Modeling of a Centrifugal Pump Body using Inventor 2014

- Spiral construction -

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This tutorial consists of one lesson that will provide you with a basic understanding of modeling special parts - like this spiral construction -, familiarizing you with multi-body concept, and introduce you to achieving advanced abilities in modeling parts.

Audience: Users accommodated with basic modeling in Inventor 2014

Prerequisites: Basic Autodesk Inventor 2014 knowledge

Time to complete: 20 - 30 minutes

Objectives

In this tutorial, you learn how to model a special part including the following:

- Loft feature with Spiral centerline
- Multi-body concept

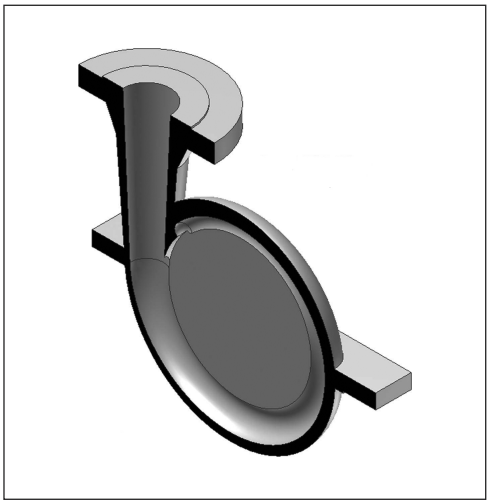
Tutorial Files

No files necessary

Modeling the Centrifugal Pump Body

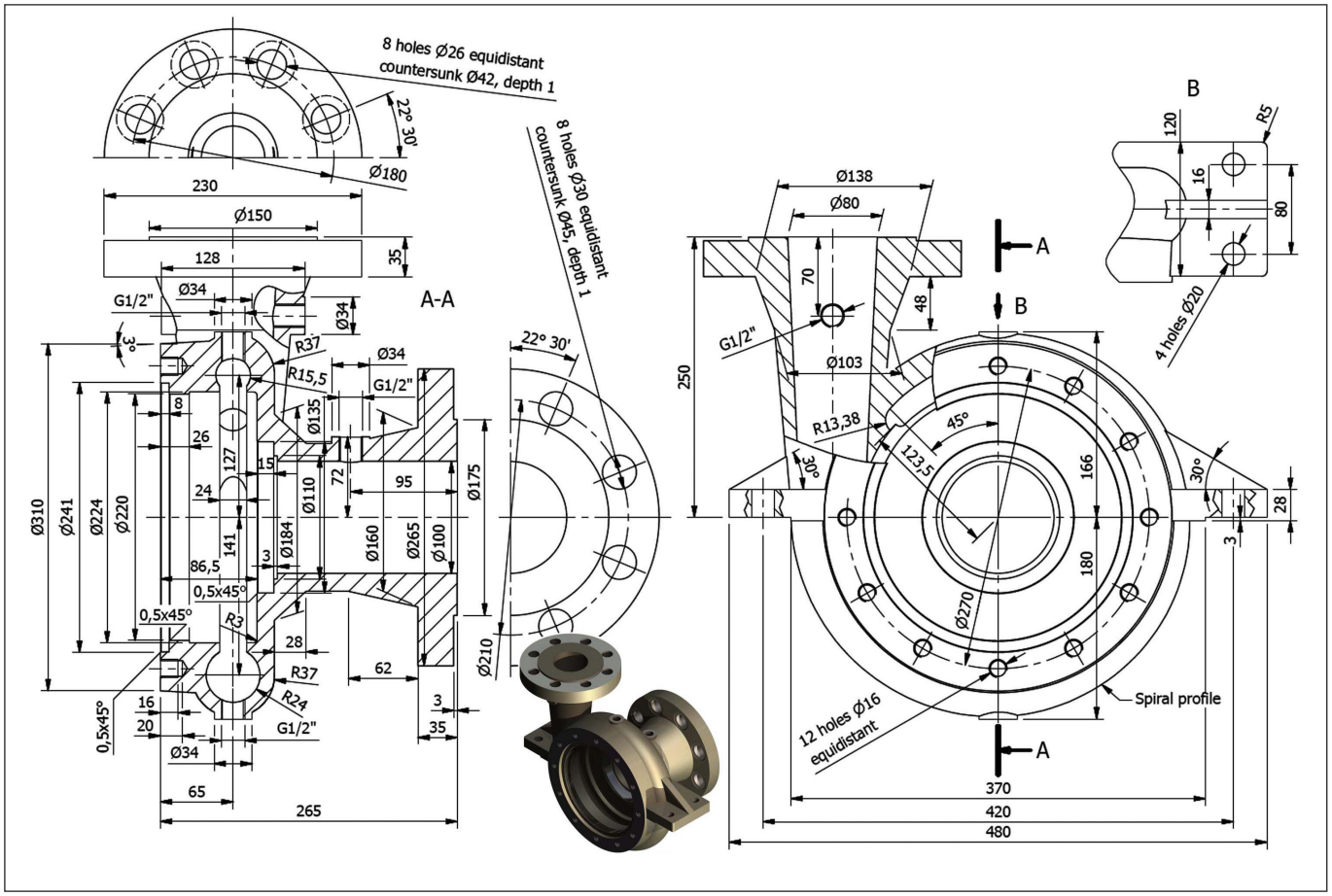
Part Analysis: A half-plane section of the main part body is presented in the image. The special construction consists of its internal and external „circular" body, which is actually a spiral one. Trying to use the **Loft** feature for creating this feature must include a *spiral* centerline, after selecting the normal sections represented by 2D sketches.

Another problem is that the internal (void) body seams to be difficult to model when the external (consistent) body is present. To solve this particular problem, the proposed approach is to begin by modeling the internal body as a first solid, then modeling the external body, and ending with subtracting the internal solid from the external one. The solution is generally welcome for parts with complicated internal void features.



Technical Drawing of the part

We have to abide all conditions from the technical drawing.

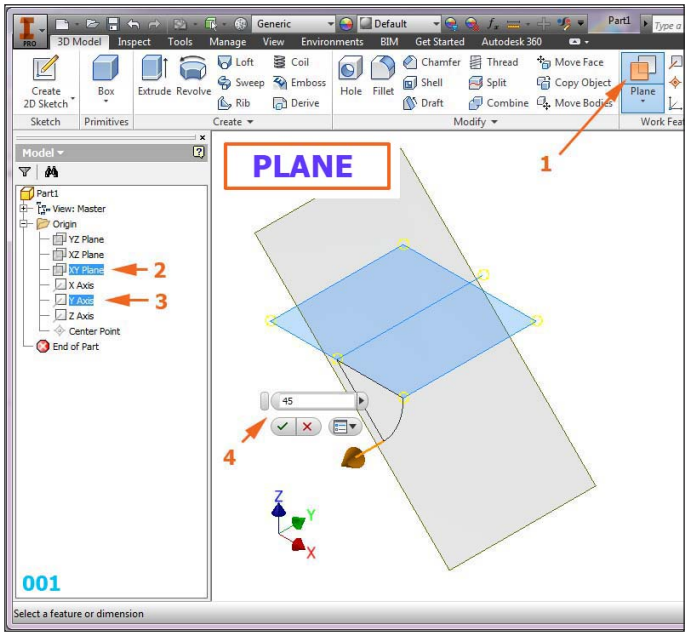


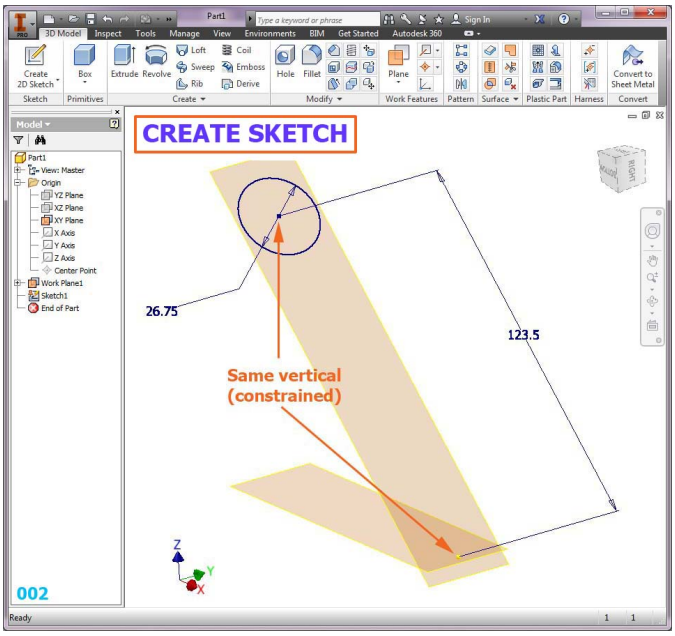
Procedure

The following 37 pictures illustrate the whole procedure. When a picture has a CAPITAL LABEL surrounded by a red border with a name of an Inventor tool, you have to prior activate that tool.

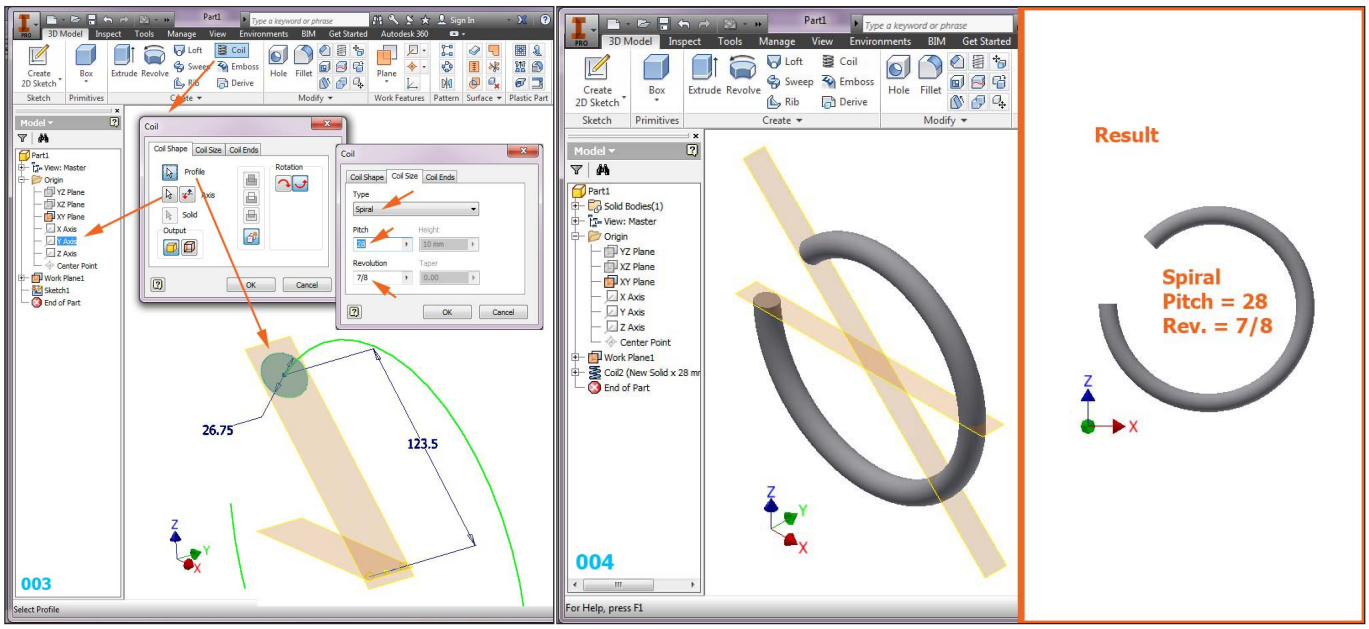
001. New *work plane* at 45° from *XZ Plane* around the *Y Axis* (see 1, 2, 3, 4 steps)

002. New sketch: Circle Ø26.75 with center at 123.5 on the same vertical with the origin *(Center Point)*

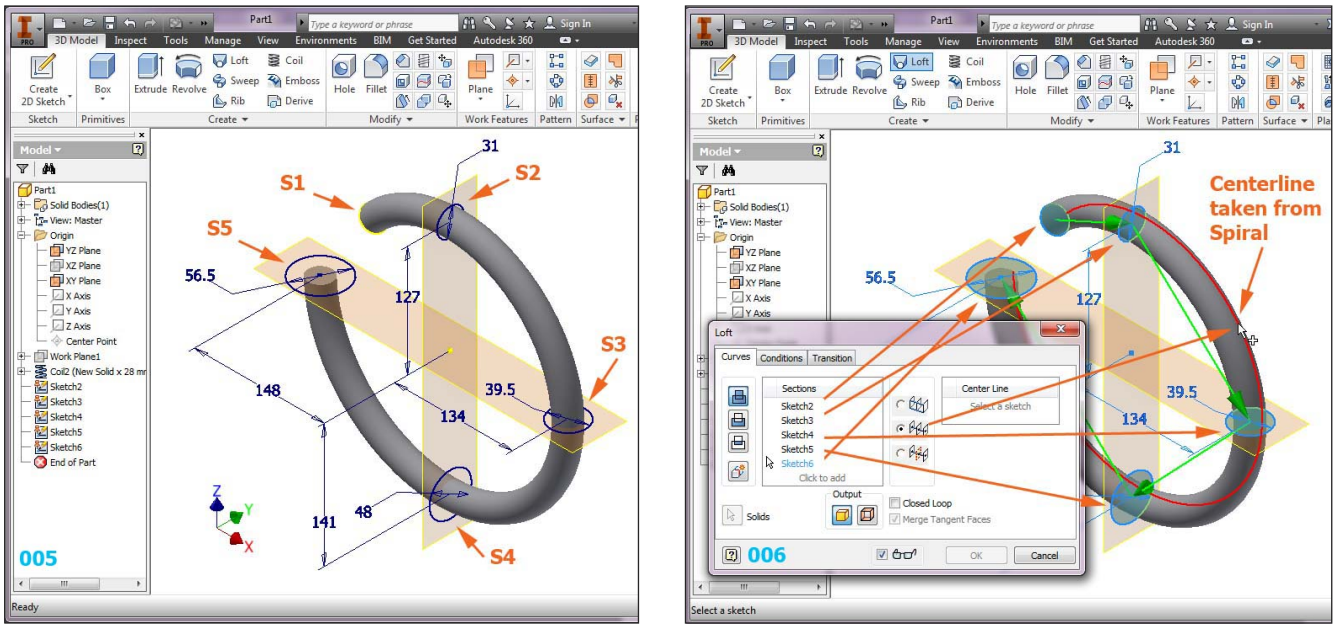




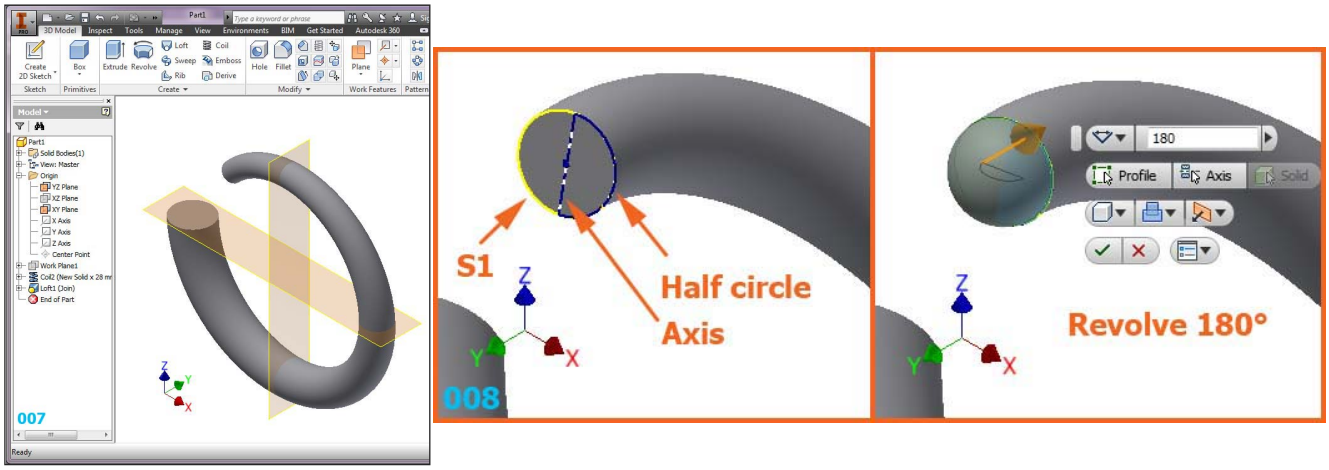
003. **Coil** tool with *Spiral* option (Pitch=28, Revolution=7/8)
004. Result from **Coil**/*Spiral* applying



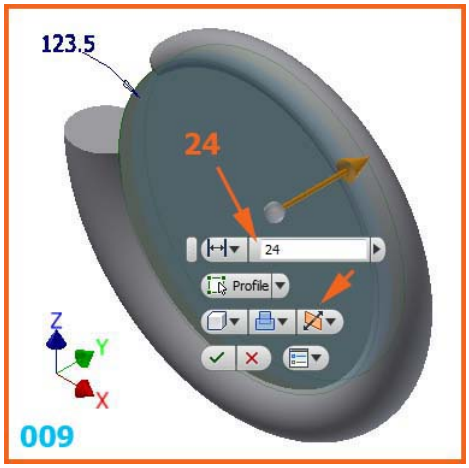
005. Creating of 5 sketches (5 circles), where the last 4 of them are related to the *Center point* only
006. **Loft** tool on 5 sketches, using the spiral curve on the spiral feature as *Centerline*

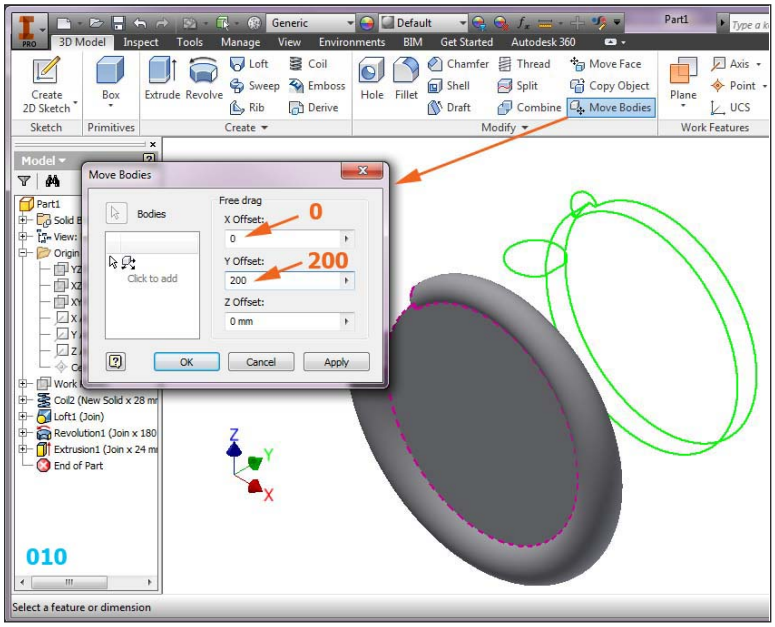


007. Result from **Loft** tool applying
008. Revolving half circle on S1 around its diameter on 180° to create a hemisphere

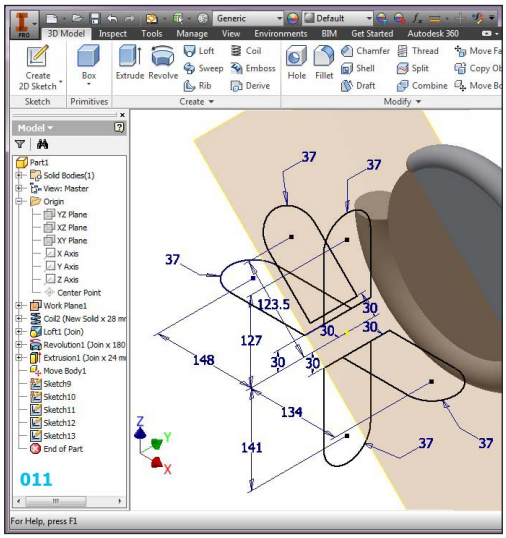


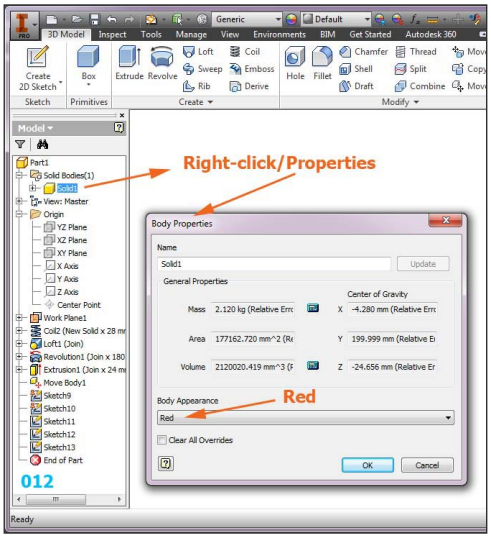
009. **Extrude** tool on a circle (radius=123.5) in the *Center point* using a 24 simmetrical distance
010. **Move Bodies** tool to move the (unique) solid 200 to the back



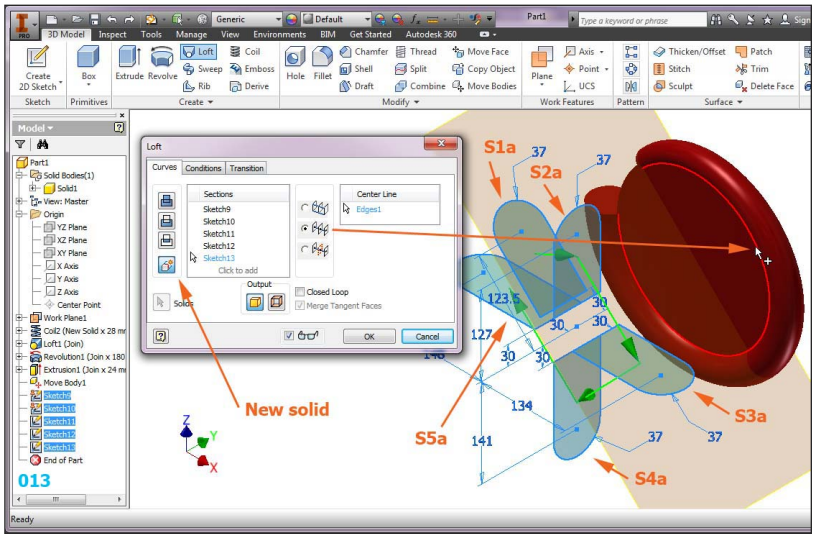


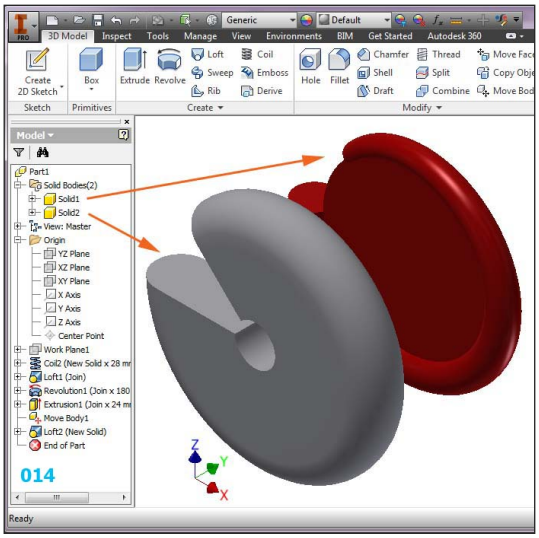
011. New 5 similar sketches (only one dimension differs), related to the same *Center point* as above
012. Changing color of the *Solid1* using *Right click/Properties*



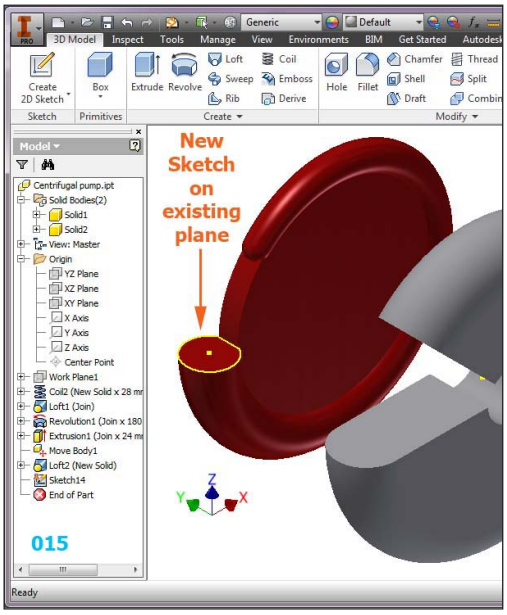


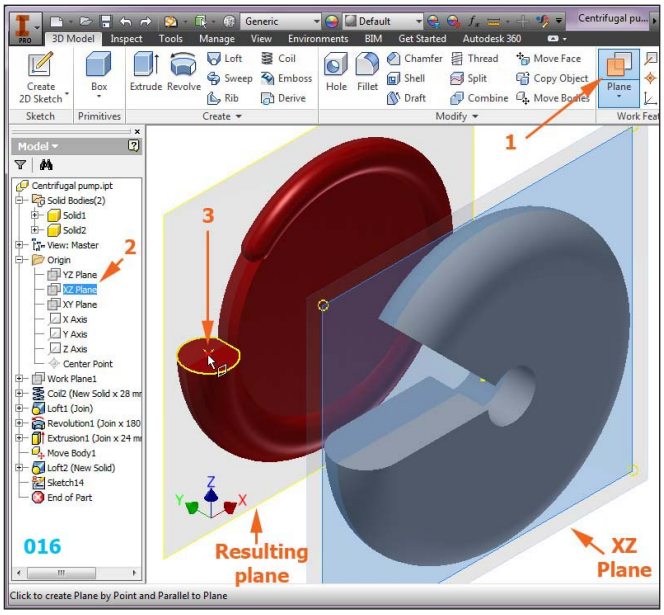
013. **Loft** tool with *New solid* option, on the 5 sketches with *Centerline* from the spiral curve
014. The result is another solid in the same `.ipt` file (or two solids of different colors)



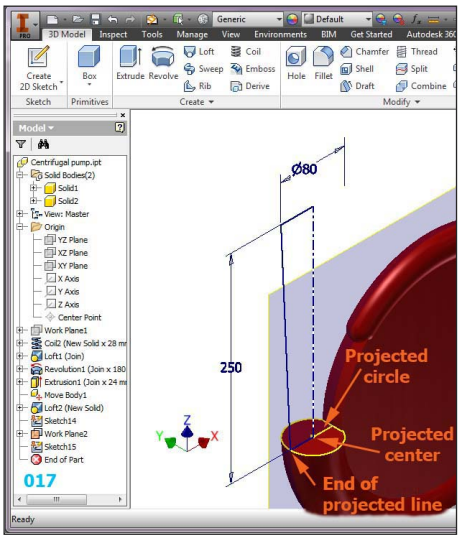


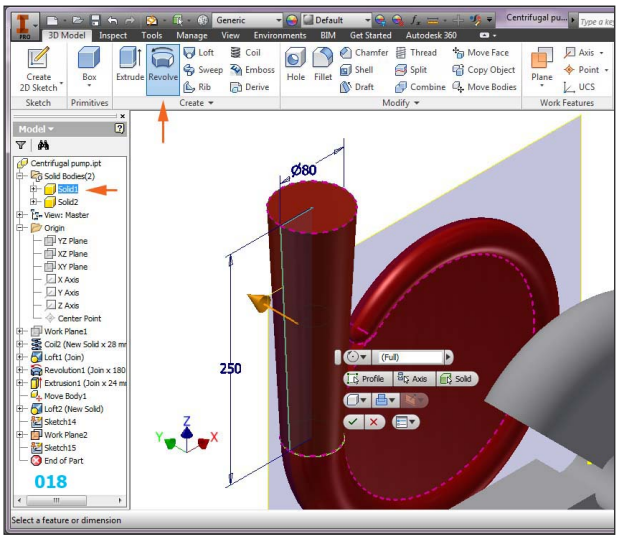
015. New sketch on an existing plane - a circle is automatically projected (like in 005 or 008 picture)
016. New work plane parallel with *XZ Plane* through the circle center (see 1, 2, 3 steps)



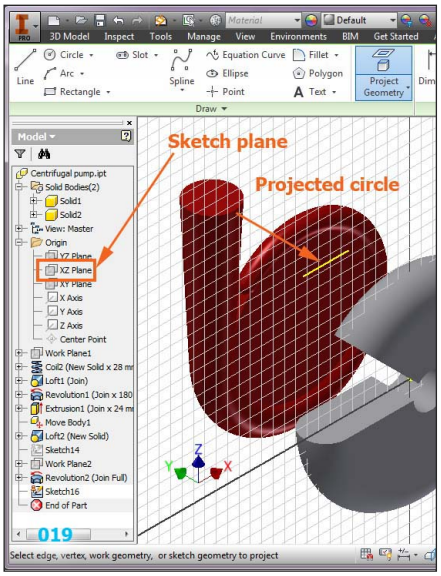


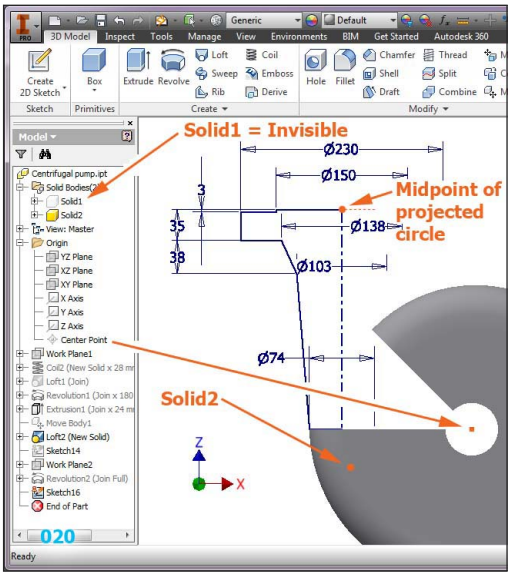
017. New sketch on *XZ Plane* - a centerline from the projected circle center, and other 3 lines
018. Revolving the closed loop to obtain a conic feature associated with the *Solid1*



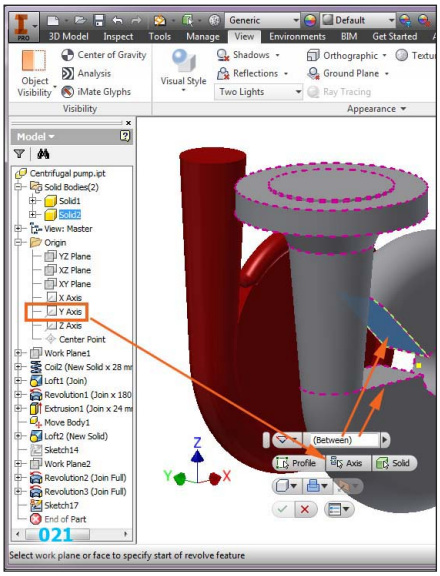


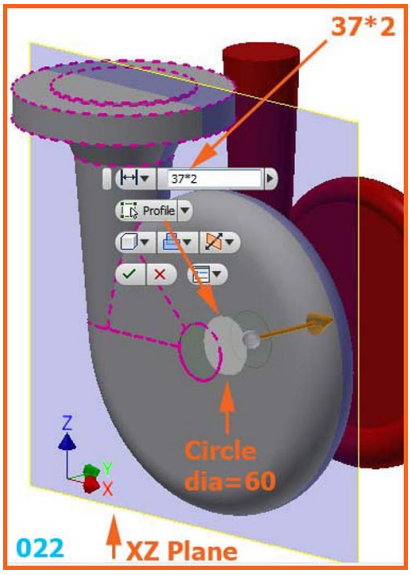
019. New sketch for the *Solid2* with a line projected from the circle of the *Solid1*
020. The sketch also related to the *Center point* will be revolved - attached to the *Solid2*



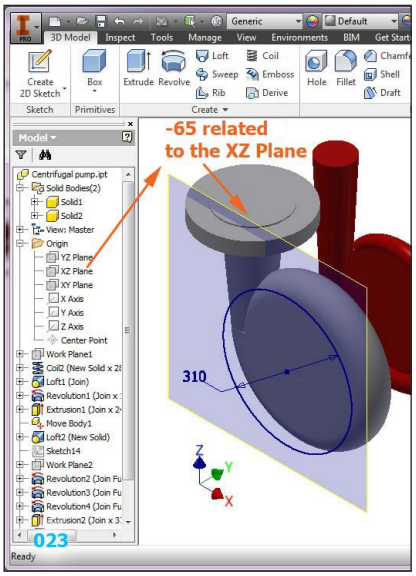


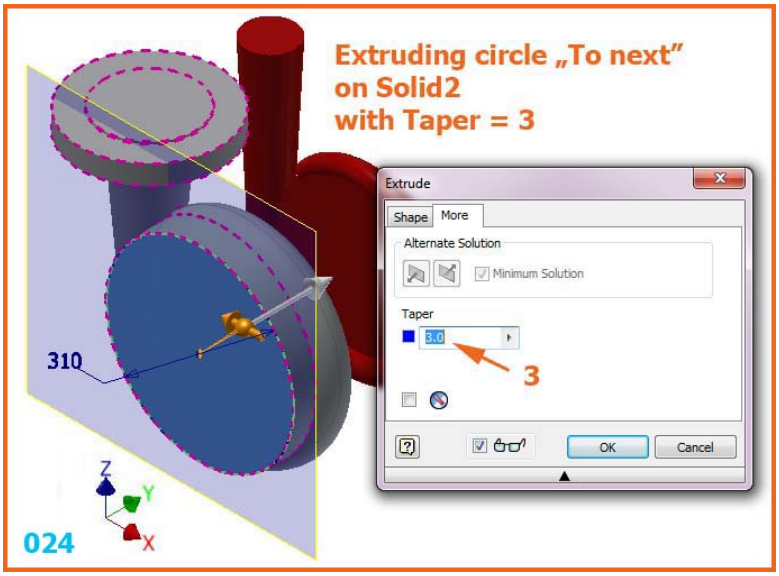
021. A sketch created on the sloped plane is revolved between it and the next one, to fill the gap
022. Filling the central hole of the *Solid2*



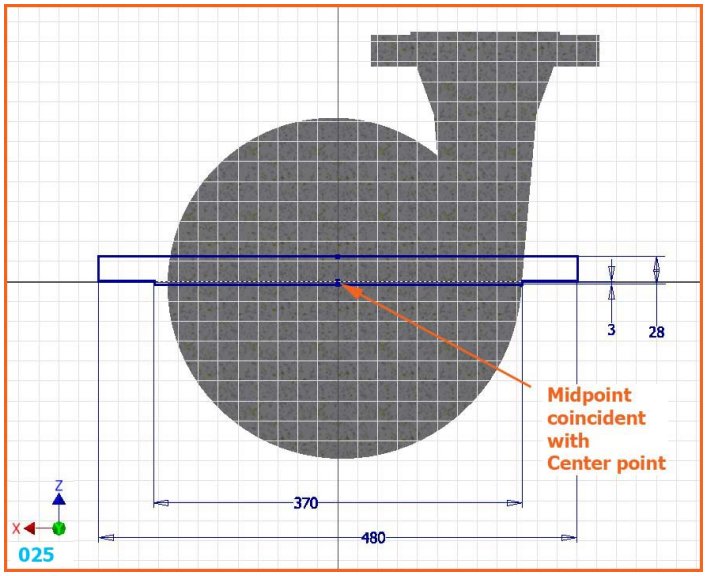


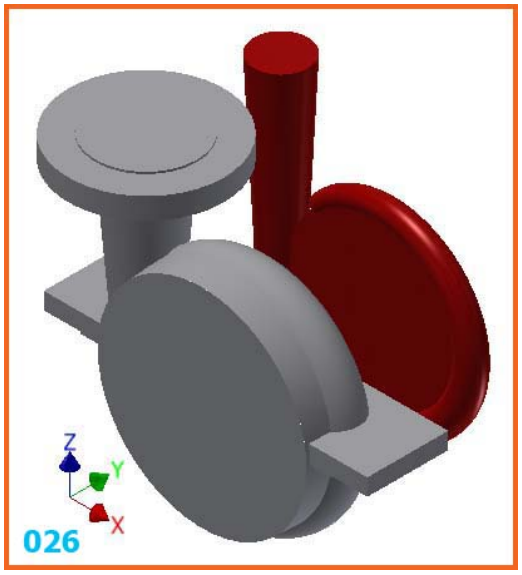
023. New sketch with a circle of Ø310 in a new work plane at -65 from the *XZ Plane*
024. Extruding the circle *To next* with a taper of 3°, as in the above part drawing





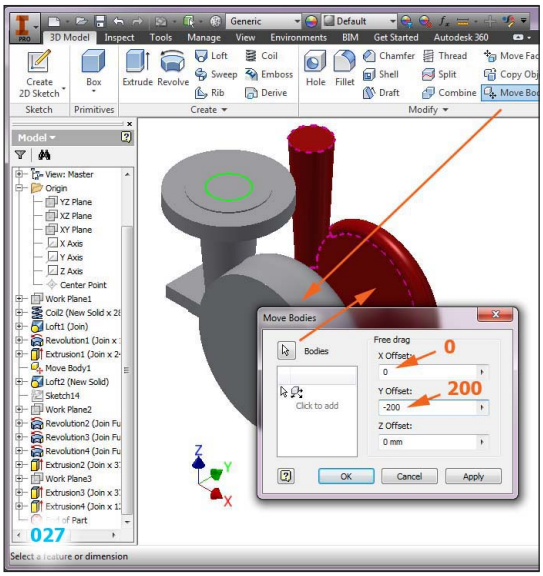
025. A new sketch in the plane XZ to extrude simmetrically on 120
026. The result of the last two extrudes

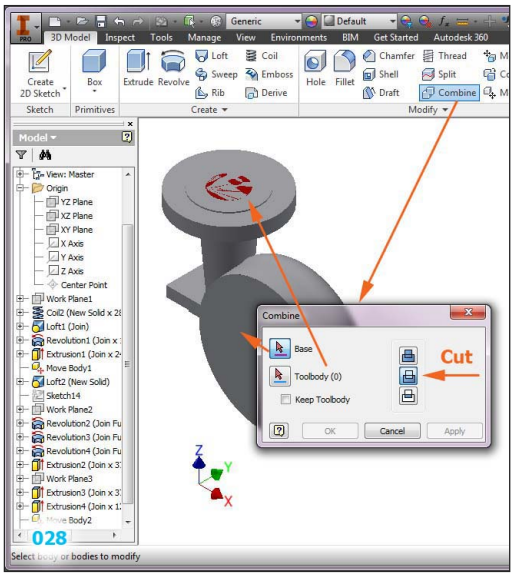




027. **Move Bodies** tool on *Y Offset* = 200 to get back the *Solid1*

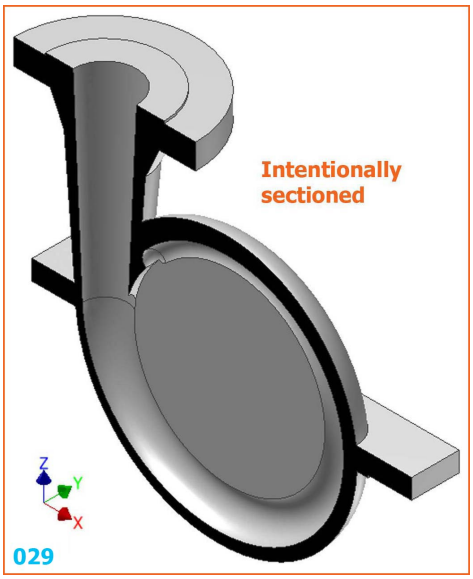
028. **Combine** tool to subtract *(Cut)* the *Solid1* (as *Base)* from *Solid2* (as *Toolbody)*

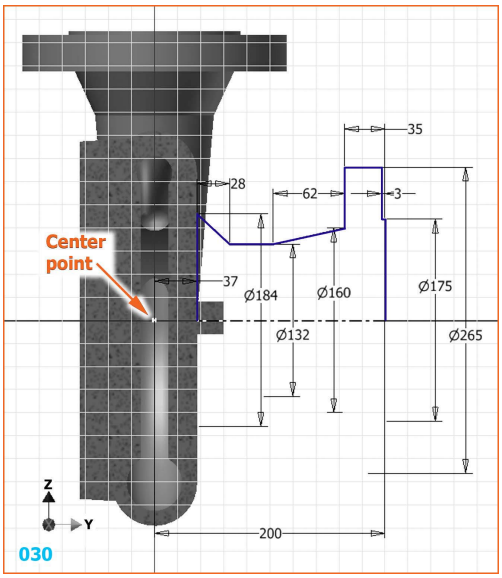




029. The result (intentionally sectioned)

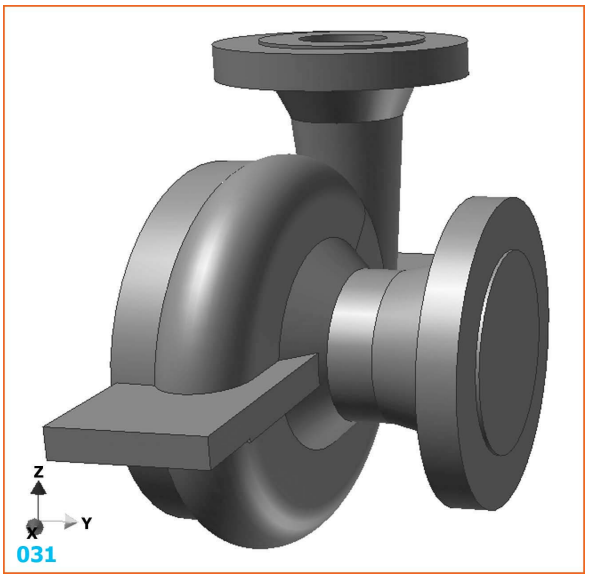
030. A new sketch for creating the revolved external body

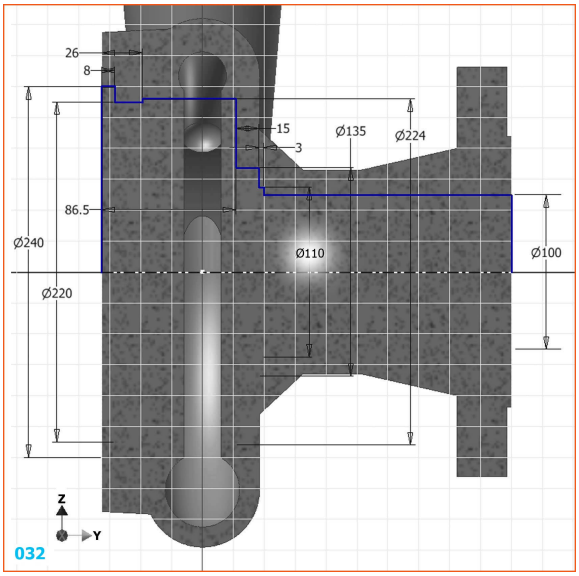




031. Result of **Revolution**

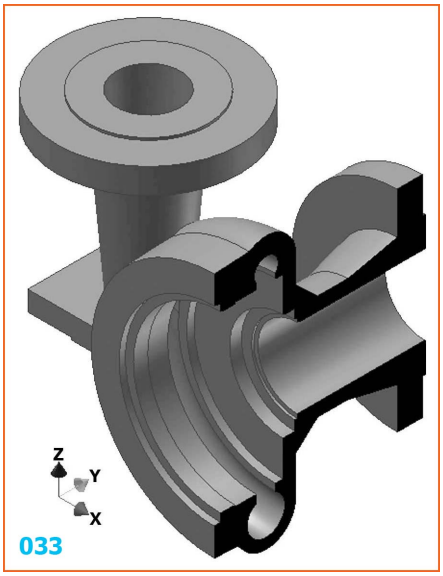
032. A new sketch for creating the revolved internal body

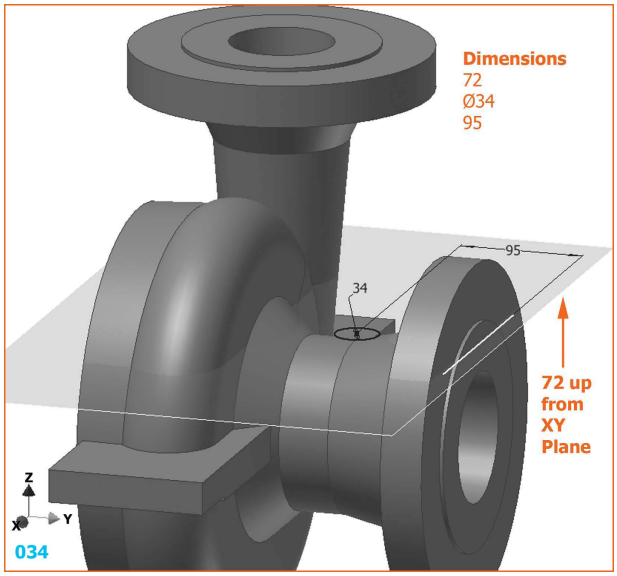




033. The result (intentionally sectioned)

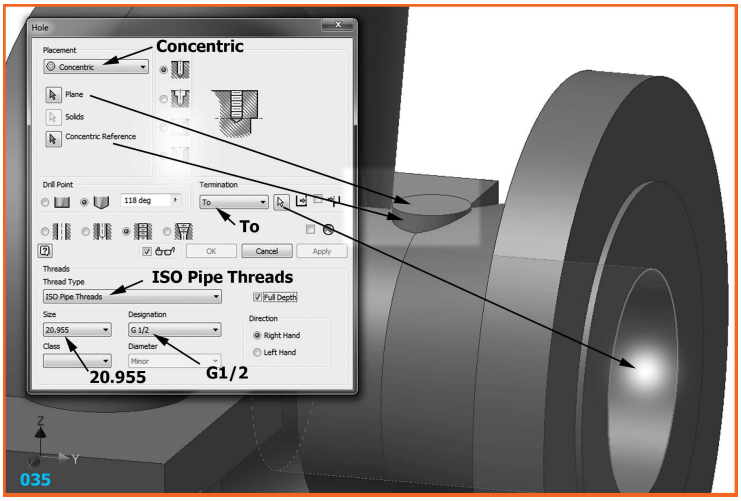
034. New sketch with a Ø34 circle to create a boss for a *G1/2"* hole

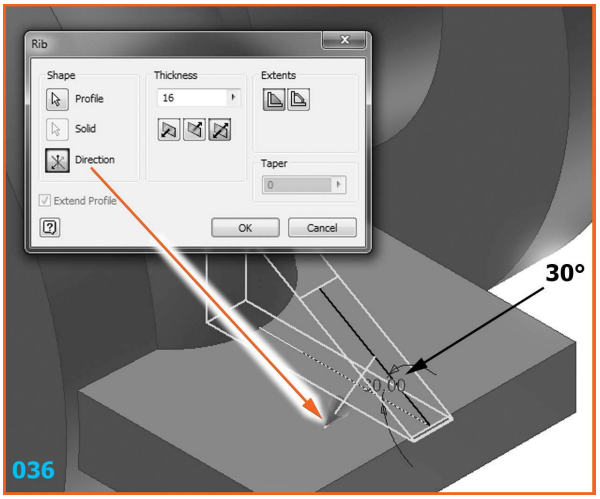




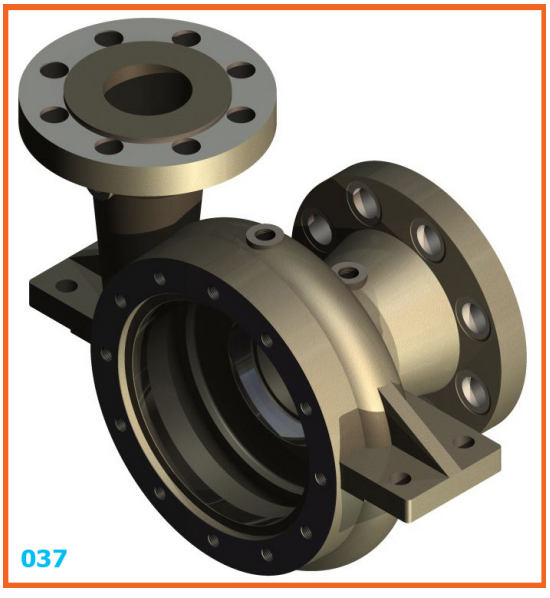
035. The *ISO Pipe G1/2"* hole, *through* but not *through all*

036. **Rib** tool to create a rib, starting from a sketch in XZ Plane with a sloped line at 30°





037. Final part



End of lesson

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