Proceeding Paper

2D Heat Transfer of an Injection mold: ANSYS Workbench and Mechanical APDL [†]

Hugo Miguel Silva [1,2,*], Hugo Luís Rodrigues [1], João Tiago Noversa [1], Leandro Fernandes [1] and António José Pontes [1]

[1] IPC-Institute for Polymers and Composites, Department of Polymer Engineering, University of Minho, Campus de Azurém, 4800-058 Guimarães, Portugal; email1@email.com (H.L.R.); email2@email.com (J.T.N.); email3@email.com (L.F.); email4@email.com (A.J.P.)

[2] proMetheus, Instituto Politécnico de Viana do Castelo, Rua Escola Industrial e Comercial

[*] Correspondence: hugo.silva@hmsilva.science

[†] Presented at the 4th International Electronic Conference on Applied Sciences, 27 October–10 November 2023; Available online: https://asec2023.sciforum.net/.

Abstract: The latest developments in additive manufacturing have enabled the creation of conformal cooling channels with improved efficiency and cost-efficiency. In the context of the injection molding process, it has been shown that conformal cooling channels (CCC) demonstrate improved cooling effectiveness when compared to conventional straight drilled channels. The primary justification for this phenomenon is from the fact that conformal cooling channels (CCC) have the capacity to adapt to the contours of a molded object, a feat that cannot be accomplished with traditional channels. Carbon-carbon composites CCC possess the capacity to alleviate thermal stresses and warpage, reduce cycle durations, and attain a more uniform temperature distribution. Traditional channels employ a design method that exhibits greater intricacy as compared to CCC. The utilization of computer-aided engineering (CAE) simulations is of paramount importance in the advancement of designs that demonstrate cost-effectiveness and efficiency. The aim of this research is to evaluate the efficacy of two ANSYS modules for the purpose of validating the acquired outcomes. The two modules exhibit comparable results when used on models with a more detailed mesh. Therefore, it is crucial to consider the objective of the research and the complexity of the computer-aided design (CAD) geometry while making a well-informed choice regarding the suitable ANSYS module to use.

Keywords: conformal cooling; injection molding; computer aided engineering; design optimization

1. Introduction

The application of additive manufacturing streamlines and decreases the expenses associated with the production of conformal cooling channels (CCC). The cold injection molding process used by CCC has exceptional performance. In contrast to mechanical devices, CCC have the capacity to adhere to predetermined patterns or molds. The utilization of CCC has been observed to be an efficient measure in mitigating heat stress and deformation. The incorporation of computer-aided engineering (CAE) simulations is vital to attain design outcomes that are both efficacious and economically viable. In their work, Dimla et al. (2005) used I-DEASTM's moldflow analysis to determine the optimal placement of channels [1]. ABM Saifullah and SH Masood conducted an evaluation of the "part cooling time" by employing the ANSYS thermal analysis modules [2]. In the year 2009, an evaluation of components was undertaken by researchers utilizing MPI simulation modules. In their research, Gloinn et al. conducted a study in which they used ABS polymer as the molten material and included a cooling water input to determine the mold temperature [3]. In 2007, a study was undertaken utilizing Moldflow Plastic Insight 3.1 software to examine the thermal effects of cooling channel design in the context of

injection molding. The concept of consistency in the design of content creation was originated by authors. The efficacy of the cooling loop was demonstrated by Wang et al. [4] through the utilization of component temperature modeling. A study was conducted by Khan et al. in 2017, wherein they used AMI modules to evaluate various cooling settings. The metrics encompassed in this study comprise cooling times, total cycle times, volumetric contraction, and temperature change. The study conducted a comparison of several cooling channel layouts, including conventional, serial, parallel, and additive-parallel arrangements, in terms of their respective properties. This research paper provides a comparative examination of the 2D transient thermal analysis functionalities offered by ANSYS Mechanical APDL and ANSYS Workbench. The study primarily concentrates on examining the steady state and transient thermal analysis aspects of both software platforms. Prior studies have also been conducted in the domain of two-dimensional analysis [5]. Previous studies have undertaken the process of design optimization in three-dimensional (3D) space [6]. The integration of Mechanical APDL with ANSYS Parametric Design Language facilitates the parameterization and optimization of models across many computer languages. The objective of this work is to investigate the optimal meshing parameters by the simultaneous cross-validation of both modules and to assess if, for this situation, if both software modules are accurate. In practice that implies that either the engineer can freely select one of the ANSYS software modules, or should carefully select one of them, based on external validation, for example, using experimental tests.

2. Methods

2.1. CAD Models (Computer Aided-Design)

The CAD model used in this project was generated by the combined utilization of ANSYS Workbench and ANSYS Mechanical APDL 2020 R2. The three-dimensional architecture comprises eight cooling channels, visually represented as circles, a mold cavity, drawn as a rectangle, and the part itself, represented by a curved plate. The assembly is illustrated in Fig. 1. Table 1 presents a comprehensive overview of the fundamental components of geometry.

Quantity	Description
1	Part
8	Channels
1	Mold
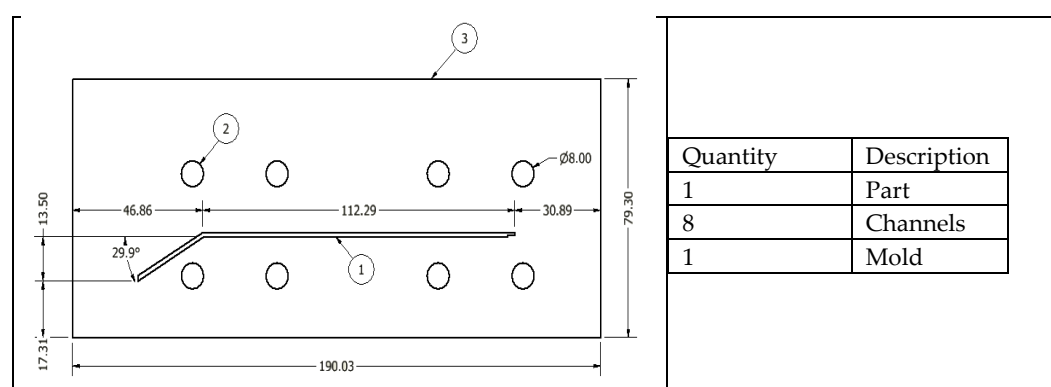
Fig. 1. Assembly drawing of the mold, showing dimensions, in millimeter and components ID (left) and Table 1–Components of the geometry used in the simulations and in the optimizations (right) [6].

2.2. Materials

Water was employed in the cooling channels of the simulations, whereas polypropylene (PP) was used in the injection part. P20 mold steel was used for the fabrication of the mold. Among these components, it is widely accepted that water exists in a liquid condition, whereas PP and steel are considered to exist in solid states. The features of the material are presented in Table 2:

Table 2. Properties of the materials used in this work [6].

Material	Water	PP with 10% Mineral	P20 Steel
Density [(kg/m^3)]	998.2	1050	7861
Specific heat [J/(kg.K)]	4182	1800, Considered constant	502.48
Thermal conductivity [W/(m.K)]	0.6	0.2 Considered constant	41.5

The cooling channels, shown as round structures, were fabricated using liquid water. The injected section was fabricated using polypropylene (PP), while it is probable that the mold cavity was constructed using p20 steel. It is widely accepted in scientific discourse that steel predominantly exhibits solid-state characteristics, whereas the remaining two substances tend to exhibit properties more like those of liquids.

2.3. Numerical Procedure

The mesh employed in this study is a quadrilateral free mesh, with a mean element size of 1 mm for the cooling channels and molds, and 0.0625 mm for the portion. Despite the utilization of identical mesh parameters in both modules, there exists a notable disparity in the overall quantity of mesh components between Workbench and Mechanical APDL. The numerical values 82,871 and 151,885 are being referred to in the context of Workbench. The mesh seen in Fig. 2 was generated with the Mechanical APDL and Workbench software.
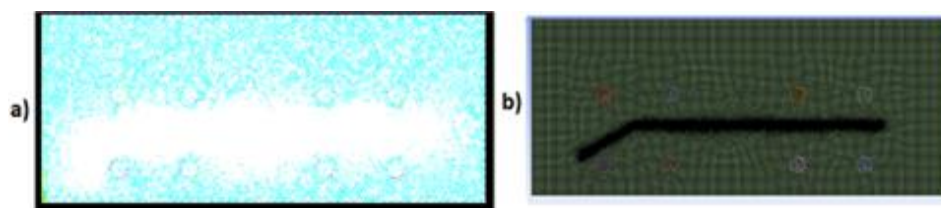
Fig. 2. Mesh in ANSYS Mechanical APDL (**a**) and in ANSYS Workbench (**b**).

The component (final part) under consideration is subjected to a temperature of 210 degrees Celsius. There is a prevailing belief that the water temperature within the cooling conduits remains consistently at 40 degrees Celsius. The methodology used in this study was designed to enable a comparison and validation of numerical outcomes acquired through the utilization of two software modules that are part of the ANSYS suite. There are two notable distinctions that may be observed between the two approaches: Given that the geometry was inherently developed in both modules, it is conceivable that the geometry environment may exhibit variations. The act of mixing gives rise to a novel differentiation. The utilization of identical mesh parameters results in meshes that exhibit discernibly disparate element quantities. One possible explanation for this disparity is that Workbench's meshing capabilities offer a far greater range of options compared to those available in Mechanical APDL. The default values of all parameters that are present in Workbench but absent in Mechanical APDL are retained.

3. Results and Discussion

3.1. Mesh Sensitivity Analysis

In order to ascertain the adequacy of the lattice for generating precise outcomes, a convergence analysis was conducted. Mesh sensitivity analyses were performed using both Mechanical APDL and Workbench in order to verify the comparable accuracy of the findings obtained from the two modules, despite the variations in their meshing modules. The mesh sensitivity analysis used four stages of refinement. Table 3 illustrates the correlation between the degree of precision and the sizes of the starting and final elements.

Table 3. refinement levels and element sizes.

Refinement Level. R_L [mm]	Esize, Initial [mm]	Esize, Final [mm]
1	1	0.5
2	0.5	0.25
3	0.25	0.125
4	0.125	0.0625

Fig. 3 shows the results of the sensitivity analysis for the maximum value of the temperature Tmax, for different element sizes and for both ANSYS Mechanical APDL and Workbench.
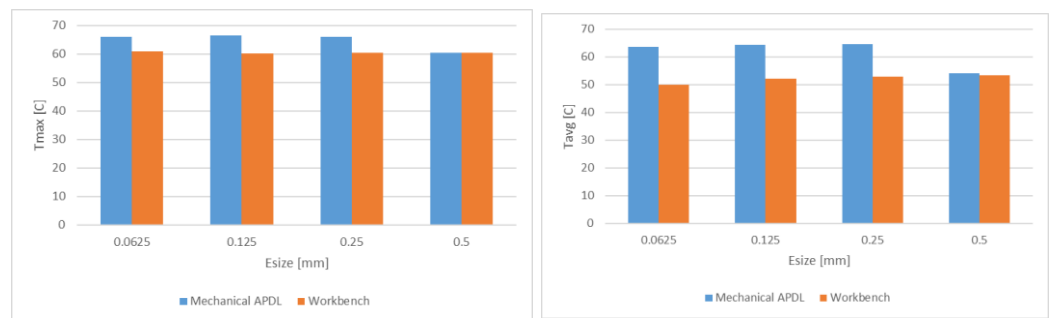
**Fig. 3.** Tmax (left) and Tavg (right), for different mesh refinement levels.

The maximum temperature in Mechanical APDL exhibits a notable increase compared to that in Workbench, as illustrated in Fig. 3 (left). The sole anomaly pertains to the mesh size of 0.0625 mm, wherein the temperature measurements exhibit a slight elevation. The conclusions of the sensitivity analysis for the average temperature Tavg are depicted in Fig. 3, located on the right side. In comparison to Fig. 3 (left) and Fig. 3 (right), ANSYS Mechanical APDL demonstrates a notably higher level of accuracy in generating temperature data when compared to ANSYS Workbench. The increased mesh effectively diminishes the significance of the discrepancy, given its mean element size of 0.0625 mm. Fig. 4 illustrates the discretization error associated with the maximal temperature in both Mechanical APDL and Workbench. The findings presented in Fig. 4 were obtained by utilizing the data from Fig. 3 and applying Equation (1).

$$Error_{Tmax_n}[\%] = \frac{T_n - T_{(n-1)}}{T_{(n-1)}} * 100\% \tag{1}$$

where n is the refinement level.

Fig. 4 illustrates the discretization inaccuracy in Mechanical APDL and Workbench, specifically in relation to the maximum (left) and average (right) temperature. The findings presented in Fig. 4 were obtained by utilizing the data from Fig. 3 and applying Equation (1).
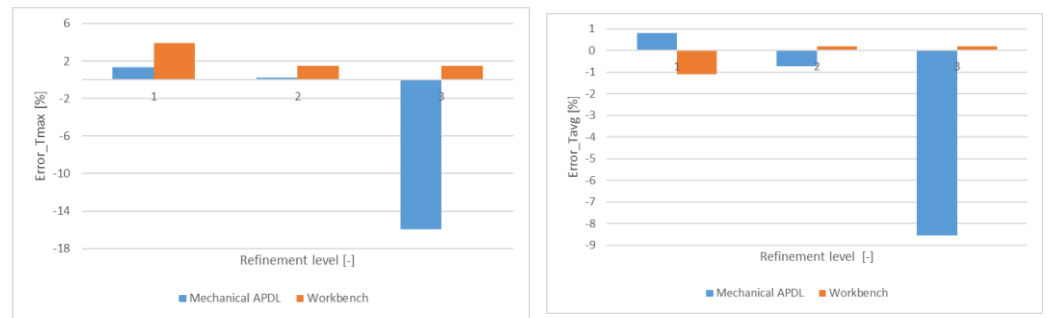
Fig. 4. Mesh Error in Mechanical APDL and in Workbench, in term of maximum temperature (**left**) and of average temperature (**right**), for different mesh refinement levels.

Fig. 4 illustrates that, save from the 0.0625 mm element size, significant mistakes are observed across all element sizes and levels of refinement. Hence, it can be concluded that a mesh size of 0.0625 mm is the sole appropriate choice for achieving accurate estimates with little error, regardless of whether ANSYS Workbench or ANSYS Mechanical is employed. Upon comparing errors between a finer mesh and a coarser mesh with a mean element size that is twice as large, it becomes evident that the errors in Mechanical APDL exhibit an increase as the level of refinement is enhanced. Conversely, the errors in Workbench demonstrate a decrease under similar circumstances. This is exemplified by both Fig. 4 (left) depicting the maximum temperature and Fig. 4 (right) illustrating the average temperature.
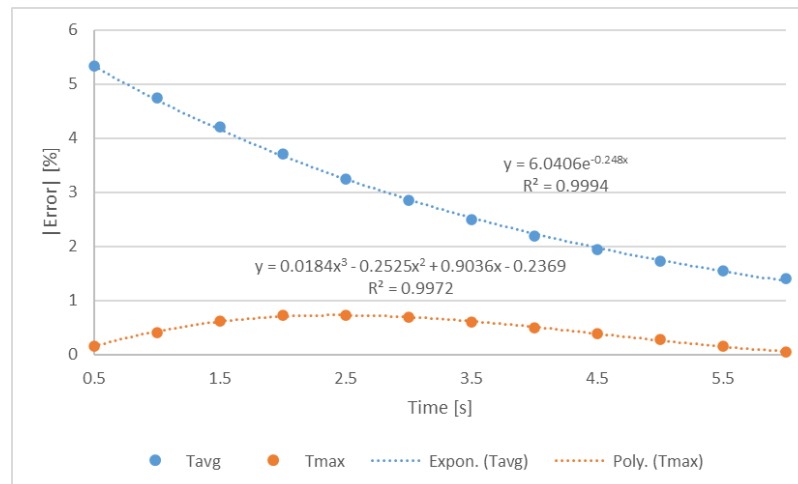
3.2. Comparison

Fig. 5 shows the maximum temperature Tmax (left) and the average temperature Tavg (right) in function of time, for both Mechanical APDL and Workbench.
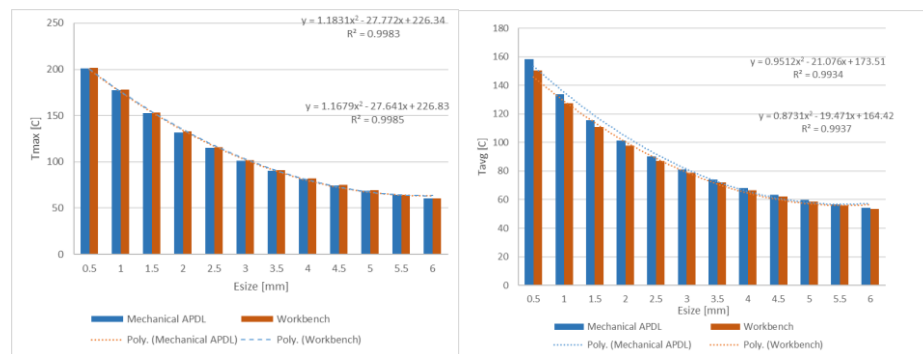
Fig, 5. Maximum temperature (**left**) and average temperature (**right**), in function of time, for both Mechanical APDL and Workbench.

Fig. 5 depicts the resemblance observed in the results obtained from both the Workbench and Mechanical APDL. The maximum temperatures of workbenches often exhibit a modest increase compared to those of APDL. In Fig. 5 (right), the average temperature Tavg is illustrated as a time-dependent variable for both the Mechanical APDL and the Workbench. According to the data presented in Fig. 5 (right), it can be observed that Mechanical APDL offers a higher level of value compared to Workbench in most instances. Nevertheless, the discrepancies between the two software components gradually decrease as time progresses. The errors regarding the results shown in Fig. 5 (left) (Tmax) and in Fig. 5 (right) are shown in Fig. 6. The errors were calculated using (2).

$$|Error|_{[\%]} = \frac{|T_{wb} - T_{APDL}|}{T_{wb}} * 100\% \qquad (2)$$

Fig. 6. Error between Mechanical APDL and Workbench, in term of maximum and average temperature, in function of time.

As depicted in Fig. 6, the temporal inaccuracy can be adequately modeled by a third-degree polynomial function for the maximum temperature (Tmax) and an exponential function for the average temperature (Tavg). According to Fig. 6, the quadratic correlation value exhibits a high degree of proximity to 1. As the study presented is purely numerical, it can be applied to part manufactured in other ways, although in real applications, conformal cooling channels will definitely be of value, due to the much larger freedom of manufacturing the desired geometry,

4. Conclusions

The ANSYS Mechanical APDL and Workbench outputs underwent cross-validation. The utilization of cross-validation, which encompasses a range of numerical approaches, presents a feasible substitute for experimental endeavors and analytical validation in complex scenarios. An element size of 0.0625 mm with high precision is required. As illustrated in Fig. 6, two equations are used to express software problems.

The main findings of the work can be summarized as follows:

- The main finding of this work is to know if in 2D Heat transfer analysis, both ANSYS Mechanical APDL and Workbench are accurate under the same conditions. They are, as the results are only slightly different. One can, therefore, conclude that, either ANSYS Mechanical APDL or ANSYS Workbench are suitable for 2D heat transfer analysis on molds.
- The aforementioned equations ascertain the potential advantages of employing mesh refinement in further processing.
- The presence of distinct module choices in the two programs poses challenges in reproducing simulation parameters, specifically pertaining to meshing qualities, despite diligent attempts. The elements and meshes experienced the most significant modifications.
- While it is true that basic meshes may not align perfectly with the two components, the most refined meshes are able to achieve this alignment. Consequently, numerical results are predominantly influenced by disparities in elements and meshes.
- Resolving this matter was straightforward. Both modules employ intermittent direct approaches for multiple substeps. In the future, it is possible to enhance designs by utilizing MATLAB and ANSYS Mechanical APDL or Workbench for optimization purposes.

In the future, the assembly presented and studied here, can be manufactured by additive manufacturing and tested experimentally. The methodology could also be applied to molds with different number of cooling channels, with different mold dimensions and

with different temperatures of the final part, to increase the extension of the study and the generalization of the findings". The use of conformal cooling channels in real application could definitely allow the minimization of temperature gradients and warpage of the final part, as well as improve its quality.

Author Contributions: Conceptualization, H.M.S.; methodology, H.M.S.; software, H.M.S.; validation, H.M.S.; formal analysis, H.M.S.; investigation, H.M.S.; resources, H.M.S.; data curation, H.M.S.; writing—original draft preparation, H.M.S.; writing—review and editing, L.F., J.T.N. and Hugo Rodrigues; visualization, H.M.S.; supervision, A.J.P.; project administration, A.J.P.; funding acquisition, A.J.P. All authors have read and agreed to the published version of the manuscript.

Institutional Review Board Statement:

Informed Consent Statement:

Data Availability Statement: The datasets generated during and/or analyzed during the current study are available from the corresponding author on reasonable request.

Conflicts of Interest: Authors declare no conflict of interest.

References

1. Dimla, D.; Camilotto, M.; Miani, F. , Design and optimisation of conformal cooling channels in injection molding tools. *J. Mater. Process. Technol.* **2005**, *164*, 1294–1300.
2. Saifullah, A.; Masood, S. Finite element thermal analysis of conformal cooling channels in injection molding. In Proceedings of the 5th Australasian Congress on Applied Mechanics, Brisbane, Australia, 10–12 December 2007.
3. Ó Gloinn, T.; Hayes, C.; Hanniffy, P.; Vaugh, K. FEA simulation of conformal cooling within injection molds. *Int. J. Manuf. Res.* **2007**, *2*, 162–170.
4. Wang, Y.; Yu, K.M.; Wang, C.C.; Zhang, Y. Automatic design of conformal cooling circuits for rapid tooling. *Comput. -Aided Des.* **2011**, *43*, 1001–1010. Jahan, S.A.; El-Mounayri, H. Optimal Conformal Cooling Channels in 3D Printed Dies for Plastic Injection Molding. *Procedia Manuf.* **2016**, *5*, 888–900. https://doi.org/10.1016/j.promfg.2016.08.076.
5. Silva, H.M.; Noversa, J.T.; Fernandes, L.; Rodrigues, H.L.; Pontes, A.J. Design optimization of conformal cooling channels for injection molds: 3D transient heat transfer analysis. *Mech. Adv. Mater. Struct.* **2023**. https://doi.org/10.1080/15376494.2023.2203686.
6. Silva, H.M.; Rodrigues, H.L.; Noversa, J.T.; Fernandes, L.; Pontes, A.J. 3D Heat transfer of an injection mold: ANSYS Workbench and Mechanical APDL [submitted].