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THE DESIGN OF AN KAPLAN TURBINE RUNNER USING AUTODESK INVENTOR

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Abstract: *The runner of the hydraulic turbine is main element of the energy transformation, with the function to convert the water energy to mechanical power at the machine axis. The main parts of the runner are, figure 1: the hub, the blades and the mechanical control system. The blades can rotate during the service period, assuring, together with the wicked gate, the double adjustment of the machine to obtain optimal regimes. The paper described the computer aided design process of the runner using Autodesk Inventor [1], [2], [3].*

Keywords*: CAD, Kaplan, runner, Autodesk Inventor.*

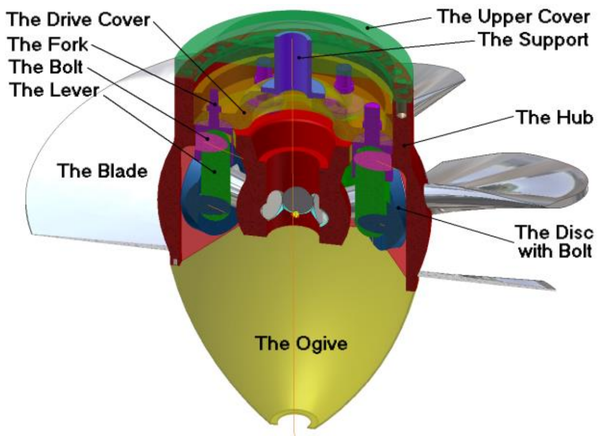

Fig. 1 The Main Parts of the Runner

1. The Hub

The hub shares a spherical shape with the blade to assure the blade's rotation. Based on the hub sketch, figure 2, with **Revolve** command, is generated the solid representation of the hub, figure 3.

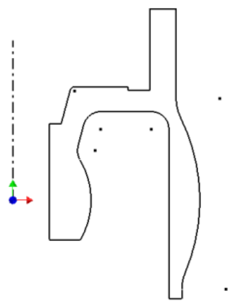

Fig. 2 The Hub Sketch

Fig. 3 The Hub

Next, in the hub must be generated the multiple holes, figure 4. The command **Hole** will be used, specifying the hole and depth diameter and the precise location of center's holes.

With **Extrude** command option **Cut** will be generated the hexagonal shape on the inner side of the hub, figure 5. The generated plane surface will assure the plane contact between the inner side of hub and the disc with bolt.

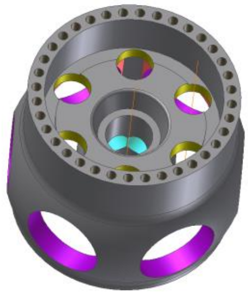

Fig. 4 The Hub's Holes

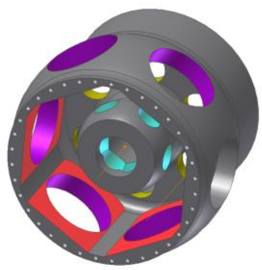

Fig. 5 The inner side view of the Hub

2. The Blade

The blade has a complex geometry, figure 6. The blade's number is imposed by constructive and functional criteria. The CAD modeling process of the blade was detailed in [4].

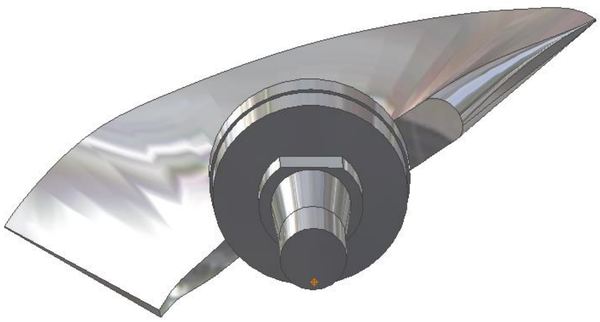

Fig. 6 The Blade

3. The Upper Cover

The upper cover has a symmetrical shape. Based on the closed profile, with **Revolve** command, is generated the solid representation of the upper cover, figure 7.

4. The Drive Cover

The drive cover has a symmetrical shape. Based on the closed profile, with **Revolve** command, is generated the solid representation of the drive cover, figure 8. Also, the **Hole** command will be used to generate holes for the forks.

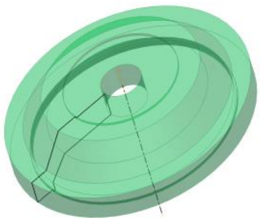

Fig. 7 The Upper Cover

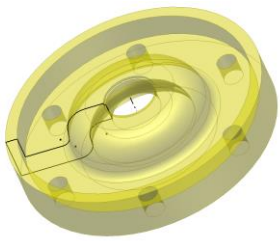

Fig. 8 The Drive Cover

5. The Support

The support has a symmetrical shape. Based on the closed profile, with **Revolve** command, is generated the solid representation of the support, figure 9.

6. The Bolt

The bolt has a cylindrical shape, with rounded shape at the ends. With command **Extrude** the circle will be extruded on **Extrude Direction 1,** to obtain the solid bolt. With the same command **Extrude** option **Intersection**, the second circle will be extruded on **Extrude Direction 2**, figure 10.

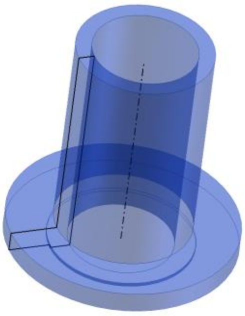

Fig. 9 The Support

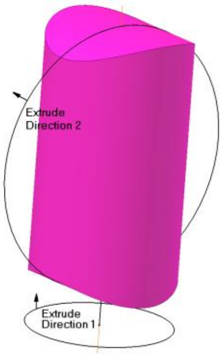

Fig. 10 The Bolt

7. The Ogive

The ogive has a symmetrical shape. Based on the closed profile, with **Revolve** command, is generated the solid representation of the ogive, figure 11.

8. The Fork

The fork has a more complex shape, figure 12. With command **Extrude** the base circle will be extruded on extrude direction **1,** to obtain the full cylindrical part of the fork. With command **Extrude** option **Cut**, the vertical circle and the closed profile will be extruded on directions **2** respectively **3,** to extract material from the cylindrical part. With command **Extrude** the circles will be extruded on direction **4, 5,** and **6** to obtain the rest of the fork's cylindrical parts.

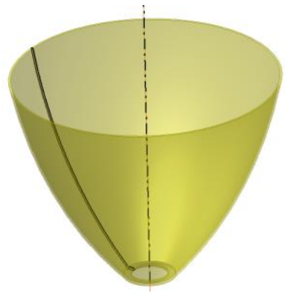

Fig. 11 The Ogive

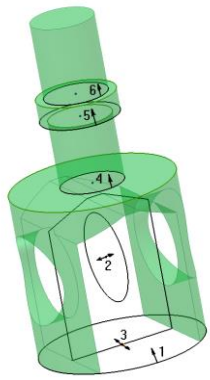

Fig. 12 The Fork

9. The Lever

The lever is presented in figure 13. In the step 1, the command **Revolve** will rotate 180^o the semiprofile around the central axe, obtaining the upper part of the lever. With command **Extrude** the complete profile will be extruded on direction **2,** to obtain the rest part of the lever.

With command **Extrude** option **Cut**, the vertical circle will be extruded on direction **3,** to extract material from the lever and create the hole. Finally, with command **Mirror Feature**, the upper side of the lever will be mirrored to obtain the symmetrical lower side.

10. The Disc with Bolt

The disc with bolt is presented in figure 14. With command **Extrude** the region between the exterior circle and inner closed contour will be extruded on **1** direction to generate the disc thickness. Next, the upper circle will be extruded on **2** direction to generate the bolt. With command **Extrude** option **Cut**, the triangular closed profile will be extruded on direction **3,** to extract material from the disc and create the bevel plane of the disc. Finally, in the step **4**, with command **Chamfer**, will be create the lower chamfer of the disc.

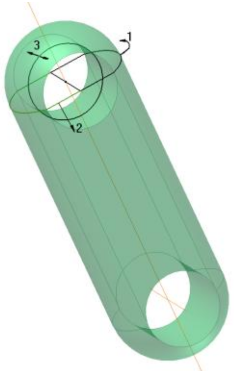

Fig. 13 The Lever

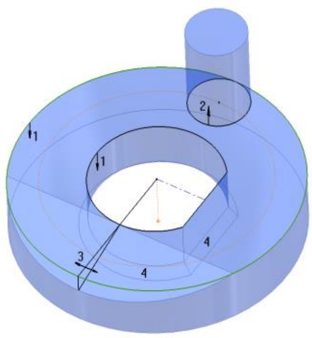

Fig. 14 The Disc with Bolt

11. The Assembly Runner

The assembly runner is presented in figure 15. To begin creating the runner assembly, will be choose the hub as the first component. When the first component is placed in the assembly, its origin is coincident and aligned with the assembly coordinate origin. All degrees of freedom are removed from the first component. The components will be placed in the assembly with command **Place Component**.

Assembly constraints determine how components in the assembly will fit together. During applying constraints steps, will be removed degrees of freedom from the components, restricting the ways that those can move.

The **Place Constraints** command creates constraints to control position and animation. Motion constraints do not affect position constraints. The **Assembly** tab has the following constraints to control the component’s position:

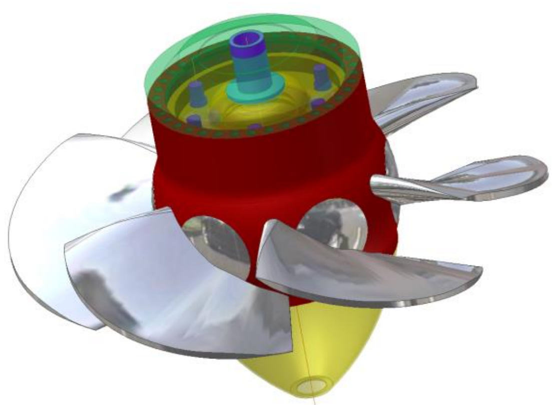

Fig. 15 The Runner Assembly

- a **mate** constraint positions selected faces parallel to one another, with faces coincident or aligns parts adjacent to one another with faces flush; the faces may be offset from one another;
- an **angle** constraint positions linear or planar faces on two components at a specified angle;
- a **tangent** constraint between planes, cylinders, spheres, cones, and ruled splines causes geometry to contact at the point of tangency;
- an **insert** constraint positions cylindrical features with planar faces perpendicular to the cylinder axis.

The **Motion** tab has the following constraints to specify intended motion ratios between assembly components:

- a rotation constraint specifies rotation of one part relative to another part using a specified ratio;
- a rotation-translation constraint specifies rotation of one part relative to translation of a second part.

Therefore, the assembly is creating by placing components in the file, applying constraints between components to fix the spatial position of every component in the context of the assembly.

The **Drive Constraint** tool is used to simulate mechanical motion by driving a constraint through a sequence of steps. This tool is limited to one constraint, but it is possible to drive additional constraints by using the **Equations** tool to create algebraic relationships between constraints. So, it is possible to study the motion of the mechanical control system of the runner and detect potential problems.

12. The Assembly Runner Drawing

A drawing is a tool to communicate engineering design to manufacturing, purchasing, service, and others. A drawing can include base views, auxiliary, detail, sections and projected views, dimensions and annotations. To create views some few click mouse are needed. Figure 16 show a section through the runner assembly.

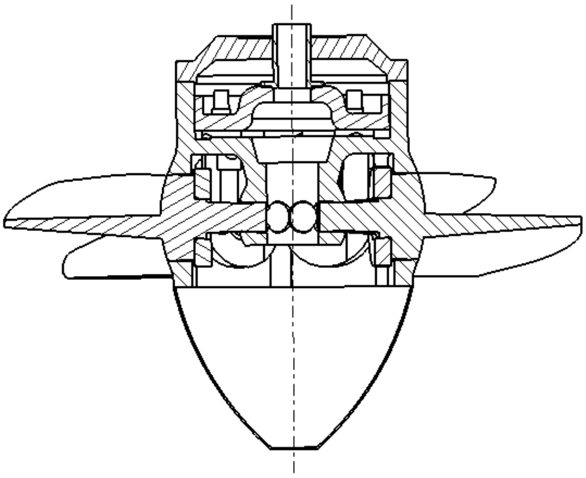

Fig. 16 The assembly runner section

13. The Runner Assembly Phases

Autodesk Inventor use presentation technique, figure 17, to develop exploded views, animations and other stylized views of an assembly to help document the design.

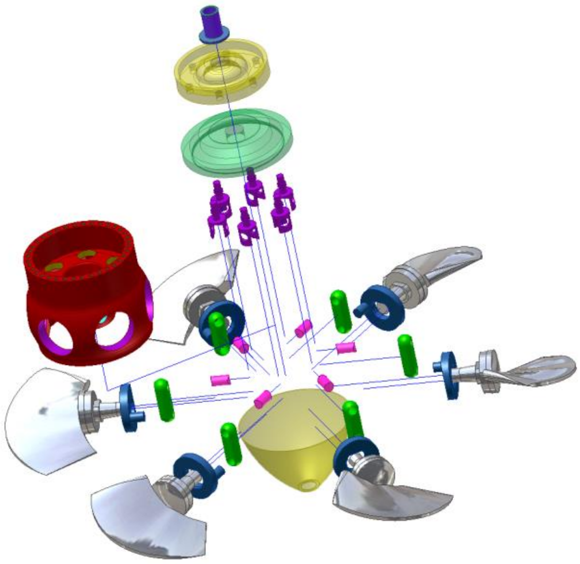

Fig. 17 The Runner Assembly Phases

14. Conclusions

Autodesk Inventor is a powerful tool to design complex parts and assembly. In a classical design manner, the engineer create the final drawing with views, sections and details, and the part or assembly is only view in the designer mind or the drawing reader. In Inventor, the process is reversed: first will be generate 3D parts and assembly as a computer representation of the real object, and on this base will be create the final drawings. The sheet metal, weldments, presentations and motion study possibilities are supplementary tools offered by Autodesk Inventor.

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PROIECTAREA ROTORULUI TURBINEI KAPLAN UTILIZÂND AUTODESK INVENTOR

Rezumat

Rotorul turbinelor hidraulice reprezintă sediul transformării energetice a turbinei, având rolul de conversie a energiei apei în putere respectiv lucru mecanic la arborele maşinii. Rotorul acestor turbine este compus din: corpul butucului, paletele rotorice şi mecanismul de reglare. Paletele se pot roti în timpul funcţionării, ceea ce asigură, împreună cu aparatul director, o reglare dublă a turbinei şi funcţionarea acesteia la regimuri optimizate. Lucrarea prezintă procesul de proiectare a rotorului, utilizând programul de proiectarea asistată de calculator Autodesk Inventor.

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