University of Natural Resources and Life Sciences, Vienna
Department of Water, Atmosphere and Environment

Introduction to OpenFOAM®

Daniel Wildt

SWARM Summer School
15 – 26 November 2021

23rd November 2021

Outline I

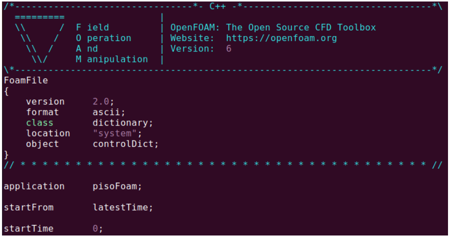
University of Natural Resources and Life Sciences, Vienna
Department of Water, Atmosphere and Environment

Open∇FOAM®

General information about OpenFOAM®

University of Natural Resources and Life Sciences, Vienna
Department of Water, Atmosphere and Environment

- **Open**-source **F**ield **O**peration **A**nd **M**anipulation
- open-source collection of CFD solvers
- includes also pre- and postprocessing tools
- no graphical user interface
- other external software used for displaying results, e. g.
  - Paraview
  - Gnuplot



```
/*--------------------------------*- C++ -*----------------------------------*\
  =========                 |
  \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox
   \\    /   O peration     | Website:  https://openfoam.org
    \\  /    A nd           | Version:  6
     \\/     M anipulation  |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    location    "system";
    object      controlDict;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

application     pisoFoam;

startFrom       latestTime;

startTime       0;
```

General information about OpenFOAM® – histroy

University of Natural Resources and Life Sciences, Vienna

- development started by Henry Weller in 1989 at Imperial College London
- commercially offered from 2000 to 2004 by Nabla Ltd. as FOAM
- Nabla Ltd. folded in 2004, code offered as OpenFOAM under GNU General Public License
  - Henry Weller, Chris Greenshields and Mattjis Janssens founded OpenCFD Ltd.
  - Hrvoje Jasak founded consulting company Wikki Ltd.

Wikipedia contributors (2021)

General information about OpenFOAM® – today

University of Natural Resources and Life Sciences, Vienna
Department of Water, Atmosphere and Environment

Two parallel version today

- Foundation
  - `https://cfd.direct`; download from `https://openfoam.org`
  - ".org-version"
  - current verion: OpenFOAM 9
- ESI group
  - `https://www.openfoam.com/`
  - ".com-version"
  - current version OpenFOAM v2106

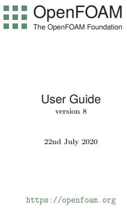
CFD Direct

get it right

General information about OpenFOAM®

University of Natural Resources and Life Sciences, Vienna
Department of Water, Atmosphere and Environment

- no support from developers
- trainings offered by different groups but expensive
- official user guide relatively short
- additional programmers and tutorials guide available
- large set of readily set-up tutorial cases covering most of OpenFOAMs' functionality


OpenFOAM
The OpenFOAM Foundation

User Guide
version 8

22nd July 2020

https://openfoam.org


OpenFOAM
The Open Source CFD Toolbox

Programmer's Guide

Version v1806
4th July 2018

General information about OpenFOAM®

University of Natural Resources and Life Sciences, Vienna
Department of Water, Atmosphere and Environment

A selection of solvers (OpenCFD, Ltd., 2020c):

- basic e. g.
  - `laplacianFoam`
  - `scalarTransportFoam`
- compressible
- incompressible
  - `icoFoam`
  - `pimpleFoam`
  - `pisoFoam`
  - `shallowWaterFoam`
  - `simpleFoam`
- lagrangian
- multiphase
  - stressAnalysis
  - `interFoam`
  - `driftFluxFoam`
- financial

check User Guide (OpenCFD, Ltd., 2020c) for full details and full list of solvers

General information about OpenFOAM®

University of Natural Resources and Life Sciences, Vienna
Department of Water, Atmosphere and Environment

Working with OpenFOAM

- cases set-up by defining variables in dictionaries (textfiles), e. g.:
  - `p`, `U`, . . . for BCs
  - `controlDict` for timestep, etc.
- basic case structure includes folders
  - `0` (initial/bound. cond.)
  - `constant`
  - `system`

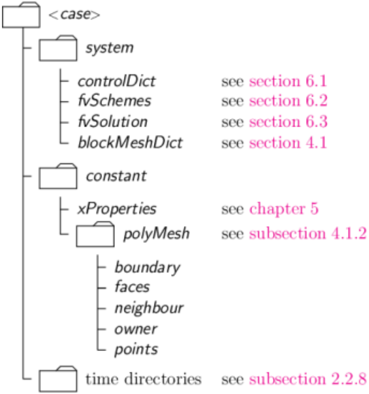


Fig.: OpenCFD, Ltd. (2020c)

Tutorials: Lid-driven cavity

University of Natural Resources and Life Sciences, Vienna
Department of Water, Atmosphere and Environment

The “Hallo World” case of OpenFOAM

- create mesh: `blockMesh`
- view mesh in Paraview: `paraFoam`
- check mesh: `checkMesh`
- run solver: `icoFoam`
- post-process in Paraview

check Tutorial Guide (OpenCFD, Ltd., 2020b) for details

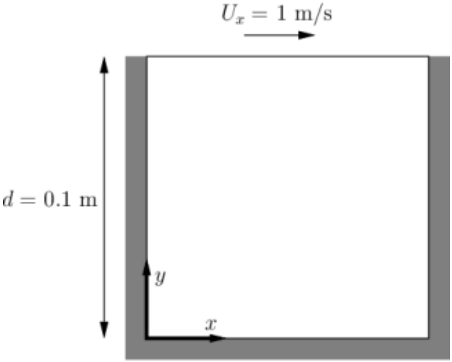


Fig.: OpenCFD, Ltd. (2020b)

Tutorials: Lid-driven cavity

University of Natural Resources

- Meshing using `blockMesh`
- mesh and geometry defined in `system/blockMeshDict`

for details check User Guide (OpenCFD, Ltd., 2020c) section "Mesh generation and convertion"

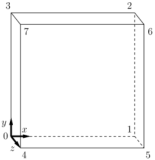


Fig.: OpenCFD, Ltd. (2020b)

```
/*--------------------------------*- C++ -*----------------------------------*\
  =========                 |
  \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox
   \\    /   O peration     | Website:  https://openfoam.org
    \\  /    A nd           | Version:  8
     \\/     M anipulation  |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    object      blockMeshDict;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

convertToMeters 0.1;

vertices
(
    (0 0 0)
    (1 0 0)
    (1 1 0)
    (0 1 0)
    (0 0 0.1)
    (1 0 0.1)
    (1 1 0.1)
    (0 1 0.1)
);

blocks
(
    hex (0 1 2 3 4 5 6 7) (20 20 1) simpleGrading (1 1 1)
);

edges
(
);

boundary
(
    movingWall
    {
        type wall;
        faces
        (
            (3 7 6 2)
        );
    }
    fixedWalls
    {
```

Tutorials: Lid-driven cavity

University of Natural Resources and Life Sciences, Vienna
Department of Water, Atmosphere and Environment

Other important dictionaries:

- boundary and initial conditions `0/p`, `0/U`
- physical properties `constant/transportProperties`
- discretization `system/fvScheme`
- solver settings `system/fvSolution`
- control `system/controlDict`
  - transient solvers set time step `deltaT`
  - steady state solvers time step corresponds to solver iteration → set `deltaT 1`

```
/*--------------------------------*- C++ -*----------------------------------*\
  =========                 |
  \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox
   \\    /   O peration     | Website:  https://openfoam.org
    \\  /    A nd           | Version:  8
     \\/     M anipulation  |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    location    "system";
    object      controlDict;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

application     icoFoam;

startFrom       startTime;

startTime       0;

stopAt          endTime;

endTime         0.5;

deltaT          0.005;

writeControl    timeStep;

writeInterval   20;

purgeWrite      0;

writeFormat     ascii;

writePrecision  6;

writeCompression off;

timeFormat      general;

timePrecision   6;

runTimeModifiable true;


// ************************************************************************* //
```

Tutorials: Dam break

University of Natural Resources and Life Sciences, Vienna
Department of Water, Atmosphere and Environment

A multiphase case, using the Volume of Fluid method (Euler-Euler)

- phase variable `alpha`
  - air: $\alpha = 0$; water: $\alpha = 1$
- initial field defined in `system/setFieldsDict`
- initialise using `setFields`

check Tutorial Guide (OpenCFD, Ltd., 2020b) for details

possible tasks: refine mesh, extend simulation time

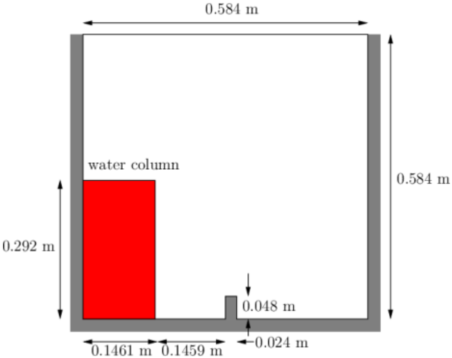
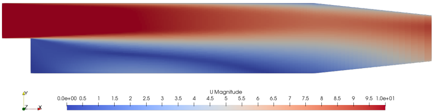


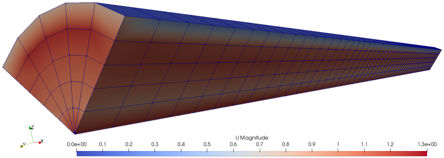
Fig.: OpenCFD, Ltd. (2020b)

Tutorials: Other tutorials

University of Natural Resources and Life Sciences, Vienna
Department of Water, Atmosphere and Environment

- steady state turbulent flow `incompressible/simpleFoam/`
  - pipe flow: `pipeCyclic`
  - backward facing step: `pitzDaily`
- transient turbulent flow: `incompressible/pisoFoam/`
  - cavity: `cavity`





see Tutorial Wiki by OpenFOAM Community (2021) at `https://wiki.openfoam.com/Main_Page`

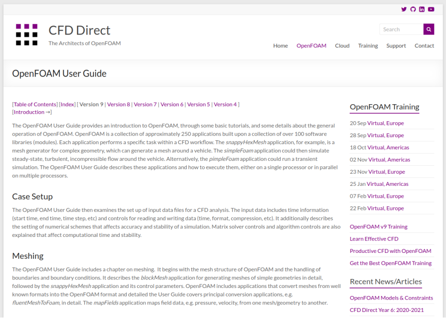
Literature and resources: Documentations by distributors

University of Natural Resources and Life Sciences, Vienna
Department of Water, Atmosphere and Environment

OpenFOAM v9 by The OpenFOAM Foundation (`www.openfoam.org` and `http://cfd.direct/`)

The OpenFOAM Foundation (2021[c]). *OpenFOAM v9 User Guide*. URL: `https://cfd.direct/openfoam/user-guide/` (visited on 28/07/2021)

The OpenFOAM Foundation (2021b). *OpenFOAM v9 C++ Source Code Guide*. URL: `https://cpp.openfoam.org/v9/` (visited on 28/07/2021)

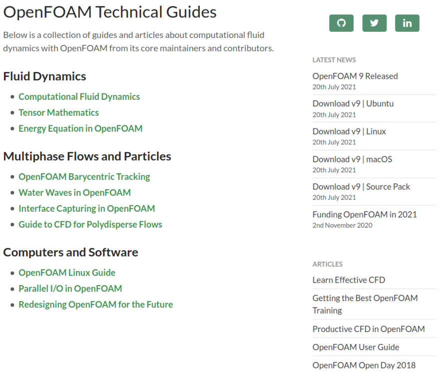


Literature and resources: Documentations by distributors

University of Natural Resources and Life Sciences, Vienna
Department of Water, Atmosphere and Environment

Technical Guides by The OpenFOAM Foundation

The OpenFOAM Foundation (2021a). *OpenFOAM Technical Guides*. URL: `https://openfoam.org/guides/` (visited on 28/07/2021)



OpenFOAM Technical Guides

Below is a collection of guides and articles about computational fluid dynamics with OpenFOAM from its core maintainers and contributors.

Fluid Dynamics

- Computational Fluid Dynamics
- Tensor Mathematics
- Energy Equation in OpenFOAM

Multiphase Flows and Particles

- OpenFOAM Barycentric Tracking
- Water Waves in OpenFOAM
- Interface Capturing in OpenFOAM
- Guide to CFD for Polydisperse Flows

Computers and Software

- OpenFOAM Linux Guide
- Parallel I/O in OpenFOAM
- Redesigning OpenFOAM for the Future

LATEST NEWS

OpenFOAM 9 Released
20th July 2021

Download v9 | Ubuntu
20th July 2021

Download v9 | Linux
20th July 2021

Download v9 | macOS
20th July 2021

Download v9 | Source Pack
20th July 2021

Funding OpenFOAM in 2021
2nd November 2020

ARTICLES

Learn Effective CFD

Getting the Best OpenFOAM Training

Productive CFD in OpenFOAM

OpenFOAM User Guide

OpenFOAM Open Day 2018

Literature and resources: Documentations by distributors

University of Natural Resources and Life Sciences, Vienna
Department of Water, Atmosphere and Environment

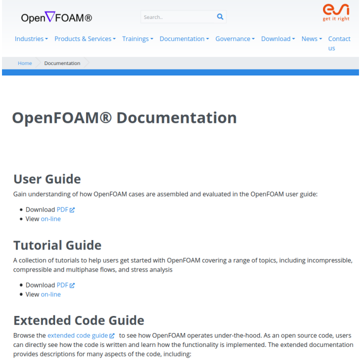
OpenFOAM v2012 by ESI Group (`www.openfoam.com`)

OpenCFD, Ltd. (2020c). *OpenFOAM User Guide*. URL: `https://www.openfoam.com/documentation/user-guide` (visited on 26/07/2021)

OpenCFD, Ltd. (2020b). *OpenFOAM Tutorial Guide*. URL: `https://www.openfoam.com/documentation/tutorial-guide` (visited on 26/07/2021)

OpenCFD Ltd. (2021). *OpenFOAM Programmer's Guide*.

OpenCFD, Ltd. (2020a). *OpenFOAM Extended Code Guide*. URL: `https://www.openfoam.com/documentation/guides/latest/doc/` (visited on 28/07/2021)

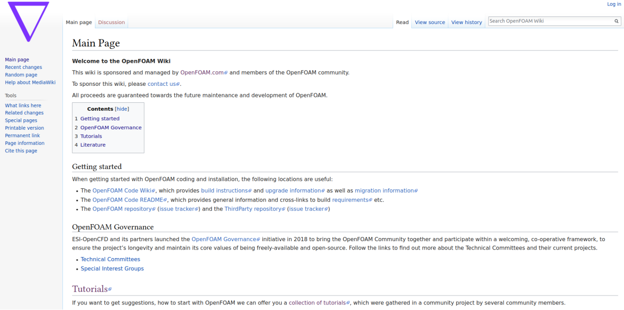


Literature and resources: Wikkis and useful tools

University of Natural Resources and Life Sciences, Vienna
Department of Water, Atmosphere and Environment

OpenFOAM tutorials collection and a “3 weeks self-learning course”:

OpenFOAM Community (2021). *Tutorial Wiki*. Ed. by OpenCFD, Ltd. URL: https://wiki.openfoam.com/Main_Page (visited on 28/07/2021)



Log in

Main page | Discussion | Read | View source | View history | Search OpenFOAM Wiki

Main page
Recent changes
Random page
Help about MediaWiki

Tools
What links here
Related changes
Special pages
Printable version
Permanent link
Page information
Cite this page

Main Page

Welcome to the OpenFOAM Wiki

This wiki is sponsored and managed by OpenFOAM.com and members of the OpenFOAM community.

To sponsor this wiki, please contact us.

All proceeds are guaranteed towards the future maintenance and development of OpenFOAM.

Contents [hide]

Getting started

When getting started with OpenFOAM coding and installation, the following locations are useful:

- The OpenFOAM Code Wiki, which provides build instructions and upgrade information as well as migration information
- The OpenFOAM Code README, which provides general information and cross-links to build requirements etc.
- The OpenFOAM repository (issue tracker) and the ThirdParty repository (issue tracker)

OpenFOAM Governance

ESI-OpenCFD and its partners launched the OpenFOAM Governance initiative in 2018 to bring the OpenFOAM Community together and participate within a welcoming, co-operative framework, to ensure the project’s longevity and maintain its core values of being freely-available and open-source. Follow the links to find out more about the Technical Committees and their current projects.

- Technical Committees
- Special Interest Groups

Tutorials

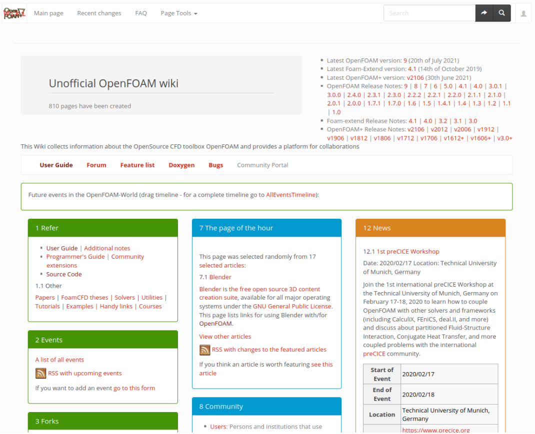
If you want to get suggestions, how to start with OpenFOAM we can offer you a collection of tutorials, which were gathered in a community project by several community members.

Literature and resources: Wikkis and useful tools

University of Natural Resources and Life Sciences, Vienna

Unofficial OpenFOAM Wikki:

> OpenFOAM Wiki (2021). *Unofficial OpenFOAM wiki*. URL: `www.openfoamwiki.net` (visited on 06/08/2021)

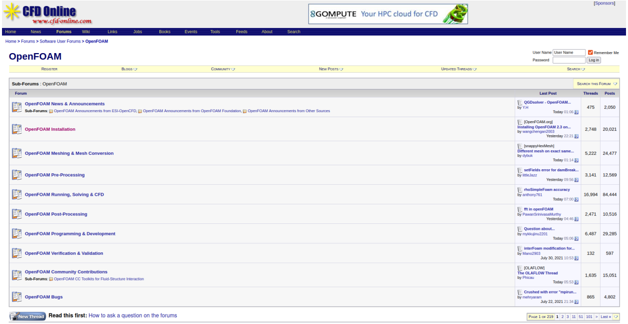


Literature and resources: Wikkis and useful tools

University of Natural Resources and Life Sciences, Vienna
Department of Water, Atmosphere and Environment

OpenFOAM Forum at CFD Online:

CFD Online (2021[a]). *OpenFOAM Forum.* URL: https://www.cfd-online.com/Forums/openfoam/ (visited on 06/08/2021)

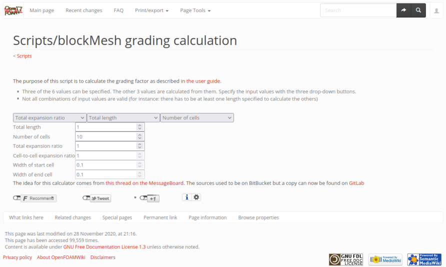


Literature and resources: Wikkis and useful tools

University of Natural Resources

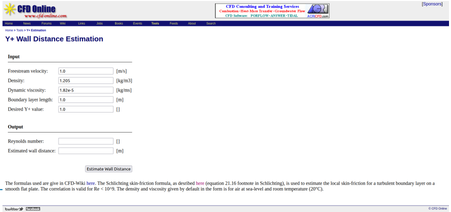
A tool for optimizing `blockMesh` grading:

> OpenFOAMWiki (2020). *Scripts/blockMesh grading calculation*. URL: https://openfoamwiki.net/index.php/Scripts/blockMesh_grading_calculation (visited on 28/07/2021)



Tool for estimating required wall resolution:

> CFD Online (2021[b]). *y+ Wall Distance Estimation*. URL: https://www.cfd-online.com/Tools/yplus.php (visited on 28/07/2021)



Literature and resources: Wikkis and useful tools

University of Natural Resources and Life Sciences, Vienna
Department of Water, Atmosphere

Prof. Håkan Nillson is offering a free course on OpenFOAM for PhD students and provides his course materials online. In the course of "The 3rd UCL OpenFOAM Workshop", 24th February 2021 he received the "OpenFOAM community contribution award" for this course.

H. Nilsson, ed. (2020). *Proceedings of CFD with OpenSource Software*. Chalmers University of Technology. URL: http://dx.doi.org/10.17196/OS_CFD#YEAR_2020

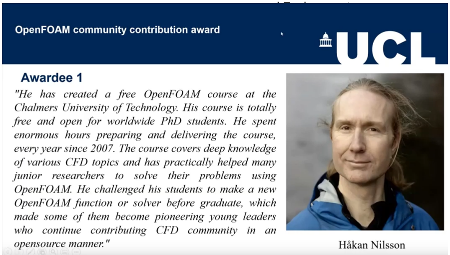


Fig.: Screenshot from the award ceremony for "The OpenFOAM community contribution award" at "The 3rd UCL OpenFOAM Workshop" on 24th February 2021

University of Natural Resources and Life Science, Vienna

Department of Water, Atmosphere and Environment
Institut of Hydraulic Engineering and River Research

Daniel Wildt, MSc

Muthgasse 107, A - 1190 Wien
Tel.: 01-47654-81935
daniel.wildt@boku.ac.at
http://www.wau.boku.ac.at/iwa/

Literature I

University of Natural Resources and Life Sciences, Vienna
Department of Water, Atmosphere and Environment

CFD Online (2021[a]). *OpenFOAM Forum*. URL: https://www.cfd-online.com/Forums/openfoam/ (visited on 06/08/2021).

CFD Online (2021[b]). *y+ Wall Distance Estimation*. URL: https://www.cfd-online.com/Tools/yplus.php (visited on 28/07/2021).

Nilsson, H., ed. (2020). *Proceedings of CFD with OpenSource Software*. Chalmers University of Technology. URL: http://dx.doi.org/10.17196/OS_CFD#YEAR_2020.

OpenCFD, Ltd. (2020a). *OpenFOAM Extended Code Guide*. URL: https://www.openfoam.com/documentation/guides/latest/doc/ (visited on 28/07/2021).

OpenCFD, Ltd. (2020b). *OpenFOAM Tutorial Guide*. URL: https://www.openfoam.com/documentation/tutorial-guide (visited on 26/07/2021).

Literature II

University of Natural Resources and Life Sciences, Vienna
Department of Water, Atmosphere and Environment

OpenCFD, Ltd. (2020c). *OpenFOAM User Guide*. URL: https://www.openfoam.com/documentation/user-guide (visited on 26/07/2021).

OpenCFD Ltd. (2021). *OpenFOAM Programmer's Guide*.

OpenFOAM Community (2021). *Tutorial Wiki*. Ed. by OpenCFD, Ltd. URL: https://wiki.openfoam.com/Main_Page (visited on 28/07/2021).

OpenFOAM Wiki (2021). *Unofficial OpenFOAM wiki*. URL: www.openfoamwiki.net (visited on 06/08/2021).

OpenFOAMWiki (2020). *Scripts/blockMesh grading calculation*. URL: https://openfoamwiki.net/index.php/Scripts/blockMesh_grading_calculation (visited on 28/07/2021).

Literature III

University of Natural Resources and Life Sciences, Vienna
Department of Water, Atmosphere and Environment

The OpenFOAM Foundation (2021a). *OpenFOAM Technical Guides*. URL: https://openfoam.org/guides/ (visited on 28/07/2021).

The OpenFOAM Foundation (2021b). *OpenFOAM v9 C++ Source Code Guide*. URL: https://cpp.openfoam.org/v9/ (visited on 28/07/2021).

The OpenFOAM Foundation (2021[c]). *OpenFOAM v9 User Guide*. URL: https://cfd.direct/openfoam/user-guide/ (visited on 28/07/2021).

Wikipedia contributors (2021). *OpenFOAM — Wikipedia, The Free Encyclopedia*. URL: https://en.wikipedia.org/w/index.php?title=OpenFOAM&oldid=1034750560 (visited on 26/07/2021).