Inventor 2011 Tutorials
iLogic Vessel - Part 2
Head Development

Part 2 of the iLogic Vessel Tutorial will guide you through the steps required in developing an iLogic vessel in Inventor 2011, focusing on Head Development. The objective will be to create heads that can be controlled through simple iLogic code. Once complete, you will understand how to use parameters, dimensions and feature names to create intelligent iLogic components for use in vessel assemblies. The seven head types you will create are

- Conical Head
- Diffuser Head
- Ellipsoidal Head
- Flat Head
- Hemispherical Head
- Torispherical Head (0.1 Dish Radius factor)
- Torispherical Head (0.85 Dish Radius Factor

Create a new part using %ProgramFiles%\Autodesk\Inventor 2011\Templates\Vessels\vessel.ipt as the template. Save the file as Head.ipt.

You can download the files here. Extract the contents to the iLogicTutorial workspace.

#95 Millbank Rd Southwest
Calgary, Alberta · T2Y 2Y9
Phone: 403.238.2758 ·Fax 403.238.8758 ·Cell: 403.612.4438
Website: www.camansol.com · Email: info@camansol.com

Head Parameter Configuration

1. Open the Parameters Dialog box
    a. Add Lip Dimension for Conical and Torispherical Heads
        i. Click *Add Numeric*
        ii. Parameter Name: **LIP**
        iii. Unit/Type: **in**
        iv. Equation: **1**
        v. Key: **Unchecked**
        vi. Export: **Checked**
        vii. Comment: **Lip Dimension**
    b. Add Dish Radius Factor (Torispherical heads)
        i. Click *Add Numeric*
        ii. Parameter Name: **DRFAC**
        iii. Unit/Type: **ul** (*unitless*)
        iv. Equation: **0.85**
        v. Key: **Unchecked**
        vi. Export: **Unchecked**
        vii. Comment: **Dish Radius Calculation Factor**

c. Add Knuckle Radius Factor (Conical and Torispherical heads)

   i. Click *Add Numeric*

   ii. Parameter Name: **KRFAC**

   iii. Unit/Type: **ul** (*unitless*)

   iv. Equation: **0.154**

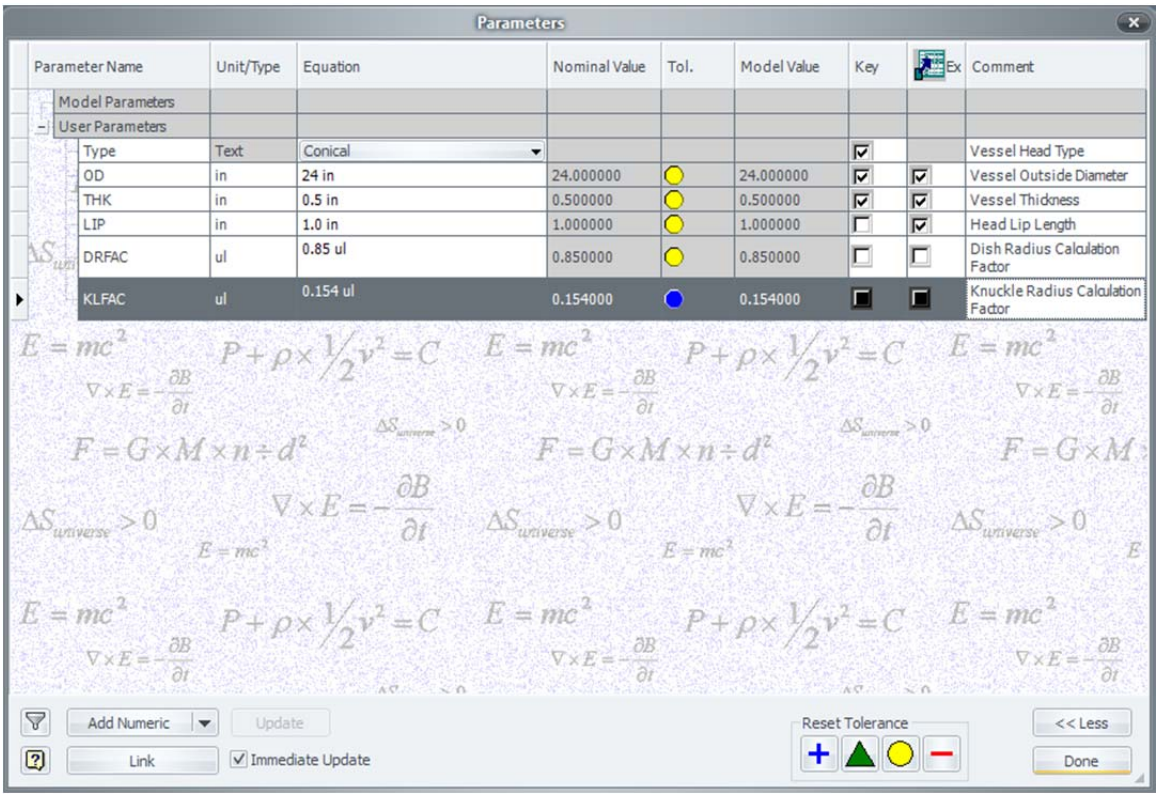
   v. Key: **Unchecked**

   vi. Export: **Unchecked**

   vii. Comment: **Knuckle Radius Calculation Factor**

d. Click *Done*.

Your Parameters Dialog box should look like the one in the diagram.

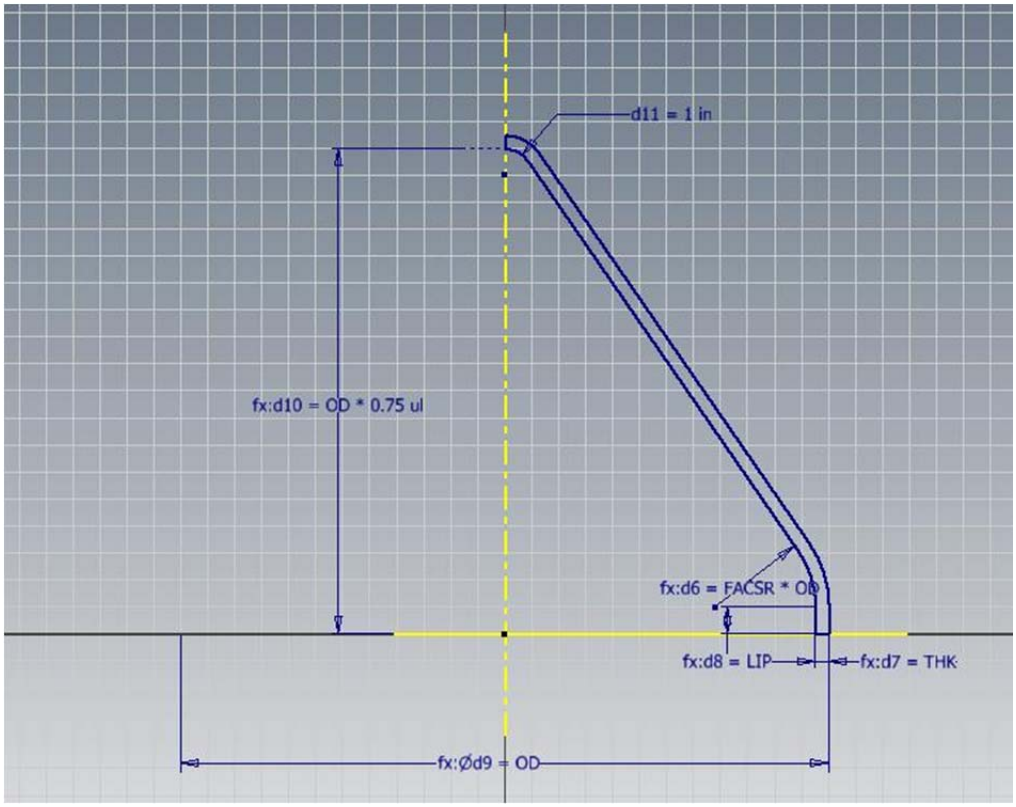


Define Vessel Head Features

The next step is to define the types of heads we placed in the Parameter Table. In the interest of consistency, all sketches should be defined on the same work plane. In this tutorial, all sketches will be placed on the XY Plane, with the X-Axis projected as a regular line, and the Y-Axis projected as a centerline. A collinear constraint will be placed between the weld face of the head and the X-Axis.

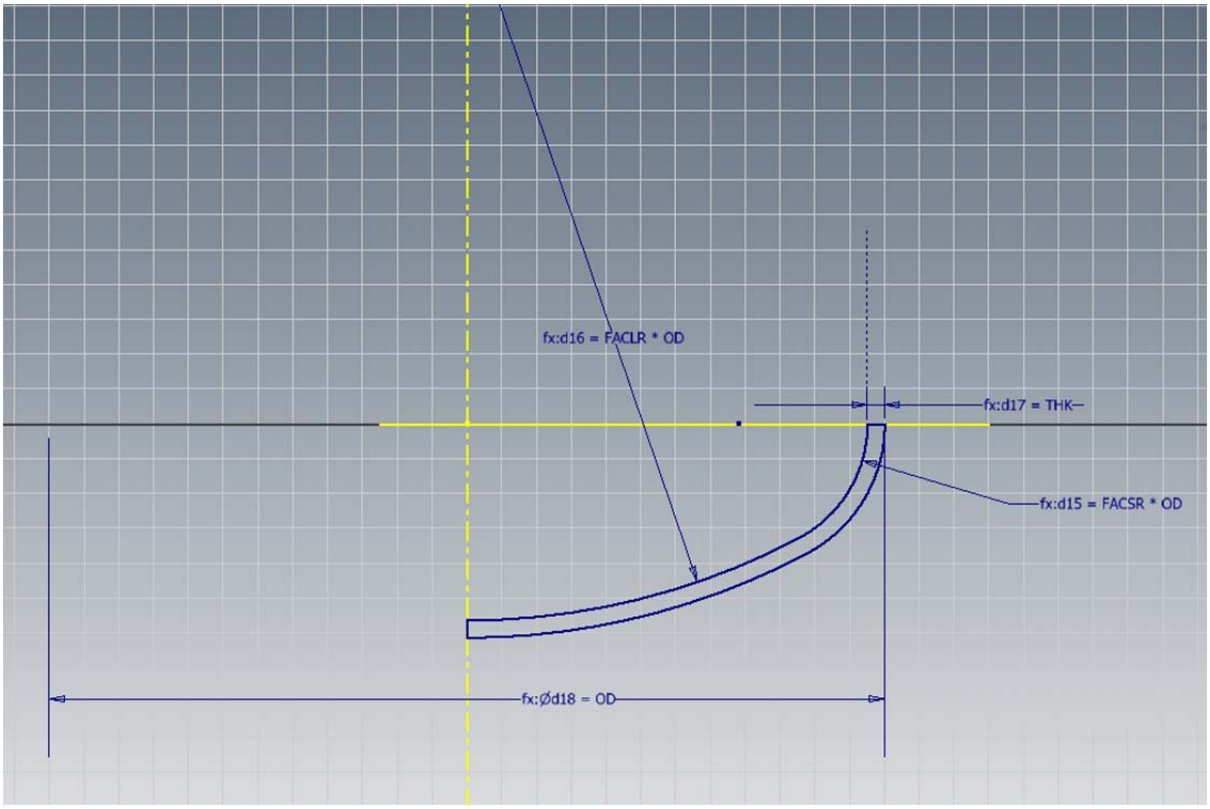
Conical

1. Rename Sketch1 to *skConical*
2. Redefine sketch to the XY Plane
3. Create the following sketch
4. Finish the sketch, and revolve the profile around the Y-Axis
5. Rename the Revolved feature to *Conical*
6. Place two **Opposed Insert** iMates on the weld face of the head
7. Rename them both *Vessel*
8. Suppress the *Conical* Feature



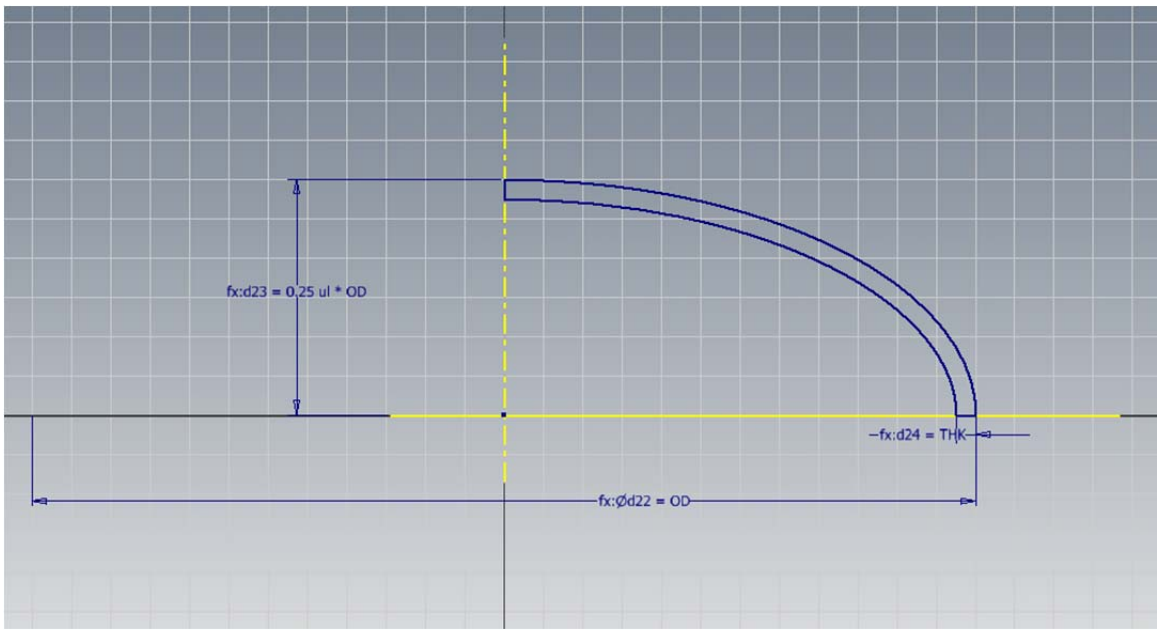
Diffuser

1. Create a new sketch on the XY Plane
2. Rename the sketch to *skDiffuser*
3. Create the following sketch
4. Finish the sketch, and revolve the profile around the Y-Axis
5. Rename the Revolved feature to *Diffuser*
6. Place two **Opposed Insert** iMates on the weld face of the head
7. Rename them both *Vessel*
8. Suppress the *Diffuser* feature.



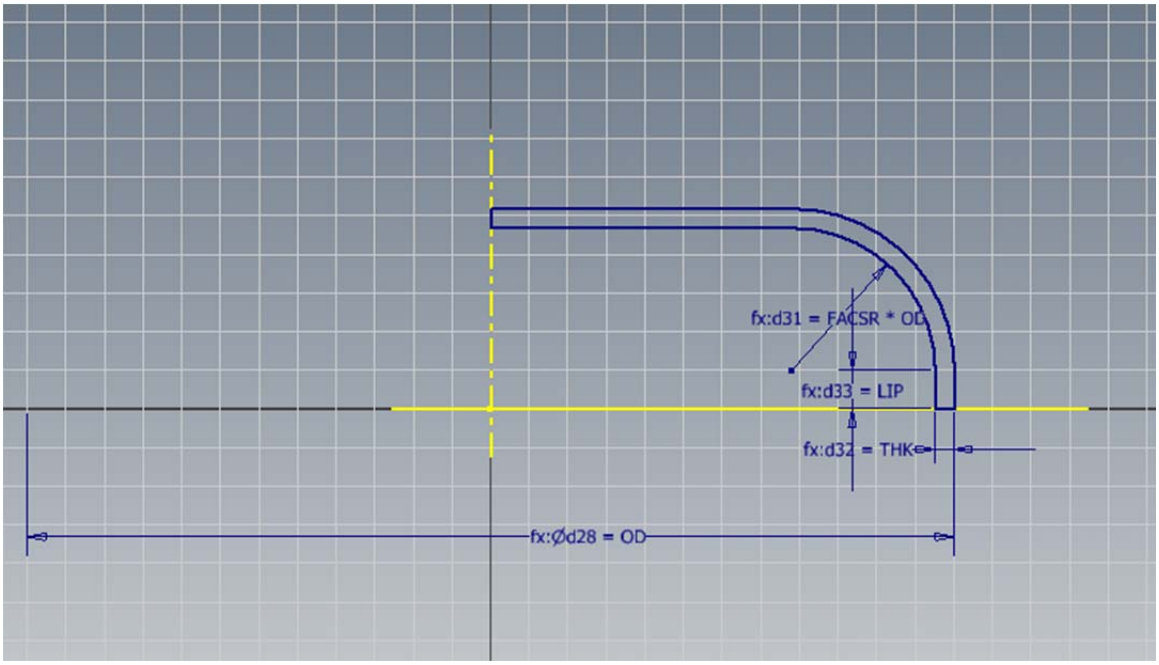
Ellipsoidal

1. Create a new sketch on the XY Plane
2. Rename the sketch to *skEllipsoidal*
3. Create the following sketch
4. Finish the sketch, and revolve the profile around the Y-Axis
5. Rename the Revolved feature to *Ellipsoidal*
6. Place two **Opposed Insert** iMates on the weld face of the head
7. Rename them both *Vessel*
8. Suppress the *Ellipsoidal* feature.



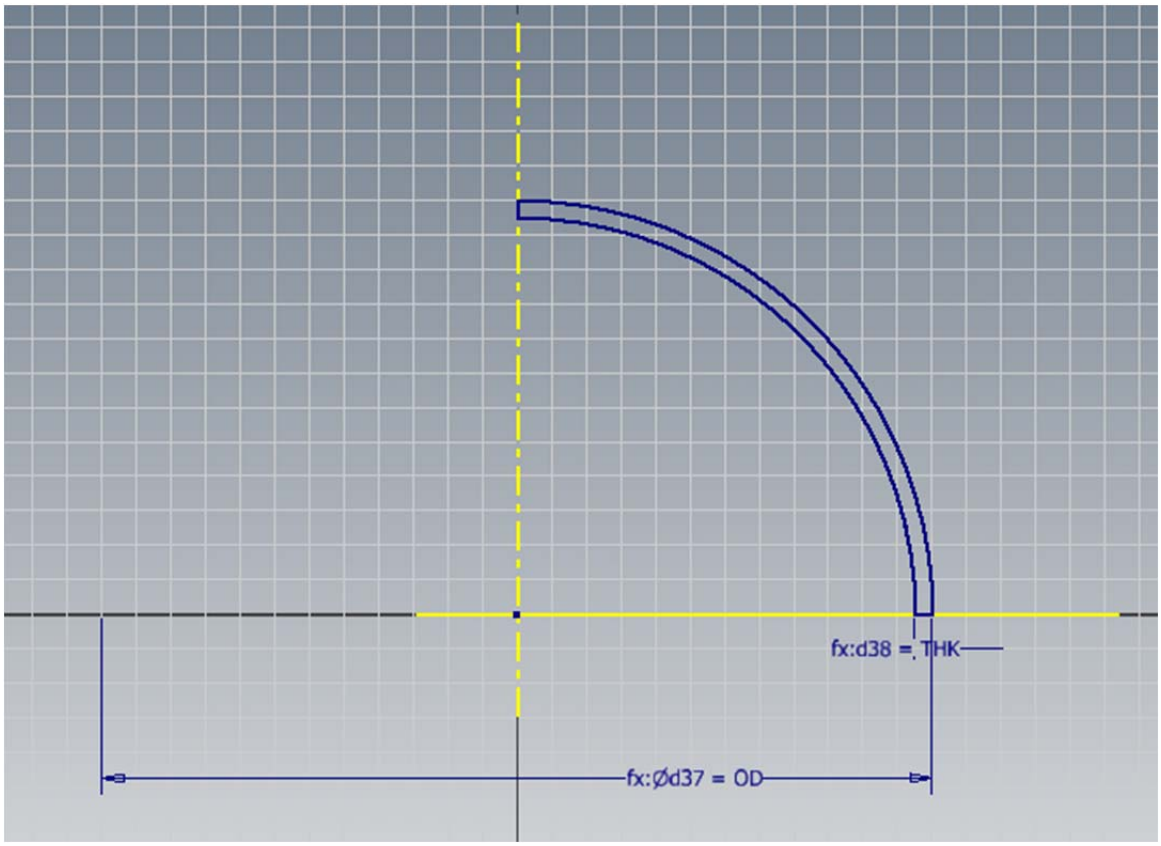
Flat

1. Create a new sketch on the XY Plane
2. Rename the sketch to *skFlat*
3. Create the following sketch
4. Finish the sketch, and revolve the profile around the Y-Axis
5. Rename the Revolved feature to *Flat*
6. Place two **Opposed Insert** iMates on the weld face of the head
7. Rename them both *Vessel*
8. Suppress the *Flat* Feature.



Hemispherical

1. Create a new sketch on the XY Plane
2. Rename the sketch to *skHemispherical*
3. Create the following sketch
4. Finish the sketch, and revolve the profile around the Y-Axis
5. Rename the Revolved feature to *Hemispherical*
6. Place two **Opposed Insert** iMates on the weld face of the head
7. Rename them both *Vessel*
8. Suppress the *Hemispherical* feature



Torispherical

1. Create a new sketch on the XY Plane
2. Rename the sketch to *skTorispherical*
3. Create the following sketch
4. Finish the sketch, and revolve the profile around the Y-Axis
5. Rename the Revolved feature to *Torispherical*
6. Place two **Opposed Insert** iMates on the weld face of the head
7. Rename them both *Vessel*
8. Suppress the *Torispherical* feature