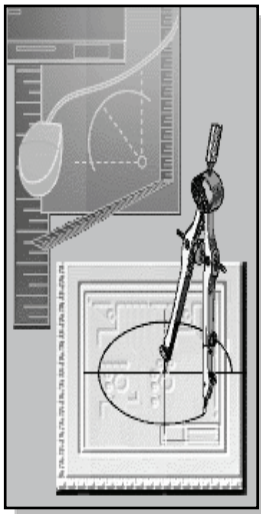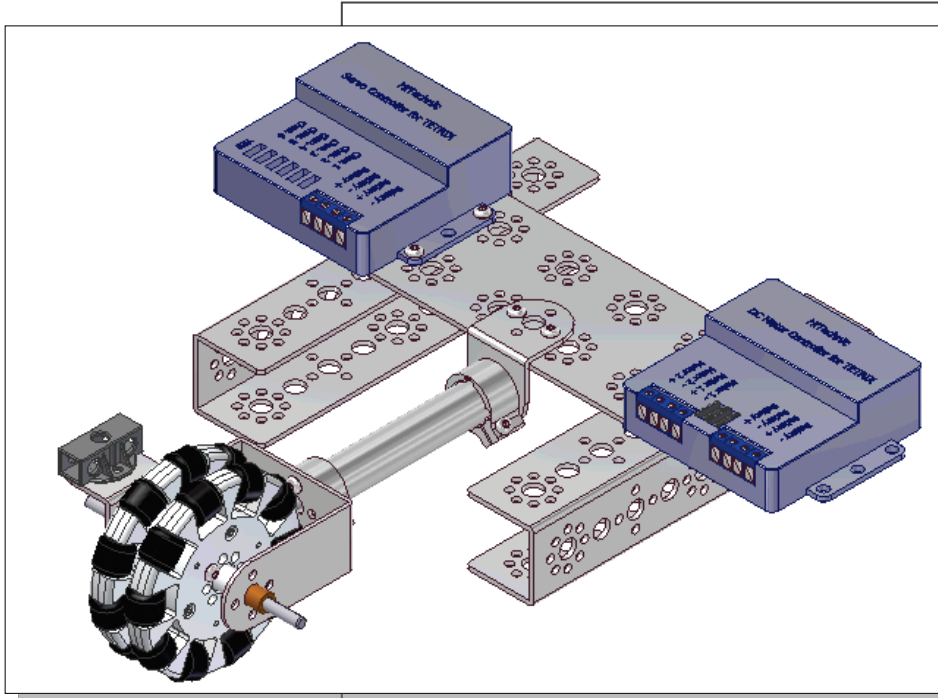# Chapter 19
Assembly Modeling with the TETRIX® by Pitsco Building System – Autodesk Inventor

Learning Objectives

- ♦ **Create and Use Subassemblies in Assemblies**
- ♦ **Create an Assembly Using Parts from the Tetrix® Set**
- ♦ **Understand and Perform Proper Constraints in Assembly**
- ♦ **Adjust the Orientation of Grounded Parts**
- ♦ **Adjust Components' Orientations**

Introduction

In this chapter, we will create an assembly with parts from the **LEGO® MINDSTORMS® NXT** and **TETRIX® by Pitsco** sets. The TETRIX® set contains many sturdy metal parts. Pitsco designed the TETRIX® system for the **FIRST Tech Challenge** (FTC), which is a robotic competition for high-school students. The TETRIX® kit is a prototyping platform that challenges the FTC teams with problems faced by real-world robotics designers. Made of heavy-duty aluminum, the structural pieces come in shapes that lend extra strength to the system. Robot builders can attach the LEGO® MINDSTORMS® NXT Set, sensors, and any other LEGO® *Technic* element to the TETRIX® system.

Through the years, many organizations have been established to provide additional assistance to the competitions. Some of these organizations were established through the help of the U.S. Education Grants; for example, the partnership between **Tufts Center for Engineering Educational Outreach** and **LEGO® MINDSTORMS® Education** which produces **LEGO® engineering (http://www.legoengineering.com)** was established through a grant from the **National Science Foundation**. Many states in the U.S. also have special programs to support the robotic competitions; in Oregon, there is the **Oregon Robotics and Outreach Program (http://www.ortop.org)**. Many helpful resources, such as workshops, training curricula, tutorials, etc., are also available through these organizations, many of which can be found at the official FTC website: **http://www.usfirst.org**,

In this chapter, we will create the *ST1* assembly model which uses parts from the TETRIX® and LEGO® MINDSTORMS® NXT sets. This design was created by members of **LEGO® engineering**. All of the required parts/subassemblies have been modeled in Autodesk Inventor, and can be found on the companion CD. Unzip all of the parts/subassemblies under a folder in the **Parametric-Modeling-Exercises** project folder before starting in the next section.

The ST1 Assembly
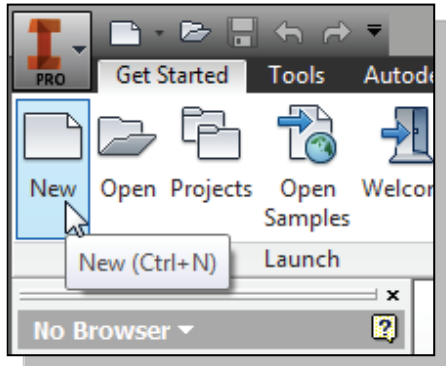(Design used with special permission from **LEGO® engineering**.)
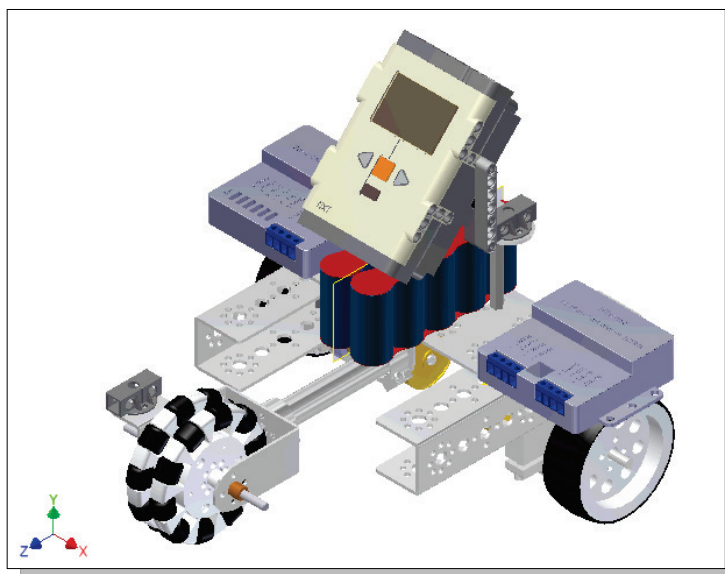
Modeling Strategy
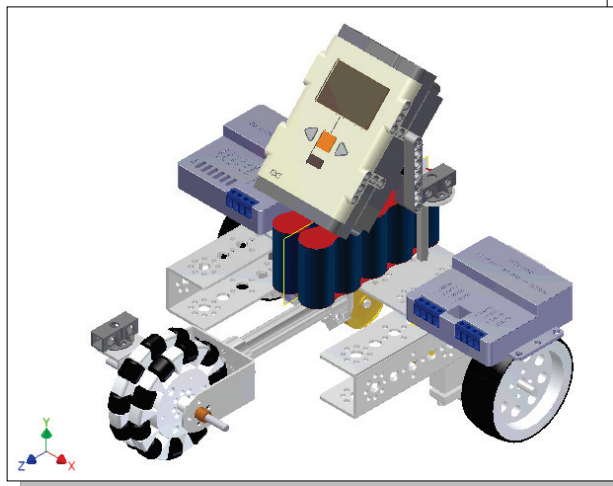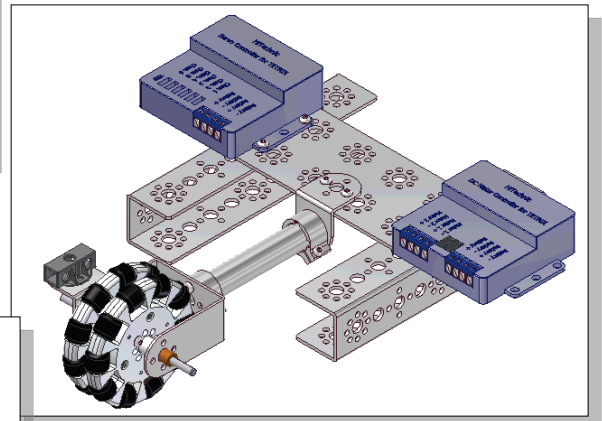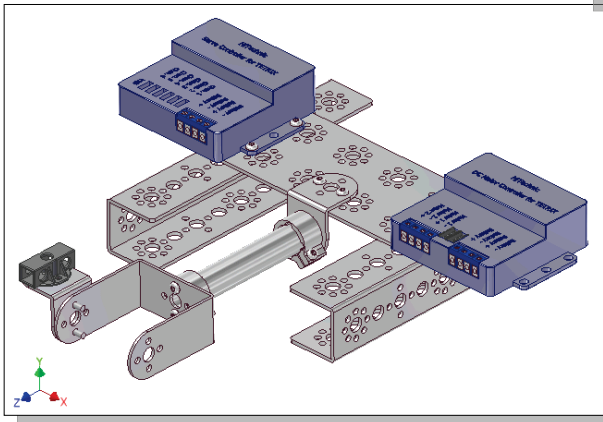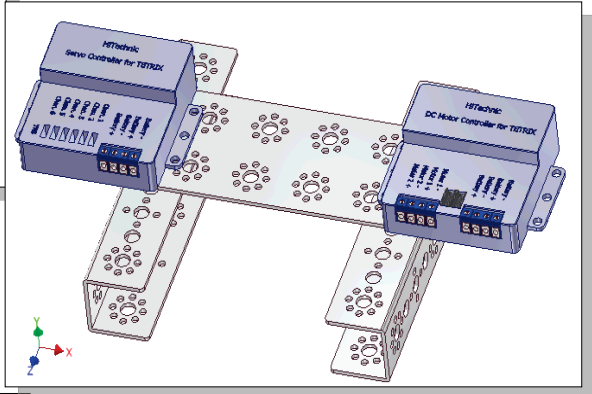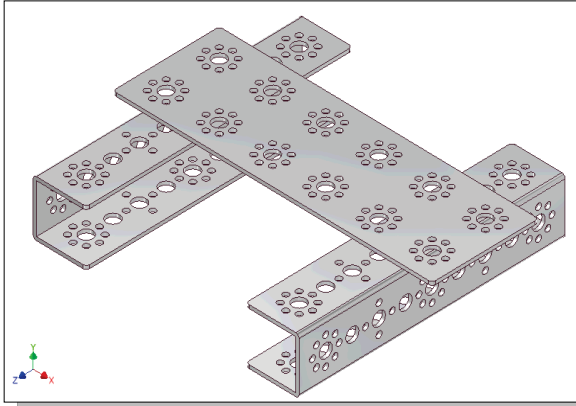
Starting Autodesk Inventor

1. Select the **Autodesk Inventor** option on the *Start* menu or select the **Autodesk Inventor** icon on the desktop to start Autodesk Inventor. The Autodesk Inventor main window will appear on the screen.
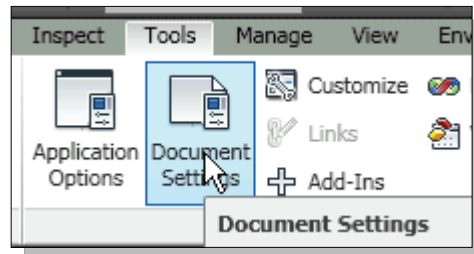
2. Select the **New File** icon with a single click of the left-mouse-button in the *Launch* toolbar as shown.

3. Select the **English** tab and in the *Template* list select **Standard(in).iam** (*Standard Inventor Assembly Model* template file).
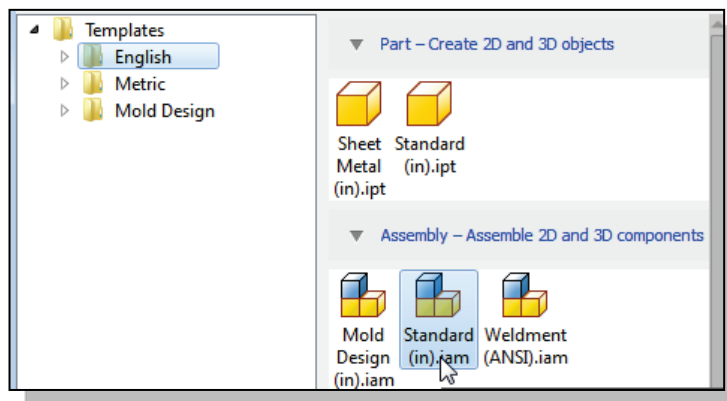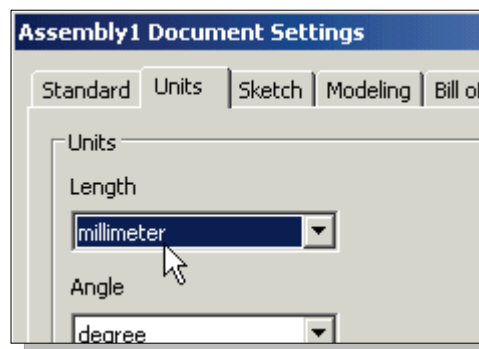
4. Click on the **OK** button in the *New File* dialog box to accept the selected settings.

5. In the *Ribbon* toolbar, select the **Tools** tab.

6. Select the **Document Settings** command as shown.

7. In the *Assembly1 Document Settings* dialog box, click on the **Units** tab as shown.

8. Set the *Length unit* to **millimeter** as shown.

9. Click on the **OK** button to accept the selected settings

- The *Standard Inventor Assembly Model* template file contains predefined settings including default 3D view orientation and default 2D drawing views setup. With Autodesk Inventor, we are allowed to mix units within the same file.

Creating a Subassembly

❖ We are now ready to assemble the components together. We will start by assembling a *DC Motor*, a *Motor Mount*, a *Hub* and a *Wheel* into a *Motor-Wheel* **subassembly**. A subassembly is a unit assembled separately but designed to fit with other units in a manufactured product. Subassemblies are commonly used to help manage complex designs and when the same configurations are used multiple times in a design. Note that a subassembly behaves just the same as a part within an assembly; i.e., a subassembly possesses only six degrees of freedom, just like a regular part. Any moving parts within a subassembly become fully locked inside the main assembly.
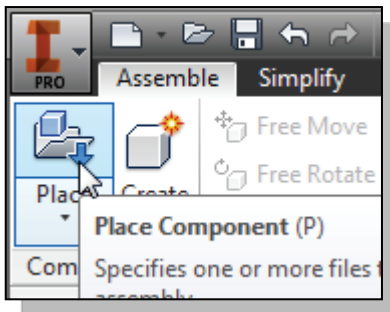
1. Switch to the *Assembly* panel, and select the **Place Component** command by left-mouse-clicking the icon.

2. Select *Motor Mount Pack.ipt* (switch to the appropriate folder if necessary) in the list window.
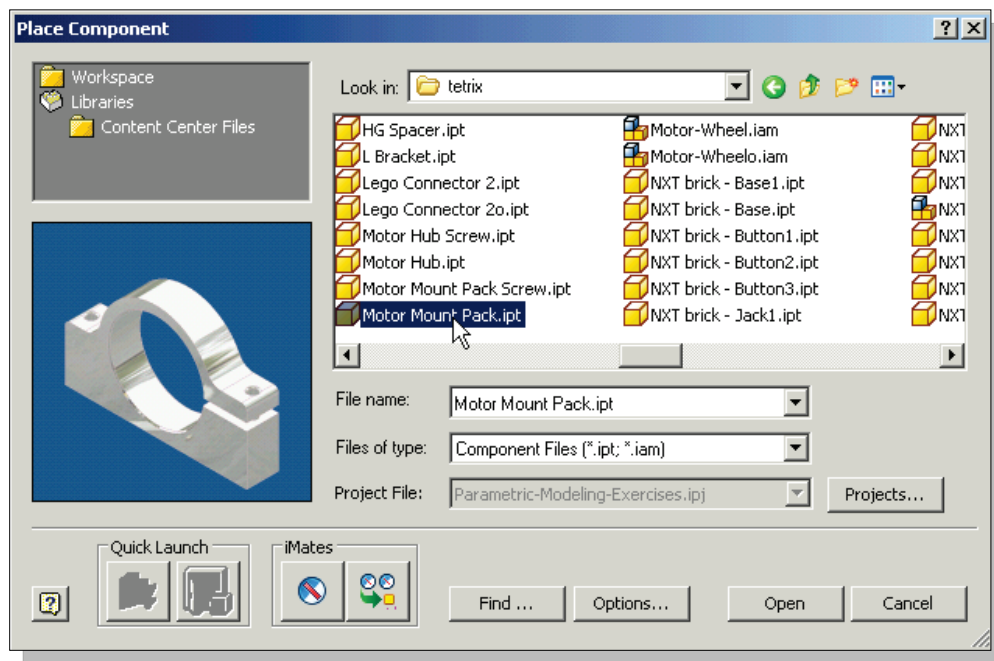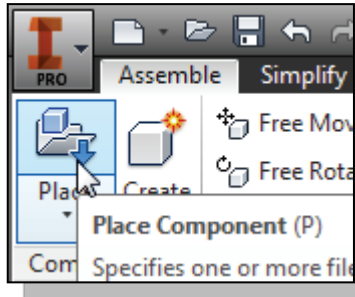
3. Click on the **Open** button to retrieve the model.
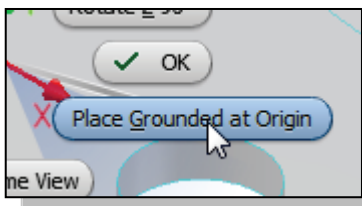
4. Right-mouse-click once to bring up the option menu and select **Place Grounded at Origin** to place the part.

5. Right-mouse-click once to bring up the option menu and select OK to end the command.

Placing the Next Component

➢ We will retrieve the **DC Gearhead Motor** part as the next component of the assembly model.

1. In the *Assembly* panel, select the **Place Component** command by left-mouse-clicking the icon.

2. Select the **DC Gearhead Motor** design in the list window. Click on the **Open** button to retrieve the model.

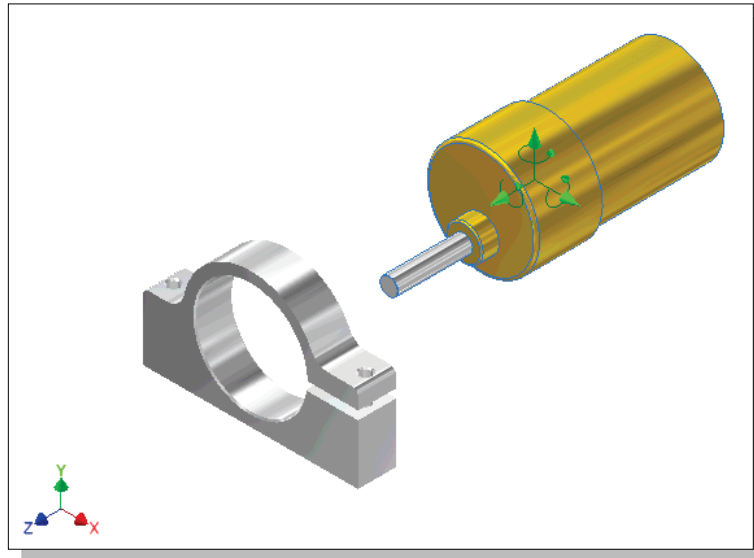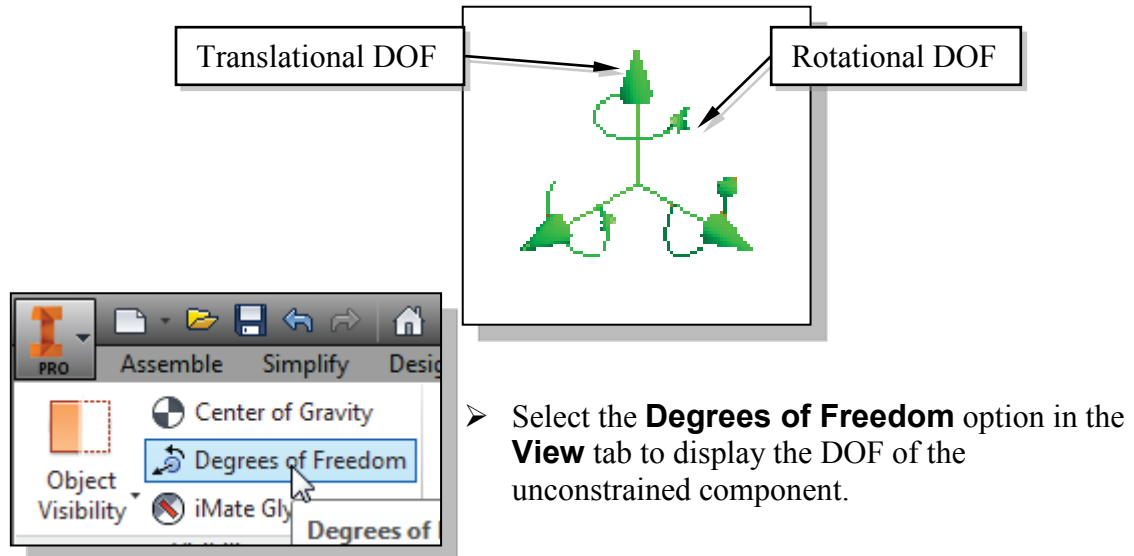3. Place **one** copy of the *DC Motor* part toward the right side of the graphics window, as shown in the figure.
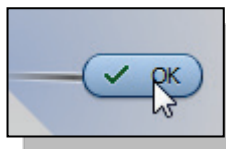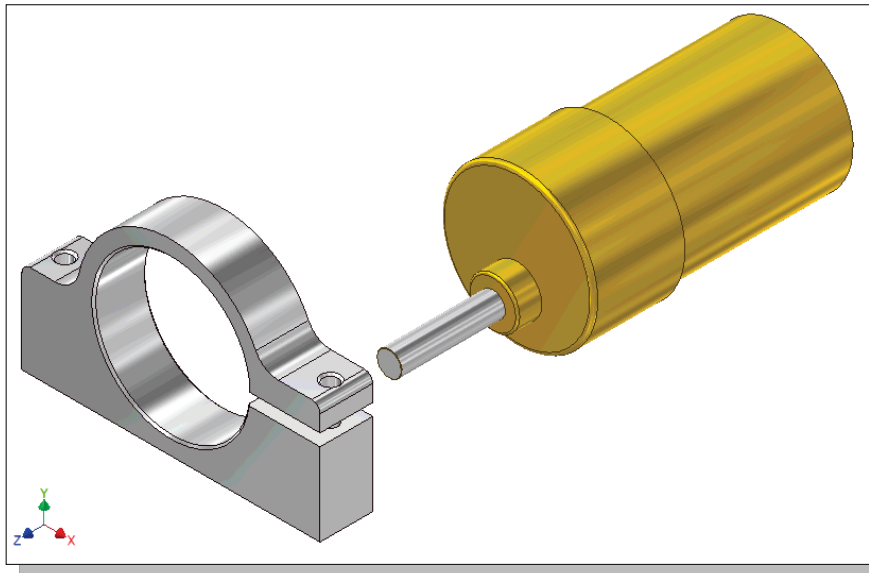
4. Inside the graphics window, right-mouse-click once to bring up the option menu and select **OK** to end the placement of the *DC Motor* part.
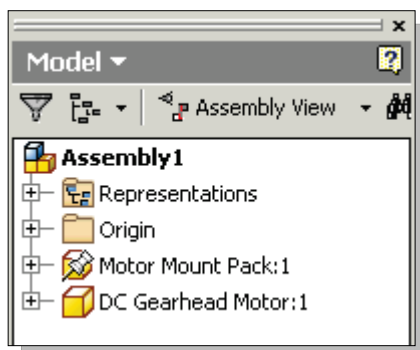
● Inside the browser window, the retrieved parts are listed in their corresponding order. The **pin** icon in front of the **Motor Mount** signifies the component is grounded and all *six degrees of freedom* are restricted. The number behind the name is used to identify the number of copies of the same component in the assembly model.

Degrees of Freedom Display

- Each component in an assembly has six **degrees of freedom** (**DOF**), or ways in which rigid 3D bodies can move: movement along the X-, Y-, and Z-axes (translational freedom), plus rotation around the X-, Y-, and Z-axes (rotational freedom). *Translational DOF*s allow the part to move in the direction of the specified vector. *Rotational DOF*s allow the part to turn about the specified axis.

Translational DOF   Rotational DOF

> Select the **Degrees of Freedom** option in the **View** tab to display the DOF of the unconstrained component.

- In *parametric modeling*, the degrees of freedom symbol shows the remaining degrees of freedom (both translational and rotational) for the components of the active assembly. Note that each component has its own degrees of freedom, even if the same part is placed multiple times within the same assembly. The assembly constraints are also applied independently to each component. The set of degrees of freedom symbols, as shown in the figure above, signifies each copy of the *DC Motor* is unconstrained and should be constrained independently.

Apply Assembly Constraints
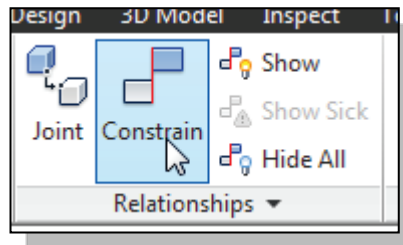
1. In the *Assembly* panel, select the **Constrain** command by left-mouse-clicking once on the icon.

2. In the *Place Constraint* dialog box, confirm the constraint *Type* is set to the **Mate** constraint.
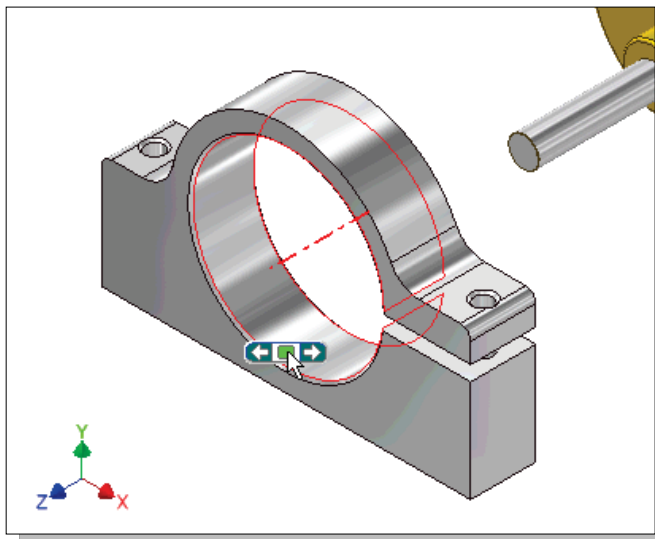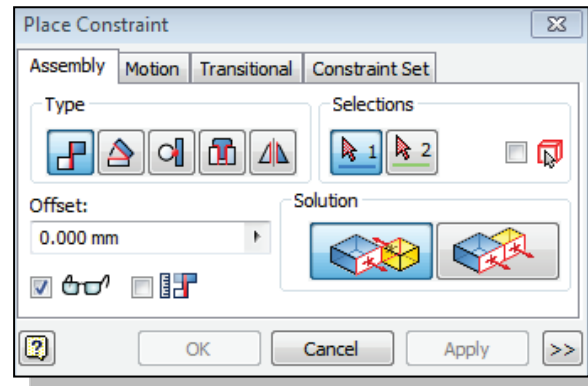
3. Select the inside cylindrical surface of the *Motor Mount*, as the first item for the Mate Alignment command.

4. Click on the outside cylindrical surface of the large cylinder of the *DC Motor* part as shown.

5. Click on the **Apply** button to accept the selection and apply the Mate constraint.
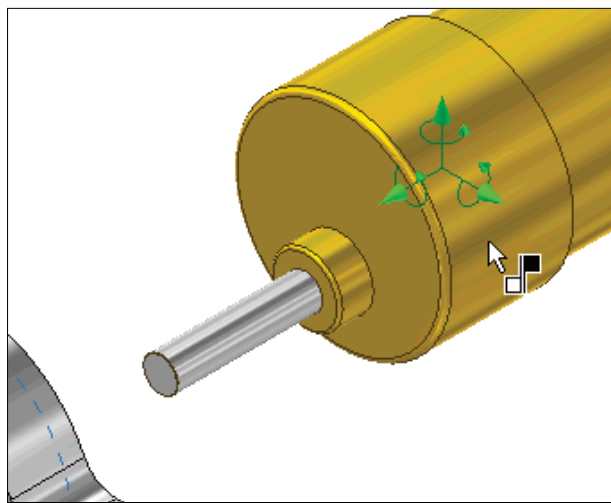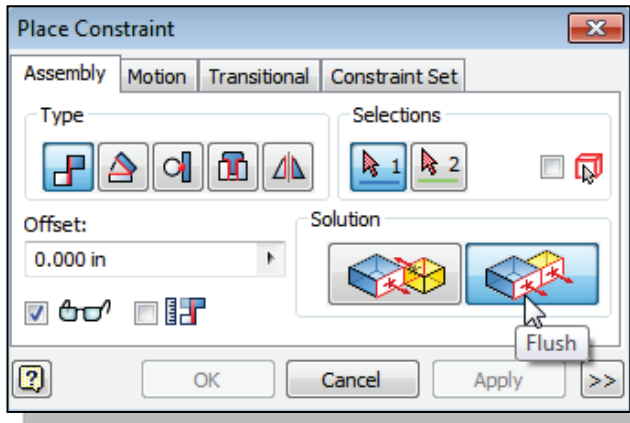
6. In the *Place Constraint* dialog box, set the constraint *Type* to **Flush**.

7. Select the **back side vertical surface** of the base component, the *Motor Mount* part, as the first item for the Mate Constrain command.
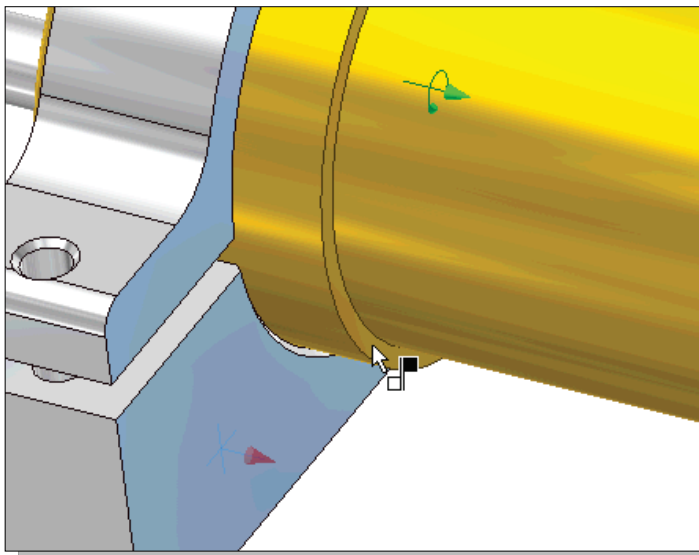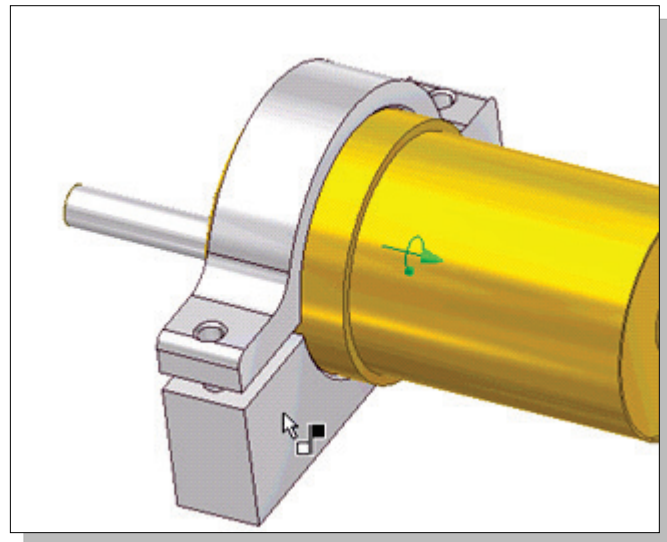
8. Click on one of the **ring surfaces** of the *DC Motor* part as the second item to apply the constraint.

9. Click on the **Apply** button to accept the selection and apply the Mate constraint.

- Note the small arrow symbol showing one degree of freedom available; the *DC Motor* can still rotate about the center axis. We will apply an additional constraint to lock the parts.
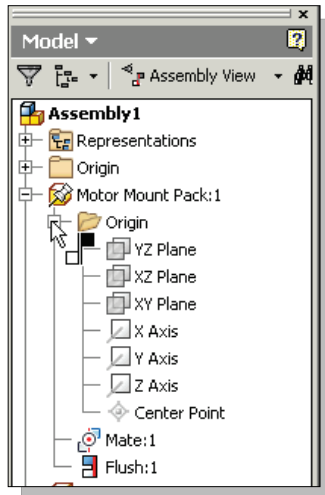
10. In the *Model* history tree, click on the [**+**] icon in front of the **Motor Mount Pack** item to show the features list of the part.

11. Click on the [**+**] icon in front of the **Origin** item of the *Motor Mount Pack* to show the features list.

12. In the *Model* history tree, select the YZ Plane of the *Motor Mount Pack* as the first item for the Mate constraint command.
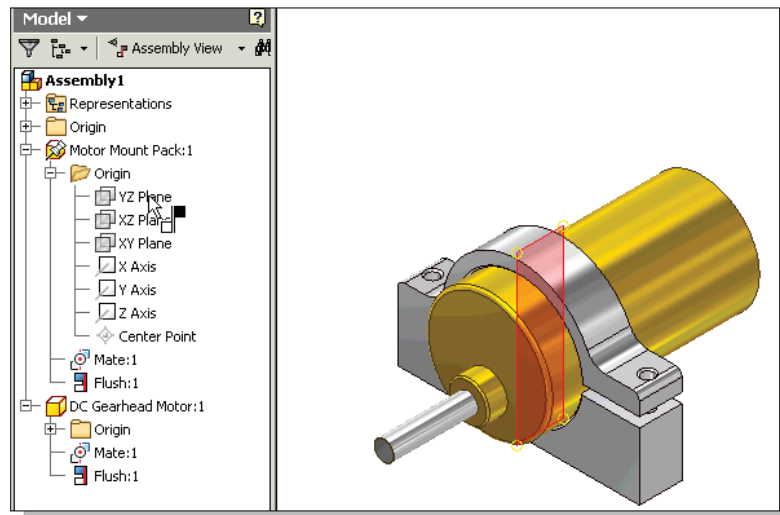
13. In the *Model* history tree, select the YZ Plane of the *DC Motor* as the second item for the Mate constraint command.
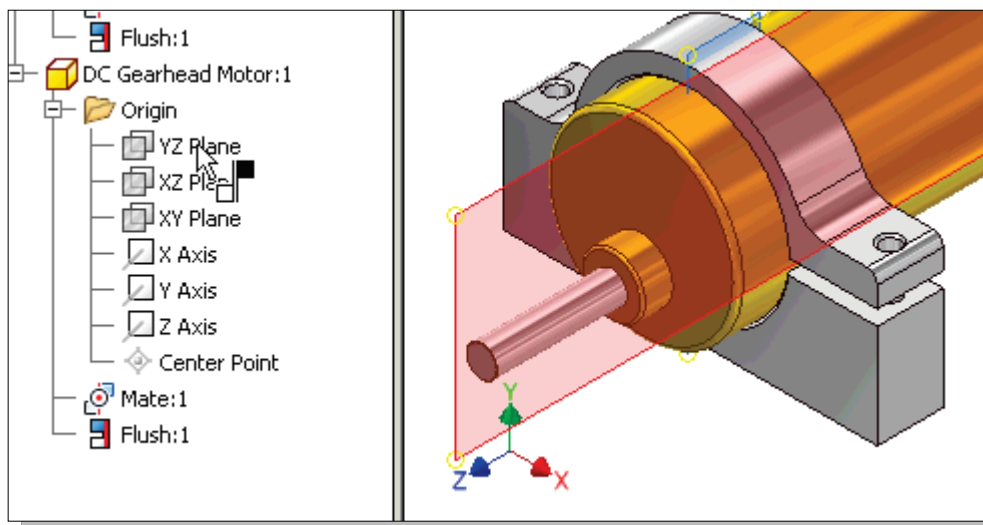
14. Click on the **Apply** button to accept the selection and apply the constraint.

15. Click on the **Cancel** button to exit the **Constrain** command.
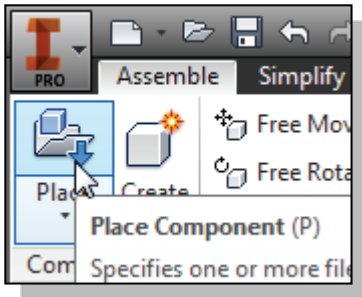
16. In the *Assembly* panel, select the **Place Component** command by left-clicking the icon.
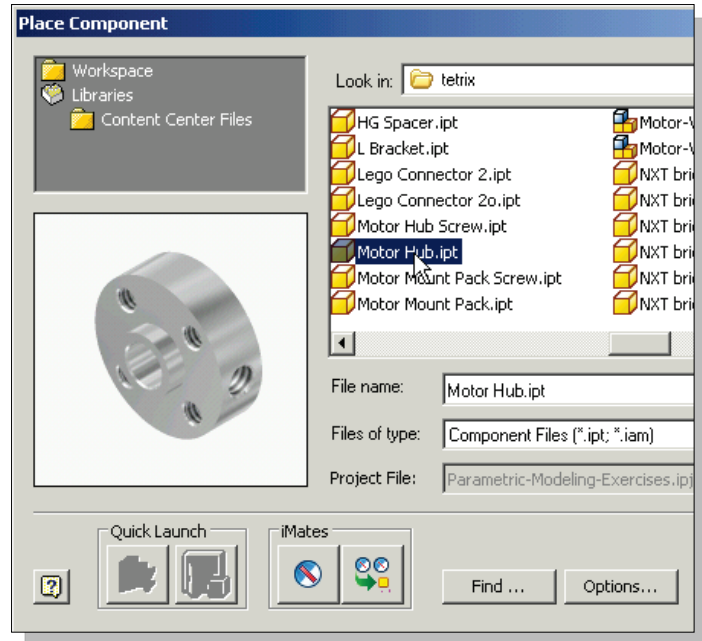
17. Select the ***Motor Hub*** design in the list window. Click on the **Open** button to retrieve the model.
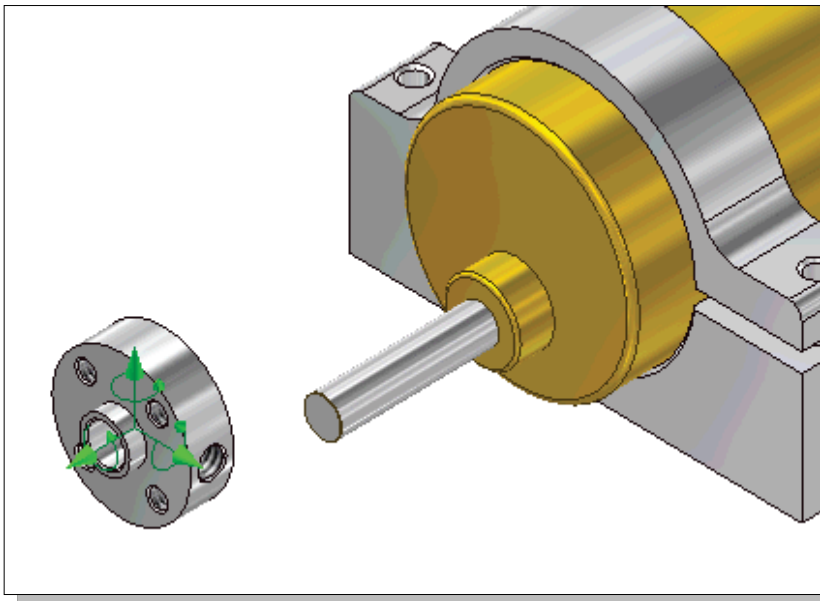
18. Place **one** copy of the ***Motor Hub*** toward the left side of the graphics window, as shown in the figure.

19. Inside the graphics window, right-mouse-click once to bring up the option menu and select **Done** to end the command.

20. In the *Assembly* panel, select the **Constrain** command by left-mouse-clicking once on the icon.

Place Constraint

Assembly | Motion | Transitional | Constraint Set

Type — Selections

Offset:
0.000 mm

Solution

OK   Cancel   Apply   >>
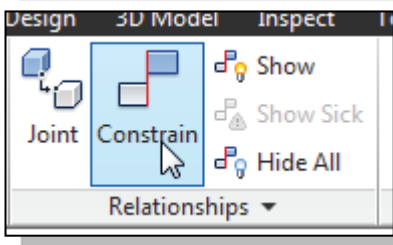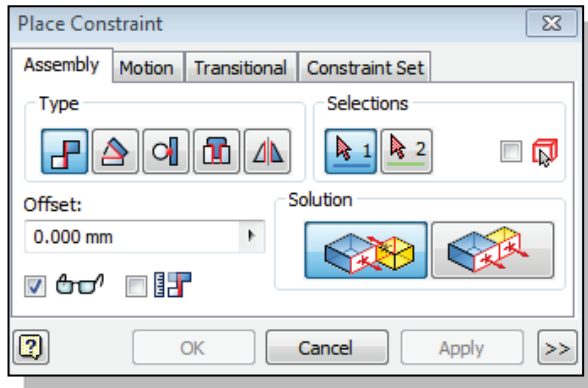
21. In the *Place Constraint* dialog box, confirm the constraint *Type* is set to the **Mate** constraint.

22. Select the ***Motor Shaft*** of the *DC Motor* part as the first item for the Mate Alignment command.
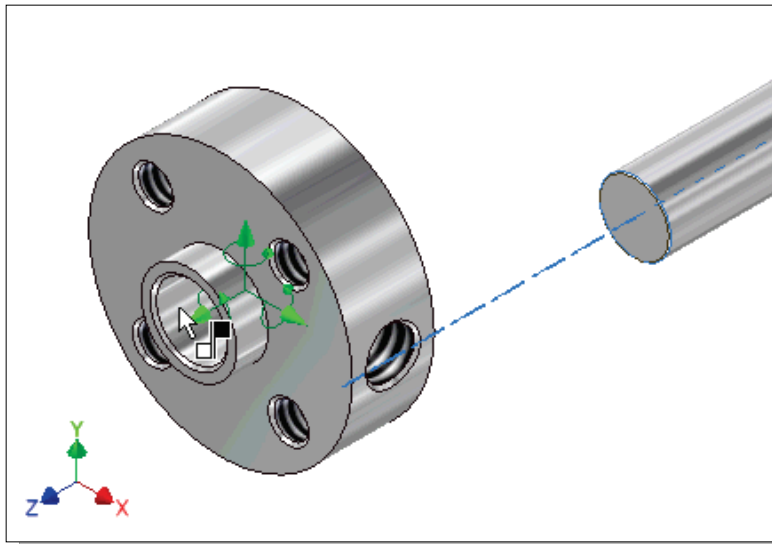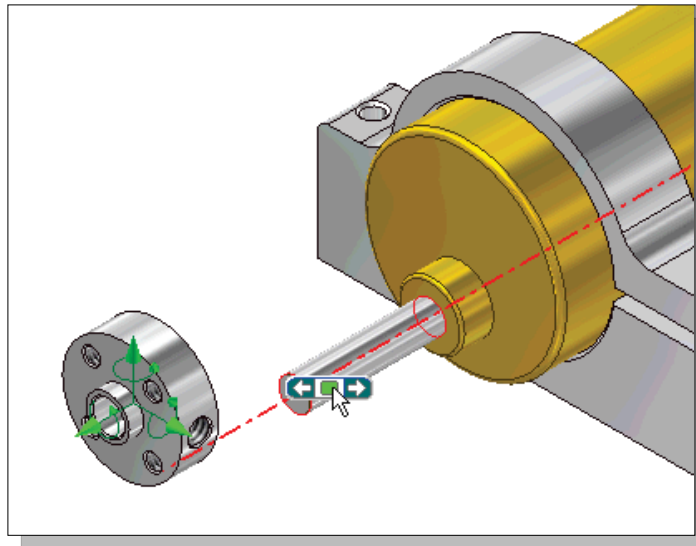
23. Click on the **inside cylindrical surface** of the *Motor Hub* part as the second part selection to align the two center axes.

Apply

24. Click on the **Apply** button to accept the selection and apply the Mate constraint.

25. On your own, repeat the above steps and align the *Motor Hub* part to push against the *DC Motor* as shown.
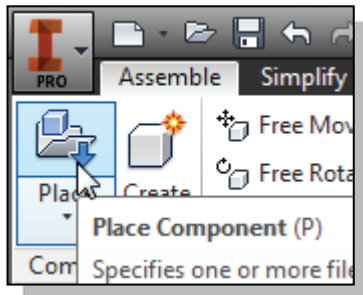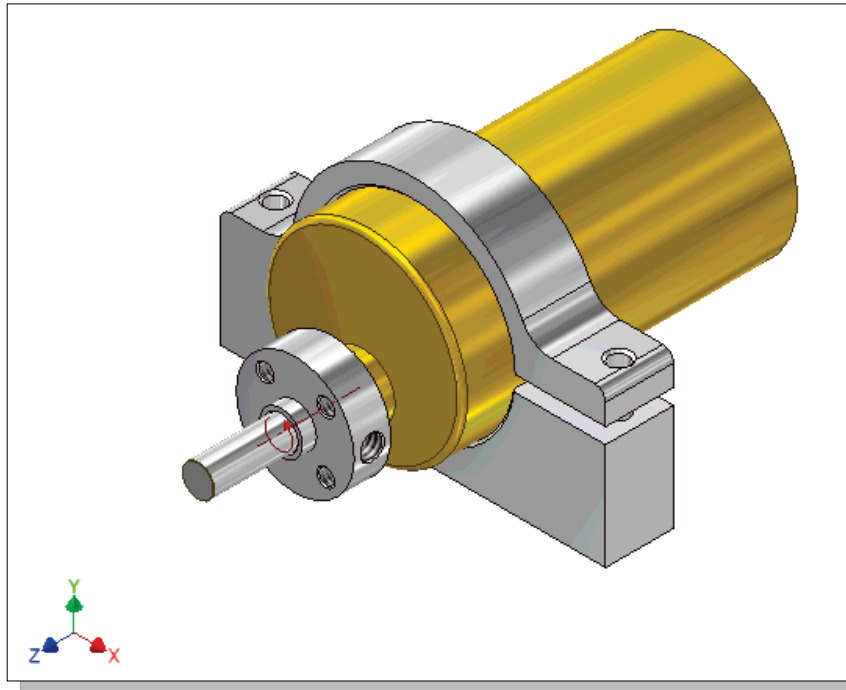
26. In the *Assembly* panel, select the **Place Component** command by left-mouse-clicking the icon.

27. Select the *Motor Hub Screw* design in the list window. Click on the **Open** button to retrieve the model.

28. Place a copy of the *Motor Hub Screw* on the screen.

29. Inside the graphics window, right-mouse-click once to bring up the option menu and select **Done** to end the command.
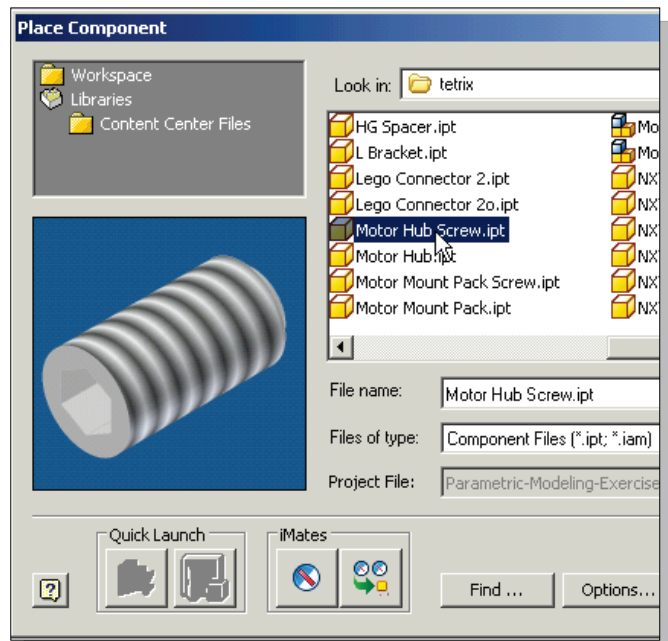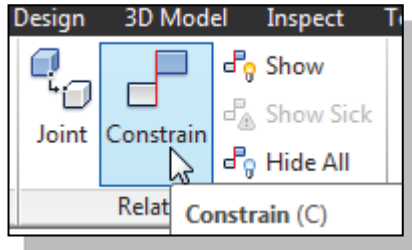
30. In the *Assembly* panel*,* select the **Constrain** command by left-mouse-clicking once on the icon.

31. On your own, align the **center axis** of the *Hub Screw* to the **center axis** of the **hole feature** of the *Motor Hub* part as shown.
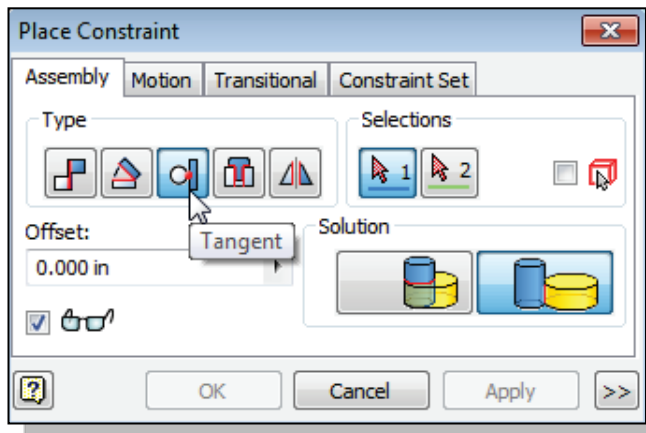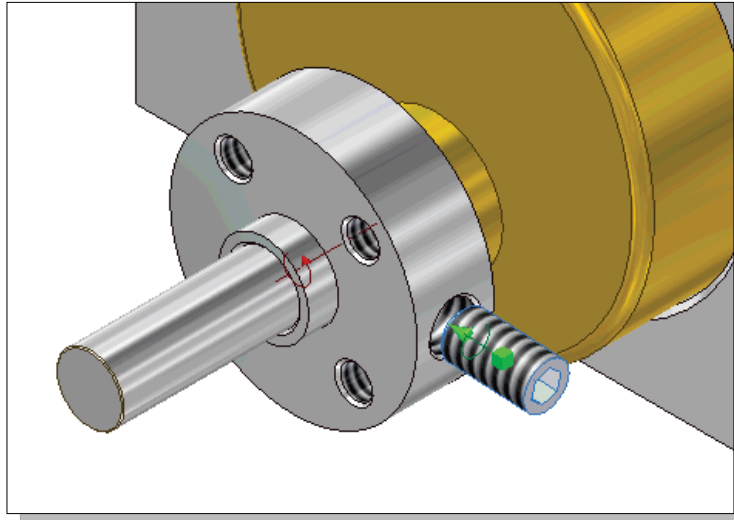
32. In the *Place Constraint* dialog box, set the constraint *Type* to **Tangent** constraint.

33. Select the **Motor Shaft** of the *DC Motor* part as the first item for the Tangent Alignment command.
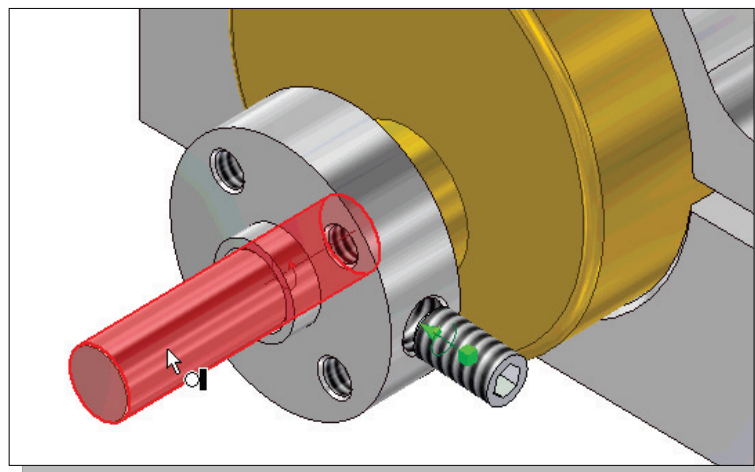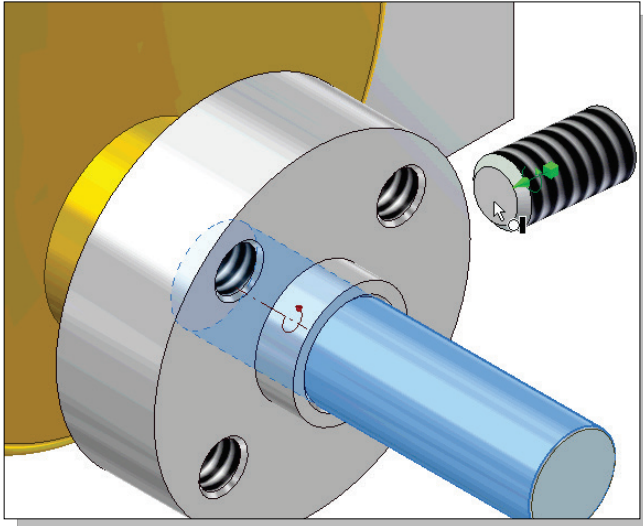
34. On your own, use the **ViewCube** and rotate the display as shown.

35. Click on the **flat end surface** of the *Hub Screw* part as the second part selection to align the two center axes.

36. Click on the **Apply** button to accept the selection and apply the Tangent constraint.

37. Click on the **Cancel** button to exit the **Constrain** command.
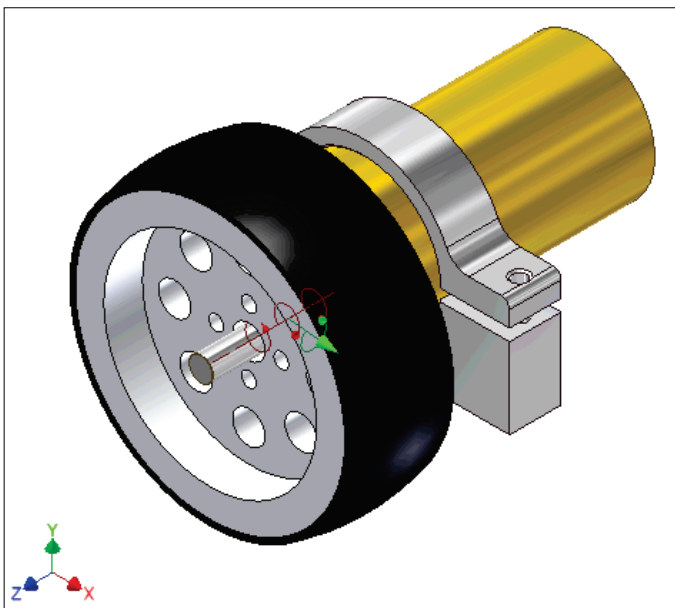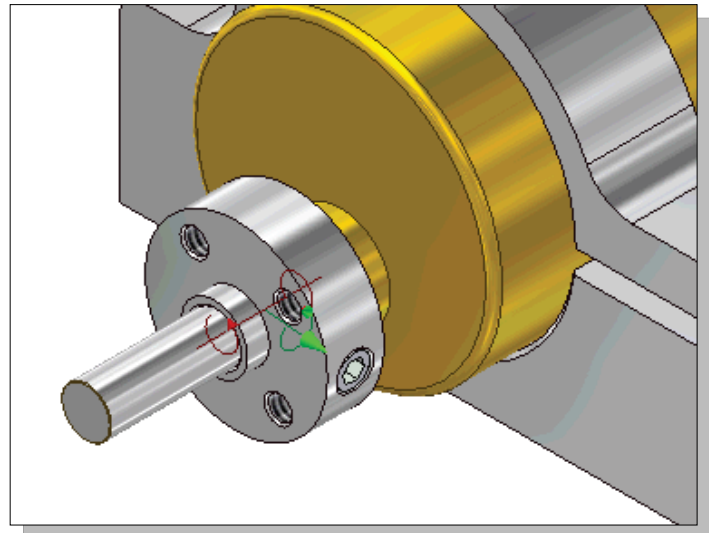
38. On your own, assemble a copy of the *Wheel.iam* subassembly as shown.
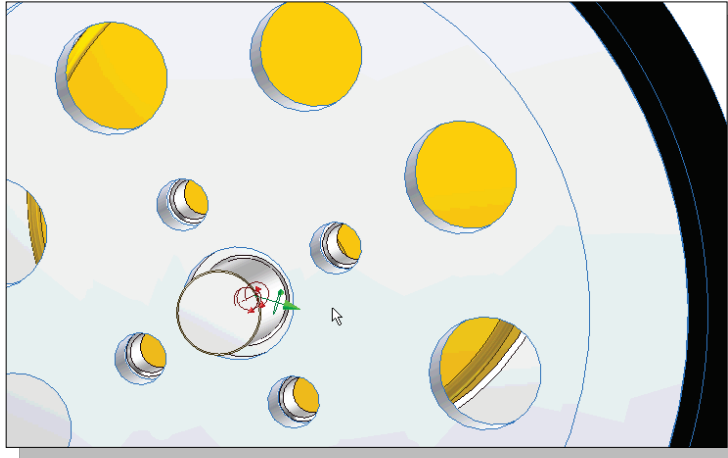
39. Align the holes of the *Wheel* to the *Hub* by adjusting the *Wheel* position, using *drag and drop*, to as shown.

40. On your own, apply another **Mate** constraint to align the center axes of one of the small holes of the *Wheel* to the *Motor Hub* as shown.
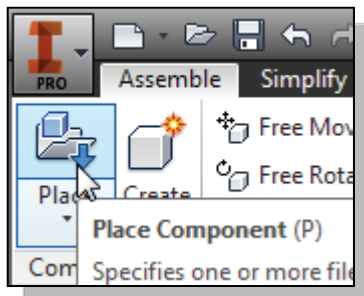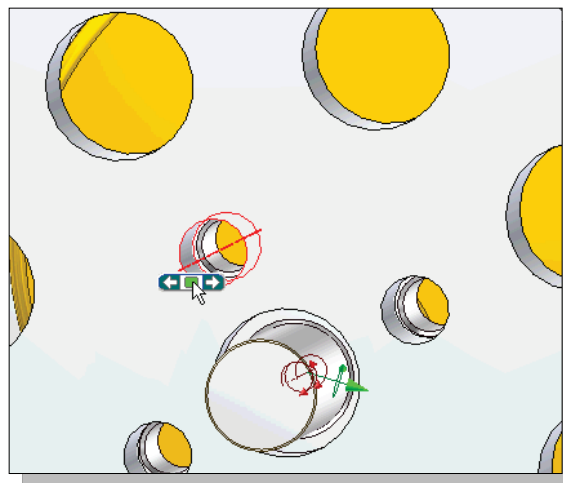
41. In the *Assembly* panel, select the **Place Component** command by left-clicking the icon.

42. On your own, place two copies of the *6-32x.3125 in SHC Screw* part and two copies of the *Motor Mount Pack Screw* part on the screen.
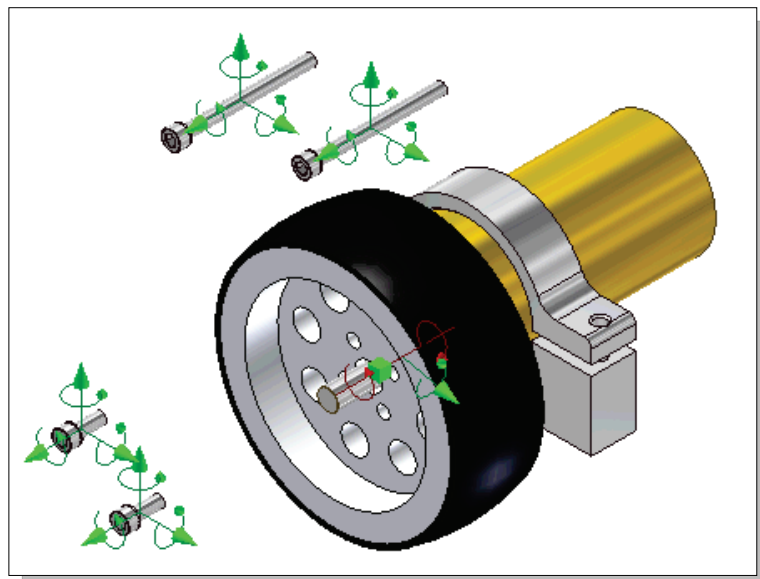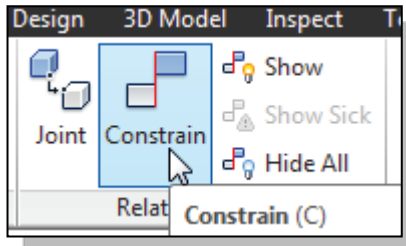
43. In the *Assembly* panel, select the **Constrain** command by left-mouse-clicking once on the icon.

44. On your own, assemble the *6-32x.3125 in SHC Screw* parts as shown.
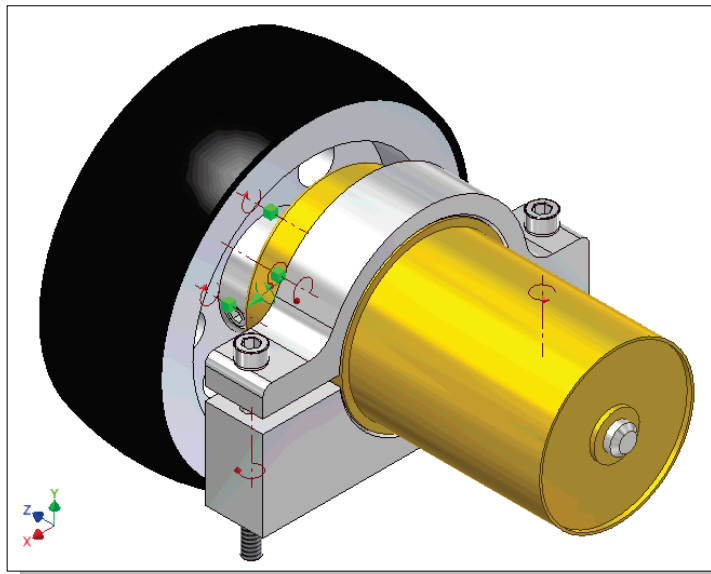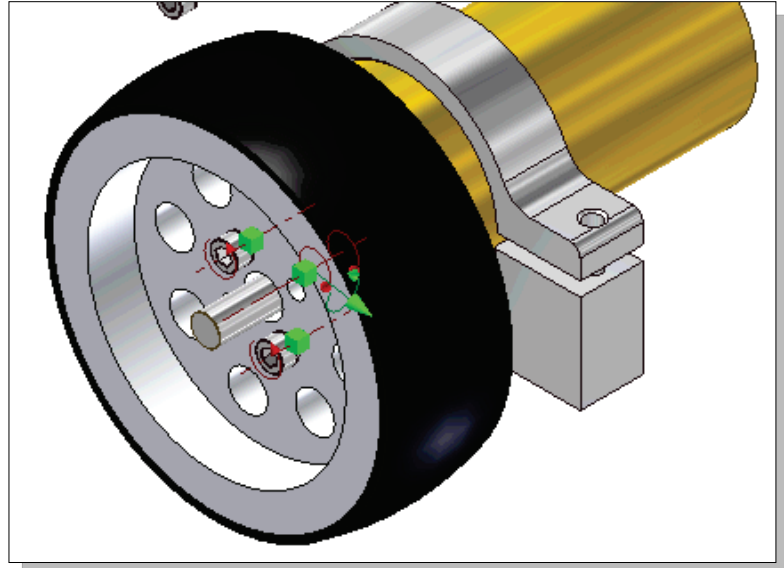
45. On your own, assemble the *Motor Mount Pack Screw* parts as shown.
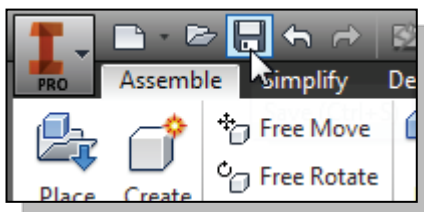
46. Click on the **Save** icon and save the completed subassembly as *Motor-Wheel.iam*.

Starting the Main Assembly

1. Select the **New File** icon with a single click of the left-mouse-button in the *Quick Access* toolbar as shown.

2. Select the **English** units set and in the *Template* list; select **Standard(in).iam** (*Standard Inventor Assembly Model* template file).
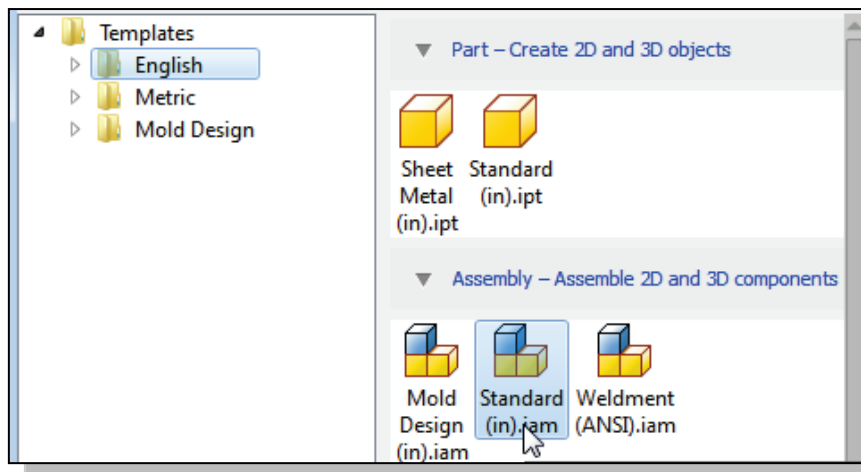
3. Click on the **OK** button in the *New File* dialog box to accept the selected settings.
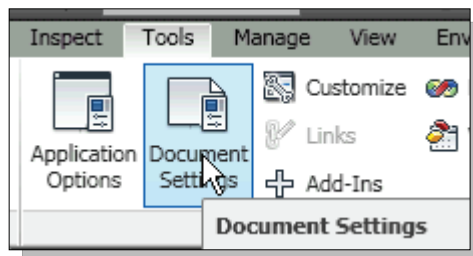
4. In the *Ribbon* toolbar, select the **Tools** tab.

5. Select the **Document Settings** command as shown.

6. In the *Assembly2 Document Settings* dialog box, click on the **Units** tab as shown.

7. Set the *Length Units* to **centimeter** as shown.

8. Click on the **OK** button to accept the selected settings.
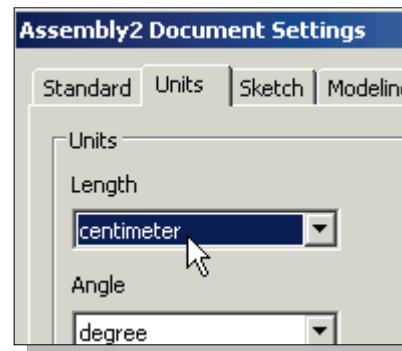
•   The *Standard Inventor Assembly Model* template file contains predefined settings including default 3D view orientation and default 2D drawing views setup. With Autodesk Inventor, we are allowed to mix different Units within the same file.
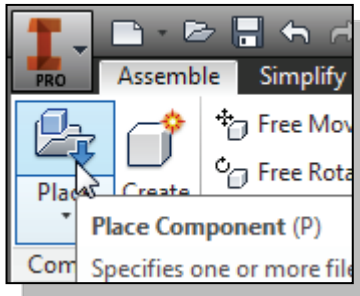
9. In the *Assembly* panel, select the **Place Component** command by left-mouse-clicking once on the icon.

10. Select the ***64mmx192mm plate*** part in the list window. Click on the **Open** button to retrieve the model.
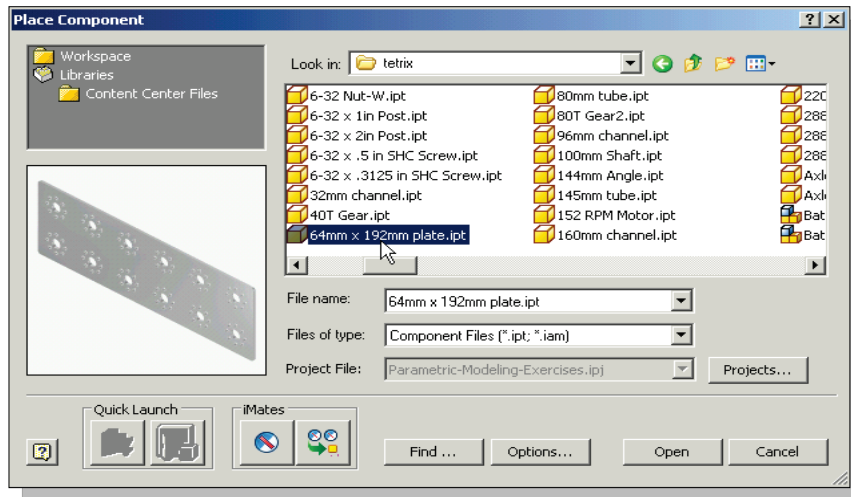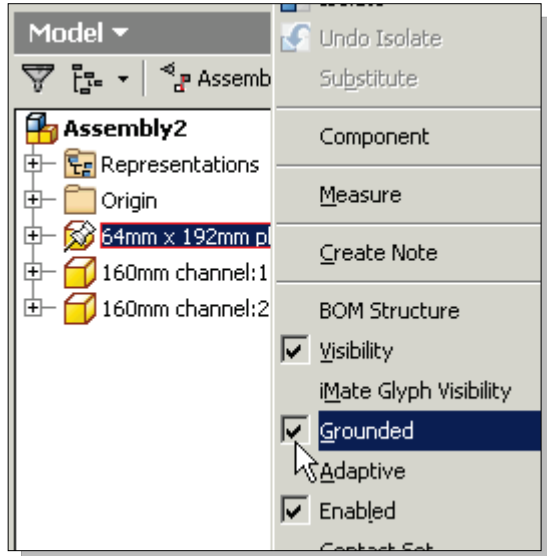
11. In the graphics area, place a copy of the part in the graphics window.

12. On your own, place two copies of the ***160mm channel*** part in the graphics area as shown.
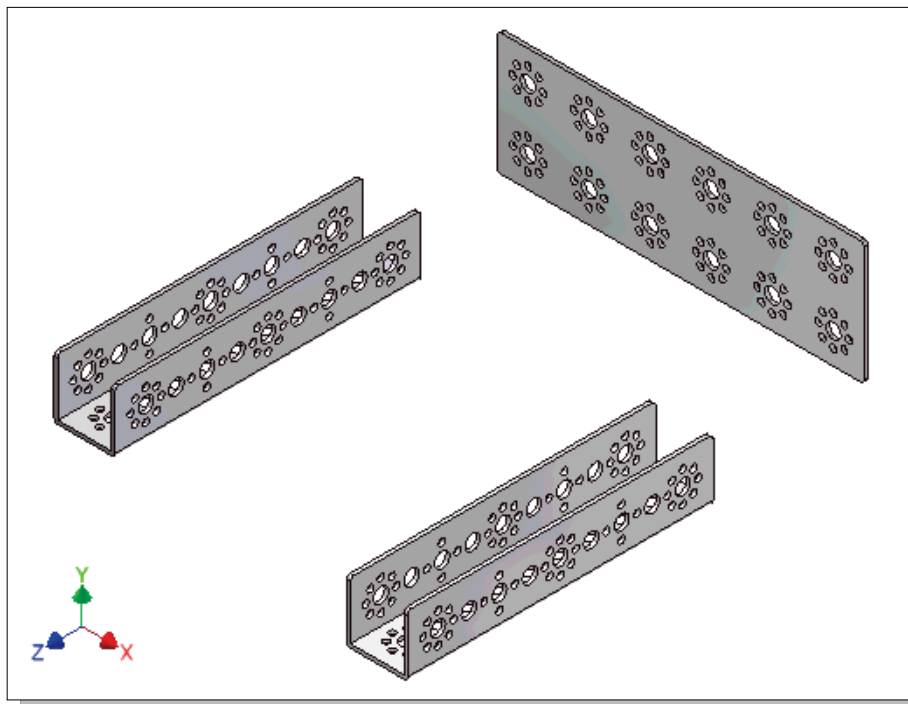
Adjusting the Orientation of a Grounded Part

- The orientation of the first component, the grounded part of an assembly, can be adjusted and additional constraints can be applied.

1. Click once with the **right-mouse-button** on the first component to show the option list.

2. Select **Grounded** to allow the part to be adjusted. Note the grounded symbol is removed in the *Model* tree.
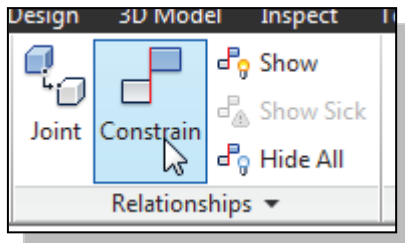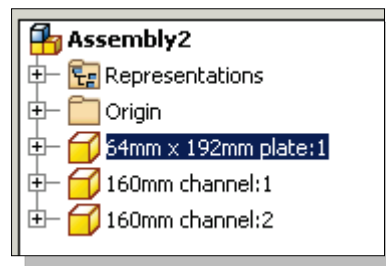
3. In the *Assembly* panel, select the **Constrain** command by left-mouse-clicking once on the icon.
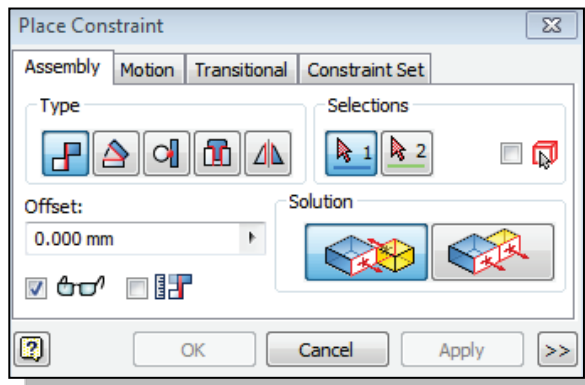
4. In the *Place Constraint* dialog box, confirm the **Mate** constraint is activated.
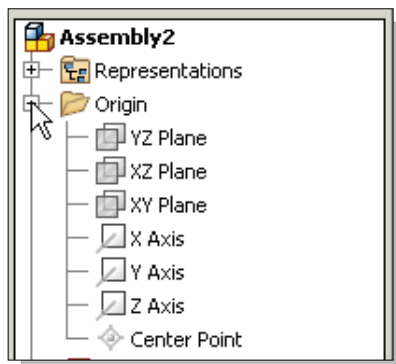
5. Click on the [+] symbol in front of the *Origin* item.

- Note that each assembly model also contains a set of datum features, just like a regular part. These datum features can be used to align the different components of the design.
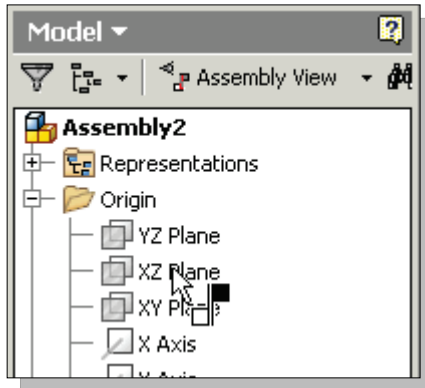
6. Select the XZ Plane of the assembly datum features as the 1st item for the Constrain command.

7. Select the XY Plane of the **64mmx192mm Plate** part as the 2nd item for the Constrain command.

8. Click on the **Apply** button to accept the selection and apply the Mate constraint. Note the orientation of the plate has been adjusted.
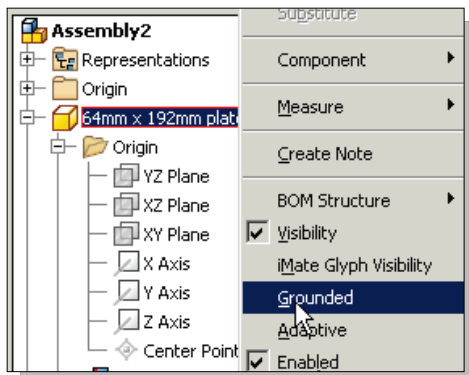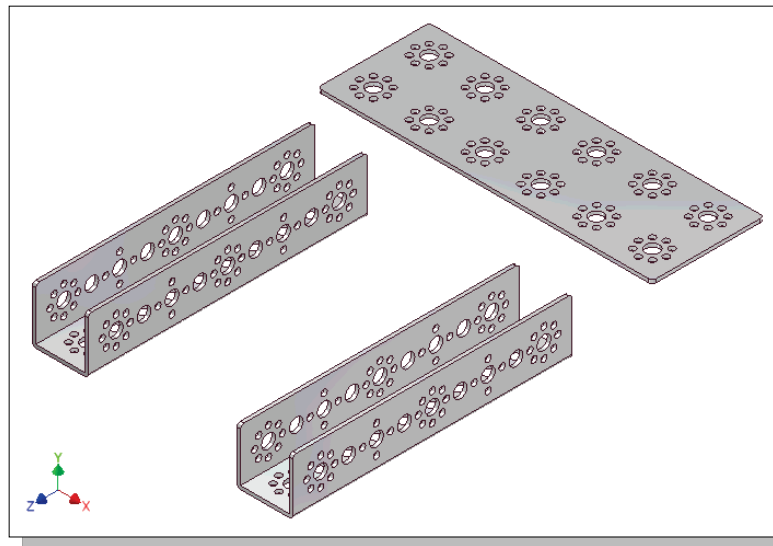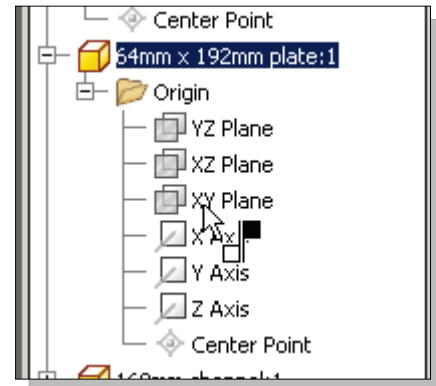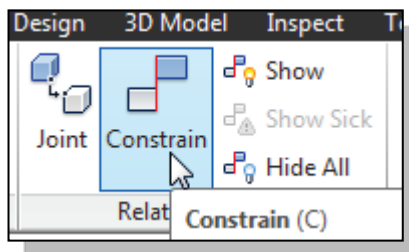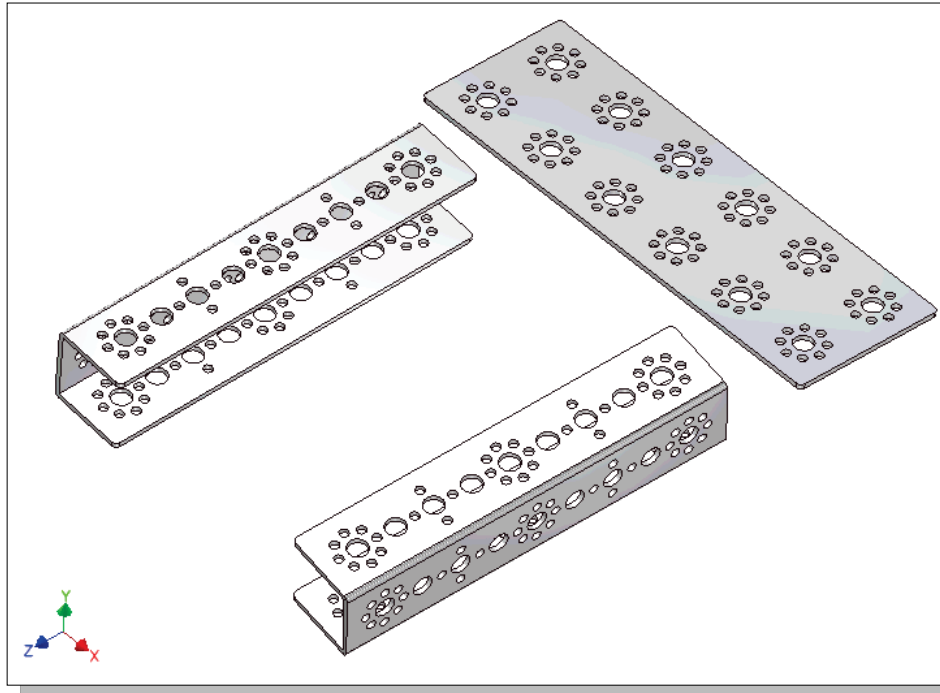

9. To make the *Plate* part grounded again, access the option through the option list in the *Model* history tree as shown.

Adjusting the Orientation of Parts
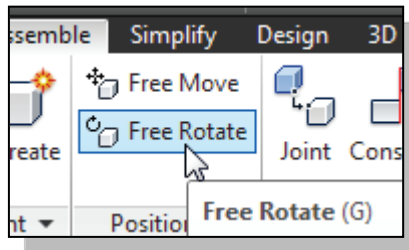
1. Select the **Free Rotate** command by left-mouse-clicking once on the icon.

2. On your own, use the left-mouse-button and rotate the *Channel Plates*, roughly matching the orientation in the figure below.

3. In the *Assembly* panel, select the **Constrain** command by left-mouse-clicking once on the icon.
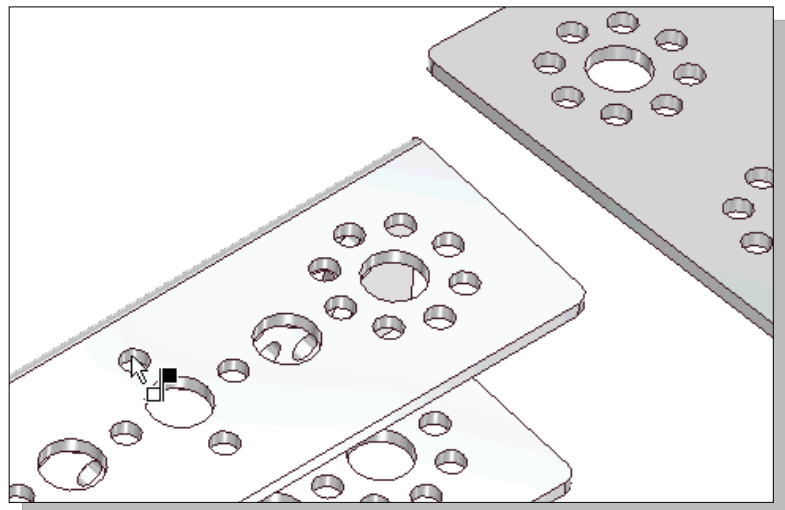
4. In the *Place Constraint* dialog box, switch to the **Mate** constraint.
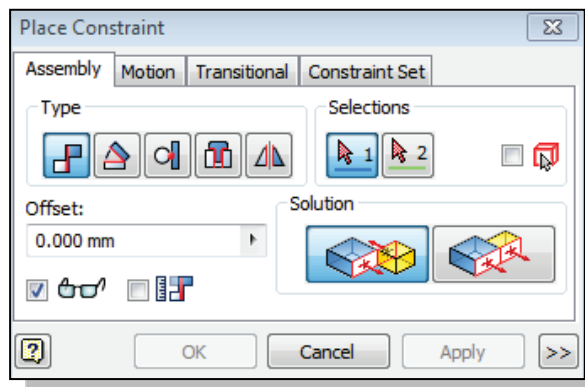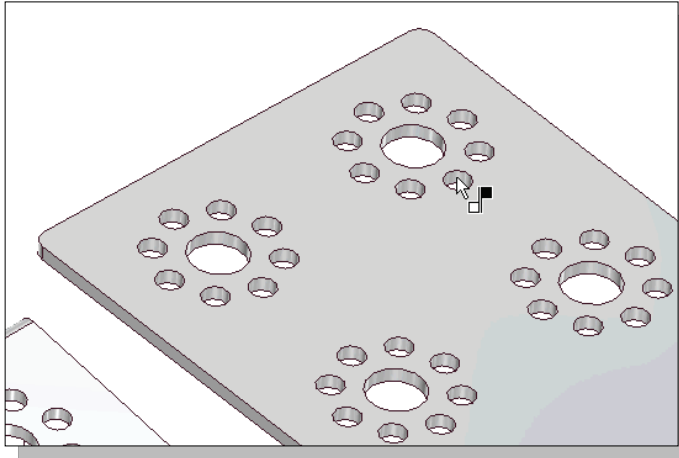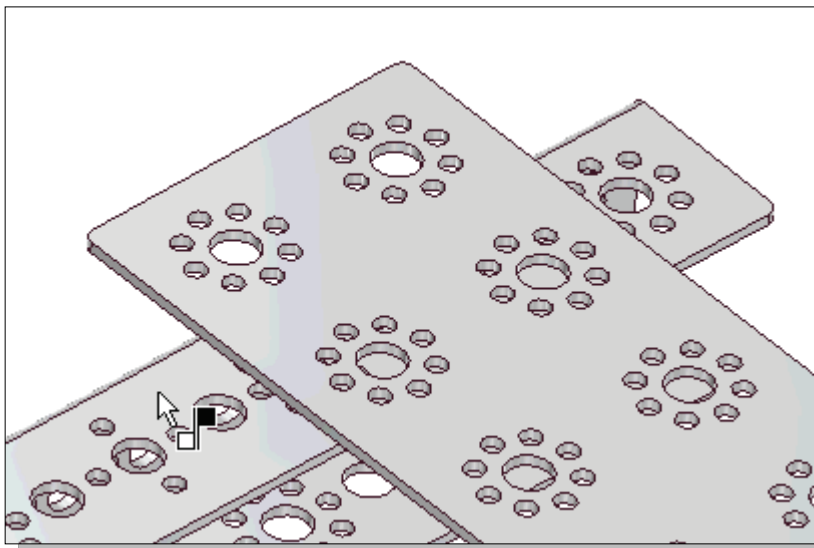
5. Select the **inside cylindrical surface** of the right small hole of the top left hole pattern of the flat *Plate* as shown.

6. Select the **inside cylindrical surface** of the left small hole of the *Channel* plate on the left as the 2nd item for the Constrain command.

7. Click on the **Apply** button to accept the selection and apply the Mate constraint.

8. Select the **top surface** of the *Channel* plate on the left as the first item to apply the Constrain command.
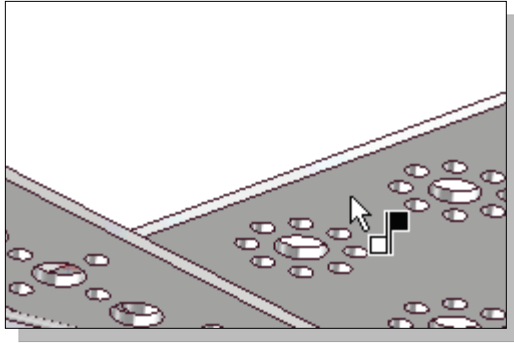
9. Select the **bottom surface** of the flat *Plate* part as the 2nd item for the Constrain command.

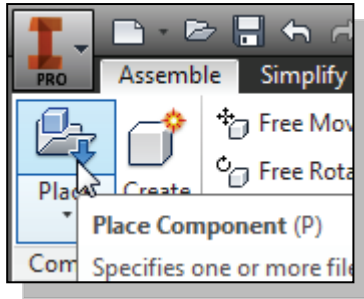10. Click on the **Apply** button to accept the selection and apply the Mate constraint.

11. On your own, repeat the above steps and align the other holes adjacent to the aligned holes of the two parts as shown.
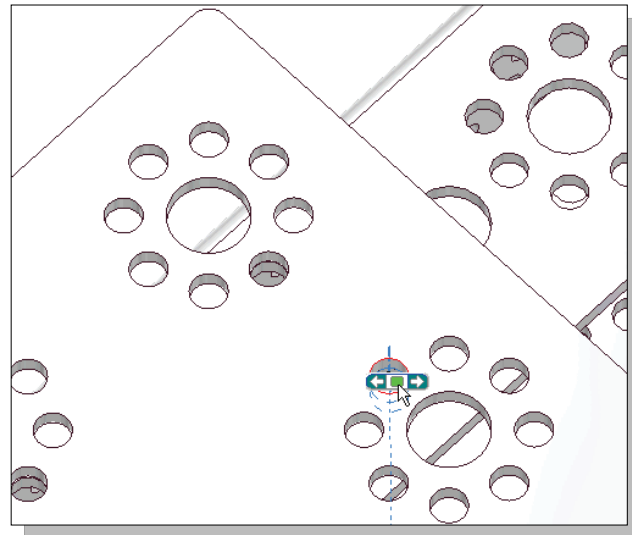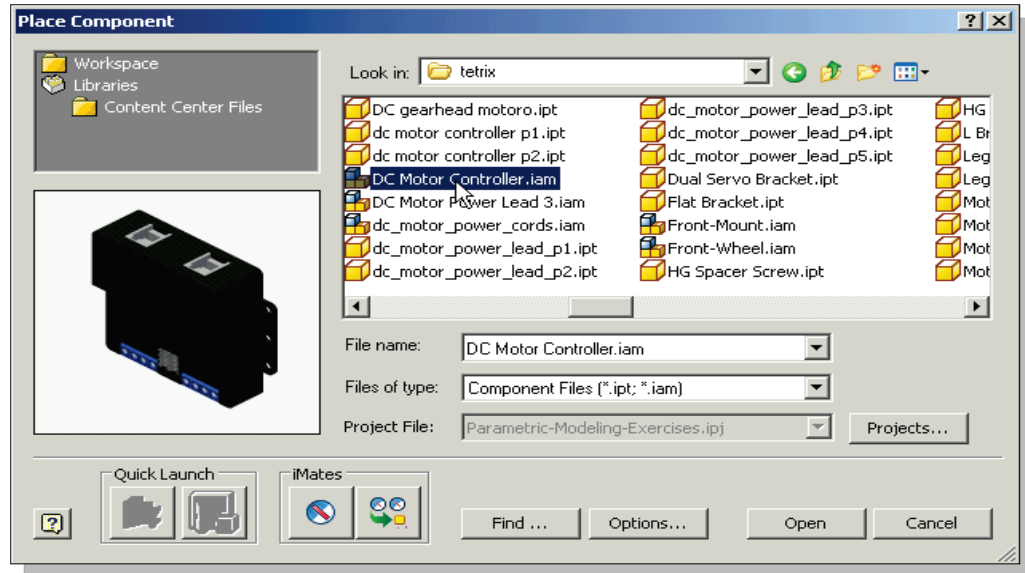

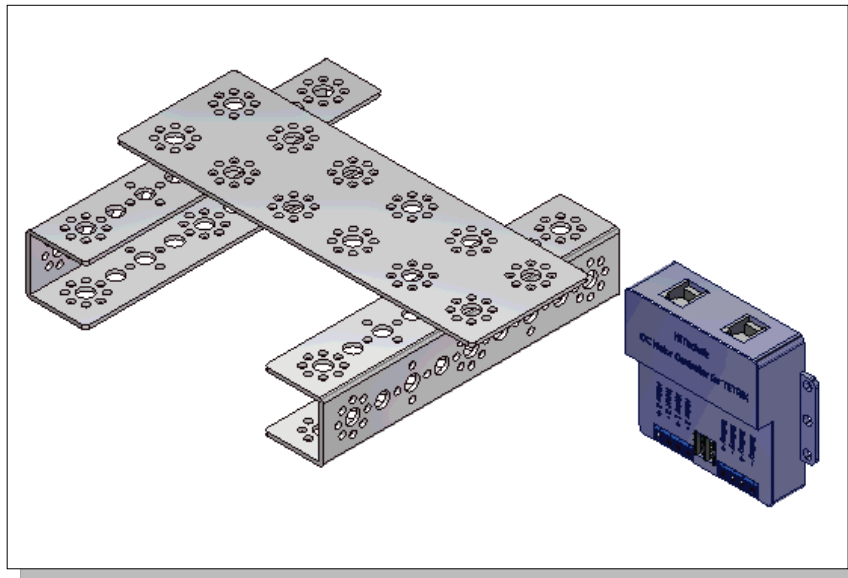12. On your own, repeat the above steps and align the *Channel* plate on the right as shown.
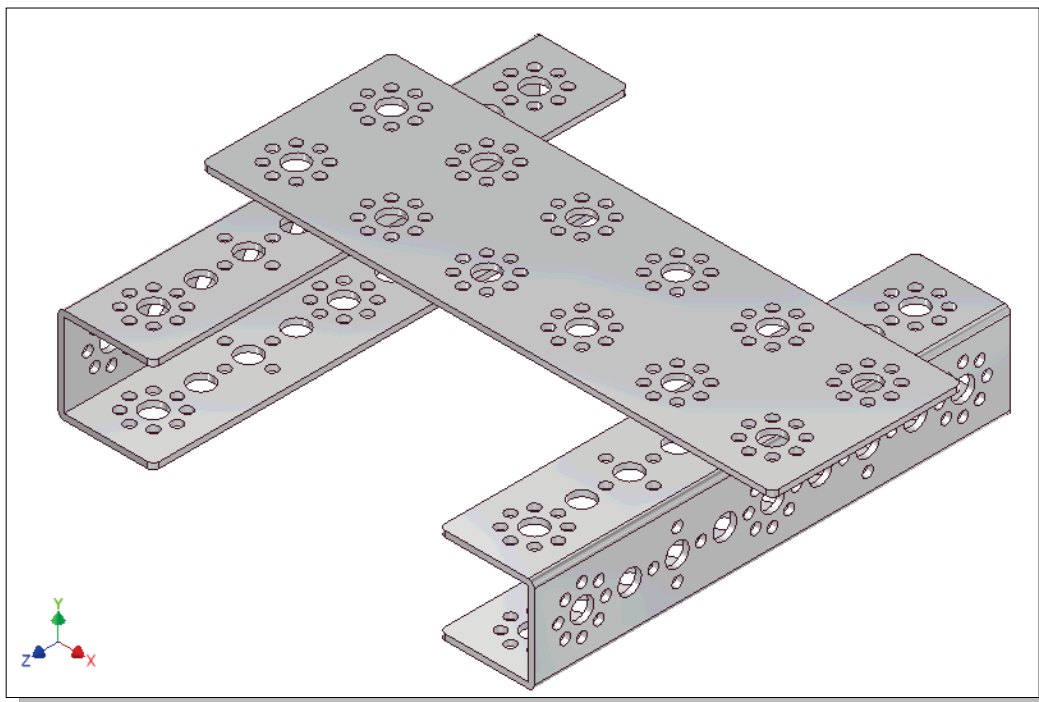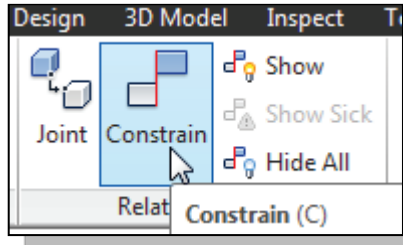
Assemble the DC Motor Controller

1. In the *Assembly* panel, select the **Place Component** command by left-mouse-clicking the icon.

2. Select the *DC Motor Controller* assembly in the list window. Click on the **Open** button to retrieve the model.

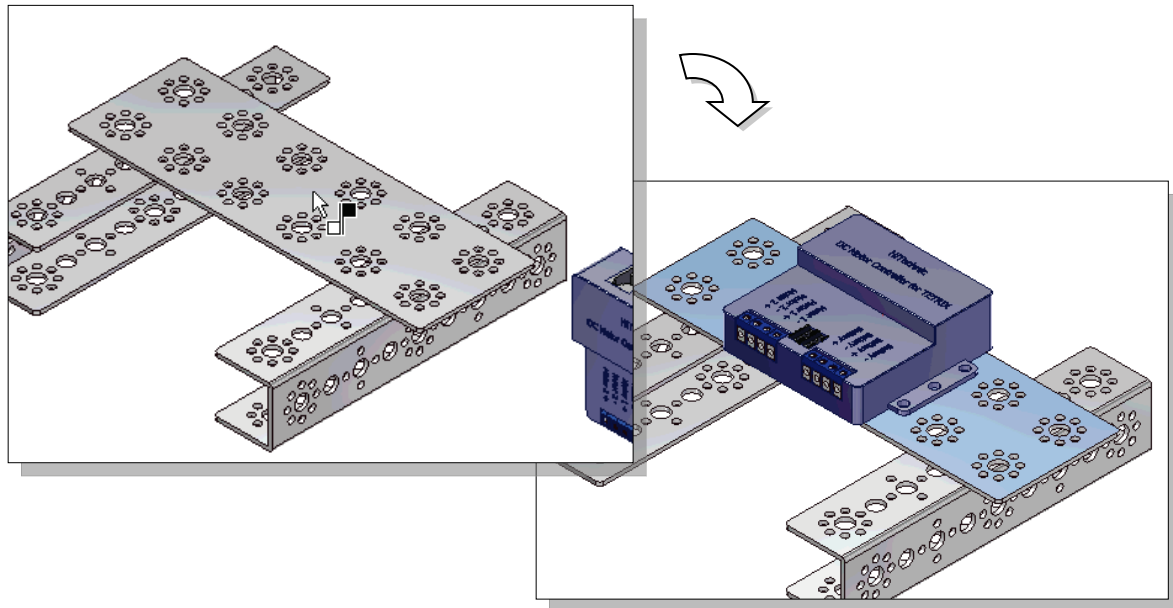3. Place a copy of the *DC Motor Controller* assembly to the right side of graphics area.

4. Inside the graphics window, right-mouse-click once to bring up the option menu and select **Done** to end the Place Component command.

5. In the *Assembly* panel, select the **Constrain** command by left-mouse-clicking once on the icon.
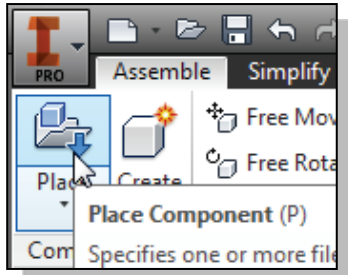
6. On your own, align the **DC Motor Controller** to the top face of the flat **Plate**.

7. Also apply a **Mate** constraint to align two of the side holes of the **DC Motor Controller** to the corresponding holes of the flat **Plate** part as shown.
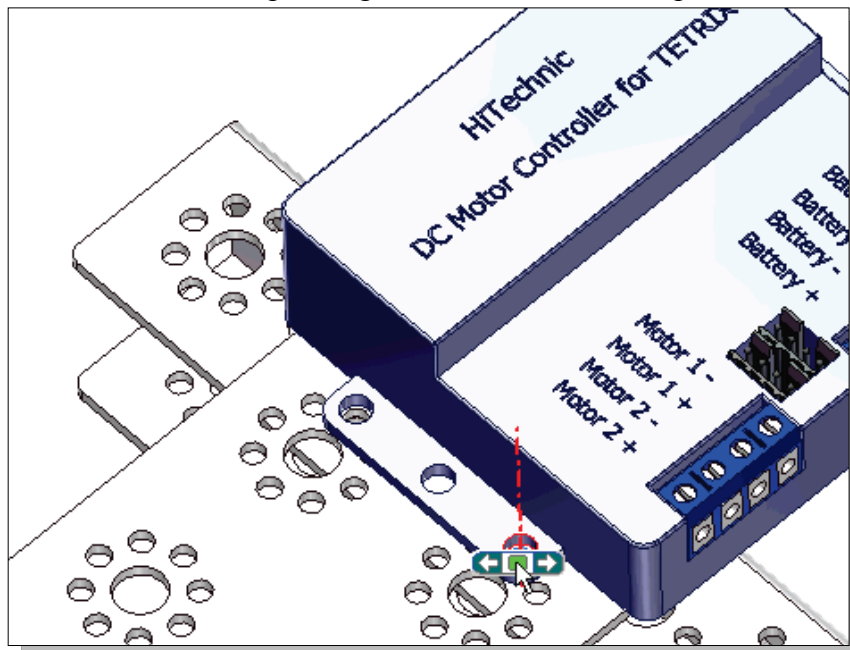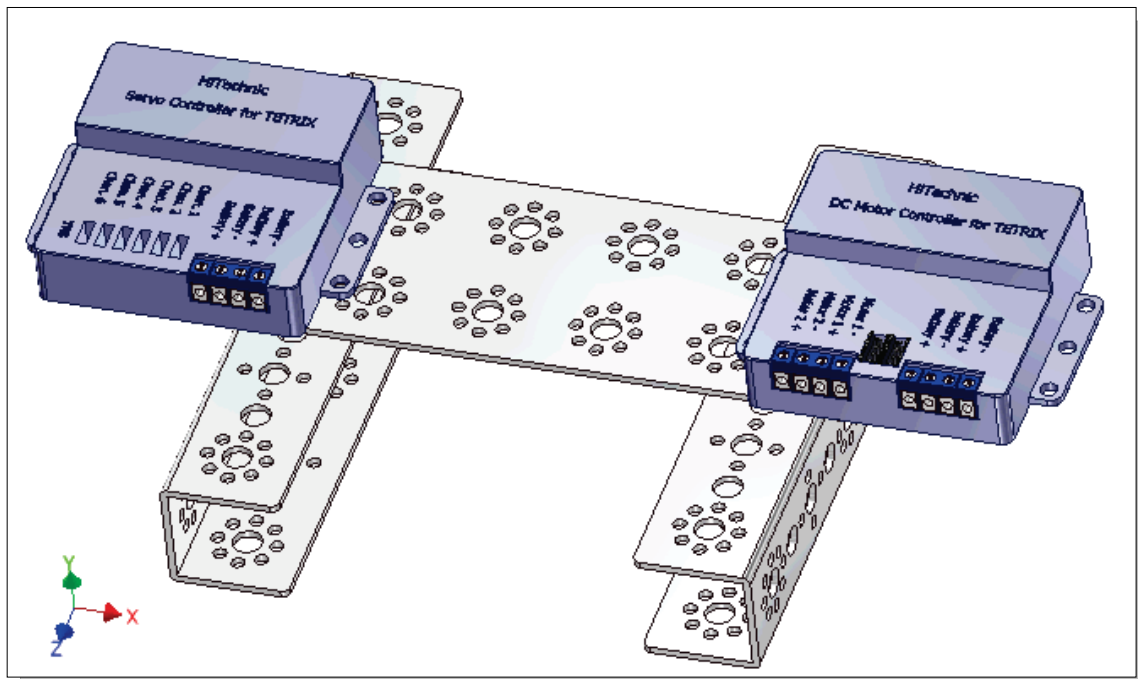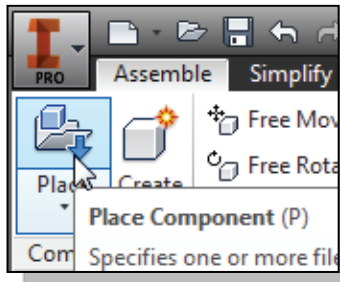
Assemble the Servo Controller

1. In the *Assembly* panel, select the **Place Component** command by left-mouse-clicking the icon.

2. Select the *Servo Controller* assembly in the list window. Click on the **Open** button to retrieve the model.
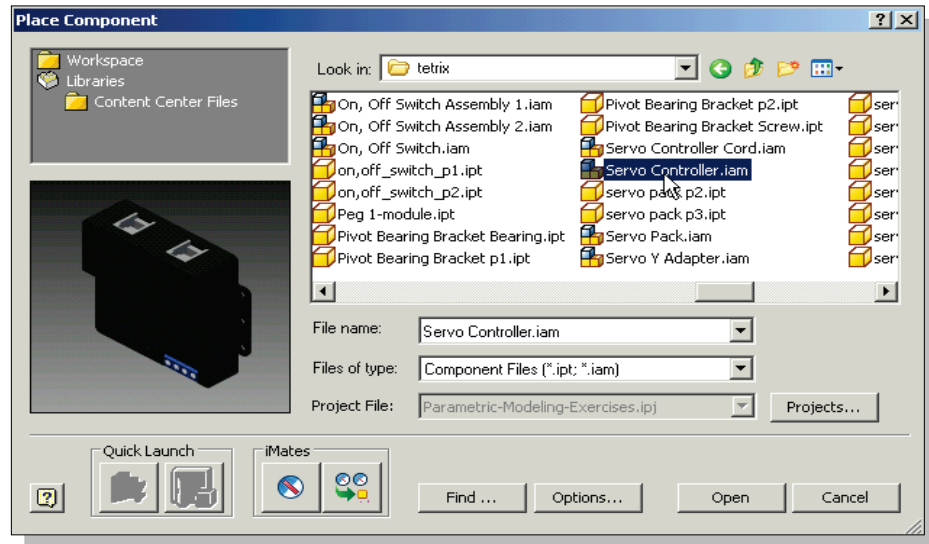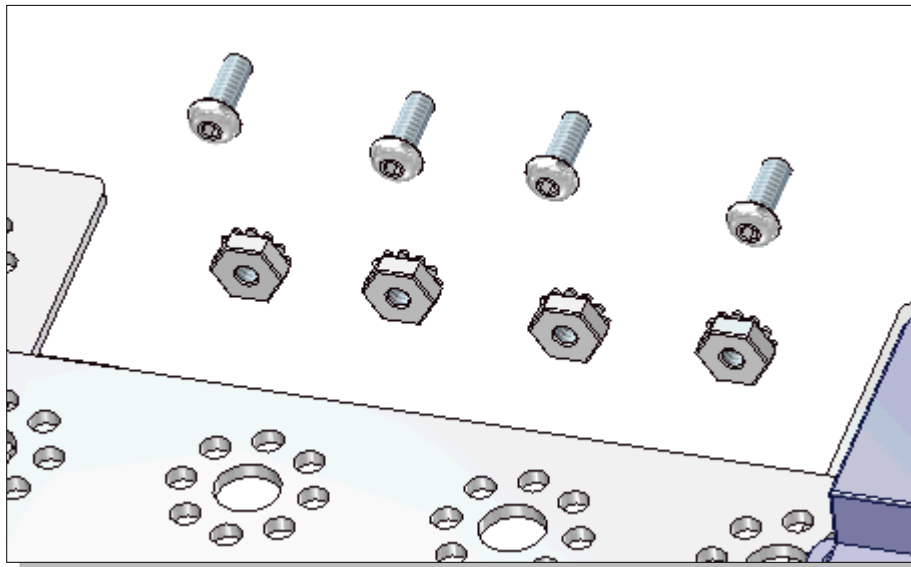

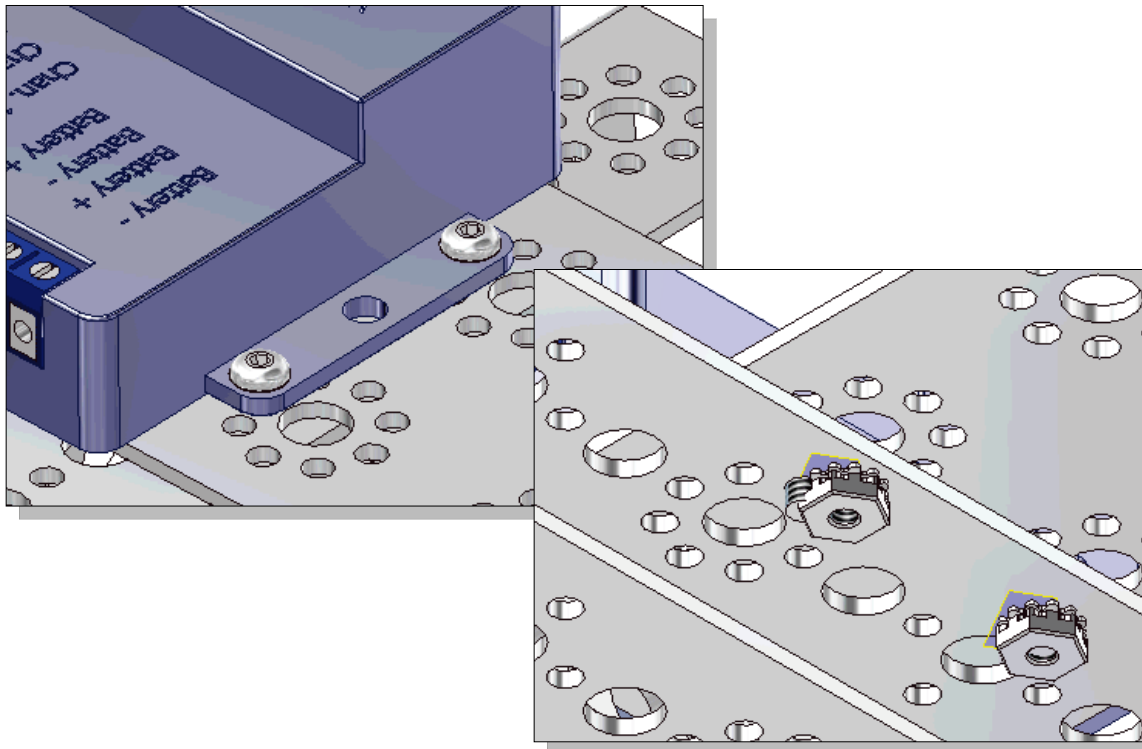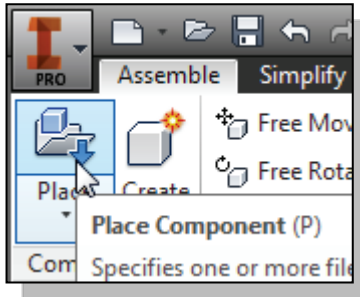3. On your own, place a copy of the *Servo Controller* and assemble it to the left side of assembly as shown.

4. In the *Assembly* panel, select the **Place Component** command by left-mouse-clicking the icon.

5. Place four copies of the *6-32x.375 in BHCS Screw* part and four *6-32 Nut* parts toward the top side of the assembly.
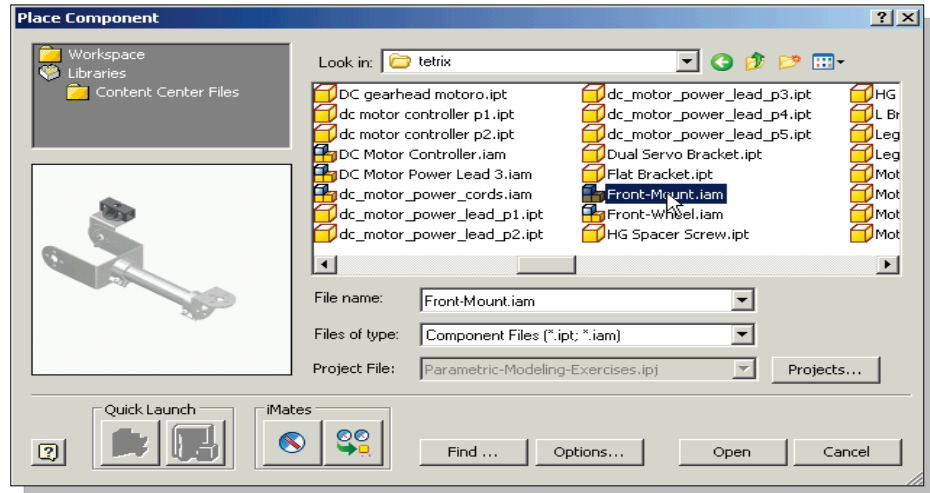
6. On your own, assemble the *Screws* and *Nuts* in the aligned holes as shown. (Hint: On the *Nuts*, align Work Plane3 to the bottom of the flat *Plate*.)
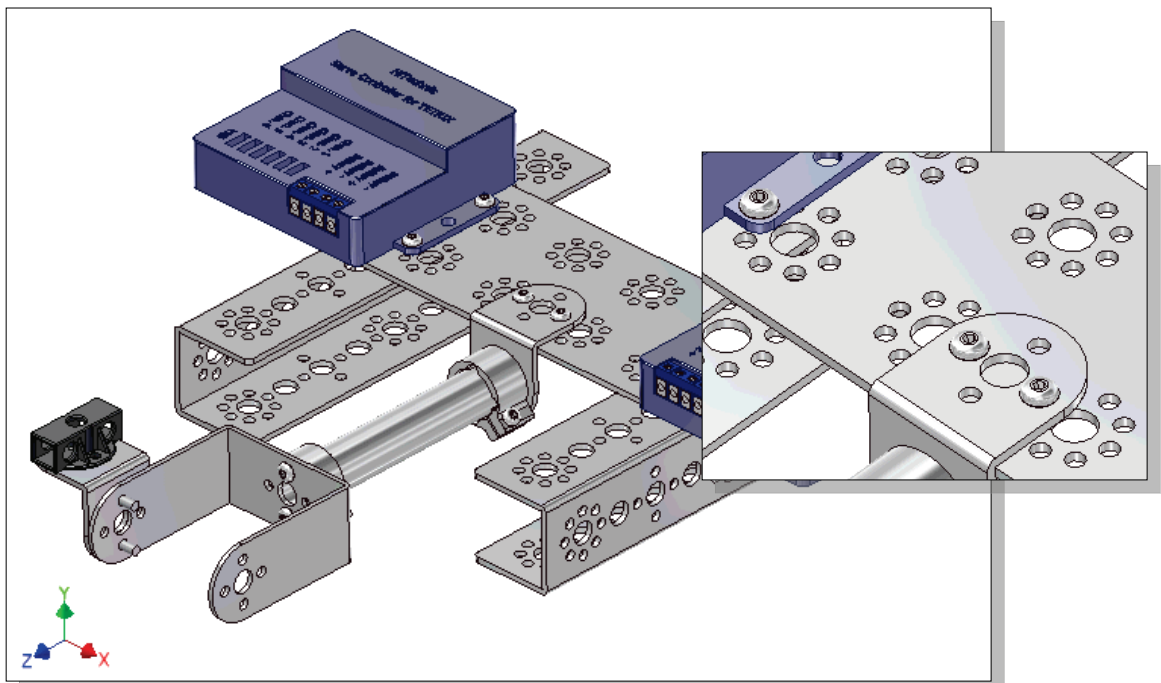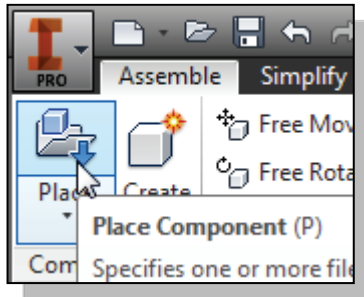
Completing the Chassis

1. In the *Assembly* panel, select the **Place Component** command by left-mouse-clicking the icon.

2. Select the *Front-Mount* assembly in the list window. Click on the **Open** button to retrieve the model.

3. Place one copy of the assembly toward the left side of the graphics window.

4. On your own, repeat the same procedure to assemble the *Front-Mount* assembly and two copies of *6-32x.375 in BHCS Screw* as shown.
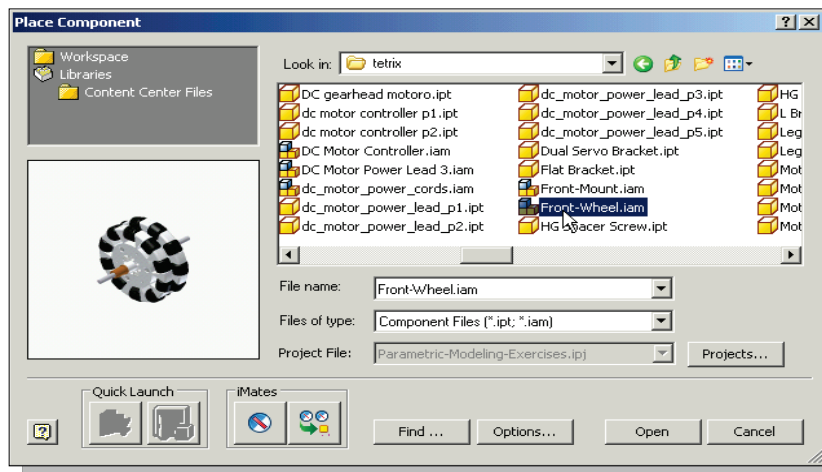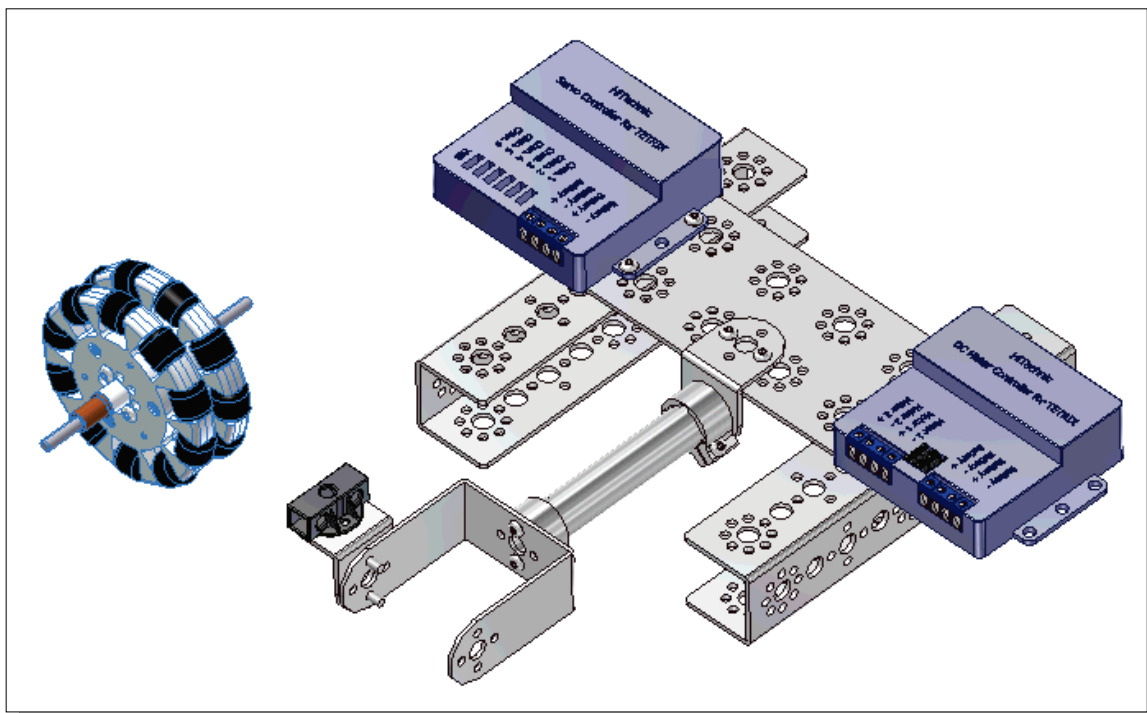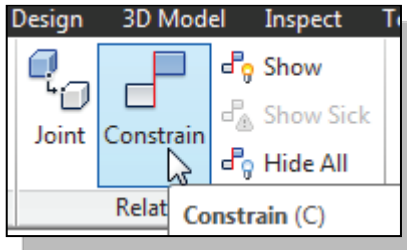
Assemble the Front-Wheel Assembly

1. In the *Assembly* panel, select the **Place Component** command by left-mouse-clicking the icon.

2. Select the ***Front-Wheel*** assembly in the list window. Click on the **Open** button to retrieve the model.
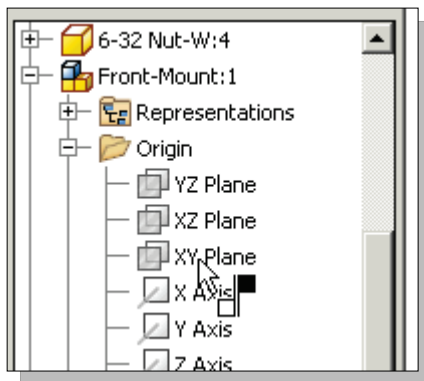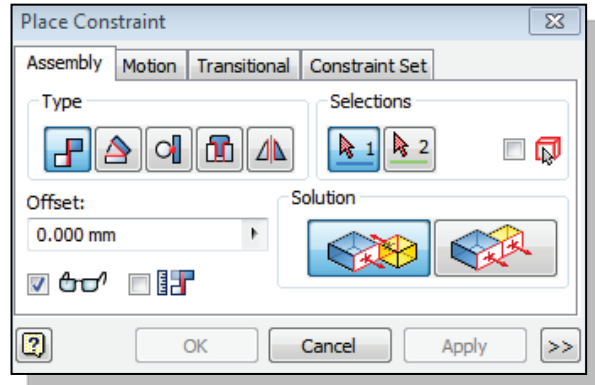
3. Place one copy of the assembly toward the left side of the graphics window.

4. In the *Assembly* panel, select the **Constrain** command by left-mouse-clicking once on the icon.

5. In the *Place Constraint* dialog box, confirm the constraint *Type* is set to the **Mate** constraint.

6. In the *Model* history tree, select the XY Plane of the *Front-Mount* assembly as the first item to apply the Mate constraint.


7. In the *Model* history tree, select the XY Plane of the *Front-Wheel* assembly as the second item to apply the Mate constraint.

8. Click on the **Apply** button to accept the selection and apply the Mate constraint.
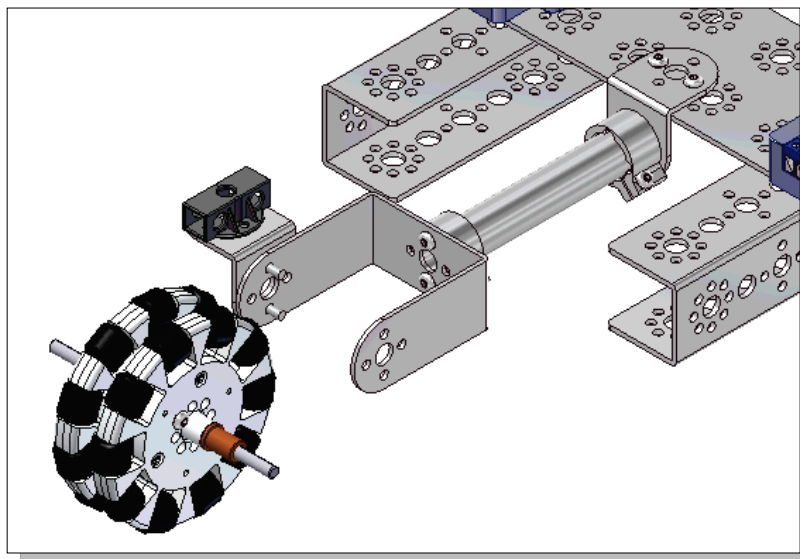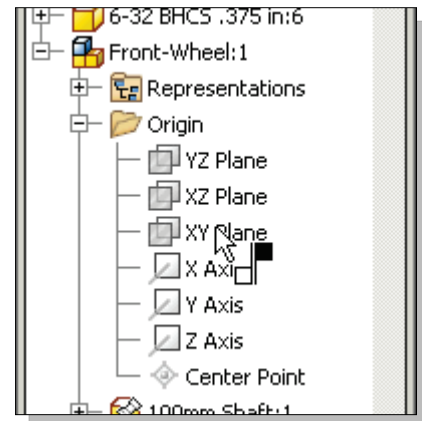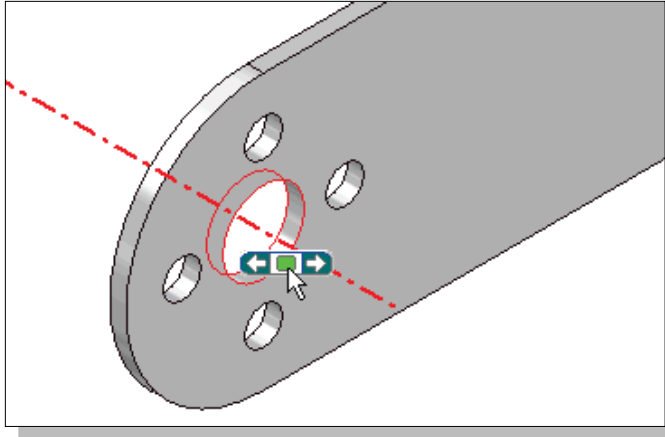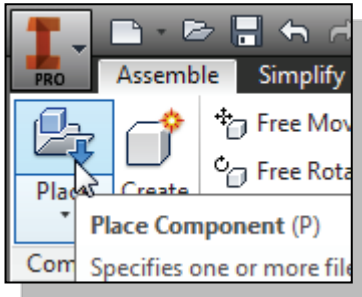
9. Select the inside cylindrical surface of one of the large hole on the *Front-Mount* assembly as the first item to constrain.

10. Select the cylindrical surface of the shaft of the *Front-Wheel* assembly as the second item to constrain.

11. Click on the **Apply** button to accept the selection and apply the Mate constraint
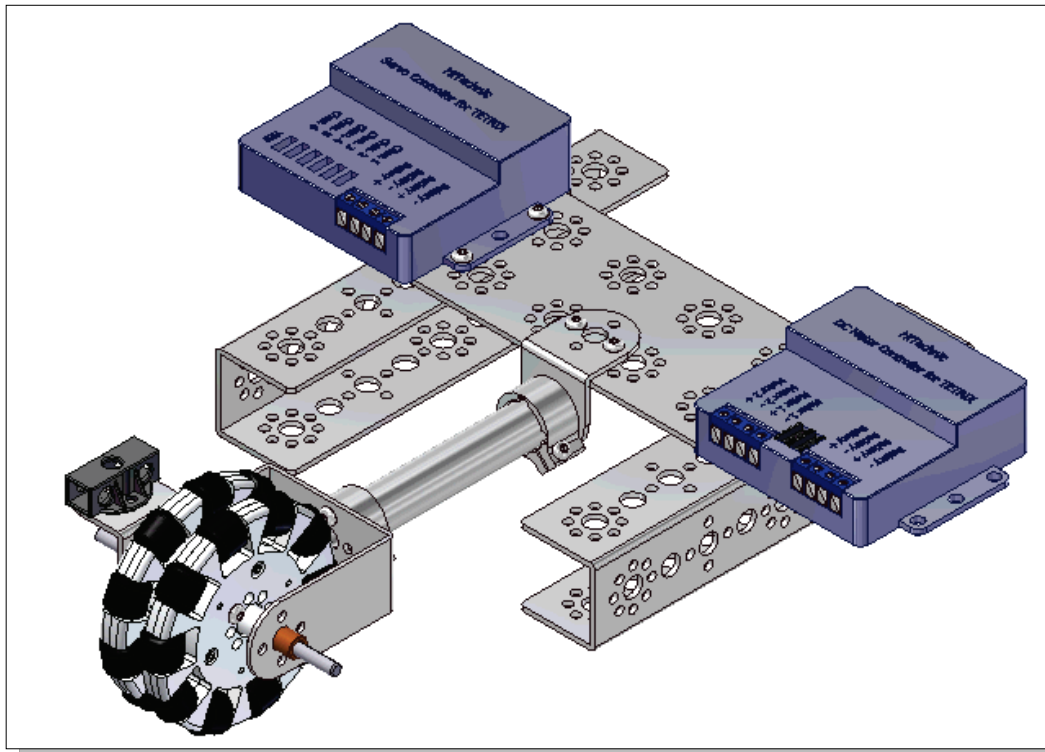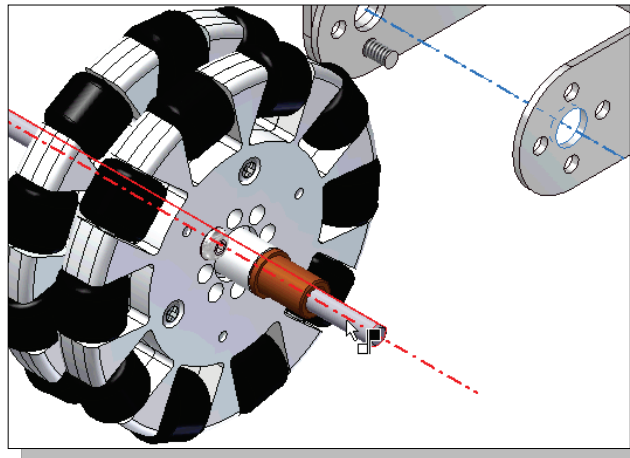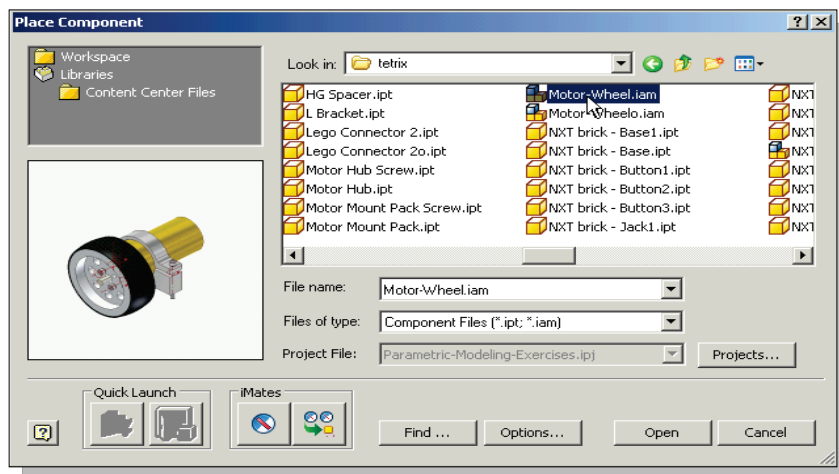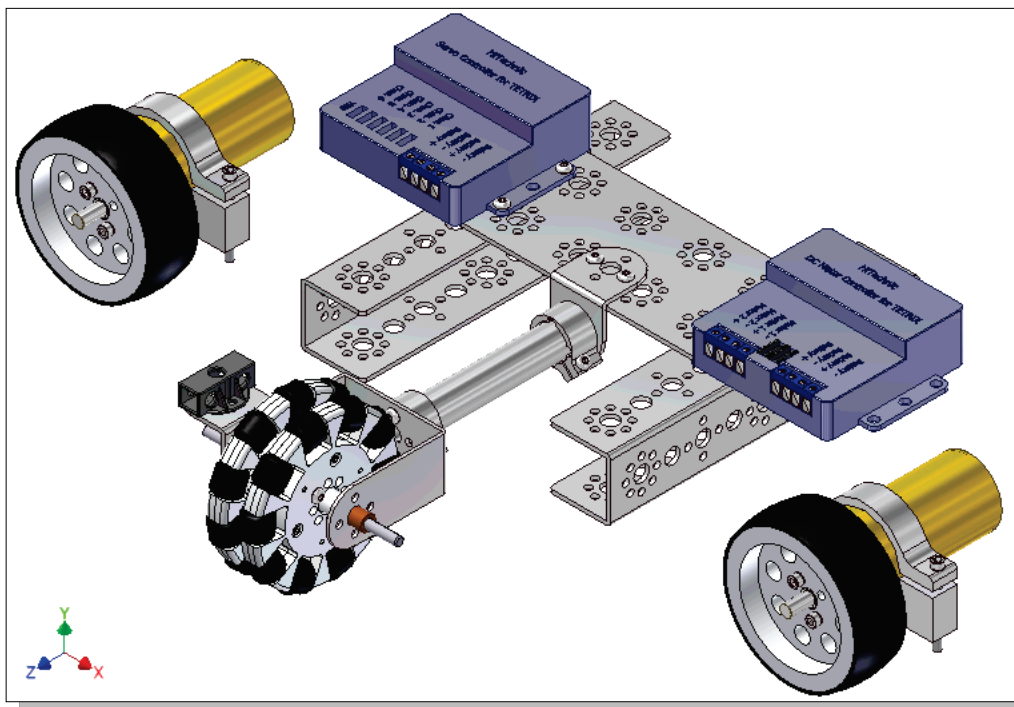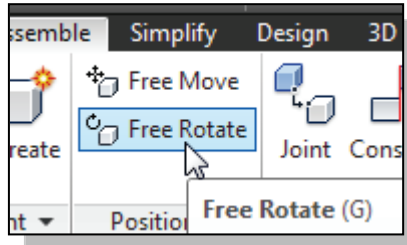
Assemble the Motor-Wheel Assembly

1. In the *Assembly* panel, select the **Place Component** command by left-mouse-clicking the icon.

2. Select the ***Motor-Wheel a***ssembly in the list window. Click on the **Open** button to retrieve the model.
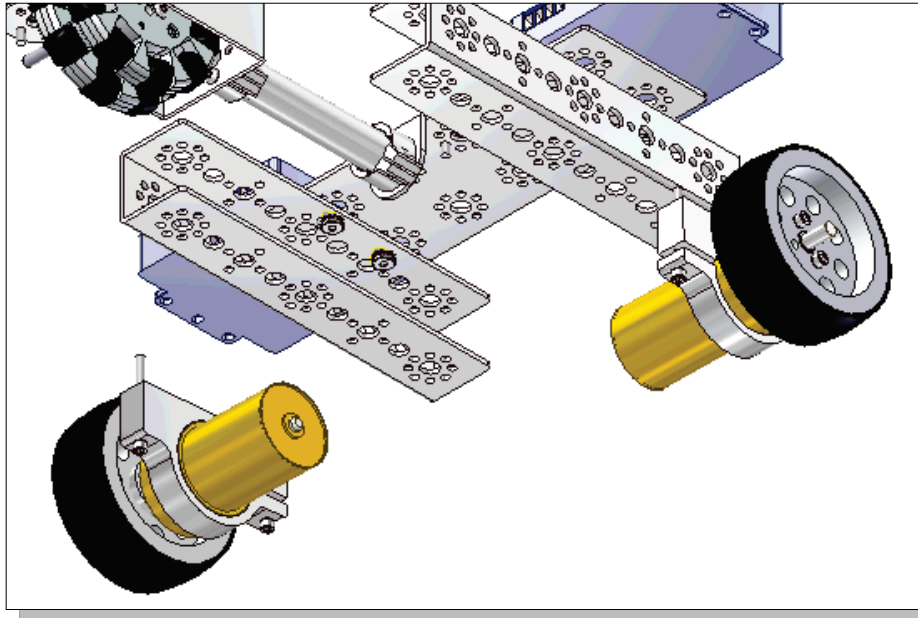

3. Place two copies of the assembly next to the design as shown.

4. Select the **Free Rotate** command by left-mouse-clicking once on the icon.

5. On your own, use the left-mouse-button and rotate the two copies of the *Motor-Wheel* subassembly to the orientations as shown.
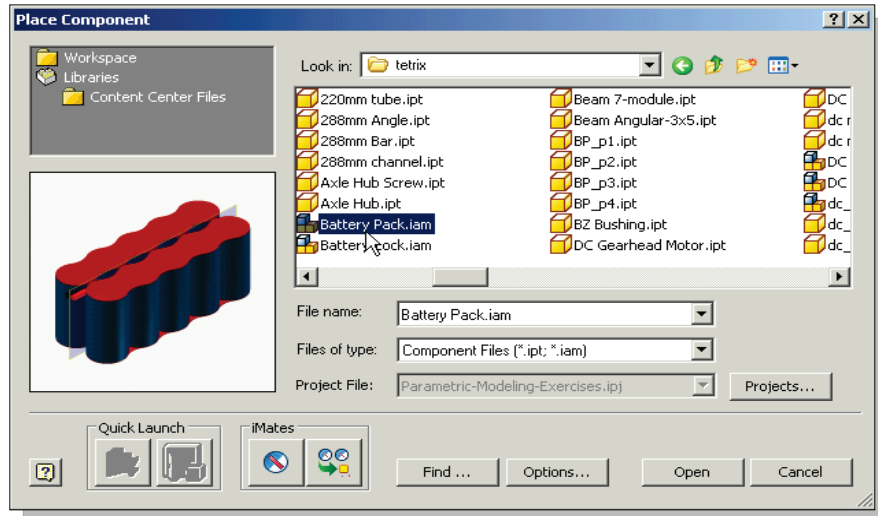

6. On your own, apply the necessary constraints to align the *Motor-Wheel* subassemblies to the two *Channel* plates as shown.
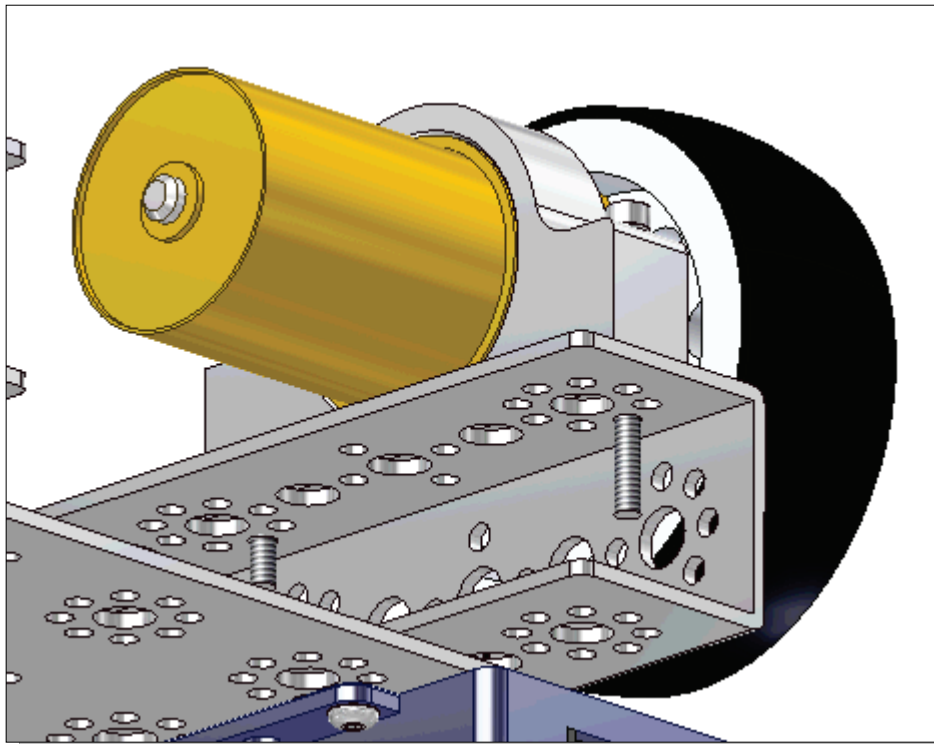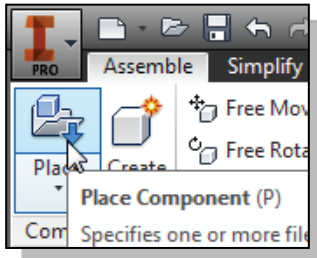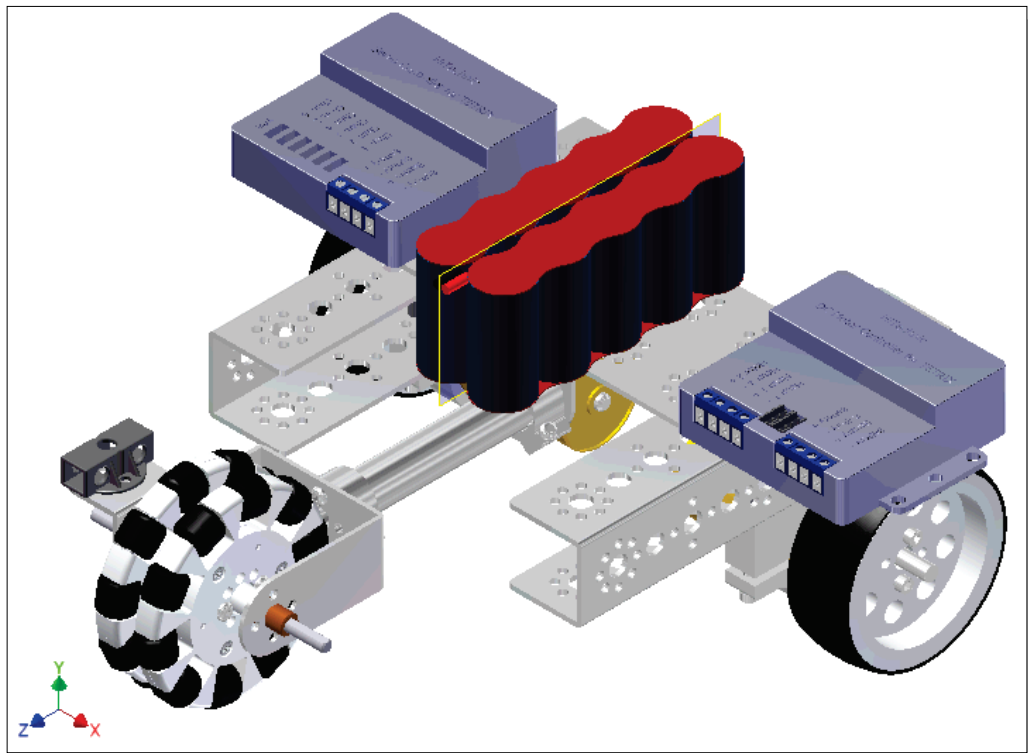
7. In the *Assembly* panel, select the **Place Component** command by left-mouse-clicking the icon.

8. Select the ***Battery-Pack*** assembly in the list window. Click on the **Open** button to retrieve the model.

9. Place one copy of the ***Battery-Pack*** assembly next to the assembly.

10. On your own, apply the necessary constraints to align the ***Battery-Pack*** subassembly to the center of the assembly as shown.
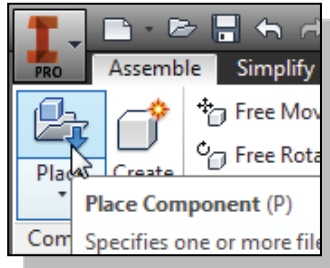
11. In the *Assembly* panel, select the **Place Component** command by left-mouse-clicking the icon.

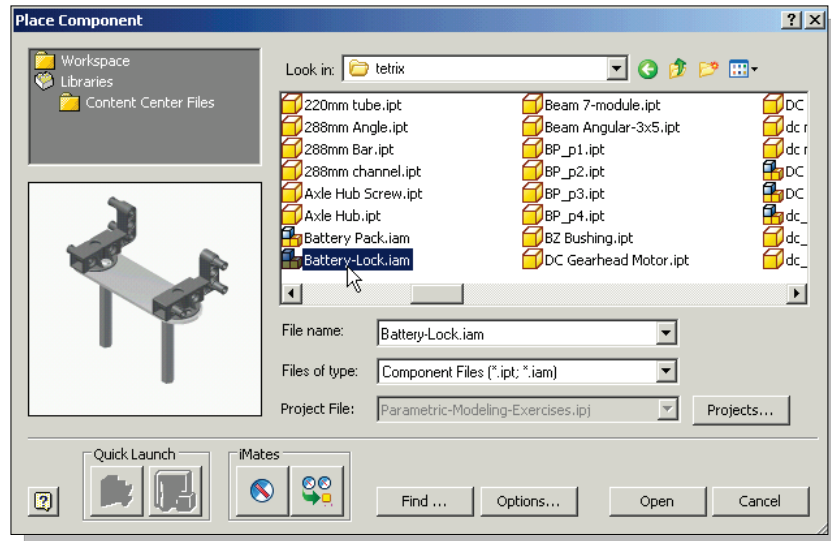12. Select the *Battery-Lock* assembly in the list window. Click on the **Open** button to retrieve the model.

13. Place one copy of the *Battery-Lock* assembly next to the assembly.

14. On your own, apply the necessary constraints to align the *Battery-Lock* subassembly to the center of the assembly as shown.
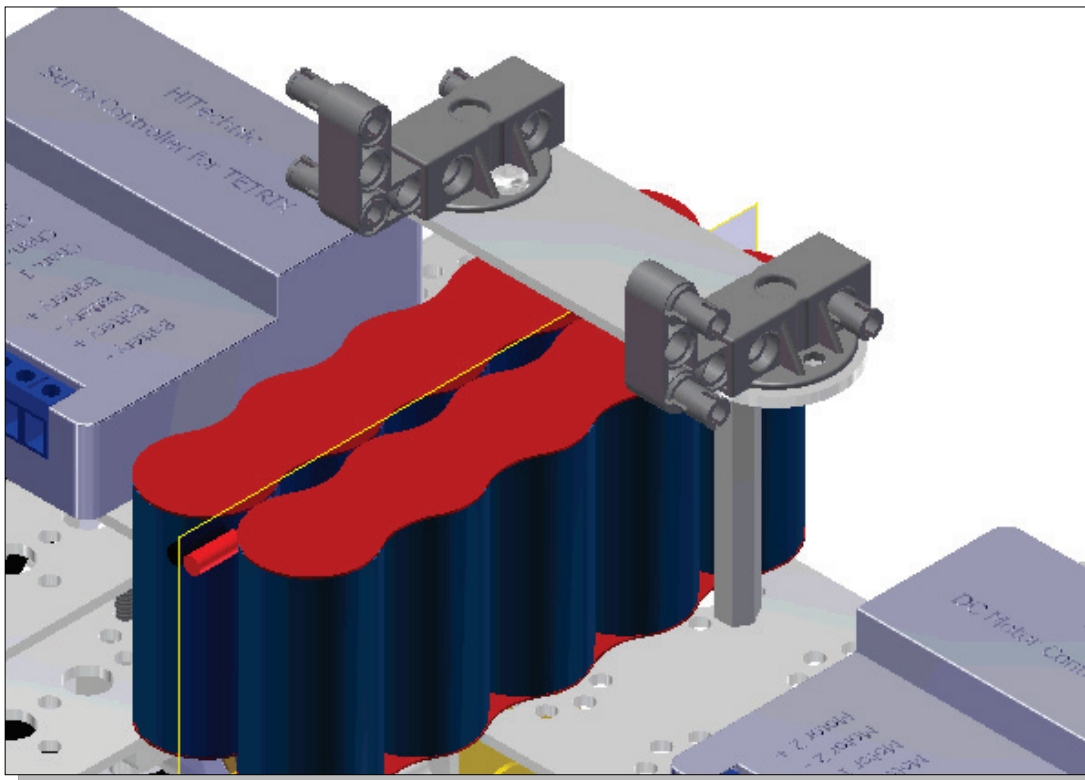
15. In the *Assembly* panel, select the **Place Component** command by left-mouse-clicking the icon.
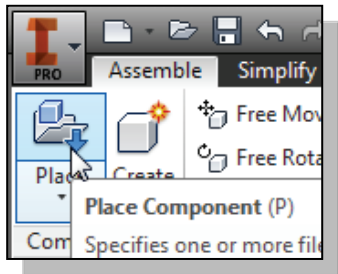
16. Select the ***Beam 7-Module*** part in the list window. Click on the **Open** button to retrieve the model.

17. Place two copies of the ***Beam 7-Module*** assembly next to the assembly

18. On your own, apply the necessary constraints to align the copies of the ***Beam 7-Module*** part to the assembly as shown.
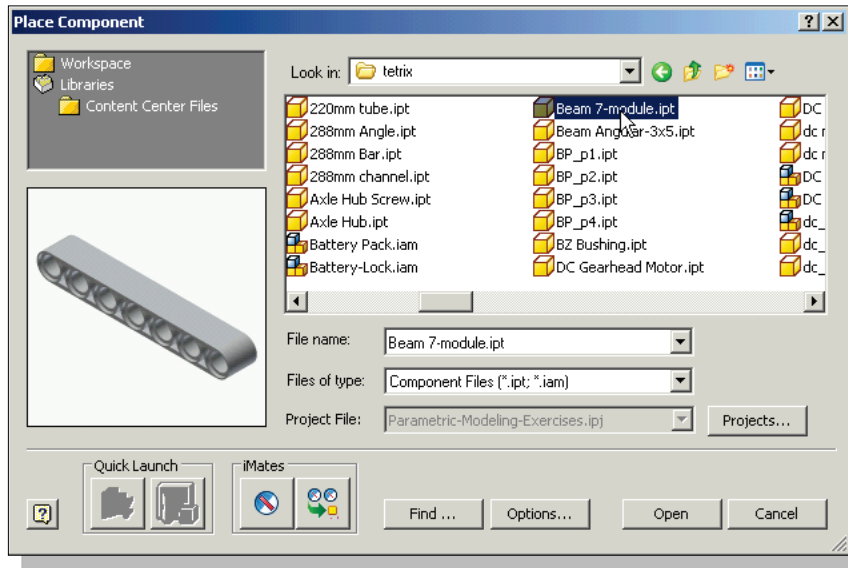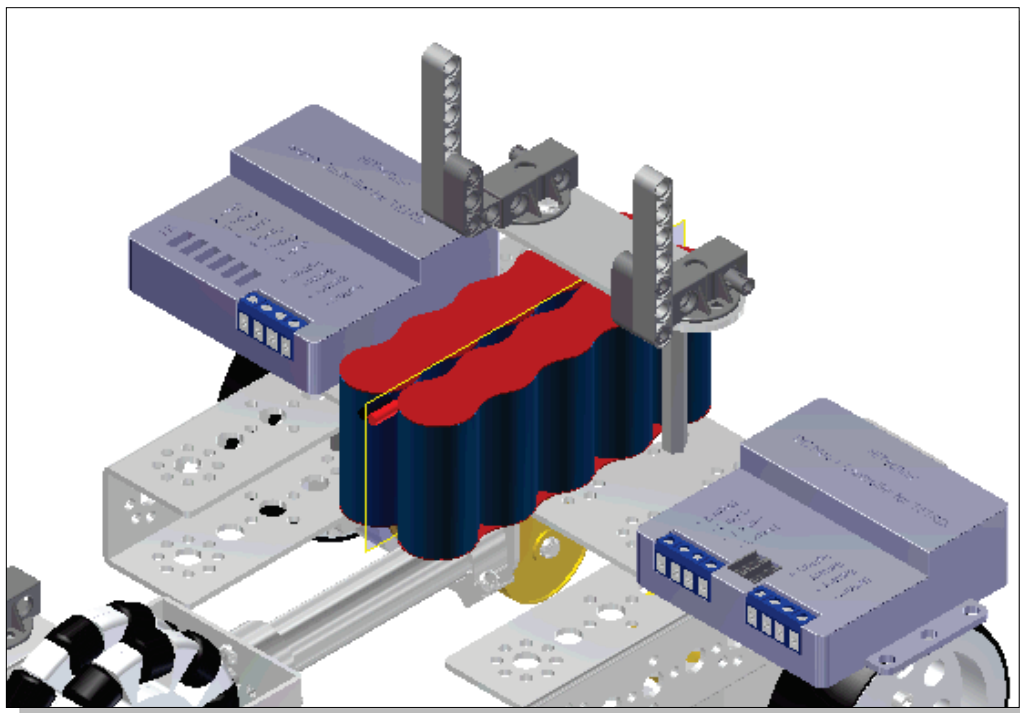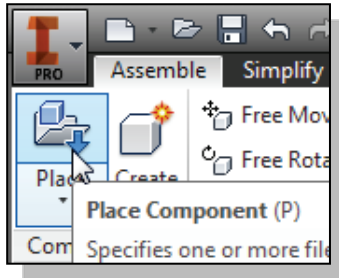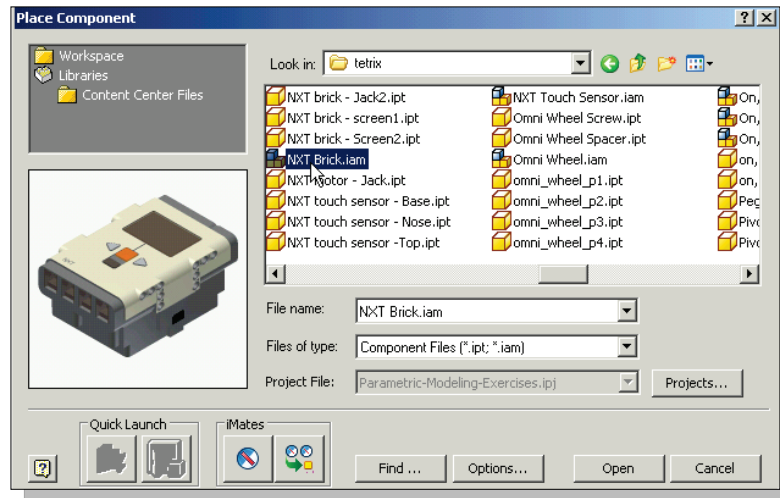
Assemble the NXT Micro-controller

1. In the *Assembly* panel, select the **Place Component** command by left-mouse-clicking the icon.

2. Select the *NXT-Brick* assembly in the list window. Click on the **Open** button to retrieve the model.

3. Place one copy of the assembly toward the right side of the graphics window.

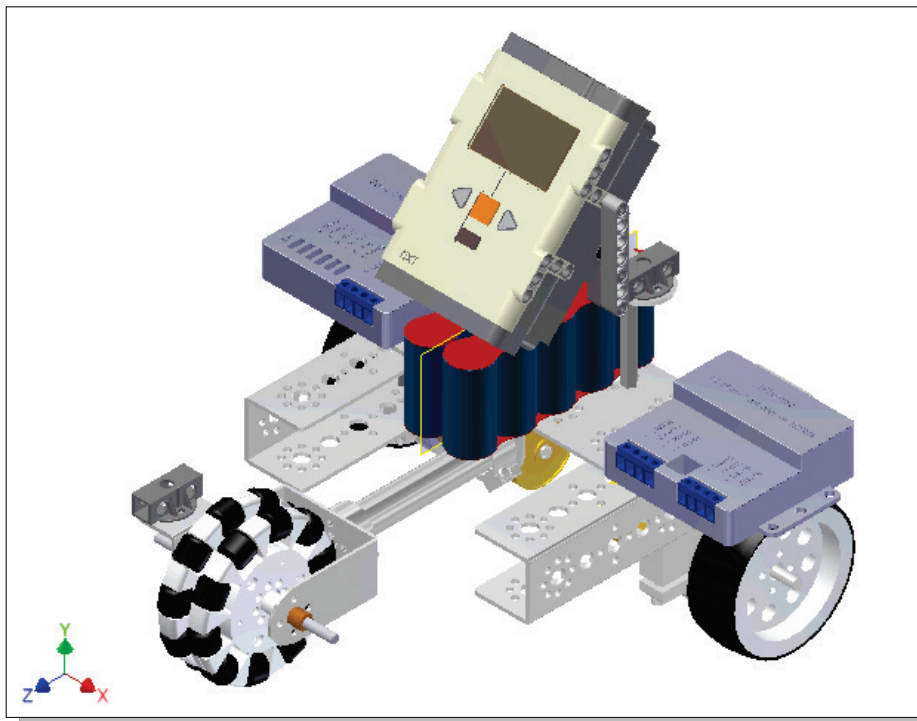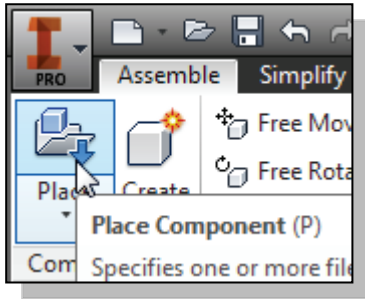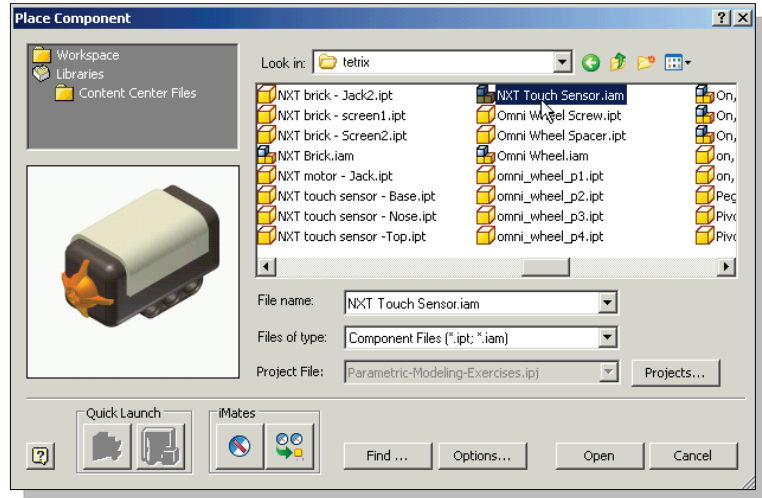4. On your own, also place two copies of *Peg 1-Module* near the assembly. Assemble the parts as shown.

Assemble the NXT Touch-Sensor

1. In the *Assembly* panel, select the **Place Component** command by left-mouse-clicking the icon.

2. Select the *NXT Touch Sensor* assembly in the list window. Click on the **Open** button to retrieve the model.

3. Place one copy of the assembly toward the right side of the assembly.

4. On your own, also place four copies of the *Peg 1-Module* part and one copy of the *Beam 7-Module* part into the current assembly as shown.
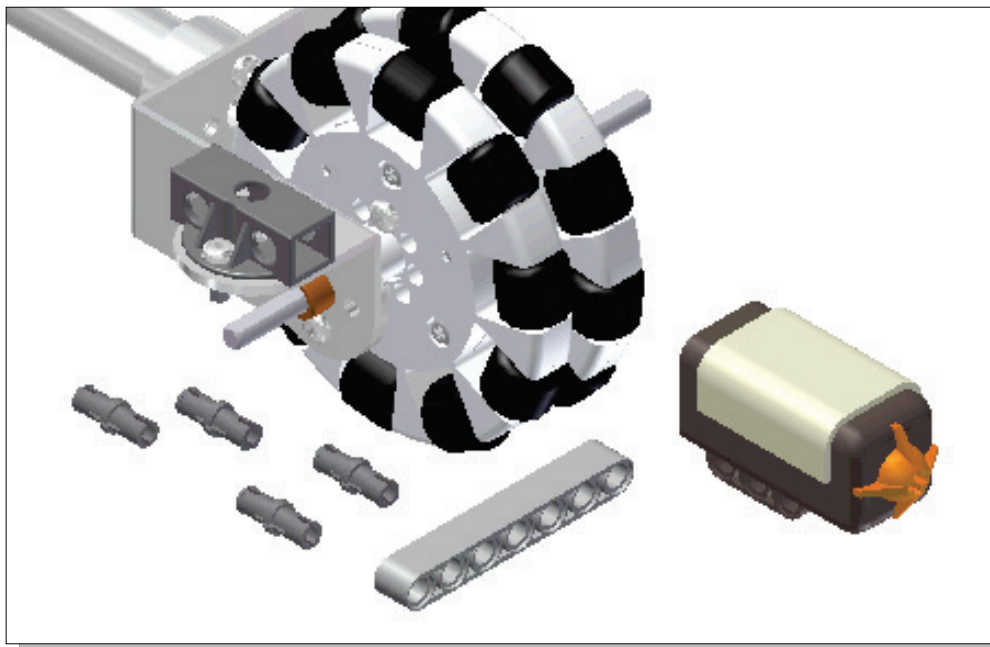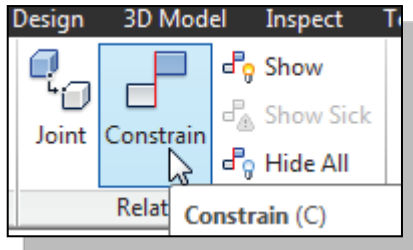
5. In the *Assembly* panel, select the **Constrain** command by left-mouse-clicking once on the icon.

6. Assemble the ***Beam 7-Module*** to the **LEGO *Connector 2*** part as shown.
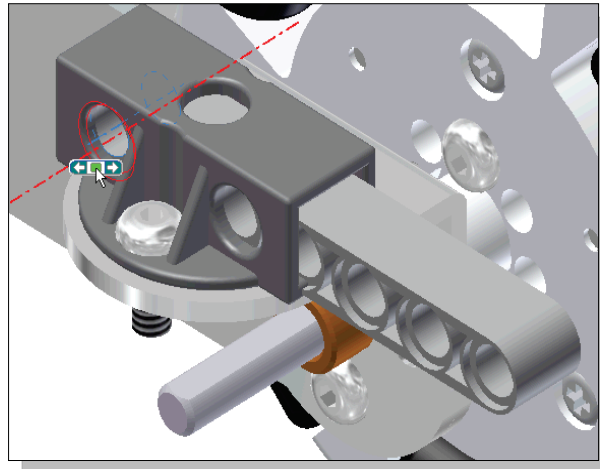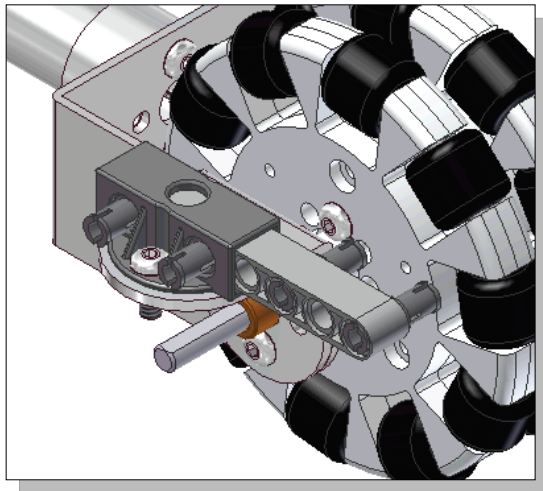
7. Assemble the four ***Peg 1-Module*** parts as shown.

8. Attach the ***Touch Sensor*** and complete the ***ST1*** assembly as shown.

Conclusion

Design includes all activities involved from the original concept to the finished product. Design is the process by which products are created and modified. For many years designers sought ways to describe and anal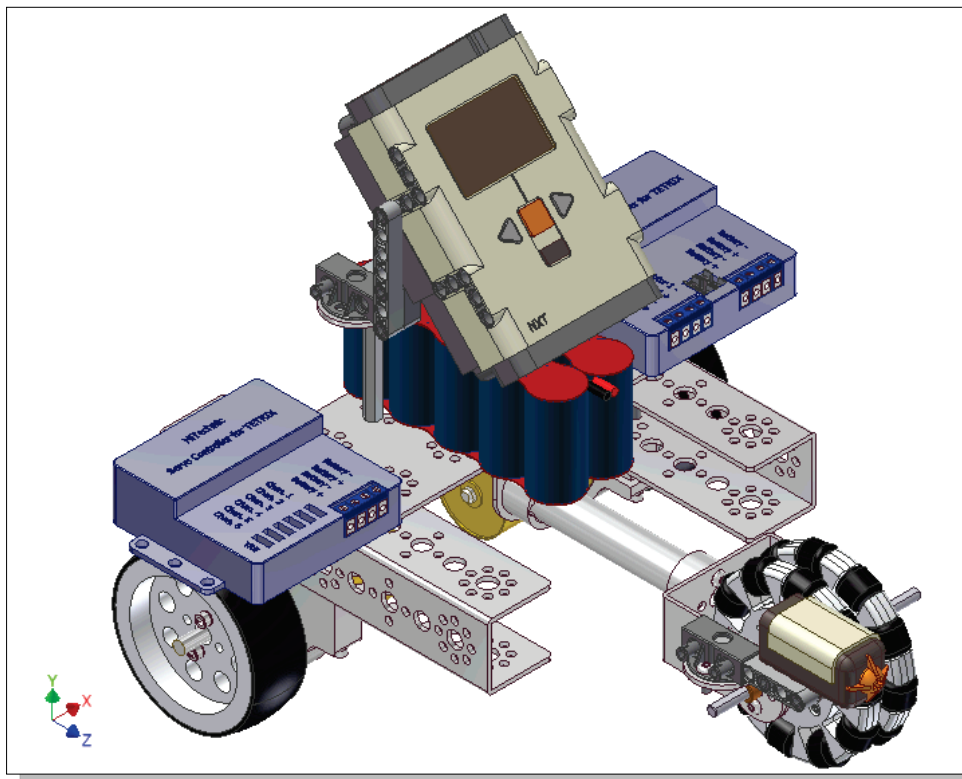yze three-dimensional designs without building physical models. With advancements in computer technology, the creation of parametric models on computers offers a wide range of benefits. Parametric models are easier to interpret and can be easily altered. Parametric models can be analyzed using finite element analysis software, and simulation of real-life loads can be applied to the models and the results graphically displayed.

Throughout this text, various modeling techniques have been presented. Mastering these techniques will enable you to create intelligent and flexible solid models. The goal is to make use of the tools provided by AutoCAD and Autodesk Inventor and to successfully capture the **DESIGN INTENT** of the product. In many instances, only a single approach to the modeling tasks was presented; you are encouraged to repeat all of the lessons and develop different ways of thinking in accomplishing the same tasks. We have only scratched the surface of AutoCAD's and Autodesk Inventor's functionality. The more time you spend using these systems, the easier it will be to perform computer aided modeling with AutoCAD and Autodesk Inventor.