Project - Design a Bike Carrier

The content in this project is derived from the course *Introduction to Engineering Design* developed by Project Lead the Way, Inc.

In this project, you design a bike carrier for the bed of a pickup truck using Autodesk Inventor software.

Design Statement

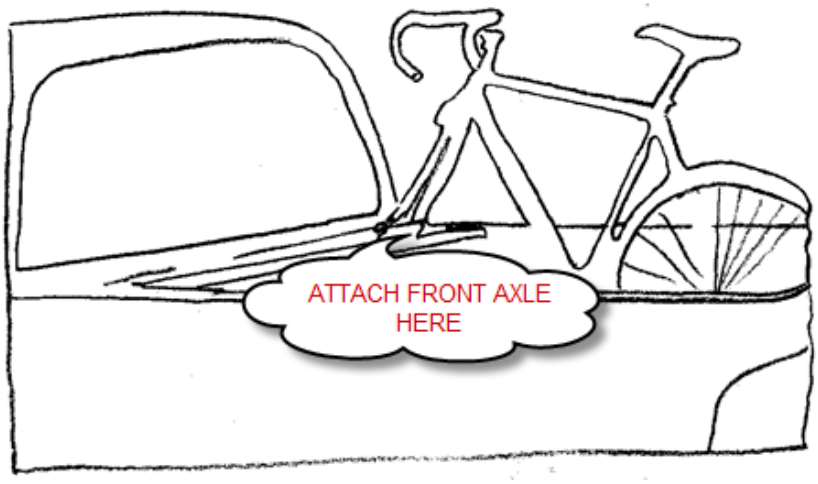
Transporting bicycles in or on a motor vehicle can be a painstaking and sometimes clumsy task. Simply throwing a bicycle in the bed of a pickup truck is unsafe and can cause damage to both the truck and the bike. There must be a secure and efficient way of placing bicycles in the beds of pickup trucks.

You are part of a design team who has the task of creating a bicycle rack/carrier that will fit in the bed of pickup trucks. Using the skills you learned in Pre-Mechanical Engineering, you present your ideas in multiple-view sketch form to the instructor. Upon approval, you translate your ideas to the solid modeling software showing all necessary views, dimensions, assemblies, and parts lists.

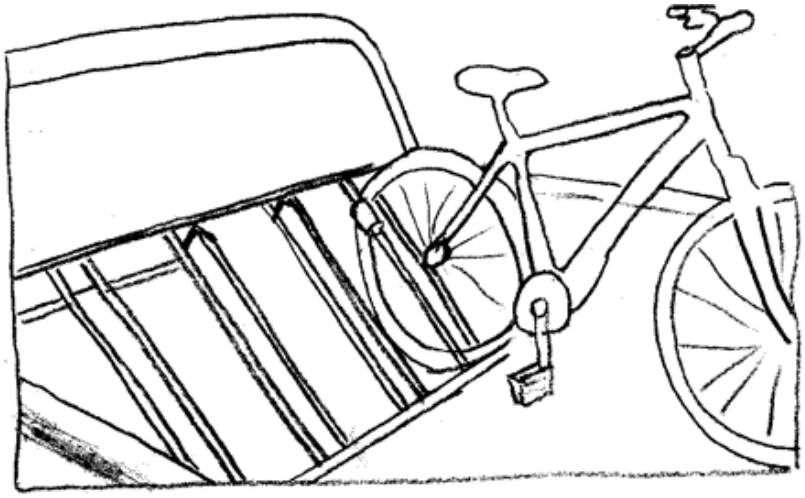
Design Constraints

- Object must be lightweight and durable.
- Easily attached, removed, and transported.
- Able to hold at least two bicycles safely.
- Fully adjustable to fit several different size pickup trucks.
- Able to hold different sizes and styles of bicycles.

The first design sketch shows a single adjustable tube stretching across the truck bed. The front wheel of the bike is removed and attached to the tubing. Ideally this should be a quick-release device.



The second design sketch shows an adjustable rack that hold up to four bikes. The wheels do not have to be removed. The rear wheel is locked using a lockable cable.



Resources

- Design Basics Exercises
- Pre-Mechanical Exercises
- Related web pages. Use **bike carrier pickup truck** as keywords in the search.
- Examples of bike carriers

Procedure

1. Define the design problem.
2. Conceptualize the design.
3. Design the bike carrier in Autodesk Inventor.
4. Document the assembly in Autodesk Inventor.
5. Create a portfolio of the design project.
6. Create a prototype model. (optional)

Suggested Outputs

1. Sketches of your design idea.
2. Documentation and drawings of the assembly.
3. A set of drawings suitable for manufacturing the assembly. Use an A or B-size sheet with suitable scales for all views. The first sheet should an isometric of the assembly. Add a new sheet for the each part of the assembly.
4. Annotate the assembly and part drawings. This includes:
   - Balloons and a parts list on the assembly drawing.
   - Center lines
   - Dimensions
   - A title block complete with iProperties.
5. The widths of the parts list columns are:
   - Description: 40 mm (2 inches)
   - Part Number: 30 mm (1.25 inches)
   - Material: 45 mm (2.25 inches)
6. A high-quality rendered image created in Inventor Studio.
7. An animation of the assembly created using drive constraints or Inventor Studio.

Project - Design a Cookie Cutter

The content in this project is derived from the course *Introduction to Engineering Design* developed by Project Lead the Way, Inc.

In this project, you design a cookie cutter using Autodesk Inventor software.

Design Statement

You have been asked to bring homemade cookies to a special function. You would like to create large uniquely shaped cookies, but are unable to find a commercially available cookie cutter in the shape you desire. After talking to your technology teachers, they have suggested that you create your own using Autodesk Inventor software for the design and 1 mm thick copper sheets as material for construction.

Design and construct a custom-shaped cookie cutter using equipment and materials available in the technology design lab. Use your knowledge of Inventor to create a solid model, orthographic drawing (with dimensions), and a sheet metal layout. The design can then be cut with the CNC laser, formed, and assembled using lead-free solder.

Design Constraints

- Four (40 minute) class periods: two for research, development, and design; one for cutting and forming; one for assembly.
- Material to be 1 mm thick copper.
- 4-5” diameter cutter.
- Must include an air hole on the top.
- Recommended height of cutter: 15 mm, handle width: 15 mm.

- Handle must be comfortable.

Resources

- Design Basics Exercises
- Pre-Mechanical Exercises
- Related web pages
- Examples of cookie cutters

Procedure

1. Define the design problem.
2. Conceptualize the design.
3. Design the cookie cutter in Autodesk® Inventor® software.
4. If required, assemble the cookie cutter in Autodesk Inventor.
5. Document the cookie cutter in Autodesk Inventor.
6. Create a portfolio of the design project.
7. Create a prototype model. (optional)

Suggested Outputs

Note: Your cookie cutter design may be a single part or an assembly. Adjust the following list of ouputs accordingly.

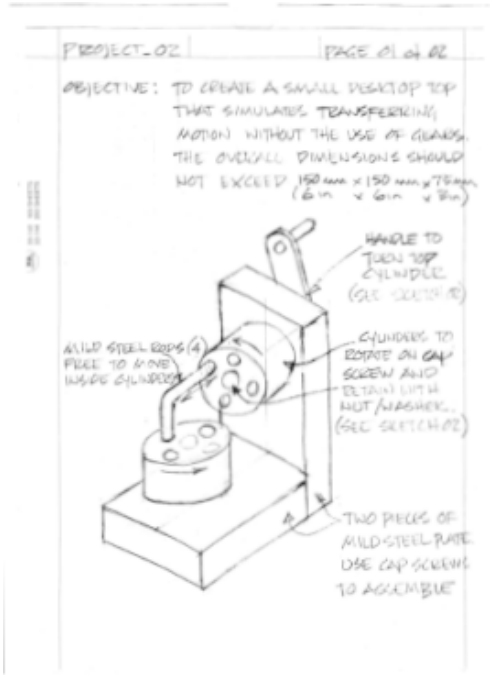
1. Sketches of your design idea.
2. Part and assembly files created in Autodesk Inventor.
3. A set of drawings suitable for manufacturing the assembly. Use an A or B-size sheet with suitable scales for all views. The first sheet should an isometric of the assembly. Add a new sheet for the each part of the assembly.
4. Annotate the part drawing. This includes:
   - Balloons and a parts list on the assembly drawing.
   - Center lines
   - Dimensions
   - A title block complete with iProperties.
5. The widths of the parts list columns are:
   - Description: 40 mm (2 inches)
   - Part Number: 30 mm (1.25 inches)
   - Material: 45 mm (2.25 inches)
6. A high-quality rendered image created in Inventor Studio.

Follow-Up Questions

1. What other products could be made using a similar technique?
2. If you changed the handle design, could you make uniquely shaped pancakes, sandwiches, and so forth?
3. What other materials could you use? What other forming processes are available?
4. Can you think of alternative assembly methods?

Project - Design a Desktop Toy

In this project, you design a desktop toy using Autodesk Inventor software.



Design Statement

Desk toys are a must-have for every office, from stress balls to desk toppers to geek gadgets. A company that designs and manufactures executive desktop toys has asked you to complete an existing design.

The design criteria are available along with some pencil and paper sketches. You complete the design evaluation and model the toy in Autodesk Inventor software.

Design Constraints

Review the requirements for the design of the desktop toy. This includes, determining overall dimensions, shape, material, and color.

Dimensions: The overall dimensions should not exceed 150 mm x 150 mm x 75 mm. This is to control the overall costs of the project and provide a suitable packaging and shipping solution.

Shape: The design criteria and sketches define what the shape of the toy will be. For example, the cylindrical rods move inside a circular barrel.

Material: The function of the toy means that the materials should be pleasing to the eye, as well as functional. This project is a good example of form and function having similar importance. For example, the rods can be made from polished stainless steel.

Color: The color of the parts depends on the steel selected. Research how steel can be colored using heat treatment.

Resources

- Design Basics Exercises
- Pre-Mechanical Exercises
- Related web pages
- Examples of desktop toys

Procedure

1. Open *project_02.pdf* and *project_02a.pdf*.
2. Review the design criteria in the sketches.
3. Model the parts in Autodesk Inventor.
4. Assemble the parts and review the design.
5. Use drive constraints to check the functionality of the assembly.
6. Document the assembly.

Suggested Outputs

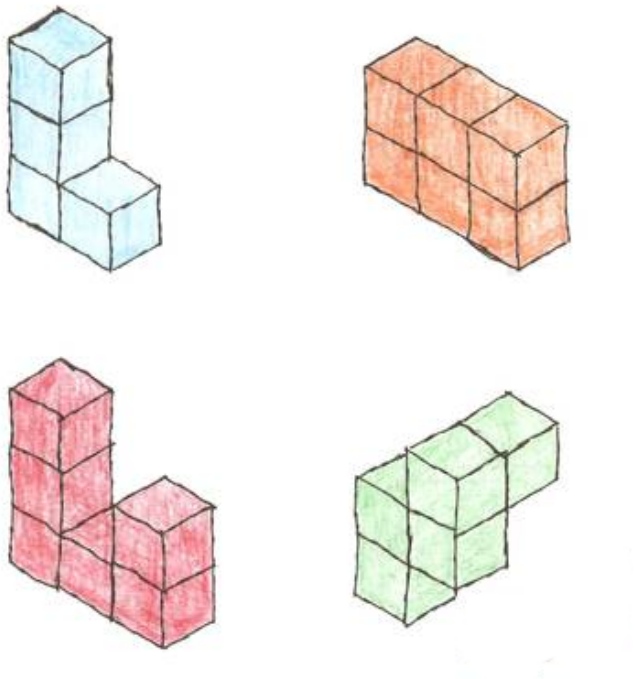
1. Sketches of your design idea.
2. Part and assembly files created in Autodesk Inventor.
3. A set of drawings suitable for manufacturing the assembly. Use an A or B-size sheet with suitable scales for all views. The first sheet should an isometric of the assembly. Add a new sheet for the each part of the assembly.
4. Annotate the assembly and part drawings. This includes:
   - Balloons and a parts list on the assembly drawing.
   - Center lines
   - Dimensions
   - A title block complete with iProperties.
5. The widths of the parts list columns are:
   - Description: 40 mm (2 inches)
   - Part Number: 30 mm (1.25 inches)
   - Material: 45 mm (2.25 inches)
6. A high-quality rendered image created in Inventor Studio.
7. An animation of the assembly created using drive constraints or Inventor Studio.

Project - Design a Puzzle Cube

The content in this project is derived from the course *Introduction to Engineering Design* developed by Project Lead the Way, Inc.

Students love to be challenged with puzzles. There are many kinds of puzzles on the market today. Are you ready to challenge someone's mind?

In this project, you design a puzzle cube using Autodesk Inventor software.



Possible puzzle cube solutions

Design Statement

The puzzle cube must meet the following design criteria:

- The puzzle must be a cube.
- The puzzle must be fabricated from twenty-seven 3/4” hardwood cubes.
- It must contain five parts.
- Each part must be made from three to six 3/4” cubes.
- Some parts should interlock.

Design Constraints

Skills Needed

Basic knowledge of sketching, creating parts, assembling parts, and creating drawings in Autodesk Inventor software.

Resources

- Design Basics Exercises
- Pre-Mechanical Exercises
- Related web pages
- Examples of puzzle cubes

Procedure

1. Define the design problem.
2. Conceptualize the design.
3. Design the cookie cutter in Autodesk® Inventor® software.
4. If required, assemble the cookie cutter in Autodesk Inventor.
5. Document the puzzle cube in Autodesk Inventor.
6. Create a portfolio of the design project.
7. Create a prototype model. (optional)

Suggested Outputs

1. Sketches of your design idea.
2. Part and assembly files created in Autodesk Inventor.
3. A set of drawings suitable for manufacturing the assembly. Use an A or B-size sheet with suitable scales for all views. The first sheet should an isometric of the assembly. Add a new sheet for the each part of the assembly.
4. Annotate the assembly and part drawings. This includes:
   - Balloons and a parts list on the assembly drawing.
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5. The widths of the parts list columns are:
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   - Material: 45 mm (2.25 inches)
6. A high-quality rendered image created in Inventor Studio.
7. An animation of the assembly created using drive constraints or Inventor Studio.

Going Beyond

Create an exploded isometric drawing based on an Autodesk Inventor presentation file.

Project - Design a Rack and Pinion Gear Assembly

The content in this project is derived from the course *Introduction to Engineering Design* developed by Project Lead the Way®, Inc.

In this project, you design a rack and pinion gear assembly using Autodesk® Inventor® software.

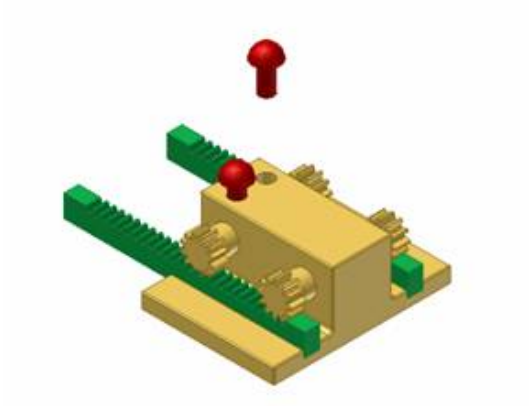
Design Statement

A customer was exceptionally pleased with the initial design of the heavy-duty rack drive assembly that you developed and animated with Autodesk Inventor drive constraints. This client was especially impressed by actuation of the assembly through one cycle when your management team presented it to them. As a result, the customer has added some additional requirements for further evaluation.

You will develop a new revision to the assembly by adding an additional driveshaft and related gear to the existing assembly.

Design Constraints

- This driveshaft/gear combination is to be inserted precisely between the two existing driveshaft/gear elements at the same height so that the gearing meshes with the rack detail teeth. Since the actuation of the original design that was submitted was a major selling point, you need to add the appropriate constraints and parameters so that the additional parts work in conjunction with the original gearing.
- Pegs must continue to work.



Possible solution

Skills Needed

Basic knowledge of sketching, creating parts, assembling parts, and creating drawings in Autodesk Inventor. The Sheet Metal application can also be used.

Resources

- Design Basics Exercises
- Pre-Mechanical Exercises
- Related web pages
- Examples of rack and pinion gear assemblies

Procedure

1. Define the design problem.
2. Conceptualize the design.
3. Design the rack and pinion gear assembly in Autodesk Inventor.
4. Create an animation of the assembly using Drive Constraints or Inventor Studio.
5. Document the assembly in Autodesk Inventor.
6. Create a portfolio of the design project.
7. Create a prototype model using a 3D printer or available materials. (optional)

Suggested Outputs

1. Part and assembly files created in Autodesk Inventor.
2. A set of drawings suitable for manufacturing the assembly. Use an A or B-size sheet with suitable scales for all views. The first sheet should an isometric of the assembly. Add a new sheet for the each part of the assembly.
3. Annotate the part drawings. This includes:
   - Center lines
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   - A title block complete with iProperties.
4. The widths of the parts list columns are:
   - Description: 40 mm (2 inches)
   - Part Number: 30 mm (1.25 inches)
   - Material: 45 mm (2.25 inches)
5. A high-quality rendered image created in Inventor Studio.
6. An animation of the assembly created using drive constraints or Inventor Studio.

Project - Design an Underwater Exploration Vehicle

The content in this project is derived from the course *Introduction to Engineering Design* developed by Project Lead the Way, Inc.

A jet-ski company has been hired to develop an underwater exploration vessel. Many people who are involved with water sports have been inquiring about this type of vehicle to explore reefs, sunken treasures, and shipwrecks.

In this project, you design an underwater exploration vehicle.

The Deep Flight Aviator from Hawkes Ocean Technologies

The Wet Flight from Hawkes Ocean Technologies

Design Statement

As the senior designer of this jet-ski company, you have been assigned to design an underwater exploration vessel.

Design Constraints

- Two person capacity.
- Maneuverability is paramount.
- A target retail price is $10,000.
- Must withstand depths up to 100 feet.
- On-board power plant is currently undetermined.

Suggested Outputs

1. Fully developed hand sketches of your design idea.
2. Documentation and drawings of at least three separate pieces.
3. Several presentation views (IPN) of the assembly.
4. Create a series of sequenced views of your final assembly that attempts to sell your product and create an AVI of your IPN file.

Skills Needed

Lofts, work planes, shell, and spline curves.

Resources

- Design Basics Exercises
- Pre-Mechanical Exercises
- Related web pages
- Examples of underwater exploration vehicles

Procedure

1. Define the design problem.
2. Conceptualize the design.
3. Design the vehicle in Autodesk Inventor.
4. Document the assembly in Autodesk Inventor.
5. Create a portfolio of the design project.
6. Create a prototype model using a 3D printer or available materials. (optional)

Suggested Outputs

1. Part and assembly files created in Autodesk Inventor.
2. A set of drawings suitable for manufacturing the assembly. Use an A or B-size sheet with suitable scales for all views. The first sheet should an isometric of the assembly. Add a new sheet for the each part of the assembly.
3. Annotate the assembly and part drawings. This includes:
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   - A title block complete with iProperties.
4. The widths of the parts list columns are:
   - Description: 40 mm (2 inches)
   - Part Number: 30 mm (1.25 inches)
   - Material: 45 mm (2.25 inches)
5. A high-quality rendered image created in Inventor Studio.
6. An animation of the assembly created using drive constraints or Inventor Studio.

Project - Design Handle Grips for a Mini-Scooter

A mini-scooter company has asked you to design handle grips for their new line of mini-scooters.

The major components of the scooter are designed and available to you as Autodesk® Inventor® parts. A partially complete assembly is also available.

In this project, you design handle grips for a mini-scooter using Autodesk® Inventor&® software.

In addition, you review the workflow required to create an animation in Autodesk® 3ds Max® Design software.

Design Statement

Review the requirements for the design of handles for the mini-scooter. This includes, determining overall dimensions, shape, material, and color.

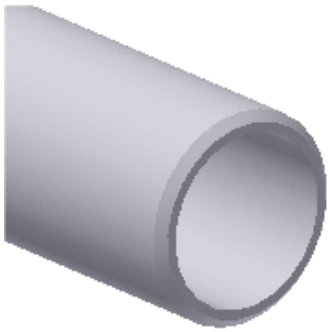
Dimensions

The handles for the mini-scooter must fit the existing handle bar.

Using Autodesk Inventor you can use a middle-out workflow to establish the dimensions of the handles. This workflow uses the existing part as the basis for the new design.

To do this, you open the scooter assembly, then create a new component. Using the Project Geometry tool, you can project the outside diameter of the handle bar and use this as the basis of your design.

A design consideration is what should the inside diameter of the handle be relative to the handle bar. Should it be the same size, smaller, or larger?



Shape

The shape of the handle can take on many forms. Do you want the shape to be a simple cylinder, slightly curved, or have a profile that matches the fingers on your hand?

Material

The material for the handle can be plastic or rubber. Each material has benefits over the other. For example, rubber may be easier to grip, just as on golf clubs. Plastic comes in many forms and colors.

Color

You should base the color you select for the handle on the current colors on the mini-scooter and the color of the wheels.

Resources

- Design Basics Exercises
- Pre-Mechanical Exercises
- Related web pages
- Examples of mini-scooters and handle grips

Procedure

1. Open *Scooter Assembly.iam*.
2. On the Component panel, click Create.
3. Sketch the profile for your handle grip design.
4. Create the handle grip feature.
5. Place assembly constraints between the handle grip and the scooter handle.
6. Place a second handle grip and apply assembly constraints between the handle grip and the scooter handle.
7. Create an animation of the assembly using drive constraints or Inventor Studio.
8. Document the assembly in Autodesk Inventor.
9. Create a portfolio of the design project.
10. Create a prototype model using a 3D printer or available materials. (optional)

Suggested Outputs

1. Part and assembly files created in Autodesk Inventor.
2. A set of drawings suitable for manufacturing the assembly. Use an A or B-size sheet with suitable scales for all views. The first sheet should an isometric of the assembly. Add a new sheet for the each part of the assembly.
3. Annotate the part drawings. This includes:
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   - Balloons and a parts list.
   - A title block complete with iProperties.
4. The widths of the parts list columns are:
   - Description: 40 mm (2 inches)
   - Part Number: 30 mm (1.25 inches)
   - Material: 45 mm (2.25 inches)
5. A high-quality rendered image created in Inventor Studio.

6. An animation of the assembly created using drive constraints or Inventor Studio.

Visualization Using 3ds Max Design

A mini-scooter company has asked you to create an animation of their latest product.

The major components of the scooter are designed and available to you as Autodesk® Inventor® parts.

Resources

- Quick Start for Autodesk® 3ds Max® Design
- Related web pages

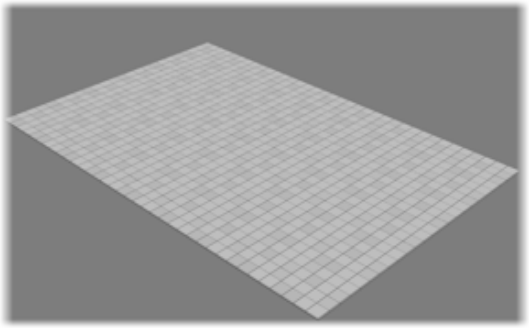
Video Demonstration

Before starting the exercise, you can refer to a video demonstration of the workflow. To view the demonstration, navigate to the Video Demonstration folder for this exercise. Double-click the HTML file to start the video.

You can pause and scroll through the video to search for help on how to complete a specific section.

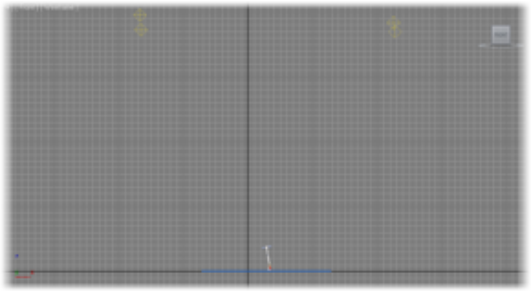
Create an Animation

1. Create a floor for the scooter. In this example, a tile material is used.

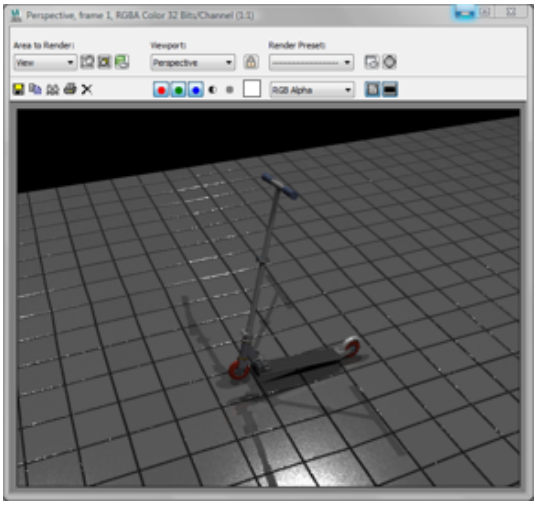


2. Import the Autodesk Inventor assembly.

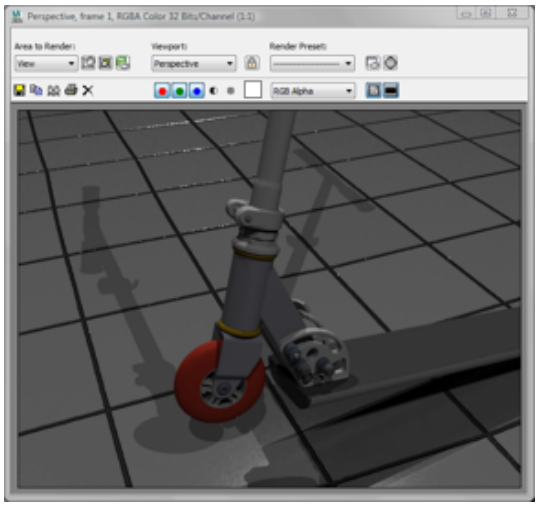
3. Add lights. In this example, four omni lights were added.



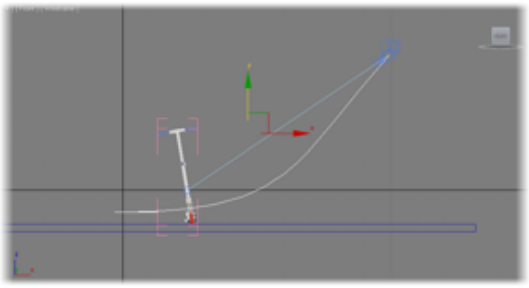
4. Render the scene to check the lighting.



5. Zoom into interesting details on the scooter and render the scene.



6. Add a camera and a path. You can then attach the camera to the path.



7. Render the animation. In this example, the animation has 300 frames.

Project - Design Playground Equipment

The content in this project is derived from the course *Introduction to Engineering Design* developed by Project Lead the Way, Inc.

Playground equipment must be designed and manufactured with several factors taken into consideration; the most important being safety, ease of construction, and age-appropriate sizing.

In this project, you design playground equipment using Autodesk Inventor software.

The EarlyWorks Cottage for 2 to 5 year olds. Image courtesy of Big Toys Inc.

Design Statement

You and your team will design and build a piece of playground equipment to be used by a 2-to-5-year-old child. Using your knowledge of the design process and model building, you construct a half-scale model and provide all the necessary documentation listed below. Extra credit may be earned by providing a fully-annotated assembly drawing of your final design using Autodesk Inventor.

Design Constraints

- Size of 2-to-5-year-old child (height, hand grip, width, and so on)
- Safety of equipment
- Originality
- Ease of construction

Skills Needed

Basic knowledge of sketching, creating parts, assembling parts, and creating drawings in Autodesk Inventor software.

Model Materials

Glue guns, clear adhesive tape, coffee stirrers, flat toothpicks, miscellaneous cardboard, and X-ACTO knives.

Please check with your teacher first before using other materials.

Resources

- Design Basics Exercises
- Pre-Mechanical Exercises
- Visit the website http://www.bigtoys.com for an example of environmentally friendly, commercial playground equipment.

Procedure

1. Define the design problem.
2. Conceptualize the design.
3. Design the playground in Autodesk Inventor.
4. Document the assembly in Autodesk Inventor.
5. Create a portfolio of the design project.
6. Create a prototype model using a 3D printer or available materials. (optional)

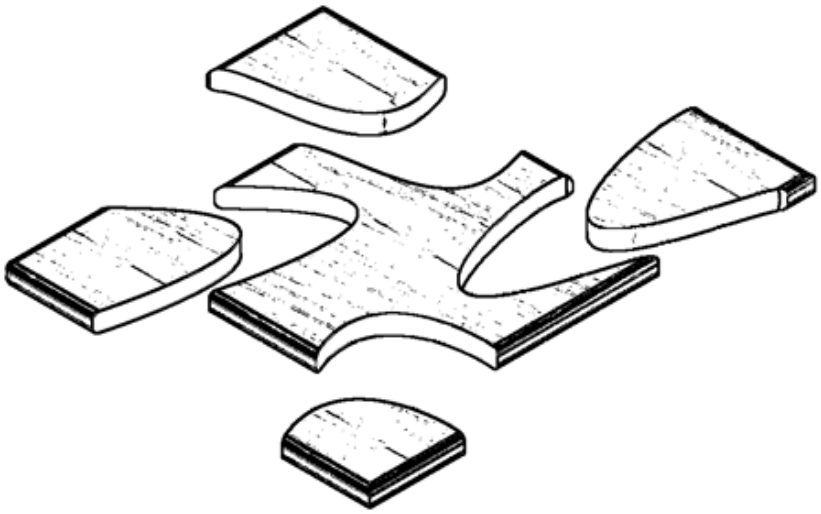
Suggested Outputs

1. Part and assembly files created in Autodesk Inventor.
2. A set of drawings suitable for manufacturing the assembly. Use an A or B-size sheet with suitable scales for all views. The first sheet should an isometric of the assembly. Add a new sheet for the each part of the assembly.
3. Annotate the assembly and part drawings. This includes:
   - Center lines
   - Dimensions
   - Balloons and a parts list.
   - A title block complete with iProperties.
4. The widths of the parts list columns are:
   - Description: 40 mm (2 inches)
   - Part Number: 30 mm (1.25 inches)
   - Material: 45 mm (2.25 inches)
5. A high-quality rendered image created in Inventor Studio.
6. An animation of the assembly created using drive constraints or Inventor Studio.

Project - Design Toy Blocks

The content in this project is derived from the course *Introduction to Engineering Design* developed by Project Lead the Way, Inc.

In this project, you develop a children's toy block system of interlocking or correlated parts using Autodesk Inventor software.



Design sketch of an interlocking jigsaw puzzle

Finished wooden jigsaw puzzle

Design Statement

A recent market survey has indicated that many younger children prefer handcrafted, wooden toys to high production, plastic and metal toys. Very young children love to play with wooden parts – sorting or building with them.

Design Constraints

- Must create a minimum of five parts.
- Must create a storage container for the parts to fit into.
- Parts must have the capability to interlock.
- Must be able to be used safely by a 2-to-5-year-old child.

Skills Needed

Basic knowledge of sketching, creating parts, assembling parts, and creating drawings in Autodesk Inventor software.

Resources

- Design Basics Exercises
- Pre-Mechanical Exercises
- Conduct an internet search on wooden toys

Procedure

1. Define the design problem.
2. Conceptualize the design.
3. Design the toy blocks in Autodesk Inventor.
4. Document the assembly in Autodesk Inventor.
5. Create a portfolio of the design project.
6. Create a prototype model using a 3D printer or available materials. (optional)

Suggested Outputs

1. Part and assembly files created in Autodesk Inventor.
2. A set of drawings suitable for manufacturing the assembly. Use an A or B-size sheet with suitable scales for all views. The first sheet should an isometric of the assembly. Add a new sheet for the each part of the assembly.
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