CFD INVESTIGATION OF FLOW FORCES ON A SAFETY VALVE

Tamás Pusztai
assistant lecturer, University of Miskolc, Institute of Energy Engineering and Chemical Machinery
3515 Miskolc, Miskolc-Egyetemváros, e-mail: tamas.pusztai@uni-miskolc.hu

Viktória Mikáczó
associate professor, University of Miskolc, Institute of Energy Engineering and Chemical Machinery
3515 Miskolc, Miskolc-Egyetemváros, e-mail: viktoria.mikaczo@uni-miskolc.hu

Balázs Szőke
MSc student, University of Miskolc, Institute of Energy Engineering and Chemical Machinery
3515 Miskolc, Miskolc-Egyetemváros, e-mail: leonfit26@gmail.com

Abstract
In this article, authors investigated a DN50/80 size safety valve with computer fluid dynamics (CFD) simulation to determine the lift forces and mass flows acting on the valve disc at 3.5 to 5.5 bar_g inlet pressures and to observe the evolution of turbulent streamlines in the outlet side of the valve, with static disc position. The simulation results were compared with measured data.
Authors studied the inner flow field of the valve with fixed 5 mm disc elevation. Lift force acting the disc and overall stream pattern were investigated with the gas vale. To eliminate backflow, authors applied a virtual streamtube on the outlet nozzle with various lengths. Differences between data of the valve measured and simulated results were investigated besides changing the length (50, 150, 250, 350 and 400 mm) of the applied tube.

Keywords: *flow force, safety valve, lift force, CFD, Fluent, simulation*

1. Introduction

The risk of disasters in the chemical industry can be significant due to the hazardous substances and complex systems involved. To avoid them, engineers implement prudent safety standards for the operation of each plant, fulfilling specific guidelines and levels of protection. Various methods can be used to ensure stable operations, and it has three levels: primary, secondary, and tertiary protection. Since probability of the occurrence cannot be eliminated perfectly, tertiary protection is essential in technologies.

On vessels where the pressure load could increase in case of any fault, pressure safety valves should be installed. Where characteristic of pressure changing is steeper, rupture panels or discs should be applied (Hellemans, 2009).

Nowadays, engineers have some opportunities to investigate safety equipment before installation or even manufacturing. Various simulation techniques and software can be used to study the designed industrial systems and their response to both external and internal effects. Besides, computer fluid dynamics (CFD) methods give the opportunity to a detailed study on flow through safety valves (Scuro et al., 2018). One of its methods is the investigation with static disc position (Siménfalvi et al., 2016), which applies steady simulation in fixed geometry to study lift forces and flow field on a given valve geometry. Dynamic modelling is a more complex way to simulate the working process, however, instabilities can be predicted with this type of method (Darby, 2013; Champeneys et al., 2014.; Siménfalvi, 2017).

Use of simulation techniques help to reveal possible malfunctions in the right time, which helps minimizing losses in case of an incident.

In this article, authors investigated a DN50/80 size safety valve with computer fluid dynamics (CFD) simulation to determine the lift forces and mass flows acting on the valve disc at 3.5 to 5.5 bar_g inlet pressures and to observe the evolution of turbulent streamlines in the outlet side of the valve, with static disc position. The relevance of this valve is its widespread use in industrial practice. The changes of the streamlines in the flow field were monitored with a theoretical streamtube connected to the valve outlet. The simulation results were compared with the data provided by the measurements.

2. Turbulence models

During the setup of a CFD simulation, numerous methods can be found for modelling a turbulent flow. These methods can be categorized according to their fundamental equations, as a DNS, LES, or RANS models. The Direct Numerical Simulation (DNS) model implements fluctuated values into the Navier-Stokes equation without any turbulence submodel. Large Eddy Simulation (LES) model solves a clarified Navier-Stokes equation used for large-scale eddies. This is an average turbulence model between DNS and Reynolds-averaged Navier-Stokes (RANS) model. RANS is a mathematical model based on average values of variables. The numerical simulation is driven by a turbulence model which is arbitrarily selected to find out the effect of turbulence fluctuation on the mean fluid flow. RANS method is the commonly used method to modelling turbulent flow. In the RANS method several turbulence models are available for the simulations. Most of the times, two-equation models are used as k- ε, k- ω or the variations of these models. One goal of this study is to compare the simulated flow force results of different two equation models with measured data and find a well working turbulence model on a safety valve.

2.1. The k-ε turbulence model

The first commonly used turbulence model was the k- ε model where k is the turbulent kinetic energy (see Eq. (1)) and ε is the turbulent dissipation rate (see Eq. (2)). In equation (1), the *u*, *v* and *w* are the velocity component of the three coordinate directions. In equation (2), *D* is the characteristic size of the flow space.

$$k = \frac{1}{2}(\overline{u'^2} + \overline{v'^2} + \overline{w'^2}) \tag{1}$$

$$\varepsilon = \frac{k^{3/2}}{0.3D} \tag{2}$$

Short overview of the k-ε turbulence model is following:

- two-equation model,
- uses wall functions,
- good convergence and low memory requirements,
- convenient,

- limitation (not accurate for no-slip walls, adverse pressure gradients, strong curvature into flow and jet flows).

The k-ε turbulence model also has some submodels, as standard k- ε, RNG k- ε or realizable k- ε models. Several authors (Song et al., 2013; Erdődi and Hős, 2017) are used standard k- ε model on them study.

2.2. The k-ω turbulence model

The k-ε turbulence model has some insufficiencies in certain cases. The model performs particularly well far from the wall conditions in the main flow, but in the boundary layer zones, it encounters problems. The model is inadequate especially in case of flows with low Reynolds-numbers, which value decreases approaching the wall, so the walls no-slip condition exerts its effect. The establishment of the k-ω turbulence model brought significant progress in CFD simulations, which replaces ε transport equation with a certain ω turbulence dissipation velocity. Characteristics of the new model are the following (Solmaz, 2023):

- two-equation model,
- solvation of the ω function is easier than to solve the ε function,
- uses wall functions,
- good convergence and low memory requirements,
- similar to k-ε model, but more accurate for internal flows, curvatures, separated flows and jets,
- limitations: sensitive to initial conditions.

2.3. The k-ω SST turbulence model

The abbreviation SST covers Shear Stress Transmission. The k-ω SST model provides a better prediction of flow separation than most RANS models, and this explains its good behavior under adverse pressure gradients. It can consider the transport of the principal shear stress in boundary layers with unfavorable pressure gradients. It is the other commonly used model in the industry, thanks to its favorable cost-to-benefit ratio (SimScale, 2022). The main characteristics of the model are:

- two-equation model (turbulent kinetic energy and dissipation),
- uses wall functions,
- the combination of k-ε (in the outer region of the boundary layer and outside it) and k-ω models (in the inner boundary layer),
- separated flows and jets,
- limitations: difficulties in convergency.

3. The studied geometry

The studied geometry was a full lift safety valve in DN50/80 size. The investigated lower section of the valve includes the inlet and outlet nozzle, the body, the spindle, the seat, and the disc (see Figure 1).

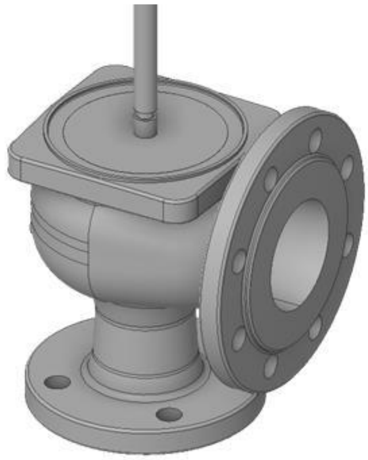


Figure 1. *3D model of the studied DN50/80 safety valve*

Authors studied the inner flow field of the valve with fixed 5 mm disc elevation. In order to shorten the simulation time, half of the geometry has been considered, with the flow field formed from the interior of the geometry. Left side of Figure 2 shows the half geometry of the solid metal, and on the right side, the flow field generated from the interior space can be observed.

Simulation preparation were performed in Ansys software environment, and for CFD simulations, Ansys Fluent were used.

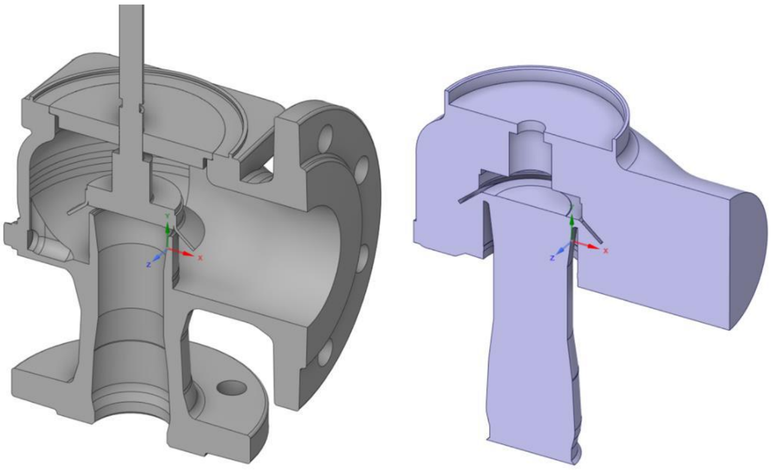
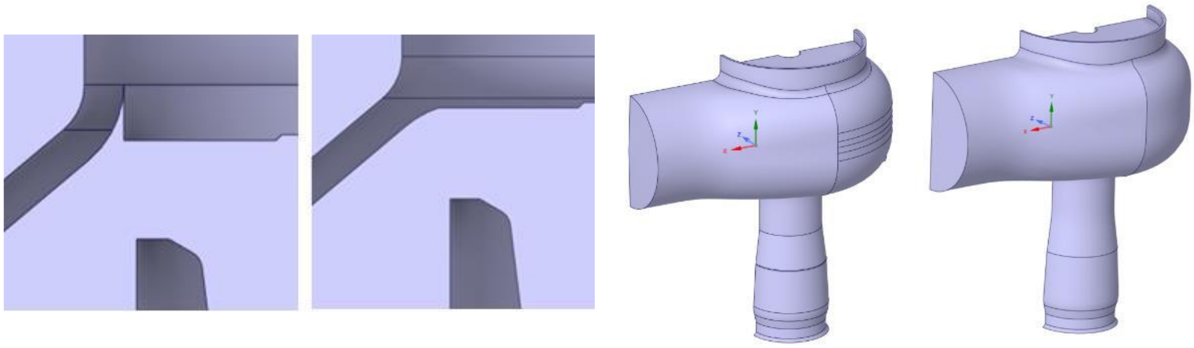


Figure 2. *Half geometry of the solid valve and the interior space*

In order to simplify the future mesh, rounding sharp edges of the geometry and merging of separated surfaces were performed in SpaceClaim environment. Example of rounding can be seen on Figure 3/a and Figure 3/b shows merging of separated surfaces.



a) Rounding sharp edges (left: before, right: after)

b) Merging separated surfaces (left: before, right: after)

Figure 3. *Examples for rounding sharp edges and merging surfaces of the geometry*

4. The simulation method

The CFD software Ansys Fluent was used to perform the simulations on the safety valve. Lift force acting the disc and overall stream pattern were investigated with the gas vale. In the three-dimensional numerical simulation, the medium was air, and assumed as ideal gas. Based on fluid mechanics, the governing equations were established as mass conservation, energy conservation and momentum conservation.

As shown on Figure 4, half of the inner valve geometry was modelled. Effect of gravity was considered during the simulation, and heat transfer through the wall was neglected. For the presented steady-state simulations, k-ε and k-ω turbulence models were used as these models give reliable results for the zones with high Reynolds-numbers, too.

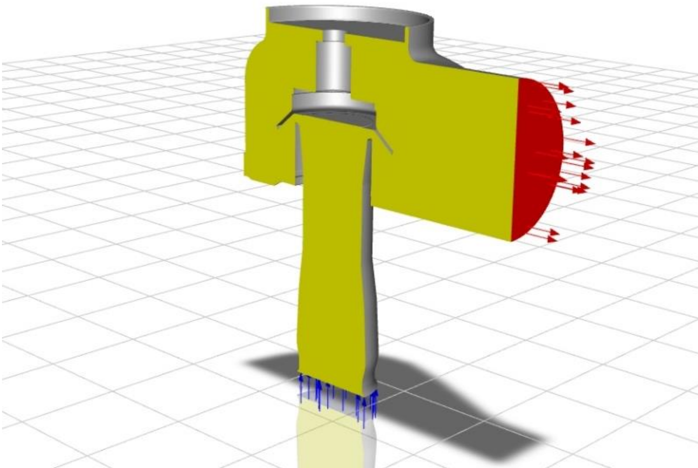


Figure 4. *Named regions of the geometry*

Boundary conditions were set as the following:

- inlet pressure was changed between 3.5 and 5.5 bar_g in each simulation (blue arrows on Figure 4);

- outlet pressure was ambient (red arrows and surface on Figure 4);
- wall condition was applied to the inner wall of the body and the surface of the disc, without heat exchange (grey surfaces on Figure 4).

The symmetry of the model was considered with "symmetry" condition (ochre coloured surface on Figure 4).

On the flow-wise critical regions, refinement of the mesh had to be increased. One of these critical volumes was the environment of the opening gap between the seat and the disc, where vena contracta can be formed. To achieve the desired refinement, an auxiliary coordinate system was fitted to the upper middle of the valve disc. The body sizing was set in its origin as a 50 mm radius sphere (Figure 5). The size of the whole-body mesh was set to 2.5 mm. Inside the 50 mm radius sphere, the mesh size was set to 0.7 mm.

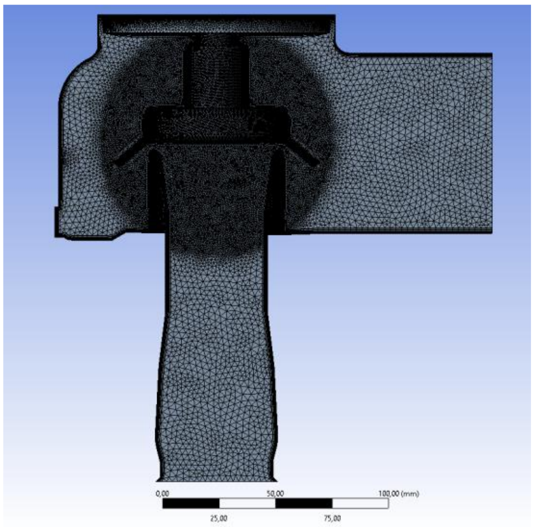


Figure 5. *Mesh refining with body sizing*

The inflation layer next to the walls (the inner wall of the body and the surface of the disc) was applied in 10 layers. The thickness of the first layer was 0.09 mm, and 1.2 increase was adjusted per layer.

The set 10^{-6} convergence tolerance value of the iterations has proved to be sufficient. Before performing the simulations, hybrid initialization was applied. In the majority of the cases, a satisfactory solution was obtained with approximately 500 iterations.

5. Results

In the previous sections, the aims of the study and the setups of the simulations were defined. Firstly, the turbulence model dependency study was performed where a well-working turbulence model was defined. On the following simulations were performed only with the chosen turbulence model.

5.1. Turbulence model dependency results

With the mentioned setups more than 25 simulations were performed with different inlet pressure conditions. During simulations, the disc displacement was fixed at 5mm. The inlet pressure was changed from 3.5 bar_g to 5.5 bar_g with a 0.5 bar_g step. The simulated lift force results with different turbulence models compared to the measured data are shown in the Table 1, and the measured data with the simulated data are shown in separated diagrams on Figure 6.

Table 1. *Measured and simulated lift force data*

Pressure [bar_g]	Measured data [N]	Standard k-ω [N]	k-ω SST [N]	Standard k-ε [N]	Realizable k-ε [N]
3.5	1076.7	1072.9	1089.1	1061.4	1090.7
4	1220.2	1225.2	1248.7	116.1	1226.9
4.5	1366.7	1325	1366.1	1328.9	1361.6
5	1512.8	1479.7	1499.3	1464.5	1496.6
5.5	1665.1	1633.5	1670.2	1603.2	1633.3

The relative error compared to the measured data are presented in Table 2. The table shows that all the tolerances were under 5%. From the results can be seen, that the best convergence was reached with the k-ω SST turbulence model. Therefore, only the k-ω SST turbulence model was used during the further simulations.

Table 2. *Tolerances of the results between measured and simulated data*

Pressure [bar_g]	Standard k-ω [%]	k-ω SST [%]	Standard k-ε [%]	Realizable k-ε [%]
3.5	-0.35	1.15	-1.42	1.29
4	0.41	2.34	-1.97	0.55
4.5	-3.05	-0.04	-2.76	-0.37
5	-2.19	-0.89	-3.19	-1.07
5.5	-1.9	0.3	-3.72	-1.91
Abs. error	7.9	4.72	13.06	5.19

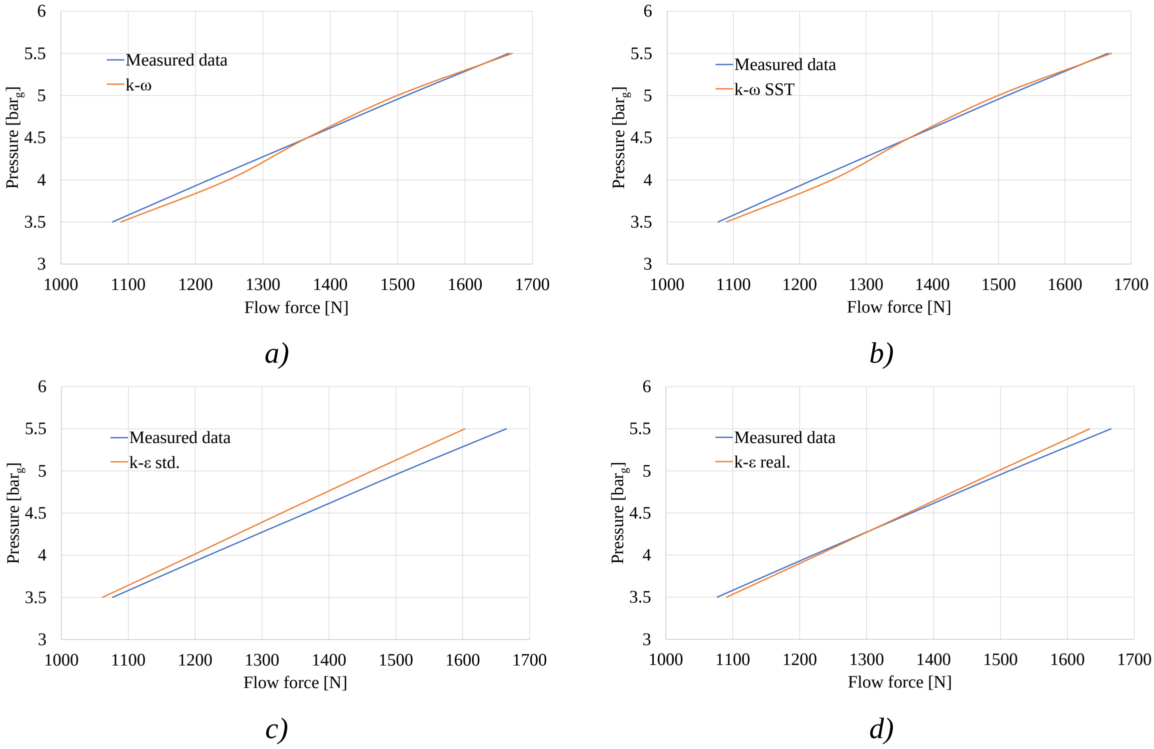


Figure 6. *Flow force results compared with measured data using different turbulent models*

5.2. Investigation of the backflow

In simulation studies of flows on safety valves, backflow through the outlet nozzle often causes numerical error. Its cause is that the outflow is rarely laminar, so the streamlines may reverse after the outflow surface. This phenomenon was also encountered while running the current simulations, so this needed further investigation.

To eliminate backflow, authors applied a virtual streamtube on the outlet nozzle with various lengths. Differences between data of measured and simulated results were investigated besides changing the length (50, 150, 250, 350 and 400 mm) of the applied tube.

After the extension of the geometry in every case, the mesh had to be regenerated and boundary zones had to be redefined. Simulations were performed with k-ω SST turbulence model, and results were recorded at 4 and 5 bar$_g$ inlet pressures.

Table 3. *Simulated lift force besides different length of virtual pipes*

Pressure [bar$_g$]	50 mm long tube	150 mm long tube	250 mm long tube	350 mm long tube	400 mm long tube
4	1252.23 (-0.51%)	1222.86 (1.85%)	1264.06 (-1.46%)	1262.202 (-1.31%)	1228.02 (1.43%)
5	1542.804 (-1.19%)	1508.66 (1.05%)	1529.21 (-0.3%)	1514.912 (0.64%)	1527.78 (-0.2%)

Table 3 shows the effect of the tube length to the simulated lift force. Deviancy of the results can be observed in the round brackets. As the results show, every investigated tube length gave adequate results. In further investigations, the minimal 50 mm length tube would be enough to eliminate the backflow problem.

5.3. Investigation of mesh independency

Mesh independency investigations needed for numerical simulations, were performed in case of 3.5 bar_g inlet pressure without additional stream tube, in case of each investigated turbulence models. Element size was halved during this study. The results are shown in Table 4, compared to the measured data. In the brackets after the simulated values, absolute deviations are indicated related to the measured data.

Table 4. *Simulated lift force besides different length of virtual pipes*

Measured data [N]	k-ω SST [N]	Mesh independency with k-ω SST
1076.744	1089.142 (1.15%)	1095.668 (1.76%)

As the results show, the evaluation of the mesh independency test differed within 2% tolerance from the calculated values with the coarser mesh structure. Thus, it can be stated that the adequate sensitivity of the original mesh resolution has been proven.

6. Conclusions

During this study, authors investigated the modelling stability of a safety valve with CFD simulations.

Different turbulence models were investigated on a safety valve with pre-defined boundary conditions.

In the first step, the simulated lift force values were compared with measured ones to validate the simulation results. The compared data were in an excellent convergence. According to the simulation results, the k-ω SST turbulence model was the most stable one. Further investigation was performed with only k-ω SST turbulence model. Backflow effect was studied on safety valve with various streamtube length. With the 50 mm long streamtube, the backflow effect was disappeared. For the stability of the simulations, it is recommended to use at least 50 mm long streamtube at the outlet side of the safety valves during CFD simulation.

Mesh independence tests were performed on the simulation model to prove the mesh goodness. Used mesh qualities were acceptable for the simulation.

References

[1] Hellemans, M. (2009). *The safety relief valve handbook*. Butterworth-Heinemann, Oxford. **https://doi.org/10.1016/B978-1-85617-712-2.00009-3**

[2] Scuro, N. L., Angelo, E., Angelo, G., Andrade. D. A. (2018). A CFD analysis of the flow dynamics of a directly-operated safety relief valve. *Nucl. Eng. Des.*, 328, 321–332. **https://doi.org/10.1016/j.nucengdes.2018.01.024**

[3] Siménfalvi, Z., Spisák, B., and Szepesi, L. G. (2016). Rugóterhelésű biztonsági szelep kísérleti és szimulációs vizsgálata. *GÉP*, 67(3), 34–37.

[4] Darby, R. (2013). The dynamic response of pressure relief valves in vapor or gas service, part I : Mathematical model. *J. Loss Prev. Process Ind.*, 26, 1262–1268. **https://doi.org/10.1016/j.jlp.2013.07.004**

[5] Champeneys, A. R., Paul, K., McNeely, M., Hős, Cs. J.(2014). Dynamic behavior of direct spring loaded pressure relief valves in gas service : Model development, measurements and instability mechanisms. *J. Loss Prev. Process Ind.*, 31, 70–81. **https://doi.org/10.1016/j.jlp.2014.06.005**

[6] Siménfalvi., Z.: Investigation of flow-force characteristics for spring-loaded safety valve dynamic modelling, 2006 *CHISA 2006 - 17th Int. Congr. Chem. Process Eng*.

[7] Song, X. G., Park, Y. C., Park, J. H. (2013). Blowdown prediction of a conventional pressure relief valve with a simplified dynamic model. *Mathematical and Computer Modelling*, 57, 279–288. **https://doi.org/10.1016/j.mcm.2011.06.054**

[8] Erdődi, I., and Hős, C. (2017). Prediction of quarter-wave instability in direct spring operated pressure relief valves with upstream piping by means of CFD and reduced order modelling. *J. Fluids and Structures*, 73, 37–52. **https://doi.org/10.1016/j.jfluidstructs.2017.05.003**

[9] Solmaz, S. (2023, June 01). *SimScale Blog. Turbulence: Which model should I select for my CFD analysis?* **https://www.simscale.com/blog/turbulence-cfd-analysis/?fbclid=IwAR3NMRPngRQqKI3-lKVJlQfT5p0rzqaFr19V9z4BmUqpJkn9t6fv3tXmAiA**

[10] SimScale, (2022, November 10). *Documentation*. **https://www.simscale.com/docs/simulation-setup/global-settings/k-omega-sst/**