# *MultiPhysX Consulting – MPX*
# *Newsletter n°3*
# *March 2026*

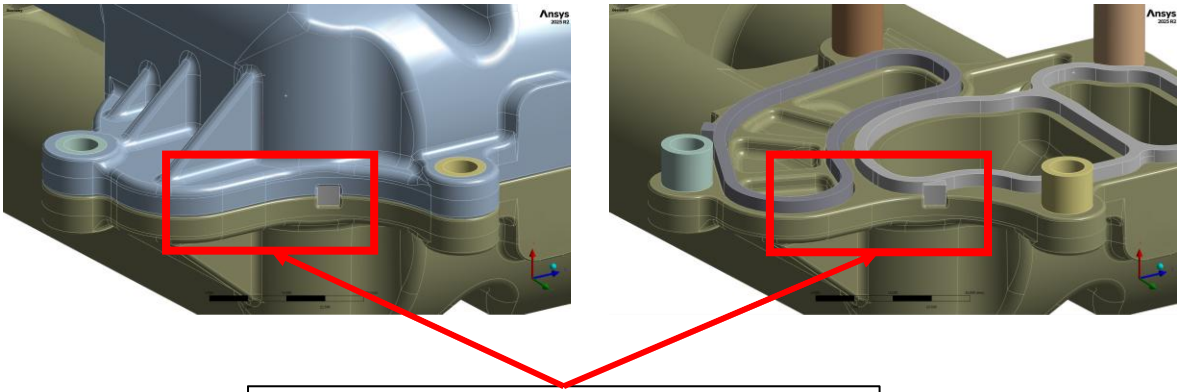
**How to solve sealing issues with glass-fiber-reinforced engineering plastics and optimize and maximize the performance of reinforced engineering plastics thanks to Multiphysics Simulation.**

## Objective of the newsletter

This newsletter highlights **the added value of multiphysics simulation** in a demanding application at **180°C**, involving the **replacement of metal parts by a glass-fiber-reinforced engineering plastic**.

Les objectifs sont multiples :

- **Reduce the weight** of the components,
- **Lower the final cost** of the part,
- **Improve the environmental footprint** (resource reduction, recyclability),
- **Solve a recurrent sealing issue**, characterized by **air leaks under extreme conditions**.



Area where the air leak was observed

## Problem: air leaks under engine hood at 180°C

During laboratory tests, an **air leak** generally appears after **5 to 10 minutes** (depending on thickness, material, and environment).

In some cases, a **failure not predicted by the simulation** is observed, worsening the issue.

**Key observation:** under **homogeneous temperature of 180°C**, the leak appears **after 3 to 5 minutes**, because thermal equilibrium is reached more quickly.

## Hypotheses and proposed solutions

Since the leak is not immediate, **creep** (related to internal pressure, gasket, and time) is often considered the main cause.

Classic solutions include:

- **Increasing the glass fiber content** (e.g.: 35% → 50%),
- **Changing the polymer family** (more expensive but potentially more performant).

However, these options **reduce the weight advantage**, **increase costs**, and do not guarantee success.

## Two critical questions:

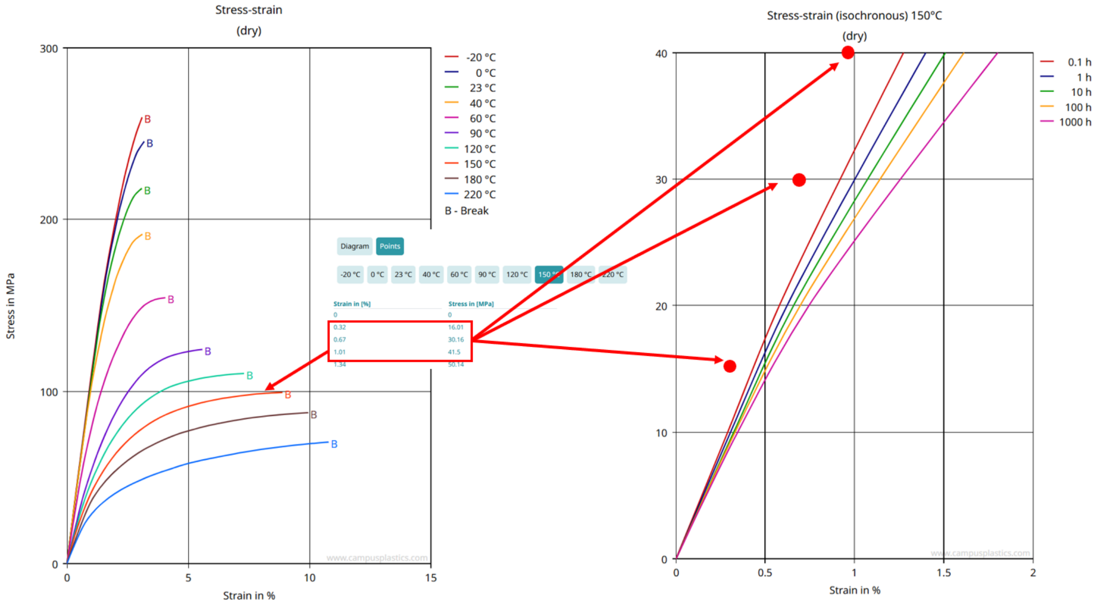
- Is **creep** really responsible for the leaks?
- Have all **physical phenomena** been included in the simulation?

## Creep behavior of a PA66 thermoplastic with 35% glass fibers at 150°C:

Creep depends on **temperature, stress level, time, and the environment (chemical exposure).** The data shows that after 6 minutes (~0.1 h), the **stress–strain curve evolves**, indicating a **progressive decrease of the modulus**, therefore an **increasing displacement of the part**.

I recommend:

- **Not starting directly with creep simulations**,
- Identifying the **dominant parameters** (temperature, stress, fiber orientation, ...),
- Reducing the stress level in critical areas through **simple design modifications (lower stress = reduced creep impact)**.



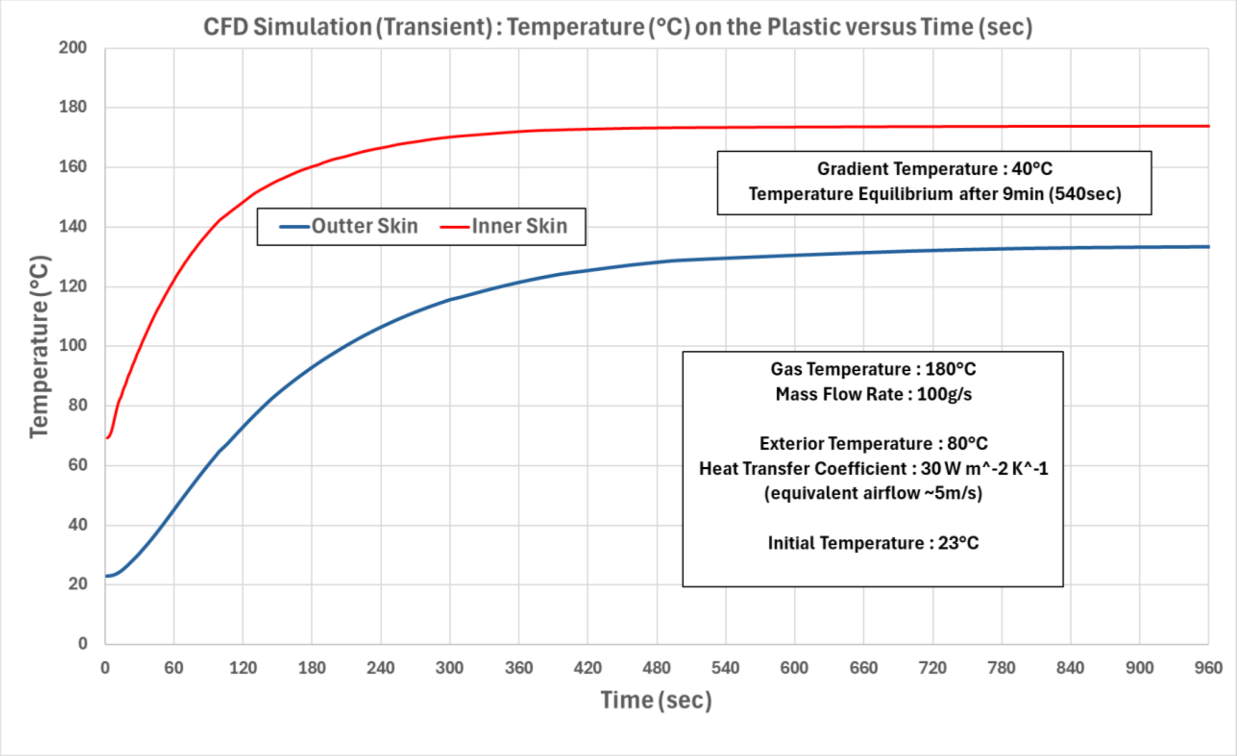
| Strain in [%] | Stress in [MPa] |
|---|---|
| 0 | 0 |
| 0.32 | 16.01 |
| 0.67 | 30.16 |
| 1.01 | 41.5 |
| 1.34 | 50.14 |

## Thermal behavior of engineering plastics vs metals

Engineering plastics exhibit **very low thermal conductivity** compared to metals, which leads to:

- Despite a **rapid temperature rise** (from 23°C to 180°C), **thermal equilibrium in the plastic part is reached in 5 to 10 minutes** (versus a few seconds for metals).
- A **strong thermal gradient** across the thickness (30 to 50°C between inner and outer skin), compared to only 3 to 5°C for a metal part.

→ Applying a **uniform temperature** in a simulation with an engineering plastic is an **excessive simplification** that may lead to incorrect conclusions or unnecessary oversizing.



## Contribution of multiphysics simulation

It allows:

- **Breaking down the mechanisms**,
- Identifying **the dominant phenomenon**,
- Implementing **simple, fast, and cost-effective solutions**.

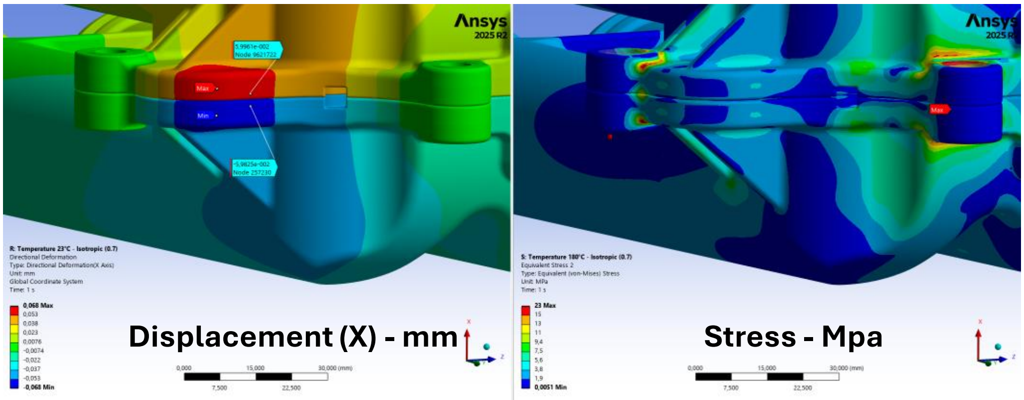
The ideal is to intervene **upstream**, before production.

## Results of iterative and comparative simulations

To understand and analyze part behavior and break down the phenomena, here are the simulation scenarios carried out, each integrating different assumptions and conditions:

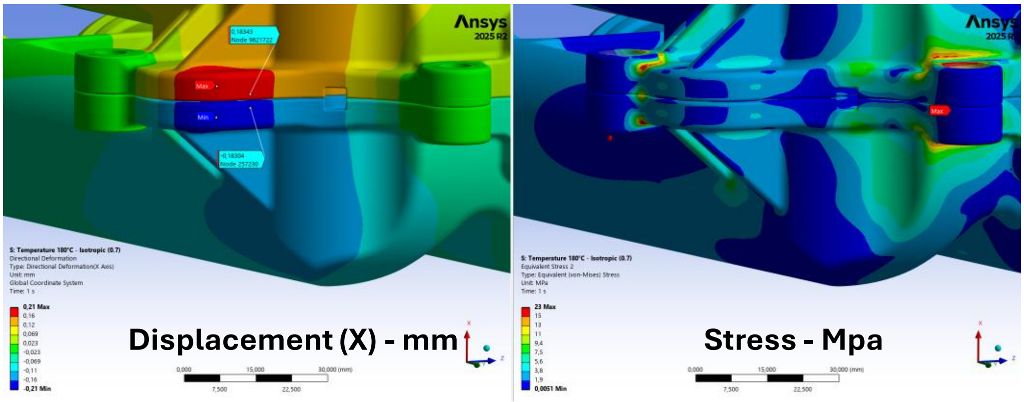
### 1.Simulation at 23°C with simplified isotropic material – *gap of 0.12 mm*

- Uniform temperature of **23°C** with internal pressure and gasket replaced by pressure.
- Material considered **isotropic**, with a **0.7 coefficient** applied to the 23°C stress–strain curve. This is a simplification because the real plastic part will not have the same orientation as the test specimen used to generate the mechanical data.



### 2.Previous simulation but with material at 180°C – *gap of 0.37 mm*

- Uniform temperature of **180°C**.
- Same approach for material modeling.

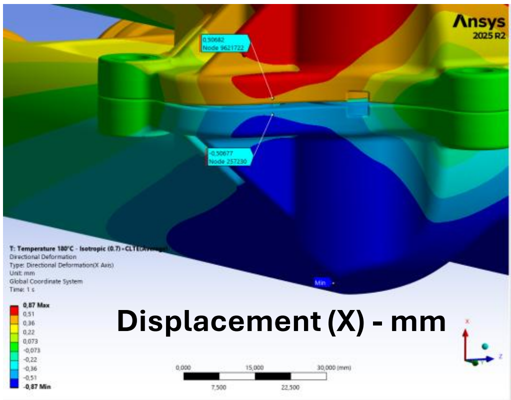


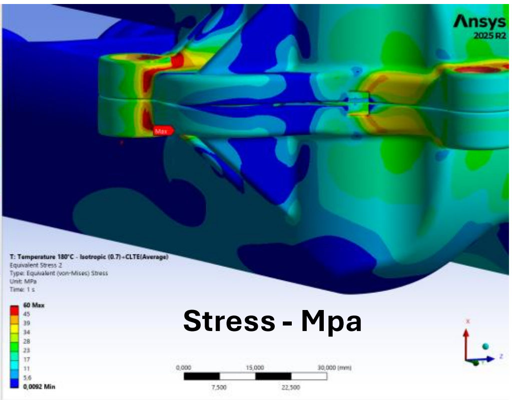
These simulations illustrate only **the impact of temperature**.
They remain **far from reality** and provide an **overly optimistic view** of the part.

### 3. Simulation with the same method but with material at 180°C and including thermal expansion – *gap of 1.1 mm*

- Uniform temperature of **180°C**.
- Same material model as previous simulations.
- Thermal expansion included, using an average of the coefficients in parallel and perpendicular directions.

The impact of thermal expansion is **major**: going from 0.37 mm to 1.1 mm shows that not including it strongly distorts the results.



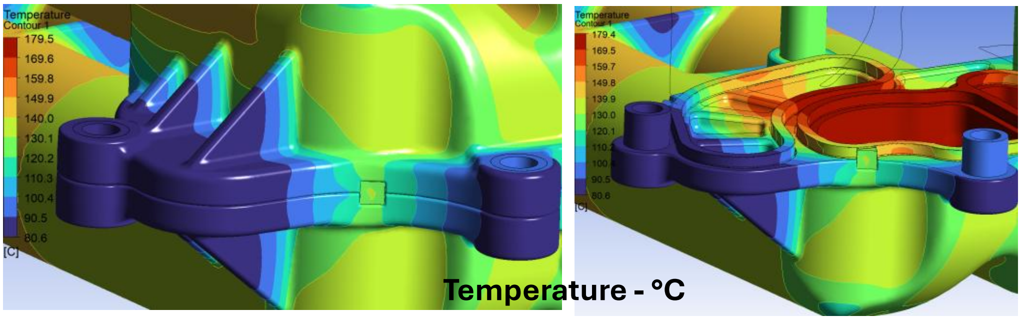


## Realism of the calculation?

- **Pessimistic** if a uniform temperature is applied (never the case in reality).
- **Optimistic** if process-related flatness defects are not considered.
- **Pessimistic** if the injection point is poorly located (unfavorable orientation).
- **Optimistic** if creep is not considered.

A CFD simulation of PA66 GF35 (photo below) clearly shows a **40 to 50°C gradient**, and that **the temperature is not uniform**.

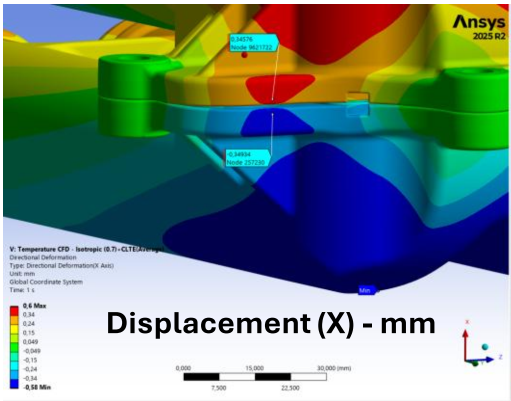
**Testing the parts at a uniform 180°C is extremely pessimistic**, can generate artificial failures, and does not reflect real operating conditions.

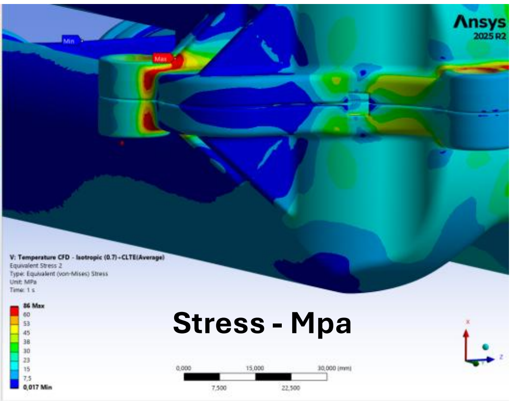


Temperature - °C

## 4. Previous simulation but more realistic using temperatures from CFD – *gap of 0.69 mm*

- In reality, the part temperature **is never uniform**.
- This simulation uses **temperatures from CFD**, allowing a much more realistic representation.
- Material still modeled as **simplified isotropic**, like previous simulations.





Now that the model is more realistic regarding temperatures, it is time to move to the next step: the calculation including fiber orientation.

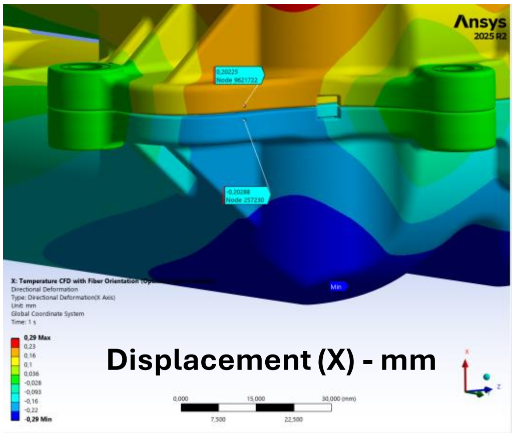
## 5. Multiphysics Simulation – *gap of 0,40mm*

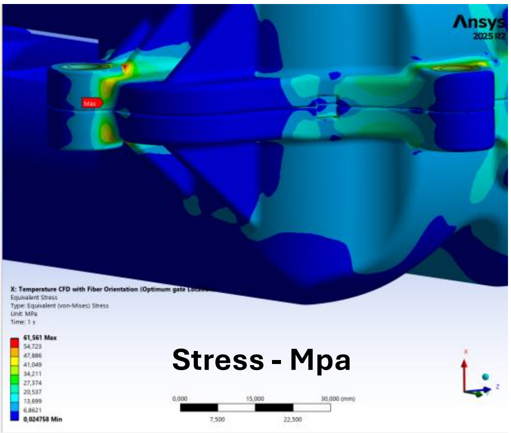
- **Temperatures from CFD,**
- **Thermal expansion,**
- **Fiber orientation** obtained from rheological studies.

**Why is the rheological study essential?**

It allows:

- **Orienting the fibers** to limit displacements induced by thermal expansion,
- **Avoiding placing the injection point** in a critical area and limiting weld lines in highly stressed zones,
- **Reducing flatness defects** in gasket zones and therefore **minimizing leak risk**.





## Summary :

| | Simulation | Thermal Expansion | Thermal realism | Fiber Orientation | Creep | Gap | Score |
|---|---|---|---|---|---|---|---|
| 1 | **Simplified 23°C** | ✗ | ✗ | ✗ | ✗ | **0,12 mm** | ★ |
| 2 | **Simplified 180°C** | ✗ | ✗ | ✗ | ✗ | **0,37 mm** | ★★ |
| 3 | **180°C + Thermal expansion** | ☑ | ✗ | ✗ | ✗ | **1,1 mm** | ★★ |
| 4 | **T°C (CFD) (realistic) + Thermal expansion** | ☑ | ☑ | ✗ | ✗ | **0,69 mm** | ★★★ |
| 5 | **Multiphysic Simulation *** | ☑ | ☑ | ☑ | ✗ | **0,40 mm** | ★★★★ |

**** Optimized technical solution without changing the design or the material.***

**In the next newsletter**, I will show more details about injection, fiber orientation, and simulations including creep.

## Why are these simulations crucial?

They allow:

- **Comparing the impact of simplifying assumptions** (isotropy, uniform temperature) with real conditions,
- **Highlighting the dominant phenomena** (creep, thermal gradient, expansion, etc.),
- **Validating corrective solutions** before production, thereby reducing failure risks and associated costs.

## Need support or expertise?

Do you have a project, questions, or would you like to benefit from the expertise of a Multiphysics Simulation specialist? **Contact me**.

And if, like me, you are passionate about **Multiphysics Simulation**, subscribe to **MultiPhysX Consulting – MPX** to stay updated on our latest news and innovations.

**Frederic BONNIN**
MultiPhysX Consulting - MPX
78000 Versailles
www.multiphysx.com
Mobile : +33 6 81 53 45 80
Email : Frederic.Bonnin@multiphysx.com