

CAD MODELLING OF ACTIVE SURFACES IN DESIGN OF DIES

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ABSTRACT

Contribution deals with procedures of CAD creating of active surfaces of dies in bulk forming. In order to design active surfaces, shapes of forging dies have been solid modelled applying their design features as defined mould joint, chamfers and radii. Two methods of CAD modelling of active surfaces are presented as mould cavity mode and assembly mode which is referred to as Merge-Cut Out. Both methods are presented consecutively and their advantages are being compared by ways of cavities' design and applied model tree, as well.

Keywords: CAD, active surface, forging die, mould cavity mode, merge-cut out

1. Introduction

Die forging technology represents wide variety of shape-changing operations to produce solid and hollow semi-finished products, i.e., process of forging, pressing, stamping etc. Semi-finished products resulting from these processes represent variety of either simple or complicated components, therefore, proper design of tooling meets requirements to spare time and costs of design. Most significant attribute of dies' design is that they are characterised by their singularity – uniqueness of design solution. In other words said, any forging allows applying different forging plans. Therefore, needs for different dies' design result from process planning because of restricted number of motion of metal forming machines. Basically, semi-finished products often result from only one motion of forming machine, however, producing complicated shapes of products in very short portion of operating time [1].

Another circumstance of tooling design in metal forming is a raw product. For instance, cold forming as stamping processes flat raw products and there is no significant change in product's thickness but change in shape due to plastic deformation. Hot forming processes bars and rods by large volume deformation as displacement of volumes and sections. Such differences in cold and hot forming must be considered in CAD design of die's active surfaces. Additionally, active surfaces preparation technique

must be taken into consideration, too [2].

2. Surface types from the view of functionality

Surfaces used in engineering design can be categorised from the system user's point of view into four types, in order of increasing degree of geometric constraint on the surface: [3]

- a) Aesthetic surfaces where appearance is the main criterion for acceptability. During the design process, modifications are made until the appearance is acceptable to the designer. Examples include many plastic mouldings such as casings of household electrical appliances.
- b) Generalised duct surfaces which are subjected to more geometric constraints at their boundaries. The precise specification of the surface is needed when it gives a smooth blending between specified boundary constraints. Examples include automotive components such as suspension arms, exhaust manifolds and other types of ducting.
- c) Blends and fillets which provide smooth transitions between neighbouring previously defined object faces. In this case the geometric constraints are clearly greater. They are particularly useful in the design of mouldings, castings and forgings.
- d) Active or fitted surfaces with high degree of geometric constraint are applied to the surface as a whole, which is usually fitted to a large number of measured or pre-computed points. Examples

include turbine blades and the aerodynamic surfaces of aircraft. Fitted surfaces are also widely used in the automotive industry for specifying car body shapes and they are called Class A surfaces. They might be thought to be of the 'aesthetic' variety, but in most companies the aesthetic criteria are imposed during the creation of model, from which subsequently digitised points are used in the generation of a 'fitted' surface. [3]

3. Surface types used in CAD/CAM systems

CAD and CAD/CAM software packages used in 3D modelling/design play significant role in dies' design. Complex mathematic descriptions of active surfaces have been implemented into CAD and CAD/CAM packages and they allow designing of complex free form surfaces with a high degree of geometric constraints [3].

Nowadays, there are two types of surfaces primarily used in CAD/CAM systems [3], [4], [5], [6], [7], [8], [9].

1. **Regular or canonical surfaces** which include revolved surfaces as cylinders, cones, spheres, toroids and extruded surfaces. These surfaces are usually created using REVOLVE and EXTRUDE commands and both closed section (or sketch) and path of extrusion or axis of revolution are necessary to define. As the primary surfaces are created, other commands can be applied to create rounds, chamfers, ribs, drafts etc. The main attribute of these surfaces is exact definition of sketch described by dimensions of lines, radii, arcs, polygons, etc.
2. **Freeform surfaces** which represent more complex shapes of surfaces. Unlike regular surfaces, they have no rigid radial dimensions. To express surface form, non-uniform rational B-spline (NURBS) are commonly applied. Freeform surfaces and curves are not stored or defined in CAD software in terms of polynomial equations, but by their poles, degree, and number of patches (segments with spline curves). CAD software packages use two basic methods of creating freeform surfaces:
 - the first method begins with construction curves (splines) from which the 3D surface is then swept (section along guide rail), or meshed (lofted) through.
 - the second method is direct creation of the surface with manipulation of the surface poles and control points by Fig.1.

From initially created surfaces, other surfaces can be constructed using either derived methods or angle methods, the former refers to as offset or angled extensions, and the latter means bridging and blending between groups of surfaces.

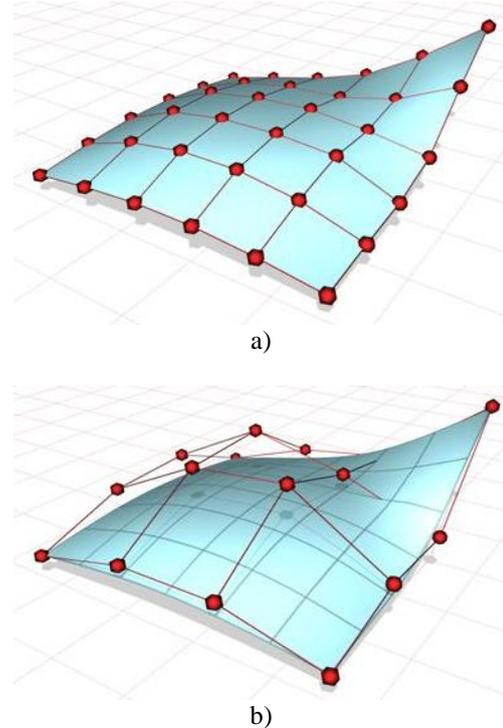


Fig. 1 Ways of NURBS surfaces manipulation
a) points shape the surface they lie on,
b) control vertices in a control lattice shape the surface it defines.

4. Modelling of forging die cavity

Commonly used principle in dies' design is that active surfaces are derived either from solid or from surface model. It is useful to note that die's active surface is negative shape to surfaces creating final semi-finished product. This procedure is typical for cavity creation of moulds and forging dies and it can be found in automotive industry. The main advantage of that progress is that active surfaces are directly derived from surface of forged piece whereas faults in 2D drawings are eliminated. Thus, two methods of design of forging die's cavity which includes active surfaces are introduced below: the first of them is based on Pro/MOLD module being known as mould cavity mode and another is based on merging positioned models by Cut Out operation, commonly known as assembly mode [10].

4.1 Forging representative

Fig. 2 shows shape of semi-finished product, a forging being taken to explain two methods of designing die's cavity. Because of needed machining allowances, semi-finished product consists from parts of regular surfaces, i.e., flange shaped head and tapered shaft. The former is of symmetrical feature with shaped noses. The latter is designed as step like cones which merge the rest into final forging.

Based on design of final product, engineering component, applicable dimensions and tolerances and surface finish, etc., were collected into tables to define machining allowances, dividing plane, bevels

and radii of sharp edges. Dimensionless feature of final shape of machined engineering component is shown in Fig. 3.

Because of complex shape of semi-finished products, no option does exist to perform only one forging operation and Fig. 2 shows results of forging

process plan to illustrate active surface design to perform the last operation of forging.

4.2 Design of die's active surfaces as mould cavity mode

Pro/Engineer Wildfire 5.0 was used to create

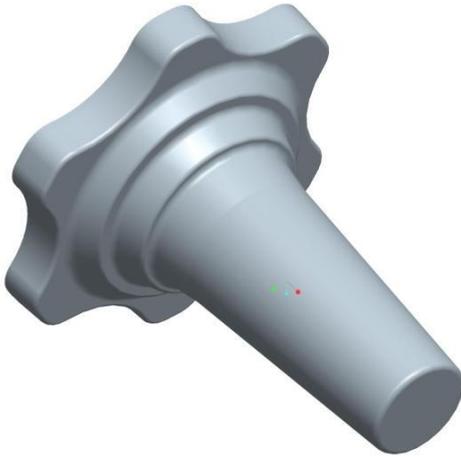


Fig. 2 Design of semi-finished product

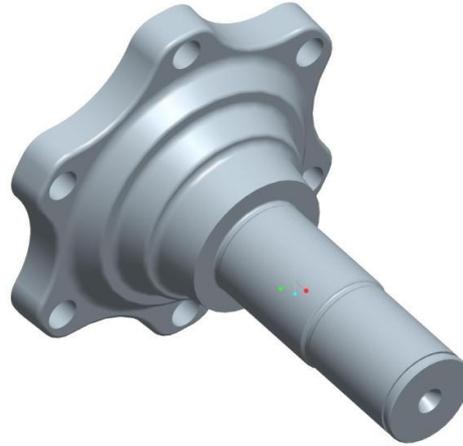


Fig. 3 Final product resulting from removal of machining allowances

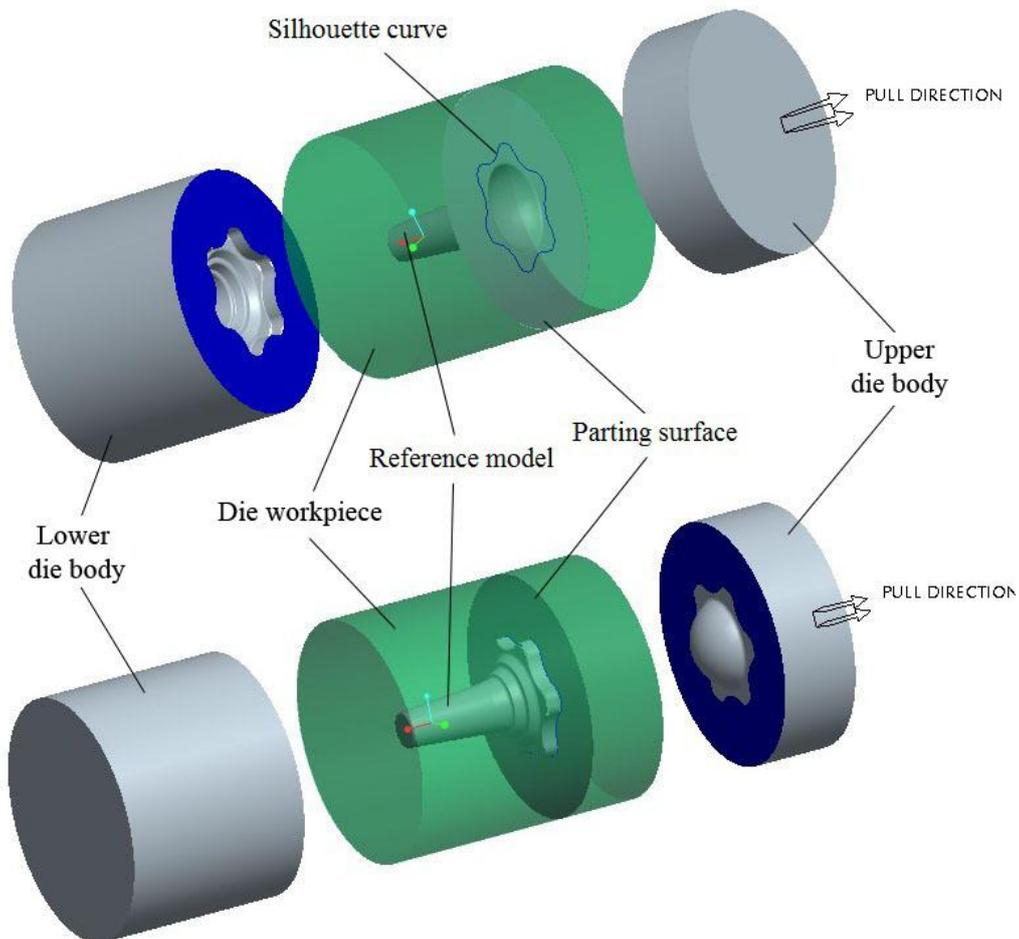


Fig. 4 Models of upper and lower forging die created as mould cavity mode

forging die cavity. Pro/MOLD represents one of optional modules to design cavities of casting moulds, injection moulds for plastics and forging dies including related components as runners, inserts, sand cores, etc. Based on the process controlled approach to cavity design, relevant icons are aligned in sequence of commands upside down. When dies cavity are created, commands are used step like in sequence of icons.

The proposed basic step of cavity design is reference part which applies “mould cavity mode”. The reference part is then used in software module to create die cavity. The part followed from basic module is loaded to Pro/MOLD as *REFERENCE MODEL*. The reference model is associated by only way for all changes in part model is respected to reference model. On the other hand, changes in reference model are not respected by part model. Final shape of upper and lower die and captions of their features are shown in Fig. 4.

Fig. 5 shows individual steps to design dies cavity by method of “mould cavity mode” which results from reference model in Fig. 4. Shrinkage of reference model as the second step is controlled by forging temperature. Shrinkage is applied in two ways: to whole part according to its coordinate system or separately to each dimension. Workpiece around reference model is defined in the third step. The workpiece represents outer shape of die created. Depending on reference model and workpiece position are then defined dividing planes and each volumes of created die. The workpiece is then divided by dividing planes and volumes onto required number of die parts that are extracted from workpiece to separate parts. These parts then create upper and lower part of dies for forging.

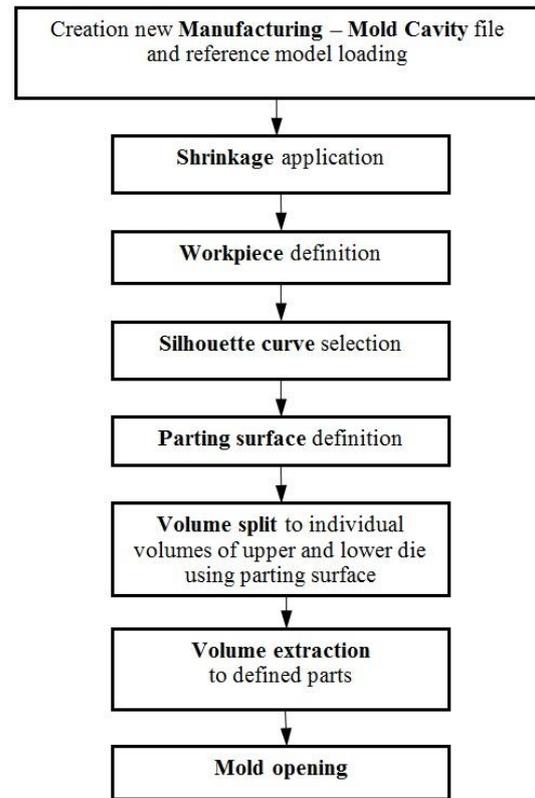


Fig. 5 Flowchart of steps at die cavity creation as mould cavity mode

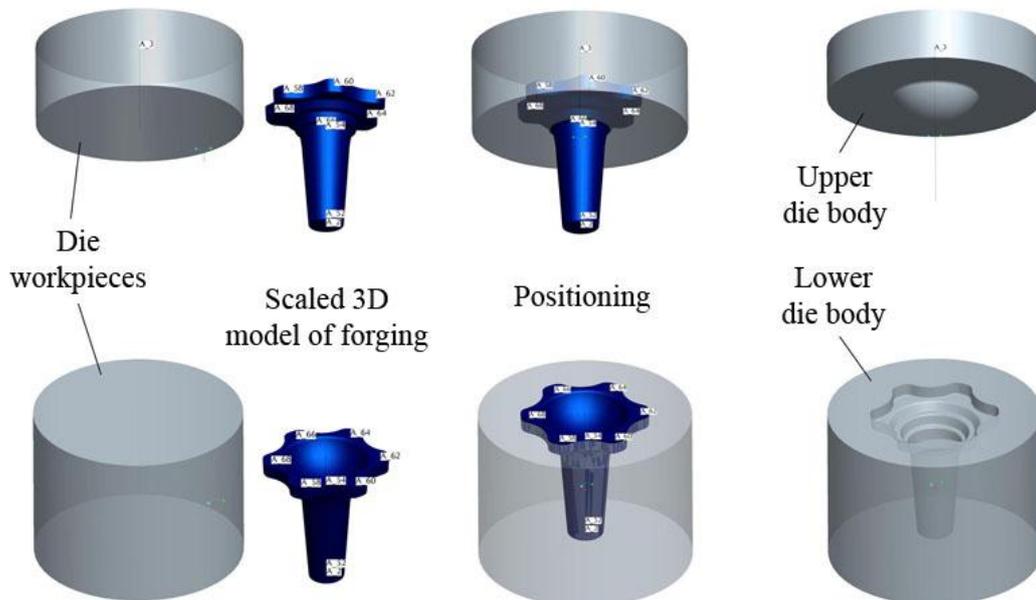


Fig. 6 Models of upper/lower forging dies created as assembly mode

4.3 Design of die's active surfaces as assembly mode

Design of die cavity in assembly mode requires further 3D models of forging which can be either solid or they include all surfaces. Additionally, both initial bodies of upper and lower die are included. The basic step is that positioning of resultant forging and bodies of upper/lower die are positioned to the parting surface in assembly mode.

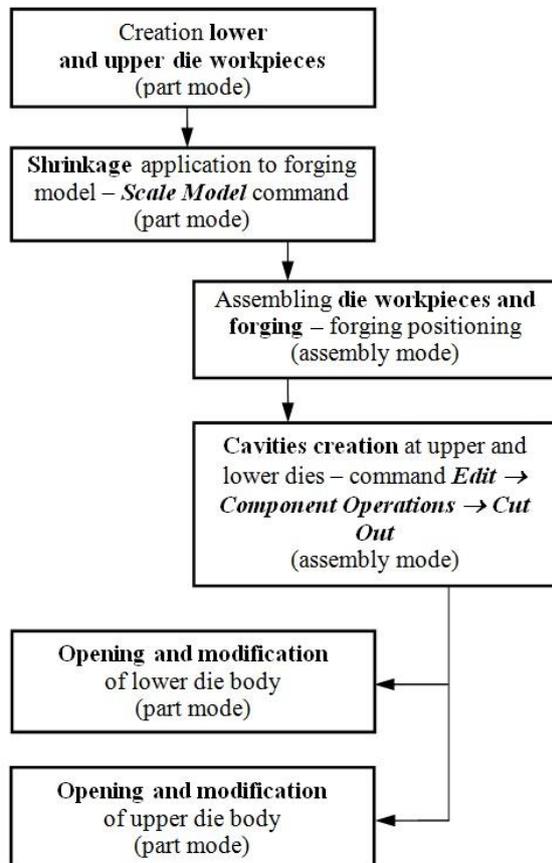


Fig. 7 Flowchart of steps at die cavity creation by Cut Out Component Operation (assembly mode)

Fig. 6 shows 3D model of /upper lower die which includes desired forging as semi-finished product. However, dimensioning of active surfaces is scaled according to shrinkage percentage and that means *Edit* → *Scale Model*. Positioning of desired forging is carried out by means of standardised assembly commands as *Mate*, *Align*, *Insert* whereas positioning of die is completely constrained. Cavity in upper/lower dies, therefore allows assuming its final shape as *Edit* → *Component Operations* → *Cut Out*. However, selection of part to perform *Cut Out* process is needed as following sequence:

- body of upper/lower die from Fig. 6 as the first step,
- 3D model of final forging with applied shrinkage as the second step of *Cut Out*.

Thus, designed cavity assumes negative shape of desired forging piece. Moreover, the body of upper/lower die is then represented by positioning of 3D model of forging piece with applied shrinkage whereas its resultant design follows from steps mentioned above. Fig. 7 shows sequence of die cavity design as “assembly mode”.

One fact which must be noted is that fully closed cavities occur when forging pieces of higher complexity than that of in Fig. 2. In such case, cavities must be re-modelled to resultant forging piece and redundant material in needed to remove.

4.4 Comparison of methods used in design of active surfaces of tooling

When die cavities are created in *mold cavity mode*, process tree represents complex procedure having items added by command sequence shown in Fig. 5. The base is reference model, being indicated by “_ref” and scaled by shrinkage percentage. The reference model is then used as core of creating die cavity. Then workpiece is necessary to define and add to process tree. Next silhouette curve is created and used for parting surface definition, referred to as *Skirt surface*. To create 3D models of upper and lower dies, workpiece splits volume and volumes extraction is done at last.

When die’s cavities are created by *assembly mode*, designer must follow two process trees. The first of them is process tree being created in assembly mode when workpiece and forging model are positioned and merged by *Cut Out* operation. The second process tree is created in part mode when die workpiece is finished. The cavity is created separately in assembly mode and process tree is very simple comparing to previous method. After material removing by command *Edit* → *Component Operations* → *Cut Out* die cavity is created in die workpiece. The *Cut Out* operation is then added to process tree in part mode when lower die part is opened for re-modeling. All operations on lower or upper die are then realized in part mode.

There are three main advantages of dies cavity creation in mold cavity mode as follows:

- creation of most complex parting surfaces,
- splitting of workpiece volume using parting surface,
- extracting split volumes to solid models of upper/lower die bodies.

Thus, the *mold cavity mode* represents process-driven, technologically oriented approach to die cavity creation. Produced solid models of lower/upper dies body maintain associative link to NC code when created; if the design part changes, the mould cavity and NC tool paths are automatically updated. Therefore, advantages mentioned above eliminate needs to translate instructions between part design, mould design and NC programming due to seamless integration of application modules.

Design of die cavity in assembly mode represents more simple process. Designer performs each operation in sequence of flowchart shown in Fig. 7 manually. Positioning of die body and forging as well as parting surface definition use standard methods and commands allowed in assembly mode. However, created die bodies require re-modelling due to closed cavities which do occur when forging pieces of higher complexity.

5. Conclusion

Nowadays, forming dies design employs mainly 3D technique. CAD and CAD/CAM software packages offer powerful tools for die designers to create complicated shapes of products including complicated shapes of active surfaces of forming dies. Procedures used in forging dies design process were analysed including their active surfaces. Main advantage of introduced procedures is that they offer associativity of transferring all changes in 3D model to software modules that had been used and no data transformation through neutral file formats is required. Substantial reduction of design – technology - cycle is then reached whereas possible mistakes in data transformation are reduced, that are factors contributing to the efficiency in tooling design.

Acknowledgement

This work was financially supported by the Scientific Grant Agency of Slovak republic under the scientific Grant VEGA No. 1/0500/12 “Research of Quality Improvement when Milling Formed Surfaces by Advanced Coated Tools”.

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