Interior volume modelling for MolFlow+ simulations

1. Direct solid body modelling approach

Description:
Solid body of interior volume is created directly using following steps:

- **Define work planes and axis** considering of future model symmetry or periodicity.
- **2D sketches, fully constrained** to the work plane, well dimensioned.
- **3D modelling features** and with appropriate Boolean operations (remember that there must be a single solid body at the end).
- Export xxx.stl file in ASCII format to MolFlow+.

Advantages:
Fast intuitive modelling using basic commands.
Solid models are parametric, so you can generate number of dimensional variations.
Disadvantages:
Difficult modelling of complex geometries (RF Fingers etc.)

Recommended application:
In case you need idealized geometry in short time or you know that you will solve geometry with the same shape but different dimensional configuration.
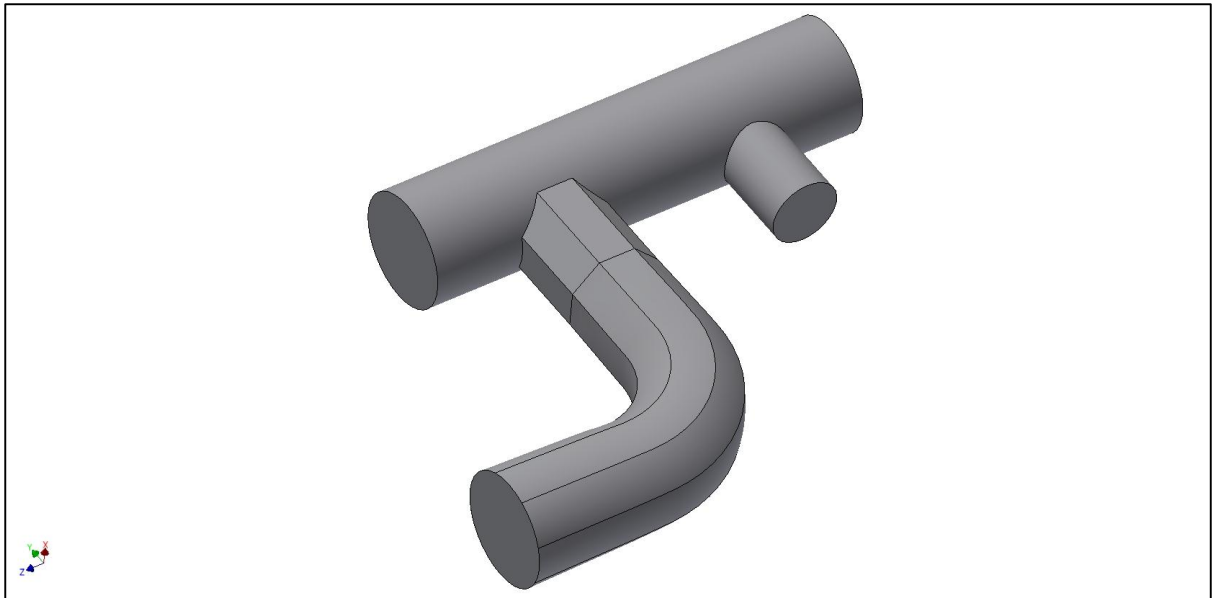

Figure 1 Solid body created by direct modelling

2. Surface to solid body approach

Description:
Based on imported geometry (CATIA Smart Team) can be interior volume created using making a copy of inner surfaces. We are reusing designers work so we are not pushed to direct modelling of complex shapes.

- **Import and save assembly** in xxx.iam format on local.

- Go to **Simplify card** and run the **Create simplified part** command (combine style is single body with seams).
- Save a new created xxx.ipt file on local.
- Using the **Delete face** command with **active heal option** to remove all unwanted features like small faces, holes etc.).
- Run **Copy Object** command with option **Select Face and Create Surface** (not a composite).
- **Select all interior faces one by one** and after apply check that no face is missing.
- Go to navigator panel, Solid Bodies and **turn off visibility for Solid1**.
- **Create inlet and outlet "plug"** face using Patch command.
- **Build single solid body** using **Sculpt** command with **Add option** selected (select all surfaces by window selection).
- Export xxx.stl file in ASCII format to MolFlow+.

Advantages:
Easy and fast approach to create interior surface from existing CATIA part or assembly.
Existing part can be easily simplified to reduce number of faces in MolFlow+.
Disadvantages:
Selecting of inner surfaces must be taken with big care.
Import form other CAD to Inventor remove all the model history – non parametric model.

Recommended application:
This approach is suitable for existing 3D models or simple assemblies without complex interior details (bellows etc.). In case that the geometry contains a lot of interior details (RF fingers etc.), the approach can be difficult due to number of faces and its selection.
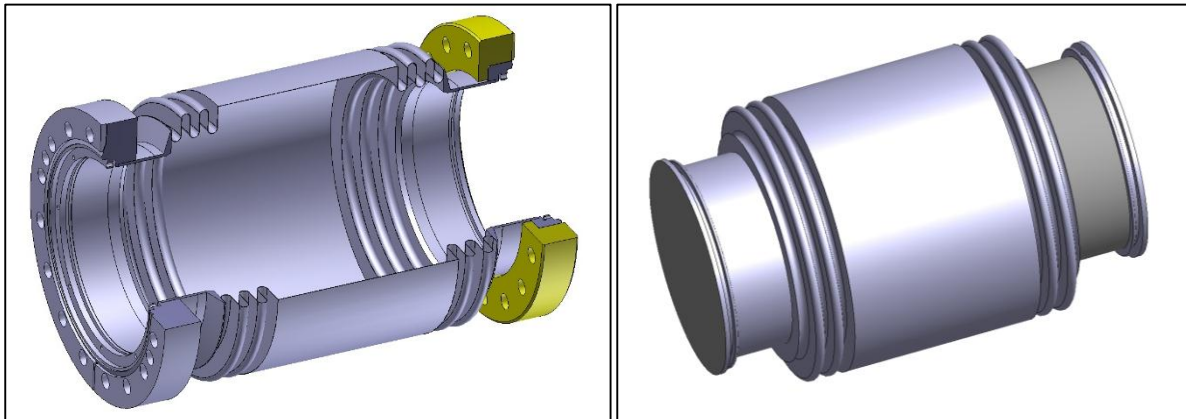

Figure 2 Imported CATIA model and solid to surface approach created interior volume

3. Imprint and sculpt approach

This approach allows to create the most complex geometries without any need of its direct modelling. Interior volume is based on imprint of imported and simplified existing geometry of part or assembly.
- **Import and save assembly** in xxx.iam format on local.
- Go to **Simplify** card and run the **Create simplified part** command (combine style is single body with seams).
- Save a new created xxx.ipt file on local.

- **Simplification:** Using the **Delete face** command with active **heal option** to remove all unwanted features like small faces, holes etc.).
- Run **Copy Object** command with option **Select Face and Create Surface** (not a composite).
- **Select ALL faces** by window selection mode.
- Go to navigator panel, Solid Bodies and **turn off visibility** for **Solid1**.
- Create **the "envelope" volume** (cylinder or box) which is **slightly bigger than the surface body geometry** (all surfaces must be inside of this envelope volume). Do not forget that **when you are creating it** a new Extrude or Rotate feature **there must be NEW SOLID option active** in the command.
- Run **Sculpt** command **with Remove option**. Surface – select all surfaces created by copy command, Solid – select an envelope volume.
- Use the section view to see and inspect the "Easter egg".
- Share the vacuum boundary existing edges: **2D sketch** and Project Geometry command **project the boundary** region lines **of your vacuum area** (sketch must be on the vacuum exposed plane which is the last one before the air side, the projected geometry is for example the edge of CF flange blade).
- Create the vacuum boundary surfaces: **3D modelling features** like Extrude or Rotation **with SURFACE output** option **must ends** or **go out of the "envelope" volume.**
- Run **Sculpt** command **with Remove option**. Surface – select the boundary surfaces from previous step, Solid – select the "Easter egg". Using a green ball with arrows **set the part of removed volume**. Arrows must be positive normal of interior volume (must go out of the axis in radial direction).
- Export xxx.stl file in ASCII format to MolFlow+.

Advantages:
No need of one by one face selection, if the imported model is without any defects you cannot do a mistake.
Existing part can be easily simplified to reduce number of faces in MolFlow+.
Disadvantages:
Little bit complex modelling approach.
Import form other CAD to Inventor remove all the model history – non parametric model.

Recommended application:
This approach is suitable for existing 3D models or assemblies with complex interior details. The modelling approach is quite complicated but you can get a nice result in max. 1 hour for a huge and interior complex assemblies.
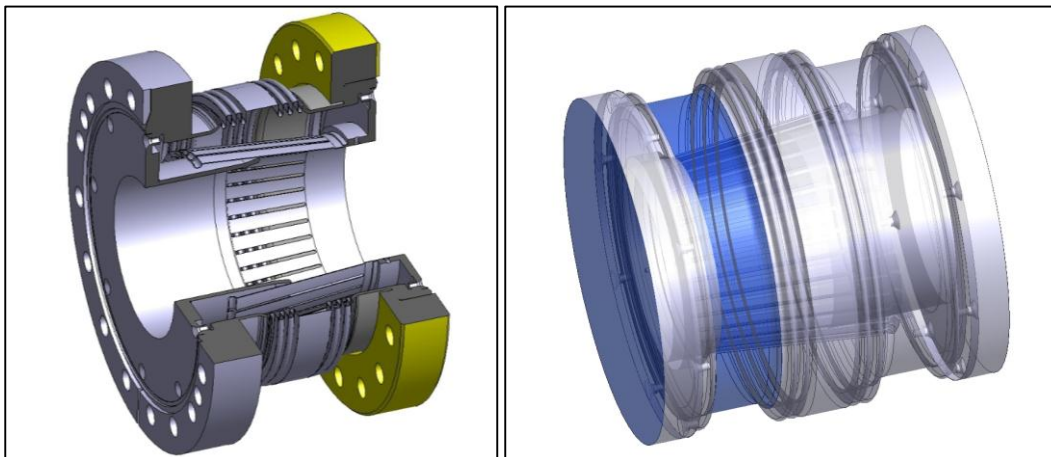

Figure 3 Imported CATIA model and Imprint and sculpt approach created volume