Topology Optimization applied to the design of Air Intake Systems for automotive applications

Dr. Carlos Massaiti Okubo Júnior
Sergio Adrian Ferreyra
MAHLE

Email: carlos.c.junior@mahle.com; sergio.ferreyra@mahle.com

ABSTRACT

The design of the flow path in a fluid system is not a trivial task. In Air Intake Systems of internal combustion engines, the fluid must flow smoothly and with the smallest pressure drop possible to the combustion engine inlet. The limitations from space and positioning of components of the car make this task even more challenging. In this scenario, optimization techniques arise as an option to treat the problems in a systematic way and advance fast in the design improvements. The Topology Optimization Method (TOM) is placed as robust strategy to give insights and help understanding the most important flow features for a given system. In this work, the TOM is used as tool to generate geometries for the flow path of part of an automotive Air Intake System. The theoretical basis is briefly explained with an objective function relevant to the problem. An example case is presented, where a design is obtained, interpreted, evaluated and compared with a traditional geometry, highlighting the gains that can be achieved by using the Topology Optimization.

INTRODUCTION

The design of Air Intake Systems (AIS) is a key area in the development of automotive products. These systems directly affect the thermodynamic efficiency, power output, and emissions of internal combustion engines. An ideal AIS must provide clean air to the turbo compressor or to the internal combustion engine or to the Air Intake Manifold (AIM), ensuring low total pressure loss between inlet and outlet and also a uniform and organized flow.

Designing such systems using conventional methods, such as parametric CAD modeling and CFD analysis of geometries generated by manual design iterations can be limiting and inefficient. The conventional approaches may loose high performing geometries for some cases, particularly in constrained packaging environments. In contrast, optimization methods can provides systematic and physics-driven ways to discover innovative designs based on performance criteria, without being constrained to predefined geometric features or traditional constructions.

In this work, the Topology Optimization Method is the strategy chosen to design AIS. Topology Optimization can place and remove material anywhere inside a design domain and is a very powerful technique to generate non-intuitive and innovative designs. As this is a gradient-based type of optimization, the total derivatives of the objective function with respect to the design variables are necessary. The adjoint method is the most appropriate calculation technique for the situation and is used here. To represent the solid parts inside the fluid volume, the Brinkman penalization is applied to the Navier-Stokes equations. Also, to be able to control the amount of final volume required to be fluid, a volume constraint is used, applied by the Augmented Lagrangian Method. In the next sections, the structure used to apply Topology Optimization is described and a case study is presented to show how the methodology can be used to produce high performing designs.

THEORETICAL BACKGROUND

TOPOLOGY OPTIMIZATION IN CFD

In Topology Optimization of fluid systems it is possible to represent solid and fluid regions within a continuous computational domain. The Brinkman penalization models the design domain as a porous medium whose permeability varies continuously with a design variable. Regions with a low permeability are interpreted as solids and regions where permeability is so high that no resistance is offered to the fluid are interpreted as fluid [1].

In practice, the penalization to represent solid regions consists in adding a term to the momentum equations. In Equations (1) to (3), the Navier-Stokes system is presented with the Brinkman penalization (last term to the right). The permeability field is represented by the compostition of the scalar value α and the scalar field γ. The “intensity” of the permeability is given by the value of α, so it usually assumes very high values (in this work, 10^7 is used). The low permeability regions are controlled by the distribution of γ, that goes from 0 to 1. Where this field is 1, the penalization term α_p is zero, the equations assume the original form and the region is interpreted as fluid. Where γ is 0, a local resistance is added to the flow (with α intensity), representing a low permeability and suggesting the presence of a solid region [2, 3, 4].

$$\nabla \cdot \boldsymbol{u} = 0 \tag{1}$$

$$\rho\nabla \cdot (\boldsymbol{uu}) - \nabla \cdot (\mu(\nabla\boldsymbol{u} + \nabla\boldsymbol{u}^T)) + \nabla p + \alpha_p\boldsymbol{u} = 0 \tag{2}$$

$$\alpha_p = \alpha(1-\gamma) \tag{3}$$

The Brinkman penalization is simple to implement and compatible with most CFD solvers. It has the advantage of not requiring mesh changes during the optimization. On the other hand, the interface fluid-solid is not well defined, so it is a good practice to perform a final evaluation of the design in a body fitted mesh [3, 4].

A scheme for an example of Topology Optimization for an internal flow case is shown in Figure 1. The image on the top left part of the figure, represents a simple flow case to be calculated. On the top right, this geometry gains a square volume in the middle, representing a design domain, which is a portion where the design of the part can be made. The bottom right part of the figure shows a possible result of the application of Topology Optimization, where a γ field defines an optimized trajectory for the flow. Finally, on the bottom right part, a possible interpretation for the design is presented, where the flow can be evaluated accurately.

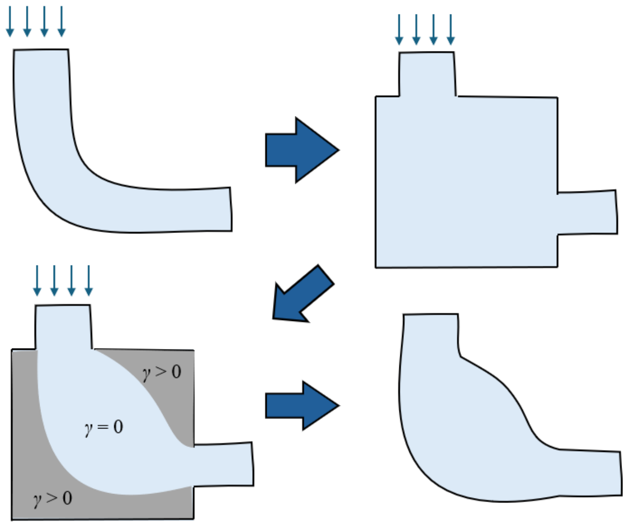

Figure 1. Topology Optimization of internal flow.

THE ADJOINT METHOD

In optimization using gradient-based algorithms, the calculation of the sensitivities is one of the most important steps and also one of the most challenging ones. In cases where the number of design variables is much larger than the number of constraints, the adjoint method is a very appropriate choice. Only two solver calls are necessary, one for the physical problem and other for the adjoint one (which has a similar computational cost) [2, 3]. Every optimization problems starts with its main definition. For this work, it is:

$$\begin{cases} \underset{\gamma}{Minimize} \quad J \\ \\ Subject\ to\text{:} \quad flow\ equations \\ \qquad\qquad\qquad other\ constraints \end{cases}$$

Where J is the objective function (in this case, total pressure drop) to be minimized and γ is the permeability field already explained (in the optimization problem, γ is the design variable).

The adjoint method transforms the constrained optimization problem into an unconstrained one where the objective is to minimize the Lagrangian function L. A new set of variables is required, the Lagrange multipliers $\boldsymbol{\lambda}$. They help define the Lagrangian as follows [1, 4]:

$$L = J + \boldsymbol{\lambda}^T\boldsymbol{R} \tag{4}$$

Where $\boldsymbol{R}$ is the residual form of the equations that govern the physical problem (Equations (1) to (3)).

Knowing that the objective function J and the governing equations $\boldsymbol{R}$ depend on the vector of state variables $\boldsymbol{U}$ (velocity and pressure) and the design variables $\boldsymbol{\gamma}$, it is possible to calculate the sensitivity of the objective function with respect to the design parameter γ. This is performed by taking the total derivative of Equation (4). After some rearranging and knowing that $\boldsymbol{R}$ must be almost zero for a well calculated flow, this equation can be written as:

$$\frac{dL}{d\boldsymbol{\gamma}} = \frac{\partial J}{\partial \boldsymbol{\gamma}} + \boldsymbol{\lambda}^T\frac{\partial \boldsymbol{R}}{\partial \boldsymbol{\gamma}} + \left(\frac{\partial J}{\partial \boldsymbol{U}} + \boldsymbol{\lambda}^T\frac{\partial \boldsymbol{R}}{\partial \boldsymbol{U}}\right)\frac{d\boldsymbol{U}}{d\boldsymbol{\gamma}} \tag{5}$$

The term d$\boldsymbol{U}$/d$\boldsymbol{\gamma}$ is computationally very expensive, so by taking term in parenthesis to zero, it is possible to avoid its calculation. This is performed by finding the correct values of $\boldsymbol{\lambda}$ and constitutes the so-called adjoint problem [4]:

$$\frac{\partial \boldsymbol{R}^T}{\partial \boldsymbol{U}}\boldsymbol{\lambda} = -\frac{\partial J}{\partial \boldsymbol{U}} \tag{6}$$

Then, the sensitivities for the optimization problem can be calculated by [1, 4]:

$$\frac{dL}{d\boldsymbol{\gamma}} = \frac{\partial J}{\partial \boldsymbol{\gamma}} + \boldsymbol{\lambda}^T \frac{\partial \boldsymbol{R}}{\partial \boldsymbol{\gamma}} \tag{7}$$

After calculating the sensitivities, the optimizer can lead to highly improved solutions.

In this work, the Adam (Adaptive Moment Estimation) update rule is used since it is readily available in StarCCM+ version 2022.1. This algorithm is designed for first-order gradient-based optimizations and it helps avoid local minimum. For a deeper understanding of this optimizer, the reader is referred to [4, 5].

In the numerical cases shown in this work, a volume constraint is used. This is implemented with the augmented lagrangian technique, which corrects the sensitivities by adding the derivative of the constraint in a progressive scheme. More details can be seen in [4, 6].

OPTIMIZATION SCHEME

The application of Topology Optimization for CFD consists in the sequence of steps below. This sequence can also be seen in Figure 2.

- Initially, a geometry with a possible region for the design to exist is created;
- The flow is calculated for the existing geometry;
- The adjoint system is calculated for the flow condition;
- The sensitivities are calculated with the results of both the physical and the adjoint system and also with the contributions from the volume constraint;
- The stopping criteria are assessed;
- If it is time to stop, the final design is ready to be interpreted and the loop is finished;
- If no stopping criteria is satisfied, the permeability field is updated and the the sequence of steps is repeated from the second item, until the stopping criteria are met.

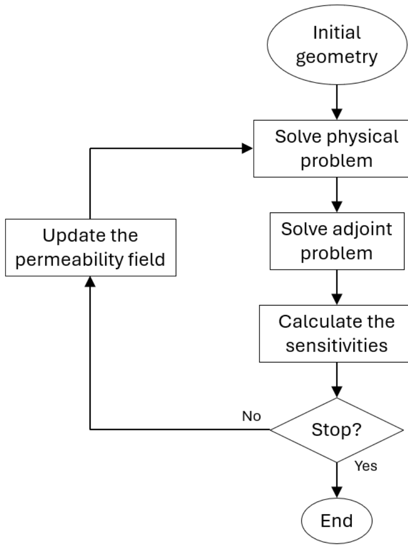

Figure 2. Topology Optimization flowchart.

CASE STUDY

In this work, a case study is presented considering realistic geometries, fluid properties and boundary conditions. Initially, a conventional design is evaluated in terms of pressure drop to establish a baseline. Then, two optimization cases are presented. The first shows optimization running without any strategies that changes the trajectory the optimizer goes until the final solution. The second is an optimization case where the initial iterations are calculated with a very high viscosity value. This changes the path the optimizer takes and leads to a different result as shown next. All calculations are performed with the commercial CFD software StarCCM+ , version 2022.1 (17.02.007-R8).

BASELINE MODEL

The AIS used as example in this work is presented in Figure 3. The full system is composed by a fresh air duct, an airbox with a filter and a clean air duct that discharges at the entrance of a turbo compressor. Only part of the clean air duct is used in the optimization, represented in red in Figure 3. The whole system could be optimized, but, by selecting the most important components, it is possible keep the model size inside low limits while still having a relevant influence in the whole system calculation.

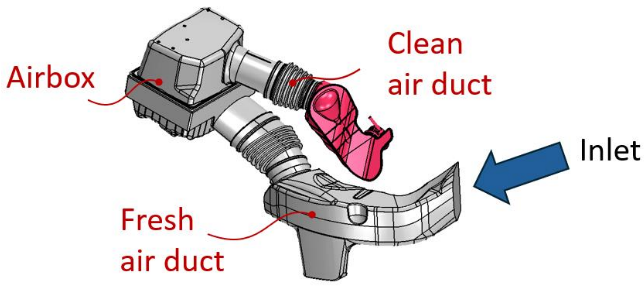

Figure 3. Air intake system.

The flow properties and boundary conditions can be seen in Table 1. The clean air duct is extracted from the system and extensions are added to the inlet and outlet. In Figure 4, the red part interior is shown. The duct used in the optimization is the part highlighted in blue.

The baseline case is meshed with a base size of 4 mm and has approximately 27 000 cells (see Figure 5). The case is calculated until the pressure drop of the part (without considering the extensions) is stable with respect to the iterations. This takes 1000 iterations and the pressure drop for the baseline case is approximately 558 Pa. In Figure 6, the total pressure and some streamlines with velocity values are presented.

Table 1. Fluid properties and boundary conditions.

Property	Value
Inlet mass flow (kg/s)	0.139
Density (kg/m³)	1.18
Viscosity (Pa.s)	1.86×10^{-5}

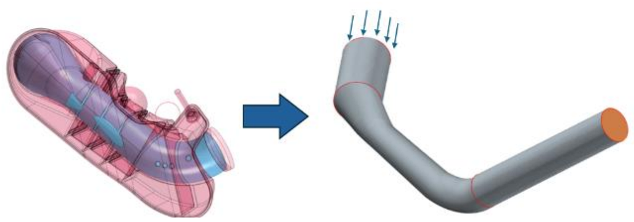

Figure 4. Part to be optimized.

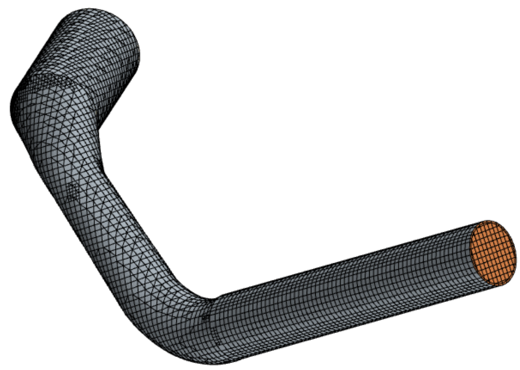

Figure 5. Mesh for the baseline case.

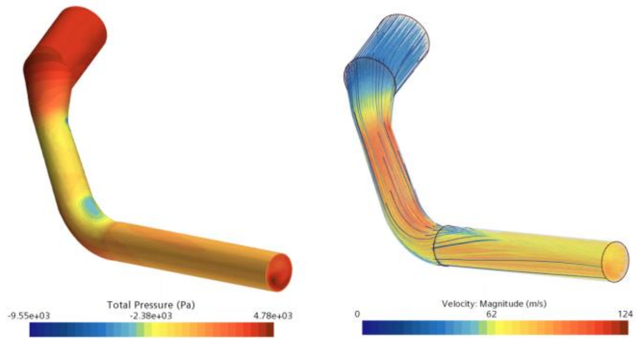

Figure 6. Total pressure (left) and streamlines (right) for the baseline

TOPOLOGY OPTIMIZATION – MODEL 1

A good use of the topology optimization features requires the existence of a design space or design domain. This space is where the final part can exist and the definition of where this part will be is the main task of the optimizer. Hence, the baseline model is changed to have a bigger volume where the clean duct can exist. The full model used in the topology optimization is exhibited in Figure 7. As can be seen, only the extensions for inlet and outlet are kept. The volume added is built around the original part, considering that the optimized geometry can exist there. The base size for the mesh used in the optimization models is 4 mm, resulting in approximately 125 000 cells (Figure 8).

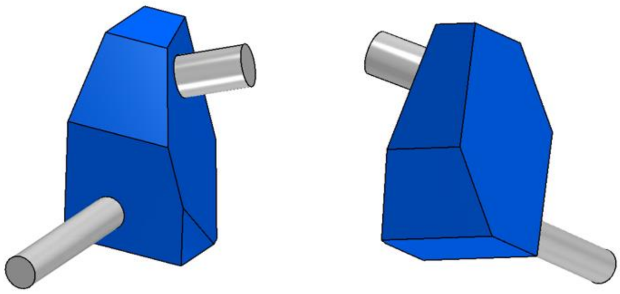

Figure 7. Design domain for optimization (blue region)

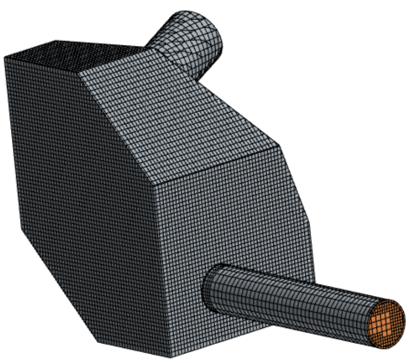

Figure 8. Mesh for the optimization model.

A volume constraint of 15% of the design space is used. This means that the final design must have 15% of the design domain as fluid volume.

For the first optimization model, the real properties are used during the whole optimization. After 75 optimization iterations, the final design obtained is presented in Figure 9.

The geometry is extracted and remeshed (with the same base size) and the final value of pressure drop obtained 268 Pa, representing 52% of reduction in the pressure drop of the original geometry. Total pressure and streamlines with velocity values are presented in Figure 10.

As can be seen, the design has a flat shaped geometry, where the volume is spread close to one of the walls. This is a consequence of the initial iterations of the pyshical problem, where the flow goes into the design domain as a jet. As the optimization proceeds, the design gets “stuck” close to this main path, rounding corners and eliminating recirculation regions until the final solution is found. Even though the design represents a relevant reduction in the pressure drop, its manufacturing may not be easily achieved. Hence, in model 2, an alternate way of conducting the optmization is explored, resulting in an optimized geometry with a simpler shape.

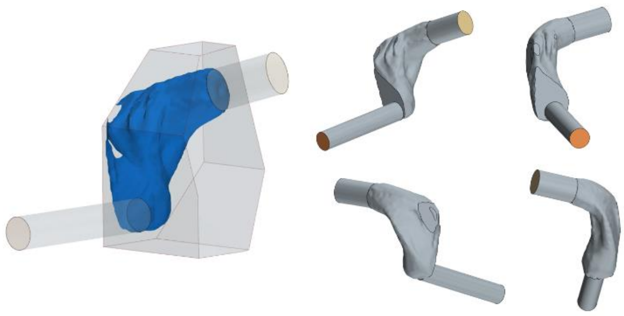

Figure 9. Optimized geometry for model 1.

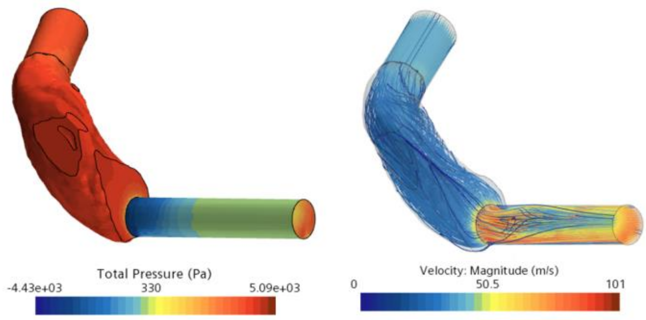

Figure 10. Total pressure (left) and streamlines (right) for the optimized model 1

TOPOLOGY OPTIMIZATION – MODEL 2

In this second optimization model, the initial iterations are calculated with a very high viscosity value (approximately 9000 times the real value). After 15 optimization iterations, the real viscosity value is used and the optimization is performed during more 60 iterations. In Figure 11, the last design considering the the high viscosity value is presented and, in Figure 12, the final geometry is shown.

Again, the geometry is extracted from the optimization model, remeshed and calculated again. The pressure drop for this case is 209 Pa, achieving 63% of reduction with respect to the original part. Total pressure and streamlines with velocities for the second optimization model are presented in Figure 13.

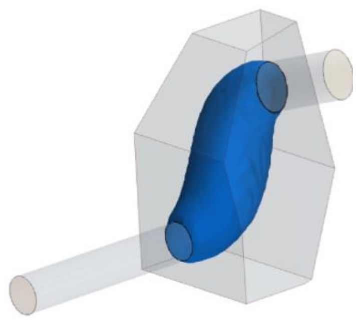

Figure 11. High viscosity optimized geometry for model 2.

As can be seen, in this case, the final geometry now has a shorter path with a wider cross section. This happens because, in the first iterations, when the viscosity was higher, flow behavior was closer to a laminar one, hence inertial effects were less important and viscous effects dominated the case. This makes the optimizer go for shorter paths instead of longer and curved ones. After the viscosity is changed, the main aspects of the geometry are kept, but some details at the interface solid-fluid are sculpted. This leads to a very different geometry (when compared to model 1) and with a lower pressure drop. This strategy of changing parameters during the optimization is known as a continuation technique. It helps to explore the possible different solutions the optimizer can generate and can be applied to other variables also. The continuation technique can be seen in the works [1, 7, 8].

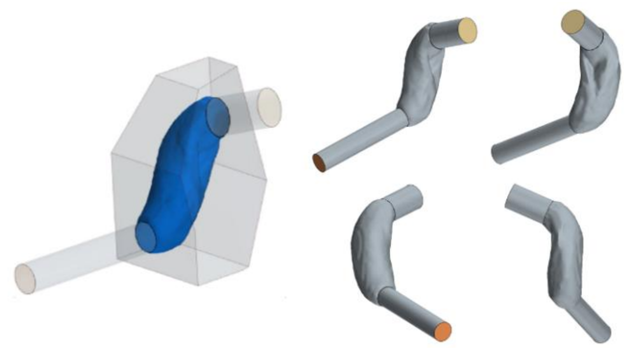

Figure 12. Optimized geometry for model 2.

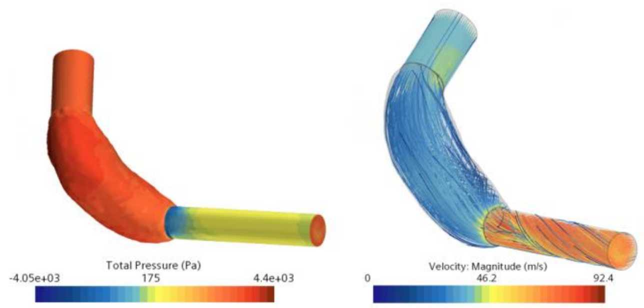

Figure 13. Total pressure (left) and streamlines (right) for the optimized model 2

CONCLUSION

In this work, the application of Topology Optimization to the design of Air Intake Systems is explored. Initially, a geometry that follows traditional ways of construction is presented and assessed by CFD. Then, by using Topology Optimization techniques, two different designs are generated. Both designs are very different from the baseline one.

The first optimized geometry shows a reduction of 52% in pressure drop when compared to the original one. Although the improvement is very interesting, the geometry is higly influenced by the initial flow, by the design domain and can be challenging to be manufactured depending on the methods available. To elaborate a new option, a continuation technique is applied where the viscosity starts with a very high value and then, after a initial shape is drawn, it assumes the real value. This final design achieves 63% of reduction with a simpler geometry, representing also a very interesting option.

The examples presented demonstrate the advantages that can be obtained by using optimization techniques in early design stages of AIS components. Additionally, it is worth to mention that, beyond the improvements in pressure drop, the use of optimization algorithms that are already implemented in the available softwares can reduce the development time, since all the iterations are performed automatically and no manual intervention is need between each design change.

Future work may include multi-objective functions, including other parameters of interest as uniformity in some sections, max velocities and flow swirls. Also, the use of complete systems when performing the optimization can help generate more interesting design options that already take into account the existence of other components.

REFERENCES

1. DILGEN, C. B.; DILGEN, S. B.; FUHRMAN, D. R.; SIGMUND, O.; LAZAROV, B. S. *Topology Optimization of Turbulent Flows.* **Computer Methods in Applied Mechanics and Engineering**, 2018.

2. OTHMER, C. *A continuous adjoint formulation for the computation of topological and surface sensitivities of ducted flows*. **International Journal for Numerical Methods in Fluids**, 2008.

3. PAPOUTSIS-KIACHAGIAS, E. M.; GIANNAKOGLOU, K. C. *Continuous adjoint methods for turbulent flows, applied to shape and topology optimization: industrial applications*. **Archives of Computational Methods in Engineering**, 2016.

4. Siemens Digital Industries Software, *Star-CCM+ User Guide*, 2021.

5. KINGMA, P. D.; BA, J. L. *Adam: a method for stochastic optimization*. ICLR 2015.

6. WRIGHT, S.; NOCEDAL, J. *Numerical Optimization*, 1999.

7. BORVALL, T.; PETERSSON, J.; *Topology optimization of fluids in Stokes flow*. **International Journal for Numerical Methods in Fluids**, 2003.

8. OKUBO JR, C . M.; KIYONO, C. Y.; SÁ, L. F. N., SILVA, E. C. N. *Topology optimization applied to 3D rotor flow path design based on the contiuous adjoint approach.* **Computers and Mathematics with Applications**, 2021.