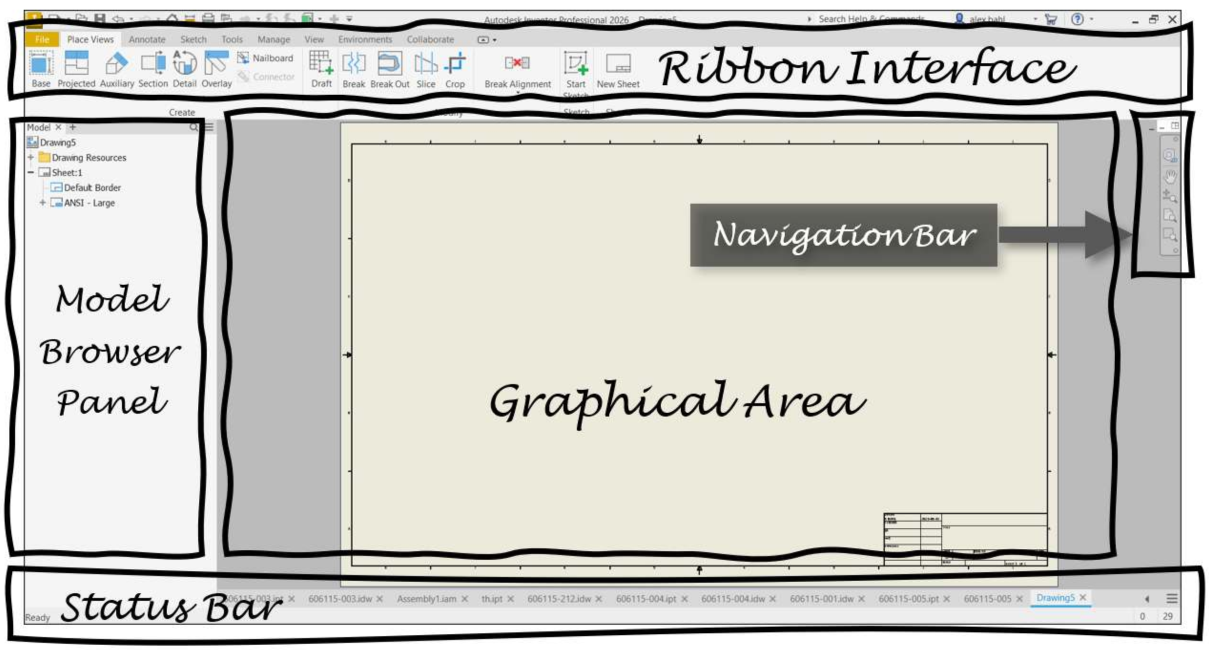
1.3.1: Overview of UI Elements

Autodesk Inventor provides a familiar User Interface (UI) to drafters for their parametric modeling, technical drawings, and assembly design. Like MS Word or PowerPoint, there is a **ribbon** above with all the commands available, and the large window in the middle, the **graphical area**, is where most of the work takes place. (Autodesk, 2025)

Key UI components:

- **Ribbon Interface:** Top of the screen. Houses modeling, sketching, and annotation tools. The tools available on the ribbon change to match the environment (part, assembly, drawing) you are in.
- **Model Browser Panel:** Left side of screen. Displays the feature history of the file you are in. For parts, it shows the feature history. In assemblies, it shows the parts and their relationships (constraints). In a drawing it shows the drawing views, and any sketches in the drawing.
- **Navigation Bar:** Right side of screen. Controls orientation and camera positioning.
- **Graphical Area:** Middle of screen. The largest portion of the screen
- **Status Bar:** Bottom of screen. Inform you of the status, depending on your environment. If you are in a sketch, it will inform you if the sketch is fully defined, or missing dimensions. If you are using a tool, it will inform you what Inventor expects you to do next. In assemblies, it will inform you how many parts, and how many unique parts are in your assembly. It also houses a few tools.

Figure 19: Autodesk Inventor User Interface



SECTION OVERVIEW

1.4: Correcting Drawings in Inventor

Topic hierarchy

1.4.1: Making Corrections to Technical Drawings

A large part of our day as designers is spent on correcting technical drawings and documenting those changes if necessary. Let us review how to make our drawings clear, complete, and concise in Autodesk Inventor.

Dimension Grips

Selecting a dimension will reveal its grips, the green dots at the end points of the extension lines and dimension line. By clicking and dragging the end points a designer can relocate the extension lines. This is useful when extension lines are touching the object.

Dimension text can be moved by clicking and dragging the text. The dimension like will automatically follow the text as you drag. To further modify the look of your dimension, you can right click the dimension and select Options.

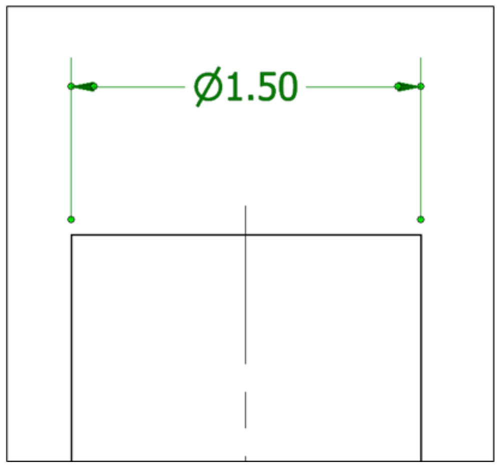


Figure 20: Dimension Grips

Leaders (Radius and Diameter, Notes, float up)

Selecting a leader will reveal grips. Selecting the arrowhead grip allows you to change what the leader is pointing to. Selecting the grip at the landing allows you to move the leader text any direction. Selecting the leader text only allows you to move the leader text left and right (not up or down). Right clicking a leader allows you to change leader Options.

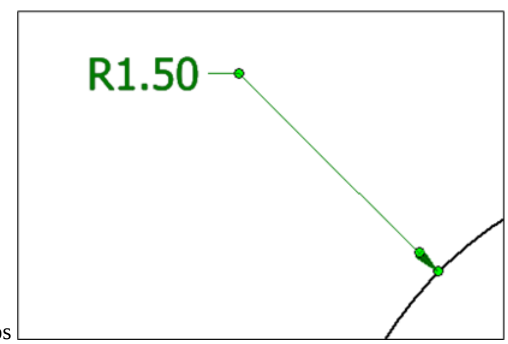


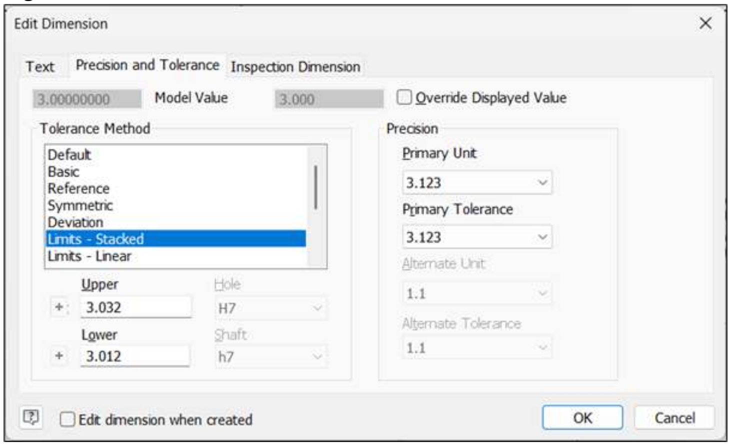
Figure 21: Leader Grips

Precision and Tolerance

Precision is the number of places shown after the decimal point of a dimension value. If precision is all the need to be changed, you can right click and choose Precision in the drop-down menu.

Tolerance is the allowable deviation of a dimension value. Double-click a dimension to open the Edit Dimension dialog box, then select the Precision and Tolerance tab to change the dimension tolerance. Reference and Basic dimensions can also be applied here.

Figure 22: Precision and Tolerance Tab in the Edit Dimension Dialog Box



Drawing View

Double click a drawing view to open the Edit Drawing View dialog box. In the File section the file name of the current model is shown. Under Representation you can choose pre-saved Model State or Design View, if any. In the Style section you can choose whether hidden lines are shown or not. In the Label section you can change the View Identifier (useful for Section Views), change the Scale, and decide if you want the View Label shown on the drawing by clicking the lightbulb icon on or off. Further edit View Label by clicking the pencil.

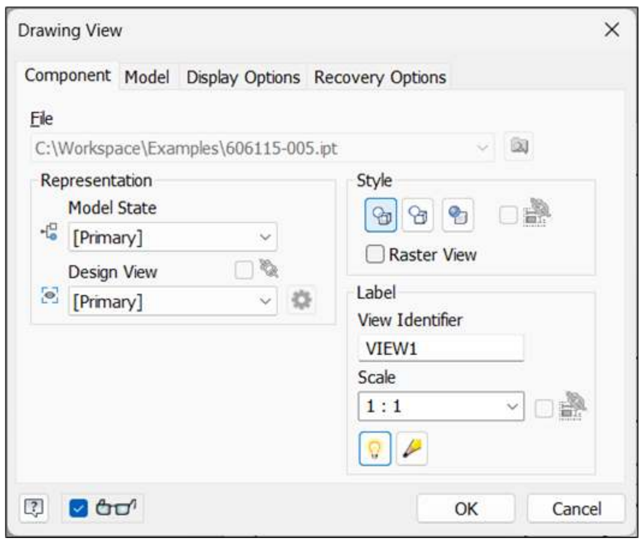


Figure 23: Drawing View Dialog Box

SECTION OVERVIEW

1.5: Project 1- The Expansion Joint

Topic hierarchy

1.5.1: Project Overview and Design Objectives

The expansion joint is a mechanical component used in piping systems to absorb thermal expansion and contraction. This project reinforces the creation of technical drawings using Autodesk Inventor, and the drawing tools you have learned this chapter.

Figure 24: Expansion Joint

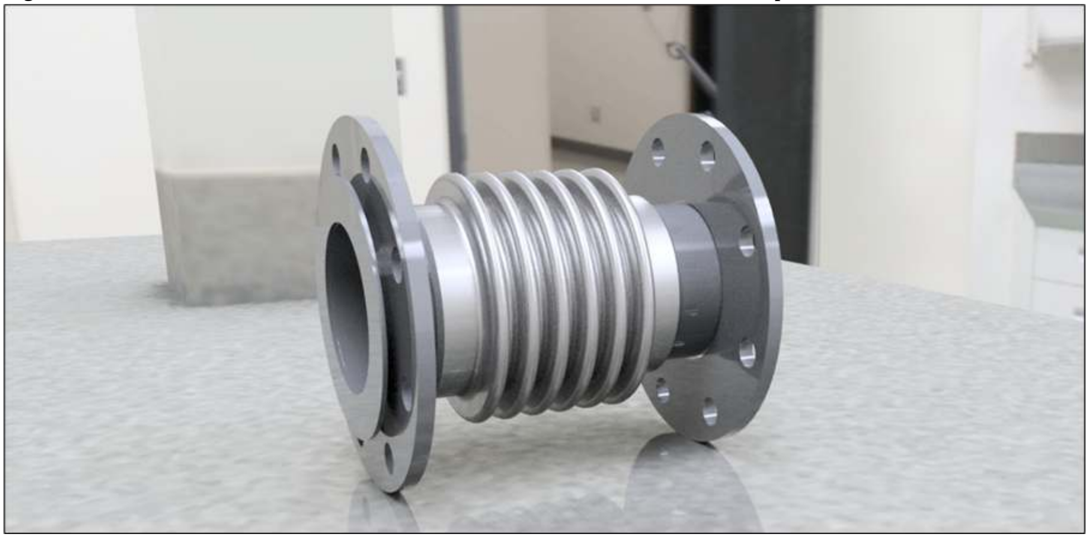
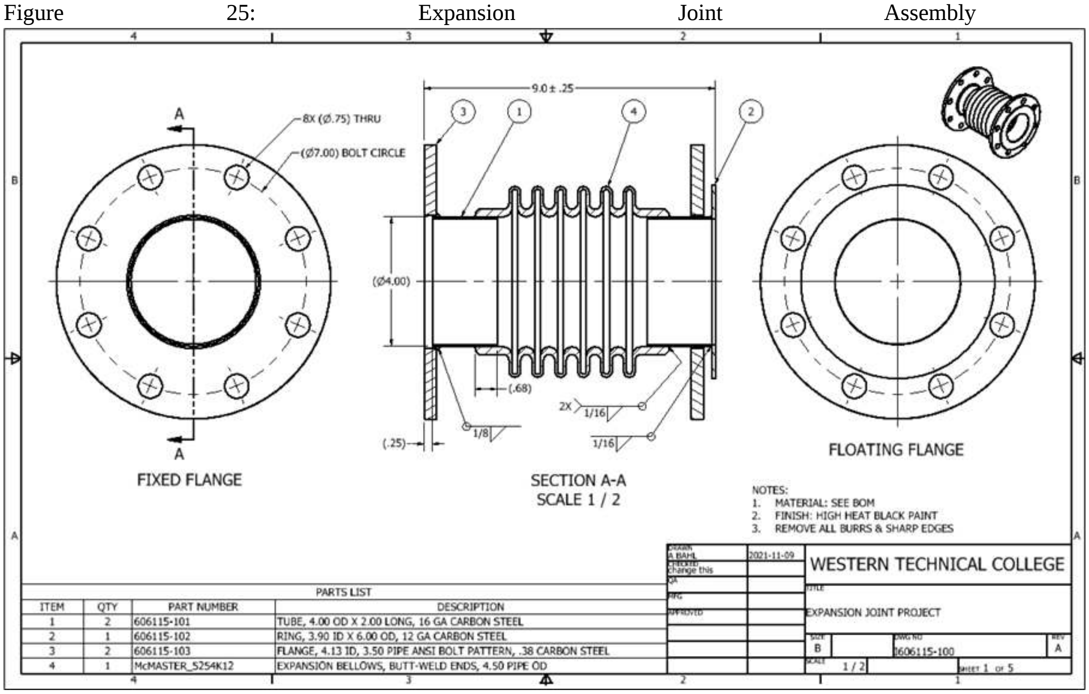


Figure 25: Expansion Joint Assembly Drawing



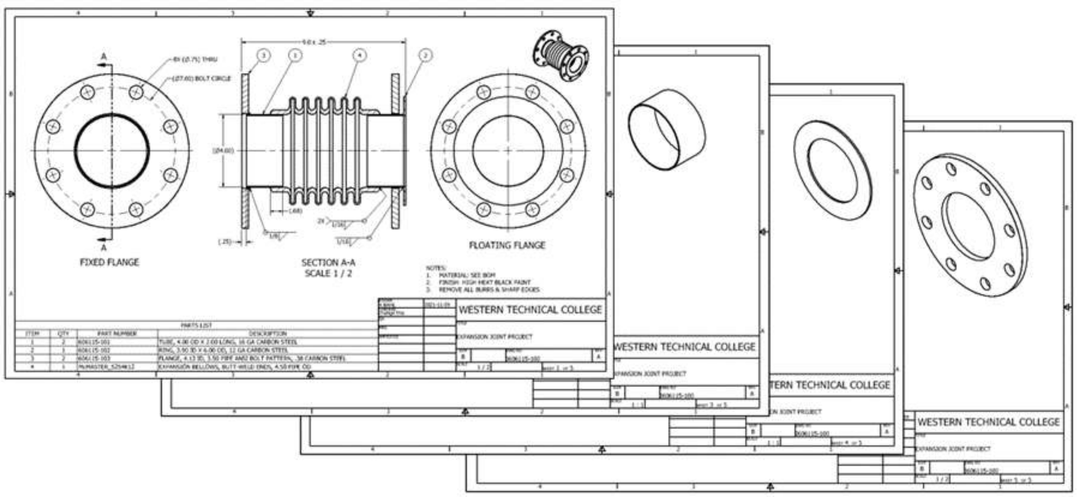
1.5.2: Step-by-Step Workflow

1. **Open** the drawing file provided by your instructor.
2. **Correct** the Assembly Drawing to match the example above.
3. **Detail** the remaining parts of the assembly. Arrange the drawing sheets in the same order as the Parts List Then place all the necessary dimensions.
4. **Print/Publish** your drawing to trade with another student for peer review.
5. **Peer Review** the other students drawings, and place your name in the Title Block as the Checker. Remember to check for The Three Cs.
6. **Make Corrections,** as necessary. Ask your checker questions if clarification is needed. Place their name in the Title Block as the Checker.
7. **Submit** to the instructor.

💡 **Final Deliverables:**

- **Full working drawing package** of the expansion joint that follows ASME standards.

Figure 26: Fully Annotated Working Drawing Package



CHAPTER OVERVIEW

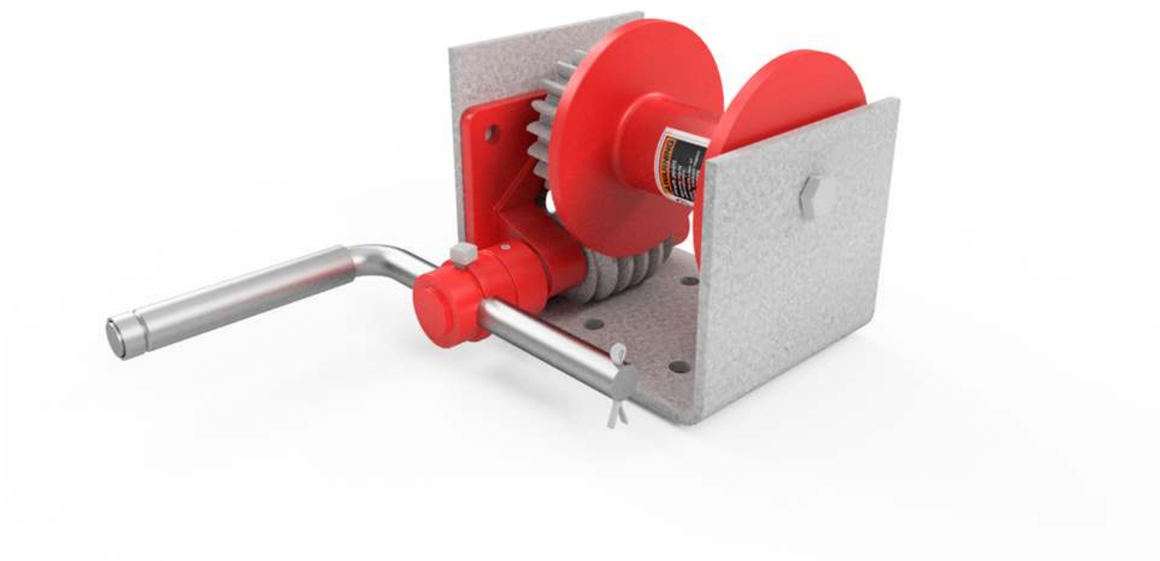
2: Bottom-Up Design

SECTION OVERVIEW

2.1: Understanding Bottom-Up Design in CAD

Bottom-up design is a **component-first** approach where individual parts are designed separately and then assembled into a final product. This method makes prudent use of standard fasteners and purchased parts, making it ideal for projects where off-the-shelf parts are integrated with final designs.

Figure 27: Worm Gear Driven Hand Winch



Topic hierarchy

2.1.1: Case Study- Bottom-Up Design

2.1.1: Case Study- Bottom-Up Design

Select a product in your home, at school, or anywhere in between. Pick something you can pick up or at least walk around and inspect closely. Now make a list or sketch of all the individual parts you can see and note if it is a fastener (screws, rivets, nut/bolt), purchased part (motors, bearings, screens), or a unique component (frame, case, drive shaft). If you can, and it is practical, safely disassemble and remove as much as possible. Be sure to remove and discharge all power sources before you take anything apart.

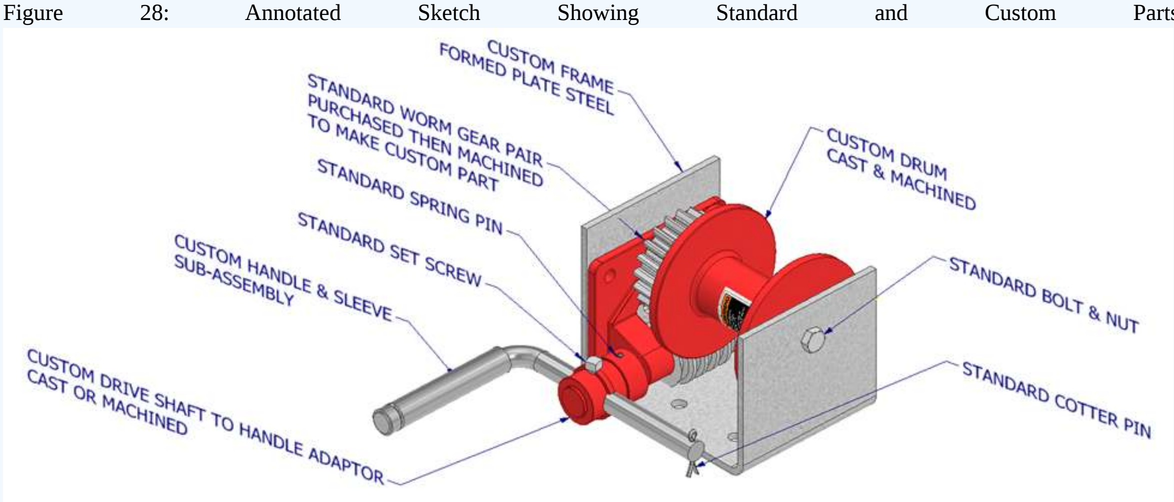
✅ **Key Takeaways from the Case Study:**

- **Standard fasteners** allow for mass production and replaceable parts.
- **Purchase parts** reduce design complexity and offload risk.
- **Unique parts** designed with parametric modeling practices allow engineers to adjust individual parts without affecting the entire assembly.

? Exercise Runaway argument for \eatSpaces?

💡 **Exercise:** Create a sketch of an assembly with components labeled as either standard or custom. If custom, guess at the manufacturing process used to make it. See the example below.

Figure 28: Annotated Sketch Showing Standard and Custom Parts



SECTION OVERVIEW

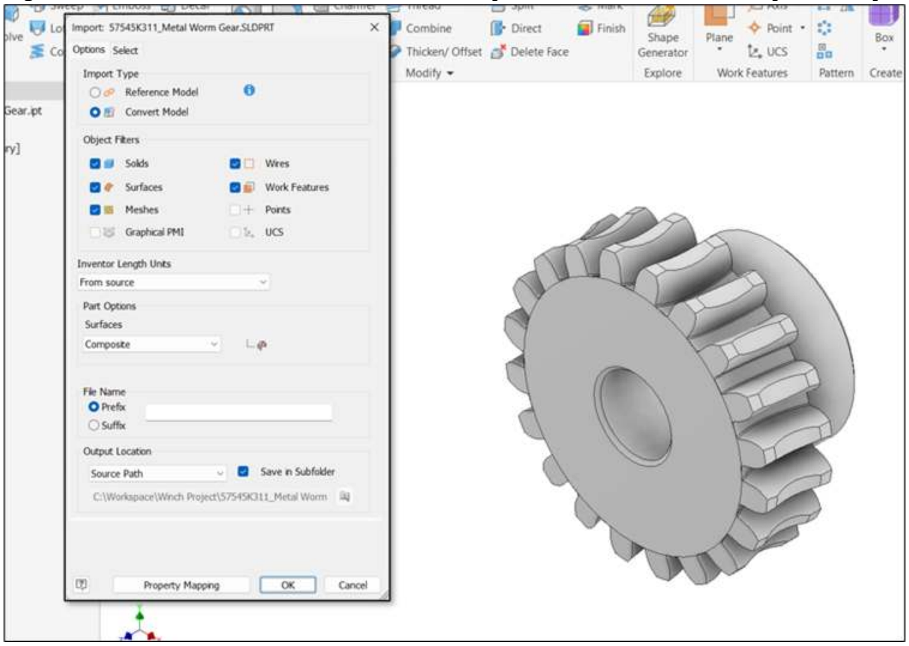
2.2: Importing Purchased Parts into Inventor

Topic hierarchy

2.2.1: Using Standard Parts in CAD

A variety of designs incorporate commercially available components, such as bearings, bolts, and gears. Autodesk Inventor allows importing models from vendor catalogs to ensure compatibility in assembly.

Figure 29: In Most Cases Accept the Default Import Options by Clicking ‘OK’



2.2.2: Updating iProperties from Vendor Models

When importing purchased components, it is essential to update their **iProperties** to maintain accurate documentation. This includes:

- **Save to Standard Parts Library:** Ensuring use in this and other designs, save imported files to a common folder in your Inventor Workspace. Name this folder ‘Library.’
- **Part Number & Description:** Updating iProperties ensures correct identification in BOMs.
- **Material Properties:** Adjusting density and mechanical characteristics ensures accurate weights in top-level assemblies.

? Exercise Runaway argument for \eatSpaces?

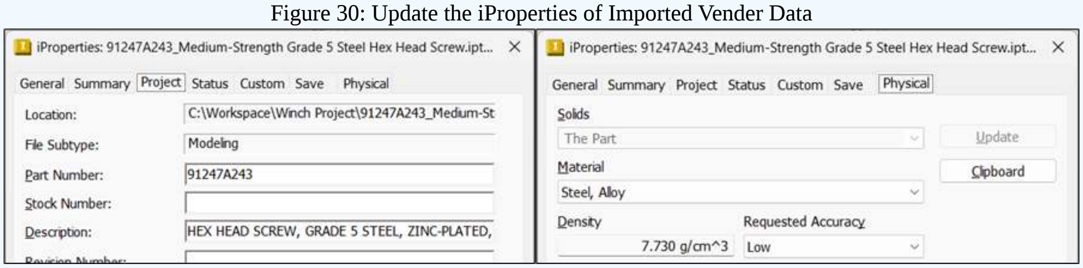
💡 **Exercise: Importing a Bolt and Nut from McMaster**

Using the McMaster-Carr website, students will import a **Hex Head Screw, Grade 5 Steel, Zinc-Plated, 3/8"-24 Thread Size, 6" Long, Partially Threaded. Then import the corresponding nut.** Then, update their iProperties accordingly. (McMaster-Carr, 2025)

✅ **Instructions:**

1. **Navigate to McMaster-Carr’s website** and locate the bolt and nut.
2. **Download the STEP file** for each component using search and filters.
3. **Import the files into Inventor** using the "Open" command.
4. **Update iProperties:**
   - Set the **Part Number and Description** to match McMaster’s catalog.
     - Part Number: “91247A243”
     - Description: “HEX HEAD SCREW, GRADE 5 STEEL, ZINC-PLATED, 3/8"-24 THREAD SIZE, 6" LONG, PARTIALLY THREADED”
   - Adjust **Material Properties** to match the vendor specifications.
     - Material: “Steel, Alloy”

Figure 30: Update the iProperties of Imported Vender Data



2.2.3: Modifying Vendor Models for Unique Assemblies

Sometimes it is necessary to modify a purchased part to suit the requirements of the final design. In this example we will download the worm and worm gear pair and modify them to fit our assembly of the hand winch we will make at the end of chapter project.

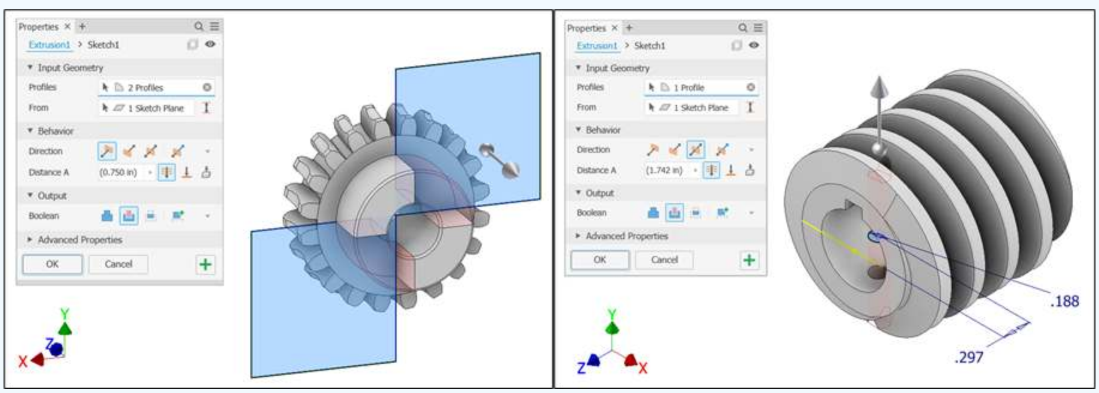
? Exercise 2.2.3.1

💡 **Exercise: Importing worm gear pair from McMaster**
Students will import a **worm gear pair** from McMaster-Carr and save and update their iProperties accordingly. Then create a copy of the vender model saved with our own part number that we will make edits.

✅ **Instructions:**

1. **Navigate to McMaster-Carr's website** and locate the worm gear pair:
   - **Turn on the following filters:** Inch; Gear Pitch = 8; Speed Ratio = 20.
2. **Download the SLDPRT file** for each component.
3. **Import the files into Inventor** using the "Open" command.
4. **Update iProperties:**
   - Set the **Part Number and Description** to match McMaster's catalog.
   - Adjust **Material Properties** to match the vendor specifications.
5. **Save** and Close:
6. **Open a new file:** Save the worm as 606115-213 and worm gear as 606115-214.
7. **Make edits** according to the markup below:

Figure 31: 606115-213 and 606115-214.



8. **Update iProperties** and save, then you are done.

2.3: Part Modeling in Inventor

Topic hierarchy

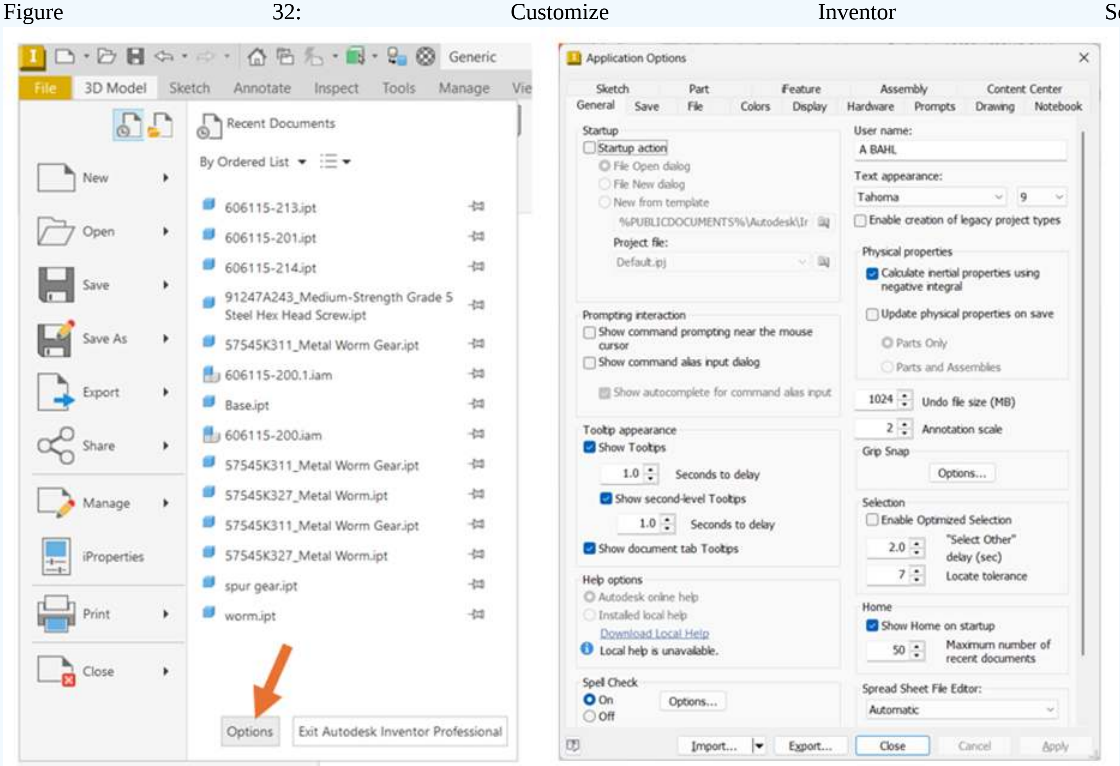
2.3.1: Customizing UI for Efficient Workflow

Designers can tailor the Autodesk Inventor interface to optimize productivity. Here is a selection of customizations that I found useful when I was a professional designer. As we make changes, read the options available in each tab so you know what customizations are available to you. Consider the changes below as a starting point for you, as you customize Autodesk Inventor to suit your own style and workflow. Customize Autodesk Inventor by making the following changes:

Exercise Runaway argument for \eatSpaces?

Exercise: Modify Inventor's UI to create a personalized workspace layout by going to File Options.

Figure 32: Customize Inventor Settings



2.3.2: Conveying Design Intent in a 2D Sketch

A well-structured **2D sketch** is the foundation of parametric modeling. Design intent refers to the **logical relationships** that ensure adaptability.

✅ **Best Practices for Conveying Design Intent:**

- **Use constraints strategically** to maintain geometric relationships.
- **Apply names to dimensions** to allow convenient modifications.
- **Avoid frivolous constraints** that do not relate to the function of the part.

2.3.3: Geometric Constraints in Inventor

Inventor provides **geometric constraints** to define relationships between sketch elements:

- **Coincident:** Ensures two points or a point and a line remain connected.
- **Collinear:** Forces two lines to lie on the same axis.
- **Parallel:** Keeps two lines equidistant along their length.
- **Perpendicular:** Ensures two lines intersect at 90 degrees.
- **Horizontal/Vertical:** Aligns lines along the X or Y axis.
- **Equal:** Forces two entities to have the same size.
- **Symmetric:** Maintains symmetry across a centerline.
- **Tangent:** Ensures a curve touches a line or another curve at a single point.

Exercise Runaway argument for \eatSpaces?

💡 **Exercise:** Apply each constraint to a sketch and observe how modifying one feature affects the entire design

2.3.4: Sketched Features

Extrude, Revolve, Sweep, and Loft are common sketched features. As the name suggests, a sketch is consumed during the construction of sketched features.

Exercise Runaway argument for \eatSpaces?

💡 **Exercise:** Create a fully constrained parametric sketch and convert it into a 3D model using Inventor's feature tools

2.3.5: Placed Features

Fillets, Chamfers, and Holes are common placed features. These features can be placed without a sketch. The Hole tool does create a sketch for the hole center locations that you can edit after placing the holes.

SECTION OVERVIEW

2.4: Assembly Creation in Inventor

Topic hierarchy

2.4.1: Why Assemble?

2.4.2: Applying Assembly Constraints

2.4.1: Why Assemble?

Building an assembly in CAD provides designers with the opportunity to explore the fit and function of a final product. Designers carefully apply **constraints** to the parts that mimic the support provided by the CAD model's real-life counterparts and can animate the assembly to better understand its performance.

2.4.2: Applying Assembly Constraints

Once parts are designed, they are assembled using **constraints** to define relationships between components.

✅ **Key Assembly Constraints:**

- **Mate/Flush:** Controls contact surfaces.
- **Angle:** Defines rotational relationships.
- **Tangent:** Defines surfaces tangent to each other.
- **Insert:** Aligns circular features like bolts in holes.
- **Symmetry:** Conveys a mirrored relationship.

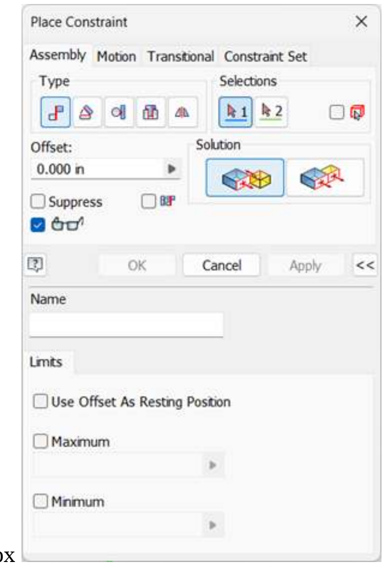
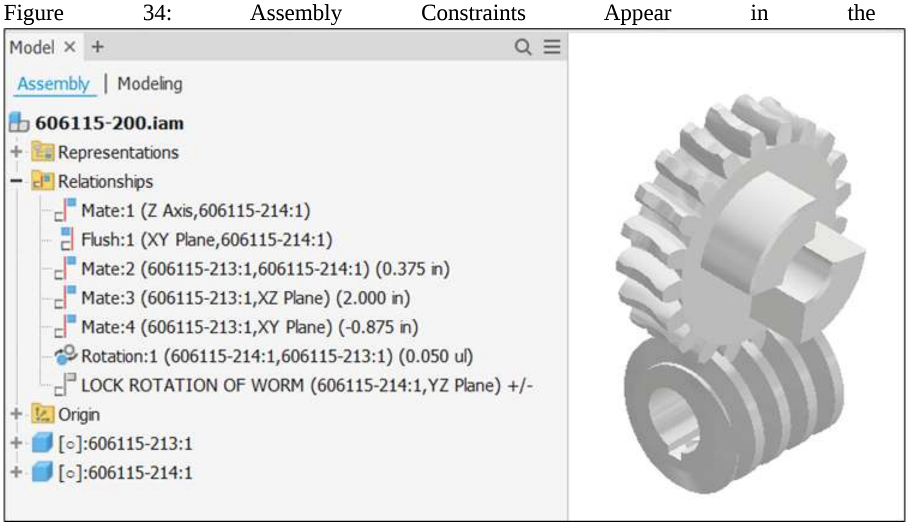


Figure 33: Place Constraint Dialog Box

? Exercise Runaway argument for \eatSpaces?

💡 **Exercise:** Assemble a worm gear pair with shafts using constraints.

Figure 34: Assembly Constraints Appear in the Relationships Folder



SECTION OVERVIEW

2.5: Project 2- The Worm Gear Driven Hand Winch

Topic hierarchy

2.5.1: Project Objectives and Design Requirements

This project integrates all bottom-up design principles:

- **Designing individual parts** using parametric modeling techniques.
- **Importing purchased components** (gears, fasteners).
- **Applying assembly constraints** for motion validation.

💡 **Final Deliverables:**

- 3D **parametric Inventor models** of all components.
- Fully **assembled and constrained winch system**.
- **Complete drawing package of technical drawings** following industry standards.

CHAPTER OVERVIEW

3: Top-Down Design

SECTION OVERVIEW

3.1: Introduction

Topic hierarchy

3.1.1: Understanding Top-Down Design in CAD

Top-down design is an **assembly-first** approach where the overall design is conceptualized first, and individual parts are derived from it. This method is ideal for projects requiring **highly integrated components**, ensuring precise interdependencies between features. It excels in custom design where changes are frequent, and components are not meant to be reused in other products.

Comparison:

- **Bottom-Up Design:** Individual parts are created separately and then assembled.
- **Top-Down Design:** The full system is designed first, and individual components are generated within the assembly.

Case Study: Top-Down Approach in Automotive Chassis Design

Automotive manufacturers use **top-down design** when developing vehicle chassis, ensuring proper integration between the body, frame components, suspension systems, and drivetrain elements. Engineers define the **structural framework** first, using multi-body part modeling and parametric relationships to guide the design of individual sections.

✅ **Key Takeaways from the Case Study:**

- **Interdependent design ensures structural integrity** while maintaining flexibility.
- **Parametric adaptability** allows engineers to modify chassis elements without affecting the overall system.
- **Streamlined assembly creation** reduces iteration cycles during manufacturing.

? Exercise Runaway argument for \eatSpaces?

💡 **Exercise:** Identify a real-world example of top-down design in mechanical engineering

SECTION OVERVIEW

3.2: Fits, Limits, and Dimensioning Principles

Topic hierarchy

3.2.1: Understanding Mechanical Fits

Fits determine how parts interact in an assembly, ensuring proper function and manufacturability.

✅ **Types of Fits:**

- **Clearance Fit:** Provides space between mating parts, allowing free movement (e.g., shaft and bearing).
- **Interference Fit:** Ensures a tight press-fit connection, eliminating unwanted motion.
- **Transition Fit:** Balances between clearance and interference, offering controlled movement.

3.2.2: Applying Limits and Tolerancing Standards

Dimensioning principles define **acceptable variations** in part size to ensure proper fitment and manufacturability. Designers apply **Geometric Dimensioning & Tolerancing (GD&T)** methods to specify tolerances in precision assemblies.

Exercise Runaway argument for \eatSpaces?

💡 **Exercise:** Look up limits in "Machinery's Handbook." Check the Internet Archive for a digital copy

3.3: Multi-Body Part Models

Topic hierarchy

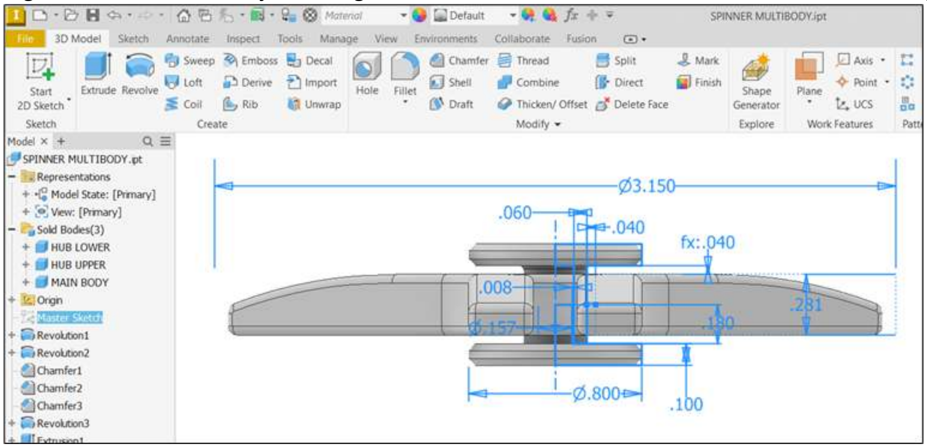
3.3.1: Why Multi-Body Modeling is Essential in Top-Down Design

Multi-body part modeling allows engineers to define complex **interdependencies** within a single part file before separating components for manufacturing.

✅ **Advantages of Multi-Body Models:**

- Maintain **parametric relationships** between components.
- Simplify **design updates** by controlling all changes within a single model.
- Improve **efficiency** by reducing time spent on part modifications.

Figure 35: Multibody Designs Start with a Master Sketch Outlining the Overall Design



SECTION OVERVIEW

3.4: Project 3- The Fidget Spinner

Topic hierarchy

3.4.1: Project Overview and Design Objectives

3.4.1: Project Overview and Design Objectives

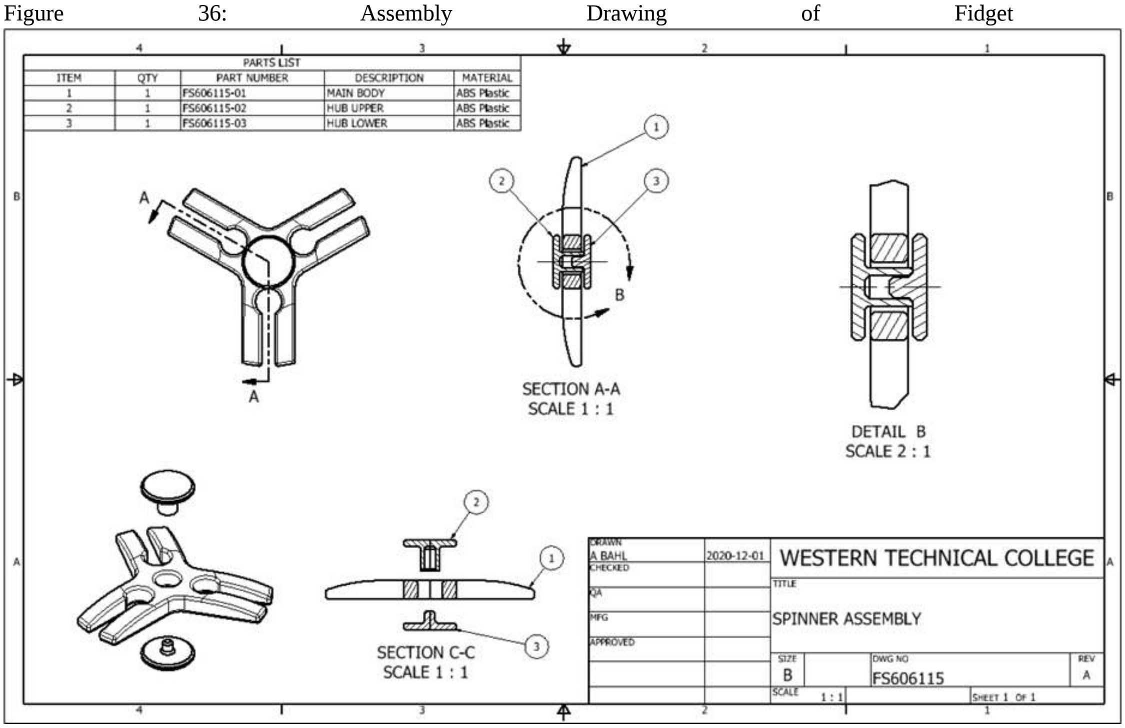
The fidget spinner serves as a **compact mechanical assembly**, reinforcing:

- **Multi-body modeling concepts**.
- **Tolerance and fit principles**.
- **Manufacturability constraints**.

💡 **Final Deliverables:**

- **Multi-body Inventor model** of key components.
- **Engineering drawings** with fits and tolerances specified.
- **3D Printed and fully assembled spinner design**.

Figure 36: Assembly Drawing of Fidget Spinner



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