



Walt Disney World Swan and Dolphin Resort
Orlando, Florida

Up to Cruising Speed with Autodesk Inventor® (Part 1)

Neil Munro - C-Cubed Technologies Ltd.
and Travis Jones

MA21-4 In part one of this three-part introduction to Autodesk Inventor, we'll demonstrate key features and workflows using a real design challenge. You will see an overview of Autodesk Inventor 10 environments (parts, assemblies, drawings), and examine best practices for new users for sketching, creating parts and assembly models, and documenting your designs. Of interest to many, the design challenge also covers working with AutoCAD and Mechanical Desktop data in Autodesk Inventor. All attendees receive a CD with the complete design challenge in tutorial format. Make sure you sign up early for part two (lab) to complete the tutorial with expert help.

About the Speaker:

Neil is a faculty member at the British Columbia Institute of Technology (BCIT) in Burnaby, B.C., where he teaches CAD and programming courses in the Mechanical Technology program. He is actively involved in the Autodesk Inventor community. Along with a variety of tutorials and other training materials, he has written a number of popular Autodesk Inventor add-in programs.

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Travis is a content manager at Autodesk. He has worked with AutoCAD® for more than 15 years and Autodesk Inventor since its inception. He has developed and delivered training courses to countless people in person and over the Internet in virtual training classes. Travis is the coauthor of eight books about Autodesk Inventor, and is a regular speaker at Autodesk University.

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Part Modeling

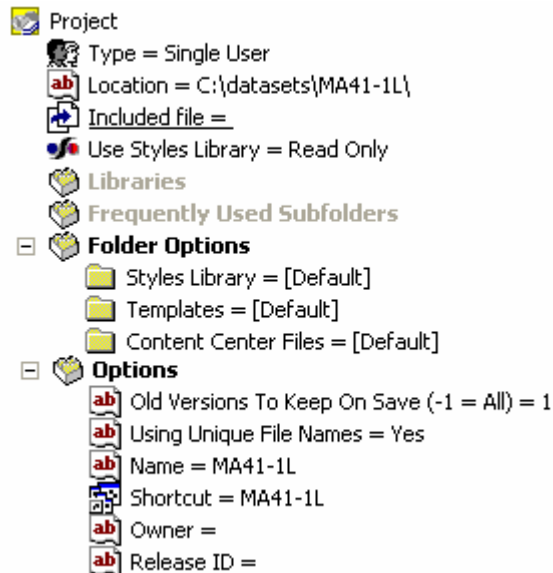
In the following section of the tutorial you will:

- Review Project files
- Create a simple part that will adapt its size to match other components later in the tutorial
- Open an Autodesk Inventor assembly file and explore the UI and viewing tools
- Review an engineer's note for creating the Claw component
- Start a new part and import an AutoCAD drawing
- Add sketched and placed features to complete the claw

Project File Review

Autodesk Inventor uses projects to logically organize files and to maintain valid links between files. A project consists of a project home directory, a project file that specifies the paths to the locations of the files in the project, and the local and network folders containing other Autodesk Inventor files that may be required in the project.

For multi-user teams, you can use Autodesk Vault to manage model versions and share the design data with all team members. We'll use a single user project for our tutorial.



Engineers Notebook

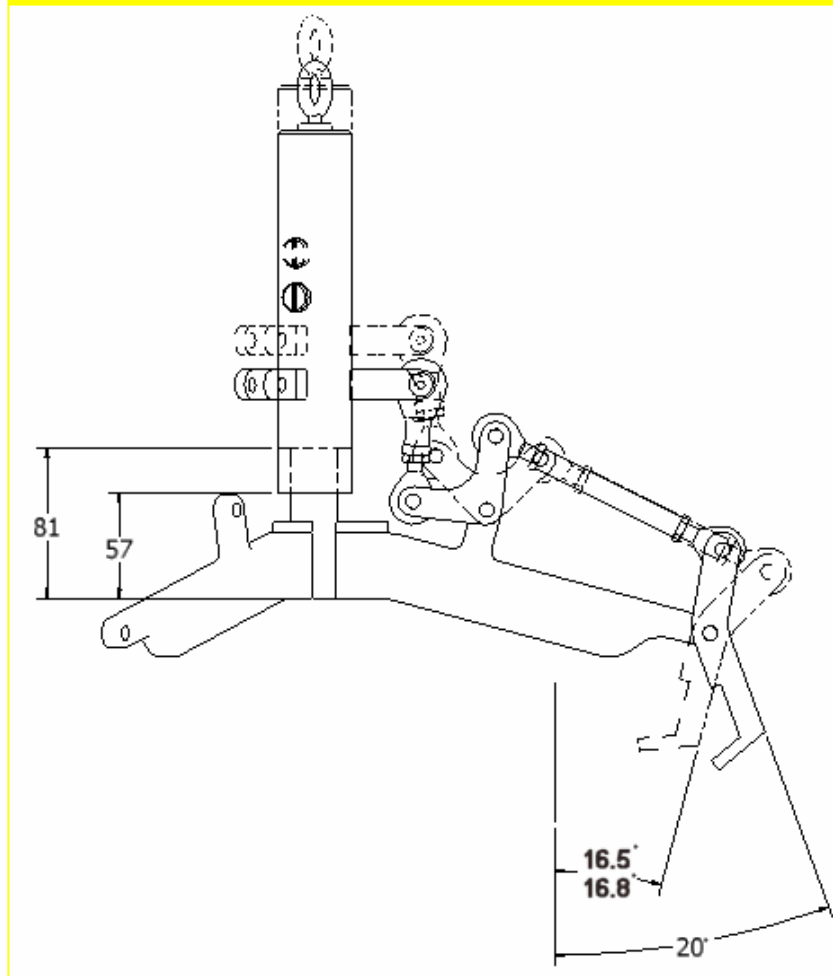
The Engineer's Notebook enables you to attach additional information to sketches, features, parts, and assembly models. You might use this to save a record of how the design was built or to add information about a component from outside sources, such as the results of an FEA analysis on a part. Notes may be used to add information about a feature of the design, or to associate a design question to a specific component in an assembly. You will review an engineer's note to begin the creation of a claw component of the assembly. You can double-click the symbol of an engineer's note to open the notebook or pause with the cursor over the symbol to view the note.

Travis,

While I'm away at Cape Carnival ... errr Canavarel for our initial design review, could you complete the design for the mirror lifting mechanism? We need the following:

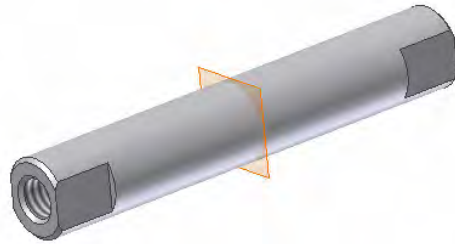
- The claw that Phil was designing in AutoCAD before he was let go (not for using AutoCAD)
- The pivot needs editing to work with the new arm design
- See the attached sketch for the open and closed states of the mechanism. We need a correctly sized link between the pivot and the claw to enable these states.

Thanks,
Neil



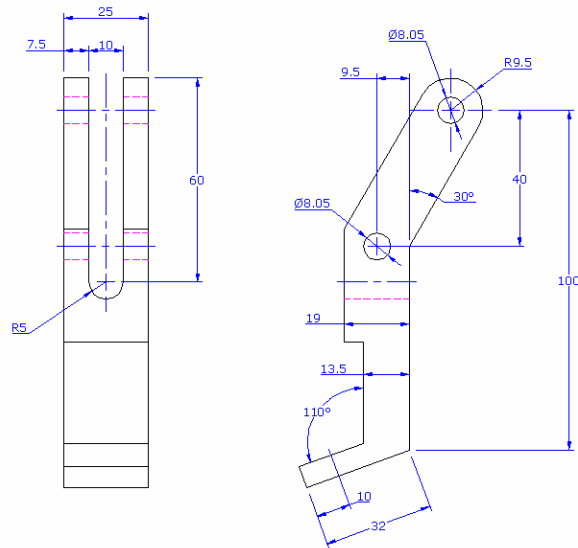
Create a Simple Part

You create a simple rod part from scratch by combining a number of 3D features. You learn some good modeling practices and how 3D modeling is different than the 2D documentation you typically create in an AutoCAD drawing. Since the final length of the rod is unknown, you specify that the length of the initial cylinder feature be allowed to “adapt” to meet design criteria later in the design.



Open an AutoCAD file in Autodesk Inventor

Many Autodesk Inventor users have a large amount of legacy data in AutoCAD or Mechanical Desktop (DWG) format. You can import 2D drawing data into Autodesk Inventor drawings or directly into a part feature sketch. Mechanical Desktop models can be translated into native Autodesk Inventor files or linked into an Autodesk Inventor assembly. You import an AutoCAD .DWG file into a feature sketch in a part, and use the information to create the claw.

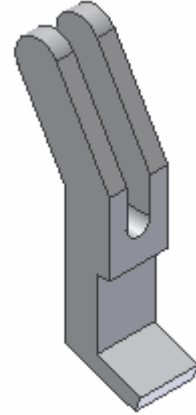


Base Feature

The first sketch of a part that is used to create a 3D feature is referred to as the base feature. In addition to the base feature, you can create sketched features, where you draw a sketch on a planar face or work plane and either add or subtract material to or from existing features in a part. Use the Extrude, Revolve, Sweep, or Loft tools to create sketched features in a part.

As features are added to the part, they appear in the Browser, showing the history of the part or assembly (the order in which the features are created or the parts are assembled). Features can be edited, deleted, or reordered from the part as required.

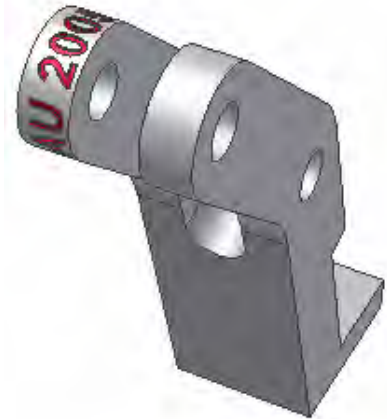
The Extrude tool, for example, extrudes a sketch along a positive or negative axis. This extrusion can form a base feature or add or remove material from a part when other features exist. You will use the extrude tool to create the base feature of the claw and to cut a slot through the top portion of the claw.



Placed Features

Placed features are features that are predefined except for specific values and only need to be located. Placed features can be edited from the Browser like sketched features. When a placed feature is edited, either the dialog box that was used to create it will open, or feature values are displayed on the part.

When creating a part, it is usually better to use placed features instead of sketched features wherever possible. To create a hole, for example, you could draw a circle profile and dimension it, then extrude it using the cut extension option. You can also select the type of hole, size it, and then place it using a dialog box. When drawing views are then generated, the type and size of the hole can be easily annotated and automatically updates if the hole type or values change. The claw contains fillets, and holes and an emboss; all of which are placed features.

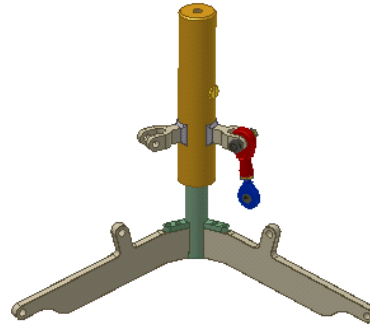


Assembly Environment and Assembly Modeling

In the following section of the tutorial you will:

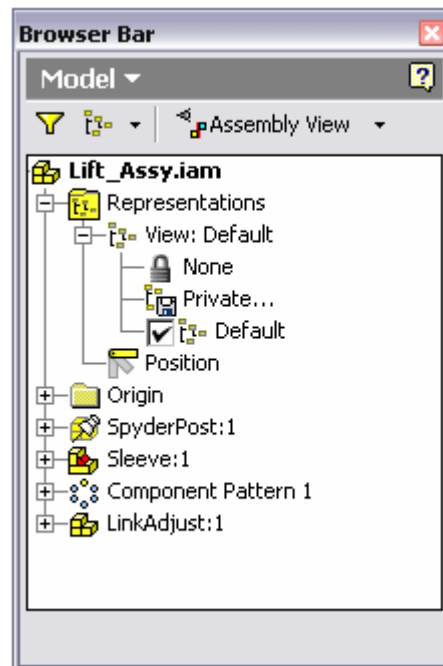
- Examine the Autodesk Inventor® assembly environment
- Place components in an assembly
- Assemble components using assembly constraints
- Test the mechanism by dragging an assembly component
- Translate a Autodesk Mechanical Desktop part to use in the assembly
- Analyze the assembly to check for component interference
- Activate and edit a part in-place in the assembly
- Adapt the size of a link to match other assembly components
- Add iParts and library components to the assembly
- Pattern components in the assembly
- Drive an assembly constraint to analyze the mechanism

An assembly file brings together individual components (parts and subassemblies) and enables you to position and link the components to build a representation of a real world assembly. The assembly file does not contain the individual components; they are referenced into the assembly in a manner similar to external referencing in AutoCAD. You build relationships between the components by adding assembly constraints between the parts. You can build static assemblies such as frames, or dynamic mechanisms that can simulate the motion of the real-world assembly.



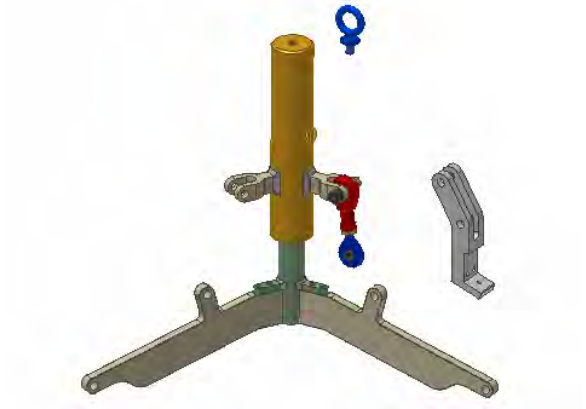
Assembly Browser

The assembly browser lists the components placed in the assembly, in the order they were added. The relationships between assembly components are also listed in the browser. As in the part environment, it is often easier to make selections from the browser rather than in the graphics window.



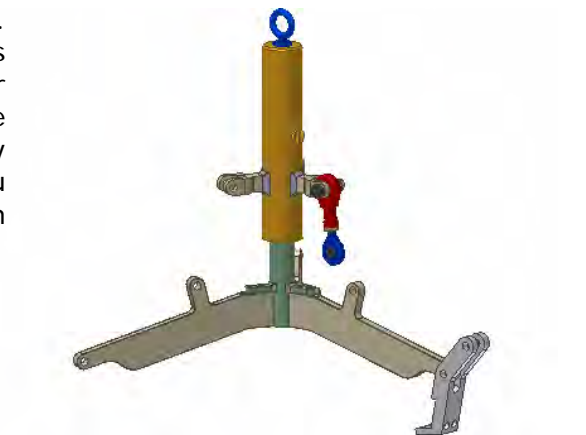
Placing Components in an Assembly

Your first task is to add the claw and lift ring parts to the top-level assembly.



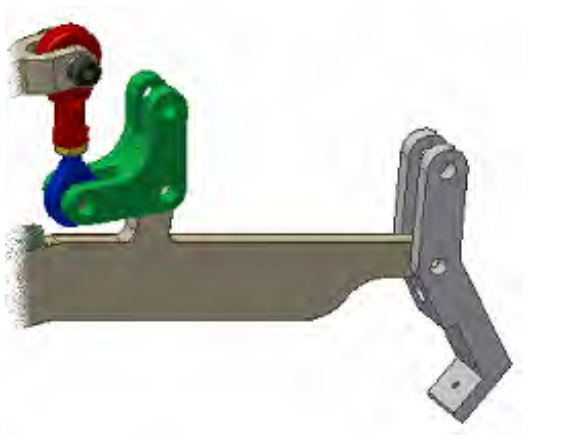
Assembly Constraints

You position these parts with assembly constraints. Assembly constraints mimic real world part relationships such as the alignment of axes, or the mating of planar faces. As you add assembly constraints, you check the remaining degrees of freedom of the component by dragging the component in the graphics window. You then constrain the components as outlined in the design note from the beginning of the tutorial.



Edit a Part in the Assembly

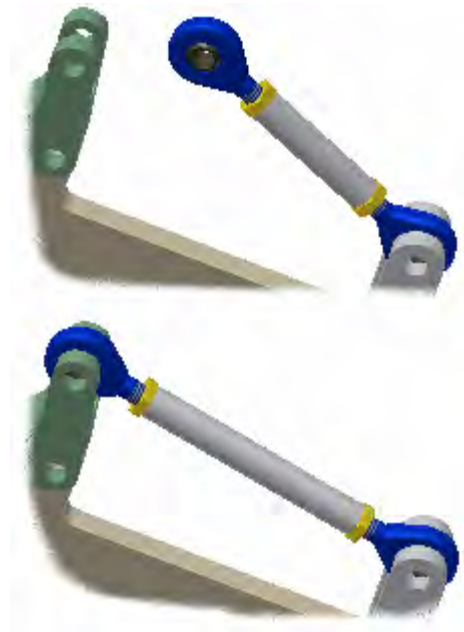
You add a partially completed pivot part to the assembly. You examine the part for interference and make edits to a part in-place in the assembly. During the edit, you add relationships between feature dimensions.



Adaptive Link

Next, you create a subassembly of the link between the pivot and the claw. Using the design data from the engineers' notebook, you set up one of the boundary states, and force the length of the link rod part to adapt to the assembly constraints positioning the link subassembly in the lift mechanism.

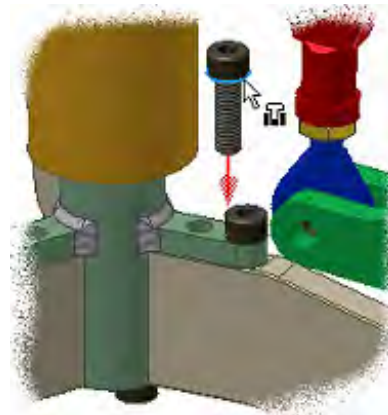
You then create Positional Representations of the open and closed mechanisms, and adjust the pivot geometry to obtain the desired design values.



iParts and iMates

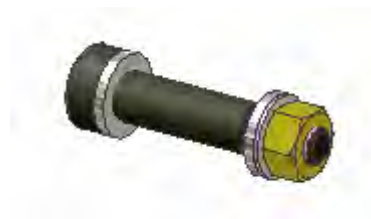
iParts enable you to create a family of similar parts from a base definition. You add a new variation to an existing iPart, and then place and assemble multiple occurrences in the assembly.

iMates are predefined assembly constraint halves that enable parts to self-assemble when placed in an assembly. You examine the iMate included in your iPart.



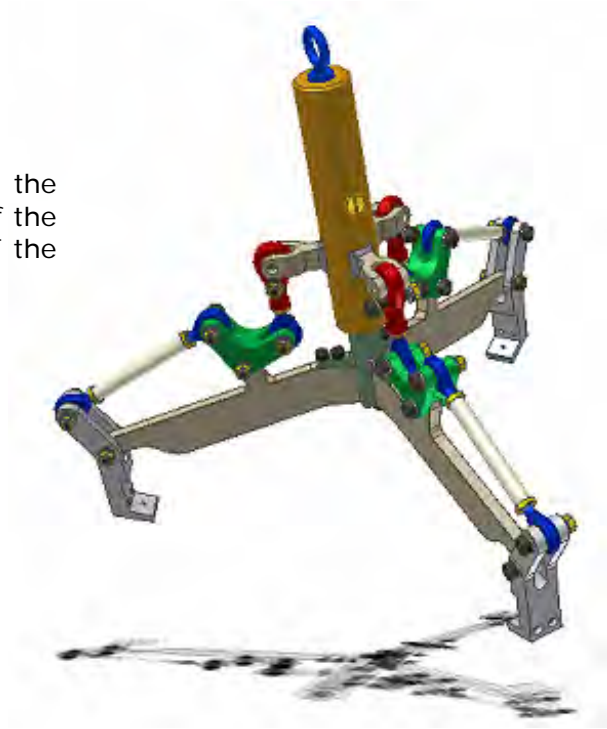
Content Center (Library) Parts

Autodesk Inventor includes a comprehensive library of standard components that can be used in place of or in addition to iParts. You place parts from the content center, and constraint them in the assembly.



Pattern Components

To complete the assembly, you pattern the linkage components. Finally, you drive one of the assembly constraints to view a simulation of the lifting mechanism.



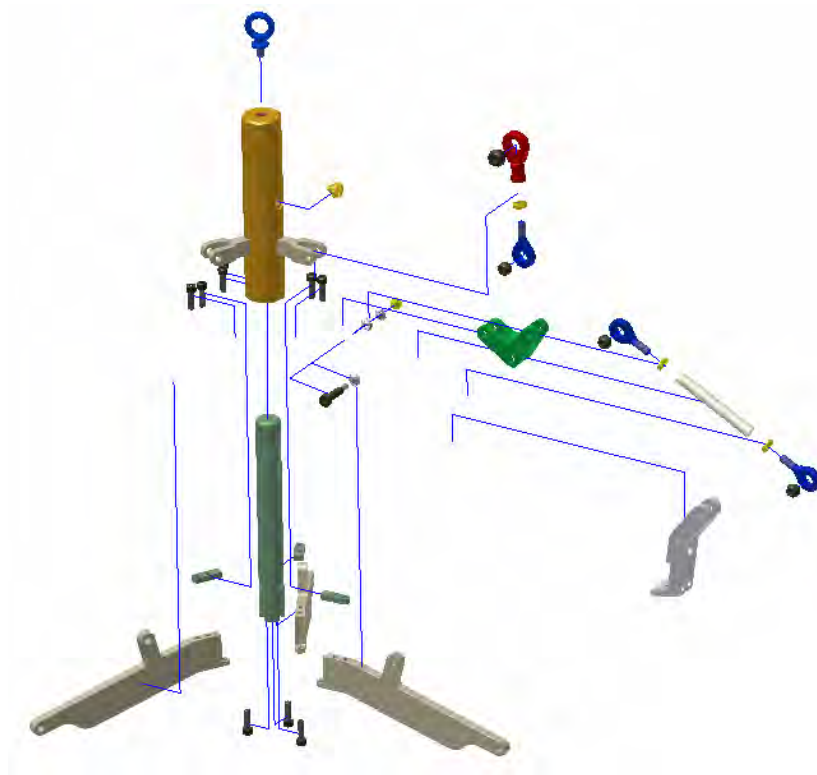
Presentation Files

In the following section of the tutorial you will:

- Create a Presentation file
- Add tweaks to assembly components to create an exploded view
- View animations of the component tweaks
- Use the Sequence view to review tweak motion

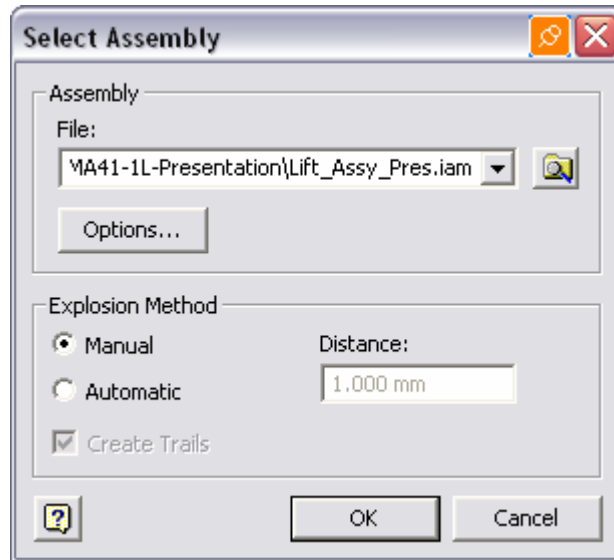
Presentation files give you the ability to illustrate how components interact with other parts and sub-assemblies within a single assembly. These interactions are represented using tweaks, which are segments of translation or rotational movement of a component within the assembly.

A presentation file is a separate file from an assembly and has a file extension of .ipn. The presentation file is associated with the assembly that it references, and changes made to the assembly file are reflected in the presentation file. You cannot create components or modify the geometry of them when working in a presentation file. To create a new presentation file, select the *New* icon from the What To Do section and then select the *Standard.ipn* icon from the Default tab.



Creating a Presentation View

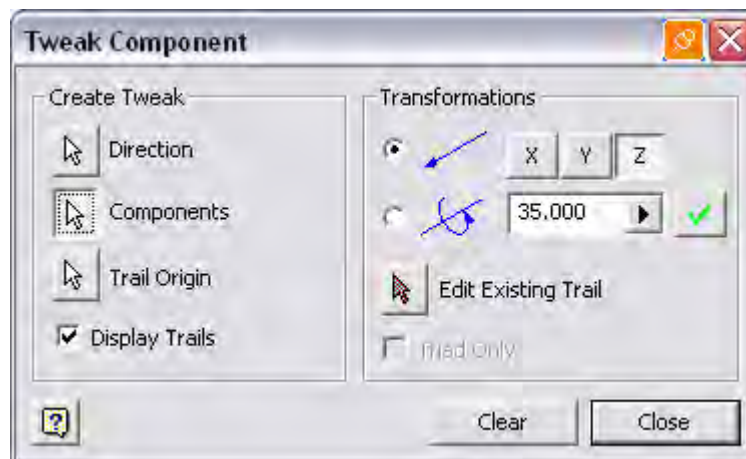
To create a presentation file (.ipn), you specify an existing assembly file (.iam) that you want to work with using the Create View tool. After selecting the Create View tool, the select Assembly dialog box is displayed.



Creating Tweaks and Trails

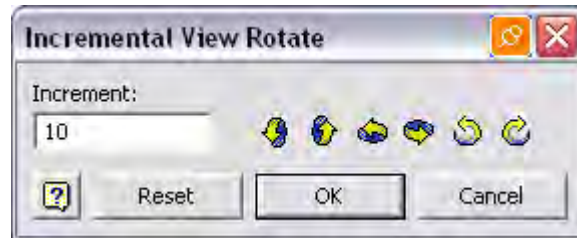
After a presentation view has been created, you add tweaks to the components to represent the assembly in its exploded state. Trails are lines that represent the path of the component from its original location to its location in the exploded state. Trails can be automatically created when tweaking components and can be suppressed to either display or not display the path of the component. If a tweak is created without the trail visibility on, it can be shown from the Browser at a later time.

To edit an individual tweak, double-click on its value in the Browser and enter a new value in the edit box that is located in the lower left corner of the Browser.



Precise View Rotation

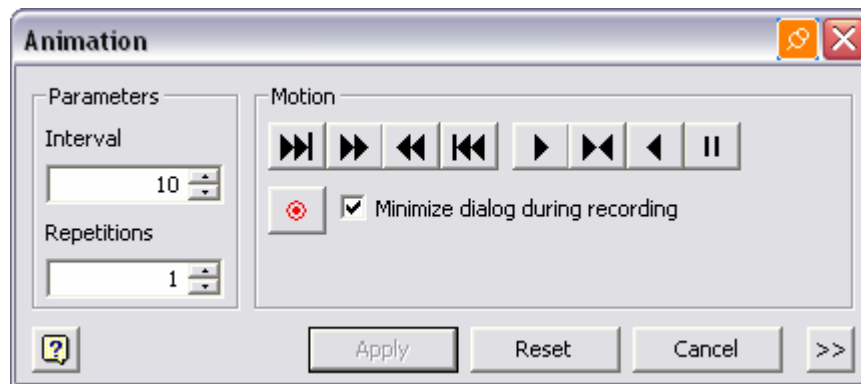
The Precise View Rotation tool adjusts the camera location or viewing angle in a presentation view. After selecting the tool from the Presentation Panel Bar, specify an increment in degrees then click one of the Rotate or Roll buttons to change the view alignment. You can also use the Zoom, Pan, Rotate and Common View tools from the standard toolbar to orient the viewing angle.



Animation

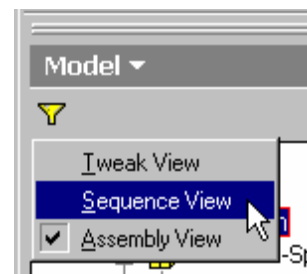
After the components have been repositioned, you can animate the components to show how they assemble or disassemble. During an animation of a presentation view, components move along the tweaks you have created. The animation can be saved to an external AVI file using the record button.

Select the Animate tool from the Presentation Panel Bar, or right-click and select Animate from the menu to create an animation. The Animation dialog box is displayed where you can control or modify the animation sequence.

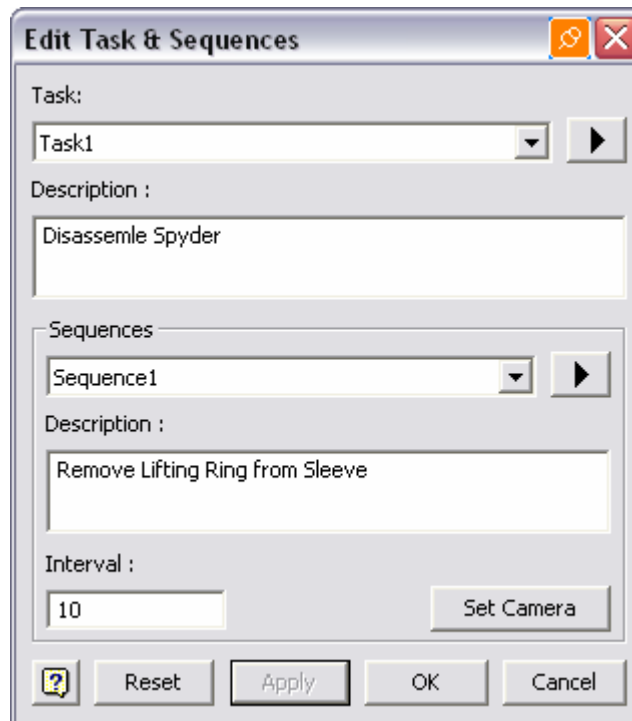


Types of Presentation Views

There are three options available for working in the presentation environment. Assembly, Sequence, and Tweak Views provide different capabilities and work flows for modifying your presentations. To navigate between the types of views, click the Browser Filters button from the top of the Browser. The viewing options effect how the Browser displays the presentation file and its components.



The Sequence View divides the Browser into Tasks and Sequences that provide additional capabilities to arrange your animation. There is no limit to the number of tasks or sequences that can be defined within the presentation file. Sequences and tasks can be combined, added, or deleted as necessary. The Sequence View provides powerful editing capabilities for your animation. Right clicking a task or sequence from the Browser and selecting Edit, opens the Edit Task & Sequences dialog box where you can assign different camera angles to the presentation, increase or decrease the speed of specific tweaks, and include assembly instructions that can be displayed as the animation is played.



Drawings – Documenting Your Model

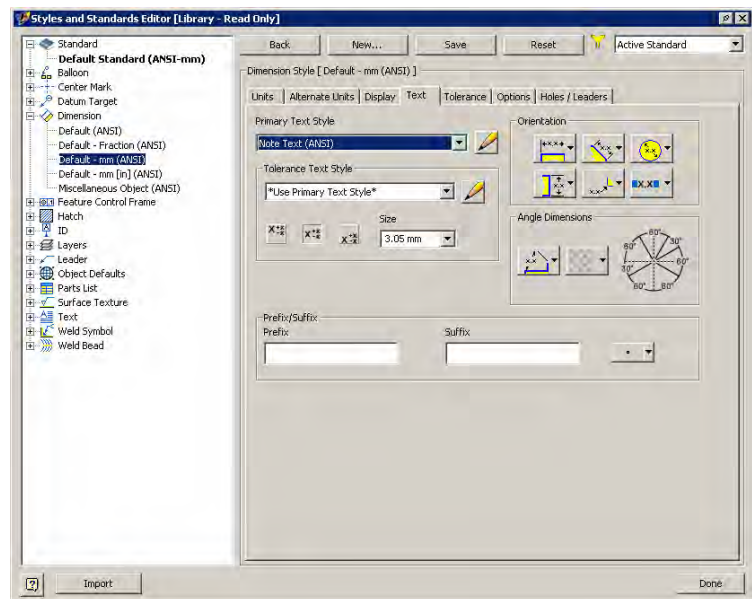
In the following section of the tutorial you will:

- Review the drawing environment in Autodesk Inventor
- Add text and dimension styles to the drawing
- Place an exploded view of the assembly from a presentation file
- Create a parts list and balloon the components in the exploded view
- Add a second sheet to the drawing
- Create various views of a part
- Add annotations to document the part

In Autodesk Inventor you can automatically create two-dimensional (2D) drawing views to represent that part or assembly. You start a new drawing file and then select a part, assembly, or presentation on which to base the drawing views. You place a base view of the model, and then add other views such as orthographic projections, sections, and detail views to document the model as required. The model and drawing views are associative. If the model changes, the drawing views automatically update to reflect the changes. A comprehensive set of annotation tools is available to complete the documentation.

Drafting Standards and Styles

You open a blank drawing and review the styles controlling the appearance of annotations in the drawing. You add a new text style and dimension style. Styles are made available from style libraries associated with the project but all styles used in the drawing are cached in the document.

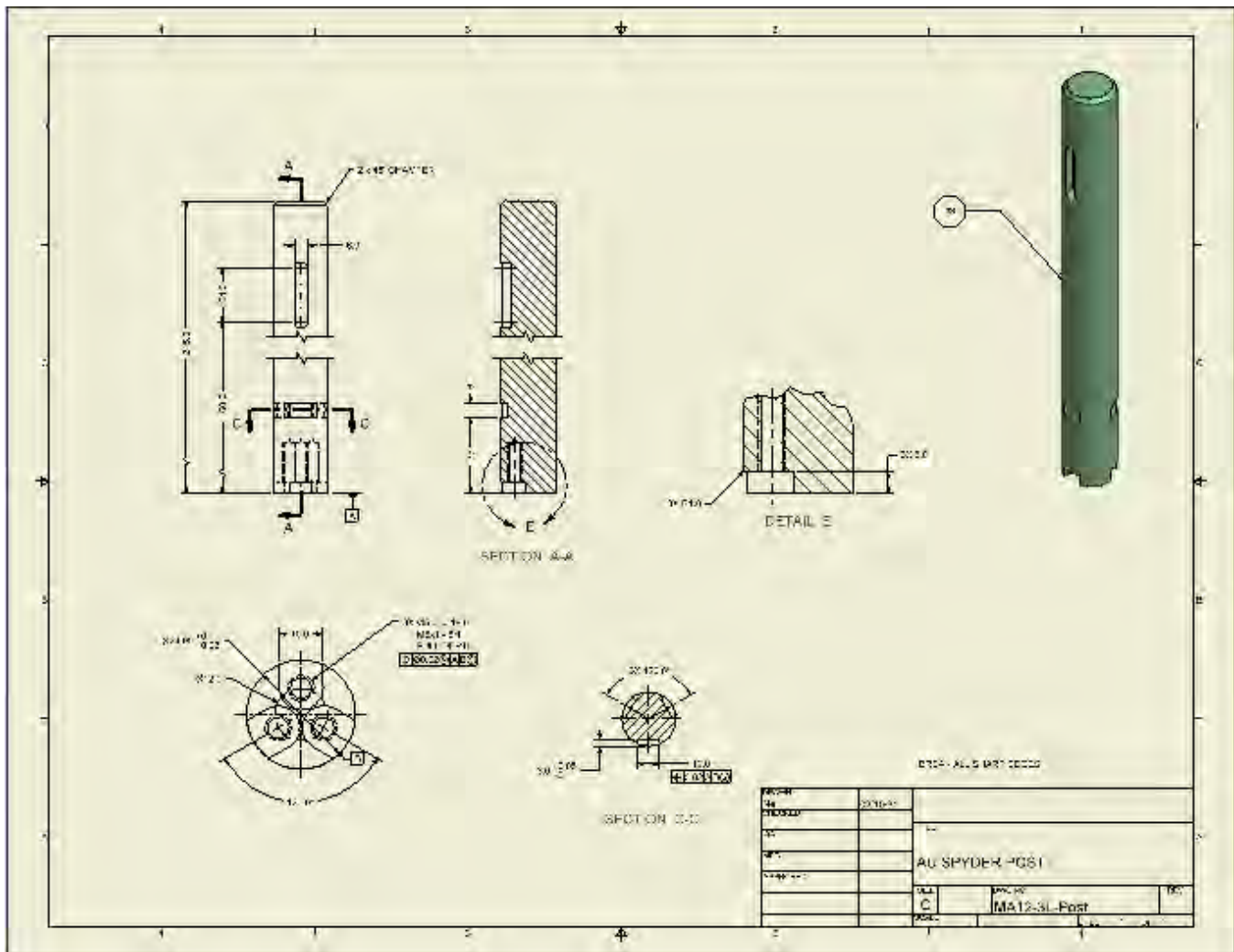


Part Drawings

You add a second sheet to the drawing. A base view of one of the parts is placed on the sheet. You then add projected, section, detail, and broken views to enable complete documentation of the part.

Annotations will include centerlines, dimensions, hole notes, text, datum identifiers and feature control frames.

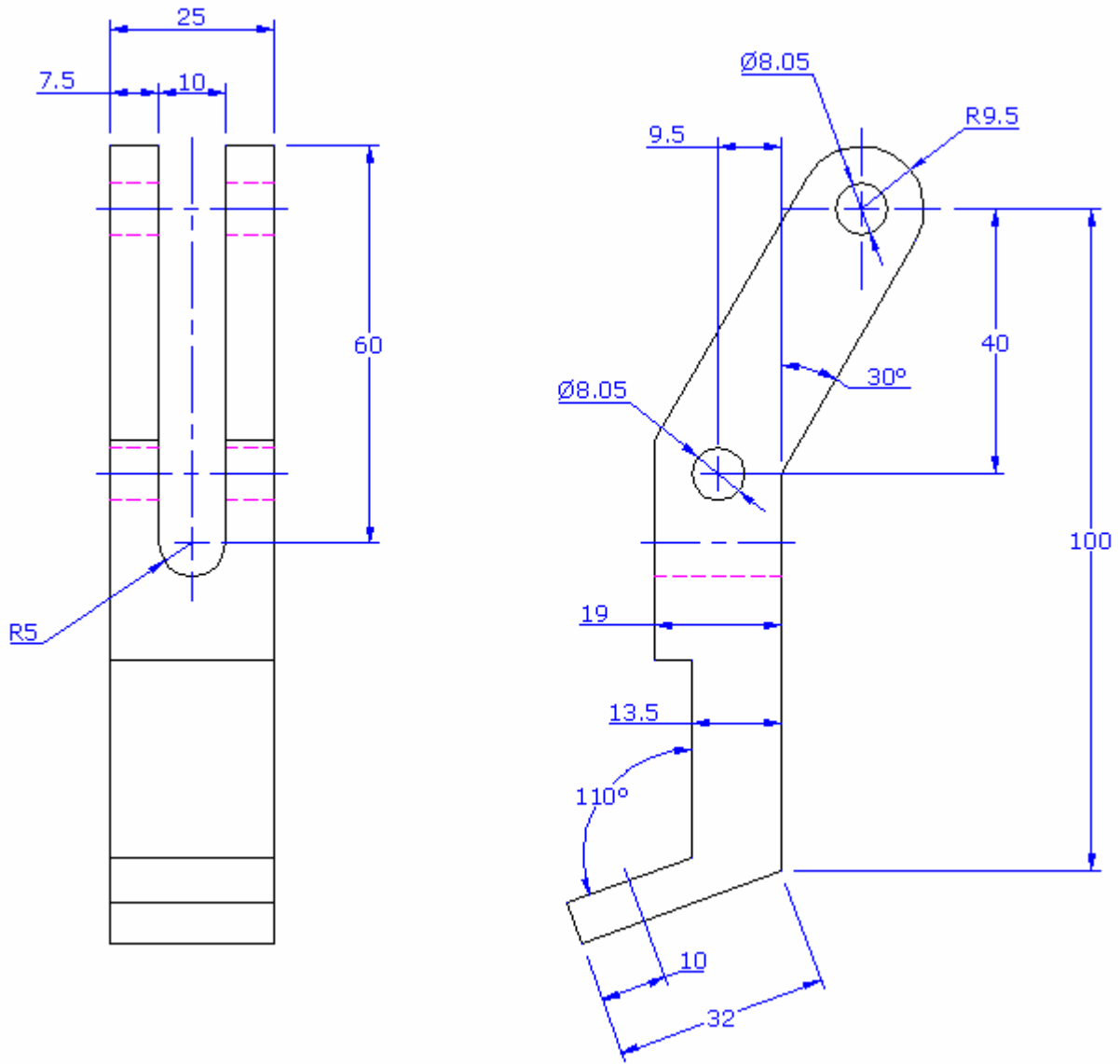
Finally, you create and place a sketched symbol that contains a model property. You change the model and the symbol updates to match the changes.



In the lab session, feel free to experiment and ask questions if you have them.

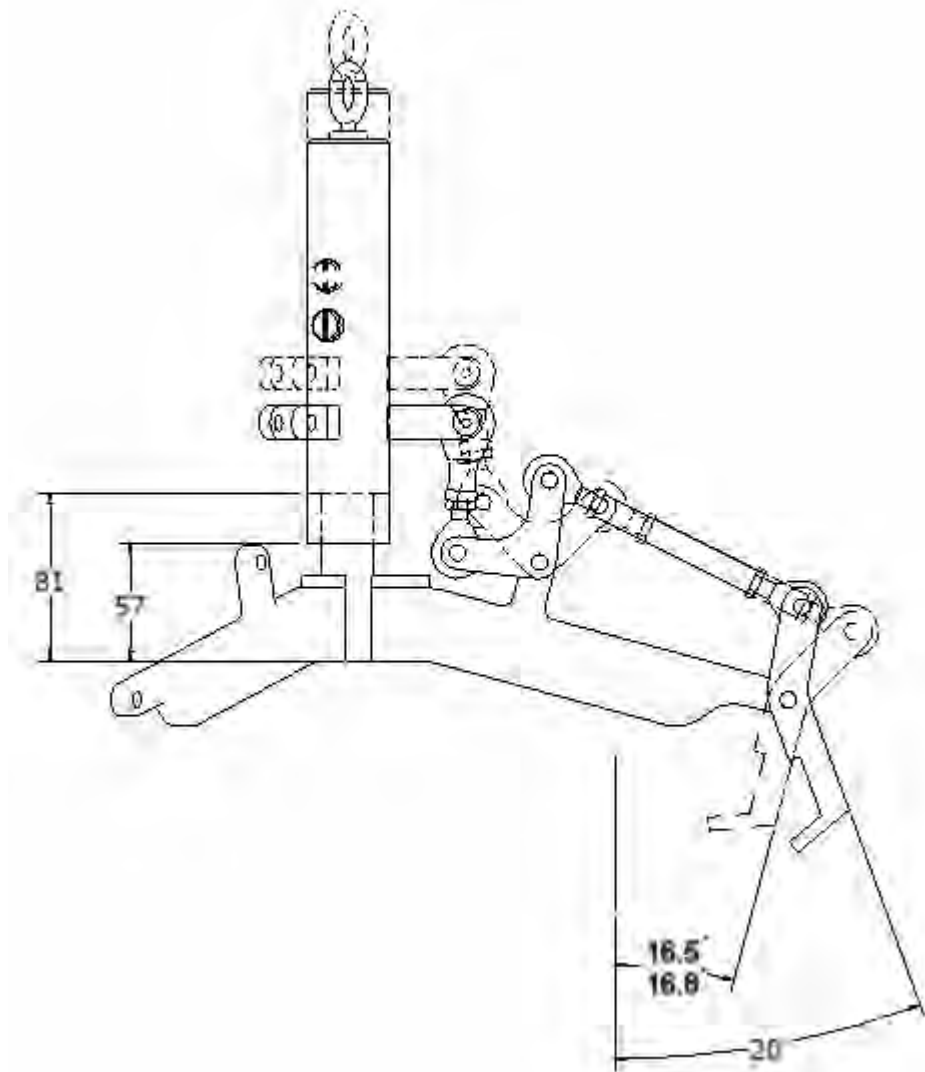
HAPPY INVENTING!

Appendix A



Claw.dwg

Appendix B



Open and closed mechanism states