

ResFrac FAQ

Updated: January 27, 2026

This document provides answers to frequency asked questions from ResFrac users. For further support, refer to: www.resfrac.com/onlinetraining or email us at support@resfrac.com. The ResFrac A to Z Guide contains step-by-step tutorials for first time users, and www.resfrac.com/onlinetraining has a one-hour 'getting started' video.

Table of Contents:

1. ResFrac License and Installation.....	4
Q: How can I use Microsoft Edge to download ResFracPro?.....	4
Q: How can I add and remove users from my company's ResFrac account?.....	6
Q: How do I check and monitor simulation hours used on my ResFrac Pro License?.....	9
Q: How do we add ResFrac as a Registered App in our Azure environment for SSO capabilities?.....	11
2. User-Interface and Cloud.....	13
Q: How do I log-in to the ResFrac user interface to be able to run simulations?.....	13
Q: How do I access the built-in ResFrac templates?.....	16
Q: How do I share a ResFrac simulation?.....	19
Q: How do I import a simulation into ResFrac?.....	19
Q: How can I troubleshoot issues with my simulation results downloading?.....	22
Q: How do I run ResFrac with 'Detailed Logging' for debugging?.....	28
Q: How do I specify an additional 'remote' location to store simulations?.....	31
Q: How do I save all of my simulations in a different location such as a network drive?.....	33
Q: How can I change the font size in the UI?.....	34
Q: I tried to create a new project, sandbox, or simulation, but it says that name already exists. Why?.....	35
3. Visualization (Viz) Tool.....	37
Q: How do I use the 'rotated stretch' option in the visualization tool?.....	37
Q: How can I save external datasets with my simulation to view using the visualization tool?.....	39
Q: How can I import external datasets into the visualization tool?.....	40
Q: Can I plot the gun barrel summary of simulation results?.....	45

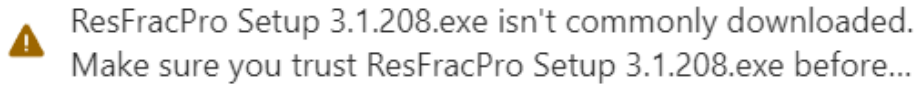
4. Simulator Inputs and Outputs.....	46
Q: I can't find some input parameters in the builder. How can I show these if they are hidden?.....	46
Q: How can I reduce the size of my simulation downloads?.....	46
Q: Where can I find a list of all outputs in the 3D viz view as well as the line plots (sim_track) file?.....	50
Q: How do I change the frequency of simulation outputs?.....	51
Q: How do I use the Economics Navigator?.....	52
Q: What is in the Raw_Res folder (nested in the Results folder of a simulation)?.....	55
Q: How do I export properties from a Petrel pseudowell to be used with the 'Properties versus depth wizard'?.....	58
Q: How can I refresh an old simulation with the most recent recommended settings?.....	59
Q: What is the fracture element fill ratio?.....	63
Q: How do I determine the true area of a fracture element using the fill ratio?.....	64
Q: How do I calculate the mass of proppant in a tip fracture element?.....	64
Q: Do the median and max fracture lengths and heights reported in the sim track file account for the fill ratio?.....	64
Q: If a fracture element spans multiple static model layers, what properties from the Geological Units (Facies List) table apply to that fracture element?.....	64
Q: What parameters are ok to change in a simulation continuing from a restart file?.....	65
Q: Why use restart files?.....	67
Q: How do I download restart files and run a simulation from a restart?.....	67
Q: Why is my treating pressure (WHP) very high at the beginning of the simulation?.....	70
Q: Why does my well keep reaching the maximum WHP and not creating any fractures?.....	71
Q: How is embedment handled in ResFrac?.....	71
Q: Does ResFrac handle proppant crushing?.....	73
Q: Where should I center my zero perm cube when using the Zero Perm Inside Cube Option when I want to simulate well spacing effects from wells that are not in the model, and can it extend beyond the edge of the matrix?.....	73
Q: Is it necessary to refine the mesh near fractures in a ResFrac simulation?.....	74
Q: How do I use local mesh refinement in ResFrac?.....	75
Q: The pressure trend in my simulation ends before the ISIP is shown, how do I see it?.....	78
Q: How do I find pressure responses (SWPM or OPM) in the simulation so I can history match to field data?.....	78
Q: How big should I make an external fracture?.....	79
Q: The water saturation column is not showing up in the static model table even after I entered the logs using the wizard?.....	80
Q: Simulations are failing after moving the wellbore and recentering the matrix.....	80
Q: After running the 'Well stages setup wizard' I receive a yellow warning message that a wellbore vertices is within one meter of another vertex.....	80
Q: My fractures are continuing to grow after the pumps are turned off and the next stage begins. Is this normal?.....	80

Q: Why do we often need to use proppant conductivity curves that are lower than lab measured values?.	81
Q: Can I output the maximum and average fracture lengths and heights for each individual fracture?.....	81
Q: How can I stagger my perforations in adjacent wells such that the perforations or clusters are not directly aligned?.....	81
Q: Is water or gas rel perm used for the steam phase?.....	83
Q: Do you include overflush volumes in the pump schedule in ResFrac?.....	83
Q: How do I use the Pressure dependent NW dP (tortuosity) option in the advanced options of the Wells and Perforations tab?.....	84
Q: If using a dual porosity (DP) option, what is the difference between 'DP matrix permeability' and 'X/Y/Vertical direction permeability' in the static model table?.....	84
Q: If I am using the fractal D parameter, should I adjust my matrix meshing if my well is not aligned with Shmin?.....	84
Q: How is uncertainty accounted for in simulations?.....	85
<u>Q: How can I quantify and visualize the size of my hydraulic fractures (e.g. half-length)?.....</u>	<u>88</u>
5. Sensitivity Analysis and Automation Workflows.....	92
Q: How can I 'restart' an automation workflow from a previously run iteration?.....	92
6. History Matching Topics.....	96
<u>7. ResApps.....</u>	<u>98</u>
Q: How do I log-in and sign-up for a ResApps license?.....	98

1. ResFrac License and Installation

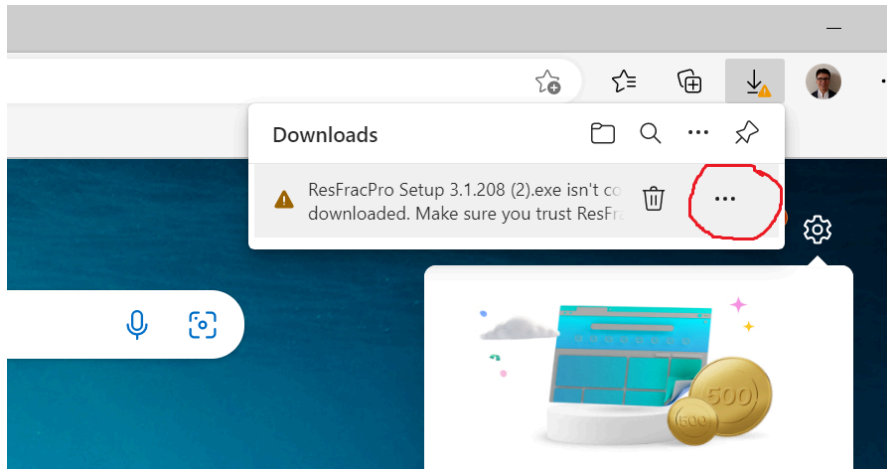
Q: How can I use Microsoft Edge to download ResFracPro?

This document describes how to install ResFracPro using the Edge browser. This warning message sometimes comes up when using the browser Edge to download the software:

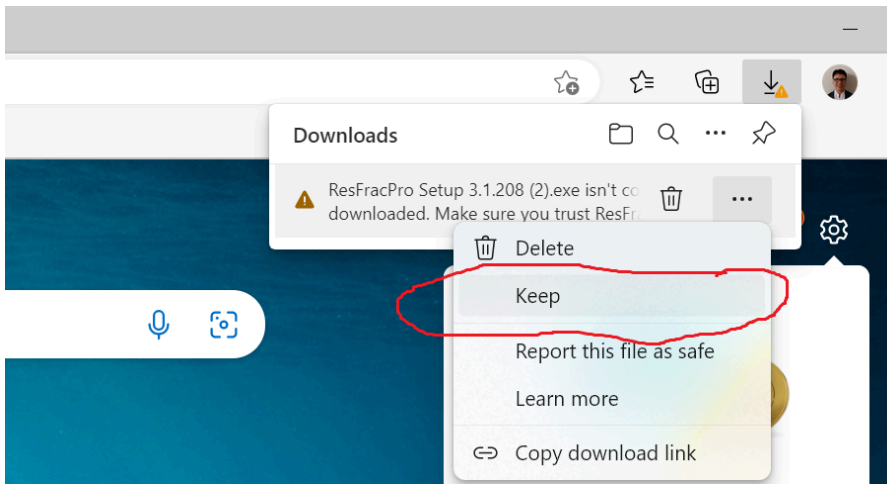


It comes up because ResFracPro is not a widely used app (compared to a consumer app). Depending on how your system is set up, you can still access the file downloaded via Edge using the steps below:

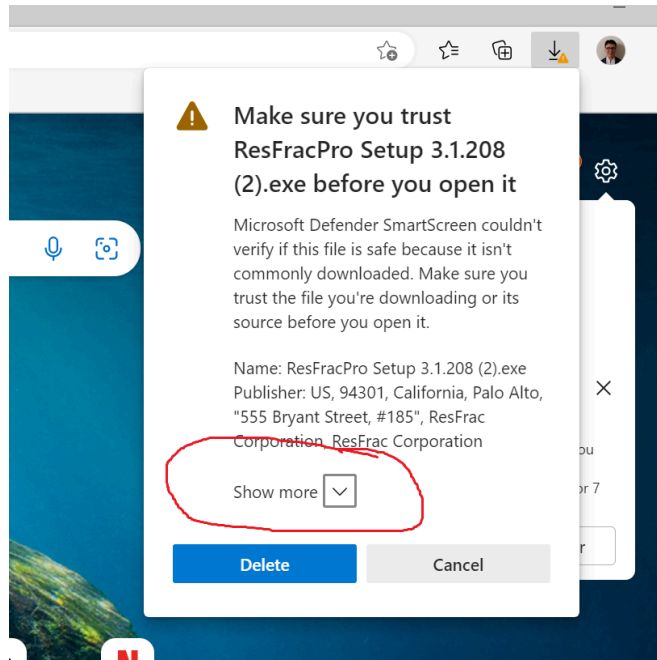
1. Hover over the warning on the "Downloads" popup. This will reveal a three-dot icon.



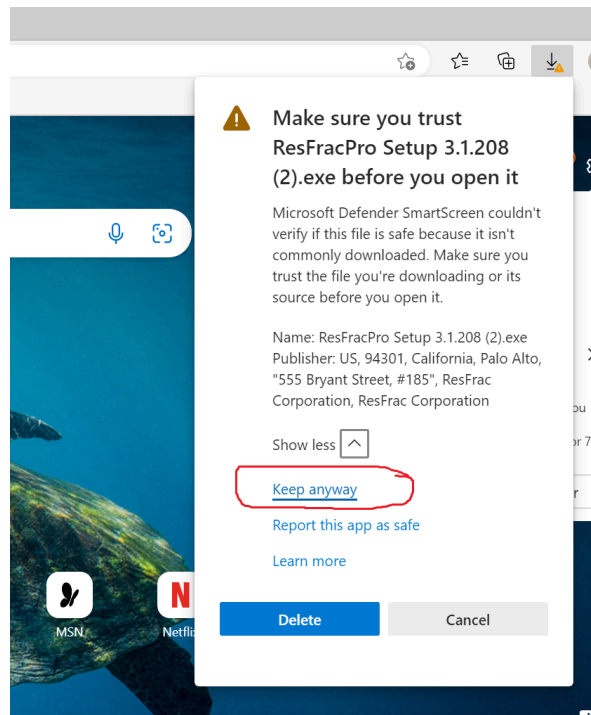
2. Click on that three-dot menu icon and then choose "Keep" on the menu that pops up.



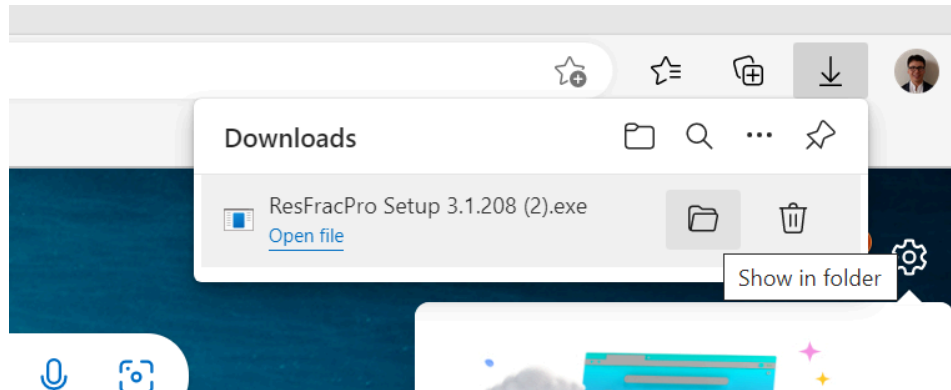
3. That will bring up another expanded warning message. Click on "Show more."



4. That will give a few more options. Click on "Keep anyway."



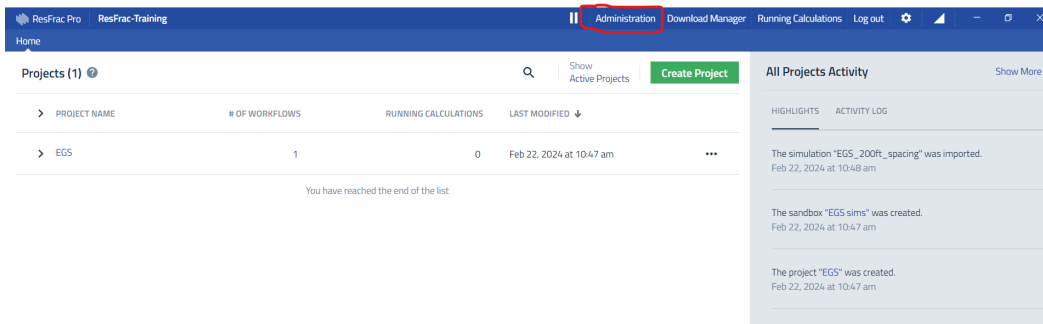
5. Then the "Downloads" popup will update. Now you can access the file in your downloads folder in the ordinary way.



Another alternative is to install Chrome and then download ResFracPro using that browser. You don't have to jump through as many hoops when using Chrome to download ResFracPro.

Q: How can I add and remove users from my company's ResFrac account?

The User Admin Tool within ResFracPro allows license admins to add and remove users. The User Admin Tool is accessed through the top tool menu within ResFracPro. You will need to be logged in and have admin privileges to access the User Admin Tool.













To add a user:

1. Open the User Admin Tool.
2. Press the "+" in the upper right of the tool:

Administration Panel ×

USERS GROUPS

Is Admin	Organization Username	User Group	Actions
No	charles@mccluregeomechanics.com	RESFRAC	 
Yes	soma@resfrac.com	RESFRAC	 
Yes	charles@resfrac.com	RESFRAC	 
No	joe@resfrac.com	RESFRAC	 
No	mark@resfrac.com	RESFRAC	 

Items per page: 5 | 1 - 5 of 112 | < > >>

3. Type the new user's email into the username input box (note, the domain for the email must match your license domain):
 - a. If you have user groups enabled for your license, you will also need to assign the new user to a user group.

Users

Admin

Username Select User Group RESFRAC

Save Close

joe@resfrac.com RESFRAC

4. After pressing 'Save', a prompt will appear that allows you to copy a link to the user interface (UI) installer. If the new user does not yet have the ResFracPro UI installed, you may send this link to them to download the software.

×

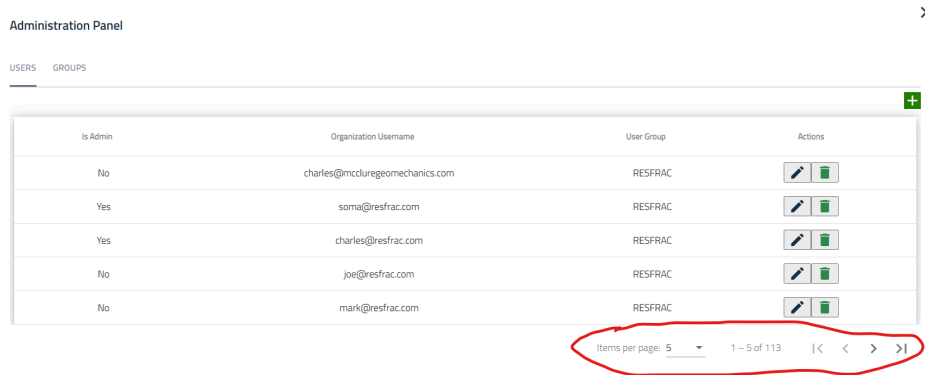
Copy ResFracPro installer link to clipboard

Would you like to copy to the clipboard the link to download the latest ResFracPro installer?

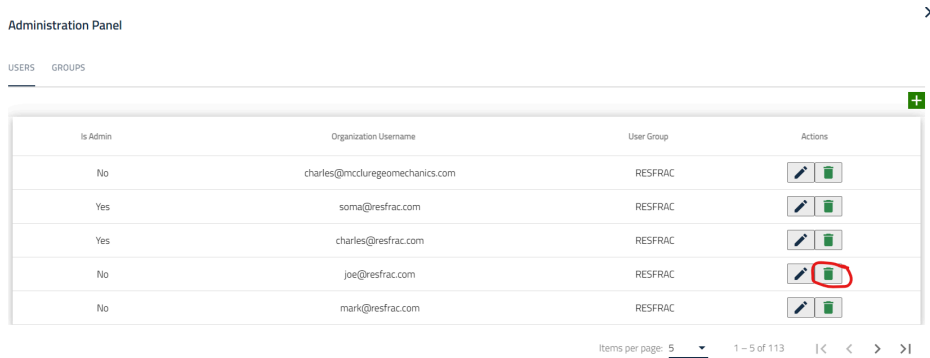
YES
Cancel

To remove a user:

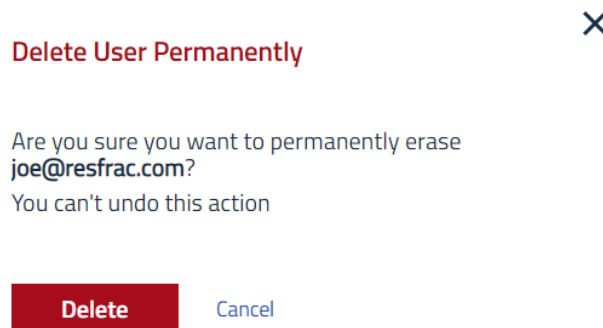
1. Open the User Admin Tool.
2. If needed, use the controls at the bottom of the window to expand the list or change pages:



3. Click the trash can next to the user that you would like to remove:



4. You will need to confirm their removal:



- This user is now removed and will no longer be able to submit new simulations. Any simulations that the user has run and have been downloaded to their computer **will not be deleted** and they will still have access to those files as well as the ability to open and view results.

Q: How do I check and monitor simulation hours used on my ResFrac Pro License?

Checking License Usage

License usage data is readily available in the ResFrac application. To check usage, click the “Settings” gear icon in the upper right corner.

The screenshot shows the ResFrac Pro application interface. At the top, there is a navigation bar with options like 'Download Manager', 'Running Calculations', 'Log out', and a gear icon for settings, which is circled in red. Below the navigation bar, the main content area is divided into two sections: 'Project Details' and 'Project Activity'. The 'Project Details' section shows information for a project named 'Training', including the number of workflows (4) and the creation timestamp (Aug 3, 2021 at 5:47 pm). Below this, there is a table of workflows with columns for 'Workflow Name', 'Type', 'Status', 'Date Created', and 'Last Modified'. The 'Project Activity' section on the right shows a log of recent events, such as 'The history matching "Midland_2Well VFR AHM" was submitted' and 'The simulation "Midland_2Well" was updated'.

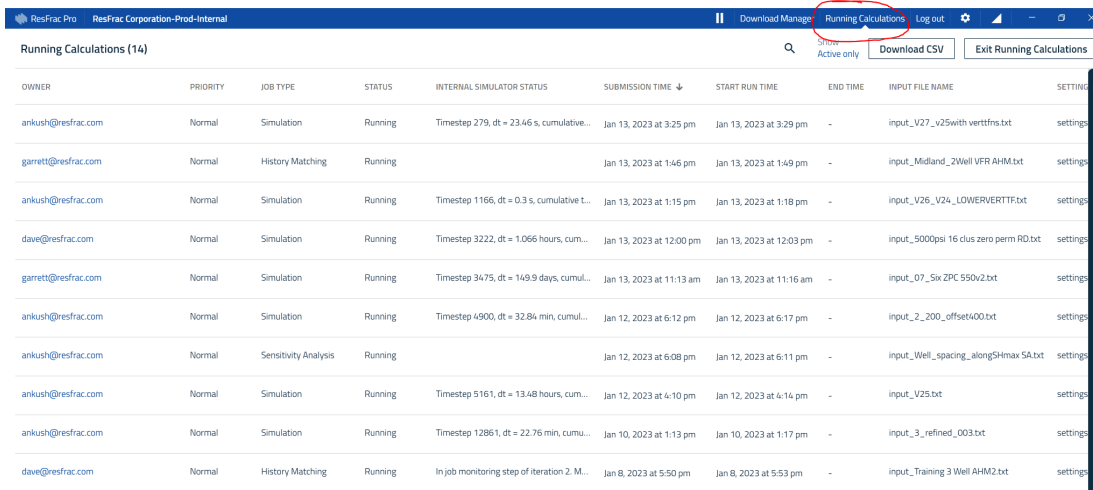
At the bottom of the settings page, a usage summary is displayed showing both the total usage across the organization for the current license period, and your individual usage during the same time period.

The screenshot shows the 'Settings' page in the ResFrac Pro application. The page title is 'Settings' and it includes sub-headers for 'GENERAL' and 'SOFTWARE VERSION'. Under 'GENERAL', there are two checkboxes: 'Turn on detailed logging? If you turn on, performance may be affected.' (checked) and 'Use local temp folder for downloads.' (unchecked). Under 'SOFTWARE VERSION', there is information about the core ResFrac simulator and the ResFrac user-interface, including version numbers and update dates. At the bottom of the page, there is a section titled 'USAGE SUMMARY' which is circled in red. This section contains the following text: 'Hourly use license - Jan 13 2023 10:42PM (UTC) - Usage since the start of the current subscription period (Jan 1 2022 12:00AM (UTC)) - 30A081476649 hours of total licensed hours - 1000000 (Surcharge starts at : 1000000 hours). User usage : 63920 hours.'

For more detailed information on usage, feel free to contact us at support@resfrac.com and we can pull detailed usage summaries.

Checking Running Jobs

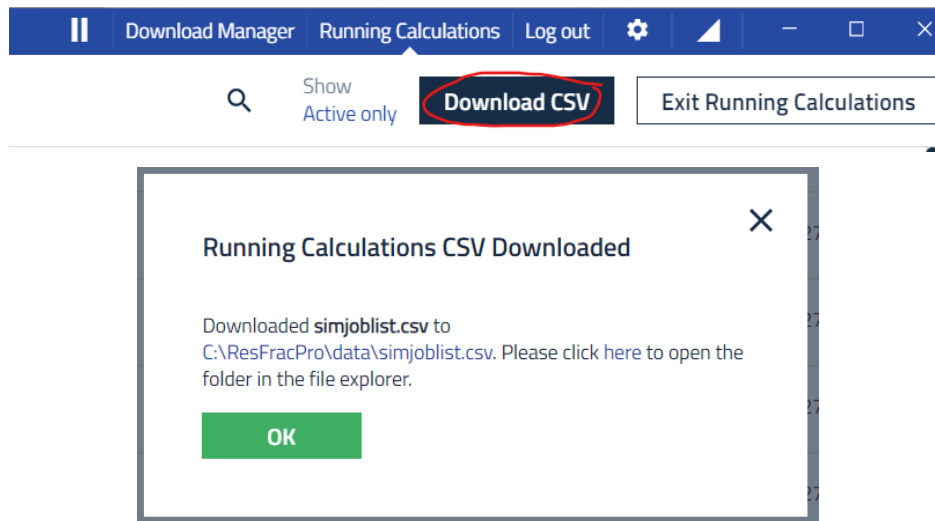
By clicking the “Running Calculations” tab in the upper right, you can easily view all jobs currently running on your license and who initiated them:



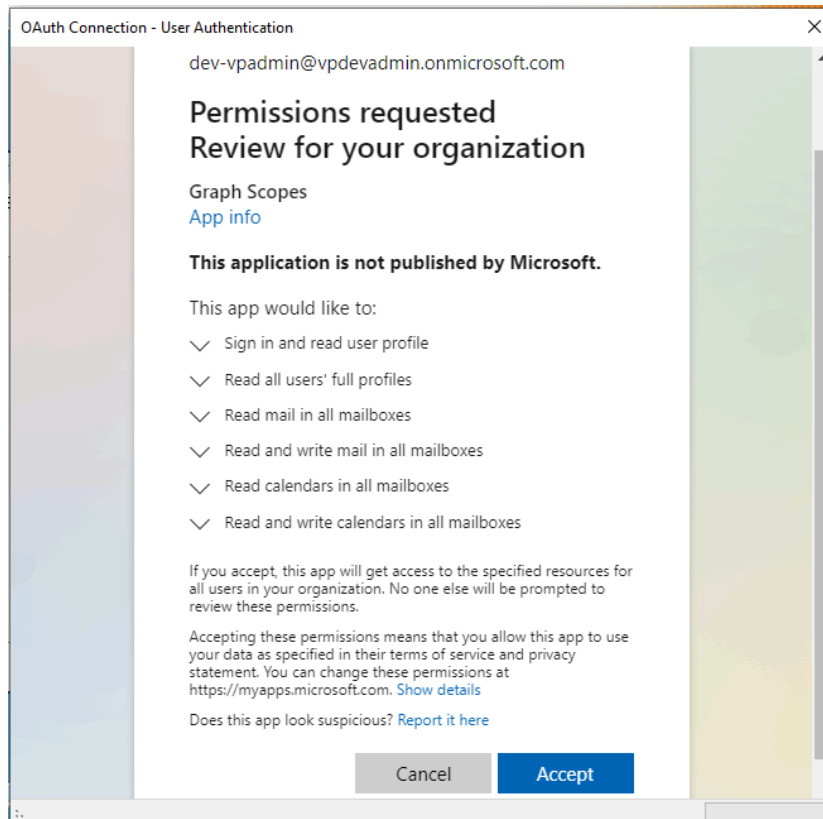
OWNER	PRIORITY	JOB TYPE	STATUS	INTERNAL SIMULATOR STATUS	SUBMISSION TIME	START RUN TIME	END TIME	INPUT FILE NAME	SETTINGS
ankush@resfrac.com	Normal	Simulation	Running	Timestep 279, dt = 23.46 s, cumulative...	Jan 13, 2023 at 3:25 pm	Jan 13, 2023 at 3:29 pm	-	input_V27_V25with verffrms.txt	settings
garrett@resfrac.com	Normal	History Matching	Running		Jan 13, 2023 at 1:46 pm	Jan 13, 2023 at 1:49 pm	-	input_Midland_2Well VFR AHM.txt	settings
ankush@resfrac.com	Normal	Simulation	Running	Timestep 1166, dt = 0.3 s, cumulative t...	Jan 13, 2023 at 1:15 pm	Jan 13, 2023 at 1:18 pm	-	input_V26_V24_LOWERVERTFF.txt	settings
dave@resfrac.com	Normal	Simulation	Running	Timestep 3222, dt = 1.066 hours, cumu...	Jan 13, 2023 at 12:00 pm	Jan 13, 2023 at 12:03 pm	-	input_5000psi 16 clus zero perm RD.txt	settings
garrett@resfrac.com	Normal	Simulation	Running	Timestep 3475, dt = 149.9 days, cumu...	Jan 13, 2023 at 11:13 am	Jan 13, 2023 at 11:16 am	-	input_07_Six ZPC 550v2.txt	settings
ankush@resfrac.com	Normal	Simulation	Running	Timestep 4900, dt = 32.84 min, cumu...	Jan 12, 2023 at 6:12 pm	Jan 12, 2023 at 6:17 pm	-	input_2_200_offset400.txt	settings
ankush@resfrac.com	Normal	Sensitivity Analysis	Running		Jan 12, 2023 at 6:08 pm	Jan 12, 2023 at 6:11 pm	-	input_Well_spacing_alongSHmax SA.txt	settings
ankush@resfrac.com	Normal	Simulation	Running	Timestep 5161, dt = 13.48 hours, cumu...	Jan 12, 2023 at 4:10 pm	Jan 12, 2023 at 4:14 pm	-	input_V25.txt	settings
ankush@resfrac.com	Normal	Simulation	Running	Timestep 12861, dt = 22.76 min, cumu...	Jan 10, 2023 at 1:13 pm	Jan 10, 2023 at 1:17 pm	-	input_3_refined_D03.txt	settings
dave@resfrac.com	Normal	History Matching	Running	In job monitoring step of iteration 2. M...	Jan 8, 2023 at 5:50 pm	Jan 8, 2023 at 5:53 pm	-	input_Training 3 Well AHM2.txt	settings

Downloading Usage for Current License Period

From the “Running Calculations” tab, click the ‘Download CSV’ button. This will download a csv of all simulations run during the last two years on the license. The sheet provides the user, simulation id, type of simulation (simulation, optimization, history match, etc.), submit time, start time, end time, status of the simulation and any failure messages, etc.



accounts for signing in. The client organization's Azure administrator will then be prompted to consent. This is a one-time action that applies to the whole organization and all users in that organization. The admin consent prompt can look like this:



Note that compared with the above example, the permission scopes will differ (ResFrac needs only a narrow set of scopes).

More details of the admin consent workflow are available at the following links:

<https://learn.microsoft.com/en-us/entra/identity/enterprise-apps/user-admin-consent-overview> and
<https://learn.microsoft.com/en-us/entra/identity/enterprise-apps/admin-consent-workflow-overview>.

Additionally, some client organizations have firewall / proxy settings that need to be configured to allow access to ResFrac service URL domains. Typically these settings are managed by someone in the IT department. Some of these domains are server-specific; for the training server, these URLs are as follows:

<https://login.microsoftonline.com/>

<https://resfracreleases.azureedge.net/>

<https://resfrac<client name>.azureedge.net/>

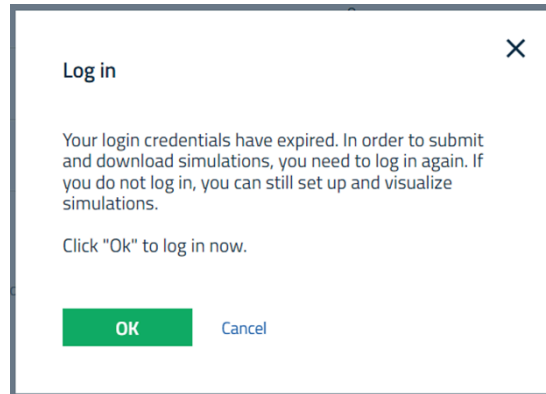
<https://resfrac<client name>.blob.core.windows.net/>

<https://resfrac-api-<client name>.azurewebsites.net/>

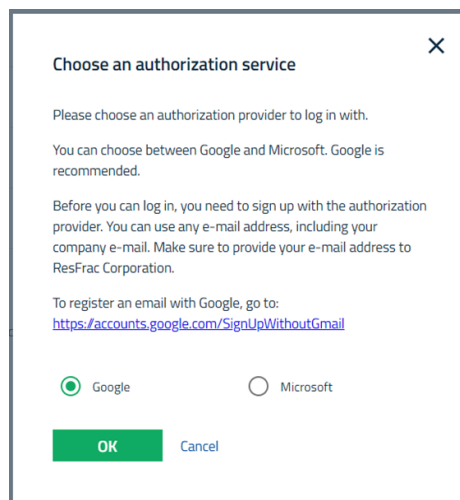
2. User-Interface and Cloud

Q: How do I log-in to the ResFrac user interface to be able to run simulations?

All ResFrac users must login and authenticate their credentials prior to submitting or downloading simulations. For users who are already logged in, an automatic logout is triggered after three days, requiring the user to log in again.

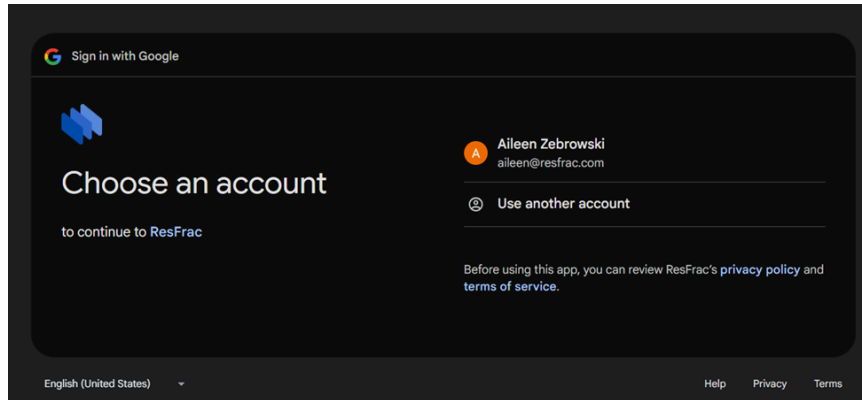


Users may choose to authenticate using either Google or Microsoft. Note that if you do not have an account with either Google or Microsoft, you can register your existing email address (from any provider) with Google in order to enable log-in, at the URL: <accounts.google.com/SignUpWithoutGmail>.



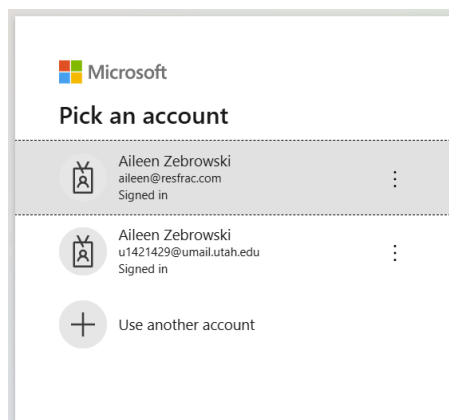
Google Authentication

1. You will be redirected to the google login page in your browser.
2. Login (or select account if already logged in) using an email address with license permissions.

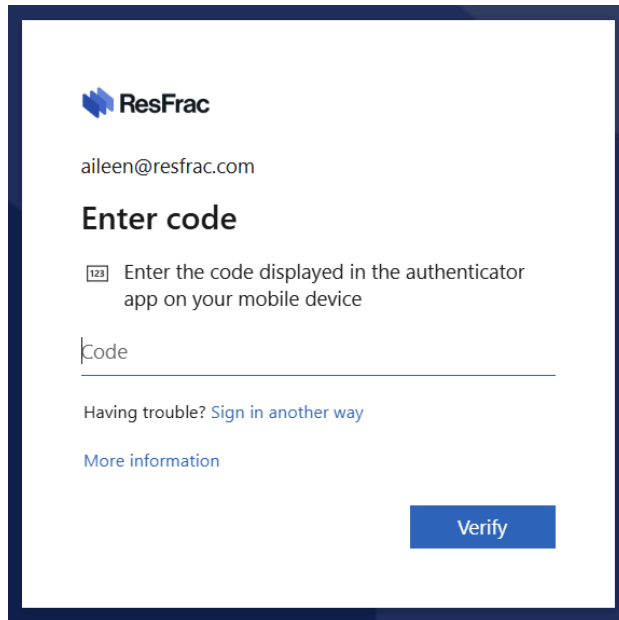


Microsoft Authentication

1. ResFracPro connects to your organization's Azure Active Directory (Azure AD) using the Application credentials (Application Name: resfraclogin)
2. The "resfraclogin" application requires a one-time [approval by Azure AD administrators](#) of the licensee organization. Once approved for the first user, all other users in the same domain can typically log in without further IT involvement.
3. Login (or select account if already logged in) using an email address with license permissions.



4. ResFracPro uses OAuth2 protocol to authenticate using Microsoft Azure AD.



The image shows a login verification screen for ResFrac. At the top left is the ResFrac logo. Below it is the email address 'aileen@resfrac.com'. The main heading is 'Enter code'. Below this is a text box with a small icon and the instruction 'Enter the code displayed in the authenticator app on your mobile device'. There is a text input field labeled 'Code'. Below the input field are two links: 'Having trouble? Sign in another way' and 'More information'. At the bottom right is a blue button labeled 'Verify'.

5. Enter the code for your 2-factor authentication to finish logging in.



For more on the admin consent workflow in Azure AD, see the following article:

<https://learn.microsoft.com/en-us/azure/active-directory/manage-apps/configure-admin-consent-workflow>

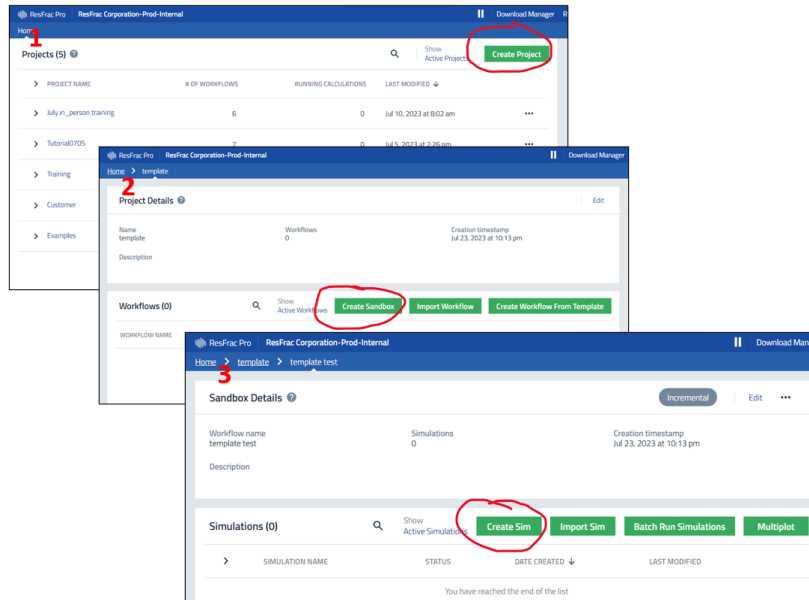
If you face any issues, we recommend connecting ResFrac with your Azure AD team so that we can provide them with necessary details of the resfraclogin client application that needs to be approved.

Q: How do I access the built-in ResFrac templates?

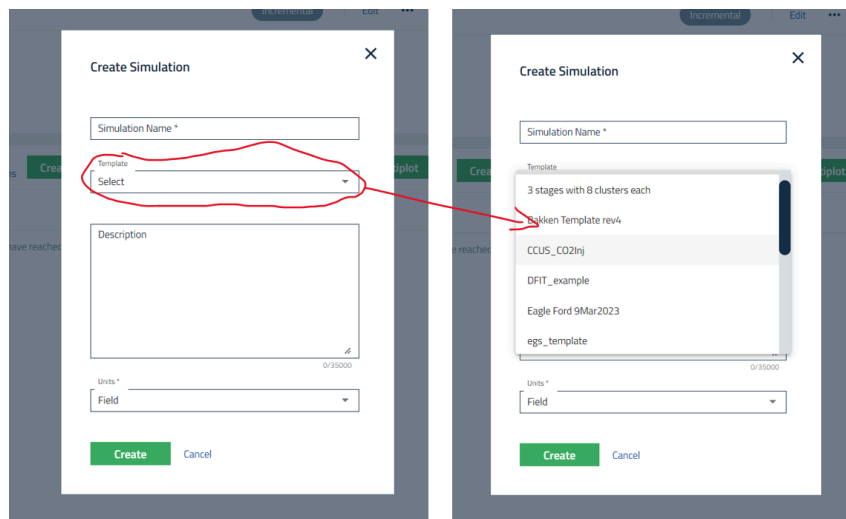
ResFracPro includes an ever-growing inventory of template simulations. These templates fall into two categories:

1. Provide basin-specific recommended values (like the Bakken or Eagle Ford)
2. Demonstrate particular processes and how to set these up (refrac, parent/child, DFIT, etc.)

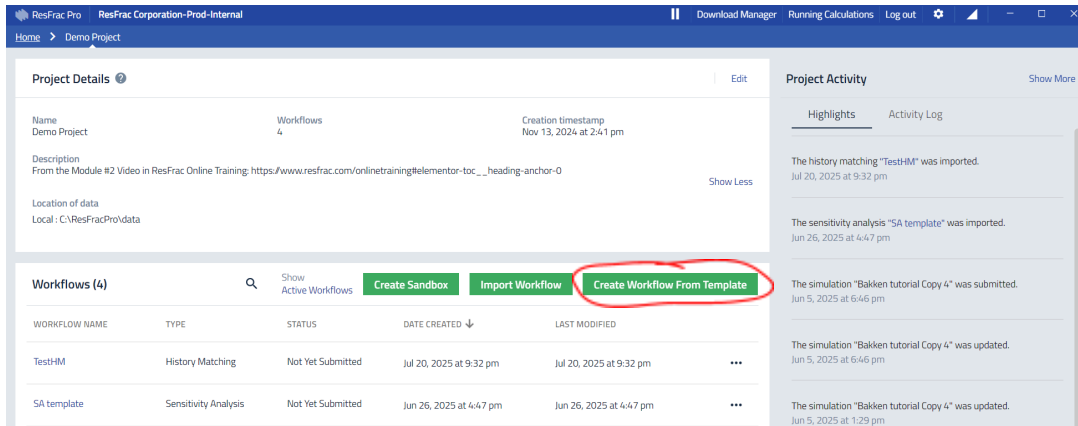
To create a simulation from a template, first create a project, then a sandbox within the project, and finally a simulation within the sandbox:



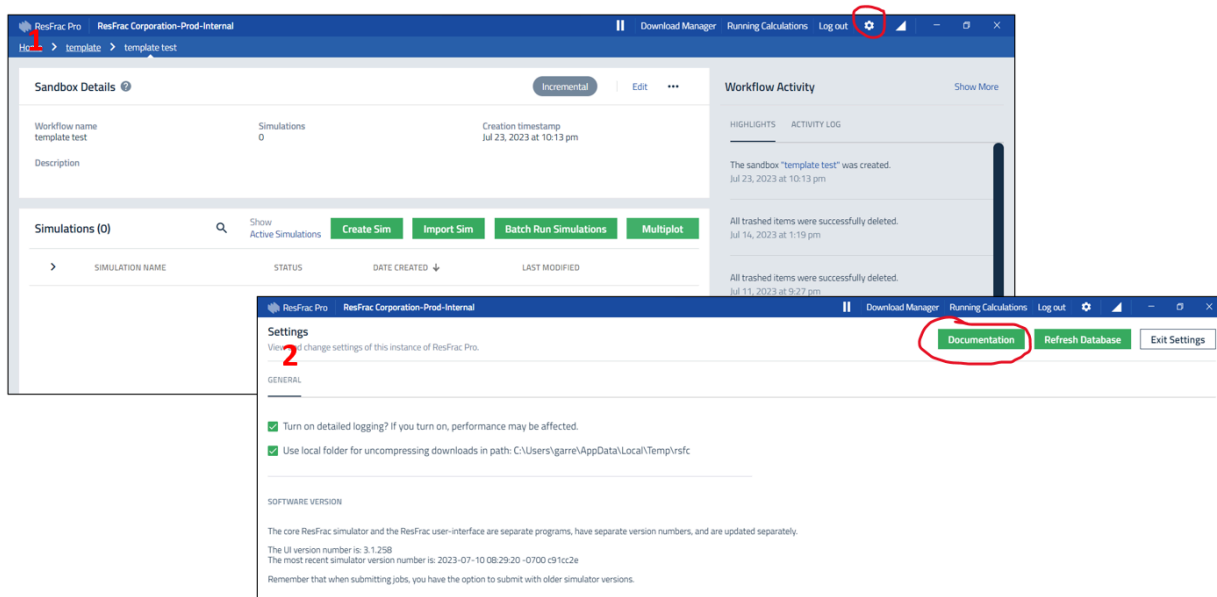
When you click the “Create Sim” function, you are given an option to select a template from the dropdown menu:



There are also built-in templates for the ‘workflows’ – history matching, optimization, sensitivity analysis, and uncertainty quantification. These can be accessed when you click ‘Create workflow’ from within the project view.



To get more details on the data or processes included in the template simulation navigate to the documentation folder on your machine by pressing the settings “gear” in the upper right and then “Documentation”:



This will open the File Explorer location on your computer that contains all release notes and documentation. The “workflow templates” folder contains individual pdf documents with the details of each template simulation.

Windows (C:) > ResFracProProd > resources > documentation >

Name	Date modified	Type	Size
Workflow templates	7/10/2023 11:46 AM	File folder	
Fracture modeling with ResFrac_March_2...	7/10/2023 11:46 AM	Chrome HTML Do...	22,818 KB
History Matching ResFrac Models 2019.1...	7/10/2023 11:46 AM	Chrome HTML Do...	910 KB
Import and plot external 2D data	7/10/2023 11:46 AM	Chrome HTML Do...	1,242 KB
McClure HFTC 2020 11-17-2019	7/10/2023 11:46 AM	Chrome HTML Do...	1,953 KB
ResFrac Release Notes April 10 2022	7/10/2023 11:46 AM	Chrome HTML Do...	1,140 KB
ResFrac Release Notes April 19 2021	7/10/2023 11:46 AM	Chrome HTML Do...	200 KB
ResFrac Release Notes April 2023	7/10/2023 11:46 AM	Chrome HTML Do...	161 KB
ResFrac Release Notes Aug 23 2020	7/10/2023 11:46 AM	Chrome HTML Do...	305 KB
ResFrac Release Notes August 8 2019	7/10/2023 11:46 AM	Chrome HTML Do...	118 KB
ResFrac Release Notes August 19 2019	7/10/2023 11:46 AM	Chrome HTML Do...	122 KB
ResFrac Release Notes December 7 2018	7/10/2023 11:46 AM	Chrome HTML Do...	167 KB
ResFrac Release Notes December 9 2022	7/10/2023 11:46 AM	Chrome HTML Do...	927 KB
ResFrac Release Notes December 20 2021	7/10/2023 11:46 AM	Chrome HTML Do...	535 KB
ResFrac Release Notes December 22 2019	7/10/2023 11:46 AM	Chrome HTML Do...	316 KB

Windows (C:) > ResFracProProd > resources > documentation > Workflow templates

Name	Date modified	Type	Size
Bakken Template	7/10/2023 11:46 AM	Chrome HTML Do...	1,193 KB
Black Oil Tracers	7/10/2023 11:46 AM	Chrome HTML Do...	778 KB
CCUS Template	7/10/2023 11:46 AM	Chrome HTML Do...	3,758 KB
ComplexEconomicOptimizationMidland...	7/10/2023 11:46 AM	Chrome HTML Do...	2,908 KB
Eagle Ford Template	7/10/2023 11:46 AM	Chrome HTML Do...	1,051 KB
EGS template	7/10/2023 11:46 AM	Chrome HTML Do...	3,393 KB
Fracture turning template	7/10/2023 11:46 AM	Chrome HTML Do...	1,313 KB
Huff and Puff Template	7/10/2023 11:46 AM	Chrome HTML Do...	1,729 KB
Midland Basin AHM	7/10/2023 11:46 AM	Chrome HTML Do...	1,663 KB
Midland Basin Template	7/10/2023 11:46 AM	Chrome HTML Do...	1,027 KB
Parent_Child_Damage_Template	7/10/2023 11:46 AM	Chrome HTML Do...	1,811 KB
Refrac Template	7/10/2023 11:46 AM	Chrome HTML Do...	1,770 KB
Sensitivity Analysis Midland Basin	7/10/2023 11:46 AM	Chrome HTML Do...	1,733 KB
Stack_AnadarkoBasin_Template	7/10/2023 11:46 AM	Chrome HTML Do...	1,582 KB
Tapered Perforating Template	7/10/2023 11:46 AM	Chrome HTML Do...	1,426 KB

These pdfs include details like where the input data came from and/or particular keywords/functions used within ResFracPro.

Q: How do I share a ResFrac simulation?

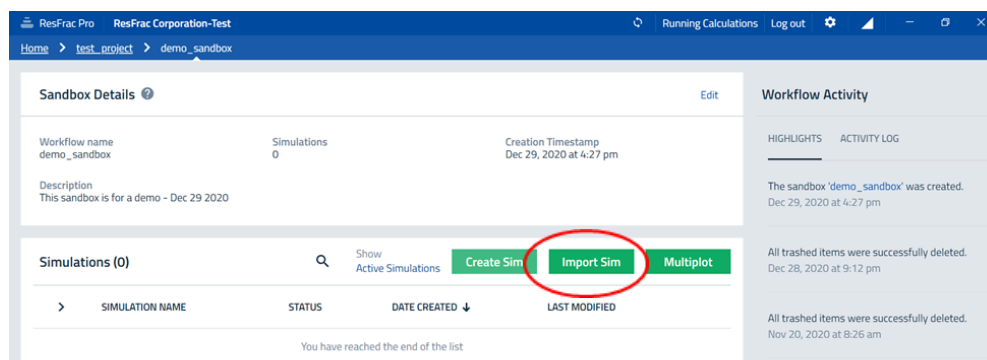
You may want to share a simulation with someone else. Click on the simulation and select 'Share simulation.' Enter a recipient email address (which by default is support@resfrac.com, though this can be changed), select a restart file to include, and add an optional message. The recipient will receive a link to download the simulation folder. Note that simulations are only kept on client servers for 30 days, in most cases. If it has been longer, then you likely will not be able to share the simulation, and will receive an email informing you that the simulation is no longer available.

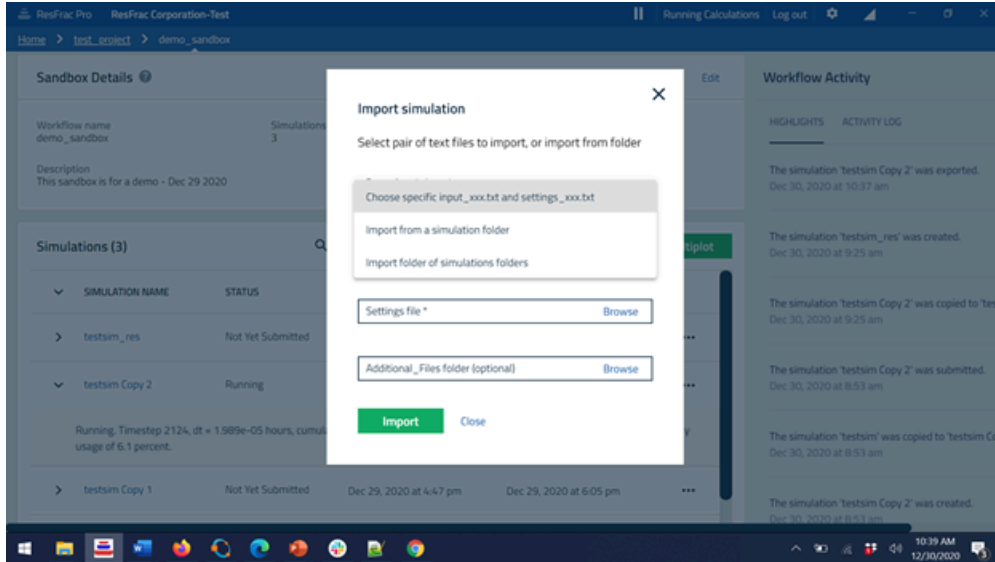
Alternatively, you may select 'Export simulation folder.' You will be prompted to select a location. The entire folder, along with an additional metadata file with information such as the 'simulation description,' will be copied to that location.

Q: How do I import a simulation into ResFrac?

To import a simulation, click on 'Import Sim' from the job manager screen in your chosen Sandbox. You have three options for importing a simulation this way. Another method is using the File Explorer directly (which will be discussed further below):

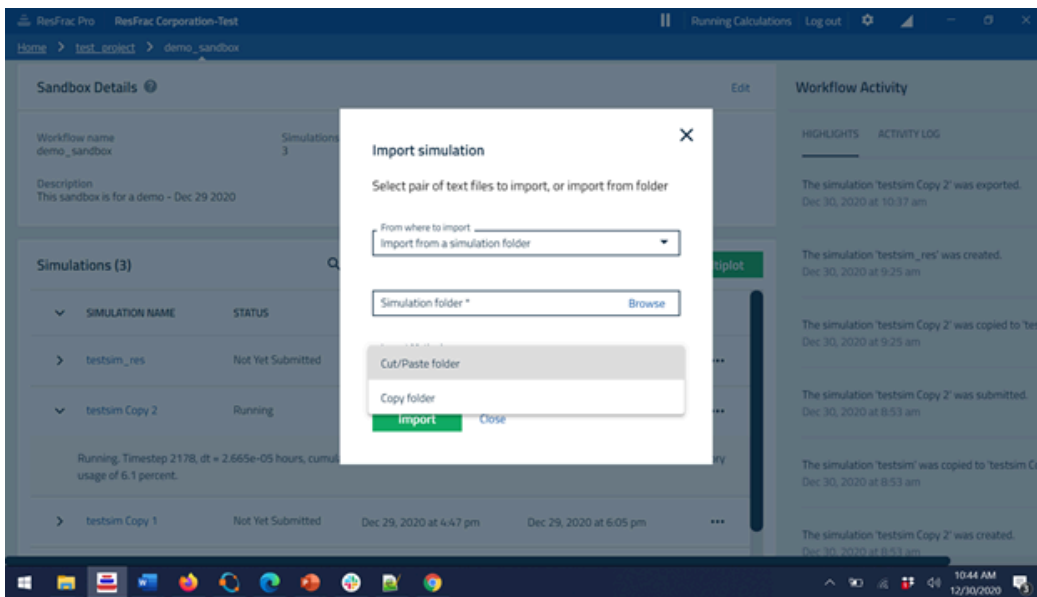
1. You can navigate directly to an input and settings file. Those two files, alone, contain all the information required to set up and run a simulation. If you are running a corner point simulation or a simulation that otherwise has additional data files as part of the setup, they must be placed in a separate folder for 'Additional_Files', and you are asked to optionally specify the location of that folder for the import.
2. You can select an entire self-contained simulation folder, such as the one created with the 'Export simulation folder' option. If the simulation has already been run and you would like to import the results in addition to the settings and input files, this is the method you should use. An alternative to using an exported simulation folder, you could simply copy the simulation folder directly from the Windows File Explorer and that would work just fine. However, it would not include some of the supplementary data, such as the 'Simulation Description' stored in the job manager.
3. You can import a 'folder of folders.' For example, if you have copied several 'simulation folders' over from another computer (and placed them all in the same folder), you could import all of the folders at once using the 'folder of folders' option.

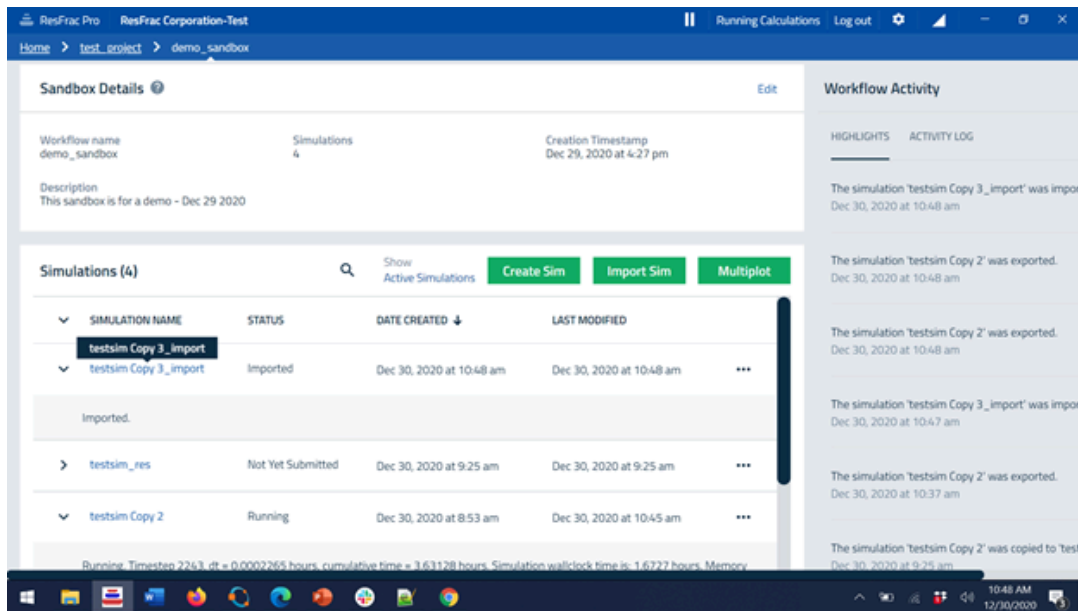
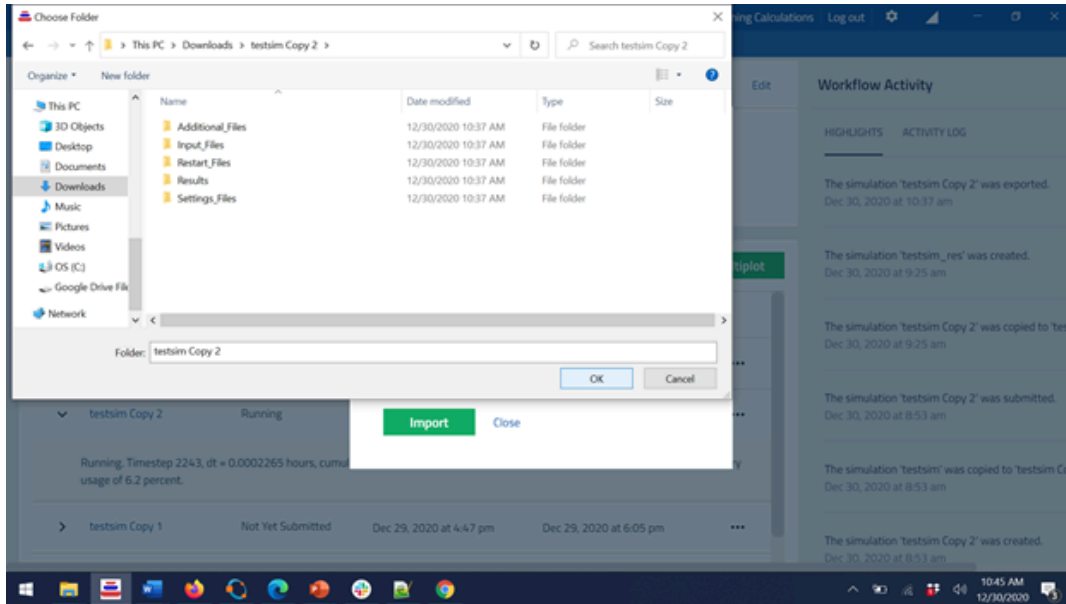




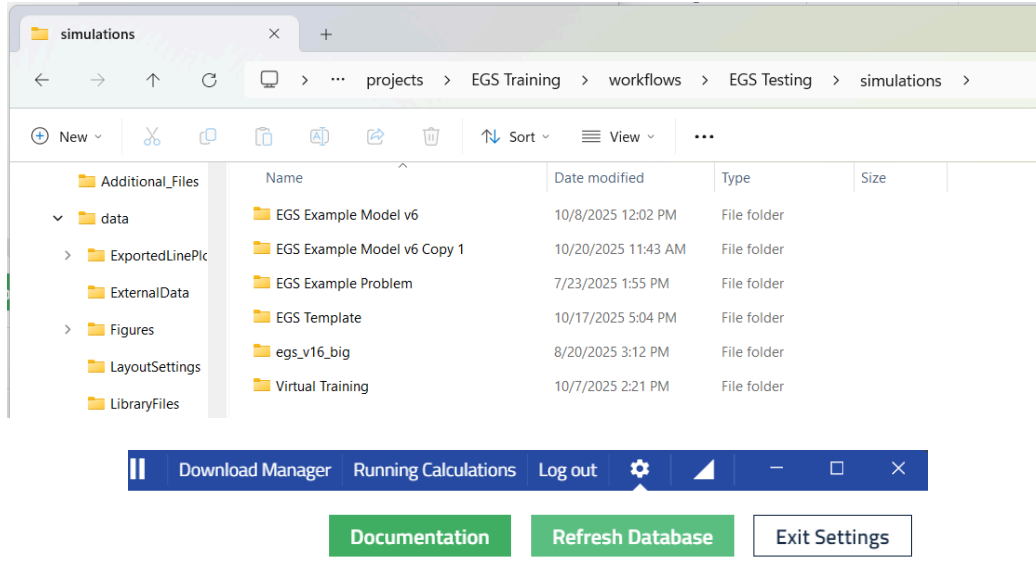
If you choose to import from a folder, you can select either the 'Cut/Paste' or 'Copy' method. The former option will remove the folder from its current location when the import is performed. The copy option copies the folder so that it remains in its current location when the import is performed.

If the simulation name was already being used in that sandbox, it will report an error. You can change the name as part of the import.





The final method for importing a simulation is to add new folders directly to your Sandbox simulations folder in File Explorer. After moving the folders to the correct location in File Explorer, to update the ResFrac UI to make your new simulation(s) visible select the option to 'Refresh database' under settings (the gear icon in the upper right corner). This is a common and useful way of adding and removing files.



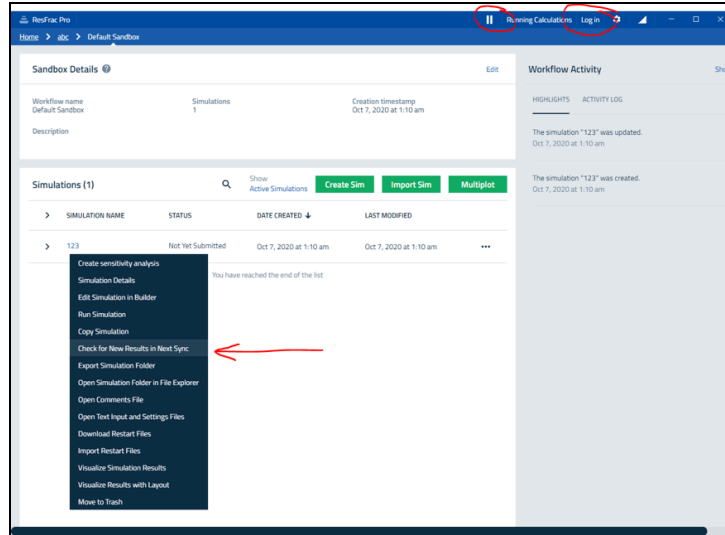
Q: How can I troubleshoot issues with my simulation results downloading?

The most common issue that occurs when viewing results is that you are trying to open a simulation with incompletely downloaded results. Incomplete results downloads can result in error messages when attempting to open a simulation in the visualization tool, limited functionality of the visualization tool after it is open, ‘odd’ results displayed in the visualization tool, and even inability to open results in the visualization tool. This section is a troubleshooting guide for resolving common download problems.

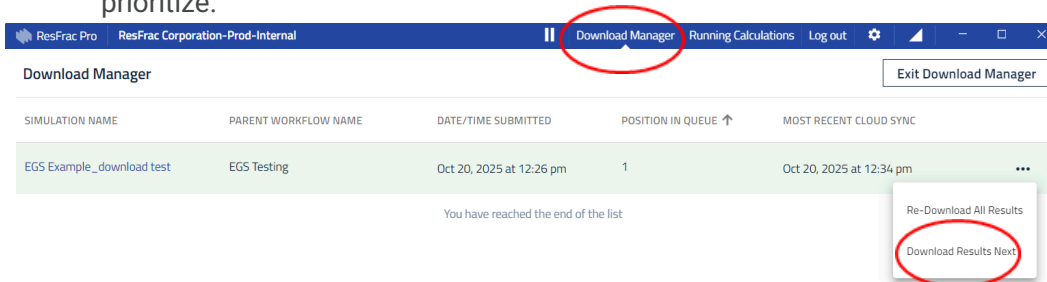
The ResFracPro UI constantly downloads results for all your running and recently completed simulations. This means that you should see new results for your simulations every few minutes or so. If you find that results for your simulations aren’t accumulating quickly, or at all, even after many hours, then something may be going wrong with results download behavior. In that case, we suggest taking the following steps to troubleshoot.

While following these steps, a temporary workaround is to [share the simulation with yourself](#), download, and [import it back into ResFracPro](#). This will allow you to make progress while troubleshooting.

1. Downloading results requires that your computer be turned on, and ResFracPro is open. If you want to get results overnight, then make sure that ResFracPro is open and your computer stays on overnight. Do not allow your computer to sleep/ hibernate/ auto-shutdown. This may require changing your power and battery settings, and ensure your machine is plugged in.
2. Make sure you don’t have downloads paused – use the “Pause” / “Sync” toggle icon in the top bar. After you unpause downloads, the automatic download cycle will kick in within a few minutes and new results should start to accumulate for your currently running and recently completed simulations. There will also be the option to permanently disable downloading for individual simulations in the next release (late 2025 or early 2026), which you should also ensure is not selected for any simulations of interest.



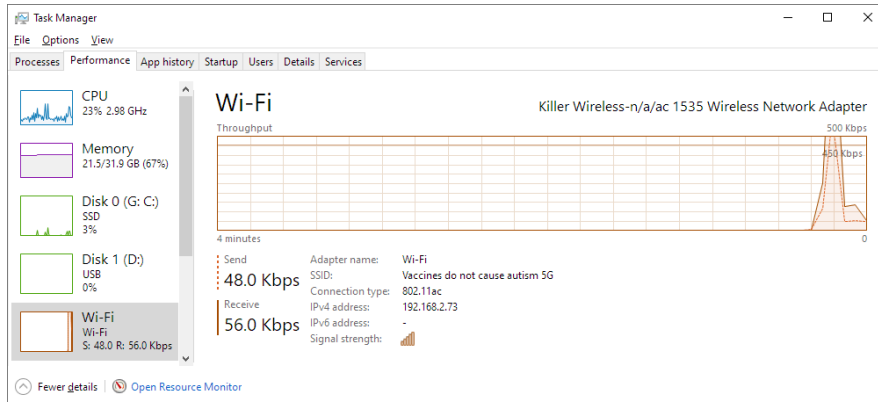
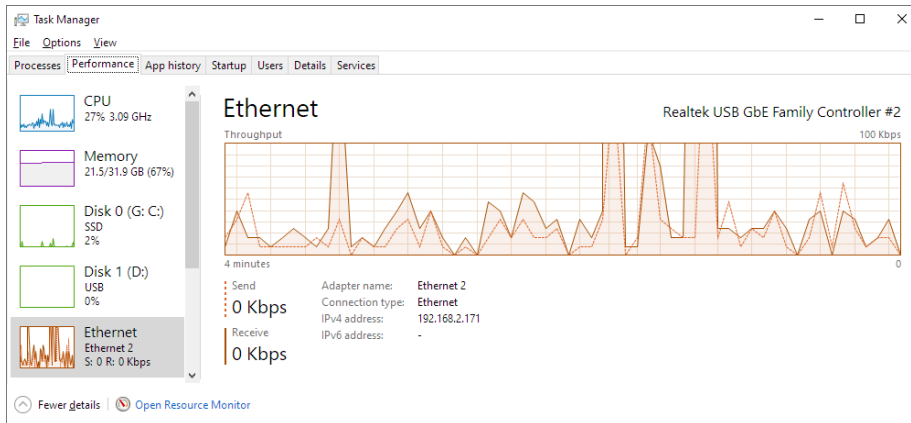
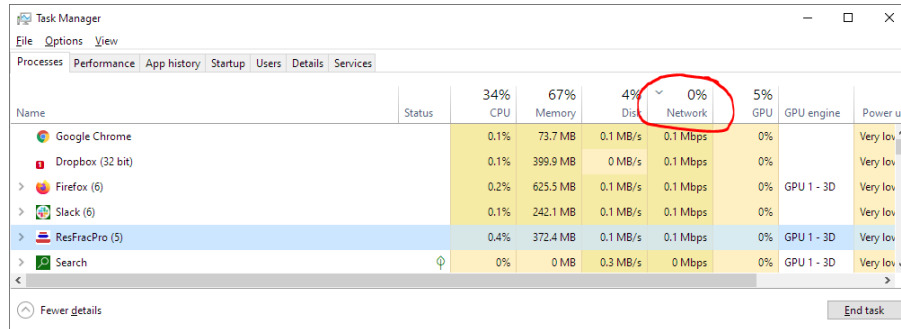
3. Check to see if your simulation has a Results folder, a Results/newviz folder, and that these folders have multiple files inside. If this is the case, then the most likely problems are that (a) the UI is taking a while to download because you have a lot of simulations, (b) the UI is stuck on something and so is failing to get the newest results for this simulation, or (c) the UI is confused about the timeliness of the local copy of certain files vs the ones available on the server.
 - If problem (a) is the case, then if you wait for at least 10 minutes or so you should see more items accumulate in the results folders for your currently running or recently completed simulations. Results are downloaded from one simulation at a time. The other simulations queue up and wait to be updated while downloads are occurring. Thus, if you are checking to confirm that results are downloading, first go to the 'Download manager' and check the list to see which simulation is currently at the top of the list and being downloaded. Check for progressive downloads in the results folder for that simulation.
 - i. There will be the option to permanently disable downloading for individual simulations in the next release (late 2025 or early 2026) which can be helpful if a simulation is downloading very slowly (or stuck), and getting stuck at the top of the queue, blocking others from being downloaded.
 - ii. You can also choose to 'Download Results Next' for an individual sim that you would like to prioritize.



- If problem (b) is the case, then exiting/reopening the app and restarting the computer will help to overcome the 'stuck' state.

- If problem (c) is the case, then you will want to try triggering a re-download, which is one of the later steps in this guide. This guide will walk you through the steps to troubleshoot all these potential problems.

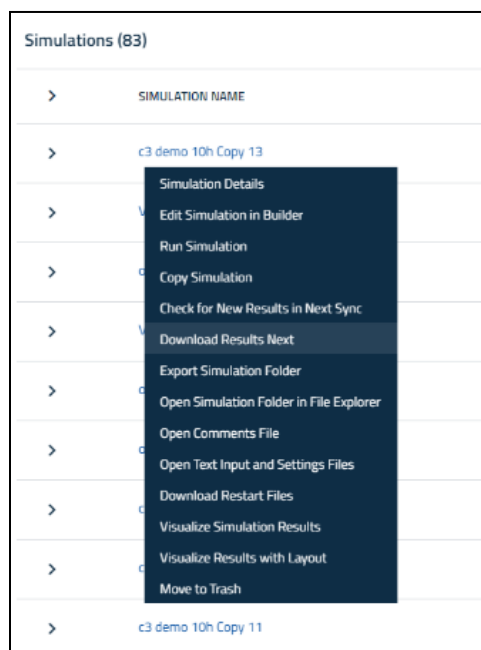
4. If you have a lot of results to download, it can take a while for the UI to get everything. For example, if you submitted 20 jobs and then turned off your computer for a couple days, there could be 50 GB of files waiting to download. That could take a long time, even with a fast internet connection. You can check to see how much bandwidth ResFracPro and other apps are using in the Windows Task Manager. Look for the “Network” column on the “Processes” tab to see by application, and look at “Ethernet” or “Wi-Fi” under the “Performance” tab to see total system usage.



If the network usage values displayed are high for ResFracPro – i.e. hundreds of KB/s or more, depending on your internet connection – then most likely you just have a lot of results waiting to download and it will take time to get your results.

5. Check internet download speeds using any online speed test. If the dial is low, check your internet connection (wifi router, range extender, etc.). Consider connecting to ethernet if you are not already, as this will provide you with the fastest possible speed.
6. You can prioritize a specific simulation in the next sync round by using the “Download Results Next” menu item. Using this on a simulation will cause that simulation to be the next simulation to download results for. Results for that simulation should start downloading within a few minutes.

After you’ve given your simulation some more time to download, you can check the Results and Results/newviz folders for that simulation and see the date modified values for the files contained within. If new files are recently modified (i.e., within the last 15 minutes), that indicates that downloading is working properly and the results should be viewable after they accumulate.



7. Log out and log back in using the “Log out” / “Log in” link in the top bar.
8. Exit and reopen the app. If there is no improvement, restart your computer.
9. Trigger a re-download.
 - a. One way to trigger a re-download is by clicking on the simulation name and choosing “Check for new results in next sync.” This is recommended for simulations that have been completed already.
 - b. An alternative way to trigger a re-download is by renaming/moving/deleting the files that are continuously modified (such as comments_xxx.txt or sim_track_xxx.txt) and then right-clicking on the simulation name and choosing, “Check for new results in the next sync.” This is recommended for simulations that are still running.

After triggering a re-download, upon the next download cycle, which occurs every few minutes, the UI should download the latest version of the repeatedly updated files locally. This works the same way for any file that gets downloaded repeatedly, including the following:

- sim_track_xxx.csv (line plot data in the UI)
- newviz/snapshots.txt (used for 3D plots in the UI)
- newviz/snapshottimes.txt (used for 3D plots in the UI)
- newviz/sobs_summary.txt (used for stress observation plane 3D plots in the UI)
- comments_xxxx.csv (not used in UI; for the user to read)
- 30_day_prod_xxx.csv (not used in UI; for user to analyze manually)
- daily_prod_xxx.csv (not used in UI; for user to analyze manually)

10. If the above steps don't help, then we will need to work with you to resolve the issue. To facilitate that, you will need to gather information for troubleshooting.

- Turn on logging with the ["Running with Detailed Logging"](#) instructions. The Windows security part is optional and is only important if turning on logging causes your system to slow down. After turning on detailed logging, wait ten minutes and send the file C:\ResFracPro\logs\logs.log to ResFrac.
- Include the file C:\ResFracPro\data\main.db
- Include a screen capture of the full UI window when sending, and the "Date modified" time and local timezone for the files that are having a problem.

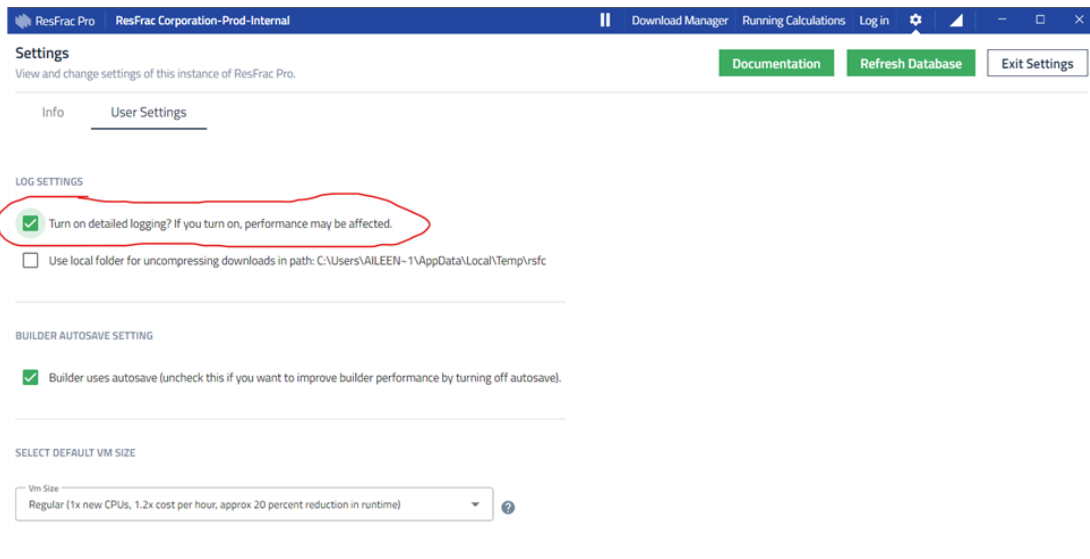
Additionally, we may request some additional files to help with troubleshooting:

- input_xxx.txt – Located in the simulation folder
- settings_xxx.txt – Located in the simulation folder
- comments_xxx.txt – Located in the "Results" folder for the simulation
- sim_track_xxx.txt – Located in the "Results" folder for the simulation
- Screen capture of the full ResFracPro UI window opened to the page that lists the simulation that is having trouble
- Screen capture of Windows Explorer opened to the simulation folder
- Screen capture of Windows Explorer opened to the "Results" folder for the simulation
- Screen capture of Windows Explorer opened to the "Results/newviz" folder for the simulation

[Q: How do I run ResFrac with ‘Detailed Logging’ for debugging?](#)

The ResFracPro user interface has a feature that causes the program to do detailed logging of everything it does. This way, if you have a problem, you can provide the log file to ResFrac so we can track down the issue. Turning on detailed logging will create a file called “logs.log” in the “logs” folder of the installed instance of ResFracPro. After you turn on detailed logging, leave the UI open for 15 minutes to accumulate some logs before sharing with ResFrac. The default path to this file is “C:\ResFracPro\logs\logs.log”.

To turn on detailed logging, go to the gear icon in the job manager and open the settings. Switch over to the User Settings tab, and select the box for “turn on detailed logging”:



In most cases, this is all you need to do to turn on detailed logging. You can turn detailed logging back off by unchecking this box.

The log file that gets created will be placed at C:\ResFracPro\logs\logs.log. After turning on detailed logging, you typically want to do whatever action that you are having trouble with. If your problem is simulation status or file sync related, wait at least 15 minutes to accumulate some results before sharing the file with ResFrac support. If your problem is with logging in, you will want to do a couple full log out / log in cycles before sharing with ResFrac

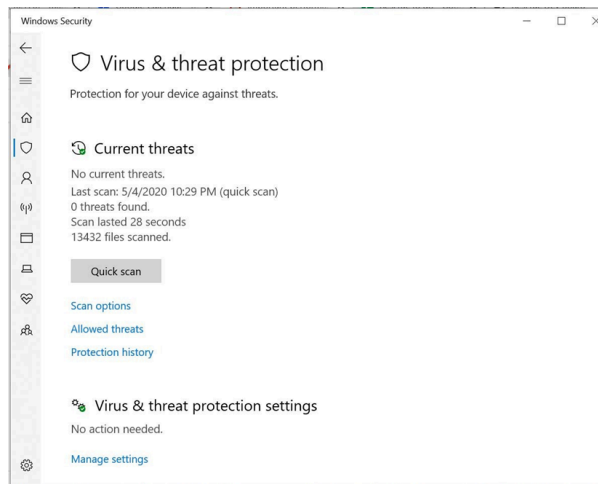
[What to do if detailed logging causes ResFracPro to be slow: Manage the Windows Antivirus](#)

We do not turn on detailed logging by default because detailed logging sometimes attracts the interest of the Windows antivirus software, causing ResFracPro to run very slowly. If you would like to send us a log, and you experience a slowdown in the UI after turning on detailed logging, you can use the steps below to tell the Windows antivirus software to back off on ResFracPro. These steps are not necessary but can be convenient if you want to continue using ResFracPro and are being hindered by the slowdown. Note that some companies prevent their users from modifying their security settings, so the below steps may or may not be available to you depending on how your company’s IT configuration is set up.

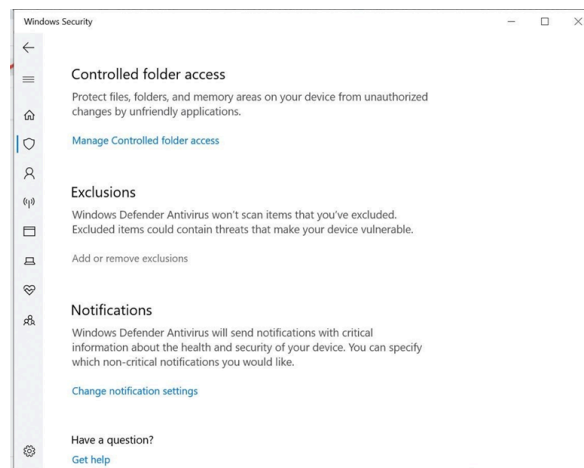
1. Go to the settings for Windows Security:



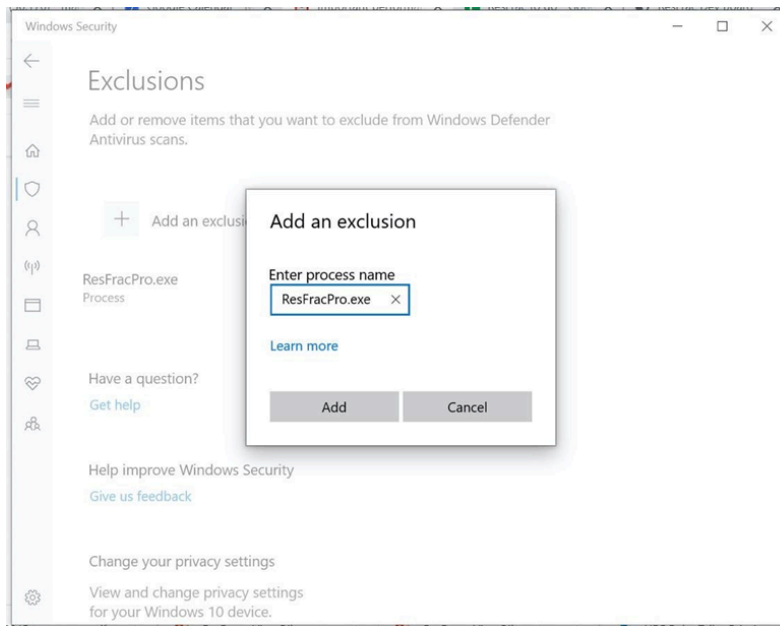
2. Go to Virus and threat protection, and click on manage settings:



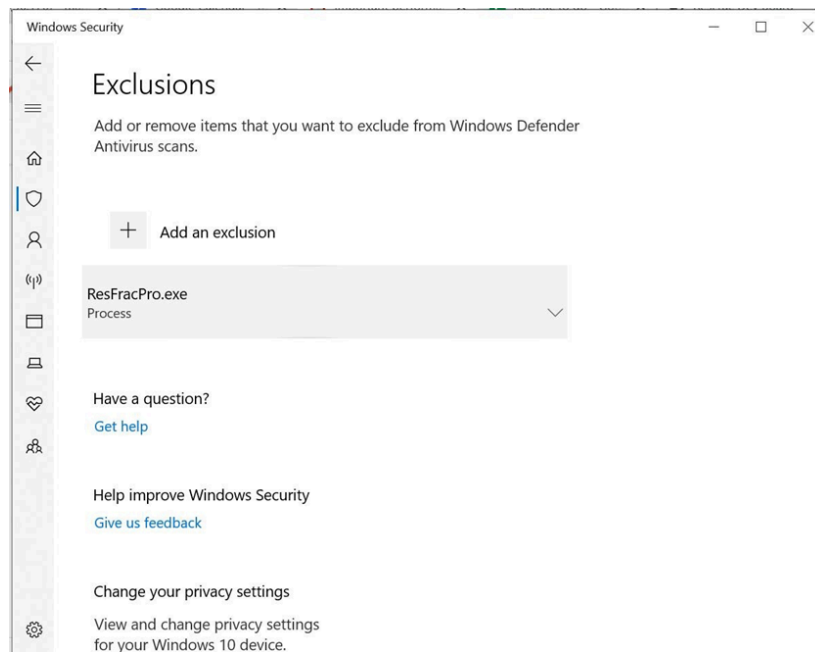
3. Scroll down and select exclusions:



4. Add an exclusion for a process, and type in ResFracPro.exe



5. It should look like this:

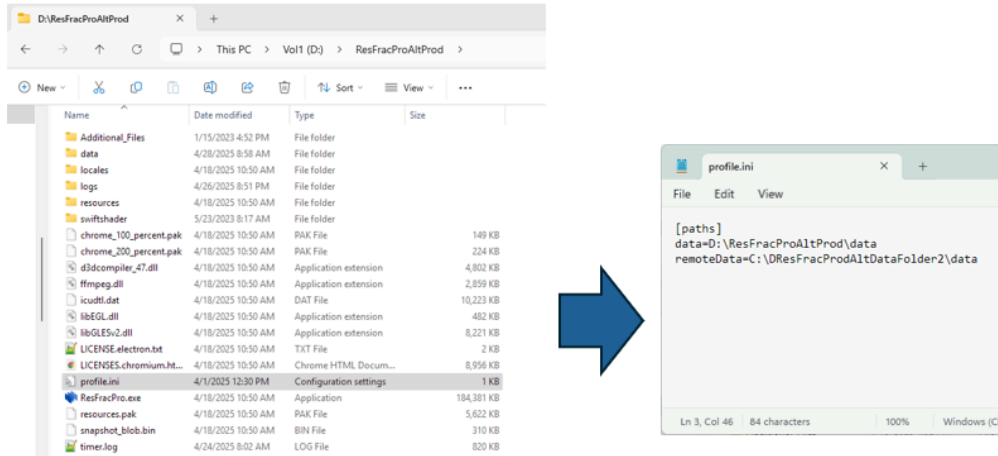


If the exclusion works properly, then ResFracPro will no longer feel sluggish. If the Windows antivirus is slowing down your machine, you'll see a process in the Windows task manager called "Antimalware Service" with high CPU usage. If that happens, try repeating the steps above and make sure you have the exclusion on the "process" ResFracPro.exe.

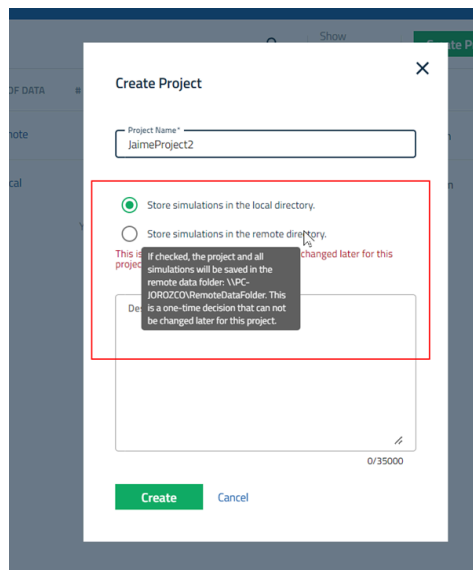
Q: How do I specify an additional 'remote' location to store simulations?

ResFracPro offers support for two data folders: a local folder, which is automatically configured, and an optional remote folder designed for use with external or network drives.

A new setting, "remoteData" (string), can be added to the profile.ini file to configure the remote data folder and activate this feature. You'll then need to restart ResFrac to see the change. Unlike the local data folder, the remote data folder does not contain a main.db file. However, data in both folders remains fully operational, and simulations are entirely contained within their respective simulation folders.

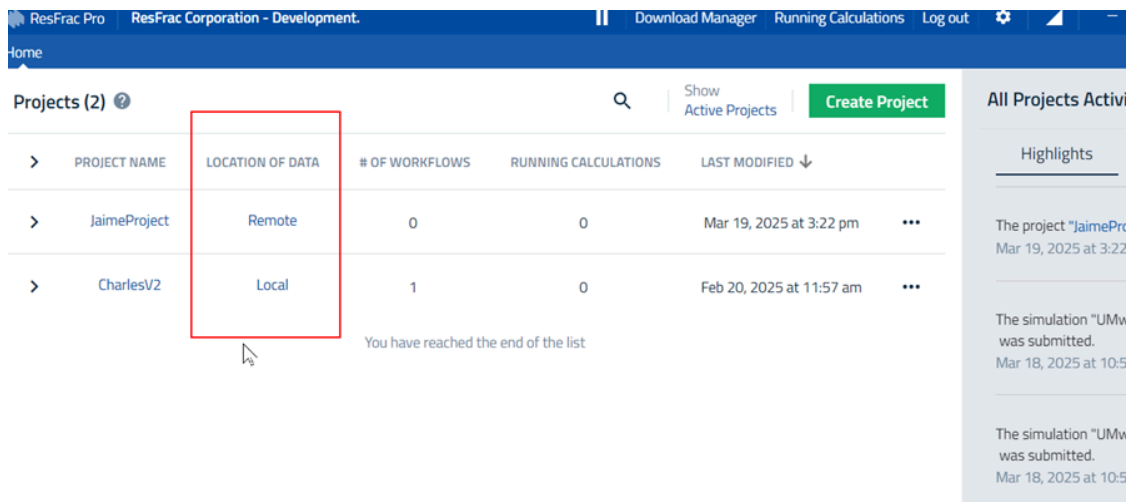


When creating a new project with the remoteData feature active, users will be prompted to decide whether the project will reside in the local or remote data folder. This decision applies on a per-project basis and is irreversible for the given project. To assist users in making an informed choice, the "Create Project" dialog includes an additional option labeled: "Store simulations in the remote directory. This is a one-time decision that cannot be changed later for this project."



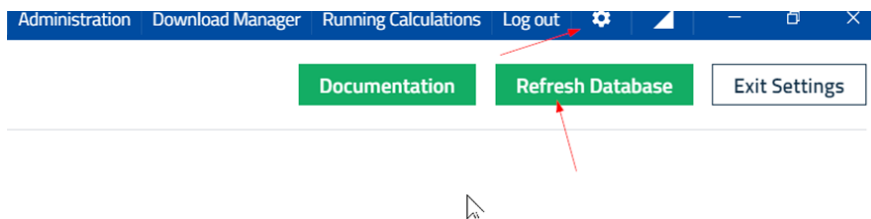
This feature also ensures that no two projects can share the same name, even if one resides in the local folder and the other in the remote folder.

To better identify whether each project is remote or local, a new column titled "Location of Data" has been introduced in the project manager panel. Users can now hover over either the project name or the "Location of Data" column to reveal the complete path to the directory where the project is stored. This enhancement improves accessibility and simplifies project management.



Manually changing file locations between remote and local projects

If users need to move a simulation or workflow from a local project to a remote project, or vice versa, this must be done manually through Windows Explorer. Afterward, they must use the "Refresh Database" button located in the Settings panel.

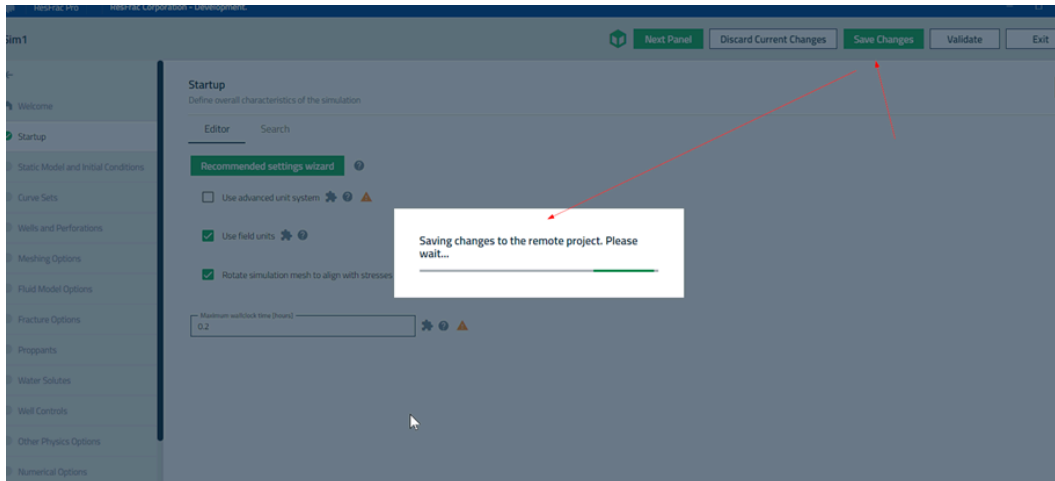


In the event of a name conflict between a project in the local and remote data folders during a database refresh action:

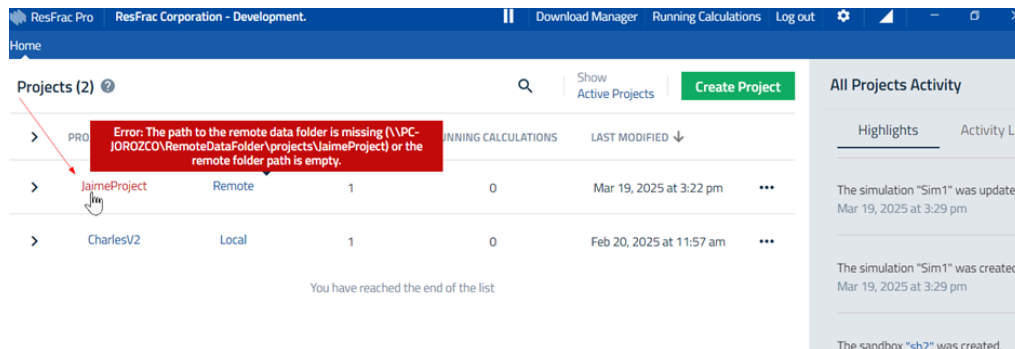
1. If a project with the same name already exists in the database, the other project is ignored, and a message is displayed in the refresh database dialog. Users are informed of the conflict and advised to rename the extra project folder before attempting to refresh the database again.
2. If neither project is in the database, the project from the local folder is loaded and added to the database. A message notifies users that the conflicting project in the remote folder was not loaded due to the naming conflict and suggests renaming the folder to resolve the issue before refreshing again.

Working in Remote Projects

When opening the builder within a simulation that belongs to a remote project, delays may occur during the reading and writing of project files due to the nature of the network folder. To address this, a progress bar has been added to indicate that changes are being saved, providing users with clear feedback during the process.



If the remote folder does not exist or it's not accessible we highlight the project in red to illustrate the error.



Q: How do I save all of my simulations in a different location such as a network drive?

Summary

ResFracPro is typically installed on the C drive of the client's machine, and simulations are saved locally. We prefer the C drive because of a peculiar limitation of Windows. If a file's 'path' in the directory is greater than 255 characters, then it will be unable to open, and you will get vague error messages from Windows. Because ResFrac uses nested folder directories (and simulation names can be pretty long), we prefer to place ResFrac in the C drive to keep the path as short as possible, and avoid the potential for a Windows bug.

Due to space constraints, it is possible to set the data folder location to be a network drive. This should only be done if the network drive is accessed over a high speed connection. If the network connection is slow, ResFracPro will feel sluggish - particularly during operations within the [visualization tool](#).

Procedure

If you want to move the data folder to a network location, here is the recommended procedure:

1. Exit ResFracPro
2. Copy or move the data folder from the default location (C:\ResFracPro\data) to the new location (let's call it N:\data) to continue working with the same simulations in the new drive.
3. In profile.ini (in main ResFracPro folder), update the path for the data folder to the new location (N:\data).
4. Reopen ResFracPro. Check to see that everything shows up as expected.
5. (optional) Check the checkbox on the Settings screen > User Settings tab (accessed via the gear icon at the top right of the main window) for "Use local folder for uncompressing downloads."
6. (optional) Remove the original data folder.

In the procedure above, step 2 is only necessary if you want to continue working with the same simulations you have from before. If you do not copy the data folder to the new location, a new empty data folder will be created there and ResFracPro will display no simulations. The optional step 5 can help with performance in certain cases when the data folder is placed on a network location, depending on the details of how the client computer is set up.

Alternatives

If you're running out of space, a good alternative to moving the entire data file to a network location would be to periodically export old simulations to a network drive for archival purposes, and delete them from the local data folder. Later, if you need to access old simulations, you can import them from the network location as required.

Q: How can I change the font size in the UI?

This option is available using the following shortcuts:

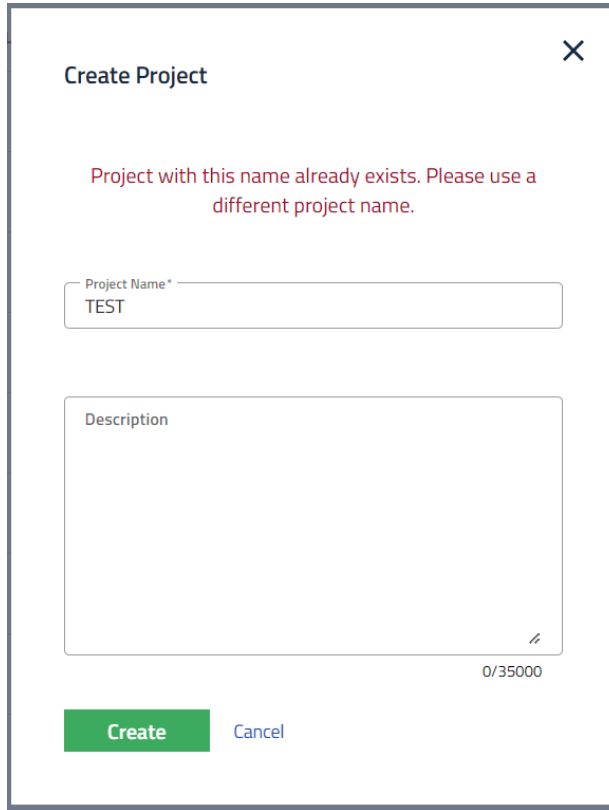
Zoom in: **Ctrl + shift + +**

Zoom out: **Ctrl + -**

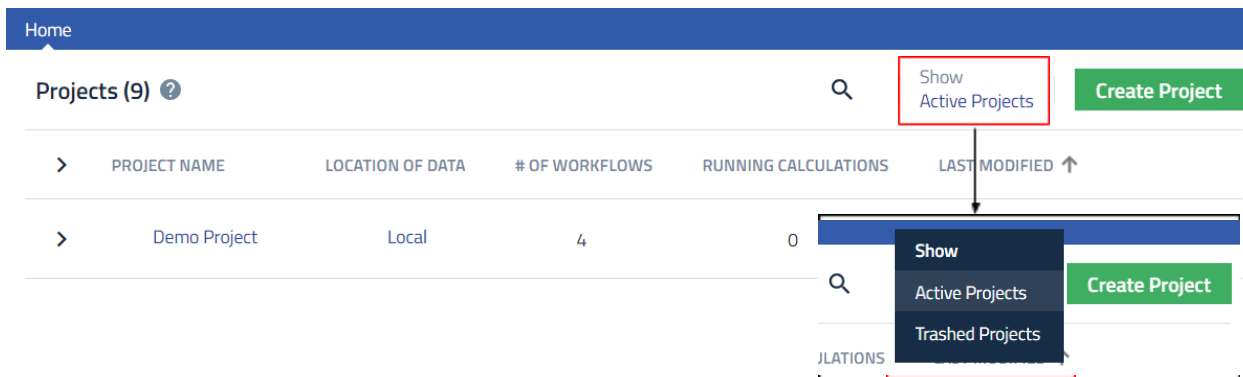
Reset: **Ctrl + 0**

Q: I tried to create a new project, sandbox, or simulation, but it says that name already exists. Why?

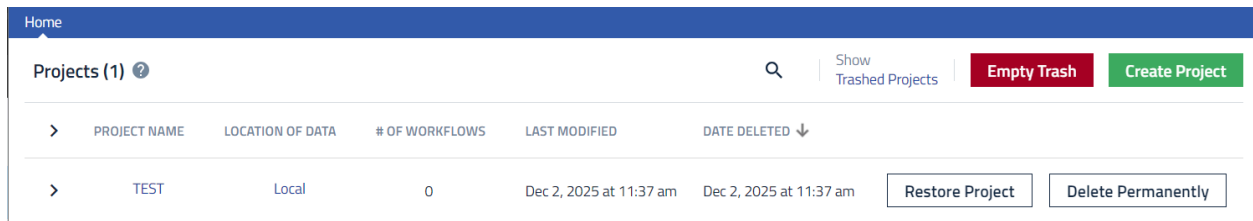
If you are creating a new project, sandbox, or simulation, you must assign it a unique name. If not, you will get the following error message:



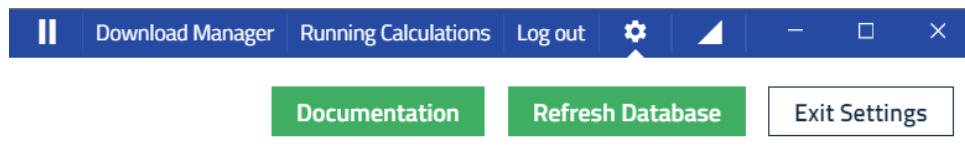
The name must be unique among both active *and* trashed items in the folder. If you do not see an item matching the name you attempted visible in the UI (listed as 'Active'), use the 'Show' option to view the 'Trashed' items instead.



Once in the 'Trashed' folder, you can choose to 'Restore' or 'Delete Permanently' each item. If you would like to name a new project (or sandbox or sim) to the name of a trashed item, you will need to permanently delete that item.



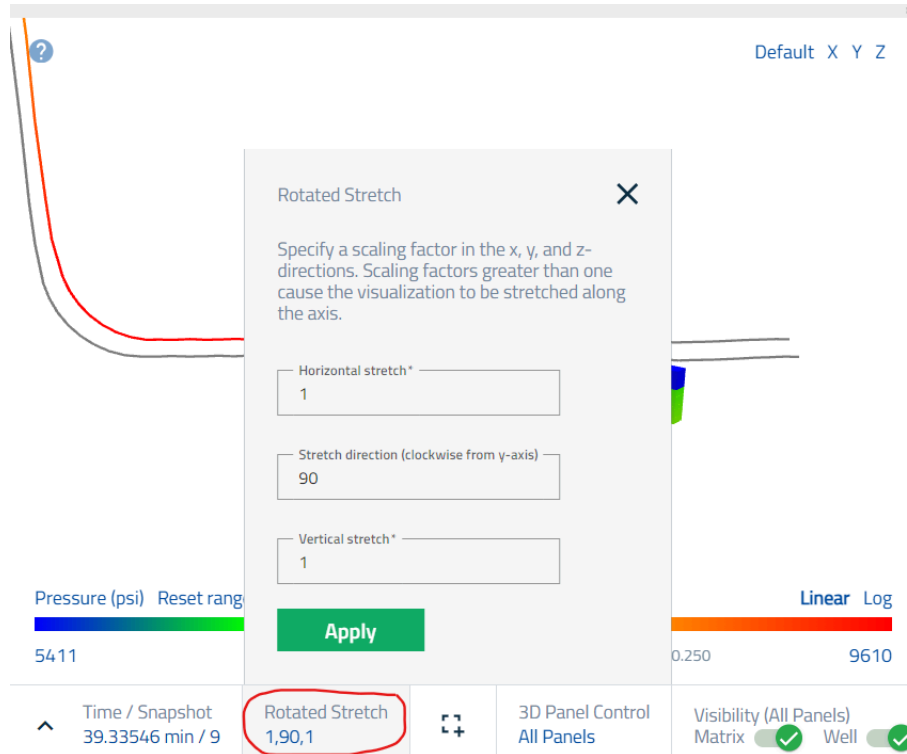
Alternatively, if you would like to archive the item, you may restore, and move the item out of that location using file explorer. If you choose that route, after moving the item you will need to 'Refresh Database' in the settings so the ResFrac file structure is up-to-date in the UI.



3. Visualization (Viz) Tool

Q: How do I use the 'rotated stretch' option in the visualization tool?

It is often useful to stretch the 3D model view in the viz tool in either the horizontal or vertical direction. The rotated stretch feature allows you to do this, and can be found near the bottom left of the viz tool window.

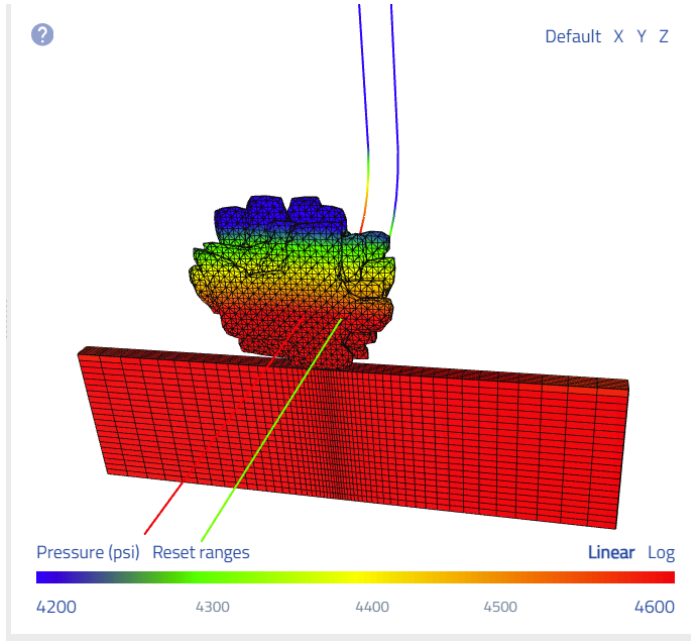


Rotated stretch takes three inputs and is based on cylindrical coordinates. The first field controls the magnitude of horizontal stretch (1 = no stretch), second field controls the direction of stretch measured in degrees clockwise from the y axis (0 = apply horizontal stretch along y axis, 90 = apply horizontal stretch along x axis), and the third field controls vertical stretch (stretch along z axis, unaffected by horizontal stretch).

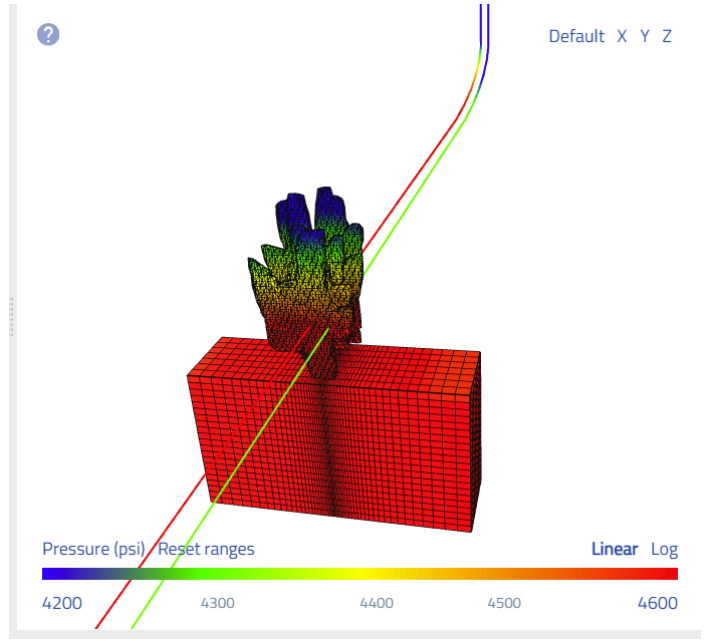
The default value populated in the second field (stretch direction) is the Shmin direction (90 degrees from the SHmax direction defined in the simulation builder). Leaving it at the default value makes it so that the stretch is perpendicular to the fracture propagation direction. Applying a horizontal stretch, therefore, causes the fractures to be spaced farther apart visually. Depending on how your well is designed, the default stretch direction may or may not be aligned with the wellbore.

The following screenshots show examples of the rotated stretch feature.

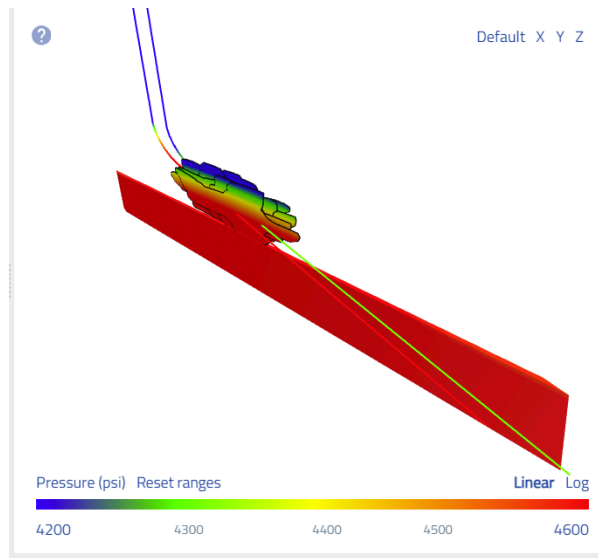
Horizontal stretch = 1, stretch direction = 90 (this has no effect if horizontal stretch is 1), vertical stretch = 1:



Horizontal stretch = 10, stretch direction = 90, vertical stretch = 2:



You can try stretching at an oblique angle to get a feel for what stretching does. Here's what it looks like with horizontal stretch = 10, stretch rotation = 45, and vertical stretch = 2:



Q: How can I save external datasets with my simulation to view using the visualization tool?

The “Additional line plots for history matching data” feature (located in the Output Options panel) allows you to append real field data (production rates, pressure, etc.) to the simulation. This data will then be available for plotting in the visualization tool to assist with history matching.

The data for these line plots can be added in two different ways: manually, or as part of the Production Sequence wizard.

The Production Sequence wizard offers the option to automatically include the production table as an external dataset in the ‘Additional line plots for history matching’ feature. This is useful for taking any real historical data you are already entering into your well controls and automatically saving them for plotting in the viz tool later. The wizard also comes with scaling and averaging options to apply to the data, and the option to append the data to a pre-existing external data set. One last feature is the option to use the production sequence wizard solely for creating additional line plot data, without simultaneously incorporating that same into the well controls.

Production sequence wizard

PRODUCTION TABLE

	Time from sequence start [days]	Water prod rate [STB/day]	Oil prod rate [STB/day]	Gas prod rate [Mscf/day]	BHP [psi]
1	0	468.490227	0.319931	0.479896	5918.984
2	1	364.749174	23.74733	35.620995	5844.022
3	2	178.701148	21.048375	31.572562	5787.42
4	3	128.285767	18.053001	27.079502	5740.888
5	4	106.33708	16.349083	24.523624	5699.391
6	5	90.006375	15.769419	23.654128	5660.86
7	6	78.909953	15.281988	22.922981	5625.851
8	7	70.306291	14.874761	22.312142	5592.733
9	8	65.001974	14.587236	21.880854	5558.954

New Row Resize Table

Include the 'Production table' in the 'Additional line plots for history matching' that are located in the 'Output options' panel.

Yes

Use the 'Averaging duration' specified in wizard when including the 'Production table' in the 'Additional line plots for history matching'.

Use the 'Optional scaling factor' specified in wizard when including the 'Production table' in the 'Additional line plots for history matching'.

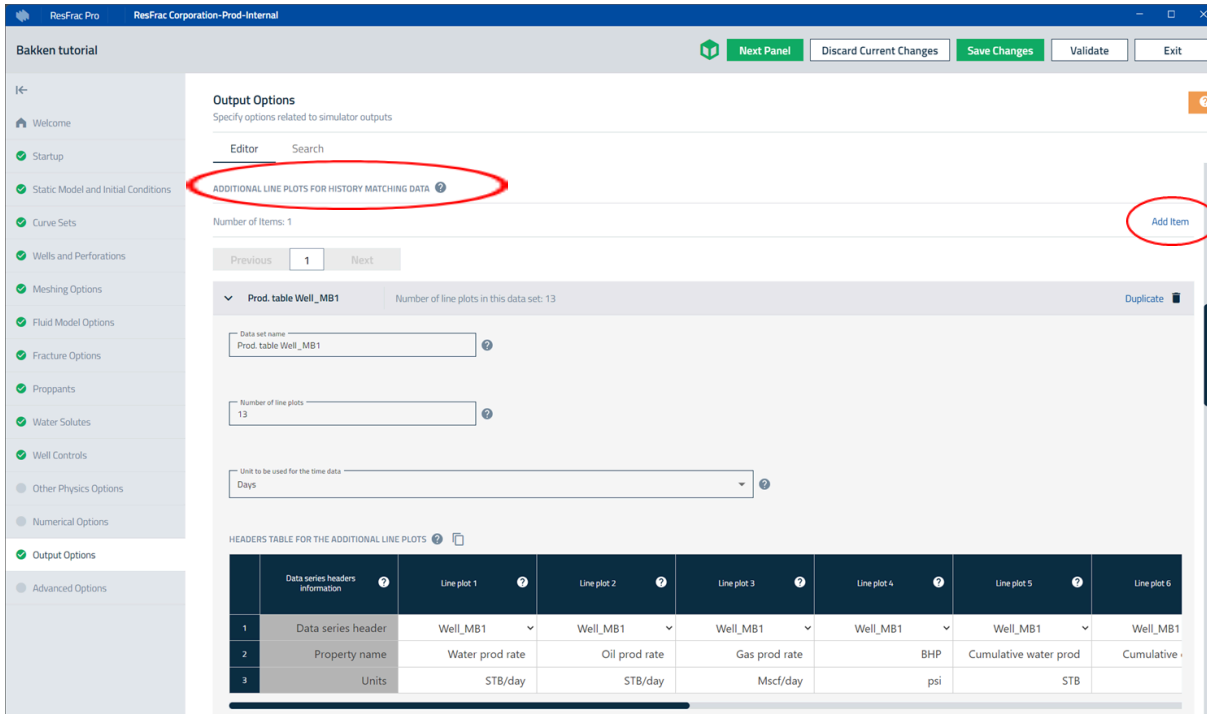
Use this wizard to ONLY include the 'Production table' in the 'Additional line plots', and NOT add a sequence to the well controls.

No

Combine to an existing 'Additional line plots for history matching'?

No

To manually add external data, go to the ‘Additional Line Plots for History Matching Data’ section in the Output Options panel. You will need to first select ‘Add Item’ to expand the section and add new datasets.



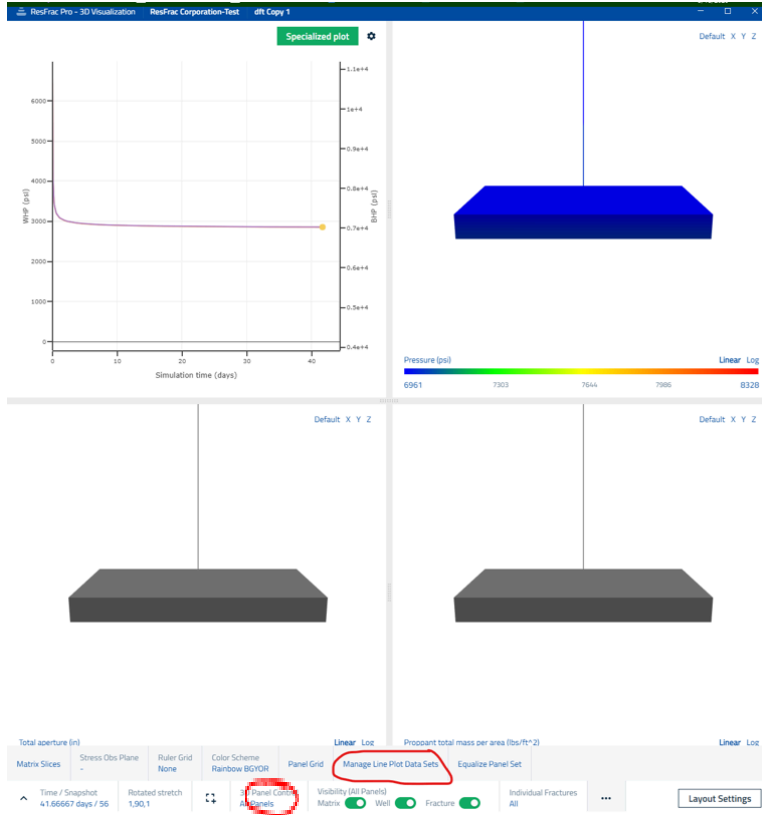
If the external data is not included as additional line plot data within the simulation prior to runtime, it can also be [imported within the viz tool](#) after the fact.

Q: How can I import external datasets into the visualization tool?

We often want to plot our simulated data alongside field data for history matching purposes. This historic data will be automatically accessible for viewing in the viz tool if it was added as “[Additional line plots for history matching data](#)” within the Output Options of the simulation builder at run time. However, if the external data was not added before running the simulation, it can be imported within the viz tool itself.

To import data into ResFrac, you will use the “Manage Line Plot Datasets” feature in the visualization tool to plot field data alongside simulation data. This guide will walk you through the process.

You will find the button for “Manage Line Plot Datasets” at the bottom of the viz tool window (if not visible, expand the ellipses ...):



Put your time series data into a CSV file formatted the same as the sim_track_XXXX.csv file in the simulation results. To make sure that you have the right format, I recommend copying an existing sim_track_XXX.csv file and clearing out the data. (To navigate to the sim_track file, [click simulation name in Job Manager window] > “Open Simulation Folder in File Explorer” > “Results” subfolder > [locate sim_track]).

The first four rows need to be header data:

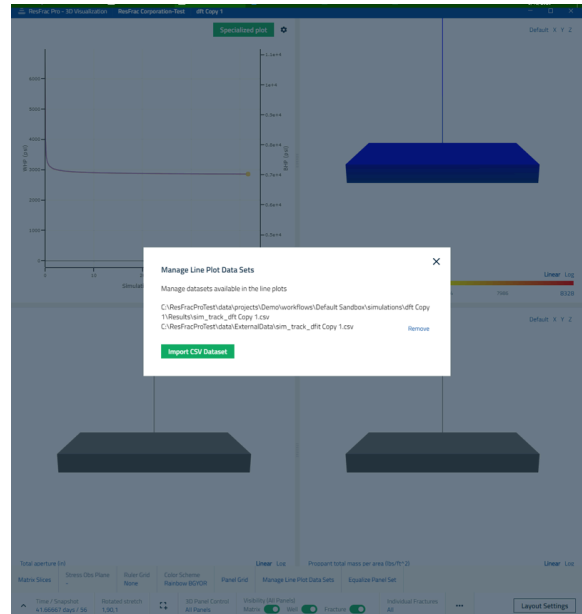
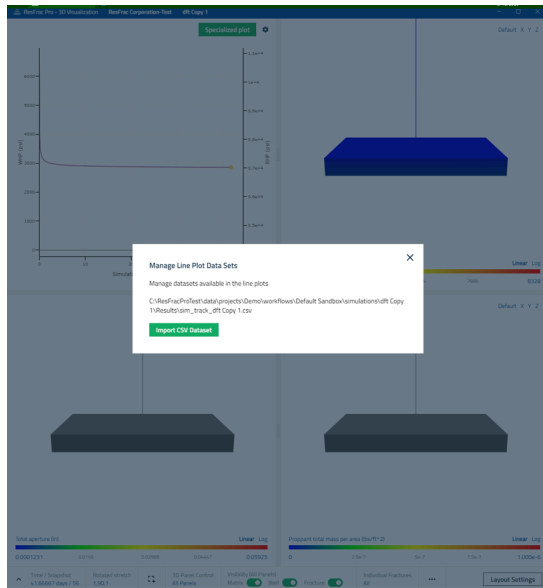
1. The first entry on the first row should say either, "Using field units", "Using metric units", or "Using Advanced Units,"
2. The second row contains well names,
3. The third row indicated the data series names,
4. And the fourth row indicates units.

Then the fifth (and following) rows contain the data. The first column has to be time. Later columns can be external data, such as pressure. Example:

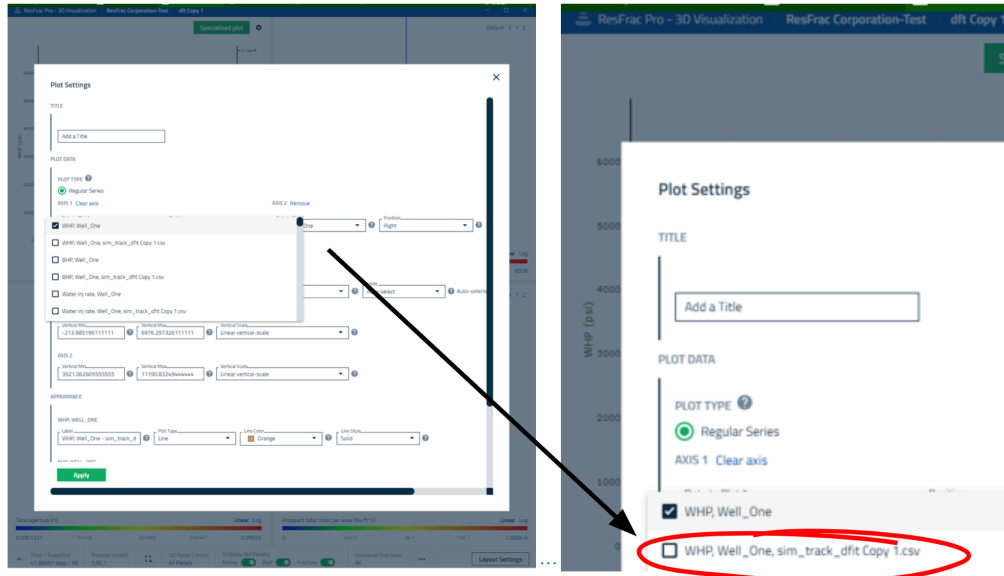
	A	B	C	D	E	F	G	H	I	J
1	Using field units	Initial reservoir pressure [psi]:	7000							
2		Well_One	Well_One	Well_One	Well_One	Well_One	Well_One	Well_One	Well_One	Well_One
3	Time	WHP	BHP	Water inj rate	Oil inj rate	Gas inj rate	Water prod rate	Oil prod rate	Gas prod rate	Instantaneous water cut
4	hours	psi	psi	STB/min water	STB/min oil	Mscf/min gas	STB/day water	STB/day oil	Mscf/day gas	unitless
5	2.78E-07	145.52893	4284.5511	0.000754777	0	0	0	0	0	nan
6	8.33E-07	145.54402	4284.5513	0.002264332	0	0	0	0	0	nan
7	1.94E-06	145.60757	4284.5514	0.005283106	0	0	0	0	0	nan
8	4.17E-06	145.81489	4284.5516	0.011319311	0	0	0	0	0	nan
9	8.61E-06	146.33799	4284.5517	0.023383101	0	0	0	0	0	nan
10	1.75E-05	147.45911	4284.5517	0.047467248	0	0	0	0	0	nan
11	3.51E-05	149.72212	4284.5517	0.095450092	0	0	0	0	0	nan
12	7.02E-05	154.21728	4284.5517	0.1906729	0	0	0	0	0	nan
13	0.000139192	163.06273	4284.5517	0.37821232	0	0	0	0	0	nan
14	0.000273146	180.20358	4284.5879	0.7421921	0	0	0	0	0	nan

Note that if you save from Excel, sometimes Excel adds in extra blank cells that get saved to the resulting CSV. Depending on where they are, these blank cells may cause problems when loading into the UI. So if you have trouble with loading a CSV file you created in Excel, try opening up the CSV file in a text editor and check to see if there are stray empty cells at the far right and end of the file.

After you have formatted your CSV file, open up the manage line plot data sets tool and tell it to import the CSV dataset. The new dataset will then show up on the list:

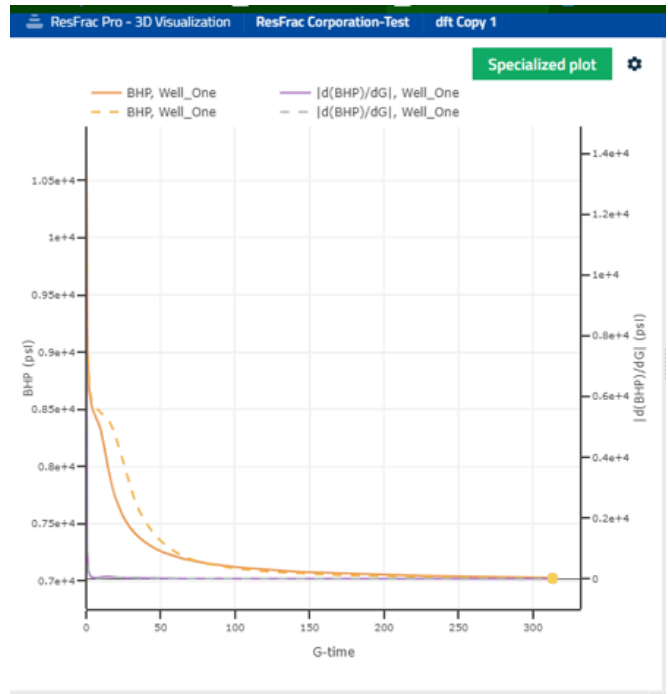
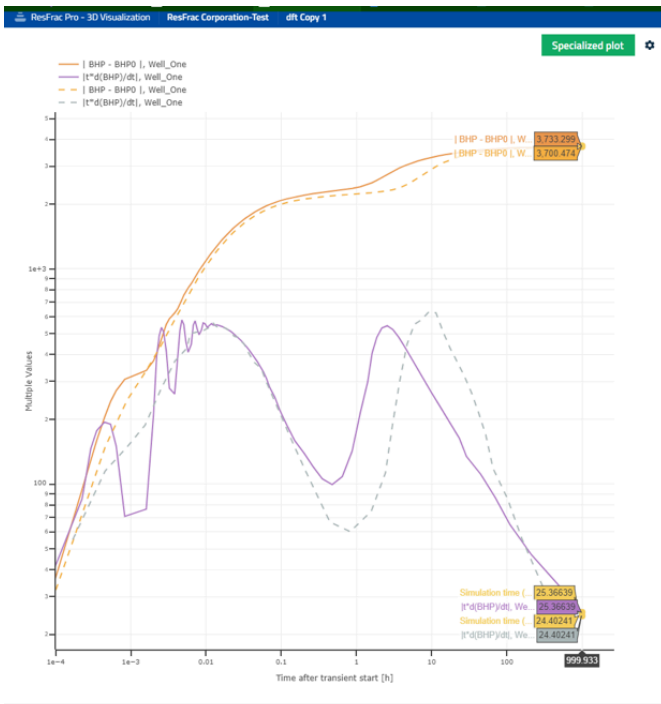


When you plot, you will be able to choose data series from both the simulation and from the imported data.

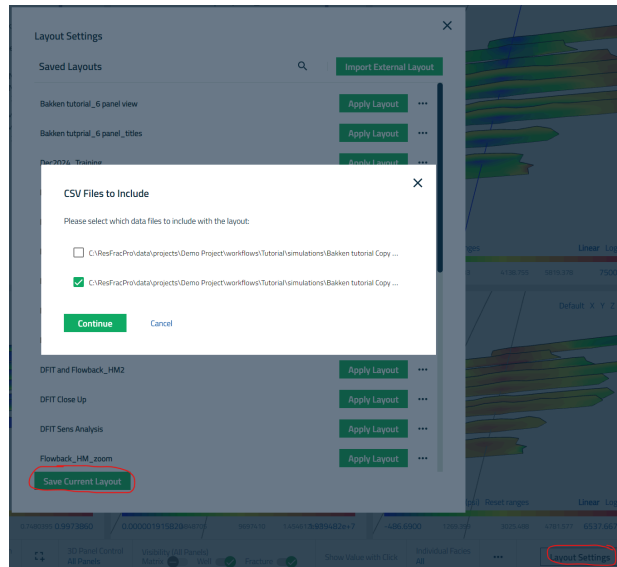


By default, the external datasets will be plotted with dashed lines:

The imported data will also show up in the specialized plots (such as the RTA, log-log, and G-function plots).



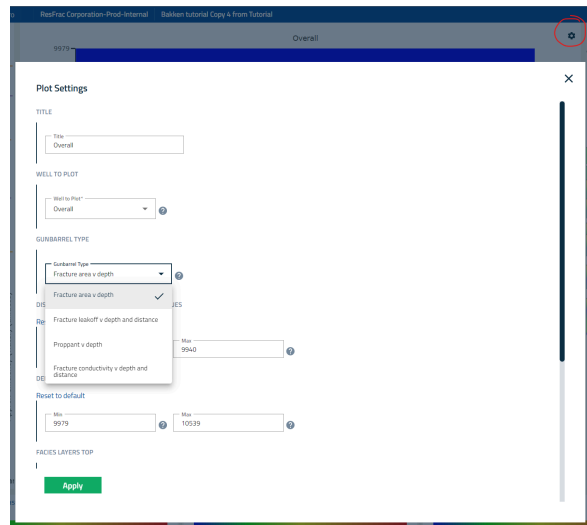
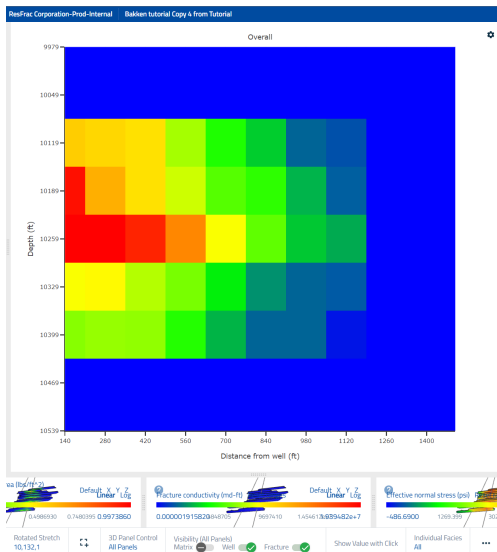
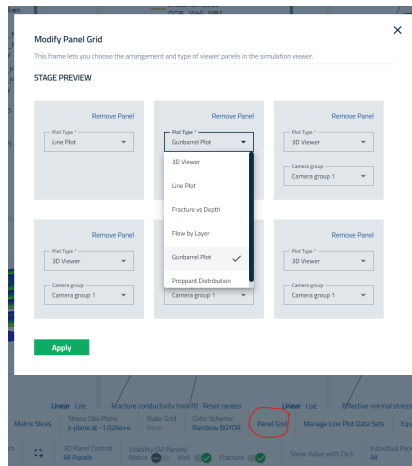
If you save your visualization layout, you can choose to save the external data files with the layout, so these files won't need to be imported every time you re-open the viz tool.



Q: Can I plot the gun barrel summary of simulation results?

Yes, there is a gun barrel plot option in the viz tool. Go to Panel Grid and change the plot type of one of your panels to “Gunbarrel Plot”. In the settings, you can choose to plot fracture area, fracture leakoff, fracture conductivity, or proppant, and change the size of the gunbarrel view. The gunbarrel view shows properties stacked along all fractures. On the x-axis, it shows ‘actual distance’ from the well, not ‘distance along strike.’ The raw data files being used for these plots are located in the [Raw Res folder](#) (and the file names end in “_v_depth_and_distance.csv”).

Note that while the plot shows only one side of the fractures, the plot is actually counting ‘both sides’, and showing properties as a function of depth and distance (regardless of which side they are on). So for example, if 2 lbs/ft² were placed on one side of a fracture and 2 lbs/ft² were placed on the other, the gunbarrel plot shown below would show 4 lbs/ft².



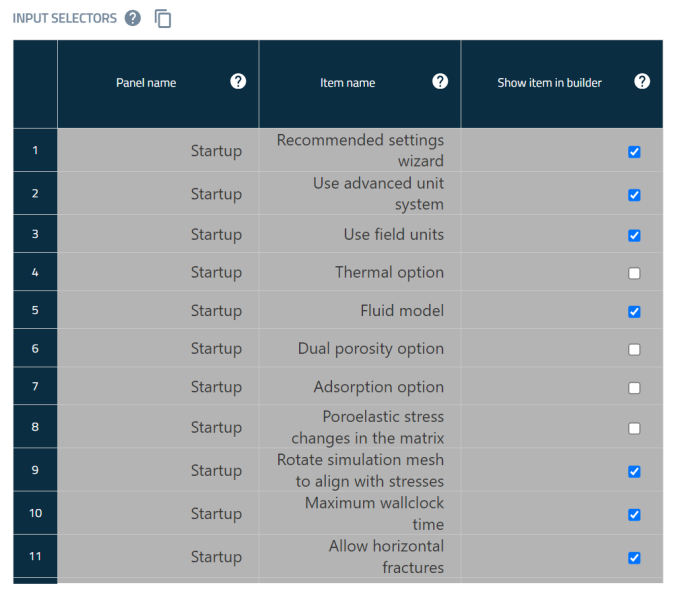
4. Simulator Inputs and Outputs

Q: I can't find some input parameters in the builder. How can I show these if they are hidden?

The builder user interface has the ability to show or hide every individual setting. This can be useful for collaboration - to emphasize inputs relevant to a colleague - or to streamline your own experience editing simulations. When an item's visibility in the 'Input Selector' table is false, it is no longer shown in the builder; however, the parameter will still be saved and loaded in the simulation's text input and settings files.

If there is a parameter that you can't find in the builder, it may be hidden. Go to the 'Advanced Options' panel, then at the top of the page is the 'Input Selectors' table. Check the box in the 'Show item in builder' column for your desired parameter. If you want to reset the settings and show everything, make sure the first row of the table is checked. Then, right click on the cell and select 'Fill down.'

The 'Input selector import/export wizard' above this table allows you to save and load preset combinations of 'hide/show' selections.



	Panel name	Item name	Show item in builder
1	Startup	Recommended settings wizard	<input checked="" type="checkbox"/>
2	Startup	Use advanced unit system	<input checked="" type="checkbox"/>
3	Startup	Use field units	<input checked="" type="checkbox"/>
4	Startup	Thermal option	<input type="checkbox"/>
5	Startup	Fluid model	<input checked="" type="checkbox"/>
6	Startup	Dual porosity option	<input type="checkbox"/>
7	Startup	Adsorption option	<input type="checkbox"/>
8	Startup	Poroelastic stress changes in the matrix	<input type="checkbox"/>
9	Startup	Rotate simulation mesh to align with stresses	<input checked="" type="checkbox"/>
10	Startup	Maximum wallclock time	<input checked="" type="checkbox"/>
11	Startup	Allow horizontal fractures	<input checked="" type="checkbox"/>

Q: How can I reduce the size of my simulation downloads?

ResFracPro defaults to outputting frequent simulation results. This is usually the safest bet, because simulations need to be rerun if you later need additional outputting. However, with large simulations or running many simulations at once, file sizes can become burdensome and you may want to choose less frequent or fewer simulation outputs. This guide will point you to parameters to reduce simulation storage size.

By typical order of importance, the following items strongly influence file size:

1. 3D view files for matrix and fractures
2. 3D view files for stress observation planes
3. '[Raw results](#)' files

First, checking 'Lite matrix export' reduces the file size of each individual snapshot by reducing the number of parameters included in the 3D visualization. Leaving this checked is appropriate for most simulations. If you are running a thermal or compositional simulation, you may want to uncheck this setting for a more detailed output.



Second, to reduce the frequency of the 3D outputs, within the 'Output Options' panel, scroll down to the 'Advanced options'. The settings shown below all control the adaptive frequency of snapshot creation. If you cannot find the settings below, please refer [here](#). Increasing the output scaling proportions or output scaling pressure, temperature, saturation or relative fracture surface area will all reduce the snapshot frequency.

<input type="text" value="4.383"/>			← Snapshots will be created at a minimum every {this parameter} hours
<input type="text" value="99999999"/>			← Snapshots will be created at a minimum every {this parameter} timesteps
<input type="text" value="1.5"/>			← Snapshots will be created at points in time defined by this geometric growth factor starting at 5 min
<input type="text" value="0.5"/>			} These parameters control the frequency of snapshots based on changing properties in the simulation. For example, with these settings, when 25% of fracture elements have had pressure change by 725psi or greater, a new snapshot will be created.
<input type="text" value="0.05"/>			
<input type="text" value="0.25"/>			
<input type="text" value="50"/>			
<input type="text" value="725.1886885"/>			
<input type="text" value="541.67"/>			
<input type="text" value="50"/>			
<input type="text" value="0.1"/>			

Starting from the Fall 2025 release, there is a new parameter 'overall output frequency scaler', which allows you to scale the frequency of all these parameters simultaneously.

Output Options

Specify options related to simulator outputs

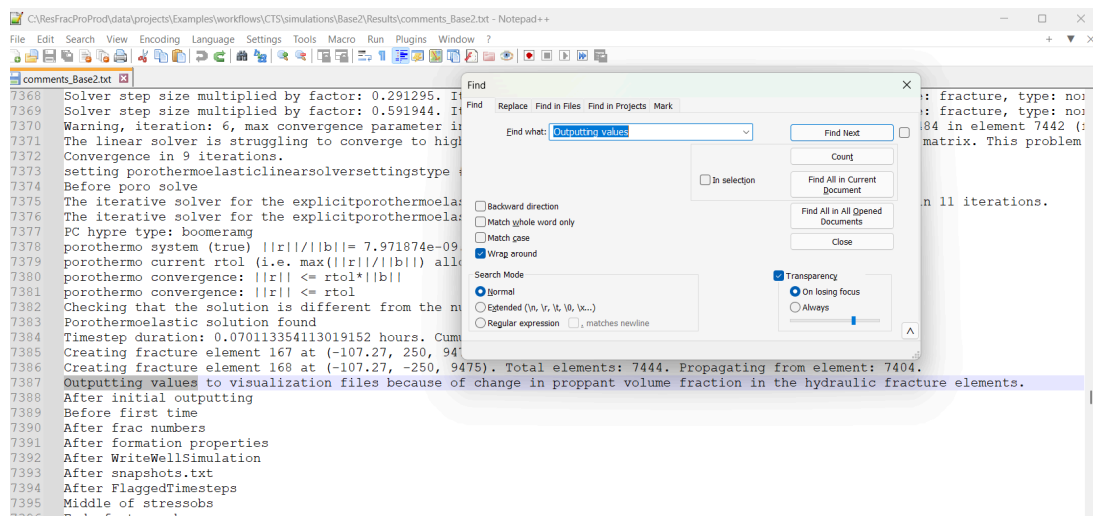
Editor Search

Output options library ?

Overall output frequency scaler

0.5 ?

If you would like to identify what is triggering visualization outputs at various times after running, open the comments file and search for “outputting values”. For each visualization step, it will be noted in the comments file and the trigger given:



For instance, if trying to reduce the number of snapshots during fracturing, you could increase the ‘Geometric output time factor’, ‘Relative minimum scaling fracture area of visualization output’ and ‘Visualization output scaling relative fracture surface area’ like below:

Visualization output time interval [hours]
4383

Visualization output timestep interval
99999999

Geometric output time factor
3

Visualization output scaling proportion in well elements
0.5

Visualization output scaling proportion in matrix elements
0.05

Visualization output scaling proportion in fracture elements
0.5

Relative minimum scaling fracture area for visualization output
500

Visualization output scaling pressure [psi]
725.1886885


Visualization output scaling temperature [F]
541.67




Visualization output scaling relative fracture surface area
500

Visualization output scaling saturation
0.1

Next, stress observation planes consume a large amount of storage, separate from the well/fracture/matrix visualizations. If you are using stress observation planes but they are no longer necessary, consider removing them. In the example below, the removal of stress observation planes reduces the simulation size by 16%.

Last, if not using the [raw results](#) (production by depth, gunbarrel fracture plots, etc.) turn off these outputs in the 'Output Options' panel:

Only output raw results on flagged snapshots   

Only output raw results on the final timestep   

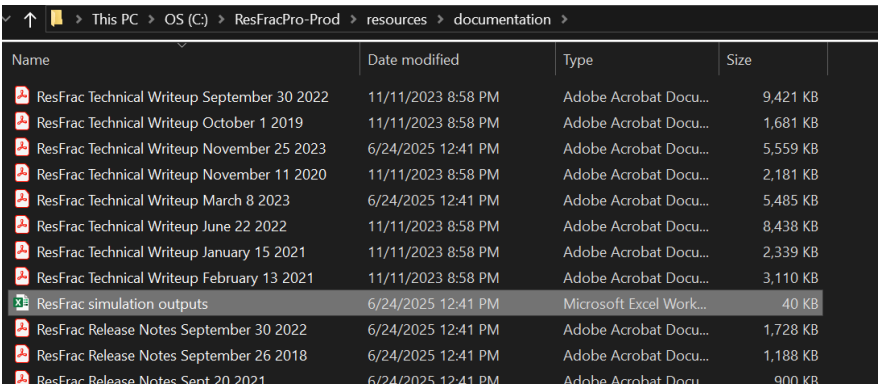
The following example shows the incremental simulation size change as a result of the steps listed above. This is based off of the Bakken [template](#) with settings changed to maximize file size for the base case:

Case	Folder size (MB)	% Reduction	File count	% Reduction	Snapshot count
Base Simulation	1945	0%	3962	0%	157

Checking "Lite matrix export"	1300	33%	3962	0%	157
Removing observation planes	994	49%	3488	12%	157
Checking "Only output raw results on the final timestep"	882	55%	358	91%	157
Adjusting output frequency to reduce snapshots	712	63%	290	93%	123

Q: Where can I find a list of all outputs in the 3D viz view as well as the line plots (sim_track) file?

If you are uncertain about what a given parameter means in the sim_track or in the 3D viz tool, please refer to the "ResFrac simulation outputs" worksheet located in the "documentation" folder in your install as shown below.



This file lists every parameter contained within the sim_track and 3D viz panels, model configuration requirements and 'Output options' parameters required for their reporting, as well as a detailed description of what the parameter represents.

Sim_track outputs example:

Aggregation level	Required model input/s	Attribute	Definition
Well	Well	WHP	Wellhead pressure. Output on a per well
Well	Well	BHP	Bottomhole pressure. Measured at the well
Well	Well	Water inj rate	Water injection rate in surface volumes
Well	Well, oil injection nonzero	Oil inj rate	Oil injection rate in surface volumes for
Well	Well, gas injection nonzero	Gas inj rate	Water injection rate in surface volumes
Well	Well	Clean fluid inj rate	Injection rate for clean fluid equivalent
Well	Well	Water prod rate	Water production rate in surface volumes
Well	Well	Oil prod rate	Oil production rate in surface volumes

3D outputs example:

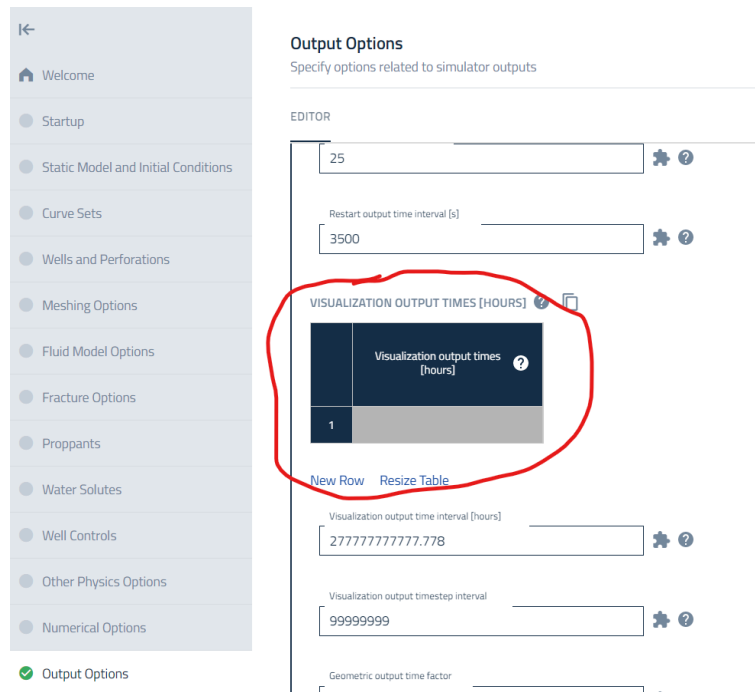
Parameter	Required input*	Shown in wellbore elements	Shown in fractures elements	Shown in matrix elements	Description
<Name of proppant> mass per area	One or more proppants must be specified		x		Amount of proppant per area of fracture
Calculated leak-off coefficient for each fracture. Average of	One or more proppants must be specified		x		Effective settling rate of chosen proppant
<Name of proppant> volume volume fraction	One or more proppants must be specified	x	x		Volume fraction of selected proppant
<Name of water solute> overall mass fraction	One or more water solutes must be specified	x	x		Mass fraction of the selected water solute
<Name of water solute> water phase mass fraction	One or more water solutes must be specified	x	x		Mass fraction of the selected water solute in water phase
<Name of component> <mass or molar> fraction in the <oil	None	x	x		Mass or mole fraction of component
<Name of component> overall mass fraction	None	x	x		Overall mass fraction of the component
<Name of component> mass fraction (excluding water)	None	x	x		Overall mass fraction of the component excluding water
<Name of component> mass fraction in the <oil, gas> phase	None	x	x		Overall mass fraction of the component in the <oil, gas> phase
Approx 'reservoir compressibility' leakoff coefficient	None		x		Calculated leak-off coefficient for each fracture
Average 1D submesh gas rel perm	1D submesh must be turned on (default)		x		Effective gas relative permeability used in 1D simulation
Average 1D submesh oil-gas mobility	1D submesh must be turned on (default)		x		Effective oil-gas mobility in the 1D simulation
Average 1D submesh oil-rel perm	1D submesh must be turned on (default)		x		Effective oil relative permeability used in 1D simulation

For description of the files in the '[Raw_res](#)' folder, there is a special file in that folder with the name 'AAA-README-For-Raw-Results'. Please refer to the file for more detailed descriptions.

Q: How do I change the frequency of simulation outputs?

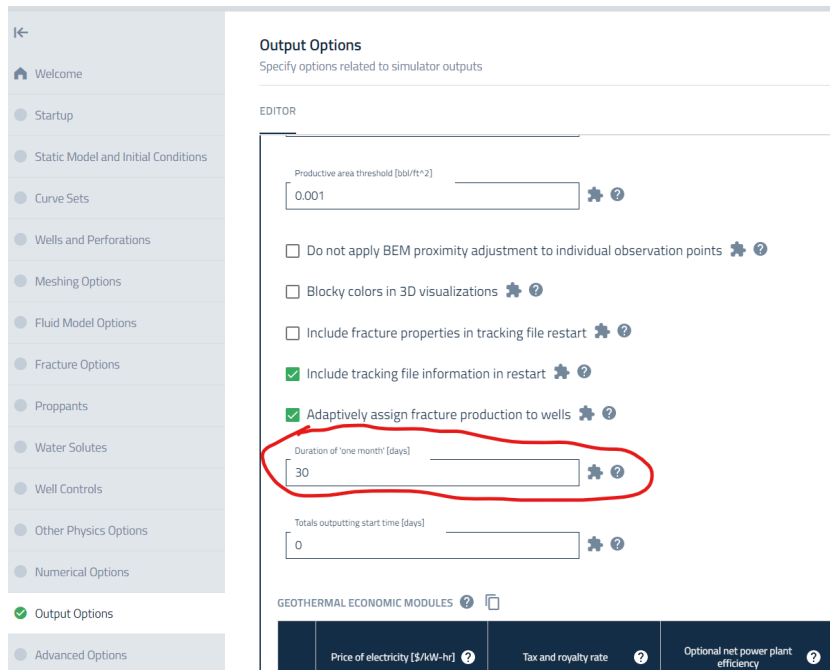
There are three locations where time series data are output from a ResFrac simulation:

1. Sim_track_{simulation_name}.csv
 - a. The sim_track file is the source of data plotted in the visualization tool (i.e. [click a simulation] > [visualize simulation results] > sim_track.csv contains the data plotted in the line plots).
 - b. Each row of the sim_track file corresponds to a timestep that the simulation has taken. You will usually see that the simulation takes more frequent timesteps during fracturing (on the order of minutes) than during production (on order of days or even months). To force the simulation to output data at a certain time, the user can specify 'visual times' in the advanced section of Output Options:



- c. At each specified visualization time, the simulation will be forced to take a timestep and print results to the sim_track file as well as to the 3D files. We recommend being cautious with this option as forcing excessive timesteps can slow down a simulation and/or make file sizes very large.
 - d. To navigate to the sim_track file on your hard drive, [click simulation] > 'Open simulation folder in file explorer' > 'Results' subfolder > [locate sim_track].
2. Daily_prod_{simulation_name}.csv

- a. The daily_prod file is also located in the Results subfolder. The simulation interpolates between points in the sim_track to estimate daily production, injection, and pressure values (and any advanced outputs), which is summarized in the daily_prod file.
3. 30_day_prod_{simulation_name}.csv
- a. Similarly, the 30_day_prod file is also located in the Results subfolder. Users can change from a 30 day interval to another by modifying the 'duration of one month' in the Advanced section of Output Options:

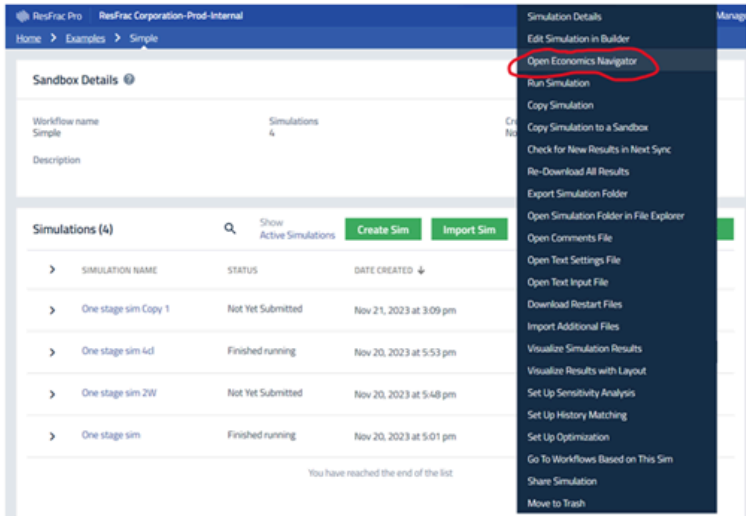


Q: How do I use the Economics Navigator?

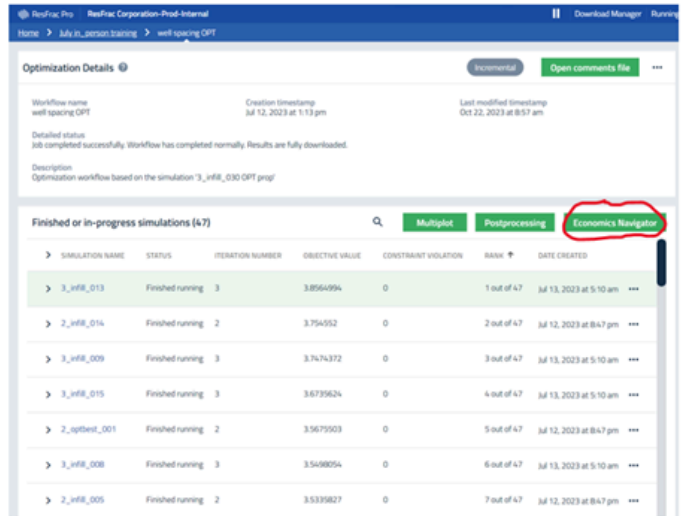
The Economics Navigator allows you to change economic models in simulations and workflows that have already been run.

From a simulation or workflow that is complete, select the Economics Navigator:

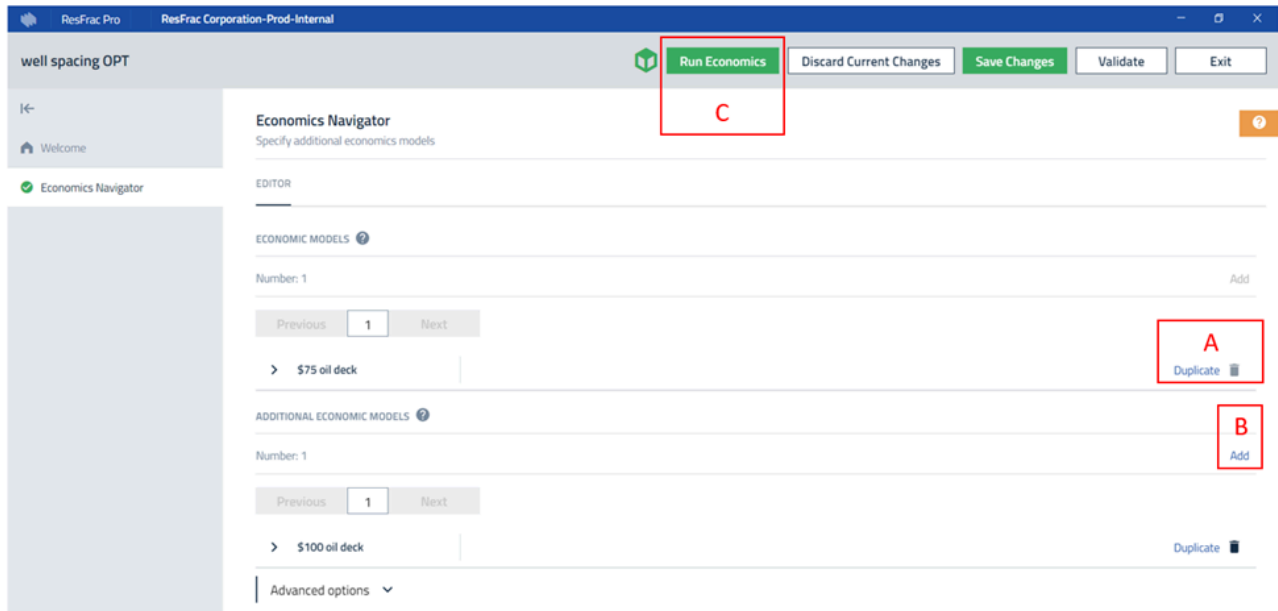
Accessing the Economics Navigator for individual simulations:



Accessing the Economics Navigator for workflows:



Within the Economic Navigator, any existing Economic Models are listed (“A” in the image below), and additional economic models can be entered by clicking the “Add” button (“B” in the image below). Once the new economic model is complete, you will need to “Run Economics” (“C” in the image below).



For each new economics model that is run, a new “addnl_econ_summary” csv is created. The daily and 30-day production files, as well as the sim_track file, will also be revised to include additional columns for each new economics model. Copies of the original production and sim_track files will remain in the Results folder of the simulation.

Name	Date modified	Type	Size	Date created
newviz	5/7/2025 9:28 AM	File folder		5/7/2025 9:10 AM
Raw_Res	5/7/2025 9:28 AM	File folder		5/7/2025 9:11 AM
30_day_prod_Bakken tutorial Copy 1	10/21/2025 10:35 AM	Microsoft Excel Co...	266 KB	5/7/2025 9:28 AM
addnl_econ_summary_\$75 Oil 10k Lateral Economics_Bakken tutorial Copy 1	10/21/2025 10:35 AM	Microsoft Excel Co...	357 KB	10/21/2025 10:35 AM
addnl_econ_summary_\$80 Oil 10k Lateral Economics_Bakken tutorial Copy 1	10/21/2025 9:56 AM	Microsoft Excel Co...	359 KB	10/21/2025 9:56 AM
comments_Bakken tutorial Copy 1	5/7/2025 9:28 AM	TXT File	5,557 KB	5/7/2025 9:28 AM
current_status	5/7/2025 9:28 AM	TXT File	1 KB	5/7/2025 9:28 AM
daily_prod_Bakken tutorial Copy 1	10/21/2025 10:35 AM	Microsoft Excel Co...	7,966 KB	5/7/2025 9:28 AM
fracture_info_file_for_testing	5/7/2025 9:28 AM	TXT File	43 KB	5/7/2025 9:28 AM
matrix_prop_names	5/7/2025 9:11 AM	TXT File	1 KB	5/7/2025 9:11 AM
misc_visualization_data	5/7/2025 9:28 AM	TXT File	7 KB	5/7/2025 9:28 AM
original_30_day_prod_Bakken tutorial Copy 1	5/7/2025 9:28 AM	Microsoft Excel Co...	115 KB	10/21/2025 9:56 AM
original_daily_prod_Bakken tutorial Copy 1	5/7/2025 9:28 AM	Microsoft Excel Co...	3,481 KB	10/21/2025 9:56 AM
original_sim_track_Bakken tutorial Copy 1	5/7/2025 9:10 AM	Microsoft Excel Co...	6,605 KB	5/7/2025 9:10 AM
sim_track_Bakken tutorial Copy 1	5/7/2025 9:10 AM	Microsoft Excel Co...	6,605 KB	5/7/2025 9:10 AM

The new economic model will also be inserted into the settings_XXX.txt file for the simulation under the keyword “additional economic models”:

```

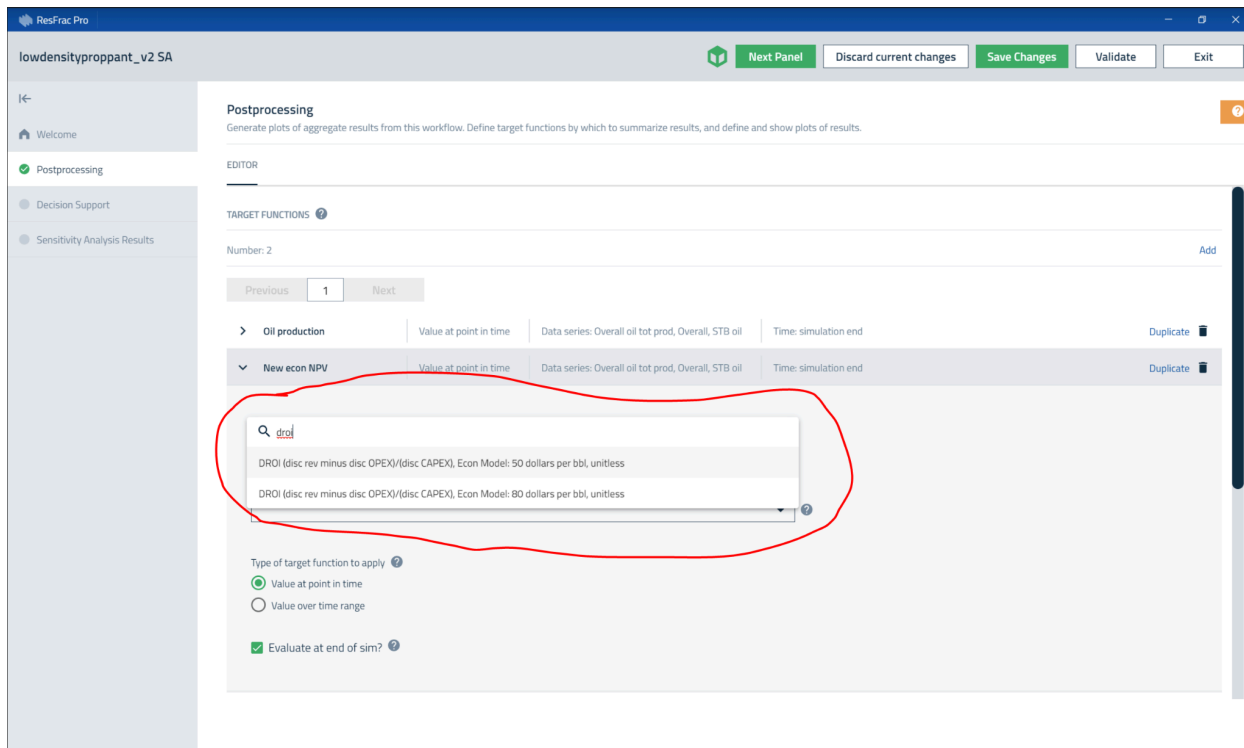
10281
10282 // ----- Begin entry -----
10283 //
10284 // Name in builder interface: "Additional economic models"
10285 //
10286 // Description:
10287 // Specify economic models used for economic analysis of the results after a simulation has run. For
10288 //
10289
10290 // ResFrac internal variable name
10291 Variable name:
10292 additional economic models
10293
10294 Length:
10295 2
10296
10297 Value(s) :
10298 //land cost ($/acre), drilling cost per well ($), fixed fracturing cost per well ($), fracturing prop
10299 25000 2500000 1000000 0.05 2.5 0 0 1 0.5 0.1 150000 30 2.5 0.2 0 0.75 50 0 0.05
10300 //operational cost per well per year (phase one), operational cost per well per year (phase two), ope
10301 50000 30000 20000 5 20
10302 //the number of wells:
10303 2
10304 //Stages per well, lateral length per well (ft), average well spacing (ft), include in model?, well s
10305 40 10000 400 true nan
10306 40 10000 400 true nan
10307 ---

```

Note that these new economics models are only stored locally and not on the cloud. If you use the “Share Simulation” function with this simulation subsequent to using the Economic Navigator, only the original sim_track (and original economics) will be shared.

Also, if you copy this simulation, it will clear out the “additional economic models” section in the new simulation. The new economics models will be added to the list of economics models in the builder for the new simulation.

Returning to your modified simulation or workflow, you can now open this simulation in the visualization tool or open the post-processing for the workflow, and both the new economics model and the original will be visible:



Q: What is in the Raw Res folder (nested in the Results folder of a simulation)?

The raw results folder contains files that are outputted in human-readable format to make it easy for users to access and post-process results. The “AAA-README-For-Raw-Results.txt” file in the ‘Results’ folder contains a detailed description of all of the following files:

- The frac_elms, mat_elms, and well_elms files output the locations and properties of every element.
- The perfsummary file outputs the locations of every perf cluster.
- The flow_by_layer file outputs net production and leakoff in each layer that is defined in the user-defined list of properties versus depth.
- The frac_v_depth file outputs various summary properties versus depth.
- The proppant_v_depth_and_distance, fracturearea_v_depth_and_distance, and fractureleakoff_v_depth_and_distance files provide gun-barrel views of proppant placement, fracture area, and fracture leakoff along the well(s).

Multiple versions of the flow_by_layer, frac_v_depth, and gunbarrel (depth_and_distance) files are outputted: one for each well and one for 'overall'. If there is only one well, then there is just one file output for each (since the overall and 'per well' files would be identical).

The output frequency of the raw results files are dictated by parameters in the ‘Output Options’ panel. By default, raw results files are created every time a visualization 'snapshot' is outputted by the simulator. The files are then numbered based on the corresponding timestep. This [output frequency can be reduced](#) by selecting

'Only output raw results on flagged snapshots' in the 'Output Options' panel in the builder. This will reduce the number of raw results files that are created to only snapshots that are flagged by the simulator as being important - this is mostly when wells are either shut-in or turned on. Alternatively, checking 'Only output raw results on the final timestep' will prevent outputting these files until the very end of the simulation.

By default, 'Output raw results files for matrix elements' is unchecked. This means that the mat_elms files are only output at the beginning and end of the simulation, and not when the other raw results files are output. This is the default behavior because the mat_elms files are fairly large, and are not ever accessed by the majority of day-to-day ResFrac users. Thus, the default behavior is to not download them as frequently in order to save hard-disk space and download bandwidth.

Output Options
Specify options related to simulator outputs

Editor Search

New Row Resize Table

RESTART TIMES [HOURS] ?

	Restart times [hours] ?	
1	98155	
2	98319	

New Row Resize Table

Never output to visualization files ?

Output raw results files for matrix elements ?

Print raw results ?

In the frac_elms files, each row represents a single fracture element. The columns specify the (x,y,z) location of the center of the element, the orientation, and then a variety of properties such as pressure and saturation. Note that because the fractures are assumed to be vertical, it is not necessary to specify dip. The length and height of each fracture element are uniform and are specified in the top row of the file (along with the timestep and simulation name).

In the well_elms file, each row represents a single wellbore element vertex. Each well element is defined between two vertices, which are contiguous. So if there are 100 well elements, there will be 101 rows corresponding to 101 vertices. Data for multiple wells are stacked on top of each other and can be distinguished based on the 'Well name' column. The first row provides the coordinates for the toe of the well and properties for the first element (defined by the vertices in rows 1 to 2). The subsequent rows follow the same pattern, with the last row for each well representing the wellhead vertex. For this last row, there are no remaining elements to define, so the properties for the last element of the well are repeated (listed in the last two rows for each well).

In the `mat_elms` file, each row represents a single matrix element. The file gives the (x,y,z) position of the center of the element, along with various element properties. In simulations using a rectilinear grid, there are columns specifying the size of the element in the x, y, and z-directions.

Two versions of the `flow_by_layer`, `frac_v_depth`, and `gunbarrel (depth_and_distance)` files may be outputted: a version for each well, and an 'overall' version (i.e., sums over all the wells). 'Per well' versions of the files are output only if the parameter 'Output per well raw results files' is checked.

Here is how the 'per well' files work: every fracture element is assigned to one well. When a new fracture is initiated from a well, it is assigned to that well. When a fracture element propagates and a new element is created, then the new element is assigned to the same well as the fracture element it initiated from. This system works well in simple cases, but be aware that when there are a lot of wells with fractures crossing each-other, the well-assignments can become confused. For example, if a fracture propagates from 'Well A', and then crosses 'Well B', and then you inject into that fracture from 'Well B', that fracture remains assigned to 'Well A'. This is an inherent limitation to the 'per well' files, and so they should be used with caution. If you turn on the option 'Adaptively assign fracture production to wells', then the assignment for `flow_by_layer` and `frac_v_depth` is calculated adaptively based on which is the closest producing well. But even with this option, the assignments are APPROXIMATE. Regardless of this option, the `fracturearea_v_depth_and_distance` and `proppant_v_depth_and_distance` are always assigned based on the original algorithm, based on where it propagated from.

The `flow_by_layer` file provides cumulative production and leakoff of water, oil, and gas in each layer that has been defined by the user in the 'properties versus depth' table. The volumes are given in both reservoir conditions and in surface conditions (listed as either 'res vol' or 'surf vol'). So for example, 1 bbl of oil produced at reservoir conditions will generally turn into less than 1 STB of oil at the surface, plus some free gas. The table lists the tops and bottoms of each layer. In corner point simulations, these are the average tops and bottoms, since the top and bottom of each layer is not uniform. Production and leakoff are calculated by adding up all the production and leakoff between all fracture elements and matrix elements in the layer. Because of cross-flow (in which fluid leaks off from fracture elements in some places, and is produced back in other places), if you look solely at total production or leakoff of phases, this can lead to a substantial overestimate of the actual amount of fluid produced or leaked off. Therefore, the file actually shows the 'net' production and leakoff. In the 'net' calculation, the cumulative production and leakoff of each phase are subtracted from each other. Either net production or net leakoff in each layer will always be counted as zero. So for example, if total leakoff is 500 bbl and total production is 3000 bbl, then net leakoff will be reported as 0 bbl, and net production will be reported as 2500 bbl.

The `frac_v_depth` file provides properties versus depth. Within each depth range, it reports average properties such as pressure in the fracture element, and also aggregate properties, such as the total propped fracture surface area, in that depth interval. It provides 'cumulative net' production and leakoff, defined the same as in the 'flow_by_layer' file (and similarly reports volumes at both reservoir and surface conditions).

The 'proppant_v_depth_and_distance' file outputs the distribution of proppant mass as a function of depth and distance from the well. It creates a 2D grid, with depth on the y-axis and 'distance' on the x-axis. This is essentially a 'heat map' of where the proppant went - how far and at what depths. It plots the results in terms of

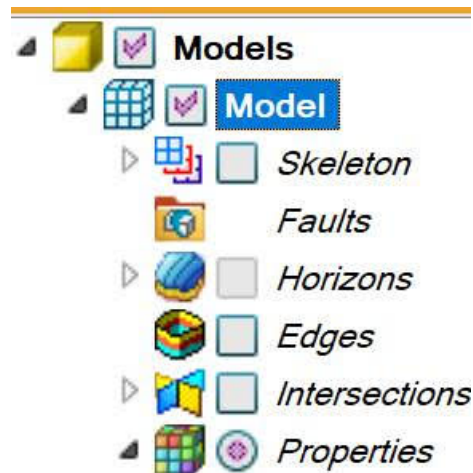
'distance from the well' - projecting onto the well to find the closest distance (not the distance along fracture azimuth). For the 'overall' version of the file, the values are summed up for all wells. The values are stacked for every fracture. So if there are 20 fractures, each with 0.5 lbs/ft², then the file would output 10 lbs/ft². Also, the values are stacked for both sides, even though the file only shows one wing. So the file sums over both sides. For example, if a fracture has 1 lbs/ft² on the left of the well and 1 lbs/ft² on the right, then the file would show 2 lbs/ft². For the 'approx well' files, these gunbarrel views are created on a well-by-well basis. For the assignments to the wells, each fracture element is assigned according to the following algorithm: when a new fracture is initiated, the new fracture element is assigned to that well. Every time a fracture element propagates, the new element is assigned to the same well as the fracture it propagated from. This assignment is heuristic, and in some cases, can lead to surprising results. These outputs should be used with caution.

The 'fracturearea_v_depth_and_distance' and 'fractureleakoff_v_depth_and_distance' files are similar, except that they instead output the distribution of fracture area and fluid leakoff, respectively. The distribution of leakoff and/or surface area could be used to estimate the distribution of microseismicity in gun barrel view down the well(s). For the 'approx well' files, these gunbarrel views are created on a well-by-well basis as described in the previous paragraph.

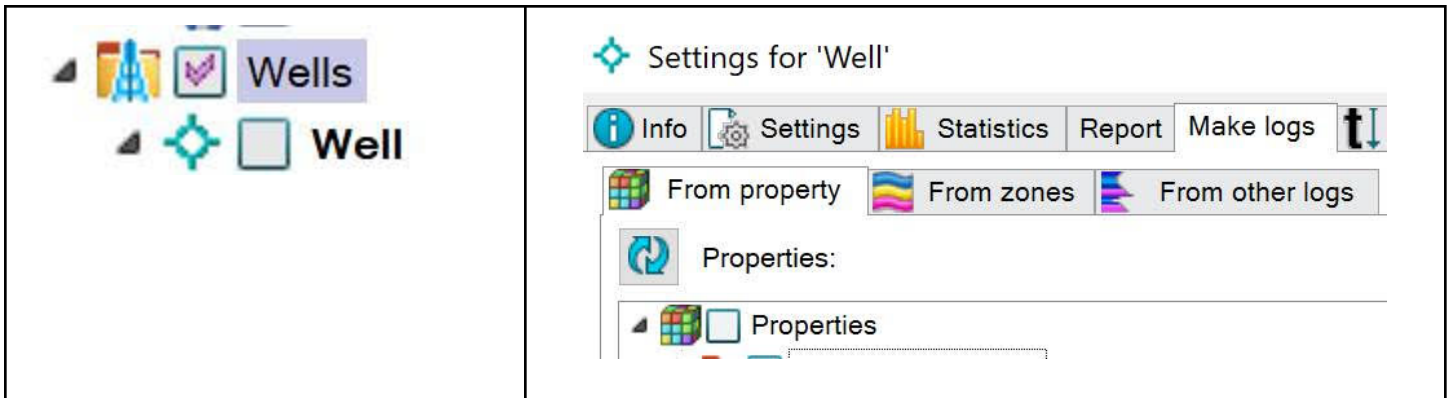
The default number of significant figures in the raw results files is 6. You can modify the number of significant figures using the setting 'rawresultsprecision'.

[Q: How do I export properties from a Petrel pseudowell to be used with the 'Properties versus depth wizard'?](#)

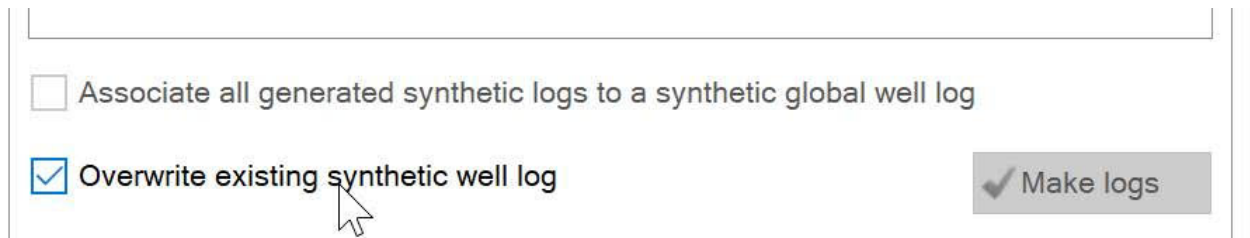
1. In Petrel, go to your model tab, and make sure the correct model that you want to extract the property from is selected by just clicking on it:



2. Then go to your wells, right click on a well > Settings > Make logs > From property:



3. A list of possible logs will appear - check the one you want.
4. There is an option at the bottom to 'override the existing synthetic well log'. I usually check that box, unless you want more than one:

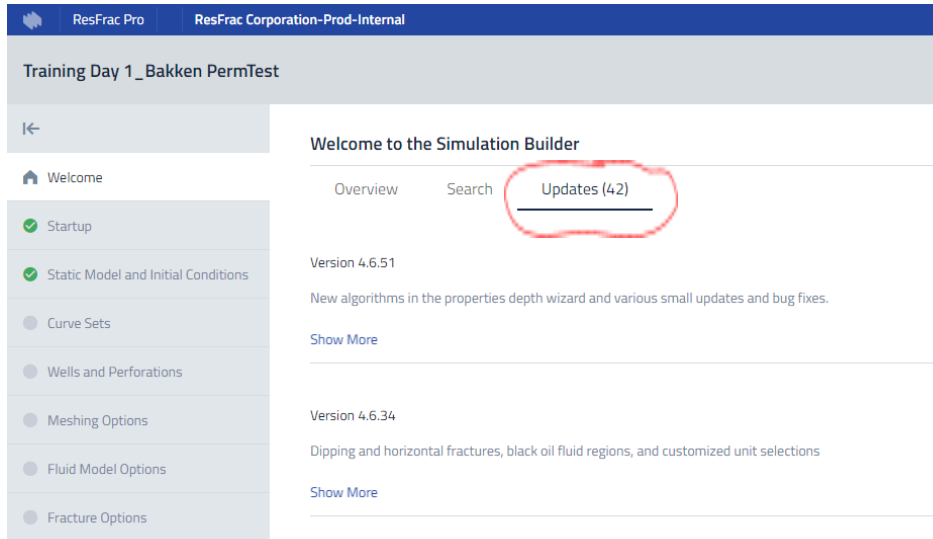


5. Then select 'Make logs'.

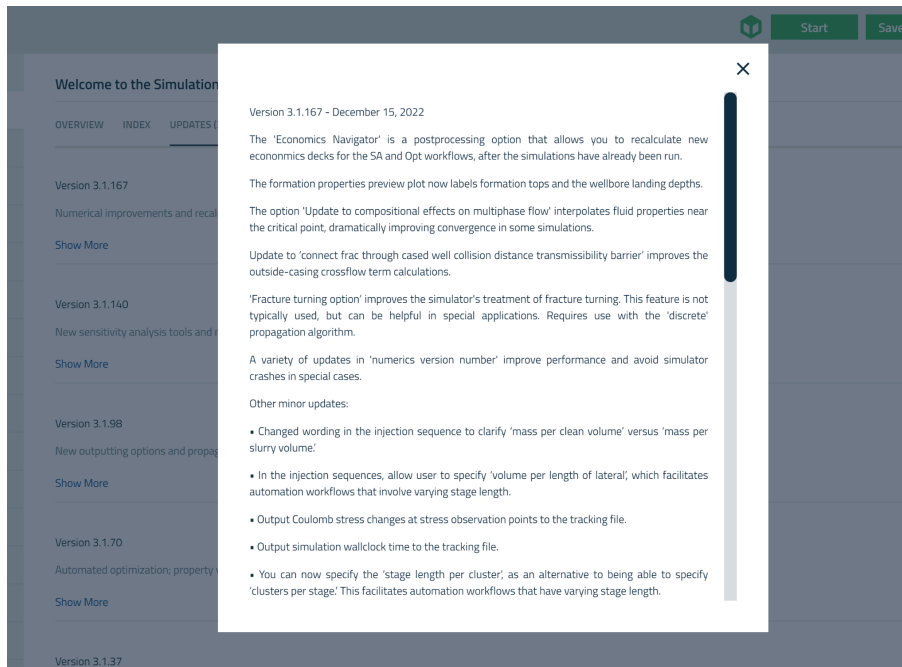
[Q: How can I refresh an old simulation with the most recent recommended settings?](#)

ResFrac regularly releases updates. These updates include new parameters as well as revised defaults to existing parameters as better calibration data become available.

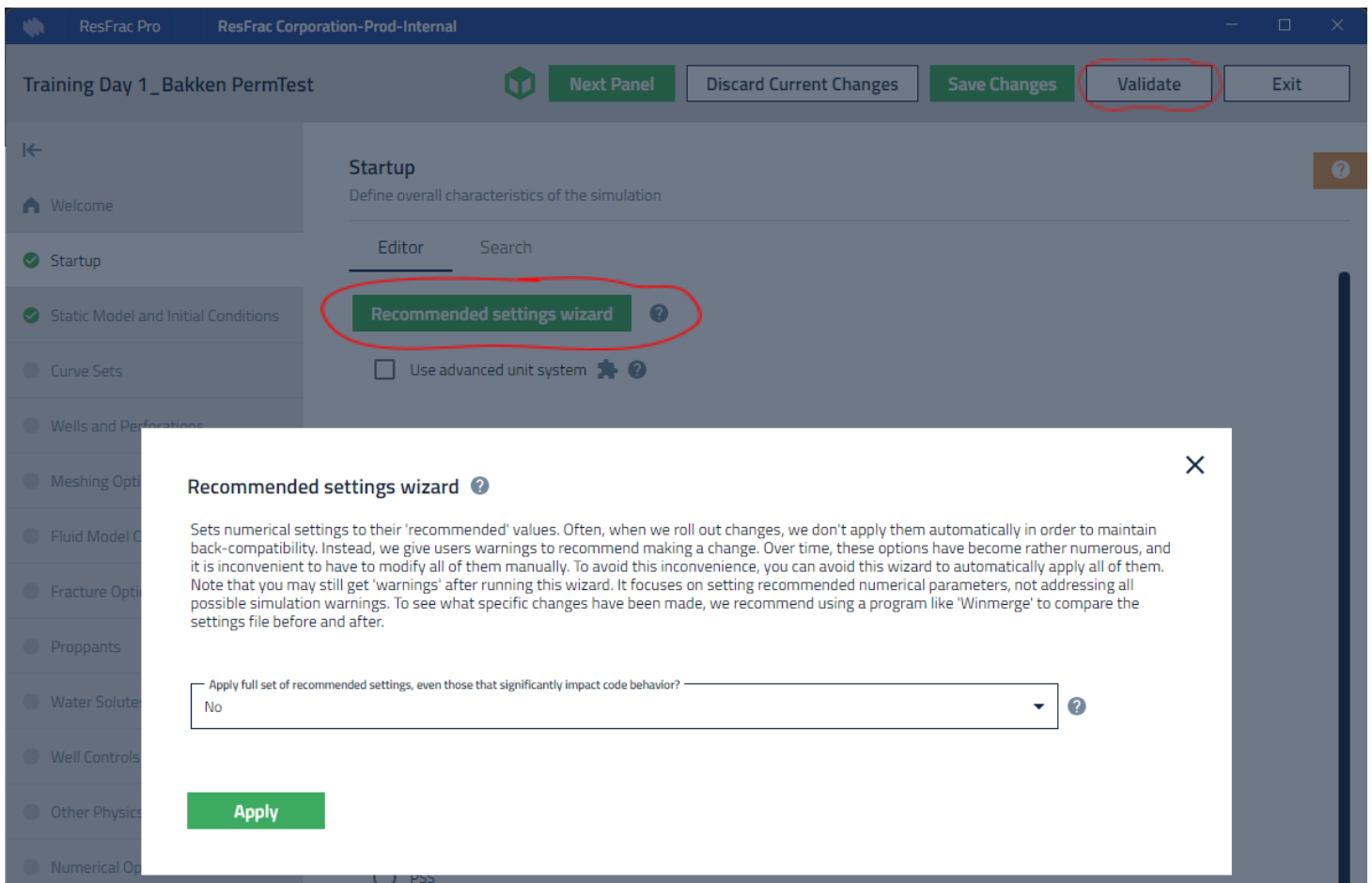
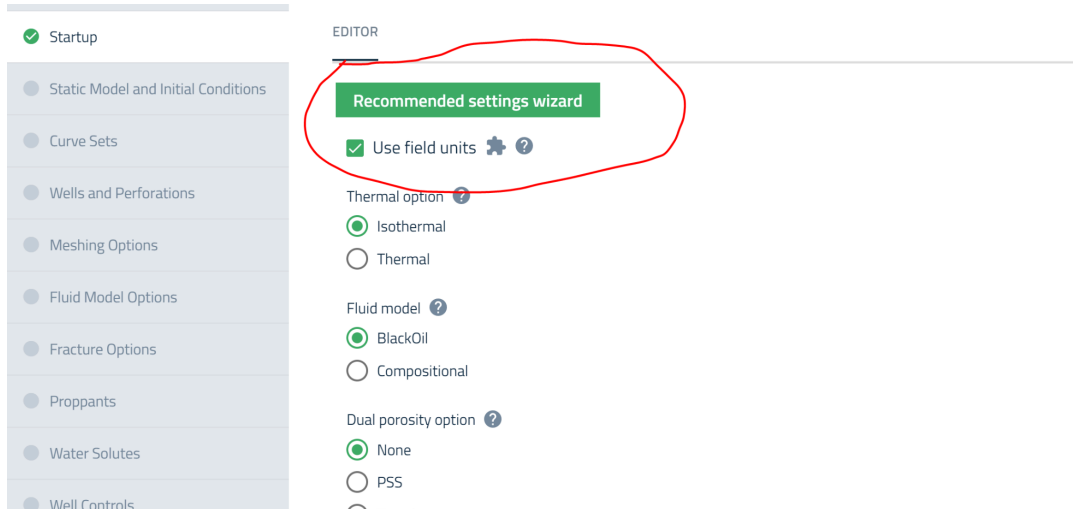
To view the recent updates and added parameters, navigate to the Updates section of the Welcome panel:



You can expand any individual release to see the details by clicking “Show More”:



An easy way to update all settings to the recommended values, is to use the “Recommended settings wizard” which will update parameter values in your model to the current recommended values.



In the Recommended Settings Wizard, you can choose to either apply all recommended settings ('Yes' in the dropdown menu), or only those that do not significantly impact code behavior ('No' in the menu). If you select

'No' (the default option), then after running the wizard we recommend clicking the "Validate" function in the upper right and reading the warnings. Any parameter values that were not automatically updated with the Wizard will show up as warnings. While generally you will want to update all your settings in order to resolve the warnings, by selecting 'No' in the Wizard you will be able to review the individual warnings to confirm you want to make each change.

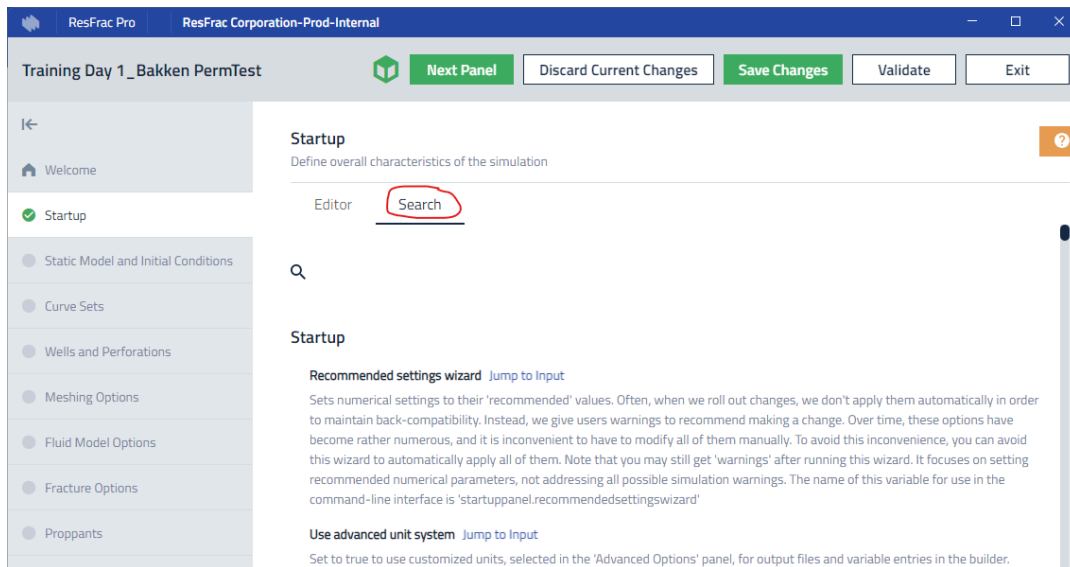
Warning(s):

The parameter 'streamline boundary condition controls' is turned on. Please keep in mind that this setting may adjust the timing of production sequence (and shut-in) control points from what you specified. This speeds up simulations (sometimes substantially), and so this is the recommended setting. But don't forget the controls may occur at somewhat different times than what you specified.

There are multiple stages in the simulation and a surface shut-in is defined with more than one stage open. Be aware - if you do a surface shut-in after the pumping of the last stage, and activate multiple stages or the full well, this opens the well and allows crossflow between the stages. This can often increase simulation runtime significantly. You might want to prevent this from happening by using an 'IsolateWellbore' shut-in, which will not allow cross flow after shut-in, and then going directly into production (preferably with a bottomhole boundary condition).

In the black oil table, the oil/gas compressibility is potentially low at the highest pressure in the table. This has the potential to cause convergence problems. Consider increasing Rs in the top row or decreasing Bo in the top row.

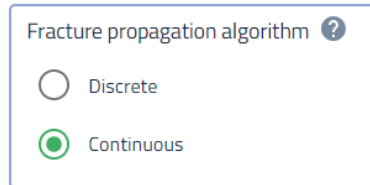
Some of these warnings are obvious while others may take a little digging into. To find any given specific parameter (always named in a validation warning), use the search function available at the top of any panel.



To see what settings changes were made, you can use WinMerge to compare the original settings file with the new settings file.

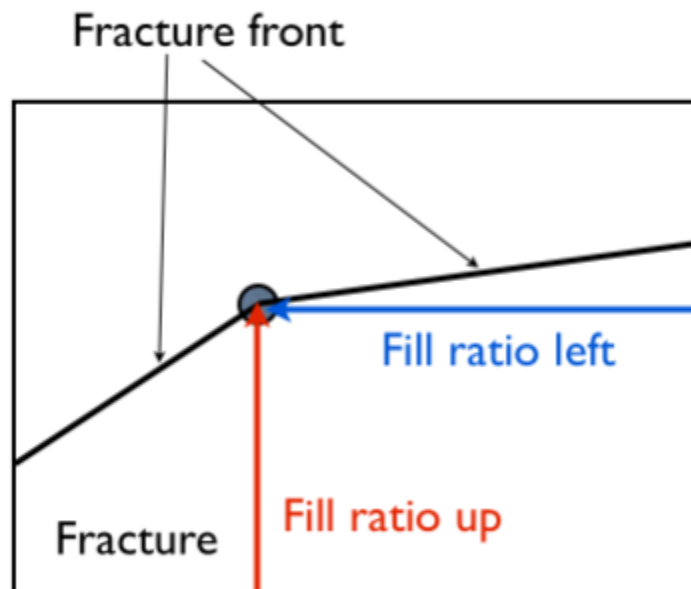
Q: What is the fracture element fill ratio?

The `frac_elms` file (in the [Raw_Res](#) folder) will report a fill ratio when using the Multi Layer Tip Elements (MulTipEI) algorithm, also known as the continuous fracture propagation algorithm (as opposed to the discrete fracture propagation algorithm). The continuous (MulTipEI) algorithm is applied by default, but this can be checked or changed in the Fracture Options panel, in the Advanced section:



A fill ratio is needed when using the continuous algorithm because the fracture propagation front is separate from the discrete fracture elements, causing the tip elements of the fracture to not be completely “filled”, or fractured. The fill ratio represents the fraction of the total fracture element area (a rectangle) that is actually fractured, as calculated by the continuous crack propagation algorithm.

The `frac_elms` file contains four directional fill ratio columns named 'fill ratio up', 'fill ratio down', 'fill ratio left', and 'fill ratio right'. These parameters define the position of the fracture front in the tip elements. Typically, only two out of the four directional values are active (the other two are NaN), as shown in the picture below. In this case, the 'fill ratio left' determines distance to the front point from the right edge of the element, while the 'fill ratio up' determines the distance from the lower edge of the element. These parameters vary from 0 to 1, to represent the fraction of the element length or height.



Another column, 'effective fill ratio', is a function of the fill ratios in the active directions. The effective fill ratio provides a viable approximation for the geometric area fraction. Consequently, the 'true area' of a fracture element is the product of the effective fill ratio and the total area of a rectangular fracture element.

Q: How do I determine the true area of a fracture element using the fill ratio?

If you want to get an area fill fraction, then use 'effective fill ratio' reported in the frac_elms.csv file in Raw_Res. It is not precisely a geometric area fraction, but a very good proxy.

Q: How do I calculate the mass of proppant in a tip fracture element?

To determine proppant mass in an element, you need to take proppant concentration (mass per unit area), multiply by the rectangular fracture element area, and then multiply it by the [effective fill ratio](#). Essentially, you need to multiply the proppant concentration by the true fracture element area, which we also output. We do not submesh proppant in these tip elements, so whatever concentration is there, we assume that it is uniformly distributed within the open part of the element.

Q: Do the median and max fracture lengths and heights reported in the sim track file account for the fill ratio?

Yes, the tracking file outputs fracture heights and lengths, accounting for the fill ratio. Similarly, the output 'max distance from well' also accounts for the fill ratio (which is slightly different from the 'length averaged' height that is outputted). Additionally, in the Output Options panel, selecting the 'Extra fracture info in tracking file' option outputs individual crack statistics into the sim_track file as well, not just the overall 'summary statistics'.

Q: If a fracture element spans multiple static model layers, what properties from the Geological Units (Facies List) table apply to that fracture element?

Some of the parameters listed in the Geological Units table are fracture properties. Therefore, they apply to the fracture elements as opposed to (or in addition to) matrix elements. And since the fracture mesh is non-conforming, a fracture element could span across multiple layers in the static model. This is handled by the code in different ways, depending on context. For flow, this is handled directly by calculating fluid flow between each pair of fracture and matrix elements. For stress, the code takes the arithmetically thickness weighted average of the stress of each layer. For toughness, the code takes the highest toughness value encountered in any layer. For elastic moduli, the 'self-interaction coefficient' is calculated with a thickness weighted geometric average. Other parameters, such as E0max, Eresmax, 90% closure stress, maximum flowing molar mass, and proppant embedment are assigned properties from the layer in which the fracture element center is located.

Q: What parameters are ok to change in a simulation continuing from a restart file?

Typically, you only want to make changes in a restart file that will not impact the simulation up to the point at which the restart is taken. For example, it is unlikely that you would want to download a restart file in between fracturing two wells, and then change the toughness. Toughness usually has a dominant impact on fracture geometry, so fracturing two wells or two stages with different toughness values would likely impact ultimate predictivity of that model.

Below are some common parameters changed in restarts.

1. Relative permeability
Relative permeability curves are one of the primary parameters used to match production data, and they have negligible impact on fracture propagation. It is very common to download a restart prior to production and change rel perm curves.
2. k_0 (in the Proppants table)
 k_0 is a linear multiplier to the conductivity of the proppant pack. Changing this parameter will not have an impact on fracture propagation or proppant transport, but will impact production and especially things like GOR.
3. Global permeability multiplier
Global permeability multiplier is another dominant parameter for production and commonly used to history-match production data. Modifying global permeability *will* impact leakoff and can subtly change fracture propagation, but is generally not the primary parameter governing fracture geometry. It is ok to change in a restart (especially if changing by less than an order of magnitude), though be aware that you should rerun the simulation from the start to confirm geometries don't change more than desired. If you want to change permeability during production, but keep 'effective permeability to leakoff' during injection constant, you can modify the permeability multiplier in the PDP downward at the same time that you increase the global permeability multiplier.
4. Permeability by zone
Similar to the global permeability multiplier, you may change the permeability of various layers of your static model in order to match production by zone.
5. Pressure Dependent Permeability
Pressure Dependent Permeability (PDP) is used in several ways in ResFrac simulations. For delta pressure above zero (fracture pressure > matrix pressure), it is common to use multipliers greater than one to accelerate leakoff and account for the presence of multiple fracture strands. If using PDP in this sense (for example to match initial water flowback), the simulation should be run from the start. PDP is also used to model stress dependent permeability reduction (fracture pressure < matrix pressure), where multipliers less than one are used. Often this phenomenon is observed in over-pressured gas shales. For these simulations, it *would* be appropriate to modify the negative delta pressure portion of your PDP table in a restart file.
6. Well controls

If you want to change a well control, restarts are a great tool. Just make sure to download a restart file prior to the time at which you are making a change.

7. Drilling new wells

If you have multiple generations of wells in your model, you can download a restart prior to the completion of a second or later generation well, and then change the location/completion design of that well in the restart simulation. Be sure to use the ‘Wellbore Drilling Times’ option in the advanced section of Wells and Perforations to do so:

WELLBORE DRILLING TIMES [HOURS] ?

	Well name ?	Wellbore drilling times [hours] ?
1	Well_P1	0
2	Well_P2	0
3	Well_C1	0
4	Well_C2	0
5	Well_C3	0

8. Numerics version number

If you update the numerics version number, the restart file will run using the later version, while the results from the part of the simulation run prior to the restart file will of course still have results based on the previous version. This may be convenient if a new nvn was provided to resolve a specific bug and running from the start is impractical.

Parameters you *should not* change in a restart (and yes, sometimes there are exceptions):

1. Meshing

Pretty much everything on the Meshing tab is a no-go. Mesh size, mesh center, meshing scheme, etc. cannot be changed in a restart.

2. Perforation locations or well locations (unless using the wellbore drilling times option above)

3. NW complexity coefficient (in the Well Vertices table)

By default this coefficient is not changeable since it is possible for erosion to change it over time. However, if you would like to reset the NW complexity coefficient from a restart, you may select the parameter ‘Near-wellbore complexity coefficients reset on restart’ under Advanced options in the ‘Wells and Perforations’ panel.

4. Anything that defines the “initial state” of the model

a. Saturations

b. Random toughness

c. Orientation of SHmax

d. Fluid model options

i. You occasionally may want to change fluid properties. If you do so, you will need to start the simulation from the beginning.

5. Everything on the 'Startup' panel (the fluid model, thermal/isothermal, etc.)
6. Physics and correlation selections on the 'Other Physics Options' and 'Numerical Options' panels

Q: Why use restart files?

Simulation compute times are often on the order of 10s of hours. And while we always suggest optimizing your model for speed, you can also often save time by using restart files during history matching.

Restart files save the exact state of a simulation. For example, if you download a restart file from timestep 12350, that file contains the exact state of the simulation at timestep 12350. If you restart the simulation with the exact set of parameters you originally ran, the result will be exactly the same. However, we often want to *change* parameters in a restart.

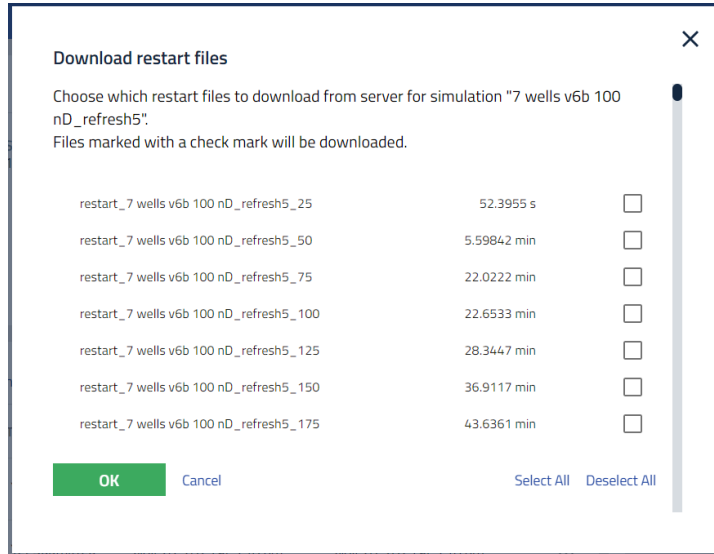
It is common to perform a restart from after fracturing the well/s in the model, but prior to production. Since fracturing usually takes most of the compute time, using restart files can save you a lot of time while you iterate on production parameters. ResFrac is a fully coupled code, and so many parameters impact both *fracturing* and *production*; however, many other parameters affect only the production response (e.g. relative permeability) and have little impact on fracturing. These are great parameters to modify in a restart simulation.

Q: How do I download restart files and run a simulation from a restart?

Restart files can be downloaded from the menu visible when clicking on a simulation in the job manager:

The screenshot shows the 'Simulations (10)' section of the job manager. It includes a search icon, a 'Show Active Simulations' link, and buttons for 'Create Sim', 'Import Sim', 'Batch Run Simulations', and 'Multiplot'. Below is a table with columns for 'SIMULATION NAME', 'STATUS', and 'DATE CREATED'. A context menu is open over the table, listing various actions such as 'Simulation Details', 'Edit Simulation in Builder', 'Run Simulation', 'Copy Simulation', 'Export Simulation Folder', and 'Download Restart Files' (which is highlighted).

>	SIMULATION NAME	STATUS	DATE CREATED ↓
>	7 wells v6b 100 nD_refresh7	Not Yet Submitted	Nov 26, 2025 at 3:20 pm
>	7 wells v6b 100 nD_refresh6	Not Yet Submitted	Nov 26, 2025 at 3:16 pm
>	7 wells v6b 100 nD_refresh5	Finished running	Nov 24, 2025 at 5:09 pm
>	7 wells v6b 100 nD_refresh4	Finished running	Nov 24, 2025 at 4:16 pm
>	7 wells v6b 100 nD_refresh3	Finished running	Nov 24, 2025 at 3:58 pm
>	7 wells v6b 100 nD_refresh2	Finished running	Nov 24, 2025 at 3:45 pm

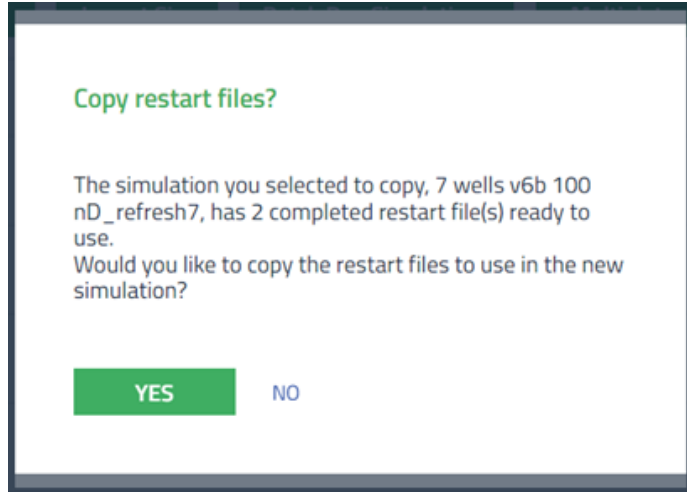


Clicking 'Download Restart Files' will bring up a selection menu where you can select one or more restart files to download. How do you know which restart file is the one you want? The numbers at the end of the restart file names are the timestep numbers, and next those are the corresponding simulation times. You can use the visualization tool to identify the timestep that you would like to use as a restart point (which may not line up exactly with the restart file times), then select the last restart file *before* the timestep that you noted in the comments file.

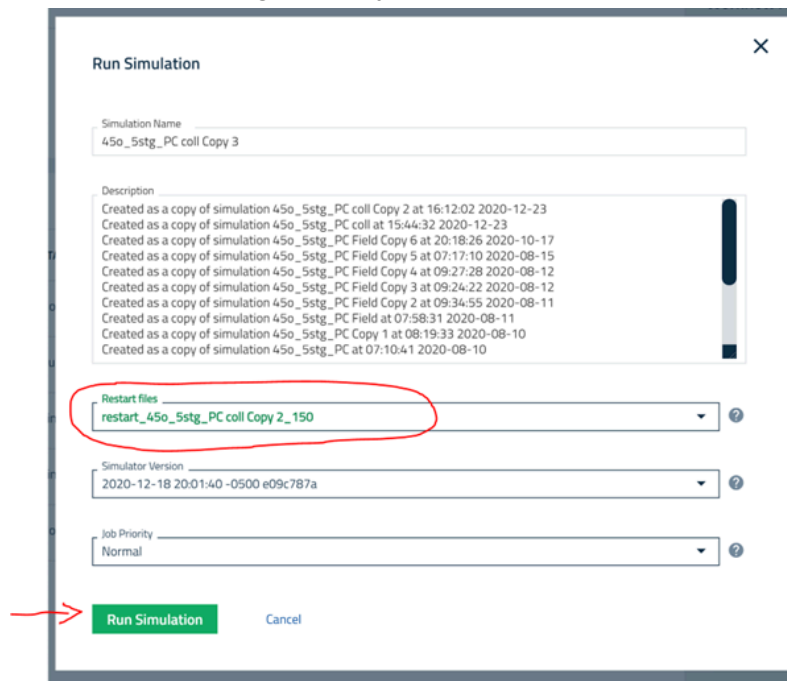
In the Output Options panel, you can select the frequency at which restart files are created. The parameter 'Restart timestep interval' indicates how many of the timesteps from the sim_track file generate a restart file. Typically, this should occur every 25 to 300 timesteps, depending on how long your simulation is. You can also force the output of additional restart files at specified timesteps (with the 'force restart timestep' table) or simulation times ('Restart Times' table). In addition, there are other (infrequently used) customizations to restart files available in the simulator, such as what data to include (or exclude) in each restart file.

Once your selected restart file(s) are downloaded, the easiest way to restart a simulation from that restart file is to:

- Make a copy of the simulation
 - Be sure to copy over the downloaded restart files (by selecting 'Yes');

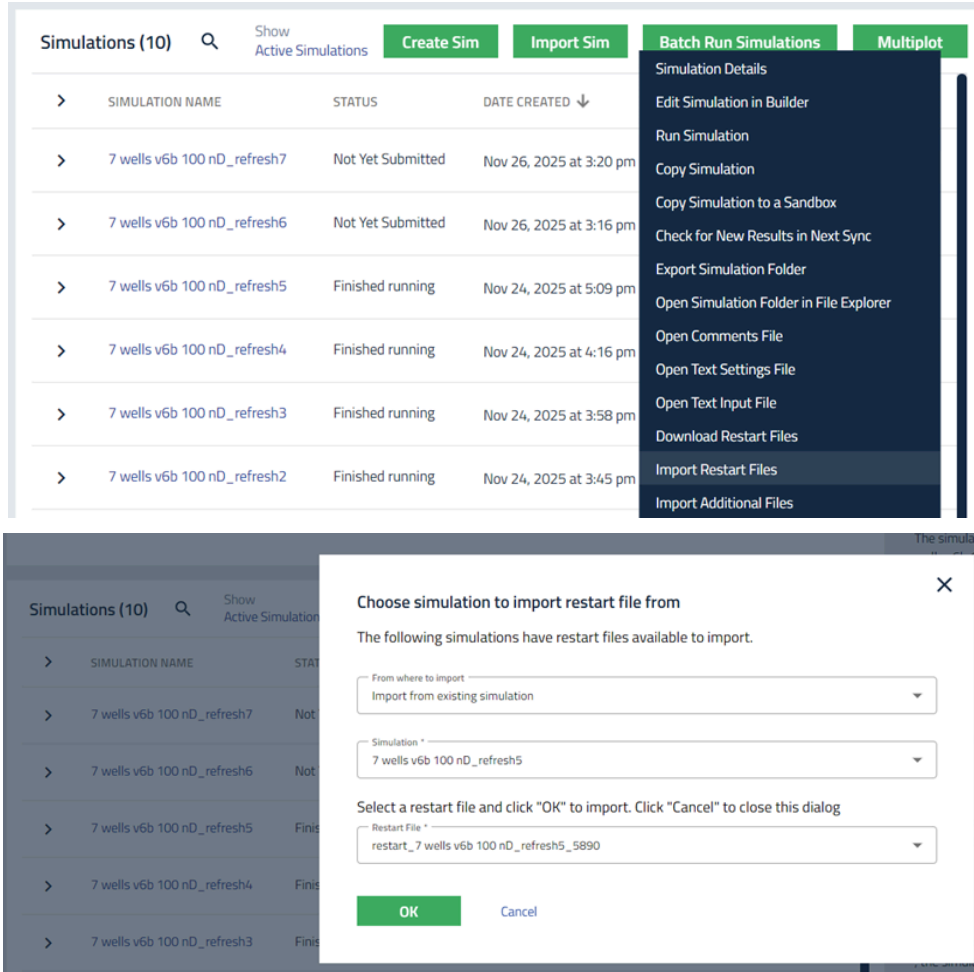


- Change the parameters of interest
- Select the restart file when submitting the copy:



Note that when you run a simulation from a restart, the UI needs to upload the restart file back to the server (along with the simulation input/settings files), so it may take several minutes to fully upload from the time you click 'Run Simulation.'

What if you made a copy of the simulation before downloading the restart file? Or selected 'No' to copying the restart files to the child sim? Click on the name of the simulation in the job manager, and select Import Restart Files from the menu. Then select "Import from existing simulation." Select the simulation that you want to import the restart file from, and then select the restart file.



If an external restart file is shared with you, do not unzip that file prior to importing it. ResFrac is expecting a .bin.zip file.

Name	Date modified	Type	Size
restart_7 wells v6b 100 nD_refresh5_1525.bin	11/26/2025 3:16 PM	Compressed (zipp...	19,029 KB
restart_7 wells v6b 100 nD_refresh5_5890.bin	11/26/2025 3:16 PM	Compressed (zipp...	58,047 KB

Q: Why is my treating pressure (WHP) very high at the beginning of the simulation?

The wellbore is initialized with water that doesn't have any friction reducer. Consider using the 'Initial Wellbore Fluid Mixtures' table to initialize the wellbore with the frac fluid that contains a friction reducer. The table can be found in the advanced section of the 'Static Model and Initial Conditions' tab.

	Well name	Initial wellbore fluid mixtures
1	Well_TF	HVFR 20cP
2	Well_MB	Default

Advanced options

Q: Why does my well keep reaching the maximum WHP and not creating any fractures?

Hitting the maximum WHP could be caused by several different factors relating to model construction, wellbore friction, or issues with the static model.

To start, some quick troubleshooting steps include:

1. Ensure that the perforations are set up correctly. If there are not any active perforations in the matrix region, then a cased/cemented wellbore will be unable to initiate fractures or flow into the matrix.
2. Set the wellbore absolute roughness to the default (because extremely high values could cause excessive wellbore friction).
3. Initialize the wellbore with a wellbore fluid mixture that contains friction reducer using the 'initial wellbore fluid mixtures' table in the Water Solutes panel. Alternatively, consider lowering the global 'wellbore friction adjustment factor'.
4. Confirm Shmin was entered correctly, and the 'Stress initial condition input method' corresponds to the value entered.
5. Confirm that toughness is not set to excessively high value.

If the maximum WHP is hit after the proppant ramp has begun, this is likely an indication of a screenout in which proppant cannot be transported into fractures. Consider reducing stress in the target zone or reducing Young's modulus; eliminating high contrasts in stress and elastic moduli can help as well.

For more general guidance on calibrating treating pressures within ResFrac, please refer to <https://www.resfrac.com/blog/calibration-wellbore-pressure-during-and-after-fracturing>.

Q: How is embedment handled in ResFrac?

Proppant embedment is the amount of proppant that does not contribute to conductivity enhancement due to the embedment of proppant particles into the fracture surface. The units used in ResFrac are mass per area, however, this can be converted to units of length (which is how embedment is often reported) by dividing by proppant density. The proppant embedment is specified on the far right side of the static model (Geological Units) table for each layer. If this parameter is left blank for a given layer, the embedment for that layer defaults to be equal to the global parameter 'proppant embedment mass per area' (which defaults to 0).

	Poisson's ratio ?	Vert over horiz modulus ?	Proppant embedment [lbs/ft ²] ?	Show in viz tool ?	Effective x-permeability [md] ?
1	0.28	1	0.02	<input type="checkbox"/>	0
2	0.28	1	0.02	<input type="checkbox"/>	0.000005
3	0.32	1	0.02	<input type="checkbox"/>	0.000015
4	0.27	1	0.02	<input type="checkbox"/>	8e-8
5	0.24	1	0.02	<input checked="" type="checkbox"/>	0.00045

Embedment typically increases with greater effective normal stress. The degree to which this occurs can be specified in the advanced section of the 'Proppants' panel by entering an embedment reference stress which controls the amount of embedment using the following equation: [specified value of embedment]*[effective normal stress/reference stress]^0.5. If the reference stress is left blank, then the amount of proppant embedment remains constant, regardless of effective normal stress. If you aren't sure what to use, 5000 psi is usually a good value.

Proppant embedment is by default irreversible. This means that the proppant embedment is a function of the maximum achieved effective normal stress in each element, as opposed to the current stress for a given timestep. The 'Irreversible embedment' option can be unselected in the Advanced options of the 'Proppants' panel.

The screenshot shows the 'Proppants' configuration panel in the ResFrac Pro software. The panel is titled 'Proppants' and includes a sub-header 'Specify parameters related to proppants'. On the left is a navigation menu with options like 'Welcome', 'Startup', 'Static Model and Initial Conditions', 'Curve Sets', 'Wells and Perforations', 'Meshing Options', 'Fluid Model Options', 'Fracture Options', 'Proppants' (selected), 'Water Solutes', and 'Well Controls'. The main configuration area has an 'Editor' tab and a search bar. Below this is a table for 'PERFORATION EROSION BETA PER PROPPANT [FT-MIN/LBS]'. The table has two columns: 'Proppant' and 'Perforation erosion beta per proppant [ft-min/lbs]'. The first row shows '40/70 White' in the 'Proppant' column. Below the table are several input fields: 'Critical Shields number in Wiberg correlation' (0.047), 'Irreversible embedment' (checked), and 'Proppant embedment reference stress [psi]' (10000). The 'Irreversible embedment' checkbox and the 'Proppant embedment reference stress' field are circled in red.

Q: Does ResFrac handle proppant crushing?

ResFrac does not handle proppant crushing explicitly. Proppant conductivity is defined with parameters ‘f’ and ‘k0’ used in the modified Kozeny-Carmen equation to calculate initial permeability (permeability at effective normal stress of zero). A ‘permeability gamma,’ c_b,k is used to control the stress sensitivity of the proppant pack.

To account for non-reversible physical processes that could irreversibly reduce proppant pack conductivity (such as proppant crushing or proppant pack diagenesis), there are several available mechanisms. First, all proppant conductivity loss can be made to be irreversible through the ‘Irreversible proppant bed conductivity loss’ parameter. Alternatively, only a fraction of the conductivity loss versus effective normal stress can be made to be irreversible through the ‘Irreversible proppant pack conductivity table versus effective normal stress’ parameter. Lastly, a time component to conductivity loss can be included with the parameter table ‘Time-dependent proppant conductivity loss’.

Irreversible proppant bed conductivity loss ?

- Reversible Proppant Bed Conductivity Loss
- Irreversible Proppant Bed Conductivity Loss

IRREVERSIBLE PROPPANT PACK CONDUCTIVITY TABLE VERSUS EFFECTIVE NORMAL STRESS ?

	Effective normal stress [psi] ?	Multiplier ?
1		

TIME-DEPENDENT PROPPANT CONDUCTIVITY LOSS ?

	Proppant Name ?	Rate constant [days^-1] ?	Damage type ?	Rate scaling with effective normal stress (optional) [psi] ?	Minimum conductivity multiplier (optional)
1					

Q: Where should I center my zero perm cube when using the Zero Perm Inside Cube Option when I want to simulate well spacing effects from wells that are not in the model, and can it extend beyond the edge of the matrix?

Yes, a zero perm cube can extend beyond the edge of the matrix. When creating a zero perm inside cube (using the table “List of ‘Zero Perm Inside Cube’ Constraints”), normally the best place to center the cube is at the edge of the matrix. This makes it simpler to accurately define the length of the cube.

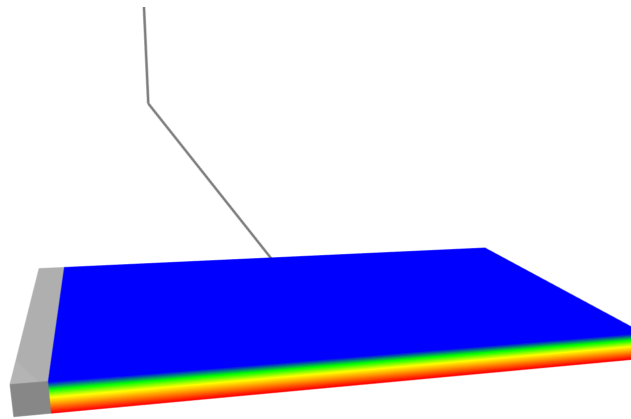
Make the length of the ‘zero perm cube’ in the SHmax direction long enough to reach ½ well spacing between the simulated well and the non simulated well. Then align the zero perm cube using the azimuth of the well. The length in the Shmin direction can be made considerably longer than the matrix since the cube can extend beyond the matrix. The length of the zero perm cube in the vertical direction can be made very long as well if using just one cube, or else it can reflect the vertical spacing of the wells if using more than one cube.

To verify the construction of the zero perm cube setup, click the green cube at the top right corner of the builder interface (next to the 'Next Panel' button) to bring up the 3D preview. The portion of the matrix that is within the zero perm cube will be grayed out.

LIST OF 'ZERO PERM INSIDE CUBE' CONSTRAINTS [FT] ? 📄 ⚠️

	x-center [ft] ?	y-center [ft] ?	z-center [ft] ?	x-direction length [ft] ?	y-direction length [ft] ?	z-direction length [ft] ?	Rotation angle [degrees] ?
1	0	-1000	10020	2000	200	200	0



New Row Resize Table






Q: Is it necessary to refine the mesh near fractures in a ResFrac simulation?

No, it is not necessary to refine the mesh adjacent to the fractures in most situations as ResFrac includes a number of features to account for this behavior. First, the 1D submesh method is effectively a mesh refinement that occurs within each timestep. For more details on how the 1D submesh method works, see Section 2.3 in the ResFrac Technical Writeup (<https://arxiv.org/abs/1804.02092>), or the original description of the method here: <https://arxiv.org/abs/1709.02493>.

Additionally, with the coarse mesh, ResFrac tracks the amount of water that has leaked off in the 'near fracture' region. This is the 'water bank' region, and it affects water flowback. The thickness can be plotted in the 3D images and is controlled by these parameters:

Water bank 'rel perm increase' scaling thickness [m]  

Water bank 'rel perm decrease' scaling thickness [m] 

Water bank immobile fraction  

With these parameters, we are able to reproduce the early-time flowback response - water cut, productivity, flowback percentage, etc. Details on these parameters are provided in the built-in help content (the ? buttons).

With this empirical/heuristic treatment, it is generally unnecessary to refine the mesh. However, under specialized circumstances, such as DFIT or huff-and-puff simulations, you may still wish to perform [localized refinement of the matrix mesh](#).

[Q: How do I use local mesh refinement in ResFrac?](#)

[Mesh refinement is often not necessary](#) because of the '1D submesh' algorithm that internally remeshes matrix elements containing fracture elements. However, the 1D submesh calculation simplifies multiphase flow and water solute transport. Consequently, for some applications you may instead want to have additional local refinement of the matrix mesh around the perforation clusters. If incorporating matrix mesh refinement, you also need to turn off the 1D submesh by deselecting the parameter '1D submesh calculation for fracture-matrix connections', found in the Fracture Options panel under Advanced options.

To have a refined mesh against each of the fractures, use the 'Mesh refinement in the Shmin direction wizard' in the advanced section of the meshing options. This wizard automatically creates a refined mesh up against each perforation cluster. Note, if using this wizard, you should set fracture initiation randomness distance to 0, to ensure the mesh refinement is centered at the fractures. An alternative method to apply a similar refinement is by setting the two 'Automatic mesh refinement perpendicular to fractures' parameters (both minimum size and growth factor). This alternative is recommended for use in Sensitivity Analysis workflows as opposed to individual simulations. Using either method, you must specify two parameters - the size of the central matrix elements (those containing the perf clusters), and the rate at which elements increase in size with distance from these center elements. Within the wizard you also have the option to select a maximum element size as well.

To refine the mesh in the Shmin direction at only a single point or perf cluster (instead of all of them), you can use the 'Apply logarithmic matrix mesh refinement in the Shmin [or x] direction' tool. This takes the same three inputs as the 'Mesh refinement in the Shmin direction wizard', but also requires the x-coordinate around which

to center the mesh refinement. Optionally, you may also re-center the matrix and change the length of the matrix in the Shmin direction.

When either of these two wizards are applied, the parameter 'Use uniform matrix elements in the Shmin direction?' is automatically deselected, and a new table called 'Matrix block length (either Shmin or x)' will appear. This table will be automatically populated with the location and size of the matrix elements in the Shmin (or x) direction, according to the size and growth rates specified in the wizard. This table can also be customized if desired. If instead of the wizards you used the 'Automatic mesh refinement perpendicular to fractures' parameters, the refinement will only occur upon running, and will not appear in the 3D preview plot. Consequently, this method of mesh refinement is recommended for use in sensitivity analysis workflows only, when the meshing wizard will not work. The refinement will override the mesh shown in the 3D preview plot, even if 'Use uniform matrix elements in the Shmin direction?' is selected.

If you use a refined mesh in the Shmin direction, it is recommended that you also set 'scale matrix fracture flow with conductivity multiplier' to 10,000 in the Fracture Options panel. This setting avoids the risk of numerical error caused by a nonconforming mesh, with minimal effect on the simulation results.

The mesh can also be refined in the SHmax direction. This feature might be recommended under certain conditions, such as in a vertical well with small or no fractures and a high permeability formation where flow is expected to be "radial". Like in the Shmin direction, there are two tools available to do this. The 'Mesh refinement in the SHmax direction wizard' automatically refines the mesh towards each of the wells. Alternatively, if there is only one well, or you wish to manually select the center of the SHmax direction refinement, you can use the 'Apply logarithmic matrix mesh refinement in the SHmax [or y] direction' tool. These wizards take the same inputs as the Shmin refinement tools, but apply them in a different direction.

The screenshot displays a software interface with two main panels. The left panel, titled 'Advanced options', contains three green buttons: 'Mesh refinement in the Shmin direction wizard', 'Mesh refinement in the SHmax direction wizard', and 'Automatic mesh refinement perpendicular to fractures - minimum size [1]'. Below these are two input fields: one for 'Automatic mesh refinement perpendicular to fractures - growth factor' with the value '2', and another for 'Automatic mesh refinement perpendicular to fractures - minimum size [1]'. The right panel, titled 'Mesh refinement in the Shmin direction wizard', includes a descriptive paragraph, three input fields: 'Center element size [ft]' with value '1', 'Element growth rate' with value '2', and 'Optional maximum element length in the Shmin direction [ft]'. An 'Apply' button is located at the bottom of this panel.

MATRIX REGION LENGTH ALONG THE SHMIN DIRECTION

Apply logarithmic matrix mesh refinement in the Shmin direction ?

Use uniform matrix elements in the Shmin direction? ?

Number of matrix elements
115 ? ⚠

Matrix region length in the Shmin direction [ft]
1152.65013532 ?

Apply logarithmic matrix mesh refinement in the Shmin direction ?

Center of matrix region in the x direction (absolute coordinates) [ft]
0 ?

Refinement center position in the x direction (absolute coordinates) [ft]
0 ?

Lc, Central element length in the Shmin direction [ft]
1 ?

Total matrix length in the Shmin direction [ft]
1152.65013532 ?

Geometric growth factor in the Shmin direction
1.2 ?

Optional: maximum element length in the Shmin direction [ft]
? ?

Apply

Q: The pressure trend in my simulation ends before the ISIP is shown, how do I see it?

When the 'isolate wellbore' boundary condition is applied in the Well Controls during shut-in, the wellbore is entirely removed from the model. Consequently, the model can no longer simulate WHP/BHP and thus doesn't output pressure data with which to get an ISIP. To access more pressure data after shut-in, include a 'surface shut-in' for 10 minutes prior to the 'isolate well' shut-in condition. If a surface shut-in was already applied, extend the duration.

Q: How do I find pressure responses (SWPM or OPM) in the simulation so I can history match to field data?

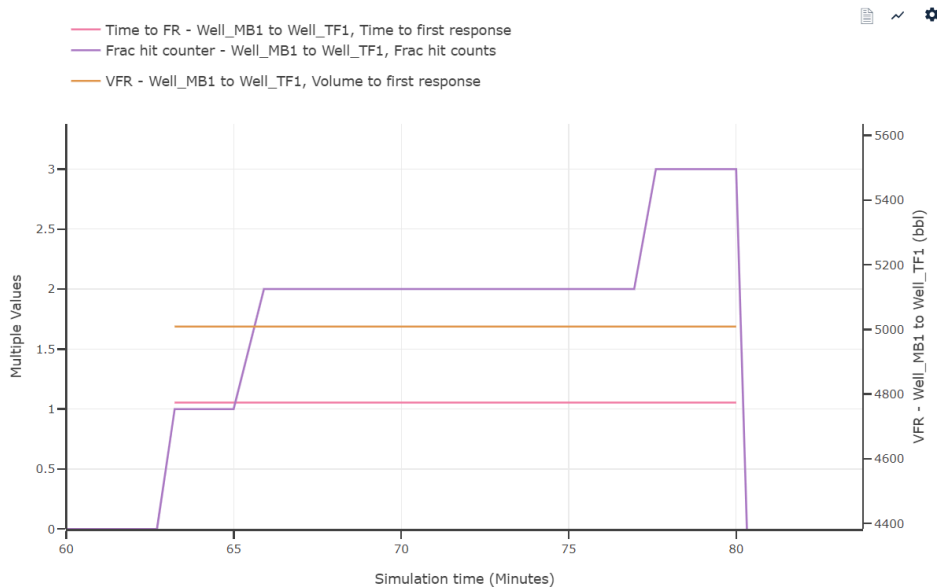
In order to track observation pressure monitoring (OPM) within the simulation, you will need to apply a surface shut-in condition in the Well Controls panel. This will permit the simulation to output a pressure response from a pressure interference test or frac hit because the wellbore is open to the formation. Conversely, the isolate wellbore shut-in from an operations standpoint is akin to having a plug in the well isolating all your perforations, like during frac operations. When an isolate wellbore condition is applied to ResFrac, it removes the wellbore from the model, and consequently does not output BHP or WHP values for pressure monitoring.

For sealed wellbore pressure monitoring (SWPM) you will not see a pressure response in a simulation since the well is not open to the formation (it is sealed). In the field, the SWPM response comes from the deflection of the casing as the fracture passes, and ResFrac does not model that process. Instead, we output the volume to first response (VFR), or time to first response (TFR), by tracking when a given fracture from an injection well physically crosses an observation well.

To monitor the simulation for VFR and Frac hits, you'll need to explicitly add them as Output Options in these two tables: 'Volume to First Response Trackers' and 'Frac Hit Trackers'. Within those tables you can indicate which well pair(s) to monitor for frac hits.

The screenshot shows the 'Output Options' configuration window in ResFrac Pro. It features a sidebar on the left with a navigation menu. The main area is titled 'Output Options' and includes a search bar and a table for 'VOLUME TO FIRST RESPONSE TRACKERS'. This table has two columns: 'Injection well' and 'Observation well', with 'Well_MB1' and 'Well_TF1' respectively. Below it is a 'FRAC HIT TRACKERS' table with the same structure. A red bracket highlights both tables. At the bottom, there is a checkbox for 'Frac hit trackers include shut-in period'.

These output values can then be plotted in the viz tool:



Q: How big should I make an external fracture?

We recommend making an external fracture the size of the region where stress shadow from a previous (unsimulated) stage will create compression on the current stage. So, roughly outlining the length and height of the anticipated drained region (similar to propped geometry). This feature is located in the ‘Wells and Perforations’ panel.

To define an external fracture, its location must be defined, either by using xyz coordinates or specifying a distance from the edge of the matrix. The size (which dictates the magnitude of the stress shadow) is defined either by length + height + net pressure in the fracture, or by volume + net pressure in the fracture. Defining an external fracture can be an iterative process, whereby you run a single simulation first to determine what a reasonable external fracture size is based on properties of the fractures that formed during stimulation. The final (optional) component when defining external fracture properties is the timing. The external fractures may be present from the start of the simulation, or may be initiated at an “activation time”. Additionally, the external fractures may be given a closure time, and potentially a residual fraction (indicating a residual stress shadow effect on the subsequent stages). The option to re-activate the external fractures up to three times is also available.

The following is a general typical procedure for how to setup and calibrate external fractures:

1. Set up one well with 3 to 5 stages.
2. Create an observation plane (Observation Planes Presentation) in the output options at the depth of the well.
3. Place the fracture ½ of the cluster spacing away from the matrix.
4. Observe the stress at the end of the fracturing and use this for a starting net pressure.
5. Observe the ‘average’ width and heights of the fracture.

6. Observe how long it takes for the stress plane to stabilize after frac'ing and use this for the closure time.
7. Observe the residual stress after fracture closing and calculate the residual fraction i.e., Residual stress ÷ Starting stress
8. Use the frac job report to determine when the last stage was frac'd and use this to activate the external fracture appropriately.
9. Use the stress plane in the UI to ensure that the external fracture achieved the appropriate stress when the stage begins to frac.
10. Use ISIP data to help confirm that the pressure in the external fracture is correct.
11. ISIP = Net Pressure + Shmin + Stress Shadowing
12. Create an external fracture for each well being frac'd.
13. As history matching proceeds, iterations may be required.

Q: The water saturation column is not showing up in the static model table even after I entered the logs using the wizard?

This depends on the 'Initial water saturation input method' selected in the Static Model panel. If 'Specify depth of water-hydrocarbon contact' is chosen for this method, then the column for water saturation is removed from the table. Instead, the water saturation is defined based on the 'Depth of water-hydrocarbon contact' parameter lower down on the panel. To define water saturation in the table, switch to the 'Specify by layer' method.

Q: Simulations are failing after moving the wellbore and recentering the matrix.

Simulations will fail if moving the wellbore has caused one or more perforations to be moved outside of the matrix. A warning box should have been present alerting the user to the problem. Rerunning the 'Well stages setup wizard' in the Wells and Perforations panel after moving the wellbore (or matrix) will alleviate this problem by reassigning stage assignments in the well vertices table. If you are using the 'inline' method for cluster setup, this along should resolve the problem; if you are using the 'one-at-a-time' method, you may need to adjust your cluster locations such that they are contained within the matrix.

Q: After running the 'Well stages setup wizard' I receive a yellow warning message that a wellbore vertices is within one meter of another vertex.

Searching the wellbore vertices table in the MD column locate the two vertices that are within 1 meter and delete the wellbore vertex that does not affect the staging of the well.

Q: My fractures are continuing to grow after the pumps are turned off and the next stage begins. Is this normal?

Yes this is normal. You can look at the following in the UI to understand the mechanisms at work. First, the fractures are probably not leaked off entirely and there is some residual aperture. You can view the 'Total Aperture' or 'Epoen' (the open part of the aperture that is not touching the other fracture wall or proppant) and

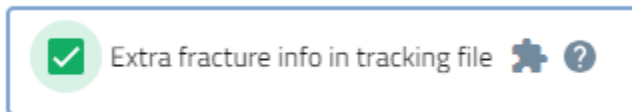
see if the fracture has closed or not. When the next stage begins a process known as ‘forced’ closure begins due to the stress shadowing increases. The aperture from the new fractures are closing the previous stage fractures and pushing the fluid to the tips of the fractures. In many cases the pressure of this fluid is sufficient enough to propagate the fractures further. To reduce the effect, consider adding/increasing the effects of reversible pressure dependent permeability (PDP) which is located under the specific curve sets in the curve sets tab.

Q: Why do we often need to use proppant conductivity curves that are lower than lab measured values?

Conductivity values are lower than the lab results for many reasons. One of the primary reasons is that proppant placement is patchy in the fractures. Some places have much more proppant than average, some places much less. The unpropped areas between the patches is probably what controls conductivity the most.

Q: Can I output the maximum and average fracture lengths and heights for each individual fracture?

Yes, but you must have ‘extra fracture info’ checked in the Advanced options of the Output Options panel.



Q: How can I stagger my perforations in adjacent wells such that the perforations or clusters are not directly aligned?

A few suggestions:

- 1) Start by setting your ‘Perforation cluster location specification method’ to the "inline" method - that ensures that the perfs are collocated with the wellbore.
- 2) If you need individual control of perfs afterward, you can use the 'Convert perforation clusters wizard' to transform your inline perfs to a table of perfs (the reason to do this would be if you had a tapered perforing scheme, like more shots in toe clusters than heel clusters).

Wells and Perforations

Specify parameters related to wells and perforations

Editor

Search

Wells and perforations library ?

Convert perforation clusters wizard ?

Use length per cluster to specify perforation cluster locations ?

Specify clusters per stage

Specify length per cluster

Perforation cluster location specification method ?

Specify perforation clusters one-at-a-time

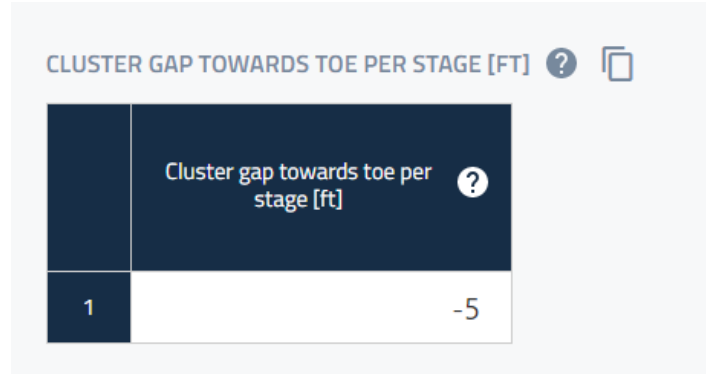
Specify using 'inline' method (e.g., number of clusters per stage)

3) In fracture options there is a fracture initiation randomness distance. This parameter does not literally shift the cluster, but instead says that the frac initiates at some random distance away from the perf (this is actually physically validated by mine-back studies):

Fracture initiation randomness distance [ft]

4) In addition or alternatively, you can systematically shift perfs using the 'Cluster gap towards toe per stage' setting in the Wells and Perforations panel (only available in the inline method):



If you use this option, and also check the 'Shift for cluster gap towards toe per stage' setting in the Advanced options, it shifts all clusters in that stage along the MD of the well by that set amount.

Shift for 'cluster gap towards toe per stage' 🧩 ?

So, for the example in the screenshot above, all perfs would be shifted 5 feet toward the heel of the stage. If you had an adjacent well, and used a 'cluster gap towards toe per stage' of +5 feet, then the clusters from the two adjacent wells would be staggered by 10 feet (in the direction of Sh_{min}), presuming both wells had the same stage length and clusters per stage.

[Q: Is water or gas rel perm used for the steam phase?](#)

The steam/water transition is handled with a pseudo-two phase approximation (treating it as one phase), with properties interpolating across the saturation line. This is an approximate treatment for handling genuine steam/water flow in the reservoir, but this practically is not significant since we so seldom encounter actual steam/water flow in Enhanced Geothermal Systems (EGS). For more detail, see Section 14.1 of the Technical Writeup. For this singular phase, water relative permeability curves are utilized.

[Q: Do you include overflush volumes in the pump schedule in ResFrac?](#)

For ResFrac, you should enter the entire injection schedule as pumped (including all chase, flush, and overflush).

- ResFrac models the entire wellbore, from the surface to the reservoir. Because injection originates at the surface, a portion of the injected volume will remain within the wellbore and will not reach the reservoir or fractures.
- We typically don't observe a significant negative impact from "overflush" in unconventional wells. This is due to the favorable dimensionless conductivity (nanodarcy permeability) and the proppant immobilization parameter. The latter prevents proppant from being entirely swept away from the wellbore in a piston-like fashion, which aligns with proppant tracer studies showing that some proppant consistently ends up near the wellbore, regardless of when it's pumped during a stage.

- We also advise setting the 'Propped wellbore-to-fracture connection conductivity multiplier' parameter in the fracture options to 100, which is the recommended default value. This mimics phenomena such as scouring and rubblizing that can occur at the fracture/wellbore interface.

Q: How do I use the Pressure dependent NW dP (tortuosity) option in the advanced options of the Wells and Perforations tab?

There are four separate parameters controlling the near wellbore tortuosity (NWT). Each of these four parameter names begin with 'Pressure dependent NW dP'. Here is our recommended workflow:

1. Set both 'minimum transmissibility' and 'reference transmissibility' to 10.
2. Set 'reference pressure' to a value around the expected Shmin without tortuosity, example 7600 psi.
3. Do a sensitivity on 'scaling pressure' between 10, 100, 300, and 1000 psi. In most of my cases the scaling pressure was between 100 - 300 psi.
4. Check that near wellbore pressure is not reducing production.

Q: If using a dual porosity (DP) option, what is the difference between 'DP matrix permeability' and 'X/Y/Vertical direction permeability' in the static model table?

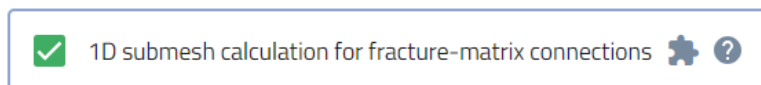
In a dual porosity model, there are three types of flow connections that can occur: (a) flow between a discretely meshed fracture (hydraulic fracture) and a DP fracture element, (b) flow between a DP fracture element and its colocated matrix element, and (c) flow between two different matrix elements. Each is governed as follows:

- A. Discretely meshed fracture <-> DP fracture element: 'X/Y/Vertical direction permeability'
- B. DP fracture element <-> colocated DP matrix element: 'DP matrix permeability' (and 'DP shape factor')
- C. Matrix element <-> neighboring matrix element: 'X/Y/Vertical direction permeability'; Note: If the 'Dual permeability' option is not enabled, there is no fluid exchange between adjacent DP matrix elements; fluid exchange only occurs between neighboring DP fracture elements.

The 'DP matrix permeability' should be lower than the 'X/Y/Vertical direction permeability' usually. The latter is the bulk system perm from the natural fractures represented in the DP model.



Q: If I am using the fractal D parameter, should I adjust my matrix meshing if my well is not aligned with Shmin?

Firstly, the fractal D parameter should only be used along with the 1D fracture submesh. It should not be used with a [refined mesh](#). This is typically on by default.



Secondly, when using the fractal D parameter you should use a relatively coarse matrix mesh, targeting a minimum size of 20 ft in the Shmin direction. That is because the fractal D parameter modifies the values in the 1D submesh in order to increase the flow ‘volume’ and ‘area’ in the vicinity of the fracture. This effect diminishes with distance, out to 20 ft from the fracture face. Since the size of the 1D submesh is limited by the size of the matrix element in the Shmin direction, setting a coarse matrix mesh ensures that the submesh is sufficiently large to represent the fracture swarms.



When considering a well that isn’t aligned with Shmin, check to see if the option to ‘Rotate simulation mesh to align with stresses’ is selected on the Startup panel. If this option is selected, then the matrix mesh will be oriented in the same direction as the fractures, and the mesh size does not need to change. The ‘Matrix region length along the Shmin direction’ just needs to be a minimum of 20 ft.

Rotate simulation mesh to align with stresses  

The submesh fractal D parameter is located in the Fracture Options panel, and can be assigned globally, or by layer in a table (in the Advance options of the same panel). Specifying values in the table will override the global value.

Submesh fractal D 

SUBMESH FRACTAL D BY LAYER  

	Layer name 	Submesh fractal D by layer 
1	overburden	
2	reservoir	
3	underburden	

Q: How is uncertainty accounted for in simulations?

There are three main categories that incorporate some level of uncertainty or stochastic behavior in simulations: (1) fracture initiation, (2) fracture propagation, and (3) wellbore fluid dynamics (perforation erosion). Each run of a simulation will initialize the various parameters as specified, but dependent on a seed

(as it is actually pseudorandom) such that each simulation run will be reproducible. If you wanted to create a different run with a different "draw" for all of these variables, you could change the 'Random seed' in the 'Other Physics options panel':



Random seed  

Here is a quick survey of most of the important uncertainty parameters. You can always read more by hovering over/clicking the question marks in the UI.


(1) Fracture initiation

For the common case of a cased and perforated wellbore, when a simulation is initialized, potential initiation points are set at every specified cluster in the simulation. If the parameter 'Fracture initiation randomness distance' is >0 , then this initiation point can be $\pm X$ feet from the location of the cluster.

During stimulation, the 'decision' to initiate a fracture at one of these points depends on several things, but one is whether the pressure at that cluster can overcome the tensile strength of the rock. If 'Randomized effective tensile strength' ('Static Model and Initial Conditions' panel) is checked, then the effective tensile strength at each cluster is varied on a uniform distribution between 0 and the value defined in the static model table.

Randomized 'effective' tensile strength  

An exception to this exists if 'Perforation phasing option' is not set to 'NoEffect'; in this case, the uncertainty is taken from the 'Tensile strength uncertainty' parameter in 'Wells and Perforations.'

Perforation phasing option 



- NoEffect
- BuiltIn
- Custom

Tensile strength uncertainty [psi]  

(2) Fracture propagation

For most models, we assume a layer-cake model with homogeneous rock properties in each layer. From a numerical modeling perspective in which we include the effect of stress shadowing between clusters, this presents a numerical instability quirk for which introducing a very small heterogeneity in the toughness of each matrix element is an elegant fix. When fractures first form, they may be perfectly symmetric, with no heterogeneity in toughness to induce asymmetry to the left or right, up or down. This is an unstable equilibria however, and accumulating roundoff errors will eventually cause asymmetry to form. Instead of allowing this to

occur from a numerical mechanism, it's more practical and physically sound to introduce some small heterogeneity in toughness. This is done using the 'Fracture toughness heterogeneity parameter' in the advanced Fracture Options panel.

Fracture toughness heterogeneity parameter  


(3) Slurry/Proppant dynamics in the well and perforation phasing

As alluded to above, changing the 'Perforation phasing option' from 'NoEffect' to 'BuiltIn' or 'Custom' actually changes the way the code calculates fluid/proppant distribution in the well. When one of these options is enabled, you specify the phasing of each perforation in a cluster, and the code calculates fluid/proppant flow and erosion for each individual perforation rather than solely on a cluster basis. When using this feature, there are a variety of other 'uncertainty' parameters that can come into play (see screenshots below).

Range for erosion alpha [ft]  


Nugget for erosion alpha  

Nugget for erosion sill  

Uncertainty distribution 

Normal



Uniform

Phasing uncertainty at perf level  

Phasing uncertainty at cluster level  

Phasing uncertainty at stage level  

Perforation diameter standard deviation [in]  

Perforation diameter standard deviation by stage [in]  

Tensile strength uncertainty [psi]  

[Q: How can I quantify and visualize the size of my hydraulic fractures \(e.g. half-length\)?](#)

There are several parameters available to quantify the fracture size in the viz tool line plots. Some of these values are aggregated from across the entire model ('overall' values), while others are 'per well' values. Additionally, [some parameters are available on an individual fracture basis](#) (the prerequisite for this option is that 'extra fracture info' must be checked in the Advanced options of the Output Options panel prior to running the simulation). All the parameters available to quantify the fracture size via line plots (on either an overall, per well, or per frac basis) are listed below:

Horizontal fracture size:

- Max horiz dist (right) from well [per frac]
- Max horiz dist (left) from well [per frac]
- Max horiz dist from well [per frac]
- Max horiz dist from well (per well)
- Max horiz dist from well at landing depth (per well)

- Largest value of 'max horiz dist from well', Overall
- Median 'fracture length (averaged across height)', Overall
- Length (averaged across height), [per frac]
- Largest value of 'fracture length (averaged across height)', Overall

Vertical fracture size:

- Max vert dist from well [per frac]
- Max vert dist above well [per frac]
- Max vert dist below well [per frac]
- Max vert dist above well (per well)
- Max vert dist below well (per well)
- Largest value of 'max vert dist from well', Overall
- Median 'fracture height (averaged across length)', Overall
- Height (averaged across length), [per frac]
- Largest value of 'fracture height (averaged across length)', Overall

Fracture area:

- Surface area [per frac]
- Fracture surface area, Overall
- Productive area, Overall

Fracture volume:

- Total volume [per frac]
- Fluid pore volume [per frac]

You may also be interested in considering the propped fracture size, which is often a better metric when considering the conductive fracture size. This is quantified with the following parameters:

Horizontal propped fracture size:

- Max propped horiz dist from well [per frac]
- Max propped horiz dist from well (per well)
- Largest value of 'max propped horiz dist from well', Overall
- Median value of 'max propped horiz dist from well', Overall

Vertical propped fracture size:

- Max propped vert dist from well [per frac]
- Max propped vert dist from well (per well)
- Largest value of 'max propped vert dist from well', Overall
- Median value of 'max propped vert dist from well', Overall

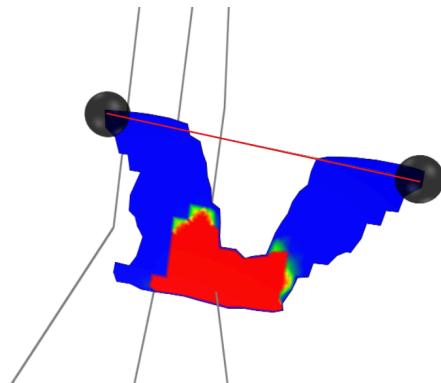
Fracture area:

- Propped area [per frac]
- Propped frac surface area, Overall

Note, there is no parameter that specifically quantifies half-length. To quantify a half-length value you can take the full averaged length and halve it, or else use the left or right horizontal distance. Note, left and right are from the perspective of the heel, facing the toe.

There are important nuances in how fracture size is quantified that you should be aware of when interpreting results. Some of these are as follows:

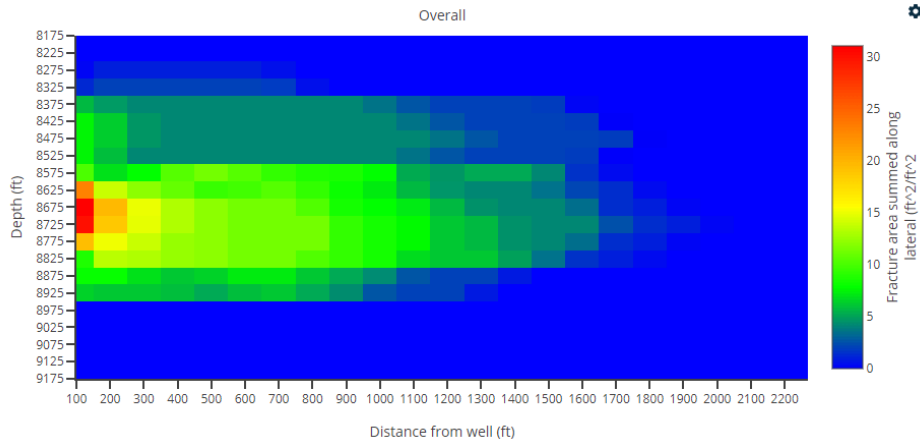
- Frac hits (frac-to-frac):
 - When two fractures hit each other, they will merge in the simulation into a single, large fracture. These fractures are thus tied to the originally stimulated well in aggregate geometry metrics. Thus, this will impact the median and maximum fracture sizes. Note, fractures that aren't merged in the viz tool but overlap with each other may still be hydraulically connected. This connection can be observed from the pressure gradient (or lack thereof) between overlapping fractures.
- Frac hits (frac-to-well):
 - When a fracture intersects an unstimulated well close enough to a perforation, future stimulation on this second well may pump into the connected fracture, instead of initiating a new hydraulic fracture. In this frac hit case, the original fracture will propagate further due to injection from the second well, but the frac will still be designated as originating from the first well. Thus, no new frac will be assigned to well 2, but the length of the well 1 frac will increase.
- Winged fractures
 - If a fracture has wings that don't connect horizontally (see image below), the length value will still be the same as if the fracture were 'filled'. In other words, ResFrac neglects to consider holes during the length (or height) measurements.



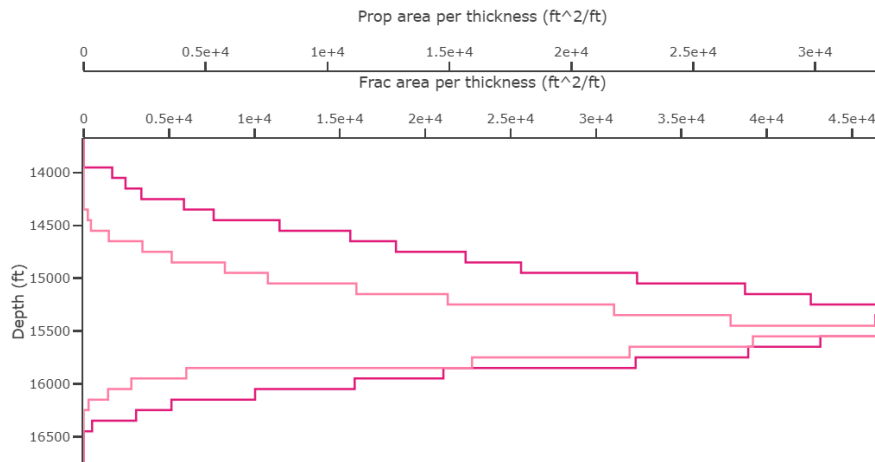
Frac hits complicate the assignment of fractures to wells, impacting the per well parameters. To get the cleanest answer and avoid the nuances associated with frac hits, you may prefer to only look at the frac lengths at the end of the first well stimulation.

Besides line plots, there are other ways you might choose to visualize the result.

Gunbarrel plots show a summary of the combined fracture distribution across the entire model. Alternatively, you may plot the proppant distribution or the fracture conductivity distribution relative to depth and distance from the well. Note, the 'distance from well' axis will be impacted by the same nuances related to frac hits as discussed above.



Another option is the Fracture vs depth plot, which represents an aggregation of the fracture attributes with depth. Both views represent aggregations of all the fractures.



5. Sensitivity Analysis and Automation Workflows

Q: How can I 'restart' an automation workflow from a previously run iteration?

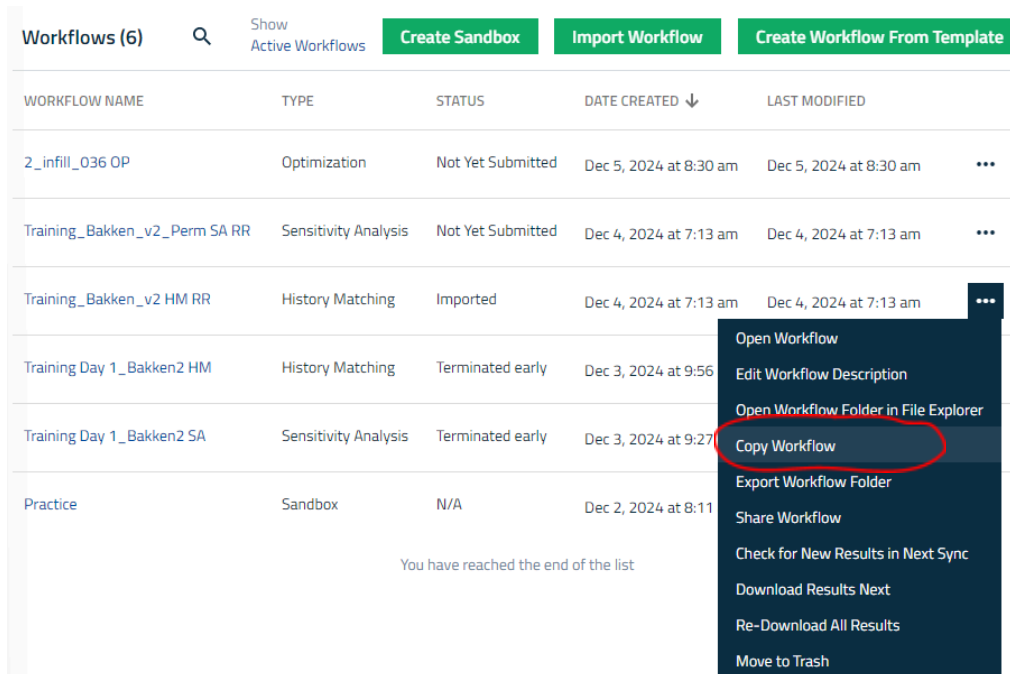
This question describes how to run an automated workflow (history matching or optimization) with an advanced start. Advanced start means that you can use the results from a partial or complete history matching or optimization workflow and run one or more additional iterations of the same workflow. The typical use case for this is that you have a history matching or optimization workflow that you want to run more iterations on because of early termination (e.g., due to a server issue that has been fixed), or because you feel that an additional iteration would improve the results of the optimization. This does not apply to sensitivity analysis workflows because those iterations are manually selected by the user.

It is very important that you do not change anything related to how the workflow is defined, such as the parameters, parameter groups, target functions, and base simulation. These must all be the same between the source workflow and the advanced start workflow. The advanced start workflow is likely to fail or give bad results if any changes are made between the source workflow and advanced start workflow.

ResFrac's development plan includes providing access to this feature within the ResFracPro UI, but at the moment that is not available so you need to take some manual steps in order to do this. The steps are as follows:

1. Copy workflow

Make a copy of the workflow you want to use for the advanced start.



The screenshot shows a table of workflows with columns for Workflow Name, Type, Status, Date Created, and Last Modified. A context menu is open for the 'Training Day 1_Bakken2 SA' workflow, with 'Copy Workflow' highlighted. The menu also includes options like 'Open Workflow', 'Edit Workflow Description', 'Open Workflow Folder in File Explorer', 'Export Workflow Folder', 'Share Workflow', 'Check for New Results in Next Sync', 'Download Results Next', 'Re-Download All Results', and 'Move to Trash'.

Workflow Name	Type	Status	Date Created	Last Modified
2_infill_036 OP	Optimization	Not Yet Submitted	Dec 5, 2024 at 8:30 am	Dec 5, 2024 at 8:30 am
Training_Bakken_v2_Perm SA RR	Sensitivity Analysis	Not Yet Submitted	Dec 4, 2024 at 7:13 am	Dec 4, 2024 at 7:13 am
Training_Bakken_v2 HM RR	History Matching	Imported	Dec 4, 2024 at 7:13 am	Dec 4, 2024 at 7:13 am
Training Day 1_Bakken2 HM	History Matching	Terminated early	Dec 3, 2024 at 9:56	
Training Day 1_Bakken2 SA	Sensitivity Analysis	Terminated early	Dec 3, 2024 at 9:27	
Practice	Sandbox	N/A	Dec 2, 2024 at 8:11	

2. (Optional) Increase the iteration count

If your source workflow ran the number of iterations you specified previously, and you want to run another iteration, open the new workflow in the builder to the decision support panel, increase the number of iterations, and save. If you are working with a workflow that did not reach the final iteration specified in the original workflow, this step is not necessary.



Number of workflow iterations  

3. Copy the pointsummary.csv file

Copy the file “pointsummary.csv” from the source workflow, located at

`<source workflow folder path>/Results/pointsummary.csv`

to the new workflow’s “Additional_Files” folder, i.e.,

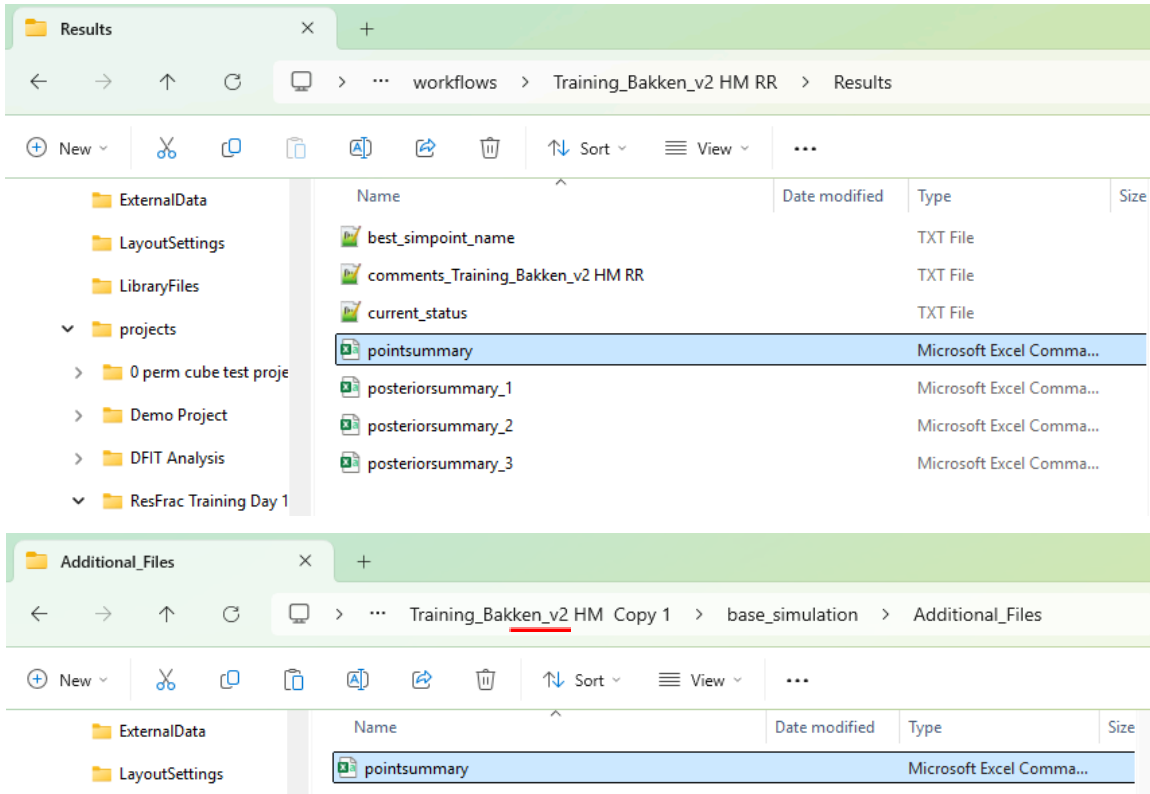
`<new workflow folder`

`path>/base_simulation/Additional_Files/pointsummary.csv`

WORKFLOW NAME	TYPE	STATUS	DATE CREATED ↓	LAST MODIFIED	
Training_Bakken_v2 HM Copy 1	History Matching	Not Yet Submitted	Jul 20, 2025 at 9:36 pm	Jul 20, 2025 at 9:36 pm	...
2_infill_036 OP	Optimization	Not Yet Submitted	Dec 5, 2024 at 8:30 am	Dec 5, 2024 at 8:30 am	...
Training_Bakken_v2_Perm SA RR	Sensitivity Analysis	Not Yet Submitted	Dec 4, 2024 at 7:13 am	Dec 4, 2024 at 7:13 am	...
Training_Bakken_v2 HM RR	History Matching	Imported	Dec 4, 2024 at 7:13 am	Dec 4, 2024 at 7:13 am	...
Training Day 1_Bakken2 HM	History Matching	Terminated early	Dec 3, 2024 at 9:56		...
Training Day 1_Bakken2 SA	Sensitivity Analysis	Terminated early	Dec 3, 2024 at 9:27		...
Practice	Sandbox	N/A	Dec 2, 2024 at 8:11		...

You have reached the end of the list

- Open Workflow
- Edit Workflow Description
- Open Workflow Folder in File Explorer
- Copy Workflow
- Export Workflow Folder
- Share Workflow
- Check for New Results in Next Sync
- Download Results Next
- Re-Download All Results
- Move to Trash



4. Add advanced start flag to dsparams.json

Open the new workflow's dsparams.json file, located at

`<new workflow folder path>/base_simulation/dsparams.json`

in a text editor such as Notepad. Add the following line into the file on a new line after the first "{" character:

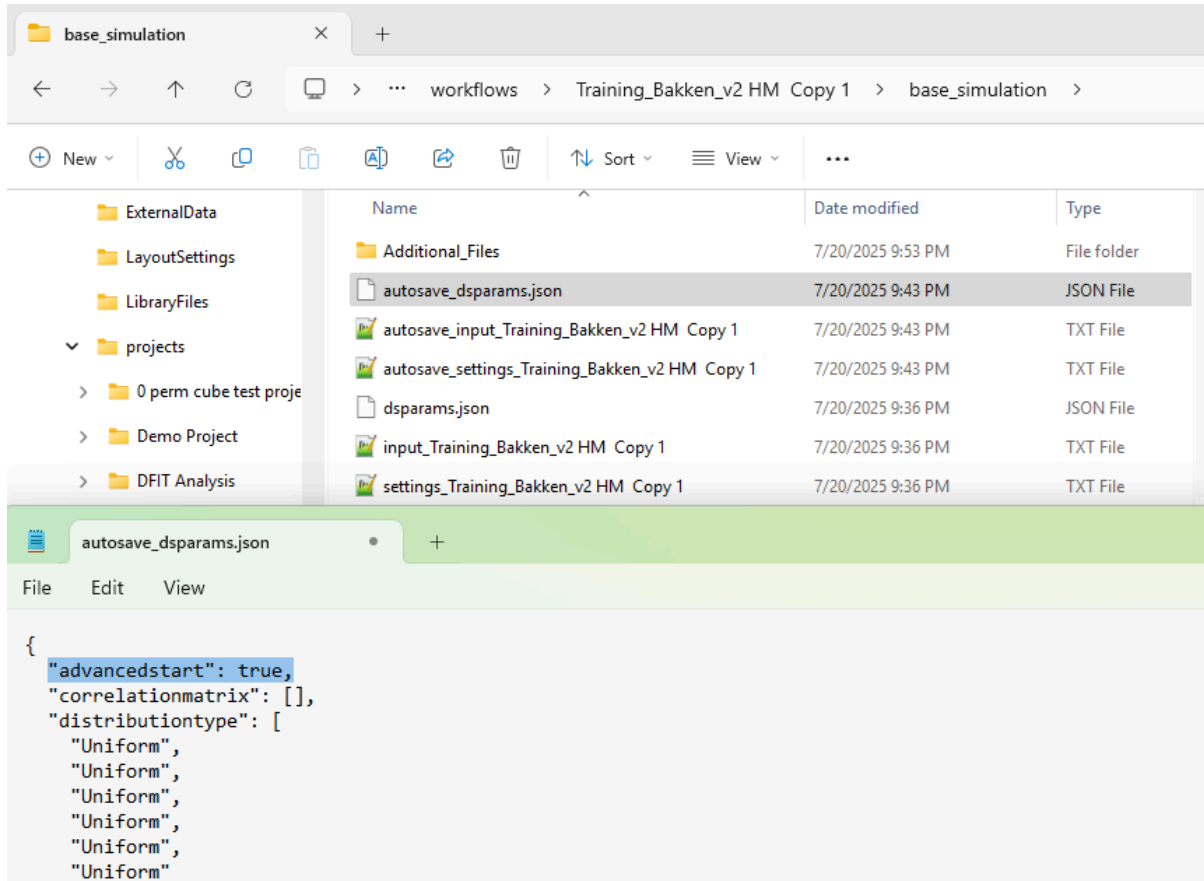
```
"advancedstart": true,
```

Make sure to use the exact text above (do not add, remove, or change any characters).

Optionally, if you want to generate a posteriorsummary_X.csv file at the start of the workflow using the advanced start points, you can also add the following line immediately after:

```
"advancedstartposterior": true,
```

As with the advanced start line, here you also need to be very careful to use the exact text. Save the file and exit.



5. Run and monitor

Run the new workflow in the ordinary way.

Open up the workflow comments file and see that it is running as an advanced start. There should be a line in the comments file that starts as follows:

Attempting to load 'advanced start' data

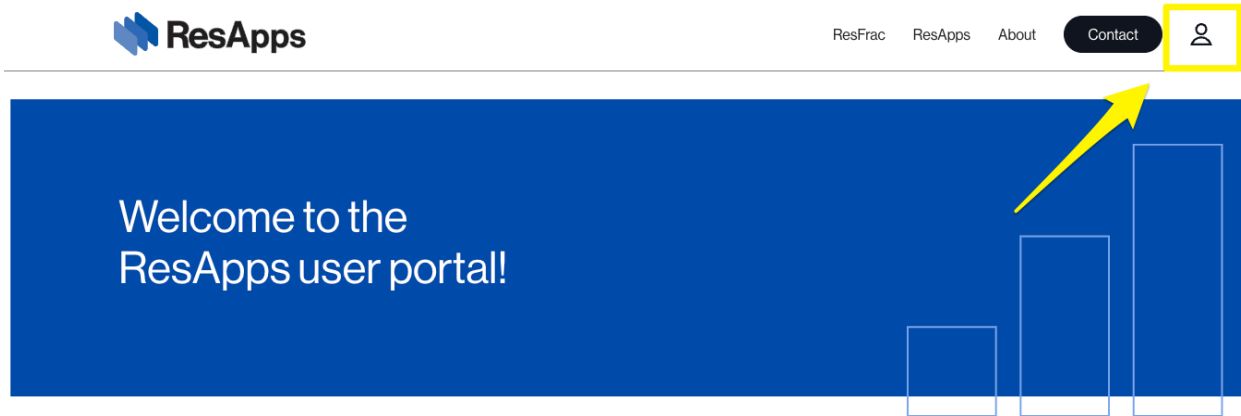
And the comments file should say which iteration number it is on (it should start with a number after the last iteration from the source workflow). The workflow should behave just the same as one started from scratch, except that the first N iterations will be skipped because their results will have been pre-loaded at the start.

6. History Matching Topics

7. ResApps

Q: How do I log-in and sign-up for a ResApps license?

To login to the ResApps portal, navigate to www.resapps.resfrac.com. Log in to the ResApps portal via the icon on the upper right hand corner of the screen:



There are two authentication options available to log in to the system, Microsoft and Google. Please use your organization assigned email address.

Login

Please log in to ResApps to view this page.

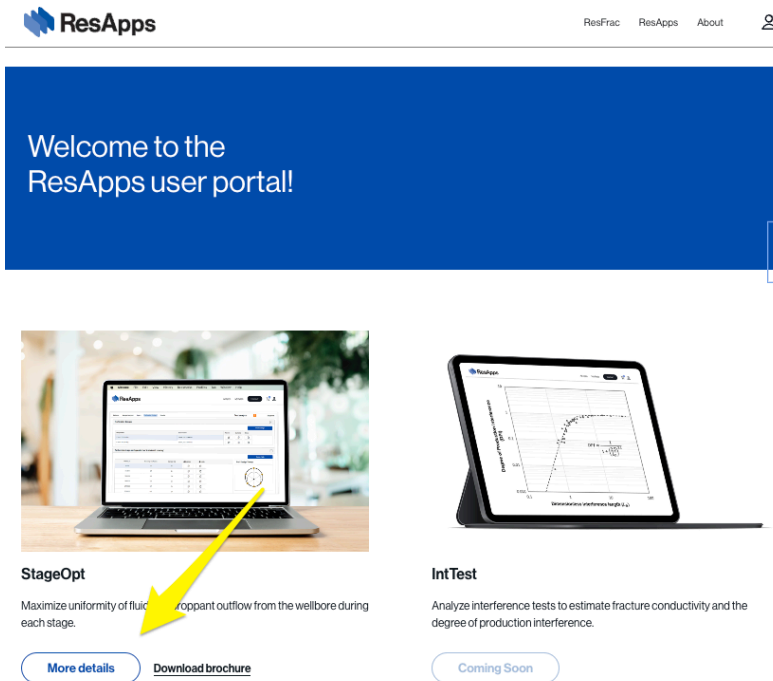
Creating an account is free. You can use any email address, but it must be registered with either Google or Microsoft. Most corporate email addresses are already registered with Microsoft by default. If you do not already have a Google or Microsoft email, then please use [this link to register](#).

 Sign in with Microsoft

or with

 Sign in with Google

Navigate to the desired app and click “More details” to get more information on the app, and to sign up.



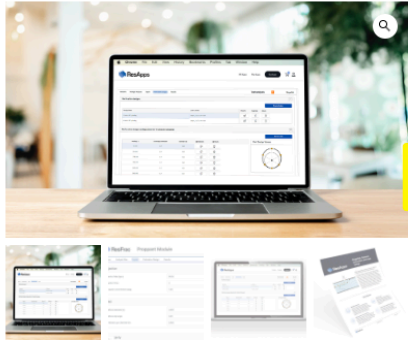
StageOpt
Maximize uniformity of fluid flow and proppant outflow from the wellbore during each stage.

[More details](#) [Download brochure](#)

IntTest
Analyze interference tests to estimate fracture conductivity and the degree of production interference.

[Coming Soon](#)

When you are ready to purchase a license for the application, click “Sign up now”:



StageOpt

Maximize uniformity of fluid and proppant outflow from the wellbore during each stage.

[Sign up now](#)

Description

Maximize uniformity of fluid and proppant outflow from the wellbore during each stage. Implements correlations for limited entry, perforation erosion, and the effect of proppant outflow from the wellbore. Implements the generalized and well-validated proppant outflow model developed by Egor Dontsov. Avoiding toe/heel bias maximizes the value of your fluid and proppant injection, with direct impact on well performance.

This will take you to the checkout page. You will fill in all details in the form to the left. If your purchasing department requires a PO, work order, or coding on the invoice in order to process, you will need to enter the information in the Order Notes section of the checkout page.

Please review and check to accept the terms and conditions, and press "Sign up now":



Billing Details

First Name * Last Name *

Company Name (optional)

Country / Region *

Street address *

Town / City *

State / County *

Postcode / ZIP *

Phone *

Email Address *

AvaTax uses this email ID for tax exemption. To receive tax exemption for the order, ensure that the email ID you enter here is applicable for tax exemption.

Order Notes (optional)

Your Order

Product	Subtotal
StageOpt	Via Invoice
Exemption Add Certificates Manage existing certificates	
Have a coupon? Click here to enter your coupon code	
Pay offline <small>This could involve: (a) paying with a check or direct deposit, (b) a corporate group license, or (c) another license type. After you submit, we will email you with further instructions, and you will not be able to access the app until your license has been manually activated.</small>	
<small>Your personal data will be used to process your order, support your experience throughout this website, and for other purposes described in our privacy policy.</small>	
Web Hosted Application – Terms Of Use <small>These Web Hosted Application Terms of Use (herein, “Agreement”) govern Customer’s (as defined below) access to and use of the proprietary web-hosted software-as-a-service application platform located at www.resfrac.com/resapps (the “Platform”)</small>	
<input type="checkbox"/> I’ve read and accept the above terms & conditions. If you do not agree to terms and conditions, your license will be put on hold.	
Sign up now	

This will submit your order, and we will email you further instructions. You will not be able to access the app until your license has been manually activated and terms and conditions have

been accepted. If you have any questions or need support in the meantime, please send us an email at resapps@resfrac.com. Thank you!